Getting started with

1 • 2•Build

By purchasing the **1•2•Build** software, the purchaser acquires a license of use. Under no circumstances can the user assign the license in part or in total to a third party without a prior written consent of the publisher.

The publisher holds no responsibility for possible faults that the program and/or the here mentioned manual may include and declines any responsibility for damages arising from misuse of the **1•2•Build** software and/or the here mentioned manual.

Contents

1 Foreword

1.1 What can 1•2•Build do for you?

1•2•Build is a program for simple, quick stability calculations. It allows the most common beams, columns and trusses used in residential construction as safe and economical dimensioning.

The calculation of such structures is done with **1•2•Build** more elastic, using the most modern computing technology (relocation method).

The final sizing can be performed for the materials, concrete, steel and wood - and always according to the Eurocodes (with verification of the ultimate limit state and serviceability limit states).

1.2 What is the difference between the demo version and full version of 1•2•Build?

The demo version is designed solely for demonstration purposes. There are also some limitations built in to ensure that the demo version is not usable in practice:

- The value of the loads are not adjustable as point loads have consistently rated at 10kN and the distributed load 10kN/m
- **This limitation is the last drop function not accessible in the demo version of 1•2•Build.**

This demo version is also entirely similar to the full version of **1•2•Build** and therefore give you a minimum of time a good look at the functionality that **1•2•Build** offers.

The full **1•2•Build** package uses a license key to a USB port on your computer.

1.3 Required hardware

1•2•Build is designed for MS Windows. Your computer must have MS Windows 2000, NT, Me, XP, Vista, 7, 8, 8.1 or 10 and is equipped with at least 64 MB of RAM. Both 32-bit and 64-bit operating systems are supported.

The speed of 1•2•Build your stability calculations is proportional to the clock speed of your processor. Preferably, your computer has an Intel Pentium 4 processor or AMD Athlon processor with a clock speed of 1 GHz or higher.

1.4 Preliminary information

Before going further, you need to be familiar with elementary commands of your MS Windows operating system and with the use of windows and icons, selected features, copy-cut-paste and using a mouse. An overview:

Icon:	Graphic representation of a program or part of a program.
Clicking with a mouse:	Pointing a given element or given area and clicking 1 time with the mouse button.
Selecting:	Clicking 1 time on an icon or element. You may also select several elements with the use of a grid: click the top left corner of a rectangle which is to comprise the chosen selection – hold the mouse button and drag it towards the opposite right corner, then release the mouse button. You may enlarge the selection by following the above procedure with pressed SHIFT key.
Double-clicking:	Quickly click your mouse 2 times. This feature is used to start a program or a part of a program.
Copy-paste:	Selecting an element and copying it to a different location in the program via Edit menu.
Dragging:	Drag a given element by selecting it and moving the mouse while keeping the button pressed.

1.5 Installation

1.5.1. Administrator rights for installation

You need your computer to log on as administrator to BuildSoft products completely and correctly installed.

1.5.2. Installation

1.5.2.1. What to install?

You should install the following programs:

- CodeMeter Control Center (= program for licences management)
- **1•2•Build**

The setups for these programs can be found on our website www.buildsoft.eu. Or you can retrieve them from the delivered USB stick when you purchased the program.

1.5.2.2. Where to install?

During installation you will be asked where you want to install the selected product. It will be proposed to perform the installation in the directory 'C:\Program Files(x86)\BuildSoft\12Build'.

We advise you to confirm this proposal, but you're free to install our products in another location.

During the rest of the installation procedure, you can confirm the proposed steps by using the button 'Next'.

1.5.2.3. Finishing up

Upon completion of the installation you will be asked whether you want to re-start the computer.

1.5.2.4. Activate your license

Your 1•2•Build license is on a CM-key (dongle) that was sent along with this manual. Place the license key into a USB port on your computer. Then start **1•2•Build on. 1•2•Build** is immediately and fully functioning.

If this is not the case, please contact us.

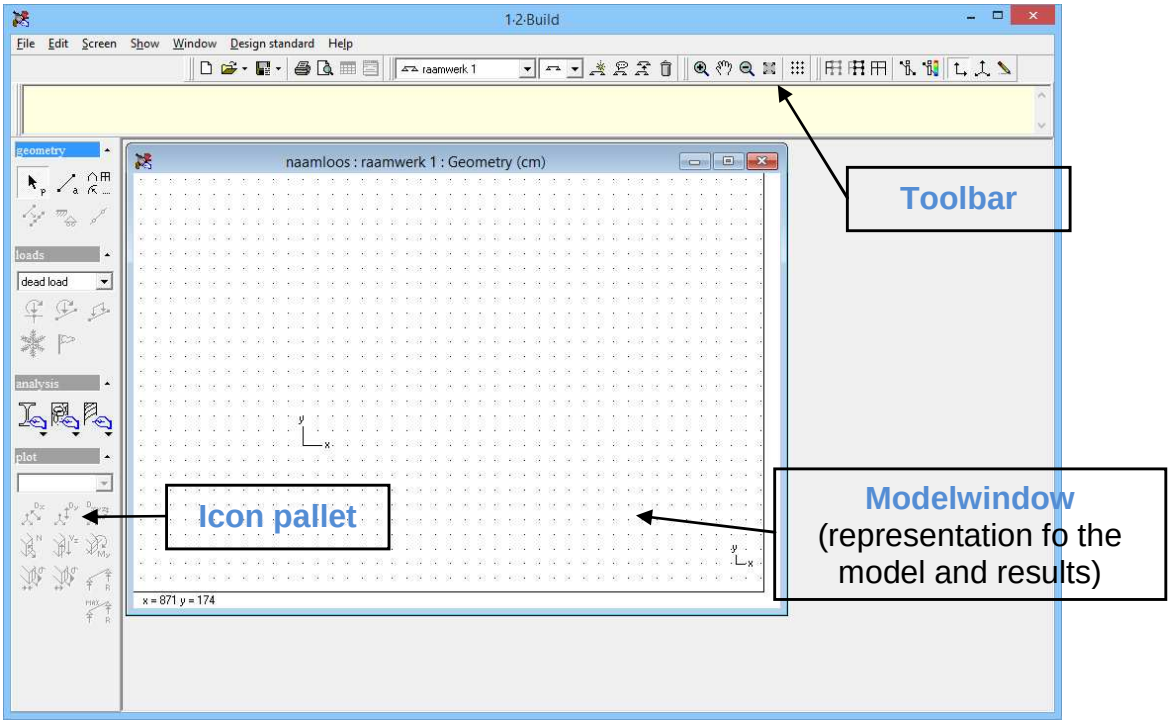
2 Dimensioning of a continuous beam

Start **1•2•Build** on Windows via the Start menu. If no licence was found, **1•2•Build** will remind you that you're working in a demo mode (= limited version) of the software.

Within the worksheet of **1•2•Build** the following things appear:

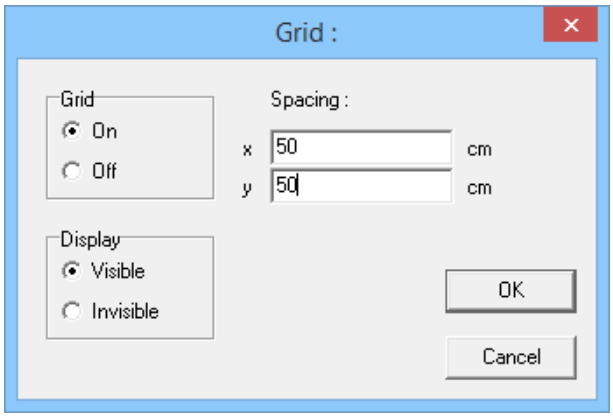
- a model window in which the computational model and the corresponding results are presented.
- an icon palette with 4 separate fields. This palette gives you access to all modelling and calculation functions of 1•2•Build and enables quick navigation through the product possible.
- a toolbar with a set of generic functions for opening and saving of 1•2•Build files, printing a calculation note, the management of computational models within a 1•2•Build file and the visualization options.

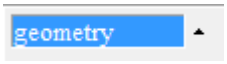
We will now use the functionality of these windows to following continuous beam can be calculated in steel, concrete and wood.



2.1 Drawing a continuous beam

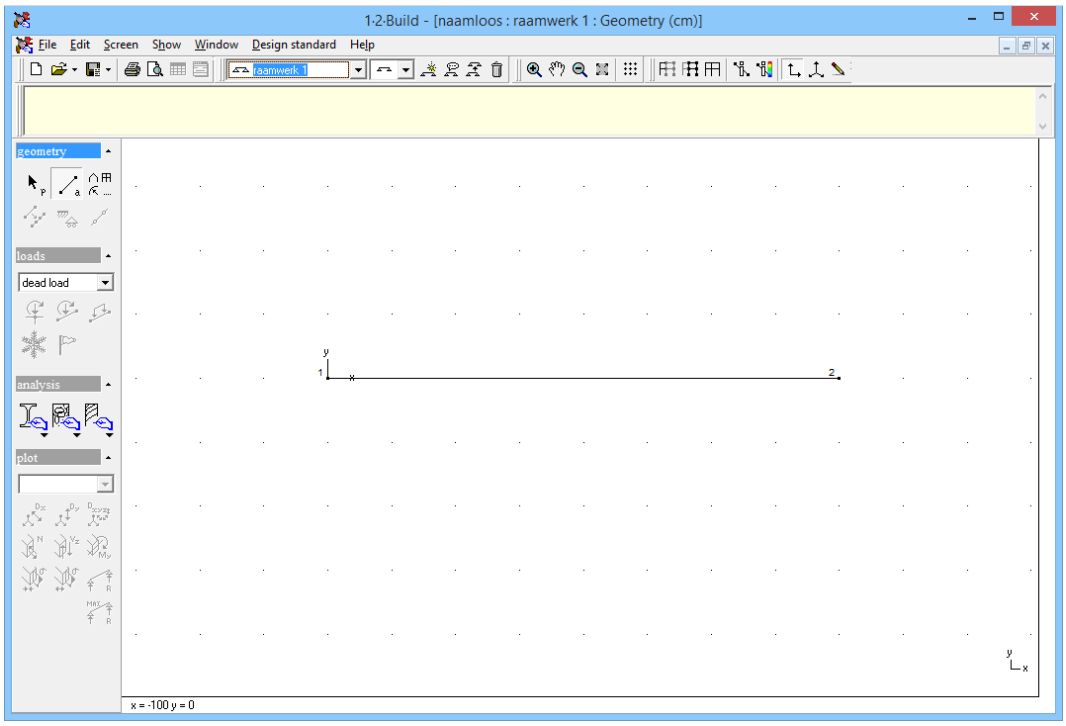
Prior to starting the actual drawing, we ensure that there can be drawn on a grid with a sufficiently fine resolution. Activate the grid definition with  and then set the resolution of the grid to 50 cm in both directions.



Select the icon field 

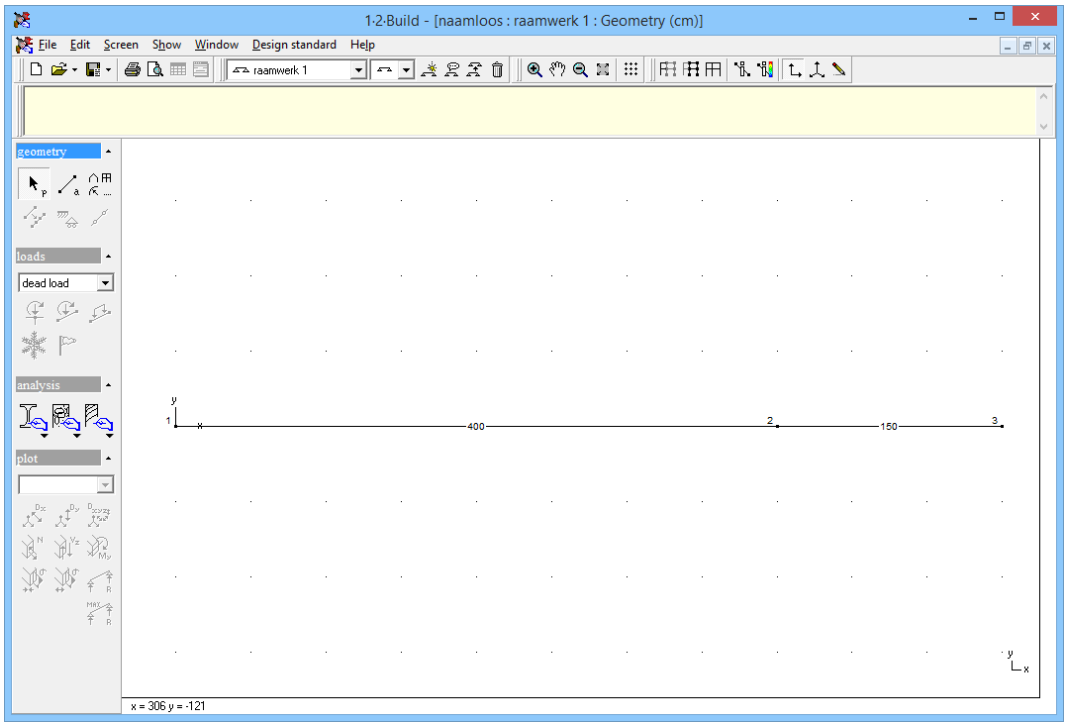
within the palette. In this field, icon three functions are actually available (). All other functions are dimmed and will only be useful after a selection was done in the window.

Click on the icon  , where after you can draw the axes of the continuous beam directly on the screen. Click the left mouse button on a point of the grid where you want to start drawing. Then drag the mouse to the right (while holding the left mouse button pressed). As you move the mouse over the window, you'll the length of the drawn line on the bottom of the window. Move the mouse to the point where dx = dy = 400 cm and 0 cm. Then release the left mouse button.

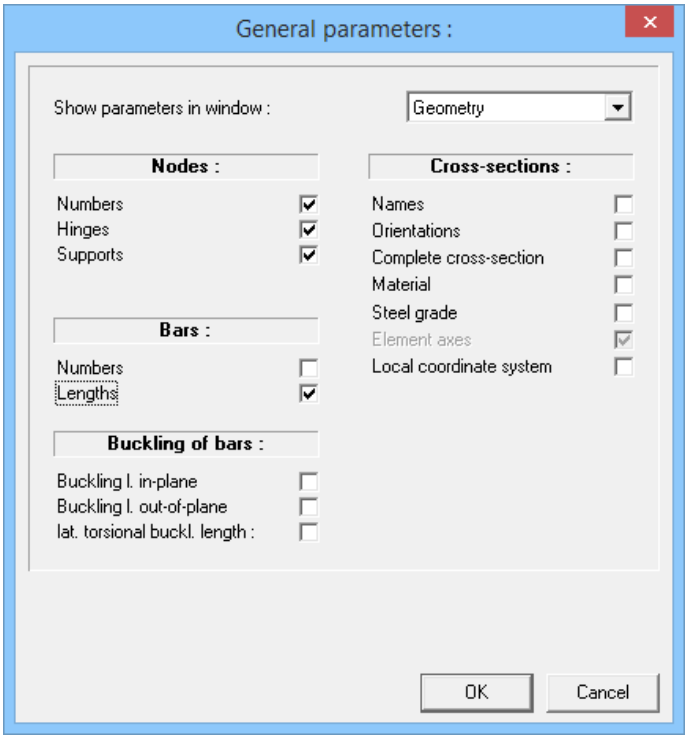


So, you now drawn the first field of your continuous beam. Draw the next field by moving the mouse to the right end of the drawn line. The moment you see a small red circle appears, press the left mouse button again and then move the mouse to the right again. Ensure that dx = dy = 150 cm and 0 cm. Then release the left mouse button and click on the icon  to the drawing function off.

To optimally visualize of the drawn axes, use the icon  of the toolbar. At this moment your model should look like this:



If the information in your model window is incomplete, use the icon from the toolbar and check if the missing information in the following dialogue (eg, the length of bars):

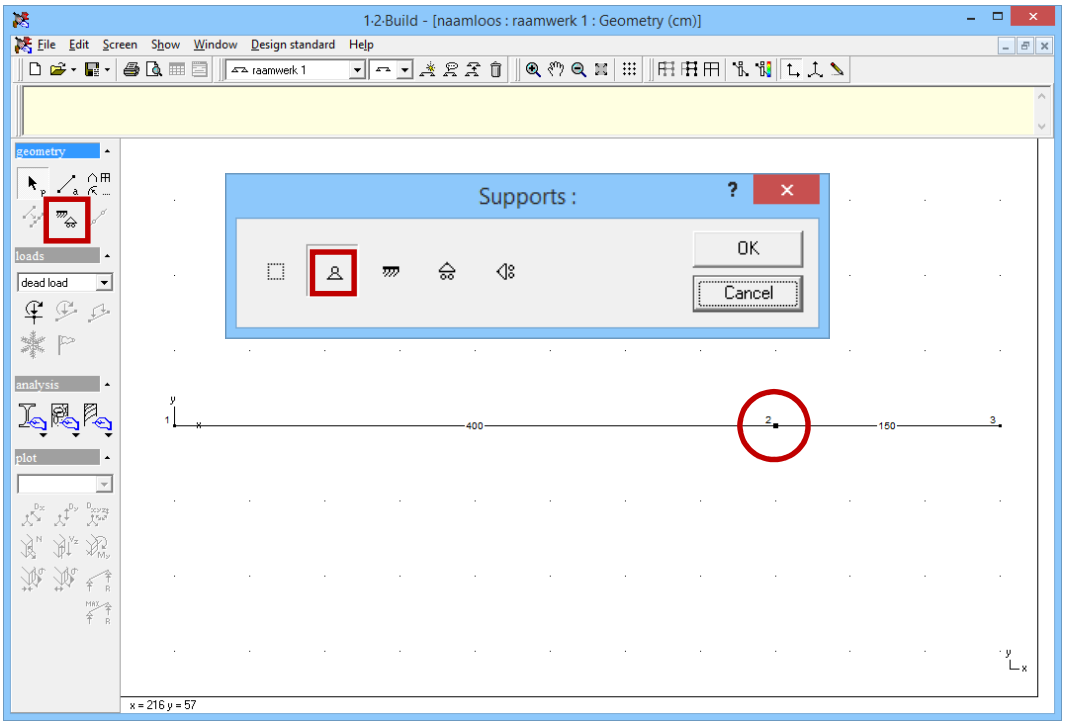


2.2 Defining support points

Before proceeding, verify that the field geometry within the palette icon is still active. To define the support, follow one of these methods:

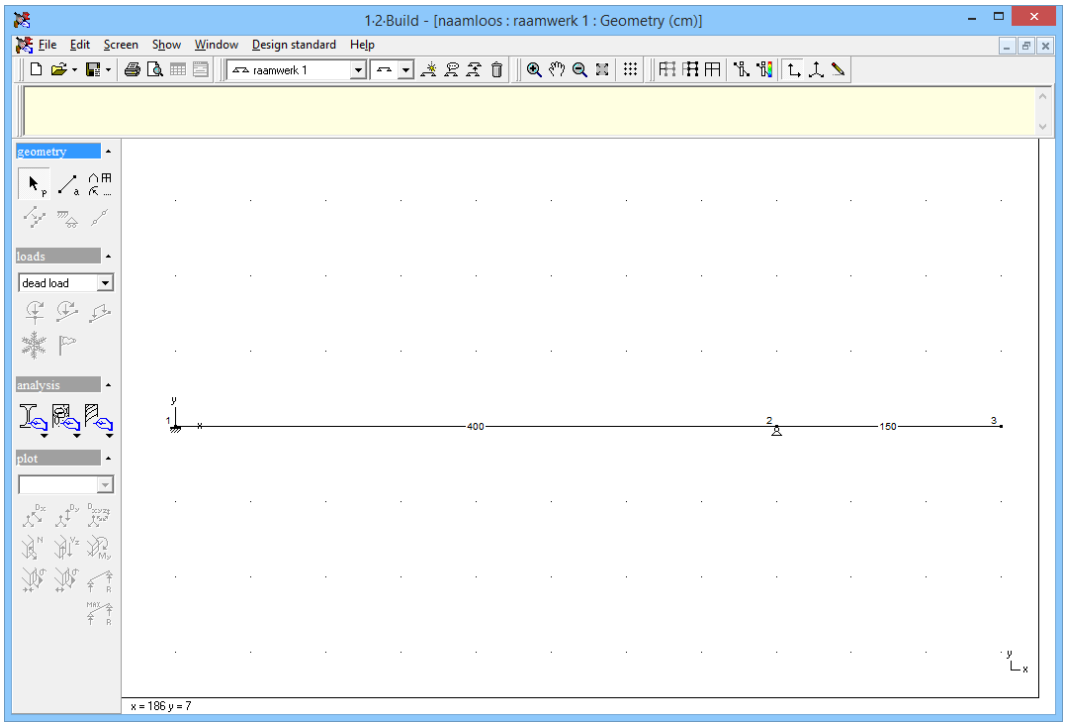
- When you move the mouse near point 2, a small red circle will appear around point 2. Now click once with the left mouse button to select the point. It is now indicated in bold on the screen.
- Select point 2 using a selection window. Click once (with the left mouse button) on the left top side of point 2. With the left mouse button pressed in, move the mouse to the bottom right side of point 2. A rectangle in dashed lines will appear in the model window. Release the left mouse button and point 2 will be selected.

When point 2 is selected, the icon  within the palette becomes active. Click on this icon to the define a simple support for point 2 using the dialog box that appears:



Select the icon  for a simple support and click OK or press the ENTER key on your keyboard to close the dialog box. Node 2 is now pivotally supported.

Select point 1 the same way, open the dialog for the support again and select a moment restrained support using the icon  . The model should look like this:



Note: 1•2•Build is based on the principle that something has to be selected before you can determine what should happen with that selection.

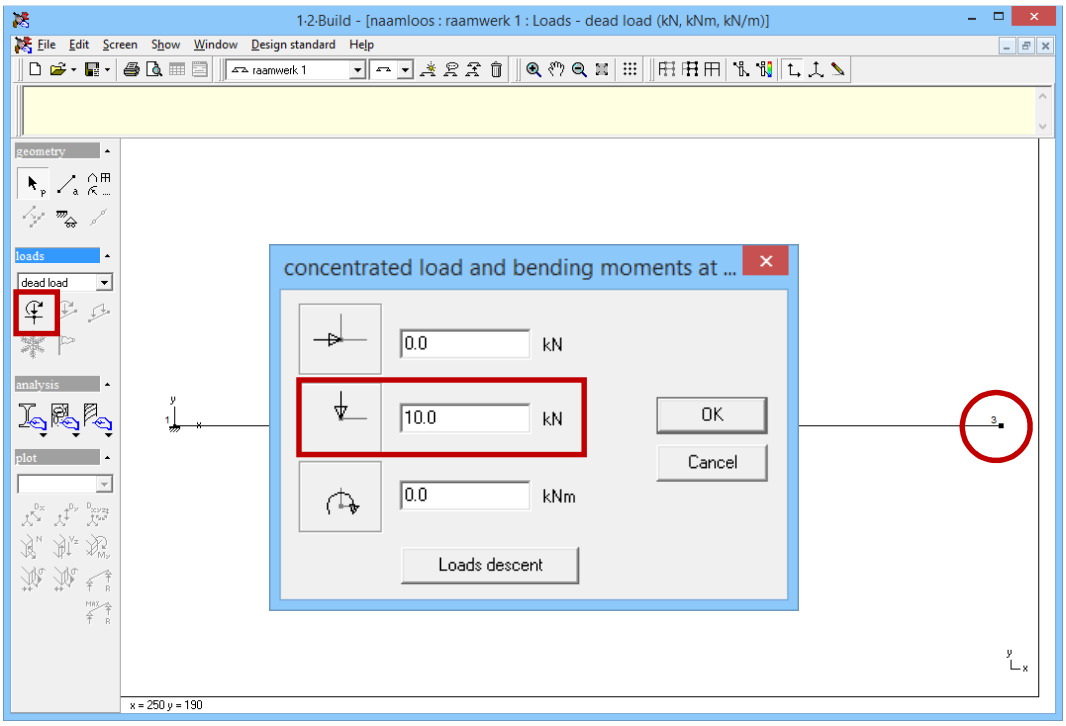
For example, to define the support, the points have to selected first and after that you can define the supports. Only after the points have been selected, there is access to the icon necessary for defining the supports.

2.3 The introduction of loads

In 1•2•Build you can define point loads and uniformly distributed. To make this possible, you must first make sure that the icon field loads within the palette active by clicking on it. Note that below loads, dead load is indicated. All loads you define, will be considered dead loads and automatically with appropriate safety factors be taken into account.

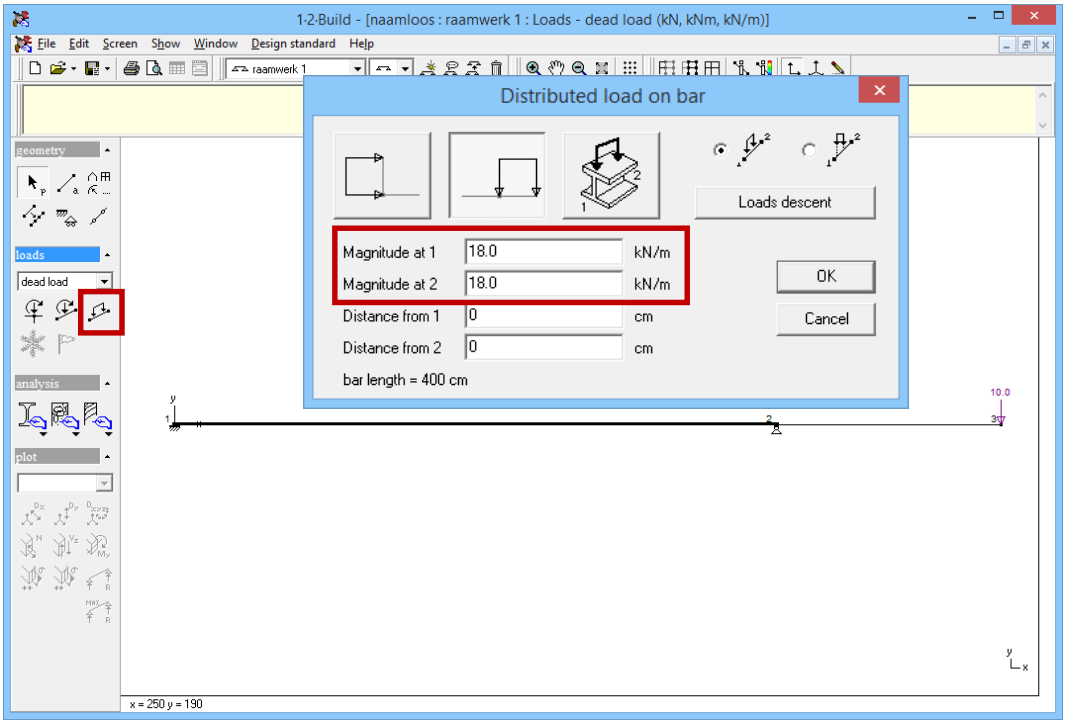
Now select the node 3 using the previously described manner (you might need to set the node numbers visible using the icon from the toolbar button).

The icon opens the following dialog allowing you to define a vertical load of 10kN. Confirm it with ‘OK’.



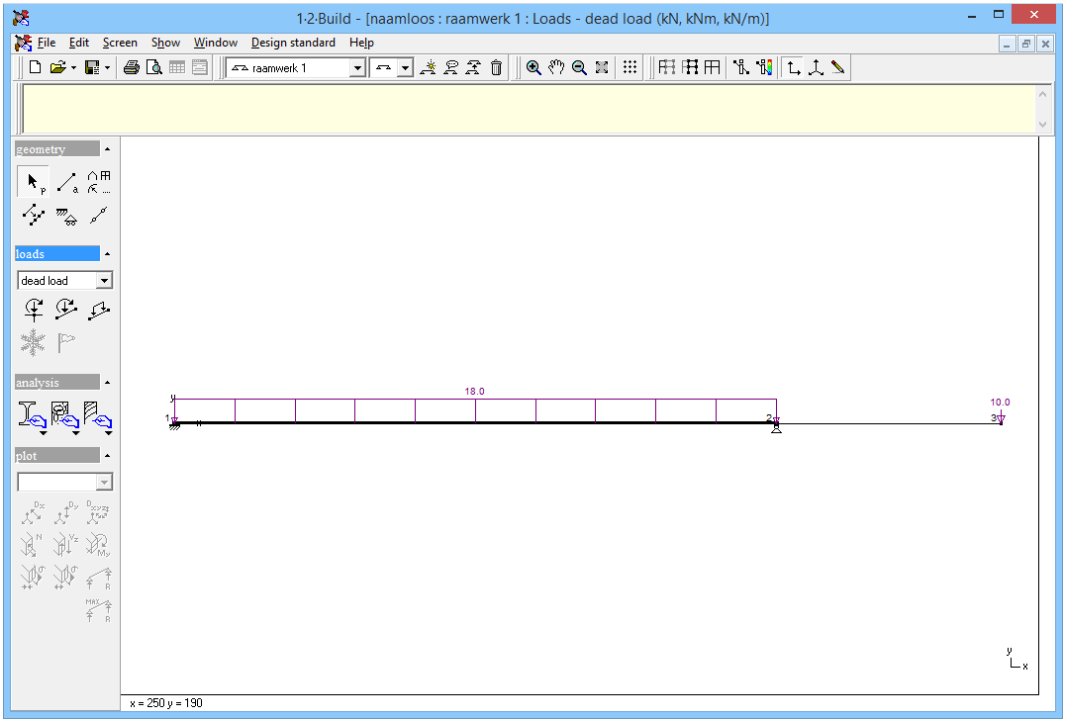
This vertical load point can also be defined on the ‘descent of loads’ function, (NOT available in the demo version), but this is described in §2.9.

Select the entire span between the nodes 1 and 2 using a selection window (from top left to bottom right). With the icon apply a distributed load of 18kN/m over the selected span:



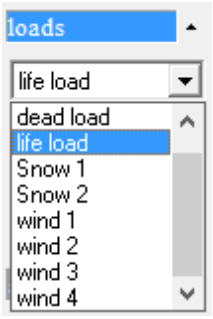
Again, you can use the ‘descent of loads’ function to define this load.

You now have the following permanent loads defined on the continuous beam:

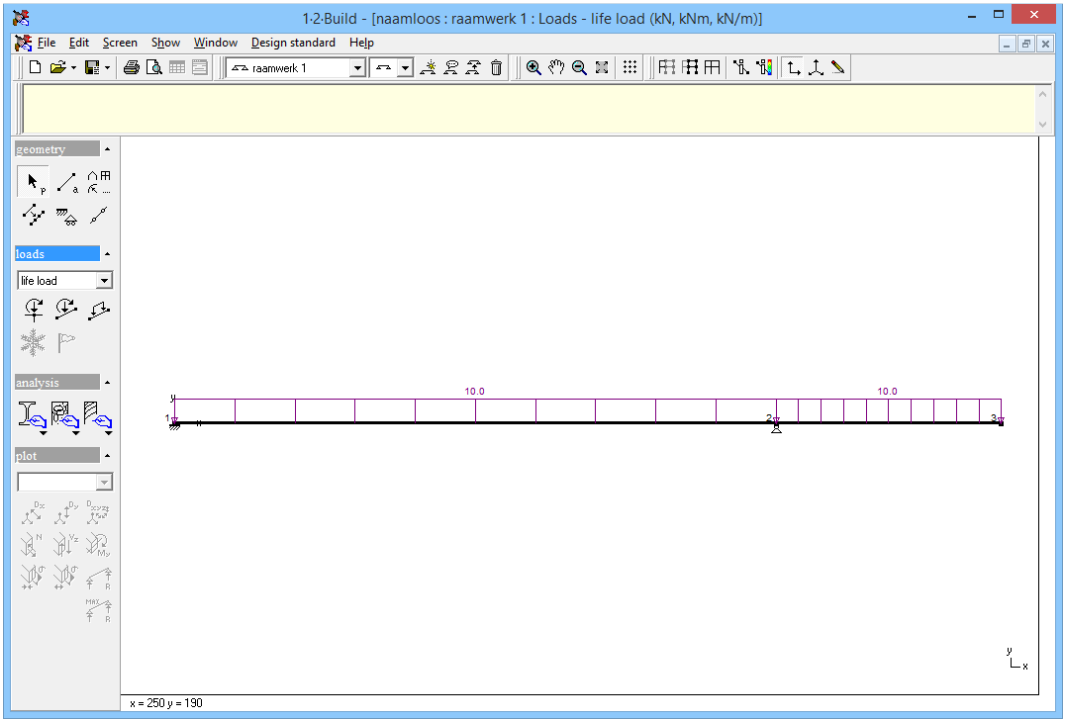


Did you make a mistake when entering the loads? Don't panic, select the point or the span for which you want to correct the load. With the DELETE key on your keyboard, you can remove the corresponding loads, to then give the correct definition.

Finally, we indicate which life loads is acting on the beam. Just click with the left mouse button on the arrow in the field  , which pull out the following list:



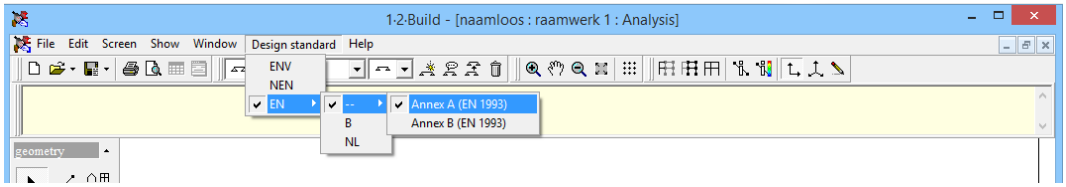
Move the mouse to 'live loads', and select it by pressing the left mouse button. Now select the entire beam and define a uniformly distributed load of 10kN/m:

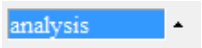


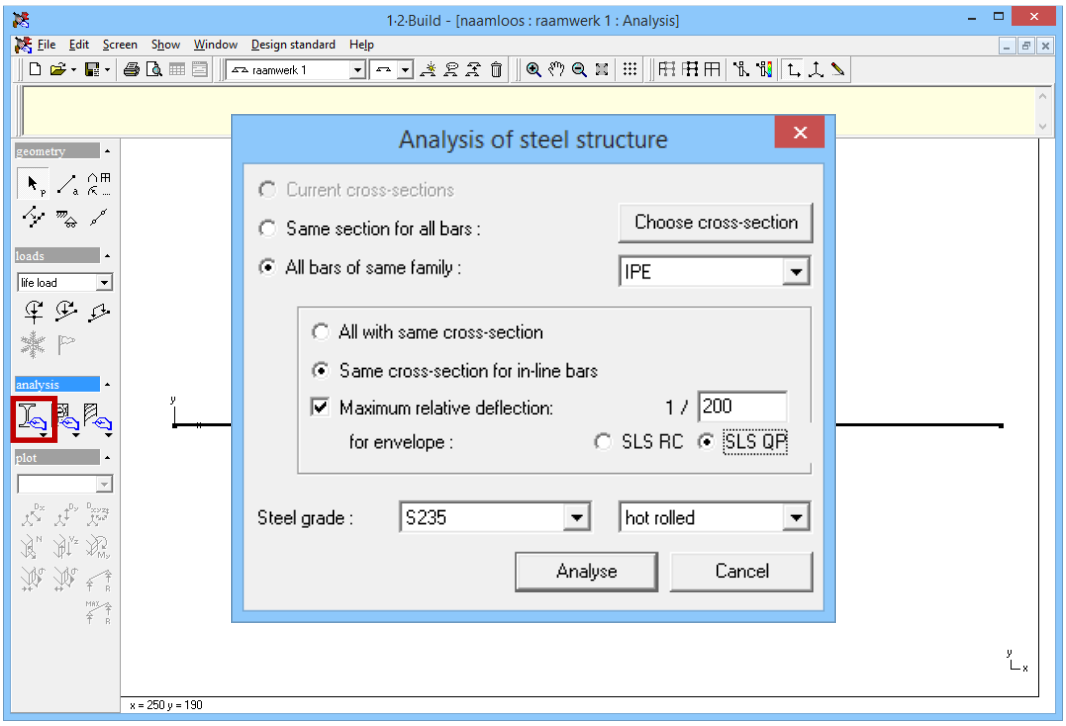
There you go. Dead and live loads are defined, while the weight of the beam automatically will be calculated as a function of the cross-section provided by **1•2•Build**. Based on these loads, **1•2•Build** will generate all the combinations according to the Eurocodes based on these loads, to safely design this continuous beam.

2.4 Design of a steel beam

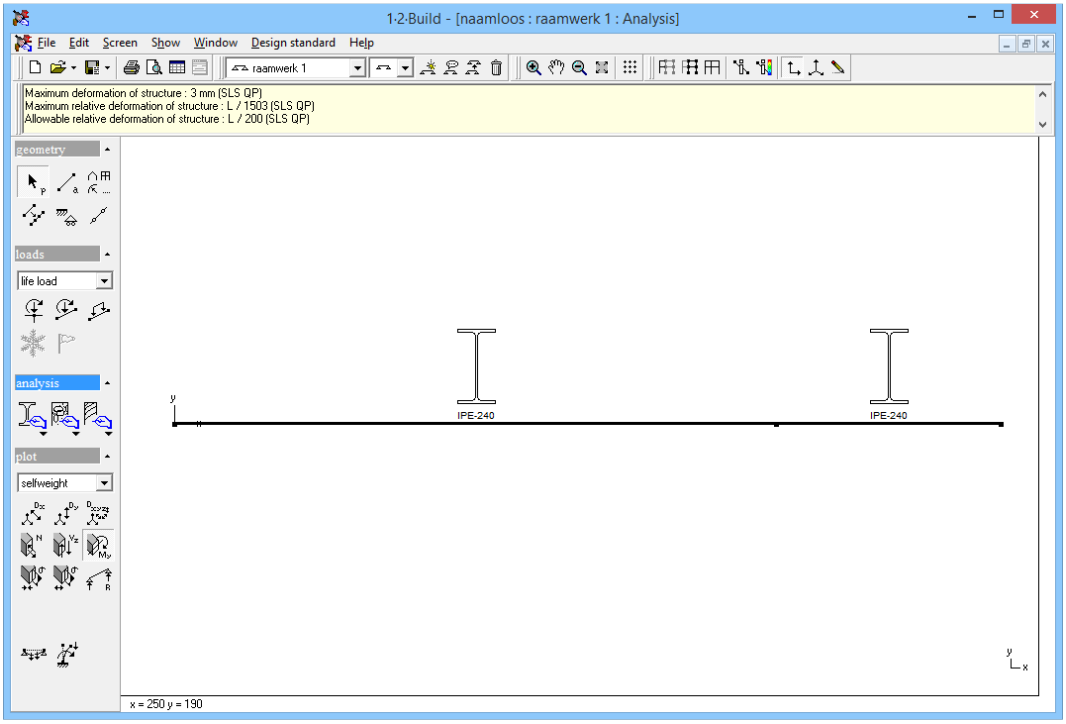
Before starting the design, we select the standard according to which we'd like to perform the calculations:



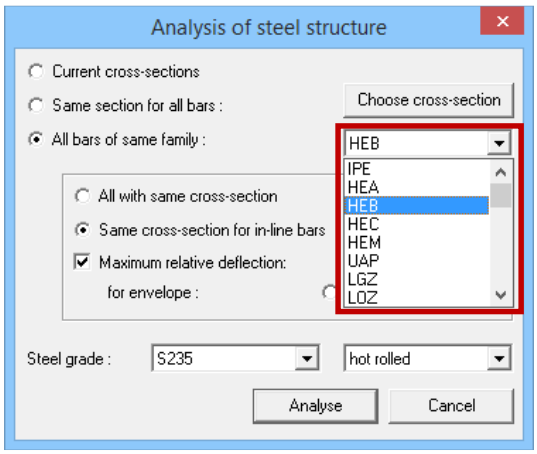
The most economical dimensions of a steel beam will be readily determined by the icon  from the  palette. Following dialog appears:



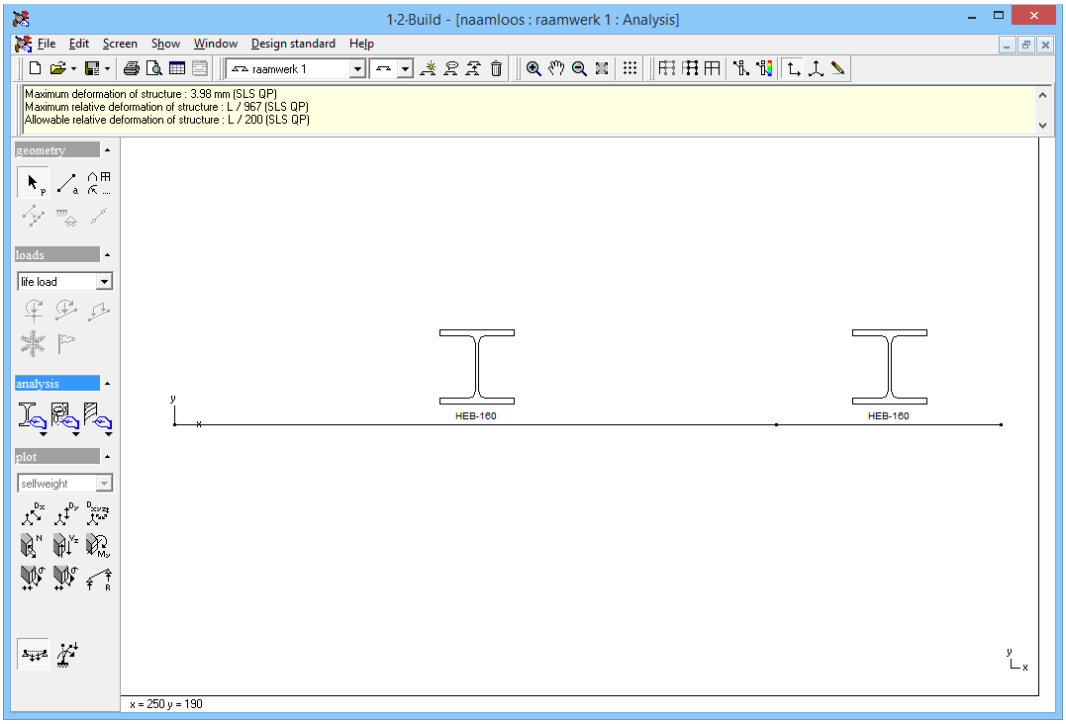
By pressing the Analyse **,** you give **1•2•Build** the command to search the most optimal IPE - profile that ensures an adequate resistance with respect to the applied loads (including its own weight). Moreover, the profile satisfies the requirement deformation limit in the serviceability limit state of your choice. For this example a service limit state quasi permanent combinations was chosen. After several iterations represents **1•2•Build** you a IPE240.



If you'd like a solution with a smaller height, then click again on the icon . Open in the dialog that appears next to the profile list of available classes and select, for example the HEB-class.



1•2•Build will propose you a HEB160.

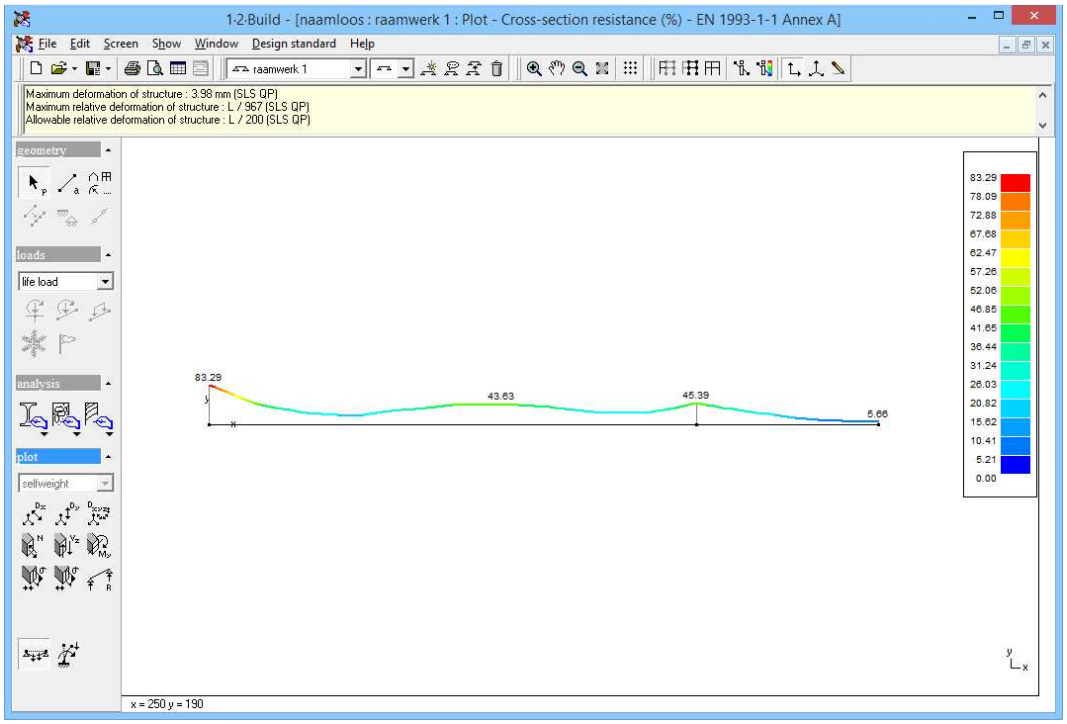


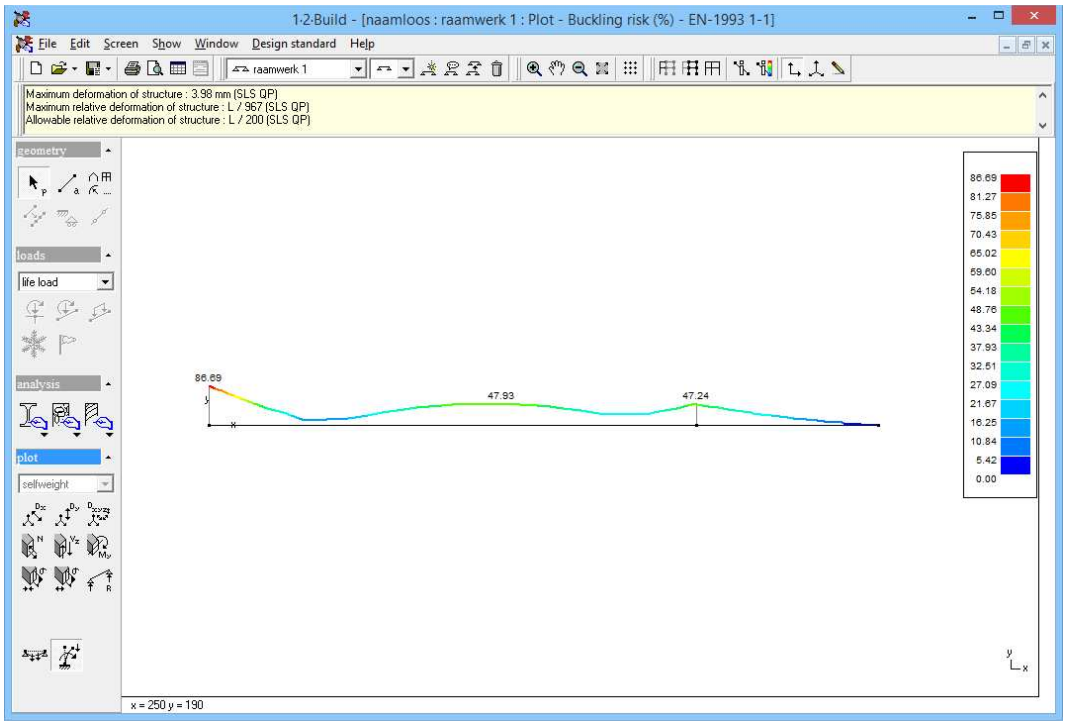
The proposed solution fully complies with the requirements set by Eurocode 3 with respect to

- the resistance of the profiles used
- the stability of the bars subjected to pressure and / or bending (articulated or chicken-stability).

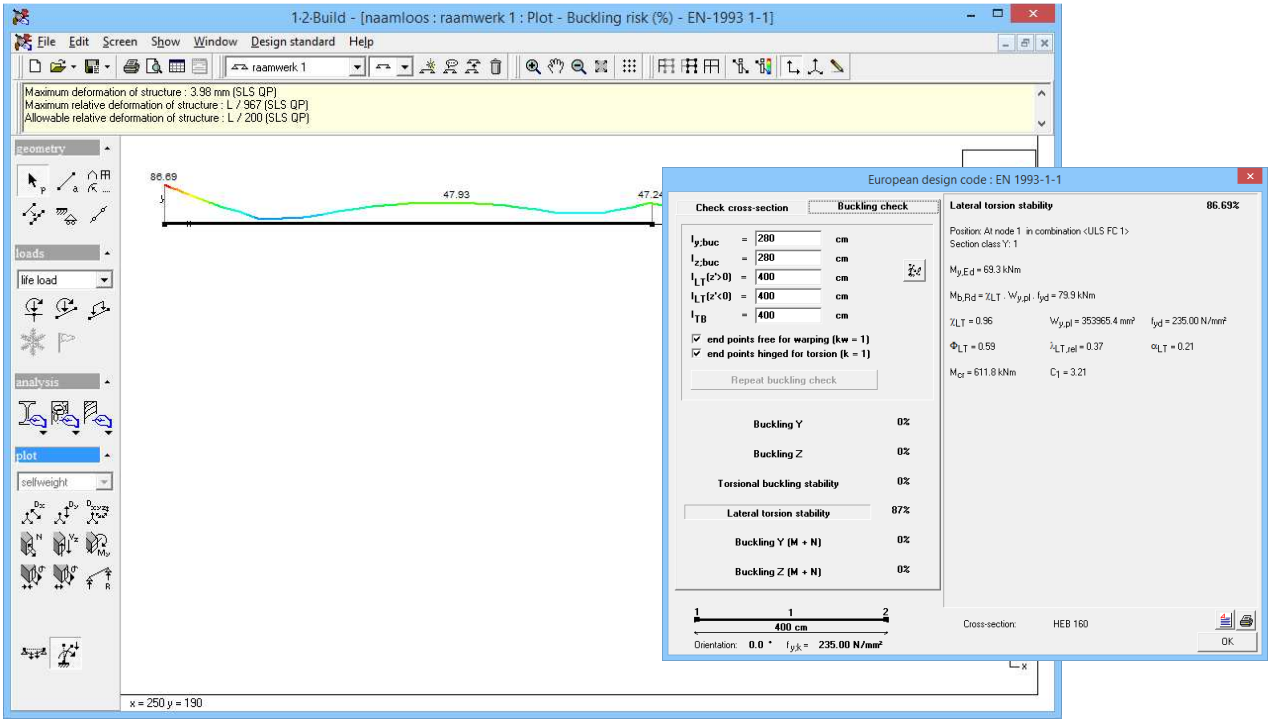
This may be documented more in detail using the icons and from the field plot. The following graphs show that both the resistance of the

sections as the stability of the rods below the 100% of the permitted values remain.

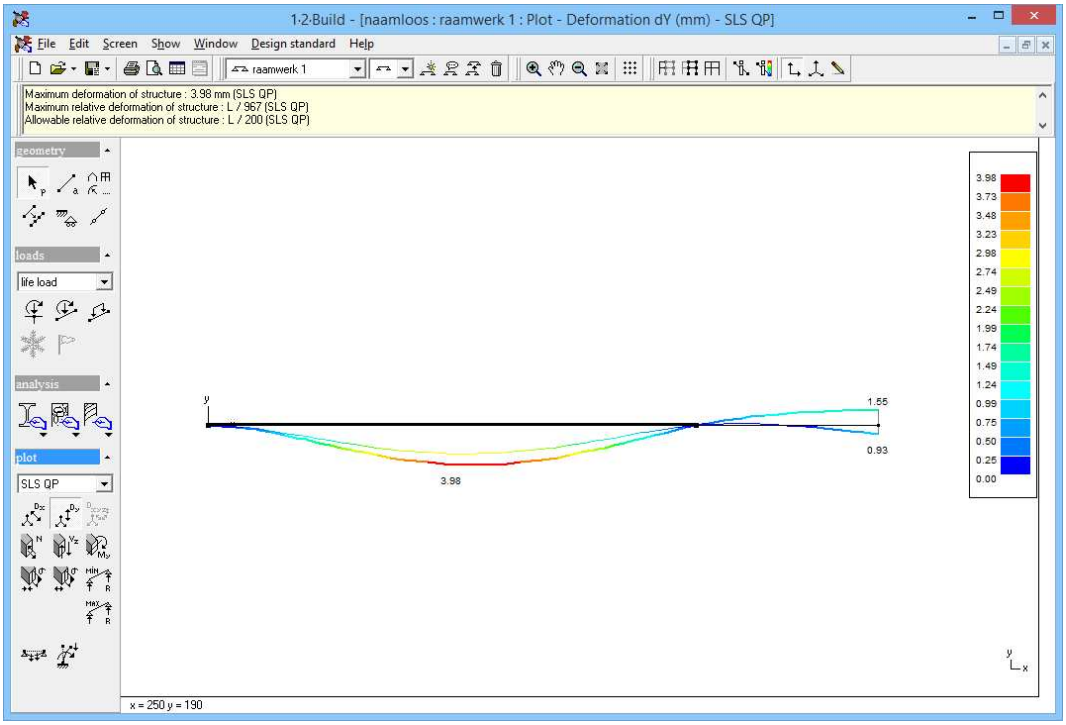




Double click one of the beams to see the details of the steel check.



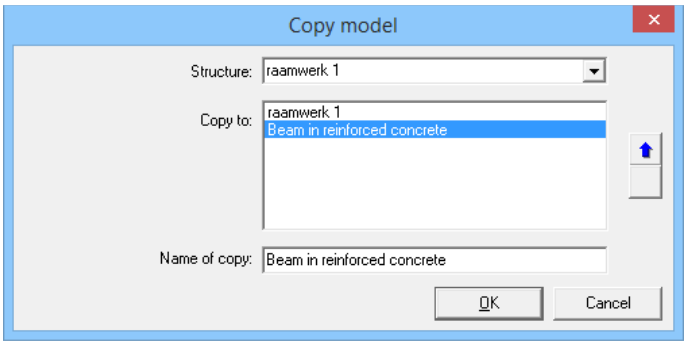
Furthermore, you can request more results in the icon field plot, in particular the deflections of the beam in the service limit state. Do this, select the icon D_y for the load combination SLS QP in order to obtain the following results. Which shows us that the expected deflection in the proposed HEB160 profile are 4 mm:



2.5 Design of a beam in reinforced concrete

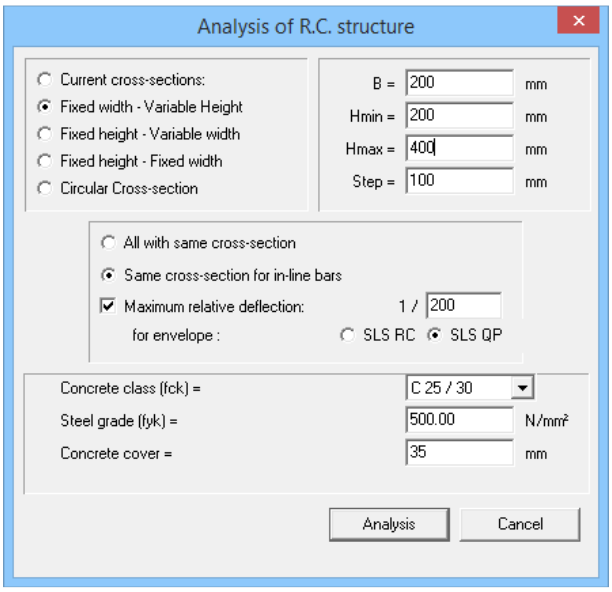
Then we wish to answer the question how the same beam can be designed in reinforced concrete. That question can best be answered by duplicating the calculated steel beam within the active **1•2•Build** project.

Use the icon in the toolbar. This enables you the following dialogue:



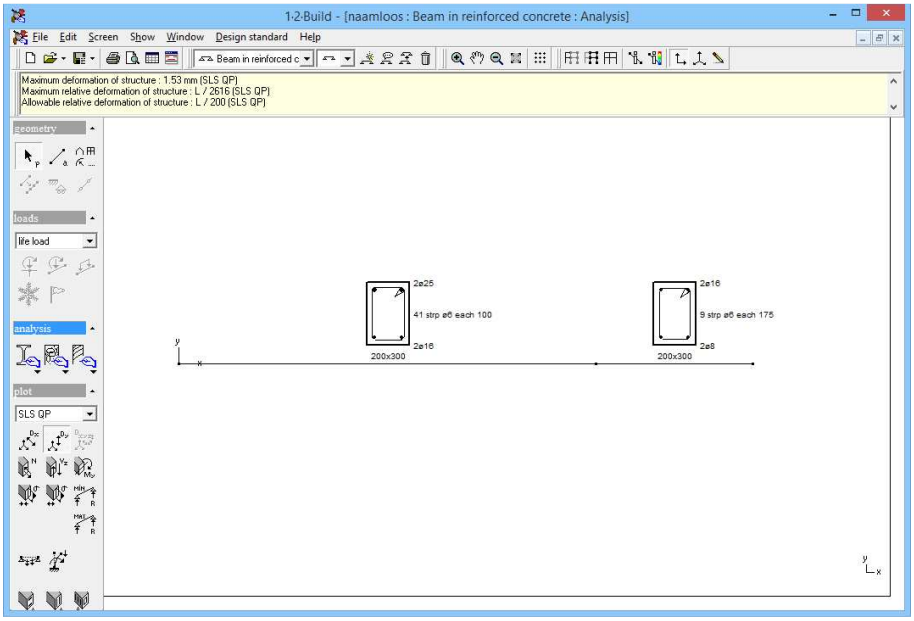
Enter an appropriate name for the copied structure (eg **Beam in reinforced concrete)** and confirm by pressing the 'OK'. You will notice that geometry, loads and profile choice were copied to a new mathematical model with the chosen name.

The optimal dimensions of this beam in reinforced concrete, can quickly be determine by the icon from the analysis palette. If you choose a section with **a fixed width of 200 mm,** but with a variable height, than you should define the width together with the upper and lower limits for the height:



In the next field you define the maximum relative deflection in SLS QP. **1•2•Build** keeps takes this limitation into account when designing. If the current size doesn't pass, **1•2•Build** will show a warning.

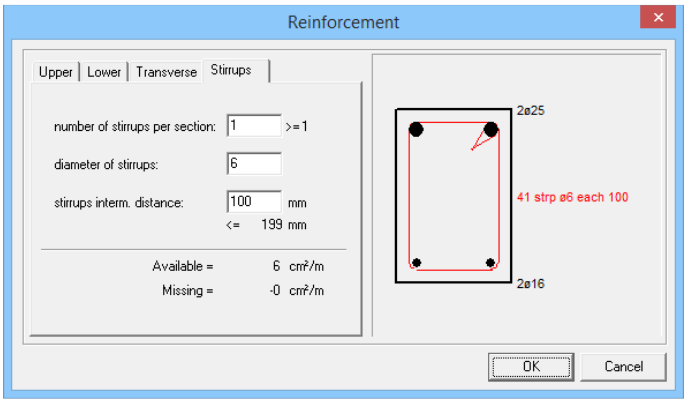
In addition you must define the concrete strength (C25/30), together with the steel quality ($f_{yk} = 500N/mm^2$). With the button Analyse you start the calculation process.



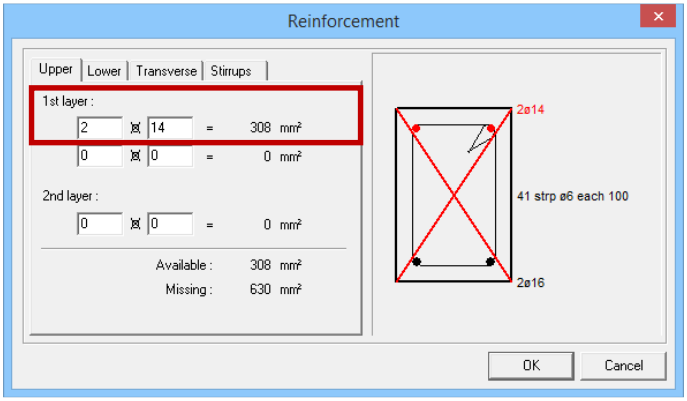
1•2•Build hasn't only calculated the optimal height of the concrete beam, but he has also included a proposal for the longitudinal and transverse reinforcement. You have the assurance that this proposal conforms to the standard EUROCODE 2 on

- the ultimate limit states
- the service limit states (limiting the stresses in steel and concrete)

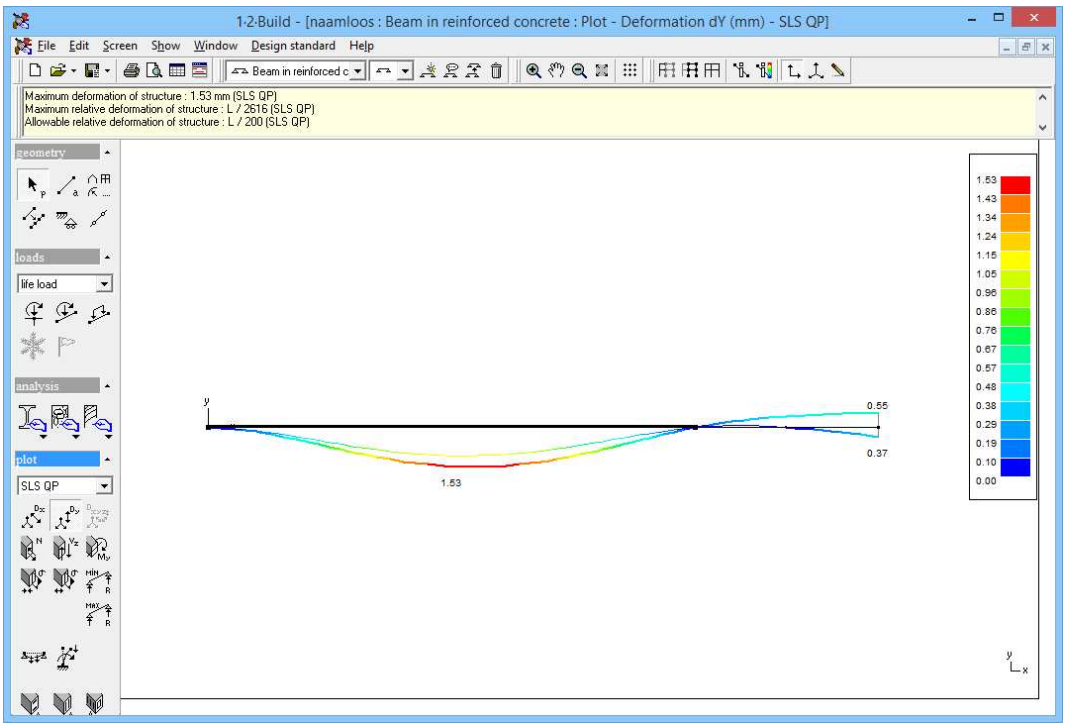
By double clicking one of the beams in the window above one of the beams, you get access to detailed information about the proposed reinforcement:



Here you can adjust the reinforcement details if you wish, both for top and bottom reinforcement as for the stirrups. If, however, you'd specify a practical reinforcement which does not meet the requirements of the calculation (for example, 2 diameter 14 instead of 2 diameter 25), this is still clearly indicated:



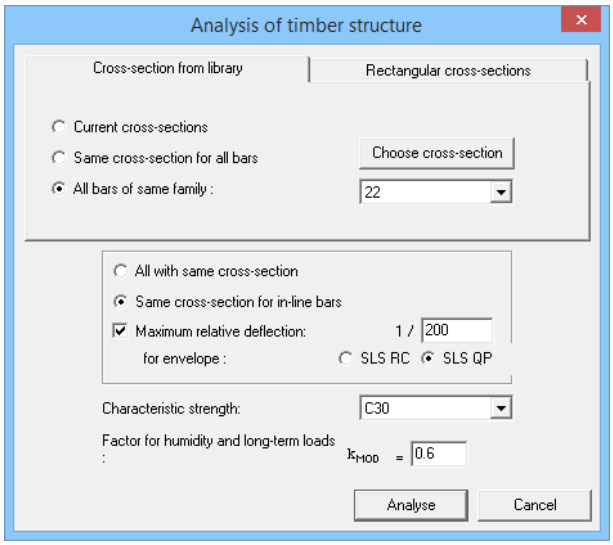
The maximum deflection of the structure is 1.5mm. The deflection is equal to L/2616, ie the structure complies with the requirement of L/200. This deflection is calculated based on the quasi-permanent load combination. It should be noted that **in 1•2•Build elastic deflections are calculated not taking into account any cracks nor creep! The actual deflections can be a factor 2 to 5 higher than the proposed elastic deflections.**



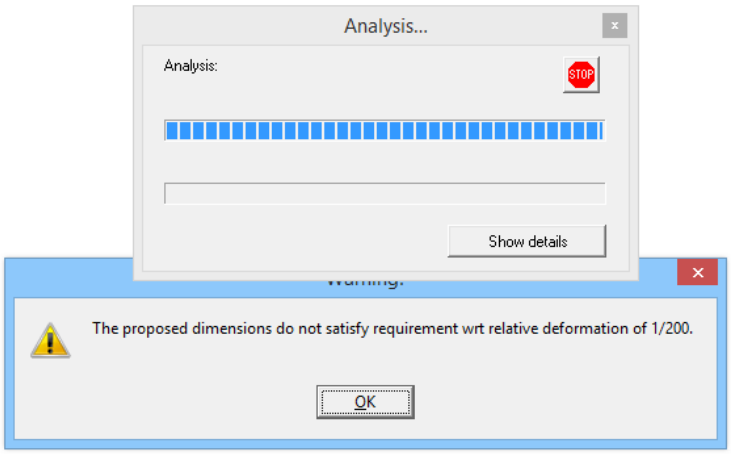
2.6 Design of a timber beam

Again, use the icon in the toolbar to create a duplicate of the existing concrete beam with the name **wooden beam.**

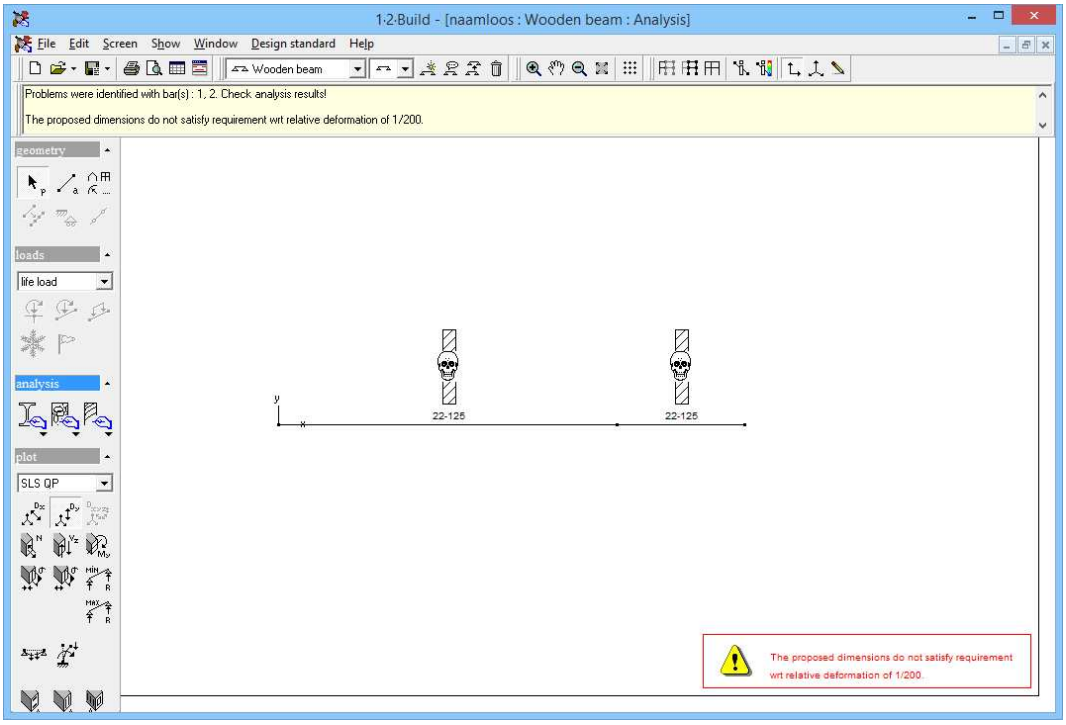
The optimal sizing of the wooden beam can be started by clicking on the icon from the analysis palette field. Just as for the concrete beam, you can define the section in terms of its width and height. Moreover you can also pick sections from a profile library and determine the most optimal section within a particular class. This requires of course that you specify in the appropriate strength class. We choose for example the profile group 22 and set the maximum deflection again equal to 1/200.



1•2•Build during the calculation gives the following warning:



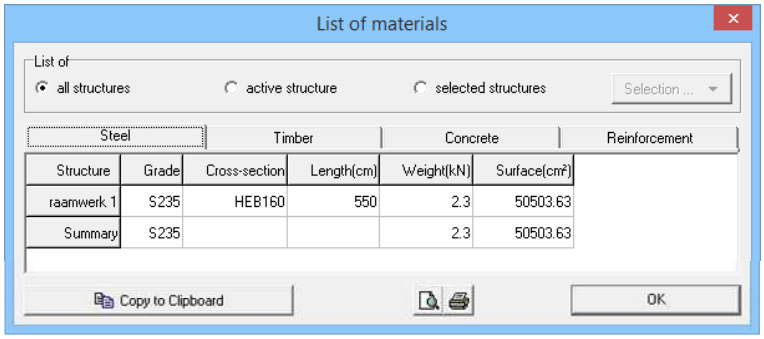
The cross-section of the profile is too small to meet the requirement of relative deflection. At the right bottom of **1•2•Build** a warning sign is shown. The sections are accompanied by a skull character because the requirements for the resistance and buckling check were not met.



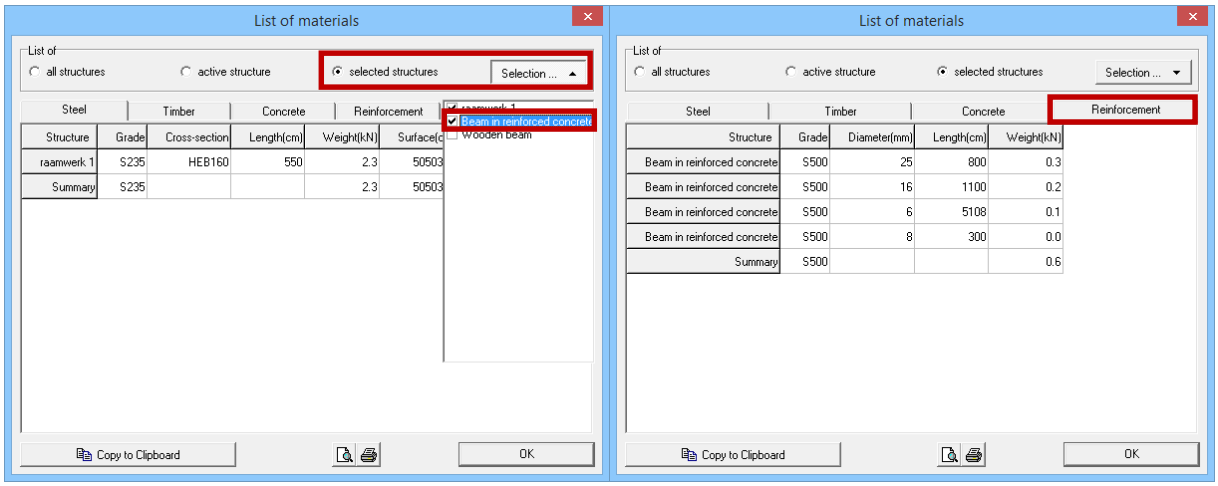
The calculation must be redone with a heavier profile.

2.7 Estimation of the materials

1•2•Build estimate the materials for you. The estimation gives an overview of all used materials in the calculation models of the project. Click on the icon in the icon bar.



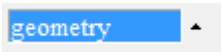
Click on the icon Selection ... that lights up after checking **selected structures** and select the beam in steel and reinforced concrete. Below is a summary of the reinforcing steel placed in the reinforced concrete beam.



2.8 Changes to a beam

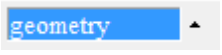
All the details of a mathematical model (for example, the length of a beam, the position and properties of a focal point, the location and the value of a load, ...) can be changed at any time.

2.8.1 Removal of a beam element

Select the field 

. Then select the beam elements that you want to delete and use the DELETE key to remove the selected beams.

2.8.2 Changes to the beam

Select the field 

. By double-clicking a beam element, a dialog box appears in which the actual length of the beam is reported. You can change this value by simply editing this field. Upon confirmation, the beam element will be adjusted by moving one the ends, namely the end that is the farthest away from the point where you clicked the beam element.

Would you want to adapt a point, then you just double-click the corresponding point. In the dialog that appears, adjust the coordinates of this point and the adjacent beam elements will follow automatically.

An alternative way is to select the point with the mouse and to move it over the drawing paper.

2.8.3 Removal of loads on a beam element

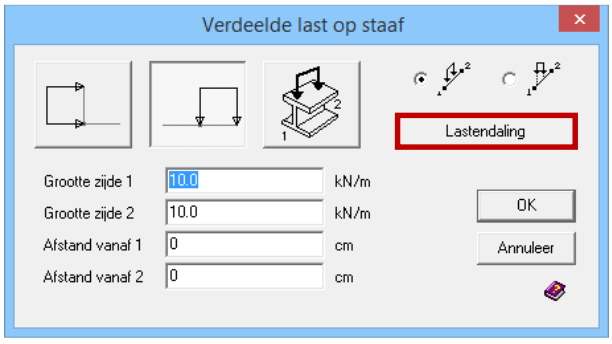
Select the field loads in the palette and then select the last group (eg dead load or live load) within which you want to remove the loads. Now select the beam elements for which you want to remove loads and use the DELETE key to remove all loads on the selected beams in the active load group.

2.9 Using function ‘Descent of loads’

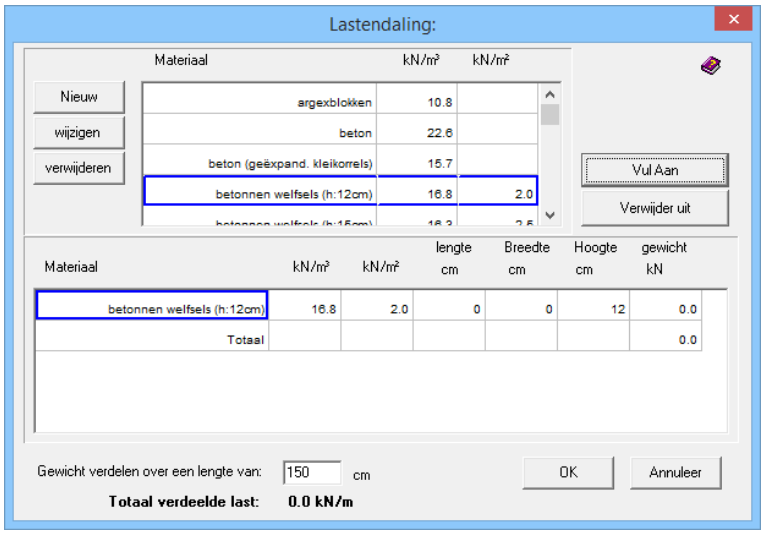
Preliminary: This section does not apply for demo licenses.

Point charges and evenly distributed loads can be easily applied to the calculation by explicitly determine their position and size. Your ease of use can, however, be increased considerably by making use of the integrated load reduction function.

Suppose you have a uniformly distributed load is defined in terms of the load drop, then you in the following dialog box directly to the button Lastendaling .



That way you will get access to the following dialogue, which you get for a material selected from the provided list. With the aid of the button Vul Aan You add the selected materials to the last drop (in the example below: concrete vault of 12 cm).



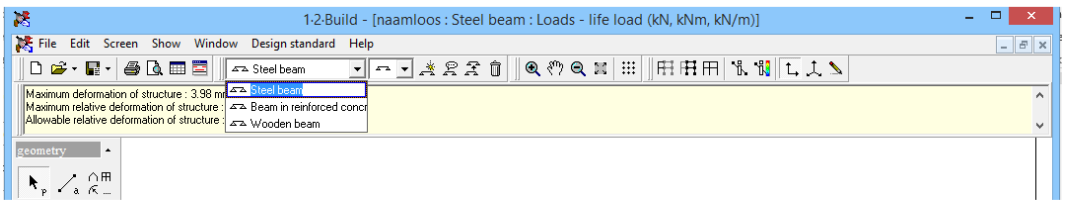
Then you define length and width of the material (read: concrete vault) whose weight is borne by the bar that you previously selected. Based on these data, **1•2•Build** the total weight of the material calculate and draw the distributed load on beam distracted.

Would you continue the weight of a screed and tiles to charge, then you select the materials one by one from the list and add them using the button to decrease the burden. After completion of the missing data (length, width and height) will **1•2•Build** the distributed loads recalculate and adjust.

Such load reduction function is also available if you wish to apply concentrated loads on the model.

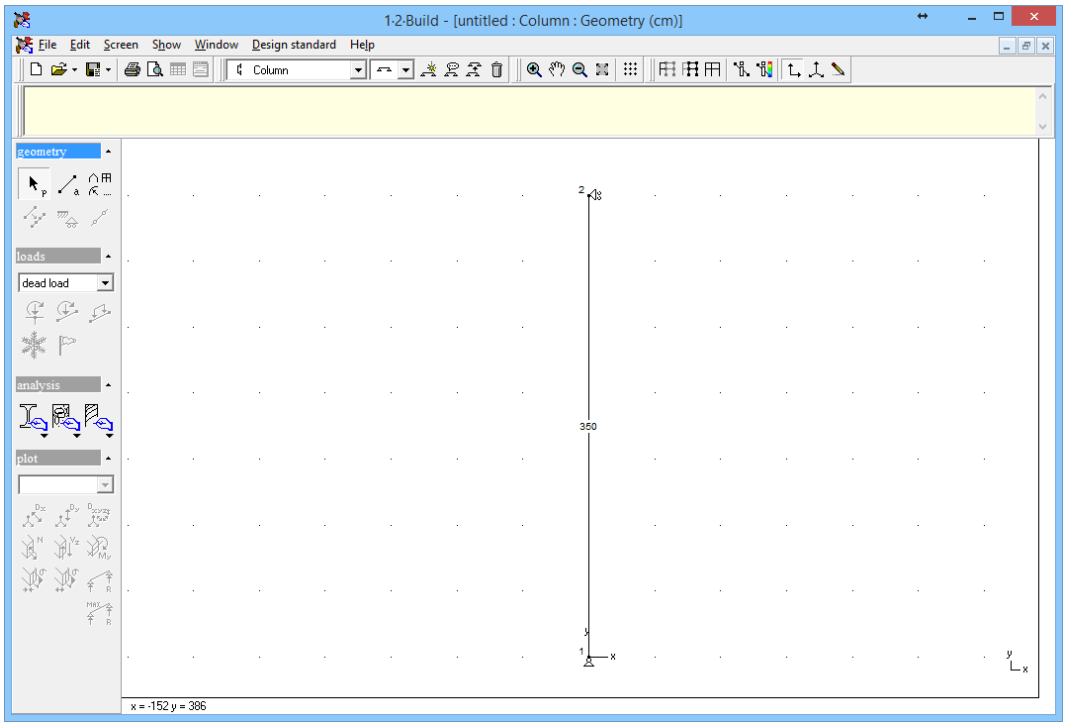
2.10 Managing projects

With the aid of the icon  from the toolbar you can save the results of your work in a project file on your hard disk. A project file contains all calculation models that you previously defined (steel beam, reinforced concrete beam, wooden beam) in this project. Will you continue in the current project, then you can always switch from one calculation model to another, by open the pull down list in the toolbar and selecting the new active calculation model:



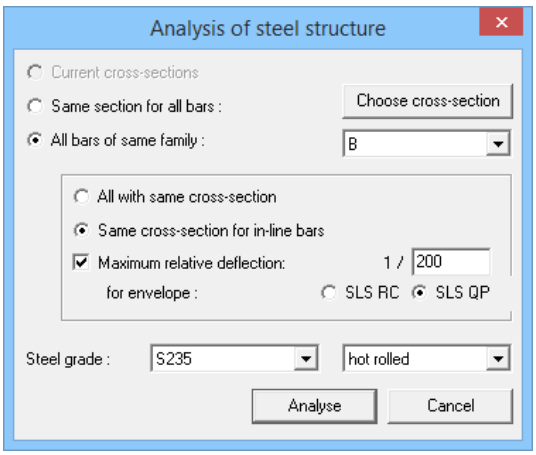
3 Design of a column

Suppose you wish to design a column next to the already calculated beams within the active project. First add a new computational model to the active project using the icon in the toolbar. In the new window that appears, then you draw a vertical rod with a length of 3.5 meters and define the supports as shown in the figure below:

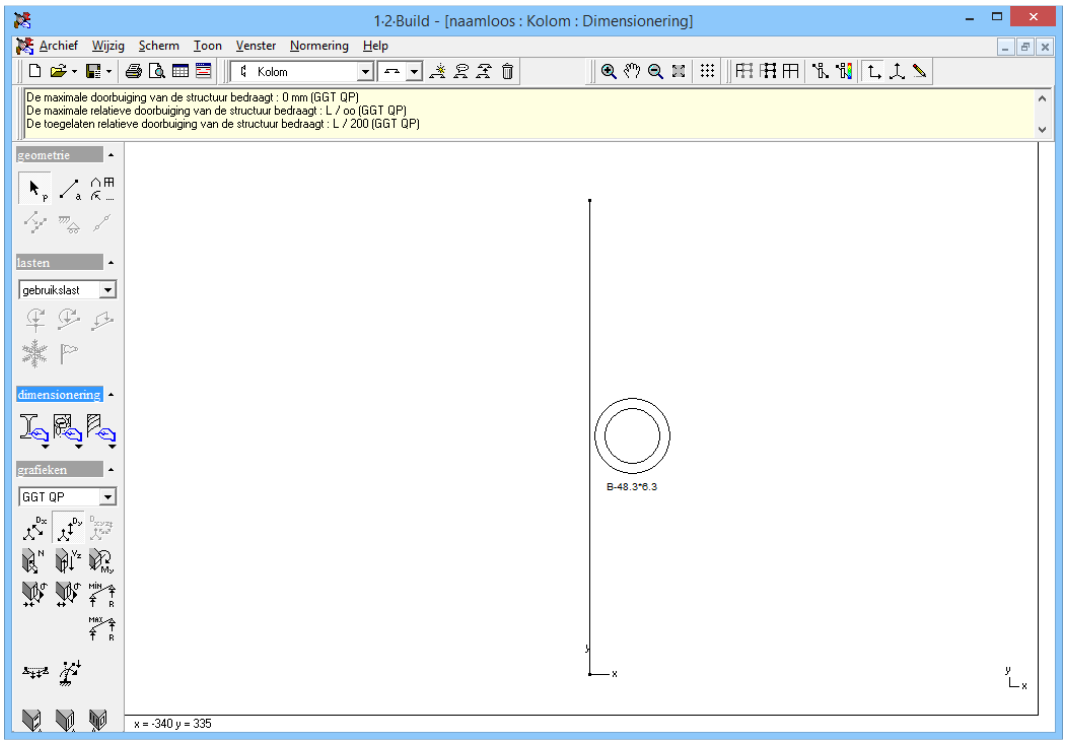


Add a dead load of 10kN and a life load of 10kN in point 2.

Suppose you want to construct this column with a steel tube profile, then you determine the most optimal section within the profile class B:



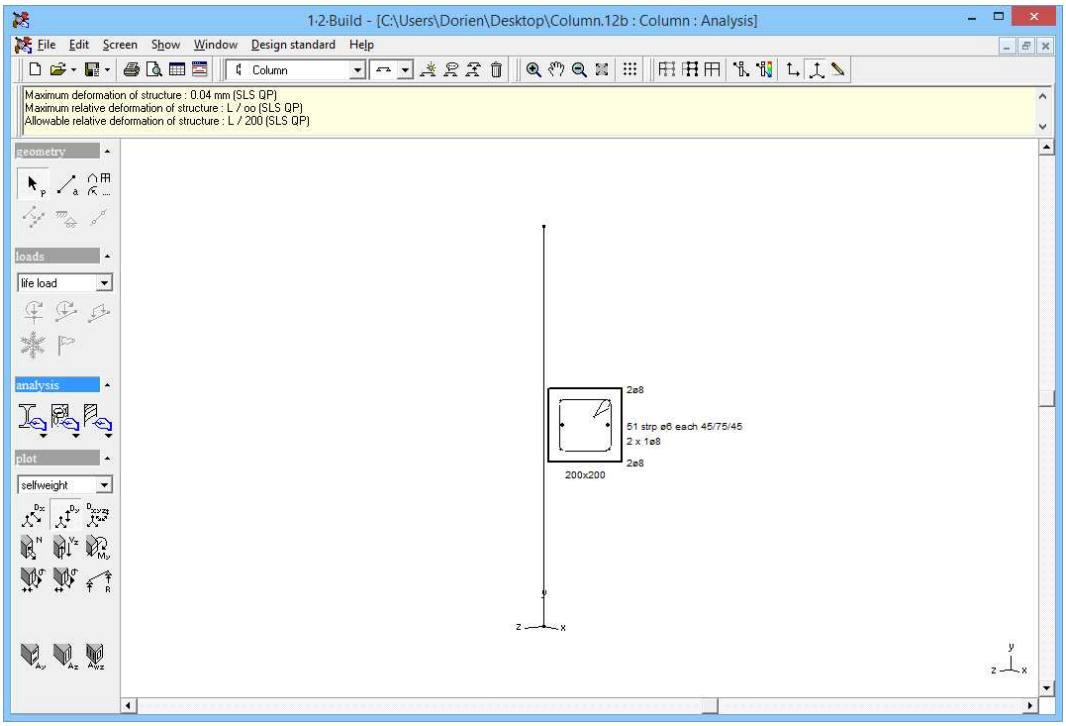
1•2•Build solution will give you following proposal:



In particular a profile tube with a diameter of 48,3mm and a thickness of 6,3 mm. This solution meets all requirements for resistance of the sections and the buckling stability of the column. If desirable, this can be check with the graphs and .

Note that the maximum deflection is L/∞ amounts. This deflection has no physical meaning.

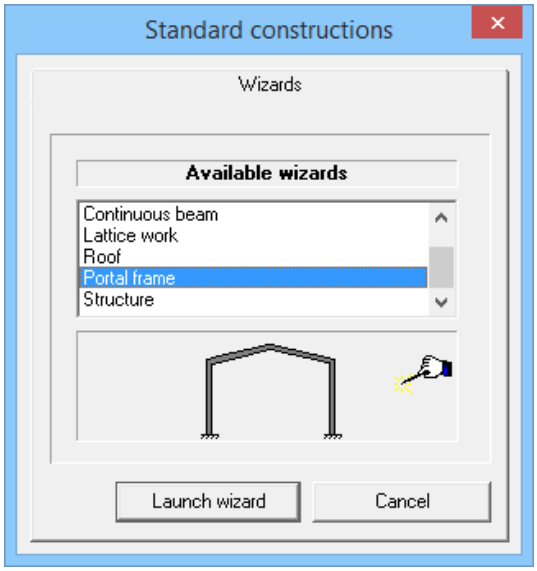
If you want to calculate the same column in reinforced concrete, than **1•2•Build** will propose reinforcement that meets all standard requirements (thereby opted for a section with a fixed width and height of 200 mm):



4 Design of a truss

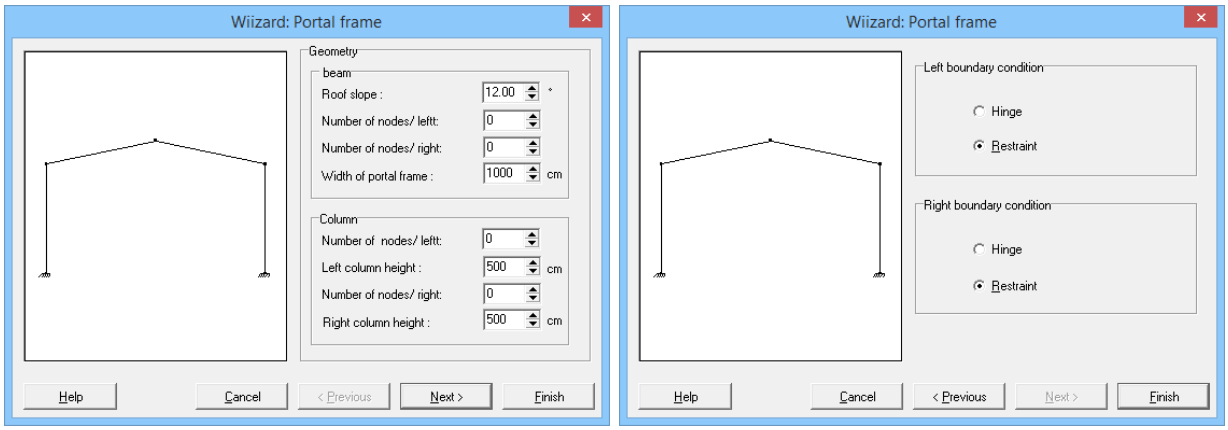
Add another calculation model to the active project, using the icon in the toolbar. In the new window that appears you can now draw the axes of the truss that you want to dimension, or alternatively you can use **Wizard** for the rapid definition of standard type structures. For this, use the icon from the field geometry.

Now select the type of structure ‘Portal frame’ in the dialogue that appears and confirm with Launch wizard:

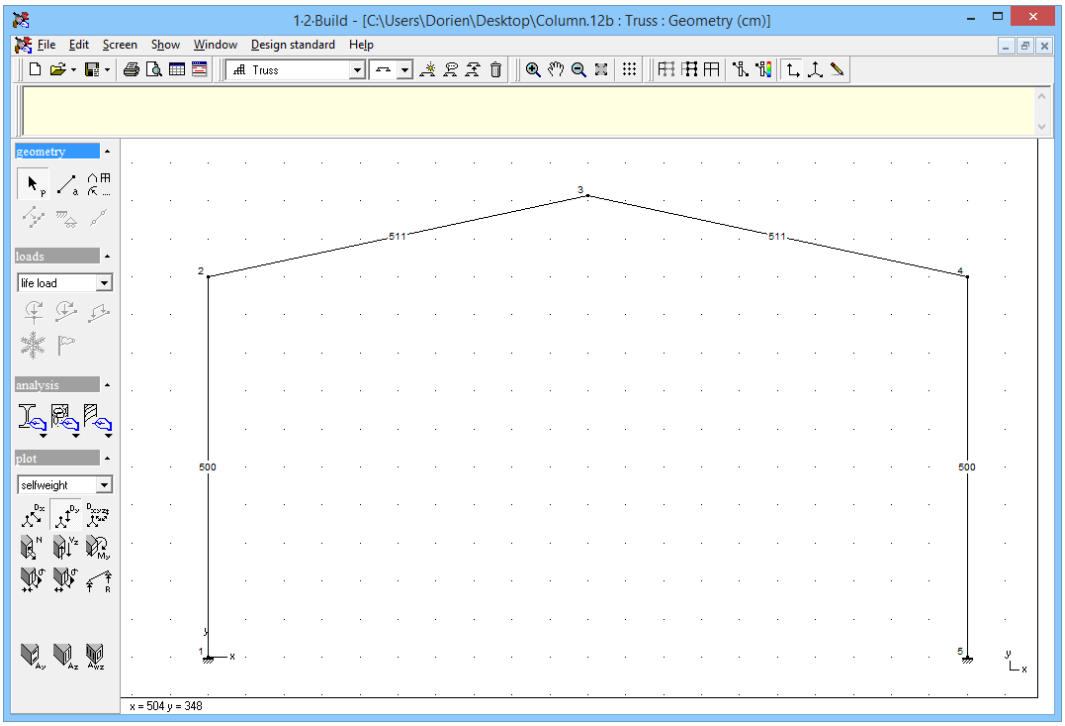


In the dialogue that appears you can change the geometry of the truss quickly on the basis of a limited number of parameters. Accept the default settings for geometry and boundary conditions.

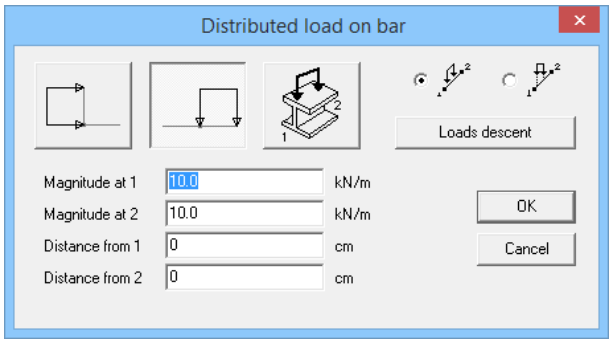
Note that in this dialogue, you can navigate with the buttons < Previous and Next >. To confirm the definition, click on Finish.



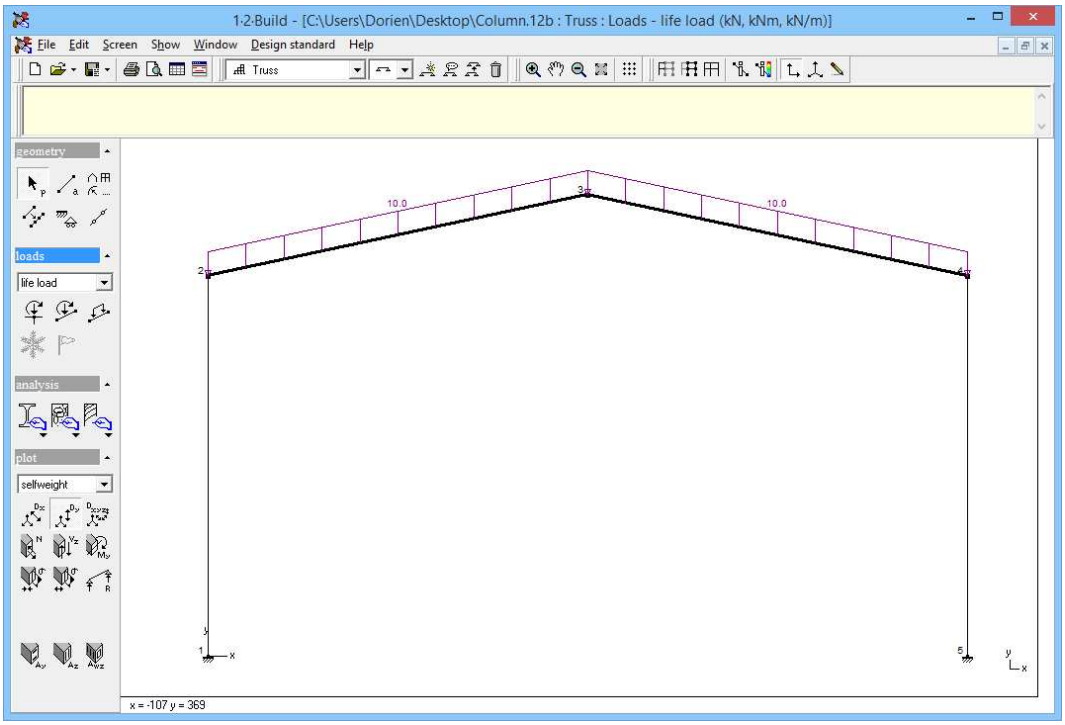
You'll end up with the following definition of the frame:



Select the end of the truss girders and click on the icon . Enter the following load data:

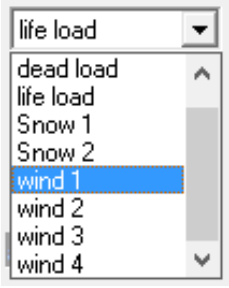


The selection  implies that the entered loads will be projected on the beam. Their value will be recalculated as a function of the angle the beam. Click 'OK' and the following loads will be shown on the beams:



Note that a wrong operation can be made undone using the instruction menu **Edit - Undo ...** or by pressing the keys CTRL+Z.

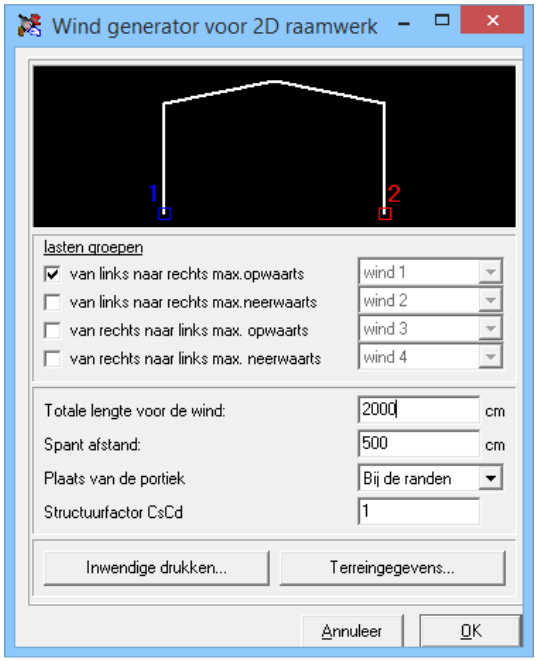
Besides the classic options for defining point loads and distributed loads, **1•2•Build** provides also wind and snow generators. To use the wind generator, select **wind1, wind2, wind3 or wind4** first from the list of possible loads:



Wind1 and wind2 may be used for a wind load from left to right and with and upward respectively downward direction. Wind3 and wind4 can be analogues used to define wind from right to left.

When 1•2•Build combines the loads, it will be automatically taken into account that wind loads can never occur together. The same remark applies also for snow1 and snow 2 when you're using the snow load generator.

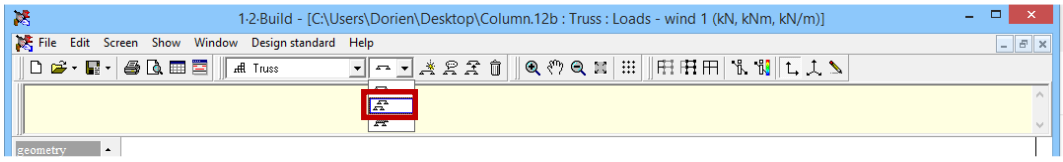
Now select the entire frame and click  :



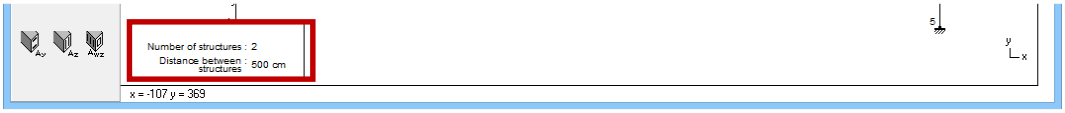
Opt for example for an upward wind load from left to right and define the total length of the structure perpendicular to the wind direction. You should also define the distance between the frames (take 5m). This way 1•2•Build knows how many area will be supported by the frame.

5 Design of a repetitive structure

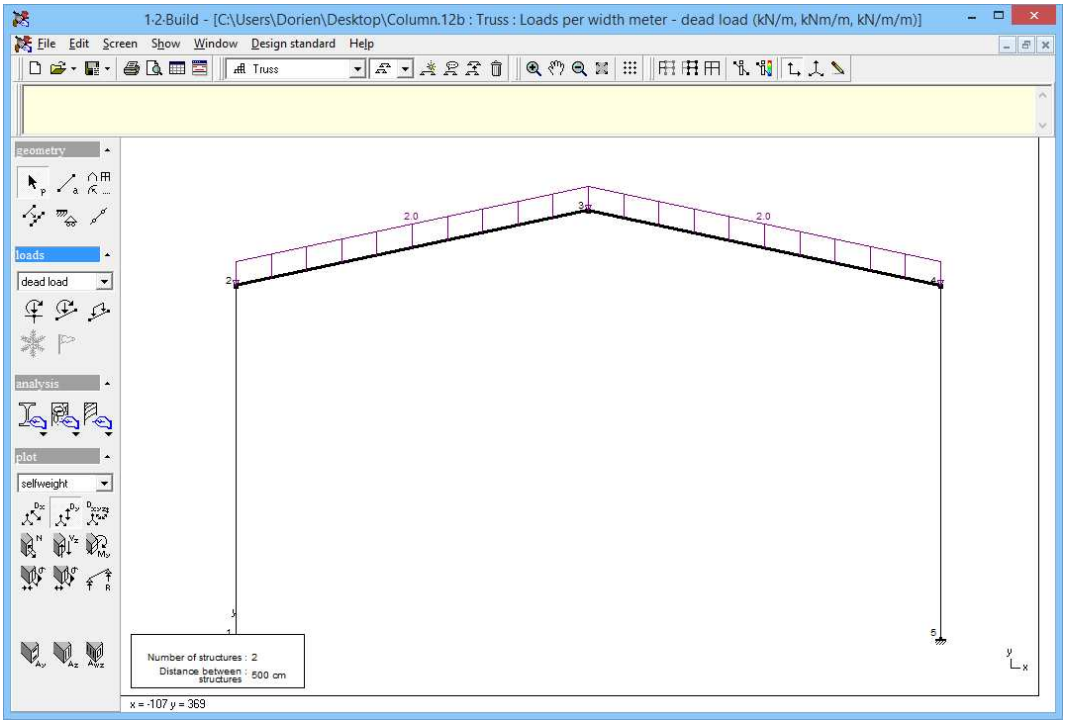
Suppose a structure consisting of a number of identical frames. For example, copy the truss that you have just defined on the basis of the icon in the toolbar. Indicate that it is a repetitive structure.



The left bottom shows a frame indicating the number of structures and the distance between them. After double clicking this box, a dialog will appear where you can modify the data.



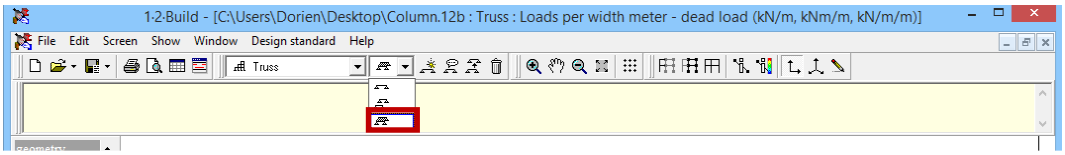
Currently no loads are defined. Therefore, select the beams of the truss and click on the icon  . We note that the loads are now expressed in kN/m/m. Place a horizontal distributed load of 2kN/m/m.



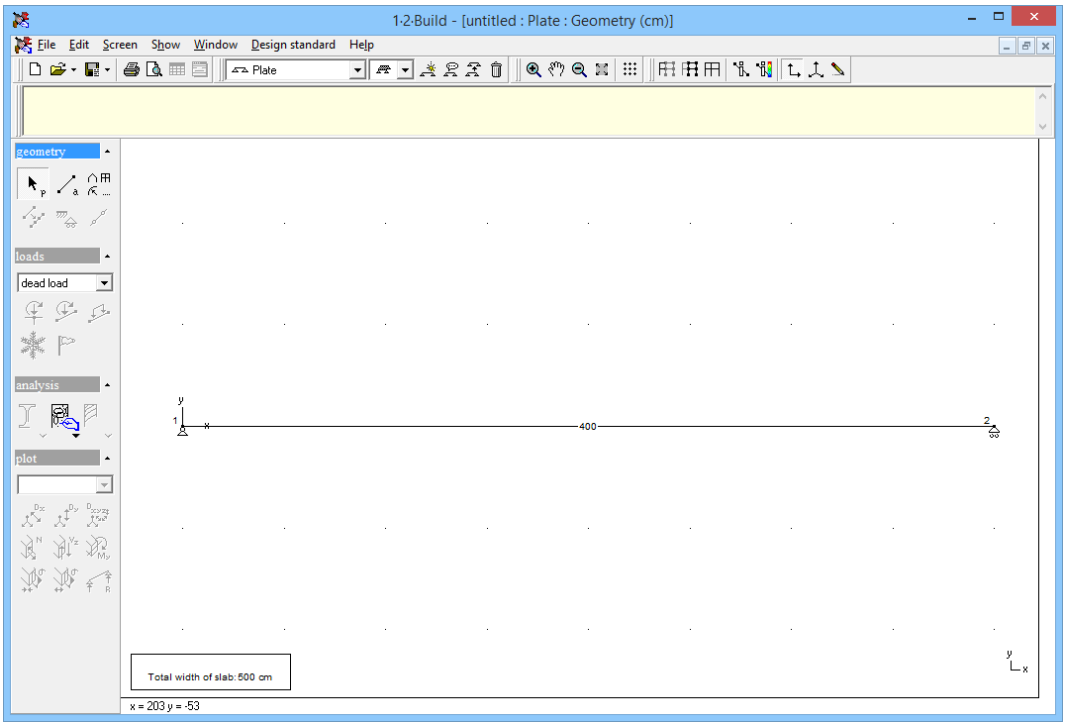
Note: 1•2•Build automatically calculates the load per running meter in the direction of the beams of the frame. The load shall accordingly be multiplied by the distance between the rafters. This means that you design the porches if they were all in the middle of the structure. This is acceptable if it is remembered that all the frames are carried out with the same dimensions in practice. If you wish to take into account that the frames on the edge only support half of the load, than divide the distance between the rafters by two.

6 Design of plates in reinforced concrete

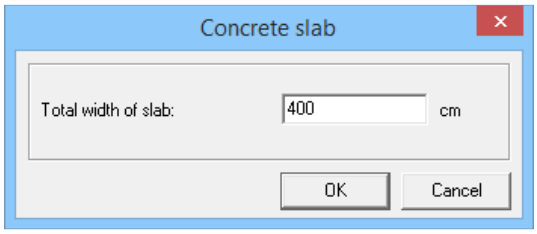
1•2•Build also allows to dimension plates bearing in one direction. Add a new calculation model for this purpose by clicking on  the icon in the toolbar.



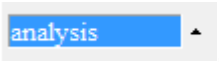
Suppose that the span is 4m, and that the plate is simply supported (in other words draw a beam with length 4m and simply supported).



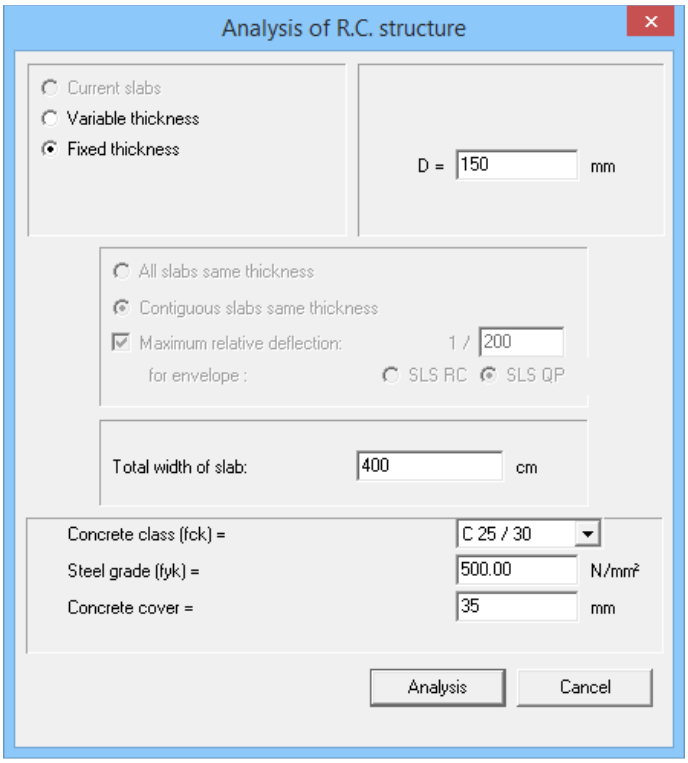
The width of the plate is written on the left bottom of the window. The default is 5m, but you can change this any time by double clicking this field. Activating the following dialogue, allowing you to specify a new value for the plate width.



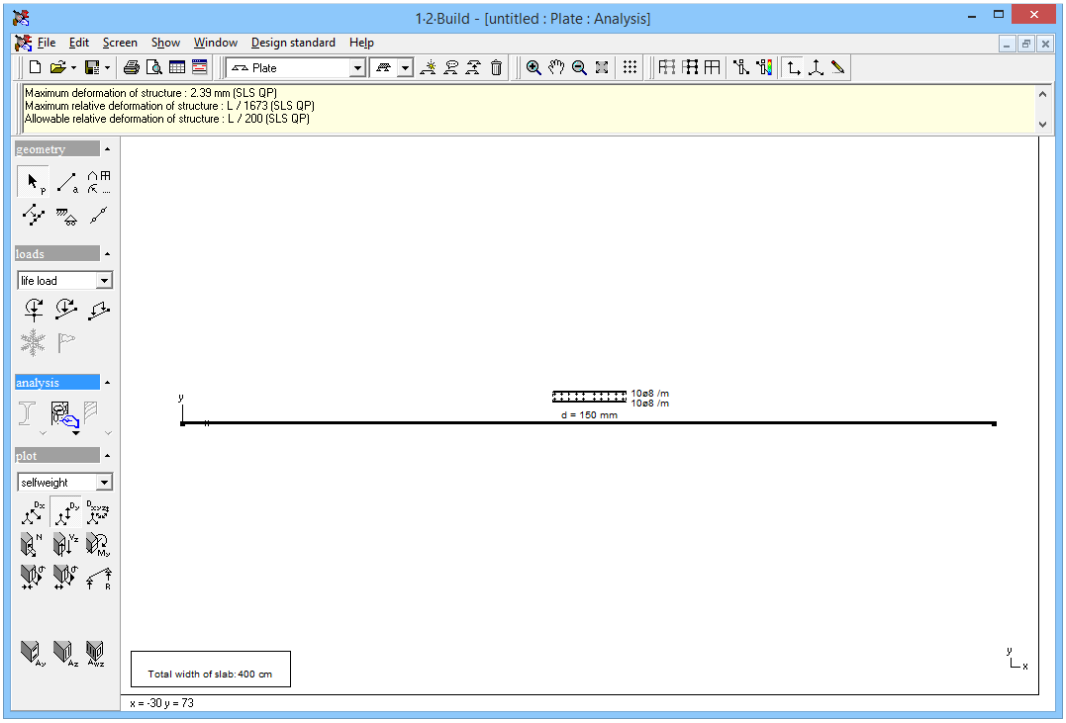
Then define a permanent and life load of 2kN/m².

The optimal size of this place in reinforced concrete, can quickly be determined by the icon  from the 

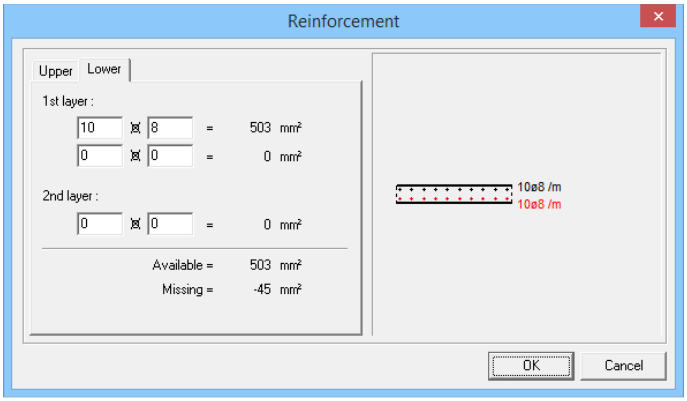
. Choose **a** plate with a **fixed thickness of 150 mm.**



In addition, define the concrete strength (C25/30), together with the quality of steel ($f_{yk} = 500N/mm^2$) and the reinforcement coverage (35mm). Start the calculation with the button Analysis .



By double clicking the plate in the window, you get access to detailed information about the proposed reinforcement:

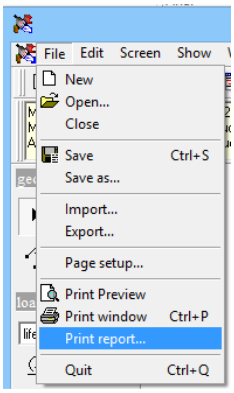


Here you can adjust the reinforcement details for both lower and upper reinforcement if you wish. However, if you specify a practical reinforcement that doesn't meet the requirements of the calculation, then **1•2•Build** will draw a red cross over the reinforcement sketch.

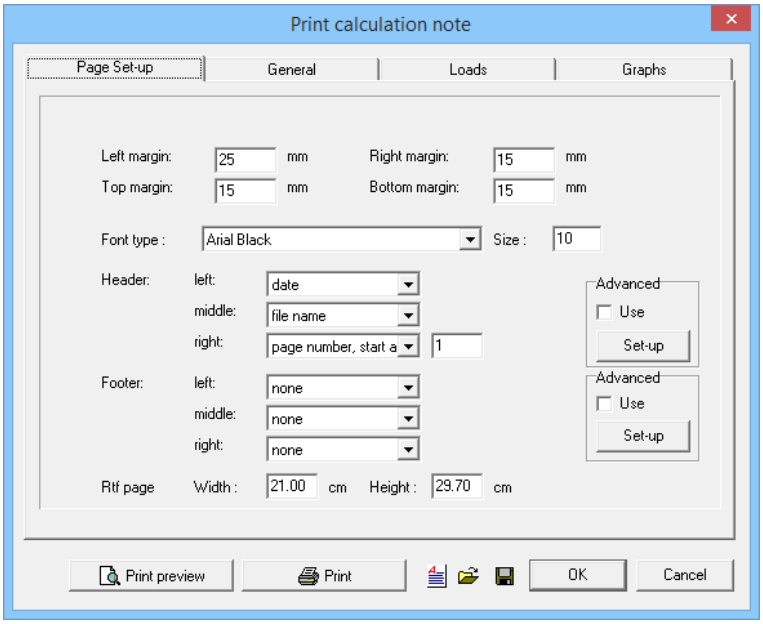
It should be noted that **in 1•2•Build elastic deflections are calculated not taking into account any cracks nor creep! The actual deflections can be a factor 2 to 5 higher than the proposed elastic deflections.**

7 Reports

Your final task is to document the results in a calculation note. From activate this feature from the menu **File - Print report ...**



And get access to the following dialogue.



This dialogue contains a number of tabs which you can specify

- what information you want to include in the calculation note
- how this information is formatted

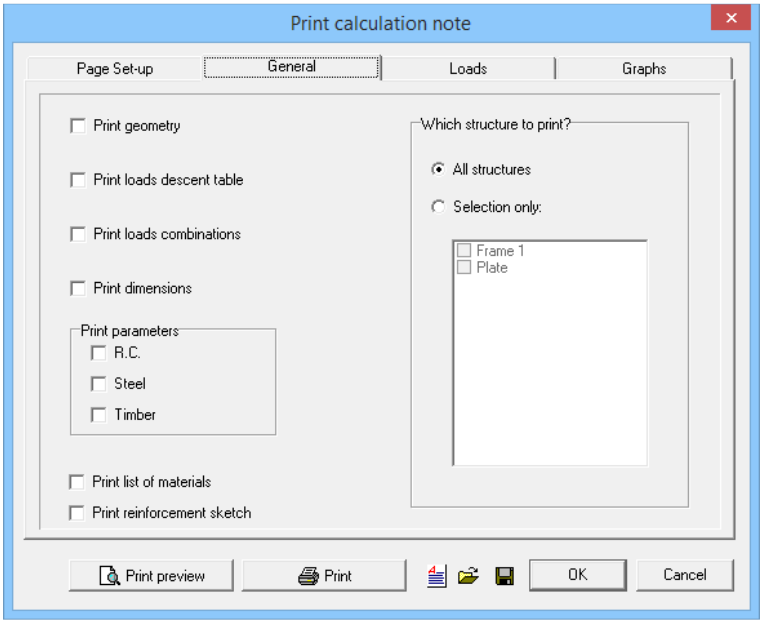
The tab page **Page Setup** shows the general information regarding the **Header** and **Footer** of the calculation note.

The actual content of the calculation note is defined via the tabs **General, Loads** and **Graphs.**

In the **General** page, you have access to a representation of the geometry, but

you can also request following information:

- the table of 'Decent of loads' that you have performed
- the description of how the load groups (dead load, life loads, …) were combined to perform the calculations



What concerns the design of the structure, you can ask the optimal dimension of the sections to be included in the report, together with an estimate of the amounts of materials:

- for steel this includes an estimation of the weight and the paint surface
- for reinforced concrete, this includes an estimation of the volume of concrete and the weight of the reinforcement

In addition you can also include the material parameters used for design in your calculation note.

Through the tab pages **Loads** and **Graphs** you have access to all graphs that were visible via the icon fields loads and plot.

After defining the details for the calculation note, you can send it to the printer by clicking on Print. In addition, you can adjust the printer settings, if necessary. However, it is always a good idea to take a preview in advance Print preview. This allows you modify the content calculation note, if it didn't meet your expectations.