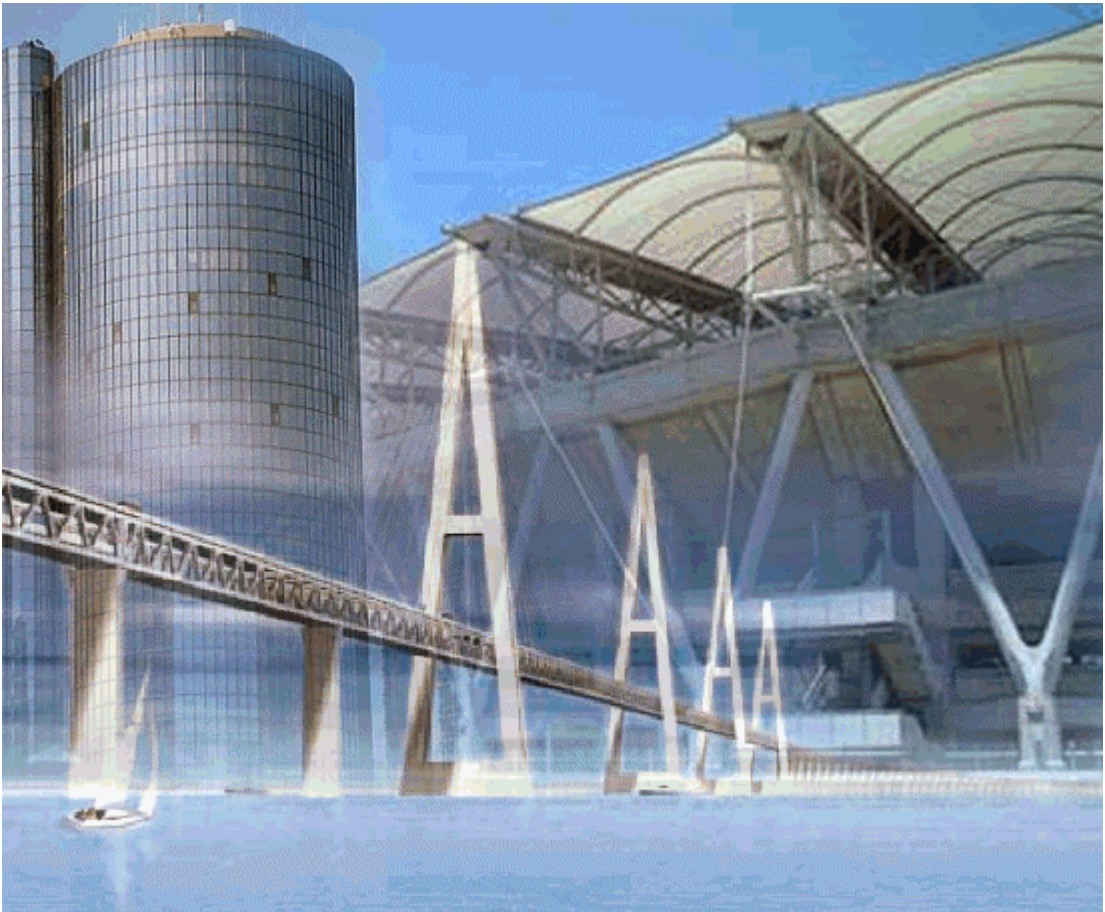




# SCIA.ESA PT



## Tutorial Frame Steel



Release : 4.x

Manual: SCIA-ESA PT Tutorial

“Example or a steel construction”

Revision: 07/2004

---

**SCIA Group n.v.**

**Scientific Application Group**

**Industrieweg 1007 B-3540 Herk-de-Stad (België)**

**Tel.(+32) (0)13/55 17 75 Fax.(+32) (0)13/55 41 75**

**E-mail scia@scia.be**

---

**SCIA W+B Software b.v.**

**Postbus 30119 NL-6803 AC Arnhem (Nederland)**

**Tel.(+31) 26-3201230 Fax.(+31) 26-3201239**

**E-mail scia@scia.nl**

---

**SCIA sarl**

**Parc Club des Prés**

**Rue Papin, 29 - F-59650 Villeneuve d'Asq (France)**

**Tel.(+33) (0) 3.20.04.10.60 Fax.(+33) (0) 3.20.04.03.36**

**E-mail scia@nc.net.tm.fr**

---

**SCIA Software Gbr**

**Emil-Figge-Str. 76**

**Gebäude I**

**Tel.(+49) 231/9742586**

**Fax.(+49) 231/9742587**

**E-mail scia@scia.de**

---

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA Software is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2004 SCIA Software. All rights reserved.

|   |           |
|---|-----------|
| <b><i>VOLUME 1: TUTORIAL</i></b>                                | <b>7</b>  |
| Welcome .....   | 7         |
| Example .....   | 8         |
| <b>INSTALLATION</b>   | <b>9</b>  |
| System requirements .....                                       | 9         |
| <b>GETTING STARTED</b>  | <b>10</b> |
| Starting a project .....  | 10        |
| <b>PROJECT MANAGEMENT</b>                                       | <b>12</b> |
| Save, save as, close and open .....                             | 12        |
| <b>SCREEN STRUCTURE</b>   | <b>13</b> |
| Introducing the user interface .....                            | 13        |
| <b>GEOMETRY INPUT</b>   | <b>15</b> |
| Input of the geometry .....                                     | 15        |
| <b>MODIFYING THE GRAPHIC REPRESENTATION OF THE CONSTRUCTION</b> | <b>29</b> |
| Modifying the image .....                                       | 29        |
| <b>CALCULATION DATA INPUT</b>                                   | <b>33</b> |
| Load cases and load groups .....                                | 33        |
| Loads .....   | 35        |
| Combinations .....  | 41        |
| <b>CALCULATION</b>  | <b>43</b> |
| Linear calculation .....  | 43        |
| <b>RESULTS</b>  | <b>44</b> |
| Viewing results .....   | 44        |
| <b>DOCUMENT</b>   | <b>47</b> |
| Documentation .....   | 47        |
| <b>CAD FORM</b>   | <b>49</b> |
| Cad form .....  | 49        |
| <b>CODE CHECK</b>   | <b>51</b> |
| Steel code check .....  | 51        |
| Fire resistance check .....                                     | 63        |
| <b>CONNECTIONS MODULE</b>                                       | <b>66</b> |
| Connections .....   | 66        |

---

|                               |           |
|-------------------------------|-----------|
| <b>GRID LINES</b>             | <b>73</b> |
| <b>IMAGE GALLERY</b>          | <b>76</b> |
| <b>PAPERSPACE GALLERY</b>     | <b>79</b> |
| <b>CUT BOX AND ACTIVITIES</b> | <b>81</b> |
| Cut box .....                 | 81        |
| Activities.....               | 82        |

---

# **Volume 1: Tutorial**

## **Welcome**

Welcome to the SCIA·ESA PT Tutorial. SCIA·ESA PT is a calculation program under Windows NT4/2000/XP with a large range of application: from checking simple frames to the advanced design of complex projects in steel, concrete, wood...

The program includes the calculation of 2D/3D frames, including the profile check and the connection check for steel constructions. The entire calculation and design process is integrated in one program: geometry input, input of the computational model (loads, supports...), linear and non-linear calculation, output of the results, elements check and optimisation to the different codes as well as generation of the calculation note...

SCIA·ESA PT is available in three different versions:

---

### Licensed version

The licensed version of SCIA·ESA PT is protected with a 'dongle', which is installed onto the parallel or USB port of your computer, either with a software licence, which is installed onto your network.

SCIA·ESA PT has a modular structure. The user chooses from the available modules to compose a calculation program, which perfectly matches his or her needs.

In the general product overview of SCIA·ESA PT you will find an overview of the different available modules.

---

### Demo version

If no protection was provided or if the program doesn't find any protection, it will be started in demo version. The properties of this demo version are:

All projects can be entered;

The calculation is restricted to projects containing 25 elements, 3 plates/shells and two load cases;

The output contains the watermark "Unlicensed software";

The projects that were saved in the demo version cannot be opened in the licensed version.

---

### Student version

The student version contains all modules and is not restricted as elements or load cases are regarded. It is also protected by a 'dongle' or through the software.

The output contains the watermark "Student version". Projects that were saved in the student version cannot be opened in the licensed versions.

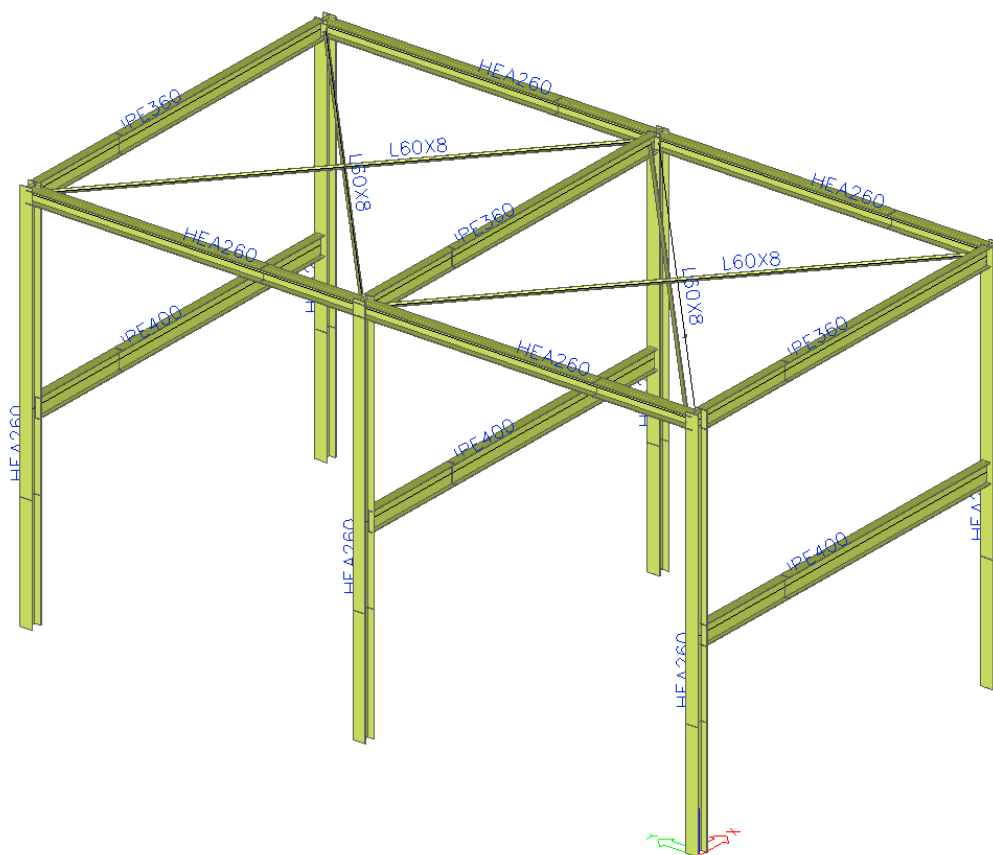
---

## Example

The example of this Tutorial can be designed with the three versions. The program must be installed onto a local computer or a network. The installation procedure is described in chapter 2. Before you proceed, you must be able working with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.

This Tutorial describes the main functions of SCIA·ESA PT for the input and calculation of a 3D steel frame.

The figure below shows the hall that we will calculate in the first volume.



**Figure 1: Model of the construction.**



---

# Installation

## System requirements

For the installation of SCIA·ESA PT, your system needs the following system requirements:

### Hardware requirements

---

|  |   |
|--|---|
| processor speed                                  | 730Mhz (recommended: > 1 GHz)   |
| RAM  | 256 MB (recommended: > 512 Mb)  |
| Graphic card                                     | 32 MB, OpenGL support   |
| Free disk space for program                      | 250 MB  |
| Free disk space for projects and temporary files | 100 MB (the required space can amount up to several GB for very large projects) |

---

### Software requirements

---

|                      |   |
|----------------------|---|
| MS Windows NT 4.0    | MS Windows NT 4.0 with Service Pack 6 or higher,<br>Internet Explorer 4.0 with Service Pack 2 or higher,<br>MDAC 2.5 (version 2.50.4403.12, completed with<br>file ODBCJT32.DLL with version number 4.0.4403.2) |
| MS Windows 2000 / XP | This system contains all required components.   |

---

### Other requirements

For the installation of SCIA·ESA PT you need administrator rights.

For working with SCIA·ESA PT you need ordinary user rights.

The general set-up program of the CD-ROM will automatically start when you insert the CD-ROM in the CD-ROM player. You will obtain a menu listing all available software on the CD-ROM. Follow the instructions on the screen to install SCIA·ESA PT.

# Getting started

## Starting a project

Before we can start a project, we first have to start the program. Once the program has started, we can open an existing project or start a new one.


### Starting the program

1. Double-click on the SCIA·ESA PT icon in the Windows Desktop.
- Or:
2. If the icon is not installed, click **[Start]** and choose **Programs > SCIA ESA PT .<version nr.> > SCIA·ESA PT**.

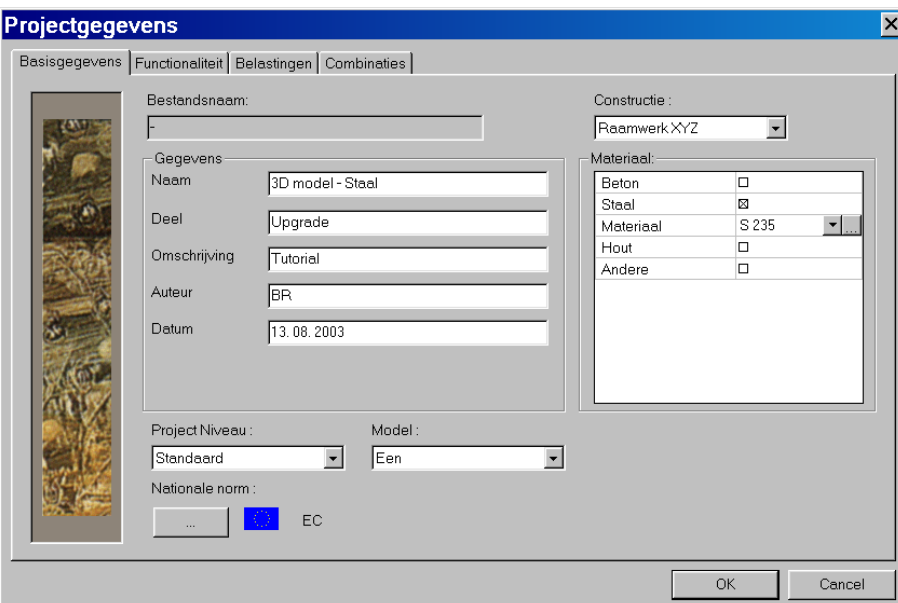
If the program doesn't find any protection, you will obtain a dialogue listing the restrictions of the demo version. A second dialogue will inform you why no protection was found. Click **[OK]** in both windows. The start screen of SCIA·ESA PT appears.

For this Tutorial we will start a new project.

### Starting a new project

1. If the dialogue **Open** appears, click **[Cancel]**.
2. Click **New**  the icon in the toolbar.

Now, the dialogue **Project data** is opened, in which general data about the project can be entered.



**Projectgegevens**

Basisgegevens | Functionaliteit | Belastingen | Combinaties

Bestandsnaam:

Constructie:

Gegevens:

Naam:

Deel:

Omschrijving:

Auteur:

Datum:

Materiaal:

Beton ☐

Staal ☒

Materiaal:

Hout ☐

Andere ☐

Project Niveau:  Model:


Nationale norm:

OK Cancel

**Figure 2: Project data**

3. In the **Data** group, enter your preferred data. These data can be mentioned on the output.

4. Choose **Project level: Default** and **Model: One**.

5. Click on the rectangular button  below **National Code** to set up a default code for the project. This code will determine the available materials, combinations and code checks. For the Tutorial project, we choose EC. The **Codes in project** dialogue appears.

- a) Choose **[Add]**.

The dialogue **Available national codes** appears.

- b) Select the European flag and click **[OK]**.

You will return to the **Codes in project** dialogue and **EC** will be added.

- c) Select the flag named **EC**.

- d) Select the option **Active code** and click **[Close]**.

You will return to the dialogue **Project data** and **EC** will be the active code.

6. Select **Frame XYZ** in the **Construction** field.  
The type of construction (2D Frame, 3D Frame, 2D Plate, 3D Shell...) restricts the possible input during the calculation.
7. In the **Material** group, select **Steel**.  
Below the item **Steel**, a new item **Material** will appear.
8. Choose **S235**.
9. Now, all information required to start the project has been entered. Confirm by clicking **[OK]**.

Note:

Settings in the Project data menu:

On the tab **Basic data** you can set a project level. If you choose default, the program will only show the most frequently used basic functions. If you choose advanced, all basic functions will be shown. On the **functionality** tab, you choose the options that you need, filtering in this way the non-selected functionalities out of the menus, thus lightening the program. On the **Combinations** tab, we will set the default values for the partial safety factors. For this “Tutorial volume 1”, the default settings will be used.

---

# Project management

## Save, save as, close and open


Before entering the construction, we first will discuss how to save a project, how to open an existing project and how to close a project. When executing the project of this Tutorial, you can save the project every now and then. In this way, you can leave the program any time and continue with the project at a later time from that point on.

---

### Saving a project

1. Click  in the toolbar.

If the project has not been saved yet, the dialogue **Save as** will open. Click on the arrow in the list box **Save in** to select the disk where you want to save the project. Select the folder where you want the project and click **[Open]**. Select sub-folders. Enter the file name below **File name** and click **[Save]** to save the (empty) project.

If you now click the  icon again, the project will automatically be saved with the same name. If you choose **File > Save as** in the main menu, you can enter a new/other disk, folder and name for the project file.

---

### Closing a project

1. To close this project, choose **File > Close** in the main menu.

A dialogue box will appear, asking you if you want to save this project. Depending on your answer, the project will be saved and the active dialogue will be closed.

---

### Opening a project

1. Click  to open an existing project.

A list of projects will appear on the screen. Select the desired project and click **[OK]** (or double-click the project to open it).

# Screen structure

## Introducing the user interface

The user interface is the part of the program, which is visible on the screen and which ensures the communication between the user and the program. Often, it is called a “graphic interface”.

We will now introduce the different components to you one by one.

### Set-up

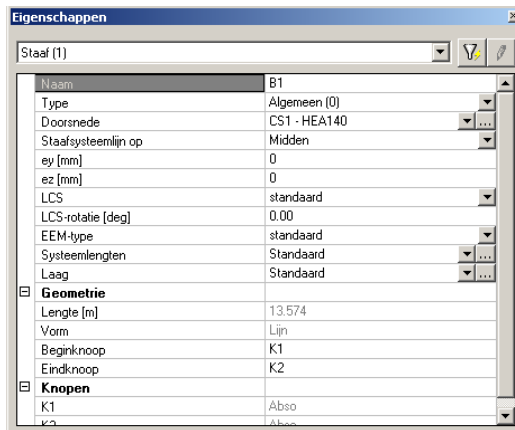
*A screen consists of the following components:*

|                      |   |
|----------------------|---|
| Title bar            | The upper part of the application window, containing the basic information regarding the application.   |
| Status bar           | Shows information related to a certain program action   |
| Menu bar             | Contains the menu that can be used to control and operate the program.  |
| Menu tree window     | Contain a tree-like menu that is used to open the individual program functions.   |
| Graphic window       | Is a kind of application window, showing the drawing of the designed object. The window shows the designed object, the calculated results and accepts mouse commands.   |
| Document window      | Is a kind of application window, showing the information about the designed object in tabular form, text comments and drawings.   |
| Print preview window | Shows all kind of information in tabular form and drawings. It can be used to edit object properties.   |
| Command line         | Can be used to type commands to control the program and also shows brief instructions regarding the required actions to be taken during individually running processes. |
| Toolbar              | Enables fast access to the most general functions   |

In addition to these default Windows application parts of a user interface, SCIA.ESA PT also uses a series of unique, especially developed control elements. These will be described in separate chapters.

### Properties table

The properties table is a unique control of SCIA.ESA PT that is used in the program dialogue screen as well as in the Properties window. The control looks like a table. The first column contains the name of each individual element. The second column shows their values.



**Figure 3: Properties table**

Generally, the values can be adapted in the “numerical cells” of the properties table. There are different ways to change the value. Furthermore, the individual component of the table are linked either (i) to another part of the program either (ii) to a graphic screen.

## Database Manager

The “Program database” manages the different libraries of the program. For instance, there is a material library, a section library, a catalogue blocs, etc. The “Database manager” contains tools to modify the entities of the program database.

|   |   |
|---|---|
| <a href="#">List of defined entities in the specific database</a> | The list shows all entities related to the database of the manager, which have been defined in the current project so far.                              |
| <a href="#">Properties table</a>                                  | This table shows a brief summary of parameters for the database entity that has just been selected in the list of already defined entities (see above). |
| <a href="#">Graphic window</a>                                    | This window shows a drawing of the database entity, whose parameters are listed in the properties table   |
| <a href="#">Control buttons</a>                                   | The button give access to functions, which are accessible from within the specific manager  |
| <a href="#">Filter</a>  | The filter ensures a legible representation of the data in the Manager  |

## Toolbar

Toolbars are small “window look-a-likes” containing a number of buttons or icons. The buttons can be used to open different functions. The toolbars can be left floating on the screen or ‘docked’ at the side of the screen.

We will introduce the different buttons in the next chapters.

# Geometry input


## Input of the geometry

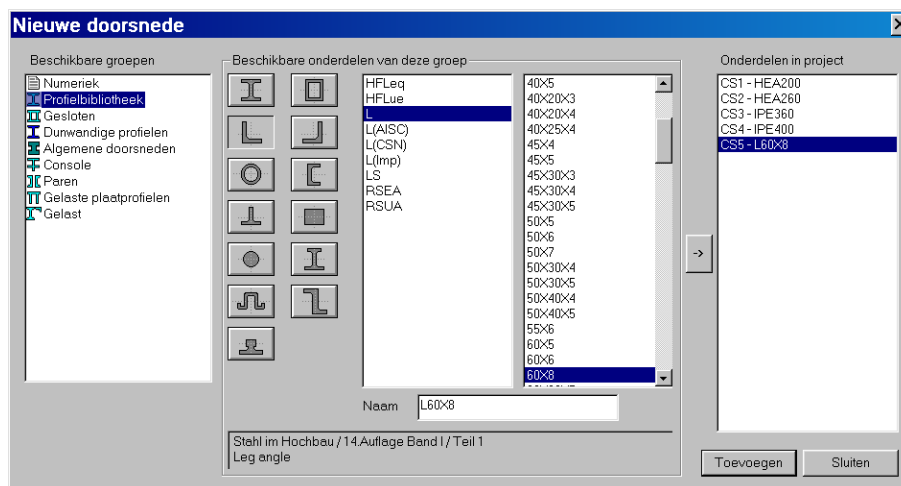
When you start a new project, you must enter the geometry of the construction. You can also draw the construction directly or use templates with parametric blocs.

### Profiles



When entering one or more 1D structure elements, a profile type is immediately assigned to each member. By default, the active profile type is represented. You can open the profile library to activate another profile type. When upon adding a structure part no profile type has been defined yet, the profile catalogue will be automatically opened.

#### Adding a profile

1. Click the **Sections**  icon in the toolbar.  
The first time you do this, the profile library will be scanned. The **Sections** manager will be opened. If no profiles have been assigned to the project yet, the **New section** dialogue will be automatically opened.



**Figure 4: The New section dialogue**

2. Click **Profile library** in the group **Available groups** to select a profile from the library. The available elements will be shown.
3. Click  in the group **Available elements of this group**  
The available I-profiles are shown.
4. Select in the list in the middle the type **HEA** and in the right-hand list **200**. Click  to add this profile to the list of **Elements in project**.
5. For this Tutorial, you add the following profiles: **HEA260**, **IPE360**, **IPE400** and **L60/6**. The list **Elements in project** shows the already selected profiles.

6. Click [**Close**] to quit this dialogue.

You will return to the **Sections** manager.

7. Click [**Close**] to close this dialogue.

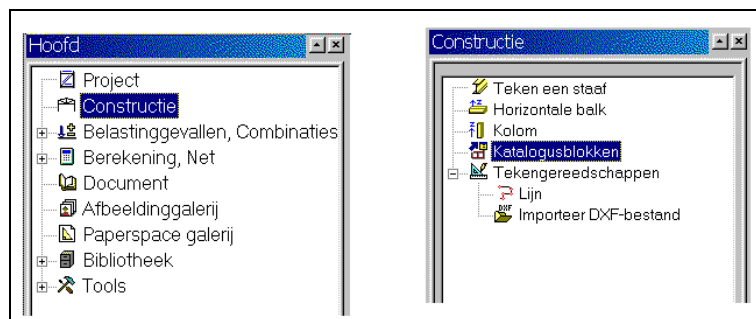
You will return to your project.

## Geometry

---

### The Construction service

1. When starting a new project, the '**Construction**' service is opened. If you want to change the construction at a later time, you must double-click **Construction** in the **main** menu.



**Figure 5:**The main menu and the service Construction

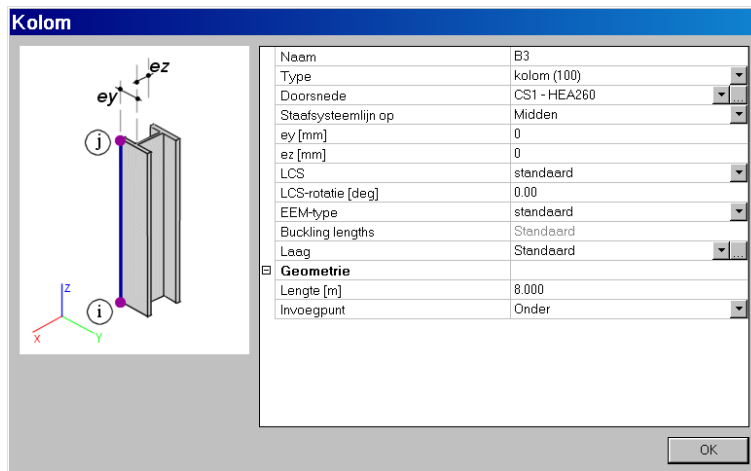
The easiest way to enter our structure is using the available blocs from the catalogue. For this Tutorial, we could use the **rRaser** from **3D frame**. For didactic reasons, we will however not use this predefined bloc, but build the construction using the simplest structure elements.



---

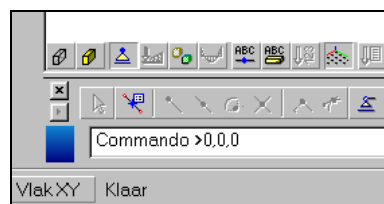
 Entering a column

1. To enter a new column, you use the option **Column** in the service **Construction**.



**Figure 6: The definition menu for a column**


2. Select **HEA260** in the library. Set the length of the element to 8 m.
3. Confirm with the **[OK]** button.
4. Position the element in the origin. To this end, you enter the coordinates **(0,0,0)** in the command line and confirm with the <enter> button. (See figure)



**Figure 7: The command line**

5. Position the element in position **(8,0,0)** as well.
6. End your input with the <Esc> key.
7. The two entered columns are selected: they are highlighted in red. Cancel the selection by clicking the <Esc> key once more.

**Notes:**

- \* The properties of the selected elements are shown in the properties window and can be modified.
  - \* If the project does not contain any section at the time when you order the definition of a structure element, the program will automatically open the dialogue box **New section**.
  - \* Any input can be ended either (i) using the ESC key, either (ii) with the right mouse button.
  - \* Click **Zoom all**  in the toolbar to see the entire construction.
- 

---

 Drawing a member

Another method that can be used to add a construction part is the option **Draw member**. With this option, you can draw members by defining the first and the end node. When defining the nodes, we can either (i) enter the coordinates in the command line, either (ii) work graphically. The setting of the pick mode for the mouse cursor here is highly important. At the bottom right you can set this mode using the button **[Pick mode]**. We choose Centres and Starting points.

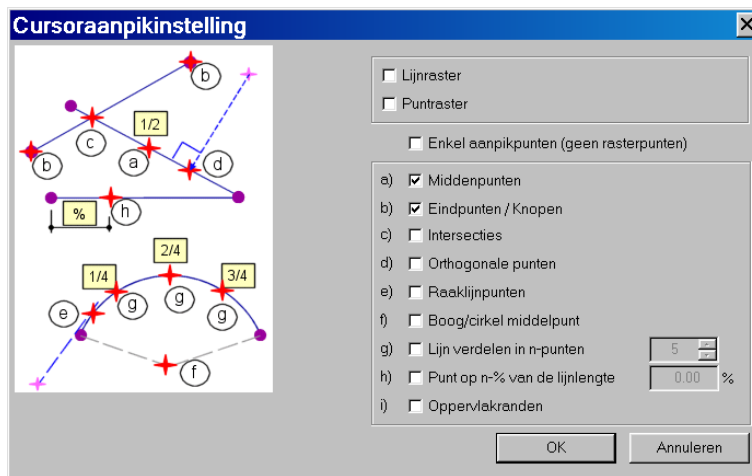


Figure 8: The menu Cursor pick setting

1. Activate the function **Draw member** in the service **Construction**.
2. Set the properties. (**IPE400**)

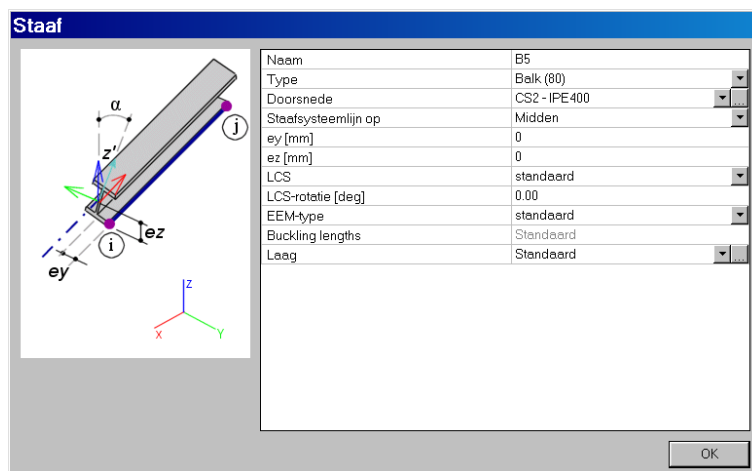
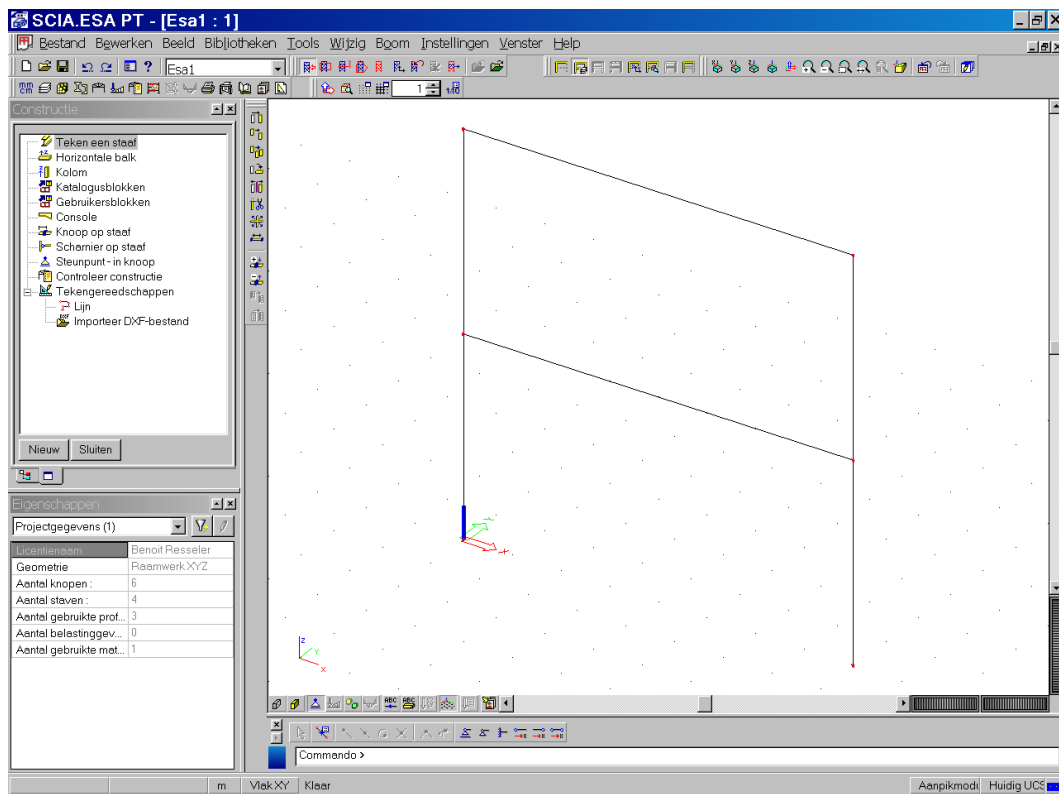


Figure 9: The Member menu

3. Draw the horizontal member connecting the centres of the columns.
4. End the drawing of IPE400 either (i) with the right mouse button either (ii) with the <Esc> key.
5. Activate the function **Draw member** in the service **Construction**.
6. This time, choose IPE360.
7. Draw the member connecting the column heads.
8. End the function Draw member either with the right mouse button either with the <Esc> key.


9. The last entered elements are selected by default. Use the Esc key to cancel the selection.



**Figure 10: Already completed input**


We already have defined one truss. The entire construction consists of three identical trusses. (See figure 1). Our next mission is to copy the already entered part twice. The members connecting the three frames will be generated during the same procedure. Therefore, we first have to activate the right profile type before we can start copying.



#### Modifying the active profile

1. Click  in the toolbar.  
The **Sections** manager appears.
2. Select **HEA200** and click [Close].

Now, we can copy the construction.

#### Making multiple copies

1. Select the construction using the button **Select all** . In this way, all members and all nodes are selected. (A selection is highlighted in red)
2. In the selected nodes the trusses are mutually connected with active section (HEA200). Deselect all nodes except the one located at the head of both columns. (See figure 11). Hold the <CTRL> key down and click with the left mouse button on the nodes concerned. You can also invert the

mouse selection mode (button ) using the switch (button ) and then click on the nodes to be deselected.

A selected member is represented in red; a selected node is surrounded by a red box.

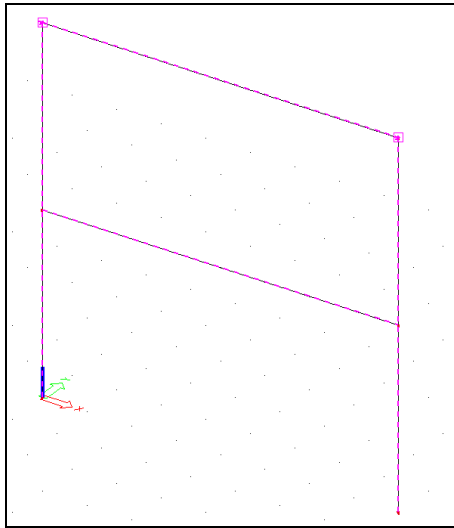



Figure 11: Image of the selection

3. Click the **Multiple copies**  button.

The dialogue **Multiple copies** is opened.

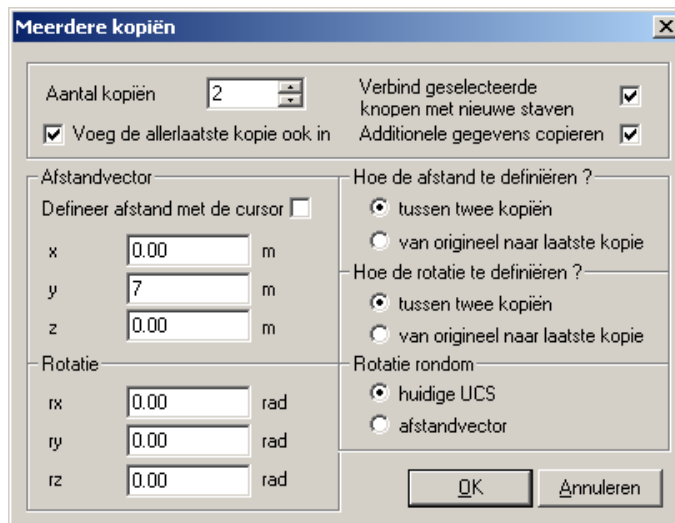


Figure 12: The Multiple copies dialogue

4. Enter the data of the figure in and click [OK].

Three copies will be generated. The distance between the trusses is 7 m in **Y** direction of the **User Coordinate System**. The different trusses are connected by the **selected nodes**.

*Note:*

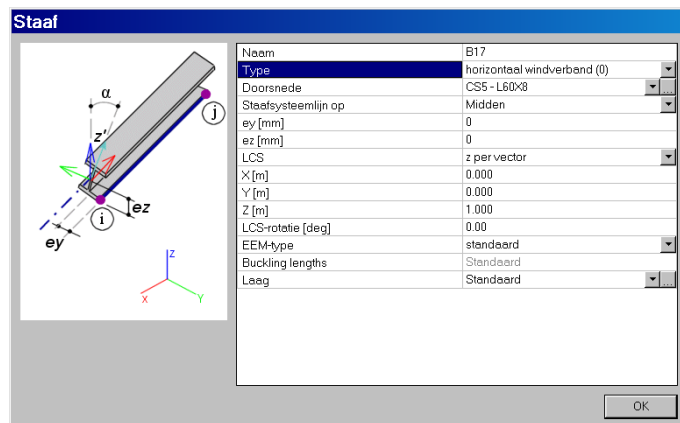
---

*For absolute coordinates enter x,y,z, for relative coordinates you use @x,y,z*

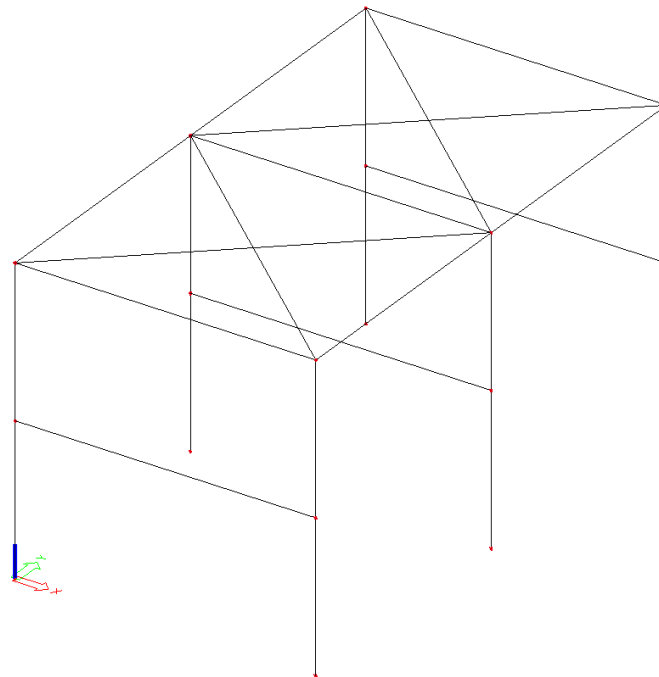
---

## Drawing bracings

1. In the service Construction, choose the option Draw member.
2. Set the parameters according to the menu below:  
type: horizontal bracing  
section: L60x8



3. Draw the diagonals in the roof



## Checking the construction

Once our model defined, we can check the construction for correctly connected members by means of the tool **Construction check**.

Apart from the correct connection of the construction elements, other errors are detected as well, such as free nodes, double nodes, double members, members with zero length...

---

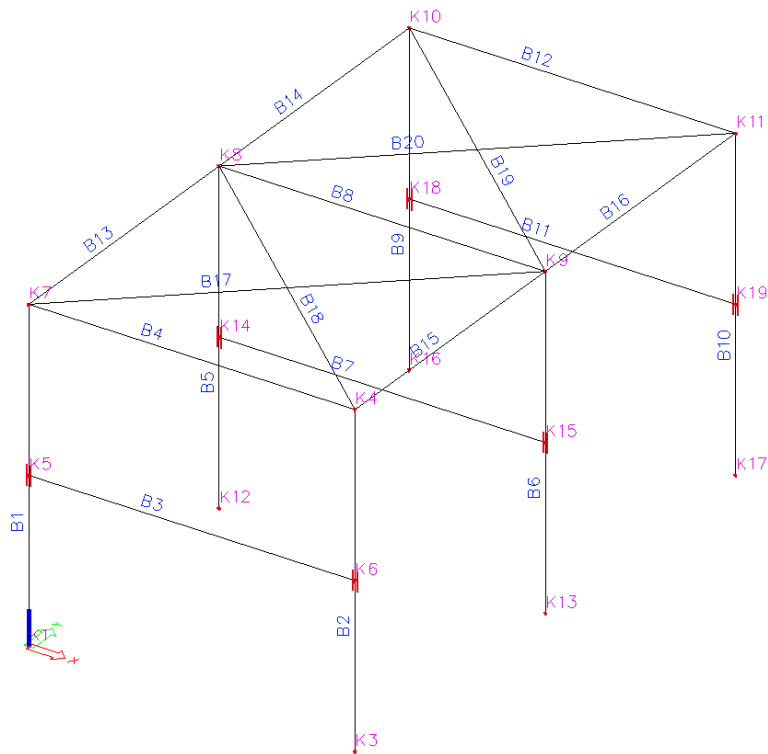
### Checking a construction

1. Click the option **Construction Check** in the **Construction** service.
2. Carry out the check.
3. The check menu indicates possible errors in the construction.

**Figure 13: The dialogue of the construction data check**


4. Click **Continue** to design the connections in the 6 nodes.
5. All entities will be corrected as indicated in the menu.

Below, you will find an image of the construction, in which all members are correctly connected. The structure is represented by means of node and member names.





---

#### Activating a label for node names

You activate the labels of the node names using the  button located at the bottom left.

---

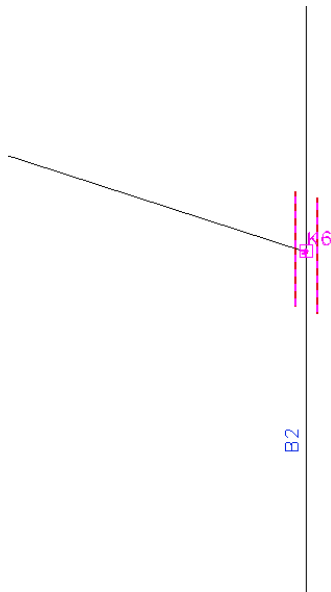
#### Activating a label for member names

You activate the labels of the member names using the  button located at the bottom left.

---

#### Relative and absolute nodes

The nodes are relatively connected to the respective members. A relatively connected node is represented as shown in the figure below.



**Figure 14: Connected node**


By selecting the node, you can consult the properties in the table. We see that Node 6 is relatively connected on member B2. It is located at half the member length. This implies that an elongation of member B2 will lead to a displacement of node K6.

| Eigenschappen     |        |
|-------------------|--------|
| Knoop (1)         |        |
| Naam              | K6     |
| Gekoppelde k...   | tot B2 |
| Coördinaat        | Rela   |
| Positie x         | 0.500  |
| GCS-coördinaat    |        |
| Coördinaat X [m]  | 8.000  |
| Coördinaat Y [m]  | 0.000  |
| Coördinaat Z [m]  | 4.000  |
| UCS-coördinaat    |        |
| Coördinaat ux [m] | 8.000  |
| Coördinaat uy [m] | 0.000  |
| Coördinaat uz [m] | 4.000  |
| Staven            |        |
| Staf              | B2     |
| Staf              | B3     |

Instead of working with relative nodes, we could have broken member B2 apart into two members, for which member B2 ends in node K6 and the new member begins in node K6.

You proceed as follows.




1. Click the “Break intersection” button ( button  ) and select intersecting members.
2. Confirm with <ESC> to connect the members with absolute nodes.



## Hinges

The members B3, B7 and B11 (IPE400 members) must be connected to the columns using a hinged connection, since the project is a 3D frame. The members transmit their forces through common nodes. The connection is rigid. Using hinges, we can liberate the moments at the member ends.

### Adding hinges

1. Double-click  **Scharnier op staaf** in the service **Construction**. The dialogue **Hinges on member** appears.

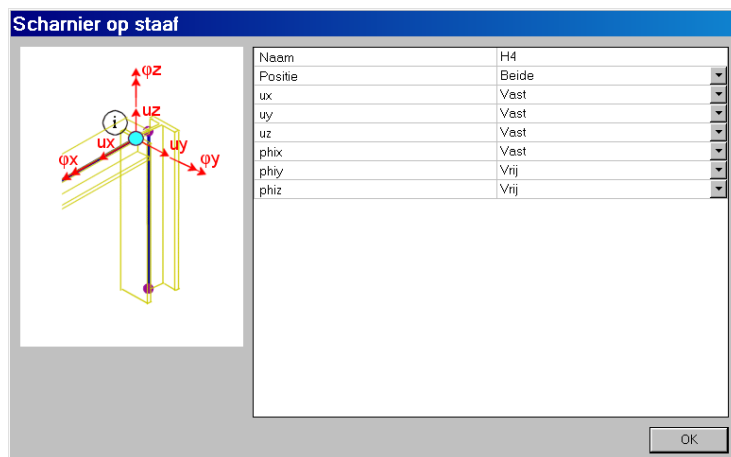


Figure 15: Hinges on member

2. Copy the data of the figure and click **[OK]**.
3. The hinges will be added when you point the beams with the left mouse button.
4. Click with the right mouse button in the graphic part to end the input.
5. The entered hinges are still selected. Press **Esc** to cancel the selection.

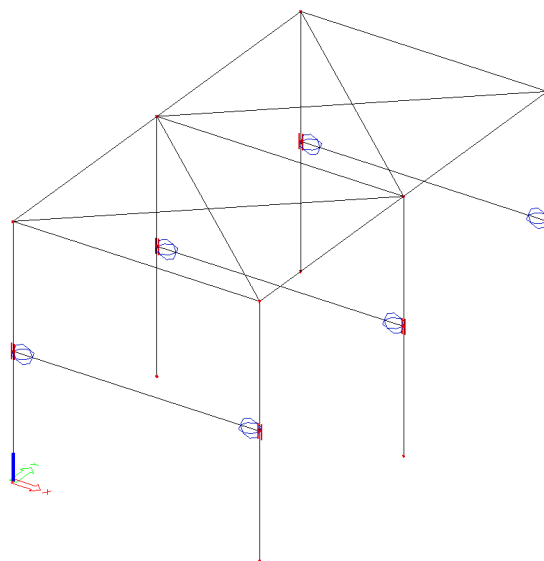



Figure 16: Position of the hinges

### Note

- 
- >Hinges are defined with regard to the local axis of the member elements
  - >The common properties of the selected elements can be modified through the properties table
- 

The bracings must be hinged in both ends for the bending moments. Instead of defining first a hinge type and then selecting the members, we can work the other way round as well.


As long as the service “Construction ” is active, the bottom of your screen will feature a number of predefined supports and hinges, which we now will use. (Similar options are available for other services. For instance, for the Load Cases service, you will find predefined loads).

1. Start from the situation, where nothing is selected. Select a bracing with the mouse cursor.
2. Use a filter selection to select all bracings. (See “Creating a filter selection”).
3. Use the option “Hinge both ends” (button ) to have hinged members in both ends.

---

#### Creating a filter selection

We will use a selection filter to select all members with a particular identical property.

1. Select a bracing with the mouse cursor.
2. Activate the “section” field in the properties table. The profile type L60x8 is assigned to all bracings, and therefore is a unique property.
3. Click the **Filter elements per property** button .

All bracings of the roof will be selected. If other members are selected as well, something is wrong with the profile assignment.


## Supports


In this project, all column bases are perfectly restraint.

We also could have used a filter to select all nodes with a Z coordinate equal to zero. This time, we will illustrate the option Selecting using a window. The procedure is as follows.

---

#### Adding supports

1. Click  in the toolbar to obtain a horizontal view.
2. Select the content of a window (see below: making a window selection).

3. Double-click the  in **knoop** item in the **Construction** service
4. Define a permanent restraint, as shown in the image below.

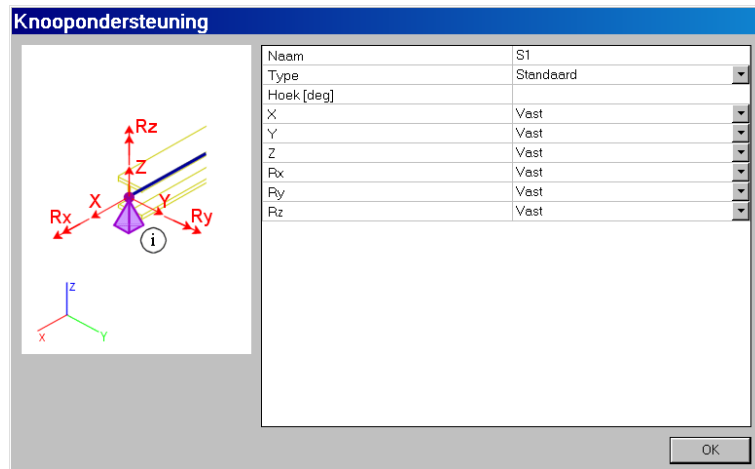

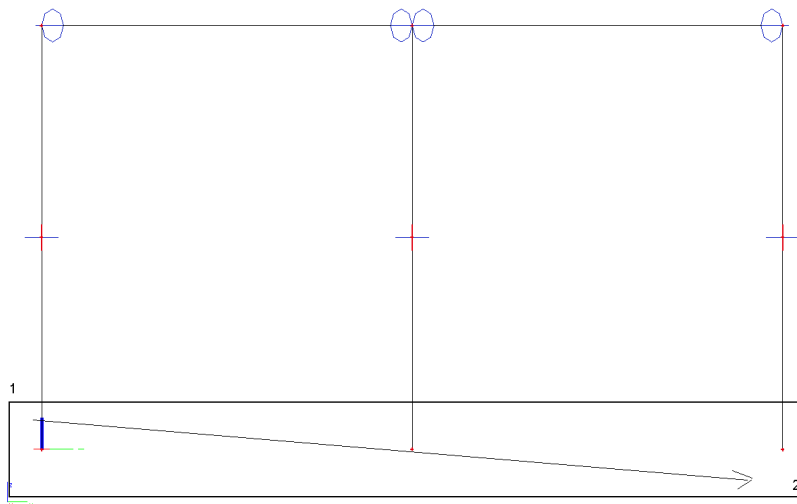


Figure 17: Supports in nodes – Perfect restraint

5. Return to the 3D view using the option .
6. Press the <Esc> key to cancel the selection.

#### Creating a window selection

1. Position the mouse cursor at the desired position in the graphic part.
2. Keep the mouse button down and slide the mouse from the left to the right. (Thus selecting only nodes and not the columns. See note below)



3. Release the mouse button.

All entities intersecting the rectangle and those who are completely within the rectangle are selected.

#### Notes:

---

*\* If the rectangle is drawn from the left to the right, only those entities located completely within the rectangle will be selected. If the rectangle is drawn from the right to the left, not only the entities located completely within the rectangle, but also those intersecting the rectangle will be selected.*

*\* A number of predefined supports are defined. For this example, we could have used the predefined*

*“perfect restraint” (button ) at the bottom of the screen.*

---

Below, you will see an image of the model with hinges and supports.

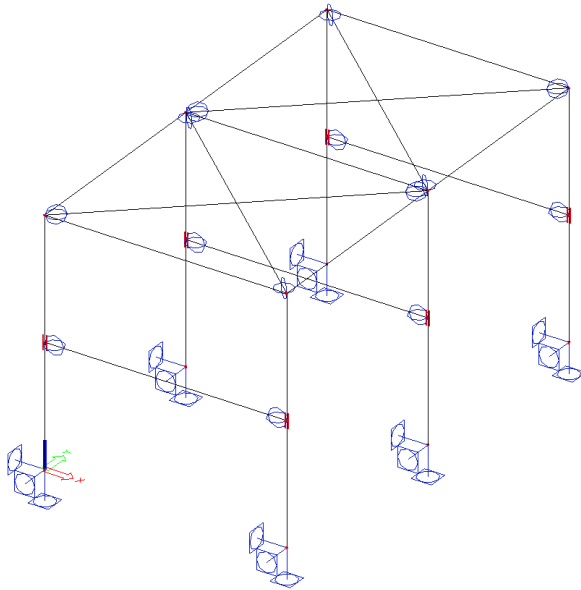


Figure 18: Model with hinges and supports

# Modifying the graphic representation of the construction

## Modifying the image

You have several possibilities to modify the graphic representation of the construction. Hereafter, we will discuss the main options:

- Manually modify the viewpoint onto the construction
- Setting a view direction
- The magnifying glass
- Modifying the image parameters through the “Image parameters” menu

---

### Modifying the viewpoint onto the construction

Setting the viewpoint using the wheels. At the bottom right of the graphic screen, three wheels are located, two horizontal and one vertical. These **wheels** enable you to **zoom** or **rotate** the construction.

1. To zoom or rotate the construction, click on the wheel (the cursor will change from an arrow shape into a hand), keep the mouse button down and move the wheel.

**Or**

Setting the viewpoint by means of a key-and-mouse combination.

1. Simultaneously keep the CTRL + right mouse button down and move the mouse to *rotate* the construction.
2. Simultaneously keep the Shift + right mouse button down and move the mouse to *move* the construction.
3. Simultaneously keep the CTR+Shift + right mouse button down and move the mouse to *zoom in or out* on the construction.

#### *Note*

---




*If you rotate the structure, while a node selection being active, you rotate the structure around the selected node.*

---



---

### Setting a view direction with regard to the global coordinate system


1. Click the button **View in X direction**  to obtain a view in X direction.
2. Click the button **View in Y direction**  to obtain a view in Y direction.
3. Click the button **View in Z direction**  to obtain a view in Z direction.


---


### The magnifying glass

Use  to enlarge

Use  to reduce

Use  to zoom in a window

Use  to get a view of the entire structure

Use  to zoom a selection

---

### Modifying the image parameters using the “Image parameters” menu

1. In the graphic screen, click the right mouse button. The following shortcut menu appears.



**Note:**

*If an element was selected previously, you can define a setting, which exclusively applies to the selected element. (An adapted shortcut menu will appear).*

2. Choose the option **Set image parameters**. The **Image parameter setting** dialogue appears. The menu contains different tabs. You can set the image parameters for all entities or only for the selected entities

---

### Image parameters – Entities

With the Entities tab, you can control the way in which the different entities are represented.

In the **Construction** group, you should pay attention to the following items:

**Style and colour:** you can represent the colour per layer, per material, per section or per CAD type

**Section:** with this option, you put the symbol of the section onto each member.

**Member LCS:** with this option, you activate the local axes of the members

**Beeldparameterinstelling**

Entiteiten | **Labels en Omschrijving** | Beeld

|                             |                                     |
|-----------------------------|-------------------------------------|
| <b>Constructie</b>          |                                     |
| Stijl en Kleur              | normaal                             |
| Staatsysteemlijn            | systeemlijn                         |
| Staafoppervlak              | geen                                |
| Rendering                   | draadmodel                          |
| Doorsnede                   | geen                                |
| Staaf LCS                   | <input type="checkbox"/>            |
| Knoop LCS                   | <input type="checkbox"/>            |
| Buckling lengths            | <input type="checkbox"/>            |
| <b>Additionele gegevens</b> |                                     |
| Stijl / Rendering           | basis                               |
| <b>Resultaatdiagram</b>     |                                     |
| Resultaten                  | <input type="checkbox"/>            |
| <b>Model</b>                |                                     |
| Modelgegevens               | <input checked="" type="checkbox"/> |
| <b>Belasting</b>            |                                     |
| Belastinggeval              | LC1                                 |

OK Annuleren

### Image parameters – Labels and Description

With the **Labels and Description** tab, you can represent the labels on the different entities. In the **Members** group, you should pay attention to the following items:

**Label:** activate or deactivate a label

**Name:** represent the name of the sections in the label.

**Section type:** represent the type of section in the label

**Length:** represent the length of the member in the label

**Beeldparameterinstelling**

Entiteiten | **Labels en Omschrijving** | Beeld


|                         |                                     |
|-------------------------|-------------------------------------|
| <b>Staven</b>           |                                     |
| Label                   | <input type="checkbox"/>            |
| Naam                    | <input checked="" type="checkbox"/> |
| Doorsnedenaam           | <input type="checkbox"/>            |
| Doorsnedetype           | <input type="checkbox"/>            |
| Lengte                  | <input type="checkbox"/>            |
| Laag                    | <input type="checkbox"/>            |
| Type en prioriteit      | <input type="checkbox"/>            |
| <b>Knopen</b>           |                                     |
| Label                   | <input type="checkbox"/>            |
| Naam                    | <input checked="" type="checkbox"/> |
| X-coördinaat            | <input type="checkbox"/>            |
| Y-coördinaat            | <input type="checkbox"/>            |
| Z-coördinaat            | <input type="checkbox"/>            |
| <b>Buckling lengths</b> |                                     |
| Label                   | <input type="checkbox"/>            |
| Naam                    | <input type="checkbox"/>            |
| Omschrijving            | <input type="checkbox"/>            |


OK Annuleren


---

Image parameters – Shortcuts


In the toolbar at the bottom left, you will find a number of frequently used options. The most important are.


**Show/hide surfaces**  to see the surfaces of the sections.


**Render geometry**  to obtain a rendered view of the members.

**Show/hide node labels**  to see the name of the nodes

**Show/hide member labels**  to see the name of the members

**Show/hide supports**  to represent supports and hinges

**Show/hide load**  to make the load case visible

**Show/hide load labels**  to show the value of the load

**Set load case for representation**  to change the active load case



# Calculation data input

## Load cases and load groups

Each load (e.g. nodal load, line load...) is assigned to a load case. A load case can contain different types of loads.

To each load case, properties are assigned, which are determinant for the combination generation. For instance, the action type of a load case can be permanent or variable.

Each variable load case is associated with a load group. The group contains information about the load category (wind, snow) and its appearance (default, exclusive). For an exclusive group, the different loads assigned to the group, cannot feature together in a normative combination. For default combinations, the combination generator allows the simultaneous occurrence of the loads of the same group.

The way, in which you define load cases, is decisive for the load combinations created by the generator. We recommend that you thoroughly read the chapter regarding loads and combinations in the reference manual.

For our example, we will define two load cases.

- The first load case includes the permanent loads. We will define the self-weight of all present members. The weight of the floor must be added to the beams of the first floor. The weight of the roof slabs must be added to the beams of the roof.
- The second load case contains a simplified schematisation of wind, coming from the left-hand side.

### Defining a permanent load case

Double-click  **Belasting** in the **Main** service.

Each load is classified in a load case. The program “knows” that the project does not contain any load cases yet and therefore will immediately open the load case manager.

By default, load case LC1 is created. This load is of the **Self-weight** type; the action type is set to permanent.

Change the value of the **Load type** property from **Self-weight** into **Default** and describe this load case as “**Static load**”.

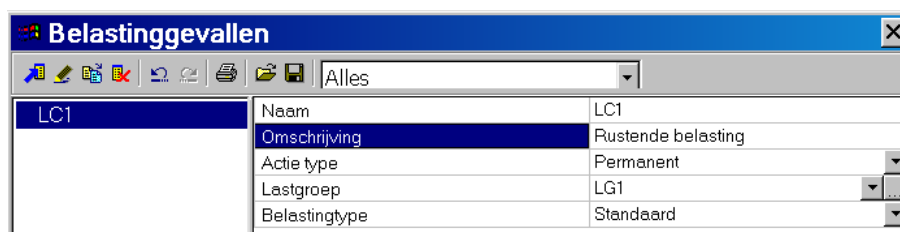
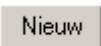

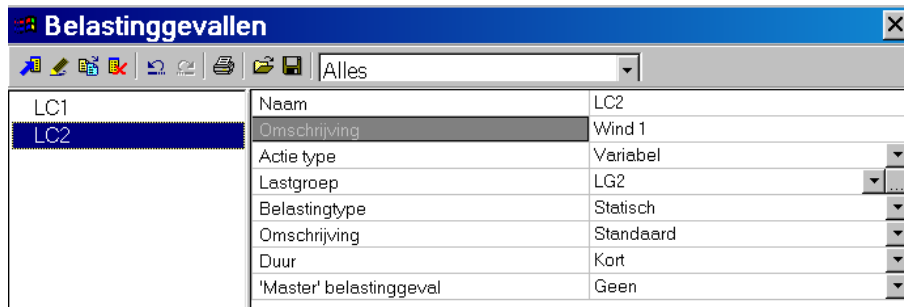


Figure 19: Load cases – Action type: Permanent – Load type: Default

---

Defining a variable exclusive load case

1. To create a second load case, click  or .  
LC2 is created and its properties are shown.
2. Change the **Action** type into **Variable** and enter “**Wind1**” in the **description** field.



**Figure 20: A variable load case**

---

Load groups determine the generated combinations

Each load is entered into a group. These groups will influence the combinations that are generated as well as the code-dependent factors that will be applied. The following logic is adopted.

Variable load cases, which are not related to each other, are associated with different variable groups. For each group, you set the category of the load (see EC1). The combination factors from the Eurocode are generated on the basis of the present load groups. As soon as two load cases, belonging to different groups, occur in a generated combination, the decrease factors for transient loads will be applied.


If the load is divisible, you enter the different part as individual load cases. As long as no variable load, belonging to another group, figures in the load combination, no decrease factors must be applied. The different load cases of a divisible load therefore are associated with one single variable group.

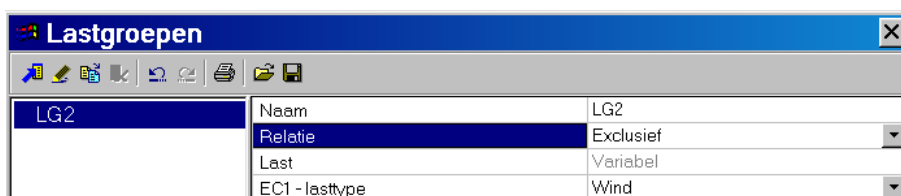
Load cases of the same type, which may not act together, are brought together in one single group, which is made exclusive. For instance, “wind x” and “wind y” are associated with one single exclusive group “wind”.

---

Setting the properties of a load group

By default, the permanent loads are put into load group (LG1).

Click  to set the properties of the variable group of load group LC2.



**Figure 21: Properties of a load group**

Set the property **Relation** to **Exclusive** and **EC1 load type** to **Wind**

Click **[Close]** to close the load case manager.

# Loads

Our example includes two load cases.

- The first load case is of the permanent action type and includes all permanent loads.
  - The self-weight of the members of the model
  - The weight of the floor must be put onto the beams of the first floor
  - The weight of the roof slabs must be put onto the beams of the roof

The second load case contains a simplified schematisation of wind, coming from the left-hand side.

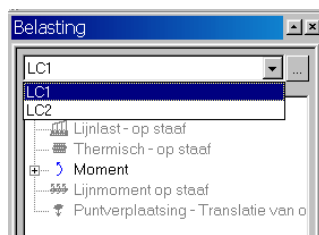
The two load cases are already defined. They only need to be filled in.

---

## Changing the active load case

Activate load case "LC1":



1. Activate LC1, by pointing the mouse cursor onto it in the list box.



**Figure 22: Active load case**

---

## Defining the self-weight through a distributed load

1. We start from a situation, in which nothing is selected. Use the  icon to cancel any previous selection.
2. Double-click  Lijnlast - op staaf . The line load on member dialogue appears on the screen.

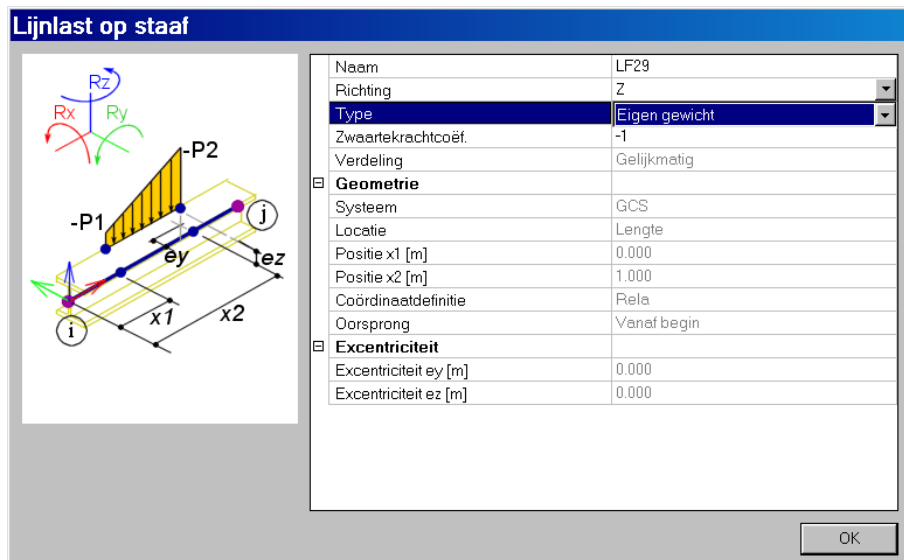





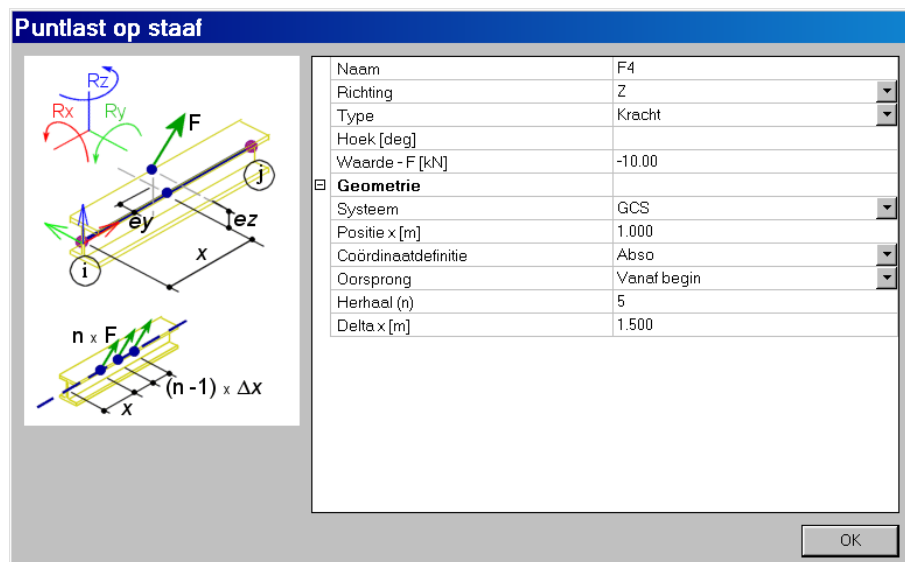
Figure 23: Line load on member

3. In the **Type** field, choose **Self-weight**  
Set the **Gravity coefficient** to **-1**.
4. Confirm with **[OK]**.
3. Select all members using the  icon.
4. Click the right mouse button to confirm your input.
5. Press the <Esc> key to cancel the selection.

#### Entering a series of concentrated loads on members

Apart from the self-weight, we will also add a number of concentrated loads to load case 1 for the modelling of the roof slabs and the floor.

1. We start from a situation, in which nothing is selected. Use the  icon to cancel any previous selection.
2. In the service **Load**, choose the function Concentrated load on member by clicking the  button.
3. The two beams of the first and the last frame are loaded with 5 concentrated loads of 10 kN. The first concentrated load of the series is located at 1 meter from the member end. The clearance between the concentrated loads amounts up to 1.5m. You obtain this configuration if you enter the parameters as shown in the figure below.



**Figure 24: Concentrated load on member**

4. Select the beams of the 3 frames.
5. Confirm your input with the right mouse button.
6. Cancel the selection by pressing the <Esc>Key.

#### Adapting the loads

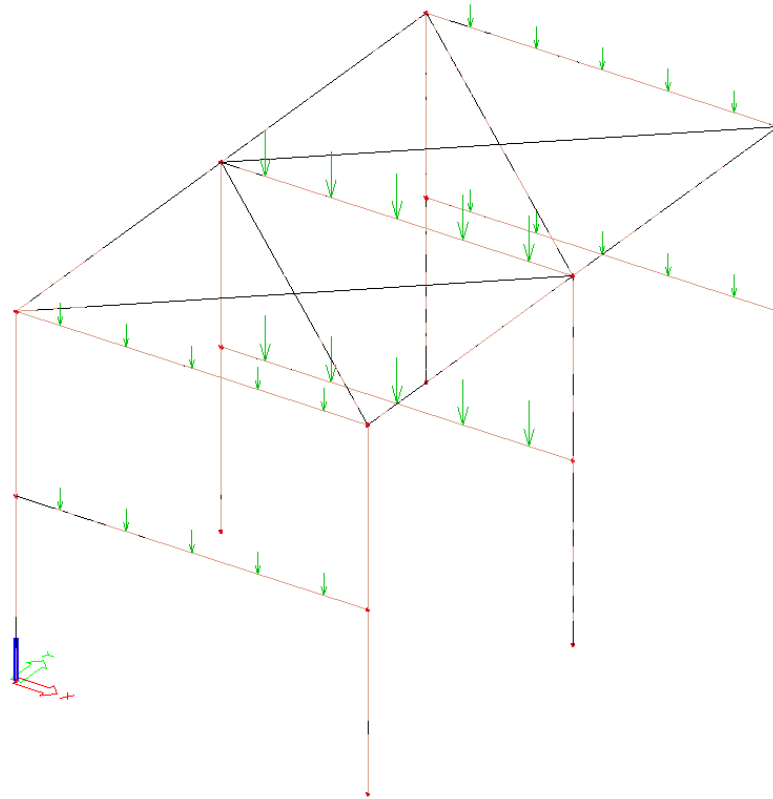
Loads are entities that can be adapted

1. Select the load(s) to be adapted
2. You can use the ingenious properties table, allowing for a fast adaptation of the properties.
3. For the outer trusses, a snow load of  $-2.25 \text{ kN/m}$  must be taken into account.

#### Adapting a series of concentrated loads

The centre beams are bearing twice as much load as the outer beams.

1. Select the two series of concentrated loads of the centre frame. The properties table shows the settings of this item.
2. In the properties table, change the value of the force into  $-20 \text{ kN}$ .



The first load case now being complete, we will complete the second load case. Activate the second load case.

Click in the field of the list box and choose **LC2**.



Now, load case **LC2** is active.

The second load case contains a simplified representation of the left-hand side wind load

The columns of the first and the last frame are loaded with 1.5kN/m

The columns of the centre frame are loaded with 3 kN/m

At the roof trusses of the outer frames, the suction force amounts up to 2.5 kN/m

The roof truss of the centre frame is submitted to a suction force of 5kN/m

Below, you see the image of the load to be defined.

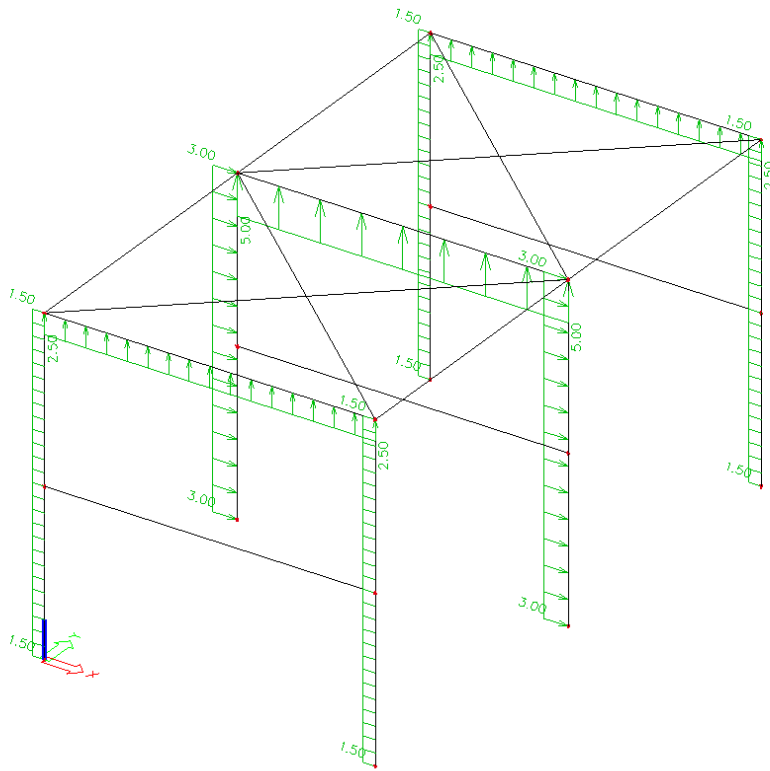




Figure 25: The simplified load case wind 1

Entering a distributed load.

1. We start from a situation, in which nothing is selected. Use the  icon to cancel any previous selection.
2. Double-click  Lijnlast - op staaf . The Line load on member dialogue is opened.
3. Set property **Direction** to **X**  
Set property **Type** to **Force**  
Set property **Value** to **1.5**

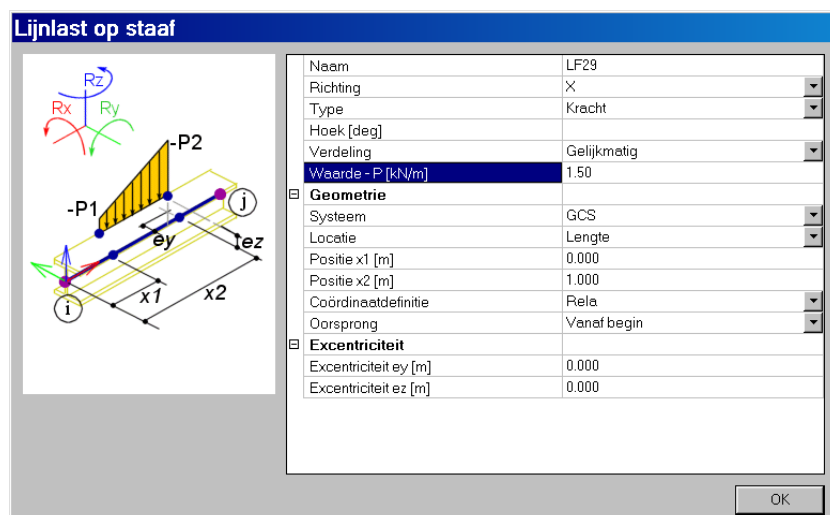


Figure 26: Menu of an equally distributed load

4. Confirm with **[OK]**.
5. Select the four columns of the outer frames.
6. Click the right mouse button to confirm your input.
7. The elements of a new input are selected. Press the **<Esc>** key to cancel the selection.

---

#### Copying loads

Load cases are entities, which can be copied. In this project, we do not need this action. If you want to copy loads, follow the procedure below.

1. **Select the load** to be copied.
2. Use the button **Copy additional data**





3. The command line asks you to which members you want to copy the selected load.



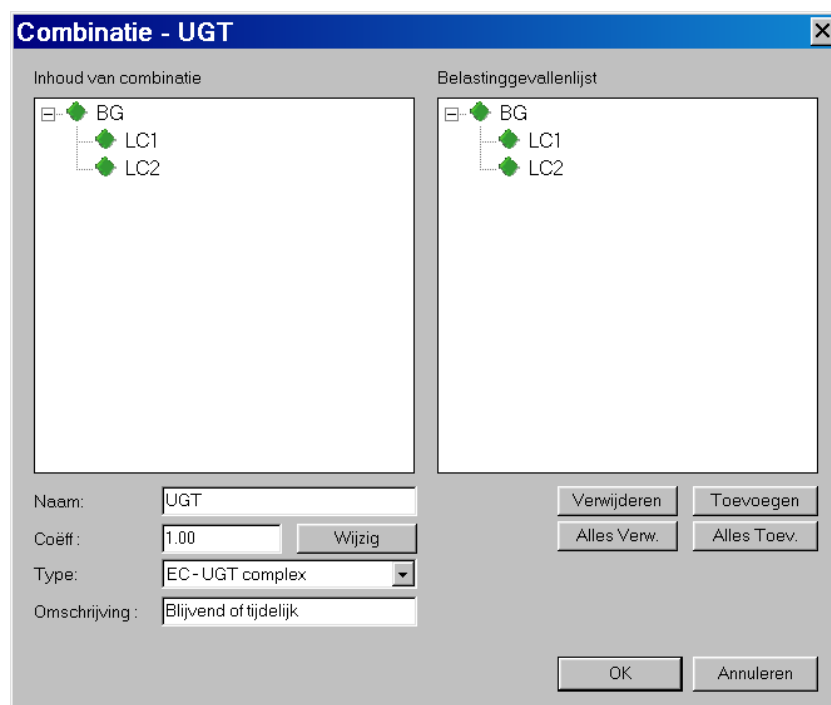
# Combinations

When all load cases are defined, we can define the combinations.


Defining combinations:

1. Choose in the service Menu  Belastinggevallen, Combinaties >  Combinaties to enter linear combinations of the defined load cases.

Since there are no combinations, a new combination is created immediately after opening the **Combination manager**.



**Figure 27: Combination - CO1**

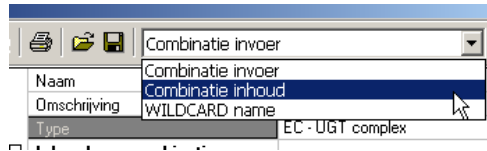
2. By default, a **Self - UGT combination** is created; this implies that the program will automatically make all possible combinations with these load cases. It will take the type (Proper weight, Permanent, Variable) into account and multiply the different load cases with the entered factors .
3. Change the **Type** in **EC – UGT Complex**  
Now, the combinations are automatically generated to EC.
4. Click **[Add all]** below the list of load cases to add all load cases to the combination and click **[OK]**.  
You will return to the Combination manager.
5. Apart from this combination, we also add the combination **EC-BGT** for the serviceability check (deformations). Click **[New]** or  , change the **Type** in **EC-GGT-Complex seldom** and click **[OK]**.
6. Click **[Close]** to close the combination manager.

---

Load rules

You can always consult the content of the combinations on beforehand.

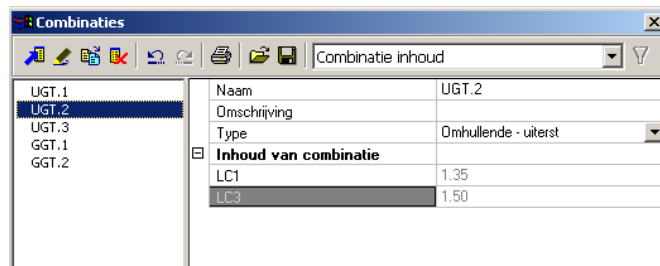
1. Set the list to Combination content



2. Now, the combinations menu contains the different load rules, with which the real combinations will be generated.

For UGT.2, which is selected in the figure below, two combinations will finally be generated:

1.35 LC1  
1.35 LC1 + 1.5 LC2



---

# Calculation

## Linear calculation

The calculation now being complete, we can launch the calculation.

---

Carrying out the linear calculation


1. Double-click  Berekening, Net >  Berekening .

The dialogue **EE-solver, net generation settings** appears.

2. Click **[OK]** to start the calculation. After the calculation, a message appears on the screen that the calculation has been carried out. Close this window with **[Close]**.


# Results

## Viewing results

When the calculation is OK, choose  Resultaten in the **Main** service to view the results of the calculation.

---

Viewing the result of forces on members, graphically represented on the structure

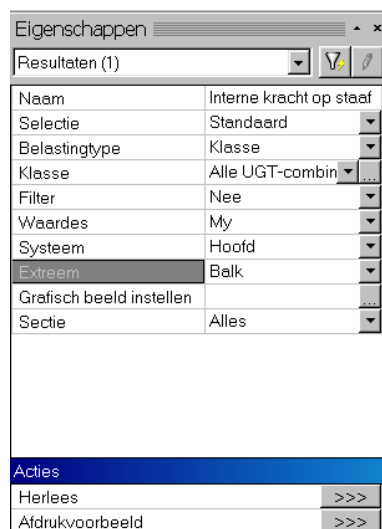
1. To view the results of forces on members, double-click  Resultaten in the **Main** service. The **Results** service becomes active.


By default, the **Internal forces** (My) are shown on all members.

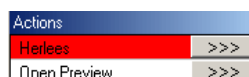
2. Select the members of one of the centre trusses.

The members are highlighted in rose.

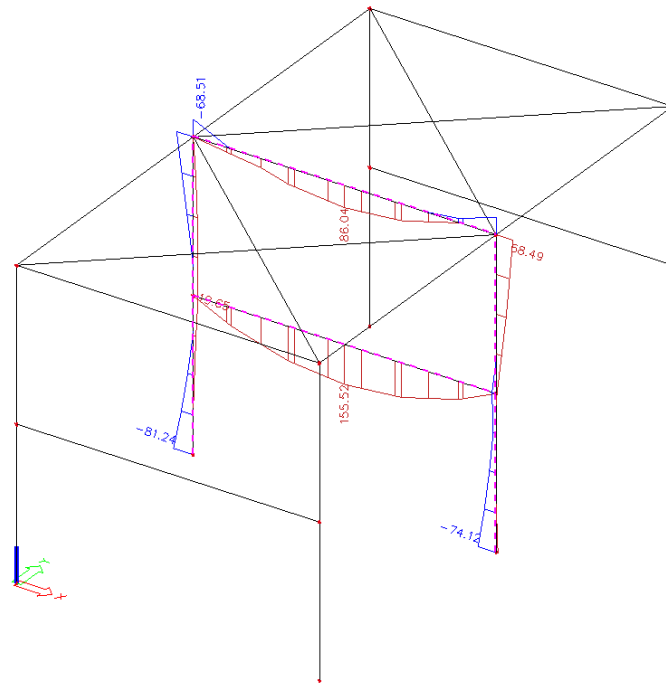
3. In the properties dialogue, configure the properties as follows:  
 Set the **Selection** property to **Default**, so that you can select yourself the elements for which you want the results  
 Set the **Load type** property to **Class** to view different combinations  
 Set the **Class** property to **All UGT-combinations** to view all UGT combinations  
 Set the **Extreme** property to **Beam** to obtain extremes per beam



4. Click  behind the **Reread** property



The beam extremes are shown for the selected members.



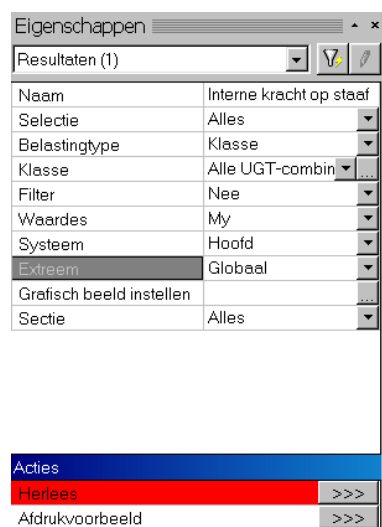
**Figure 28: Internal forces on member**


#### Viewing the result of forces on members in tabular form

This time, we will search the global extremes of the entire structure

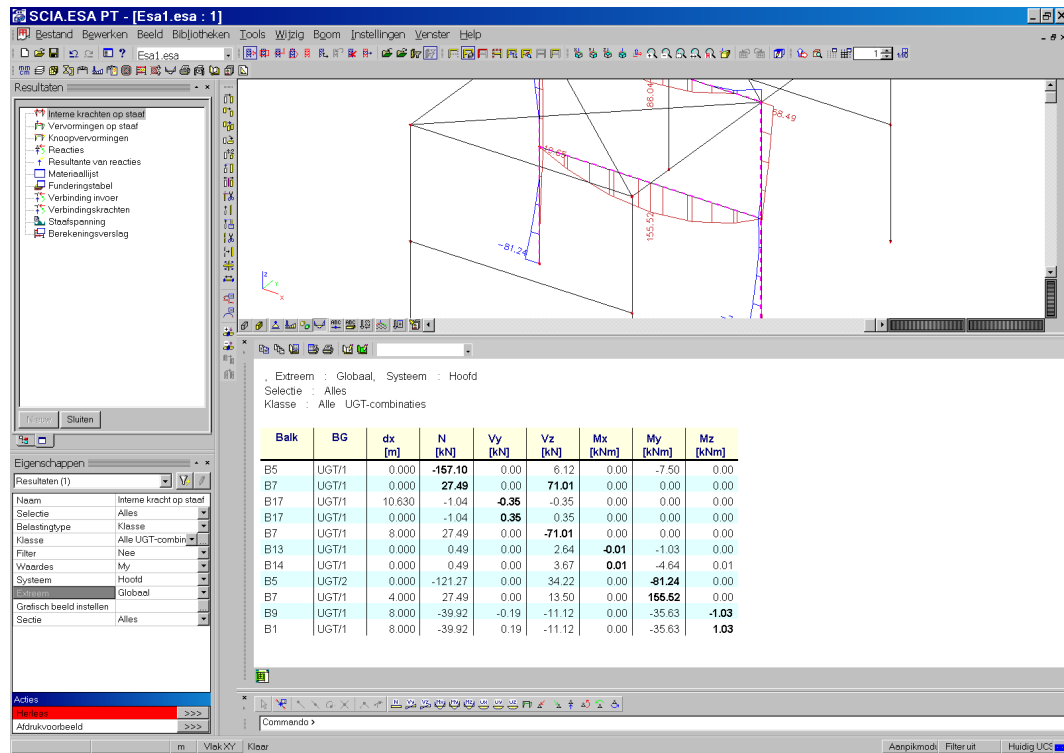
Set the properties as follows:

1. Set the Selection property to All  
Set Extreme to Global



2. Use the Print Preview service, by clicking the  button to obtain a numerical output of the internal forces on the screen. A new frame will appear between the graphical part and the command line.

5. To see a larger part of the results, enlarge this frame.



6. Click the cross in the upper left corner of this frame to close it.

# Document

## Documentation

When the calculation is finished, we can generate a calculation note.

---

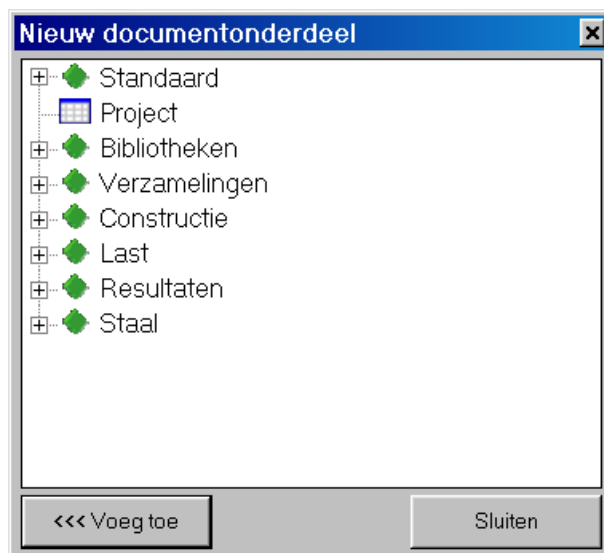
### Creating documentation

1. Choose  **Document** in the toolbar.

The screen will change and the Document service will be opened. The Document service will show the content of the document.

2. Click with the right mouse button in the white part of the **Document** service.

The dialogue **New document part** appears.



**Figure 29: New document part**

3. Choose **Project** and click **[Add]**.
4. Open the **Libraries** group and select **Section**. Add it.
5. Open the **Construction** group and select **Members**. Add twice.
6. Open the **Results** group and select **Internal forces on member**. Add.
7. Close the dialogue. Click **[Close]**.

All chapters will appear in the service to the left side of the screen. You can move an element by dragging it with the mouse. At the right side of the screen, a print preview of the document is shown. You can browse the document by clicking the elements in the menu to the left or using the scroll bar at the right-hand side of the screen.

---

### Modifying the table of contents

1. Move the chapter **Members** and drag it towards the **Sections** chapter.

The **Members** item becomes a paragraph in the **Sections** chapter. For each section, the members are shown with that section.



2. Move the chapter **Internal forces on member** and drag it towards the second added chapter **Members**. The internal forces are shown for each member.

---

### Modifying the properties of a table

1. Select the paragraph **Internal forces on member**.


The properties are shown.

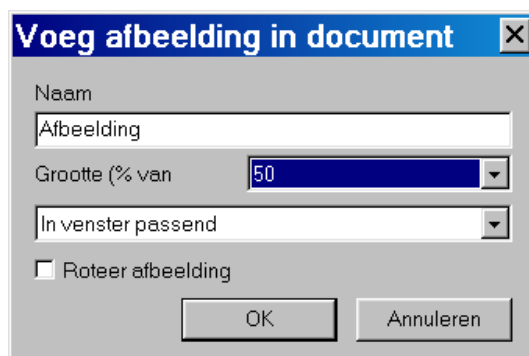
2. Choose for the **Load Type** property the value **Class** and for the **Extreme** property **Beam**.

For each member, the results of that member are shown for the class all ULS combinations and only the extremes of the beam are shown.

---

### Adding an image to the document

1. At any time, you can click  in the toolbar and choose **Image to document** to send the content of the graphical window to the document. The following dialogue will appear.



**Figure 30: Add image to document**

2. In the **Name** field, you can enter the text for the image. The size of the image (**Size (% of page)**) can be chosen as a fraction of a full page (50% is half a page). The image can be **fit to the window**, **fit to the range** or you can zoom everything (**Zoom all**). Furthermore, you can rotate the image ('landscape').
3. If you click **[OK]**, the image is inserted in the document.



# Cad form

## Cad form

By means of the Cad form options, you can generate true drawings of the structure. This means that, apart from the computational model, the program can contain the drawing model as well.

The options to generate a Cad form are only available when you have selected the **Cad form** option on the **Functionality** tab of the project data.

---

### Parameters to set a Cad form

The main parameters to configure the drawing model are:

- The priority of a member:

The priority of a member is set in the member properties. This is a value, which is proportional to the importance of the profile. In this way, you determine the members that should be drawn continuously in the graphic representation. A member with a smaller priority is cut off of a member with a higher priority value.

A series of default priorities are implemented, but you can set the priority yourself as well

- Geometric parameters:

For the properties of a member, you will find in the Cad model group a number of parameters allowing for a correct alignment of the element on the structure. These parameters are independent of the default parameters for rotation and eccentricity.

---

### Setting a default priority type

1. Select the columns
2. Set the Type property to Column(100)

The beams of the different frames are set to beam (80). The roof bracings are set to Horizontal bracing (0)


---

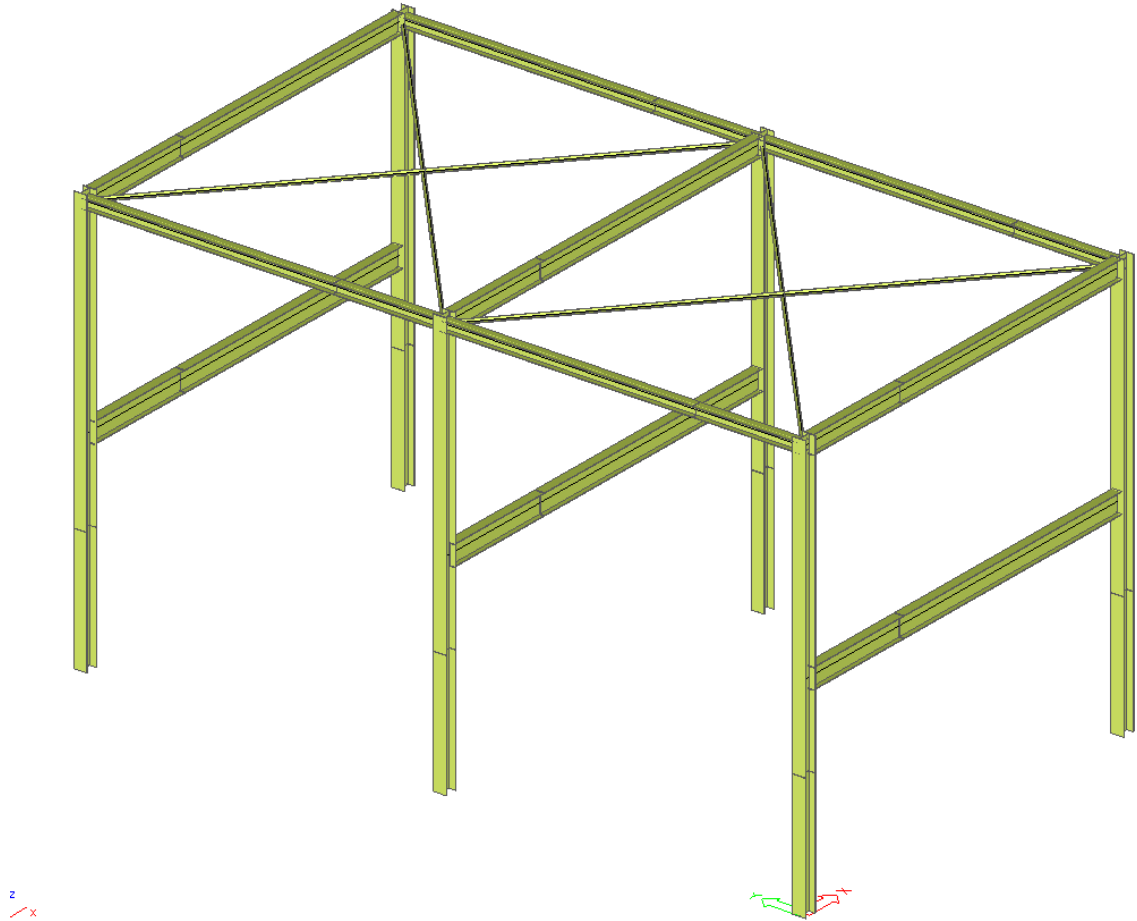
### Setting a priority value

1. Select the beams connecting the different frames.
2. In the properties table, in the Cad model group, set the **Priority definition** property to **manual**.
3. Set the value of the property **Priority value** to 75.

---

### Generating the Cad model

After having set all parameters, you can generate the Cad model using the option Generate Cad form (button ) in the toolbar. In the figure below, you can see an image of the Cad form.




**Figure 31: Image of the CAD model**




# Code check

## Steel code check


The steel code check is implemented for different codes. In this Tutorial, we will only discuss EC.

For a stress analysis of the steel, the program must dispose of results. The options for the steel code check can be found in the Main service under  **Staal**.


The following checks are provided:

-  **Controle** : This option includes a stress analysis and a stability check.
-  **Slankheid** : This option includes a slenderness check
-  **Relatieve vervorming** : This option includes a check of the relative deformations

The program is able to apply local reinforcements. The following possibilities are implemented.

-  **Console**
-  **Kipsteunen**
-  **Verstijving**
-  **Verbindingen**
-  **Beplatingen**

Before you can start a check, the element must be properly configured. The strut type and the system length are highly important.


- The default configuration of strut type is implemented and can be set through the option  **Instelling**.
- The program calculates the system lengths. These system lengths can be represented through the image parameters (**image parameters > entities > buckling lengths**)

The program contains the library objects 'buckling', enabling you to overwrite the default settings for strut type, system length and deformation length.

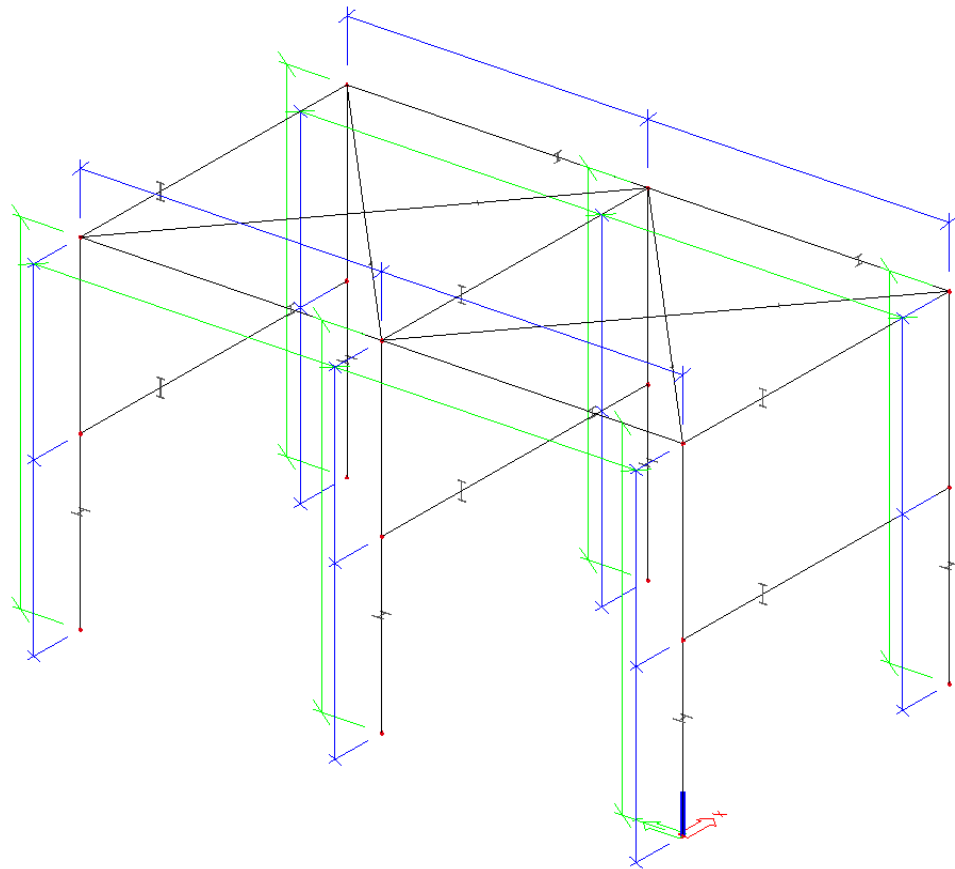


---

Representing the system lengths on the structure

1. Start with a situation, in which no object is selected. Click to this end in the toolbar on the option **Cancel selection** (button ) . If nothing was selected, the button will not be active.
2. Click with right mouse button in the graphic part. The menu **Image parameter setting** will be opened.
3. Choose the Entities tab.
4. In the group **Construction**, choose the option **Buckling lengths**

The system lengths are represented on the structure. In the image below, the representation of the section was activated as well.



**Figure 32: structure with system lengths**

---

Notes:


*The system lengths are only represented on elements, which are connected with other members in their field. For elements consisting of one single part, such as the horizontal beams of the frames in our structure, no system lengths will be represented.*

*The system lengths are automatically cut off in the hinges.*

---

---

## Default configuration

The default configuration for all members can be set in  Instelling

- The default strut type is set in the tab: **Setting for steel member check.**

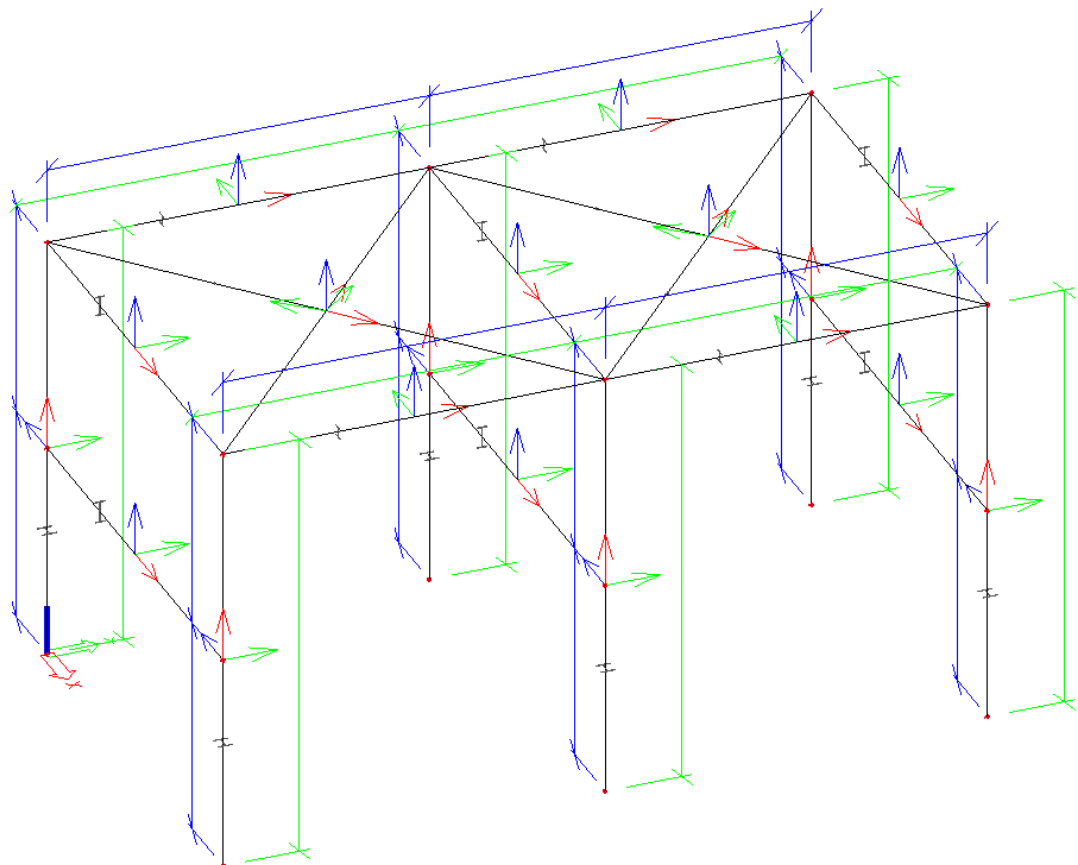
If you take YY non-strutted, by activating the radio button, this implies that the strong axis of the profile will be non-strutted.

- The default system lengths and deformation lengths are set in the tab buckling settings.
- The requirements for the relative deformation check are set in the tab setting of the relative deformations.

---

## Configuration of the columns

Below, you will find an image of the structure. The following image parameters are activated: member names, member LCS, section and system lengths.




The figure shows the default configuration.

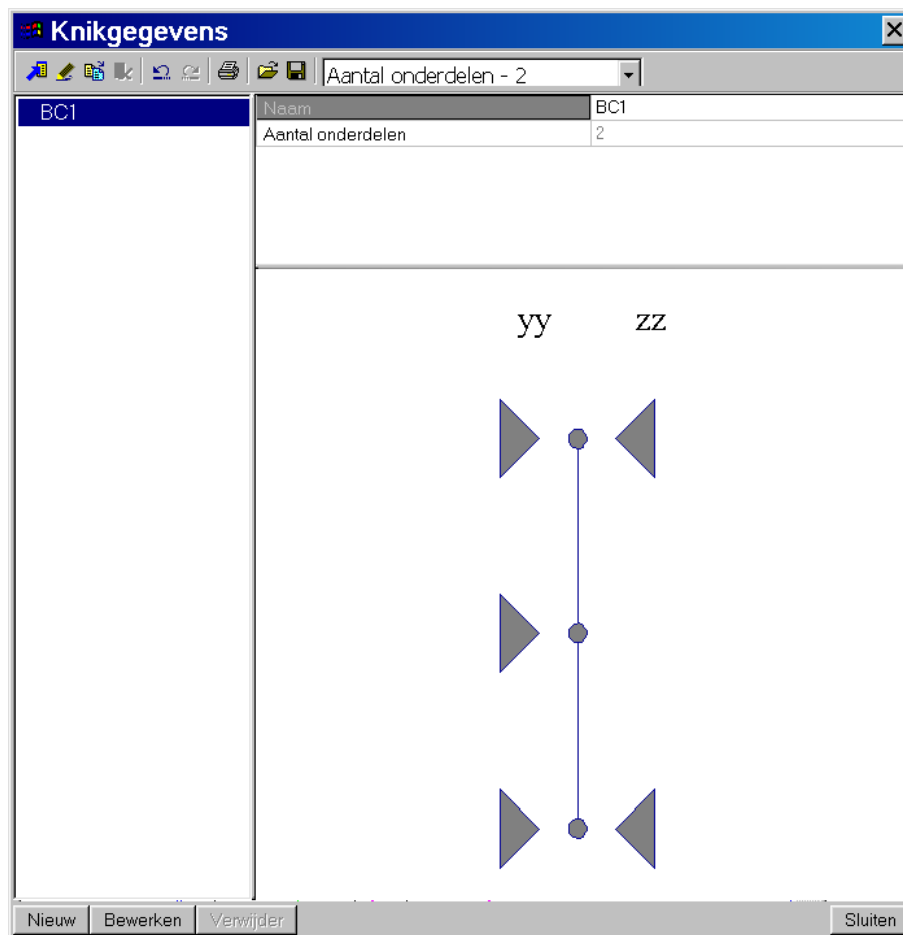
The program finds for the system lengths of  $L_y$  (for the strong axis) a length of 4m and for  $L_z$  (the weak axis) a length of 8m.

The default strut type is strutted for the strong axis and non-strutted for the weak axis. This does not comply with our structure. In this model, the columns of the strong and the weak axis are non-strutted. This can be set using adapted buckling parameters.


### Setting the buckling parameters for an element

The program contains a library with buckling configurations. A number of buckling parameters can be set. Next, the setting can be linked to different members.

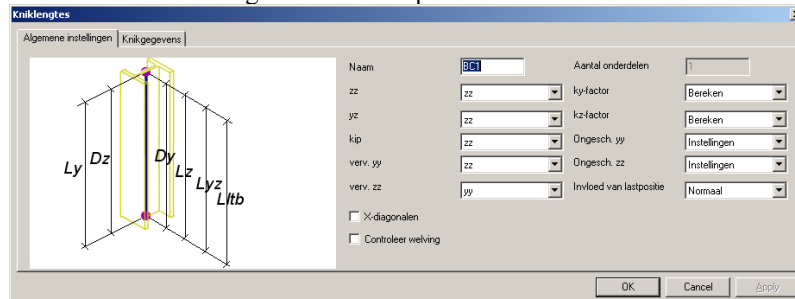
1. Select a column with the mouse cursor
2. The **Buckling lengths** property of this member indicates the value default. This means that the default configuration is linked to this element.
3. Click the  button behind default to create a new series of buckling parameters. Since the library does not contain any objects yet, the **Buckling data** menu is opened. A new series of buckling parameters is created. By default, the program had found that this object consists of two parts, whereof the Y-axis is connected at half-height. The new object is a copy of the default configuration and is called BC1. Below, an image of the library containing BC1.



**Figure 33: Buckling data**

4. Click the  button to set the properties for this object. The menu Buckling lengths will be opened.

5. The tab General settings must be interpreted as follows:



The field next to **Name** contains the name of the object. Enter a recognisable name.

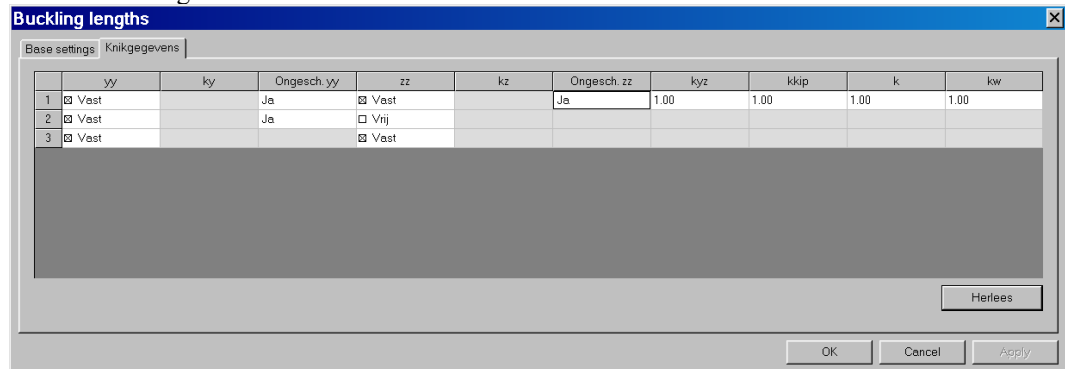
The value **yy** represents the system length of  $L_y$  and does not figure in the menu, since the system length for  $L_y$  is always calculated by the program.

The value **zz** represents the system length for the weak axis. We can take it equal to  $L_y$ . This means that the value of the **zz** fields must be set to **yy**. In our model, we want the program to calculate the system length for  $L_z$  as well. Enter value **zz** for the property **zz**.

The other system lengths,  $L_{yz}$  and  $L_t$ , are taken equal to  $L_z$ .

The relative deformation in Y-direction has the system length of  $L_z$  as basis. Therefore, set the value of **verv.yy** to **zz**. The same applies to **verv.zz**.

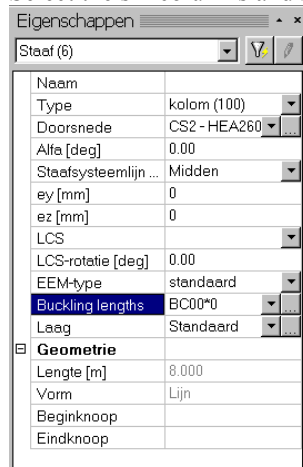
6. The tab buckling data is set as shown below:



The column has a length of 8 m. The strong axis has a connection at 4m. This one is fixed at the start (1) on the location of the connection in the field (2) and at the end(3).

The member parts between (1) and (2) and between (2) and (3) are non-strutted. Therefore, we set the property **Non-strutted** to **YES**. (Same consideration for the weak axis.)

7. Confirm your input with the **[OK]** button.
8. Select the six columns and set the **Buckling lengths** property to BC00\*0





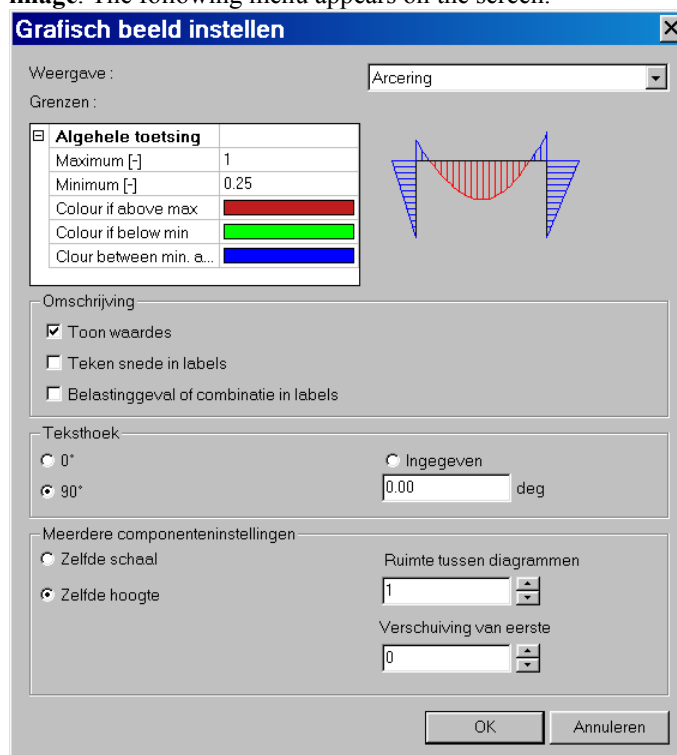
Also configure the other elements

First, we will carry out a global check. We want the result to be represented in graphic form, drawn directly on the members of the structure. In this way, we can easily localise possible problems.

### Configuring the graphic output

Before carrying out a check, we will configure the graphic output. This will be done just once.

1. Select with the mouse the option  **Controle** in the Steel service.
2. The properties table shows the properties of a check. Click the  button next to **Set graphic image**. The following menu appears on the screen.



3. The result of a check is shown as a unity check. The program allows the configuration of two limit values, an upper limit and a lower limit. Sections of members that do not comply with this code have a unity check higher than 1. This will be our upper limit. Members whereof the unity check is inferior to 0.25 are badly used. This is our lower limit. The upper and lower limit unanimously delimits 3 zones. We can associate a colour to each one of these zones. A unity check of less than 0.25 means that the member is poorly used. We choose a blue colour. A unity check between 0.25 and 1 shows that the result is optimal. We take a green colour. For a unity check of more than 1, the member is not code-compliant. We choose a red colour.

Now configure the values and colours in the menu **Set graphic image**.

The graphic result of a check on sections of members will now always be represented in one of the chosen colours.

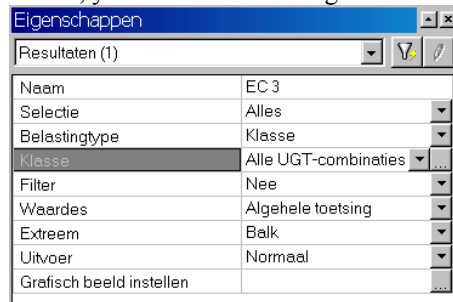


## Graphically representing the result of a check

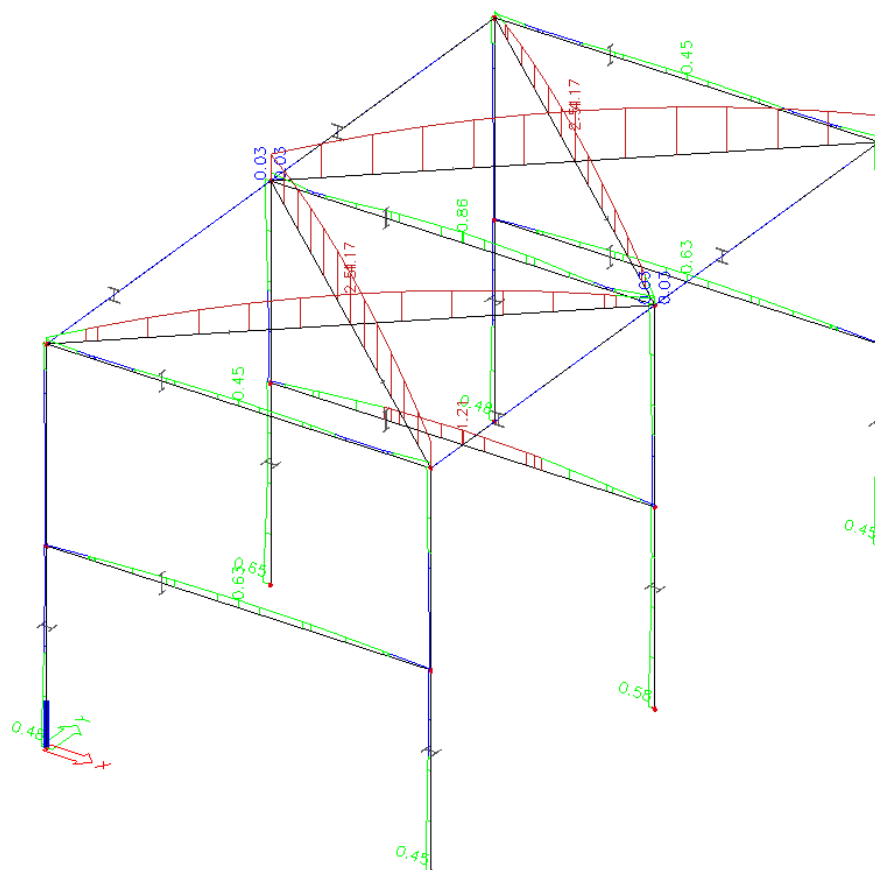
The check must be carried out for all members all UGT combinations.

1. Set the properties of the check.  
 Set **Selection** to **All**: we want to check all members  
 Set **Load type** to **Class**: we want to consider a set of actions  
 Set **Class** to **All UGT- combinations**: we want all UGT combinations

Below, you will see the settings.



2. Click **>>>** behind the property **Reread**.



The result shows that there is a problem with the floor beam of the first floor. We also remark that the roof bracing do not satisfy.

We analyse the problem of the floor beam by means of a section check.

---

## Section check

For a section check, you only need to set the combinations or load cases for which the check must be carried out.

- 1 In the properties table, set the **Load type** property to **Class** and the **Class** property to **All UGT-combinations**.

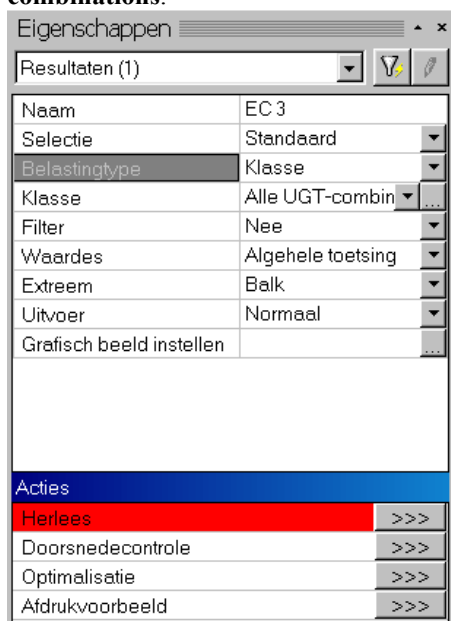
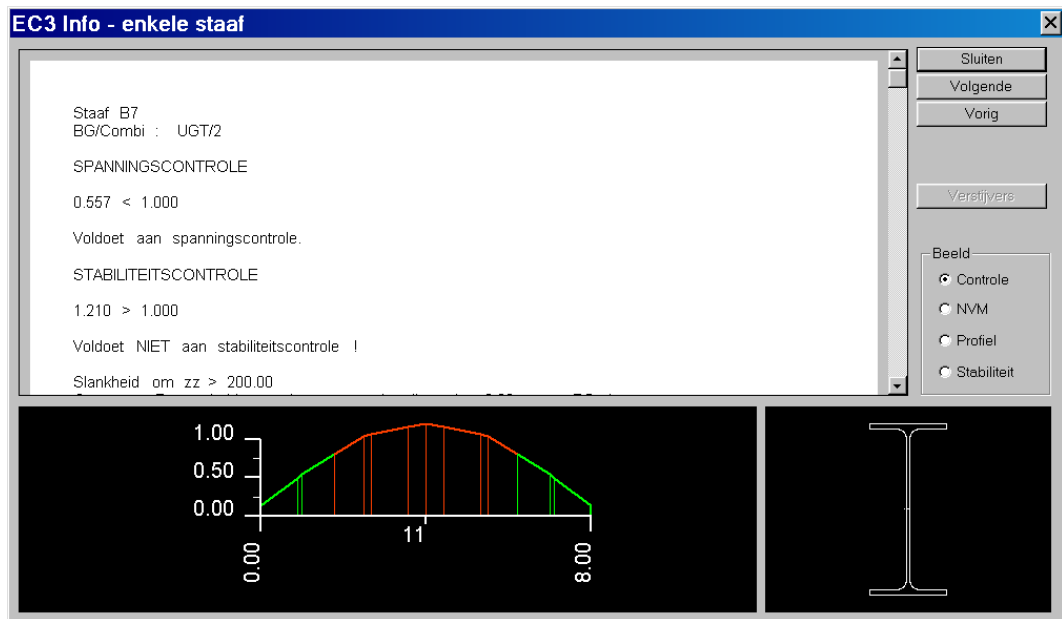


Figure 34: Properties steel check

- 2 Click **>>>** behind Section check. The command line will ask you for which entities the check must be carried out.
- 3 Select the floor beam of the second truss. The following menu will be opened




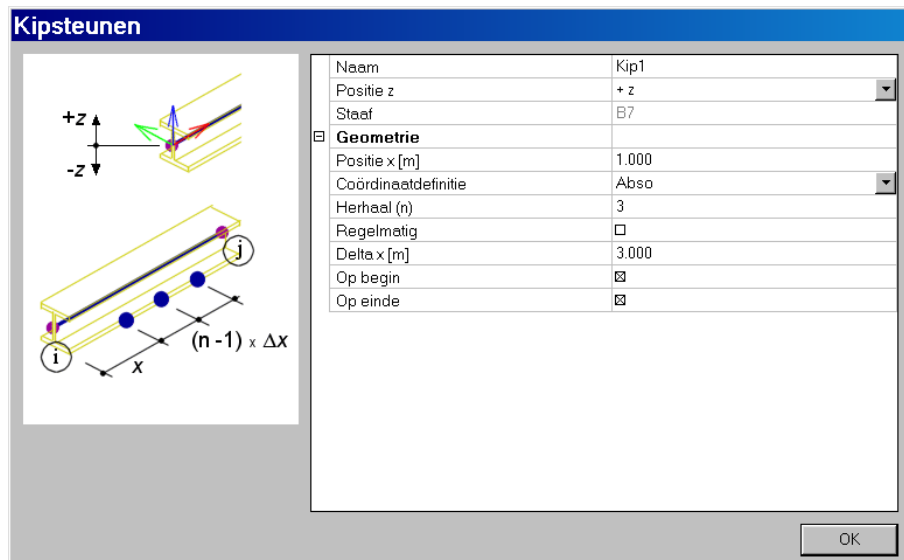
- 4 In the **Image** group you can choose between:
- Check** if you want to obtain a concise report of the checked member
  - NVM** to view the section forces of the representative combination.
  - Profile** if you want to see a detailed report of the stress check.
  - Stability** if you want a detailed report of the stability check of the element.

The report shows that the stability check does not comply. The member buckles.

#### Adding buckling supports

In reality, the upper flanges of the floor beams are fixed through floor slabs, which constitute a stability against buckling. In the model, we will add buckling support to the upper flange of the beam.

1. Select  Kipsteunen
2. We will define three buckling supports. The first at 1 meter from the start of the beam. The intermediate distance of the supports is 3m. The configuration is represented in the image below.



3. Select all floor beams of the first floor.
4. A new section check shows that the element now complies.

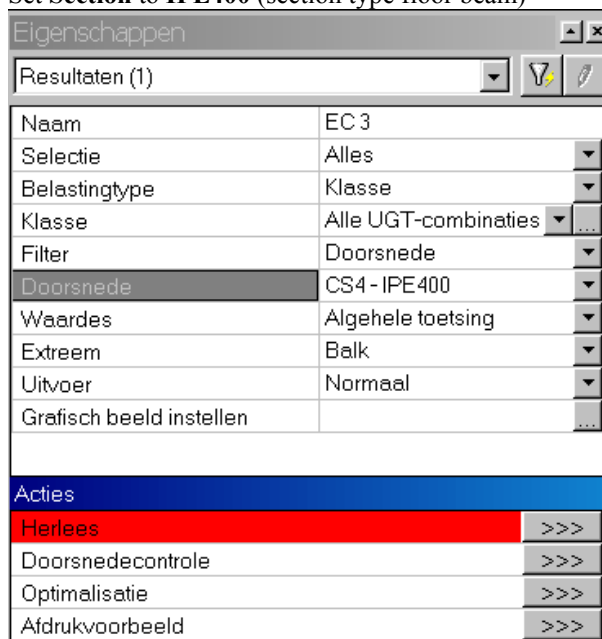
Through an optimisation, we can found out if it is possible to lower the profile types.

#### Carrying out an optimisation

1. First, we have to set the properties to carry out an optimisation.

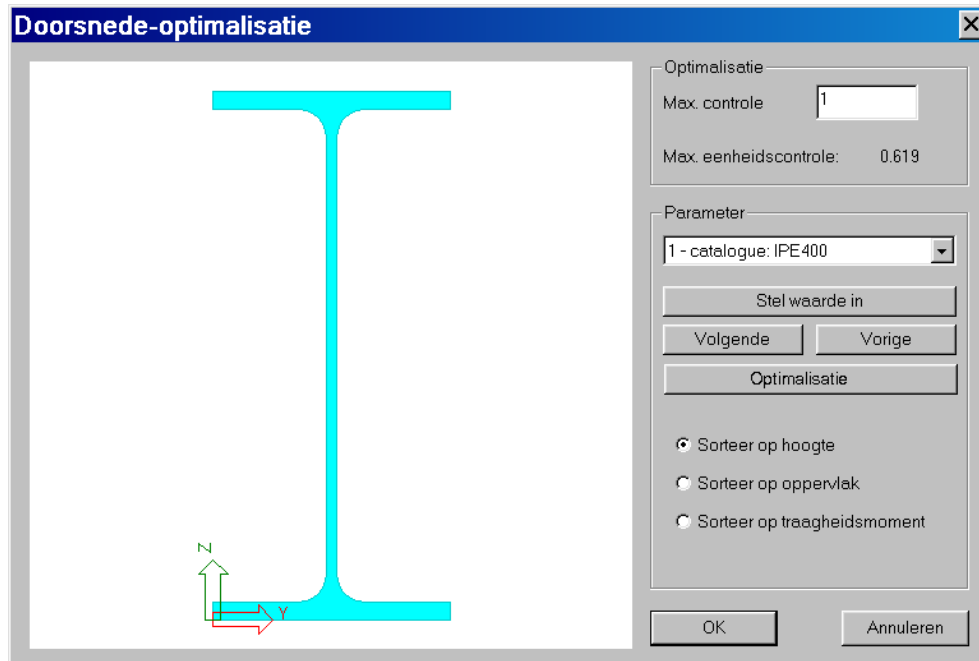
Set **Filter** to **Section**

Set **Section** to **IPE400** (section type floor beam)




2. Click **>>>** behind optimisation. The menu section optimisation appears.

- Click the button **Optimisation** to start an optimisation. The program will search the lowest IPE profile that still complies with the check. The new profile type is immediately assigned to the beam.



The program thinks that an IPE360 is an appropriate choice. Leave this menu by clicking **[OK]**

- Since the profile has been changed, the construction needs to be recalculated, before we can continue. Start the recalculation using the **Hidden calculation** button  in the toolbar. Now, the program can continue

You can further optimise starting from the new forces distribution.

We want a print preview with a short report of each member

---

Print preview of a code check

- In the properties table, set the parameters as follows:

Eigenschappen

Resultaten (1)

|                          |                   |
|--------------------------|-------------------|
| Naam                     | EC 3              |
| Selectie                 | Alles             |
| Belastingtype            | Klasse            |
| Klasse                   | Alle UGT-combin   |
| Filter                   | Nee               |
| Waardes                  | Algehele toetsing |
| Extreem                  | Balk              |
| Uitvoer                  | Kort              |
| Grafisch beeld instellen |                   |


Set **Selection** to **All**  
 Set **Filter** to **No**  
 Set **Output** to **Short**

Open the report by clicking >>> behind Print Preview.

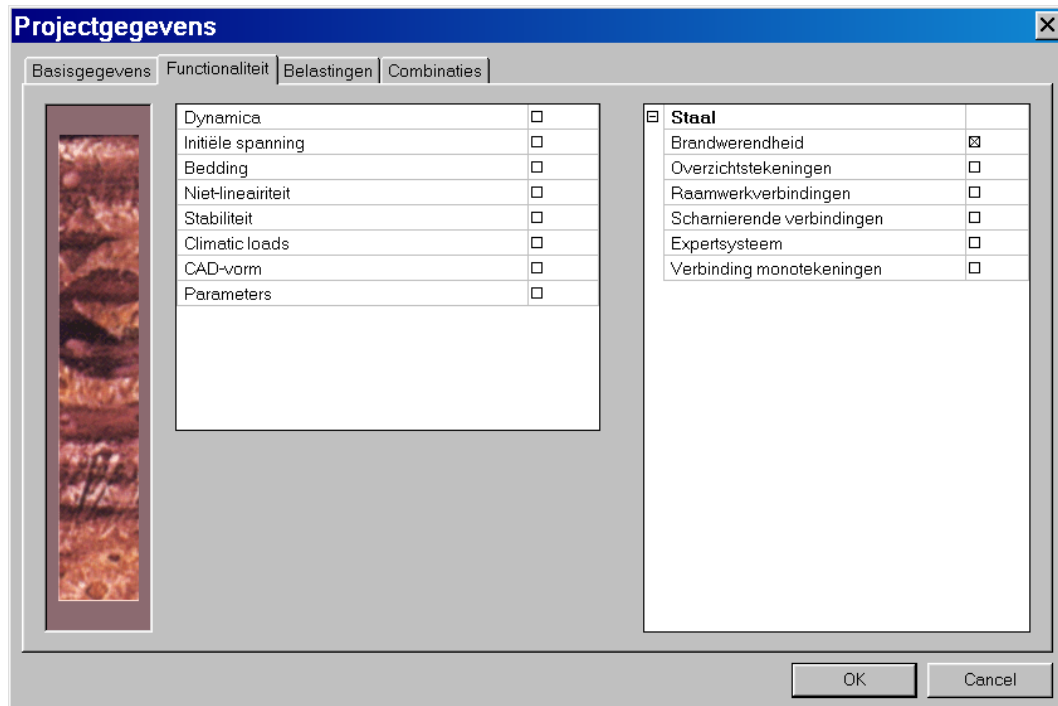
| Balk | BG    | dx<br>[m] | Algehele toetsing<br>[-] | Doorsnedetoetsing<br>[-] | Stabiliteittoetsing<br>[-] |
|------|-------|-----------|--------------------------|--------------------------|----------------------------|
| B1   | UGT/1 | 0.000     | 0.48                     | 0.11                     | 0.48                       |
| B2   | UGT/1 | 0.000     | 0.45                     | 0.10                     | 0.45                       |
| B3   | UGT/2 | 4.000     | 0.42                     | 0.37                     | 0.42                       |
| B4   | UGT/2 | 4.000     | 0.45                     | 0.04                     | 0.45                       |
| B5   | UGT/1 | 0.000     | 0.65                     | 0.41                     | 0.65                       |
| B6   | UGT/1 | 0.000     | 0.58                     | 0.37                     | 0.58                       |
| B7   | UGT/1 | 4.000     | 0.80                     | 0.71                     | 0.80                       |
| B8   | UGT/2 | 4.000     | 0.86                     | 0.39                     | 0.86                       |
| B9   | UGT/1 | 0.000     | 0.48                     | 0.11                     | 0.48                       |
| B10  | UGT/1 | 0.000     | 0.45                     | 0.10                     | 0.45                       |
| B11  | UGT/2 | 4.000     | 0.42                     | 0.37                     | 0.42                       |
| B12  | UGT/2 | 4.000     | 0.45                     | 0.04                     | 0.45                       |
| B13  | UGT/1 | 7.000     | 0.03                     | 0.01                     | 0.03                       |
| B14  | UGT/1 | 0.000     | 0.03                     | 0.01                     | 0.03                       |
| B15  | UGT/1 | 7.000     | 0.03                     | 0.01                     | 0.03                       |
| B16  | UGT/1 | 0.000     | 0.03                     | 0.01                     | 0.03                       |
| B17  | UGT/2 | 5.315     | 2.51                     | 1.12                     | 2.51                       |
| B18  | UGT/1 | 5.315     | 4.17                     | 1.13                     | 4.17                       |
| B19  | UGT/2 | 5.315     | 2.51                     | 1.12                     | 2.51                       |
| B20  | UGT/1 | 5.315     | 4.17                     | 1.13                     | 4.17                       |



## Fire resistance check

The program includes a module to carry out a fire resistance check for steel structures.

The tools to configure and check the fire resistance are gathered under the option  **Staal** in the **Main** service.

The options are only available if you have selected the option **Fire resistance** on the **Functionality** tab of the project data.




The basic configuration for fire resistance can be found under  **Stalen** in  **Instelling**.

The definition of fire resistance is an additional data of a member (just as load, hinge...). When carrying out the check, only those members, for whom the fire resistance was defined, will be considered.

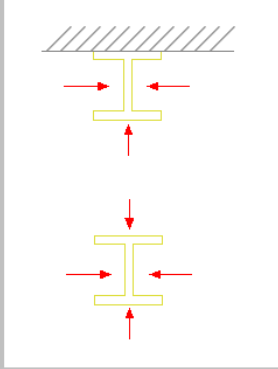
The fire resistance check must be carried out with UGT - incidental combinations. These combinations only make sense in case of an incidental load. In this example, we will NOT create a third load case. If we would create a third load case, we would be unable calculating the project in demo mode. We will discuss the procedure, although with an ordinary UGT combination

### Defining the fire resistance

1. Double-click the option  **Brandwerendheid** in the **Main** service.  
The fire resistance menu appears on the screen.
2. Set the parameters in the **Fire resistance** menu.

In our example, we will install **Plate material of 10mm**  
We will check an 1800-s fire resistance.

**Brandwerendheid**

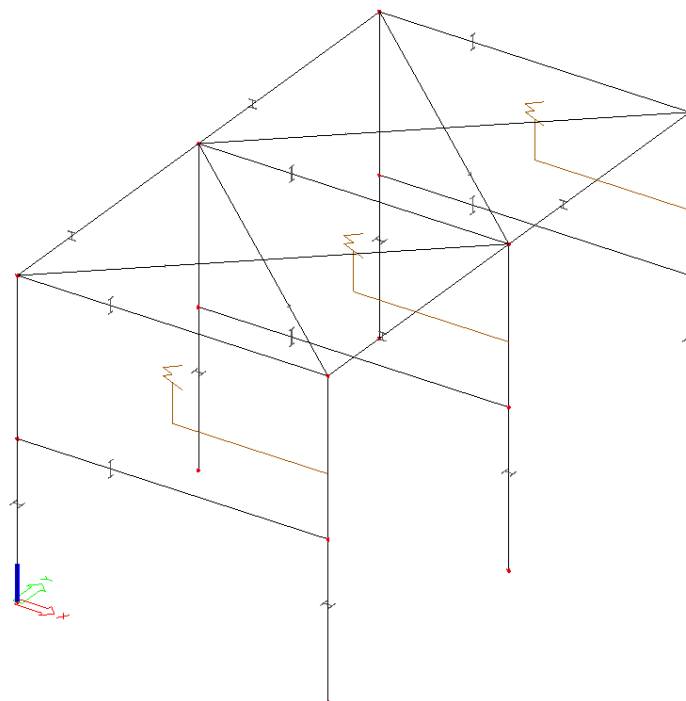


|                      |                |
|----------------------|----------------|
| Naam                 | FR1            |
| Tijdweerstand [sec]  | 1800.00        |
| Knikverhouding $k_y$ | 0              |
| Knikverhouding $k_z$ | 0              |
| Brandblootstelling   | Alle zijden    |
| Protectie            | Ja             |
| Isolatie LIB         | Plaatmateriaal |
| Dikte [mm]           | 10             |
| $k_2$                | 0.85           |
| Staal                | B2             |

OK


If you set the buckling ratios  $k_y$  and  $k_z$  to zero, the buckling ratios defined for steel will be used. If you redefine  $k_y$  and  $k_z$  in this menu, the default buckling ratios will be overwritten.

3. When the parameters are configured, confirm with **OK**.
4. Select the members that must feature the defined fire resistance.




On the selected members, the label is represented.



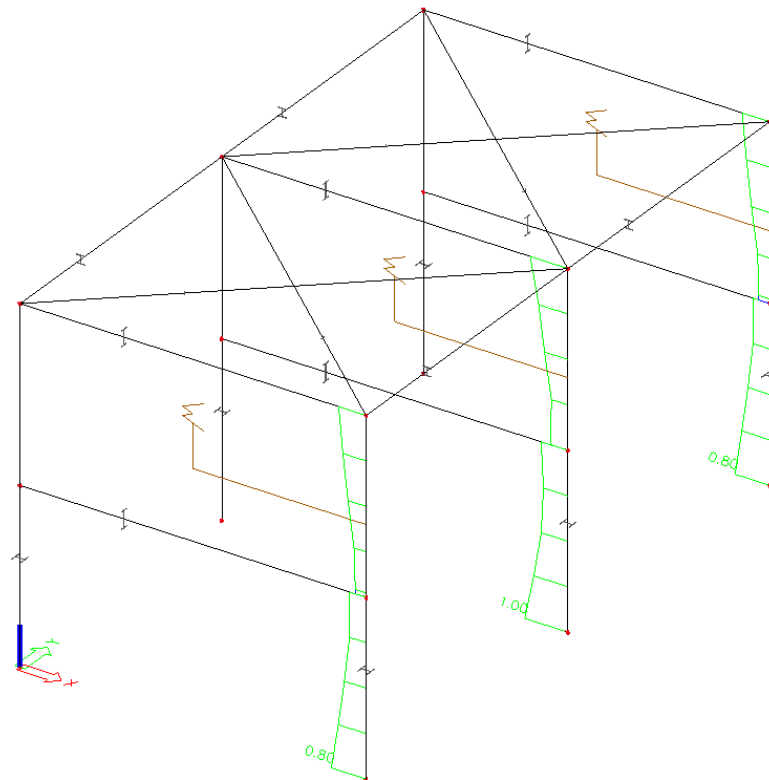
1. Double-click the  **Controle - brandwerendheid** option in the **Steel** service to activate the function.
2. Set up the Properties table for the result configuration.



We will choose a general check with one extreme per beam, for UGT combinations.

3. Start the check by clicking the  button behind **Reread**

In this way, the result will be displayed in a graphic manner.



# Connections Module

## Connections

The Connections module of SCIA.ESA PT has been developed to design frame connections from steel constructions.

The options to define and calculate the connections are normally not activated. If you want to activate the connections modules, you must select them in the project data menu on the functionality tab. When selecting a functionality for a type of connection, the CAD form option is automatically selected as well.

The CAD form option serves to generate a drawing model of the structure. For more details, please refer to the Cad form chapter.

A connection is designed with the results of UGT combinations. If the results are no longer available, you must start a calculation.

The priority assigned to the members to be connected is highly important. For instance, if you want to design a beam-to-column connection, the column should have a higher priority than the beam. For a beam-to-beam connection, the beams must have an identical priority. For more information regarding the priority, please refer to the Cad form chapter.

We will design a bolted connection strong axis-to-strong axis in node K4.

---

### Copying bolts from the system database

The program integrates a bolt library, containing a set of predefined bolts. You can define additional bolts as well.


It is useful to previously select a number of bolts, thus facilitating the design when you want to configure bolts.

1. In the main menu, choose the **Bolts** library under the option **Bibliotheken**. The **Bolt positioning** menu is opened.


---

#### **Note**

*In the SCIA.ESA PT object libraries, you can*



Create new objects using the  button

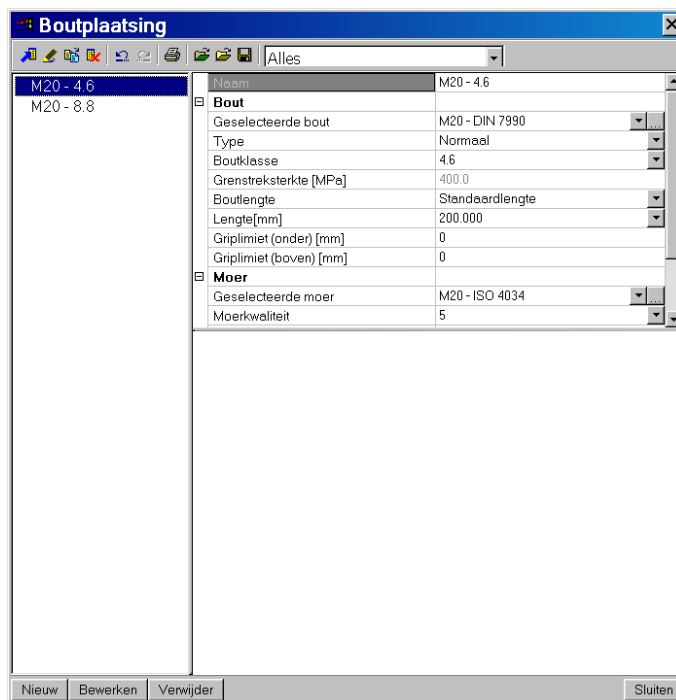
Load objects from the pre-programmed database using the  button.

Open objects from your own database using the  button

---

Save objects in your own database using the  button.

2. Open the system database by clicking the  button.
3. Search the M20-4.6 bolt in the system database and select the line with the mouse cursor.
4. Click the [**<<Copy to project**] button to copy bolt M20-4.6 in the project database.
5. Position bolt M20-8.8 in the project database using the same procedure.
6. Quit this menu with the [**OK**] button.
7. In the **Bolt positioning** menu, remove all other bolts, by selecting them and then clicking on the  button. Below, you will see an image of the user library, which only contains the two desired bolts.



**Figure 35: Menu Bolt positioning**

8. Quit the **Bolt positioning** menu with the [**OK**] button.

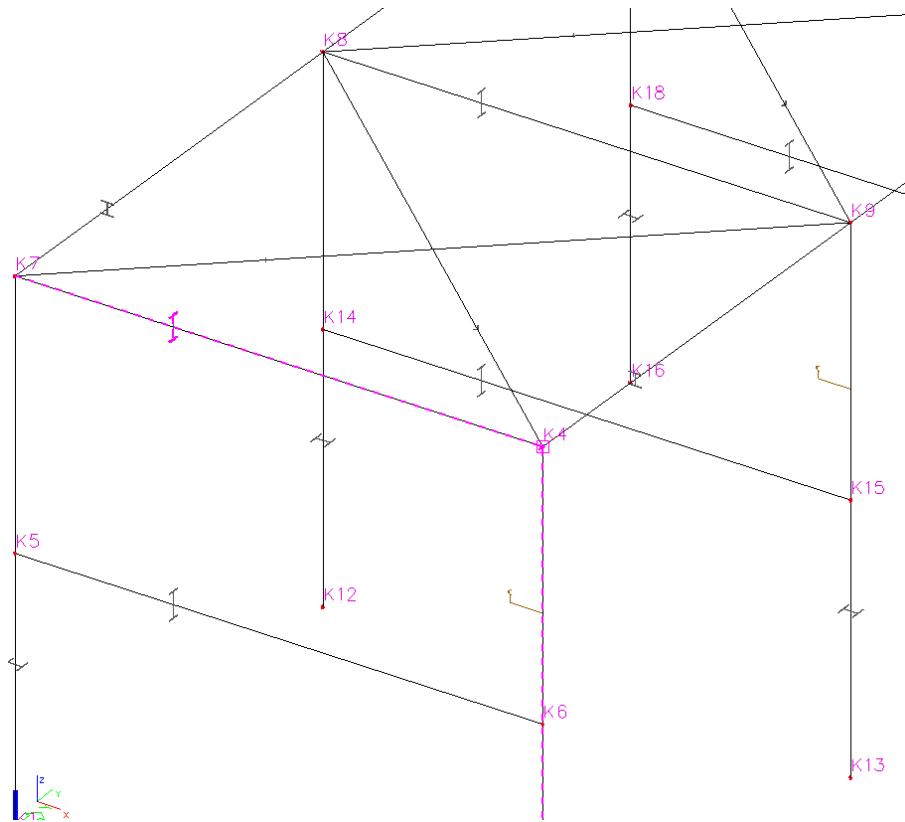
*Note:*

*The last element that is added to the library will be active by default. If an application uses a library object, the active object is taken. You can activate any other object from the library by double-clicking its label.*

## Defining a connection

1. Start the function **Frame bolted/welded** ( Raamwerk gebout/gelast - Sterke as ) using the menu function **Menu tree > Steel > connections > frame bolted / welded**.

2. Select the node where the connection must be designed. In our example, we will take node K4.
3. The program automatically selects all members arriving in the selected node. If necessary, you must deselect the members that are not part of the connection to be designed. In our example, we will deselect the bracing and the buffer beam realising the connection with the next frame. The figure below shows an image of the selection.



4. Use the right mouse button or press **Esc** to confirm the selection. The program automatically creates the connection in the selected node. In the properties table, you will find a new part that is highlighted in green. These are new options to configure the current connection. (The properties screen only shows those parameters, which are relevant to the selected connection type). Hereafter, we will set the parameters to obtain a correct connection.

Below an image of the properties table.

| Naam                            | Waarde                   |
|---------------------------------|--------------------------|
| Naam                            | Conn                     |
| Knoop                           | K4                       |
| Belastingtype                   | Combinaties              |
| Combinaties                     | UGT                      |
| Load case/Combination           | Alle                     |
| Raamwerktype                    | geschoord                |
| Verbindingsgeometrie            | Knie                     |
| Zijde ->[B4]                    |                          |
| Verbindingstype                 | Gebout raamwerk          |
| Kopplaat                        | <input type="checkbox"/> |
| Bovenverstijving                | <input type="checkbox"/> |
| Onderverstijving                | <input type="checkbox"/> |
| Diagonale verstijving           | <input type="checkbox"/> |
| Opdikplaat                      | <input type="checkbox"/> |
| Herlees stijfheid               | <input type="checkbox"/> |
| Lengte voor stijfheidsklassi... | 8.000                    |
| Las                             |                          |

*The type of members to be connected is highly important. The possible connections proposed by the program directly depend on the type configured.*

*In our example, we are designing a beam-to-column connection. The element that must be calculated as column should have a higher priority than the beam. The members of the connection in our example are configured as follows:*


*The column is of the default type **Column(100)**.*

*The beam is of the default type **Beam (80)**.*

*For more information regarding the configuration of priority types, refer to the **Cad form** paragraph.*

5. Activate a cap plate by checking the check box behind **Cap plate**.

*Each time you add a component to the connection, it will be shown in the graphic part. If you zoom the connection, you will see the components in the detail.*

6. Activate bolts by checking the check box behind **Bolts**. The currently active bolt in the bolts library will be used for the connection. If you did not change the active bolt, the current connection will contain M20-8.8 bolts.
7. First, we want to try M20-4.6 bolts. Click the button  behind **Bolts**.


8. In the **Properties** menu, set **Selected bolt** to M20-4.6

| Eigenschappen            |                                     |
|--------------------------|-------------------------------------|
| Naam                     | XXX                                 |
| <b>Bouten-&gt;[B4]</b>   |                                     |
| Geselecteerde boutgting  | M20 - 4.6                           |
| Lengte [mm]              | M20 - 4.6                           |
| Boutpatroon              | M20 - 8.8                           |
| Referentie               | Onderkant van de staaf              |
| Interne boutafstand [mm] | 100                                 |
| 1.Rij                    | <input checked="" type="checkbox"/> |
| 2.Rij                    | <input type="checkbox"/>            |
| 3.Rij                    | <input type="checkbox"/>            |
| 4.Rij                    | <input type="checkbox"/>            |
| 5.Rij                    | <input checked="" type="checkbox"/> |
| 1.Locatie [mm]           | 307                                 |
| 2.Locatie [mm]           | 251                                 |
| 3.Locatie [mm]           | 194                                 |
| 4.Locatie [mm]           | 138                                 |
| 5.Locatie [mm]           | 53                                  |
| <b>Acties</b>            |                                     |
| Herlezen positie         | >>>                                 |
| OK Annuleren             |                                     |

9. Click **[OK]** to leave the **Properties** menu.
10. Click the button **>>>** behind **Reread** to update the data.
11. Click **B4** to open the check menu. Below, you will see the report menu of the check.




| Staalverbinding                           |                         |
|---|-------------------------|
| Naam                                      | Conn                    |
| Grensmoment Mrd [kNm]                     | 42.82                   |
| Grensdwarskracht Vrd [kN]                 | 52.68                   |
| <b>Unity check M/MRd [-]</b>              | <b>1.25</b>             |
| Unity check V/VRd [-]                     | 0.56                    |
| Resultaat van stijfheid: Sj [MNm/rad]     | 3.5585e+000             |
| Resultaat van stijfheid: Sj,ini [MNm/rad] | 1.9413e+001             |
| M [kN]                                    | -53.51                  |
| V [kN]                                    | -29.64                  |
| Controleer M                              | Verbinding voldoet NIET |
| Controleer V                              | Verbinding voldoet      |
| Check of stiffness                        | OK                      |
| Zwakste onderdeel in trekzone             | Bouten in trekspanning  |
| Zwakste onderdeel in drukzone             | Kolomlijf in druk       |
| Load case/Combination                     | UGT/2                   |
| Balk                                      | B4                      |
| OK  |                         |

The connection is not OK. The moments' capacity is too small. The line of the unity check is highlighted in red.

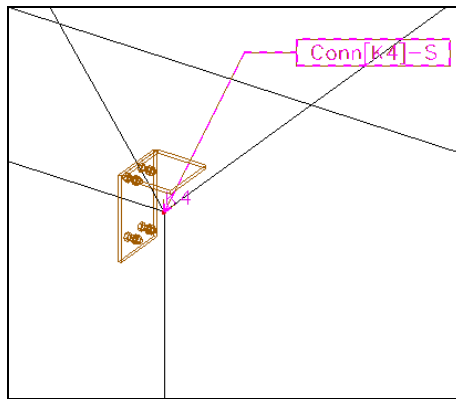
Before starting the check of a connection, first click the  button behind **Reread**. In this way, you are sure that the check is carried out using the current data.

If the connection is not code-compliant, the non-compliant element is highlighted in red in the result screen.

The result menu always indicates the weakest link of the connection.

12. The check menu indicates that the bolts under tensile stress are the weakest part.
13. We will use stronger bolts. Click  behind **Bolts**. Now choose M20-8.8 bolts.
14. Carry out another check. Click  behind **Reread** to update the data. Click  to open the check menu. The connection is OK.
15. We want to increase the stiffness of the designed connection by installing a stiffener at the column head. You can activate this stiffener by checking the check box behind **Cap plate**.


Each time when you activate a new element of the connection, it will be represented in the graphic environment. Below, you will see an image of the current connection.



**Figure 36: The designed connection**

---

Print preview with the report of the check of the connection

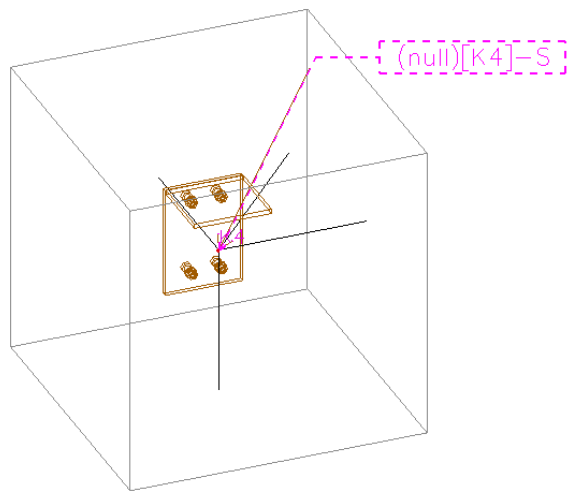
1. Select the connection(s) by clicking on the label(s).
2. You can choose the extensiveness of the report in the properties table. By default, the **Output** property is set to **Normal**. (You can choose between concise, normal and detailed).
3. Click  behind **Open print preview**. You will obtain a print preview of the calculation.

---

Isolating the connection using the cut box




1. Activate the connection by clicking on its label. If the connection is already selected, skip this step.
2. In the main menu, choose the option **Around selected entity in Image > Cut box**.

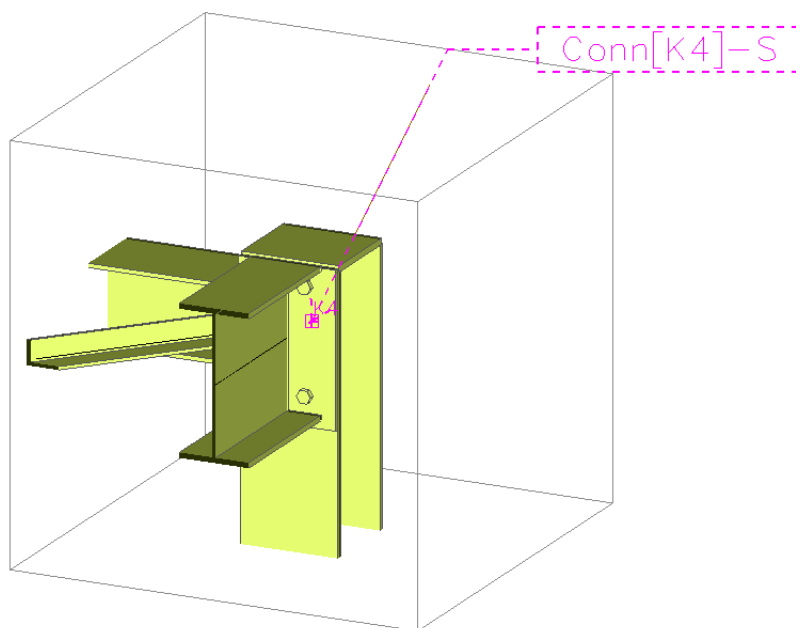
For more options of the cut box, refer to the Cut box paragraph.




---

#### Generating the Cad form from a connection

1. Set up a cut box to isolate the current connection. If you already have set up a cut box, skip this step.
2. Set the image parameters for a rendered structure.  
 With the  button, a volume model will be represented.  
 With the  button, the volume model becomes opaque
3. Use the option “Generate Cad form” (button  in the toolbar) to activate the Cad form.






# Grid lines

The grid lines are useful tools that can be used for the geometry input. They enable you defining a two- or three-dimensional grid, whereunto you can directly draw the structure.

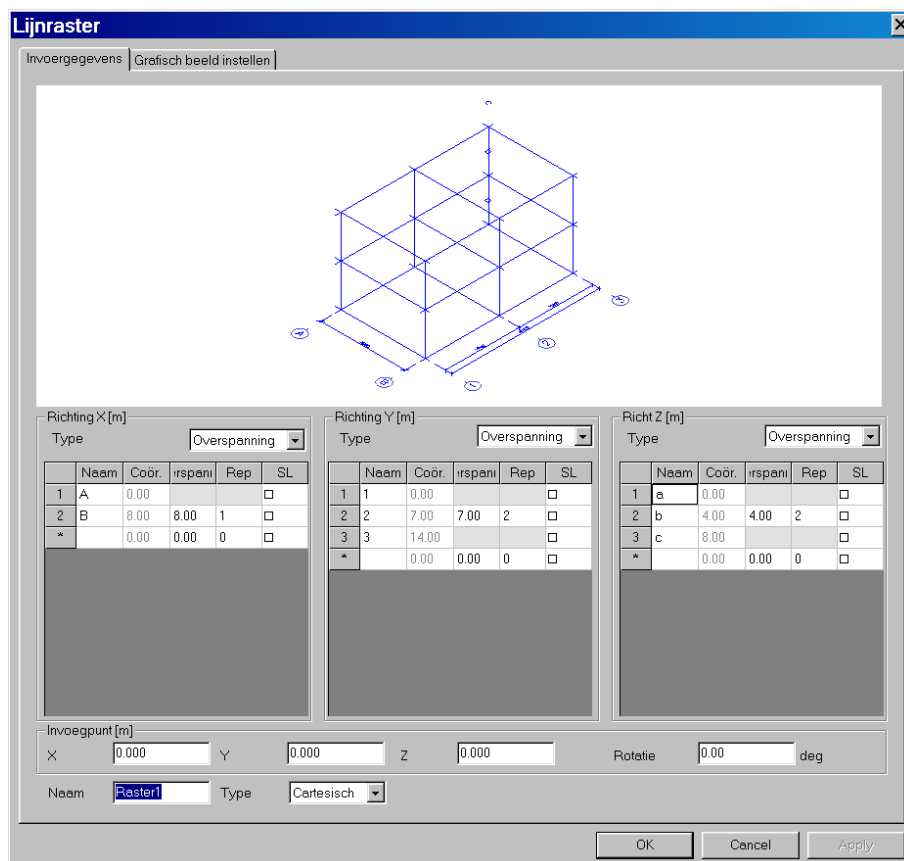
Upon the input of the structure of this example, we did not use grid lines. In this paragraph, we will define the grid line, on which you could have drawn the model of this Tutorial.

## Creating the grid

1. Choose  **Line grids** in the toolbar.

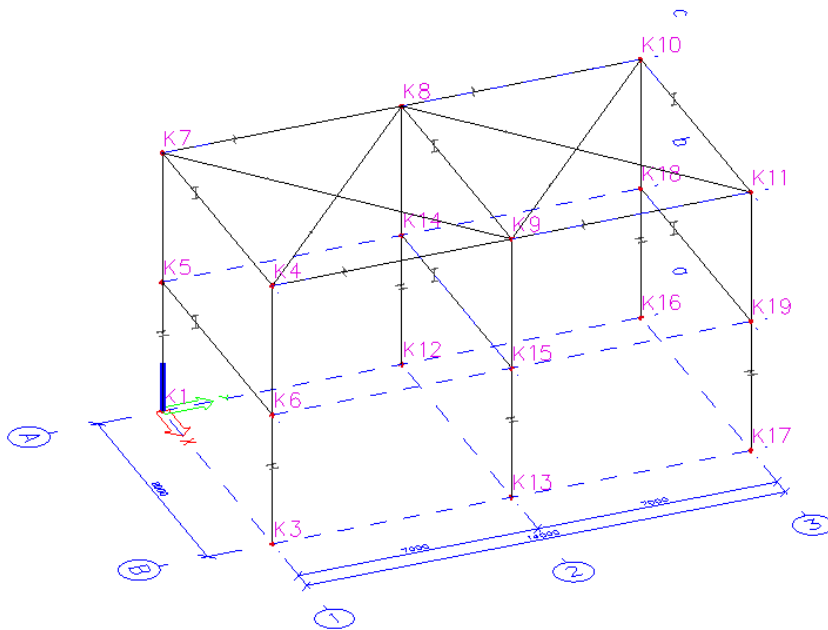
The screen will change and the line grid dialogue will appear.

2. Enter for the X, Y and Z-direction the different distances with their repetition factor. For **Span**, enter the distance, for **Repetition**, the number of times that this distance must be repeated. We want to enter a line grid of (8m x 1; 7m x 2; 4m x 2). To obtain this, enter the parameters as shown in the figure below.




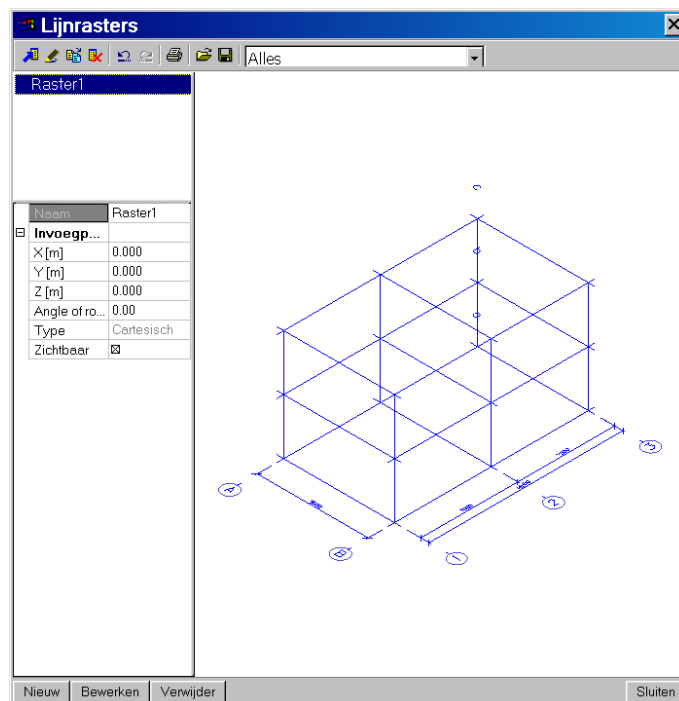
**Figure 37: The line grid**

3. Confirm your input with **[OK]**.
4. The **Line grid manager** is opened. You can modify the input coordinates and the property **Visible**. Take the insertion point at coordinate (0;0;0) and the property **visible** activated.



#### Showing / hiding the line grid

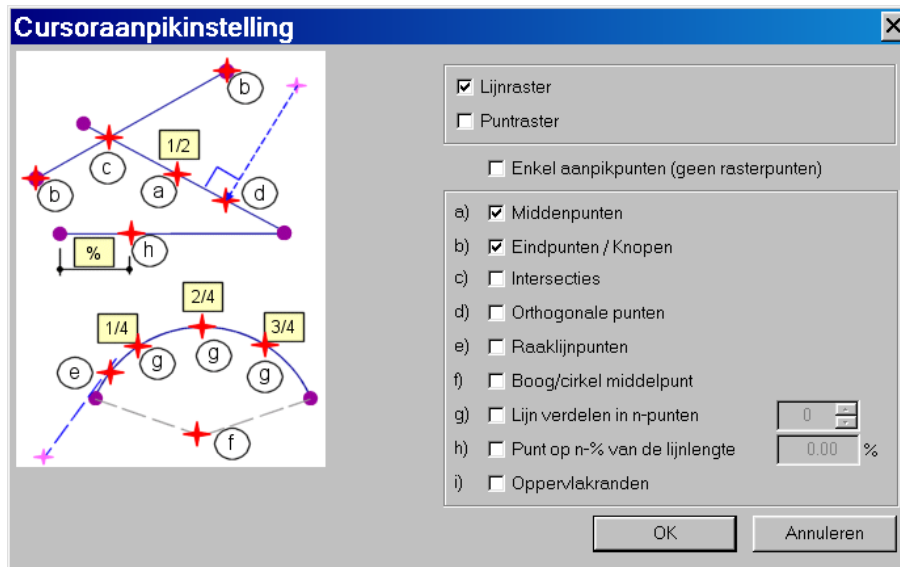
1. Click  to open the line grid library.
2. Check the check box of the property **Visible** to show the grid (make it visible). (Uncheck the check box to hide the line grid, i.e. make it invisible.)



#### Setting up the snap mode to snap on a line grid

In order to enable snapping with the mouse cursor on the line grid, this option must be set.

1. Click the **[Pick mode]** button at the bottom of the screen. The **Cursor pick setting** is opened.
2. Activate the check box of the **Line grid** property.
3. Confirm the settings with **[OK]**.



**Figure 38: The Cursor pick settings menu**

# Image gallery


The image gallery is a tool to create, edit and save drawings.

The image gallery includes a wizard enabling a smooth generation of layouts. To this end, the **advanced** project level must be activated in the project data and the option **Layouts** must be selected in the **Functionality** tab.

The image gallery contains a second wizard, enabling a smooth generation of mono-drawings of connections. To this end, the option **Connection mono-drawings** must be selected in the project data.

---


## Sending a drawing to the gallery

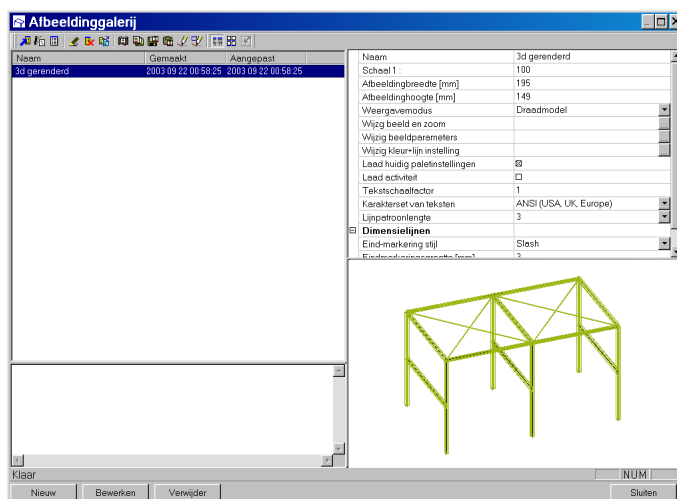
1. Configure the graphic part of the model. We take a rendered image of the structure.
2. Click the  button at the bottom of the screen to send the drawing to the gallery.
3. The **Image name** menu is opened.




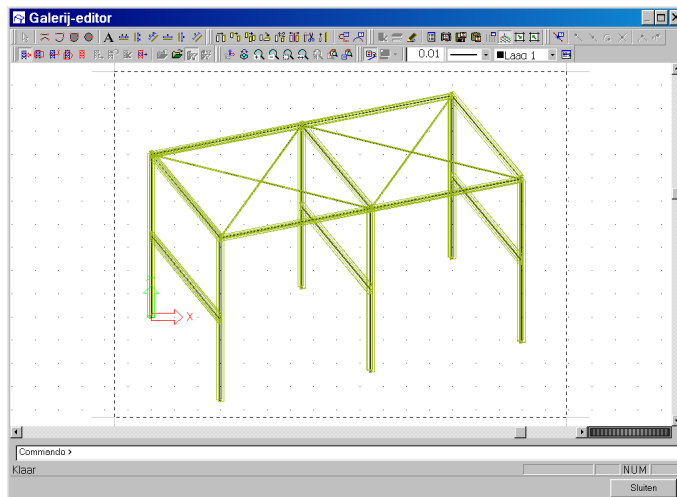

---

## Editing a drawing of the gallery.

1. Click the  button to open the gallery.
2. Select the drawing to be edited by pointing it with the mouse.



3. Activate the drawing in the gallery using the  button





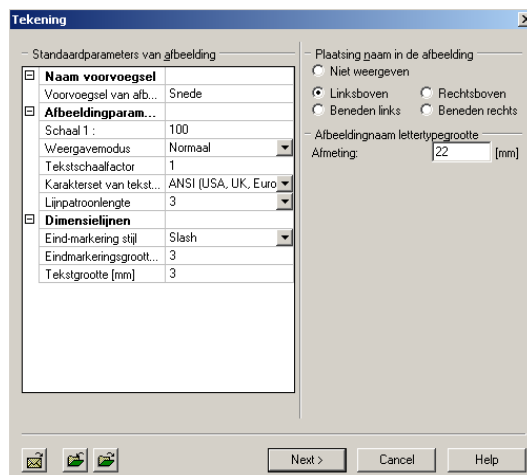
The gallery contains options to

- Draw lines, arcs, ornaments...
- Position texts
- Position dimensioning

### Using the Layout Wizard

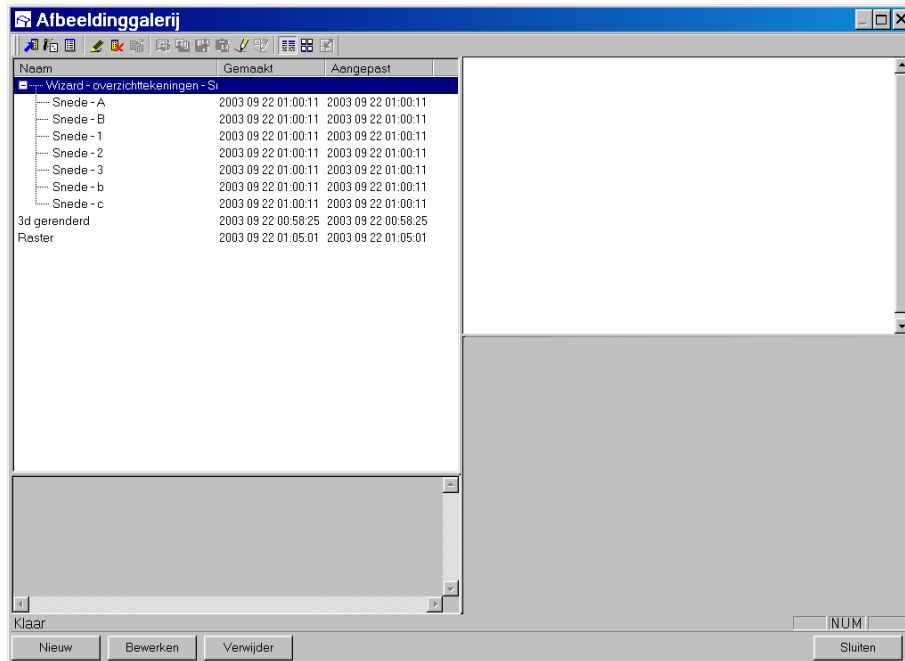
The Layout Wizard is based upon the line grid. For all planes of the line grid, drawings of the structure including dimensioning are created.

1. Double-click  **Afbeeldinggalerij** in the tree menu. The image gallery service is opened.
2. We want to create the layouts using the wizard. Double-click **Cuts per planes according to line grid (layouts)** (button ).
3. The first screen of the wizard appears, in which you can set the drawing options.



4. Click **Next >**. The second screen appears. This screen contains options to change the screen settings.

5. Click **Next >**. The third and final screen is opened. Here, you have to choose the cuts to be created. By default, all cuts are chosen.
6. Click **Finish**. The layouts are created.






---

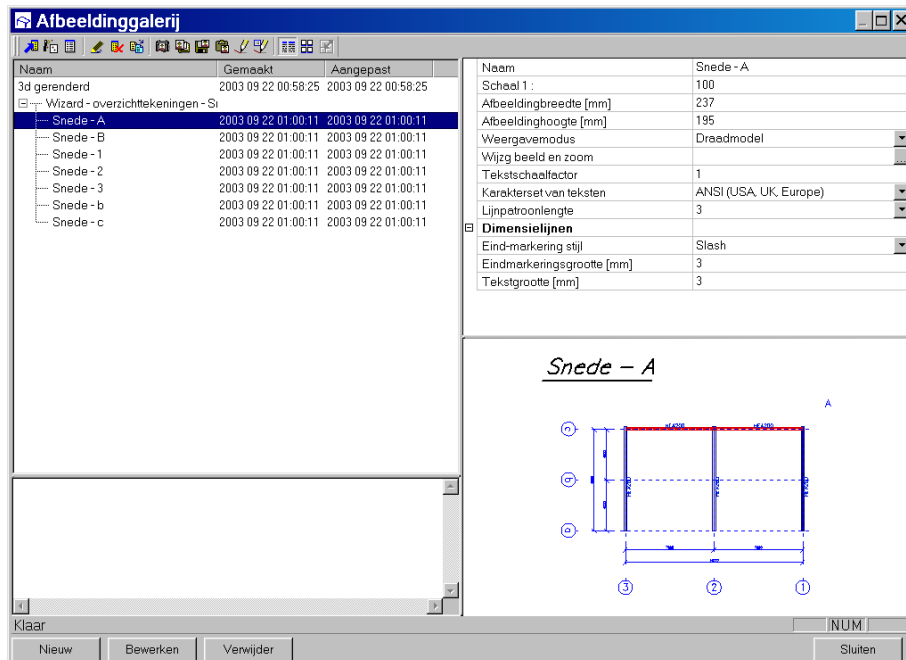
#### The Connection Mono-drawings Wizard

The Wizard for Mono-drawings of connections works in a similar way.

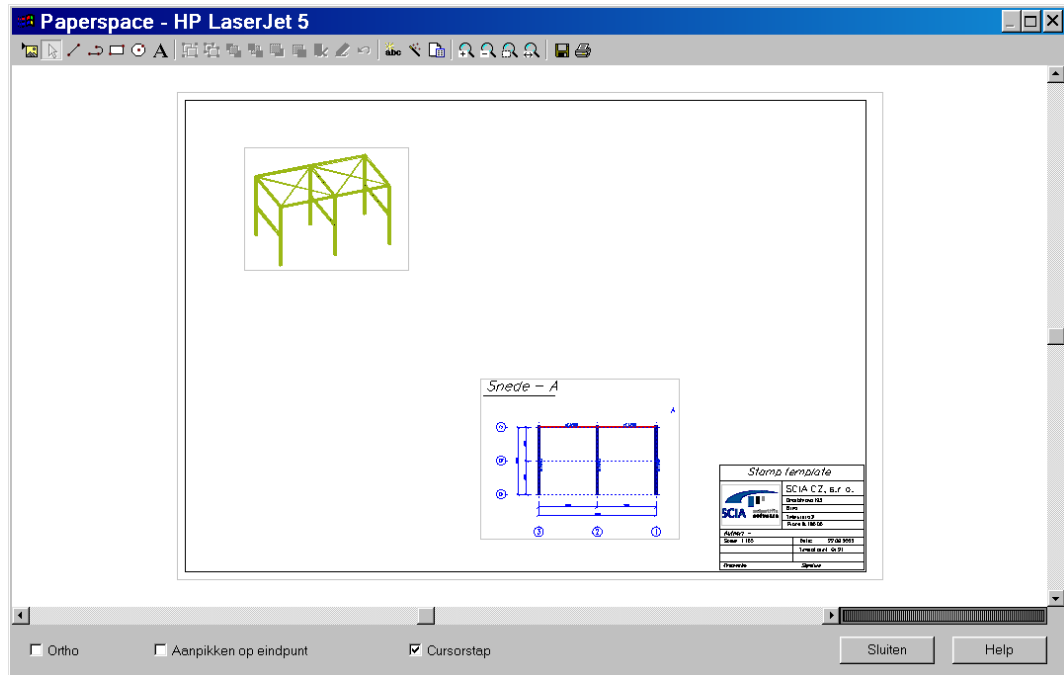
# Paperspace gallery

In the paperspace gallery, you compose the definitive drawings from previously created entireties. The composed drawings are saved here together with the project. You can print a drawing at any given moment.

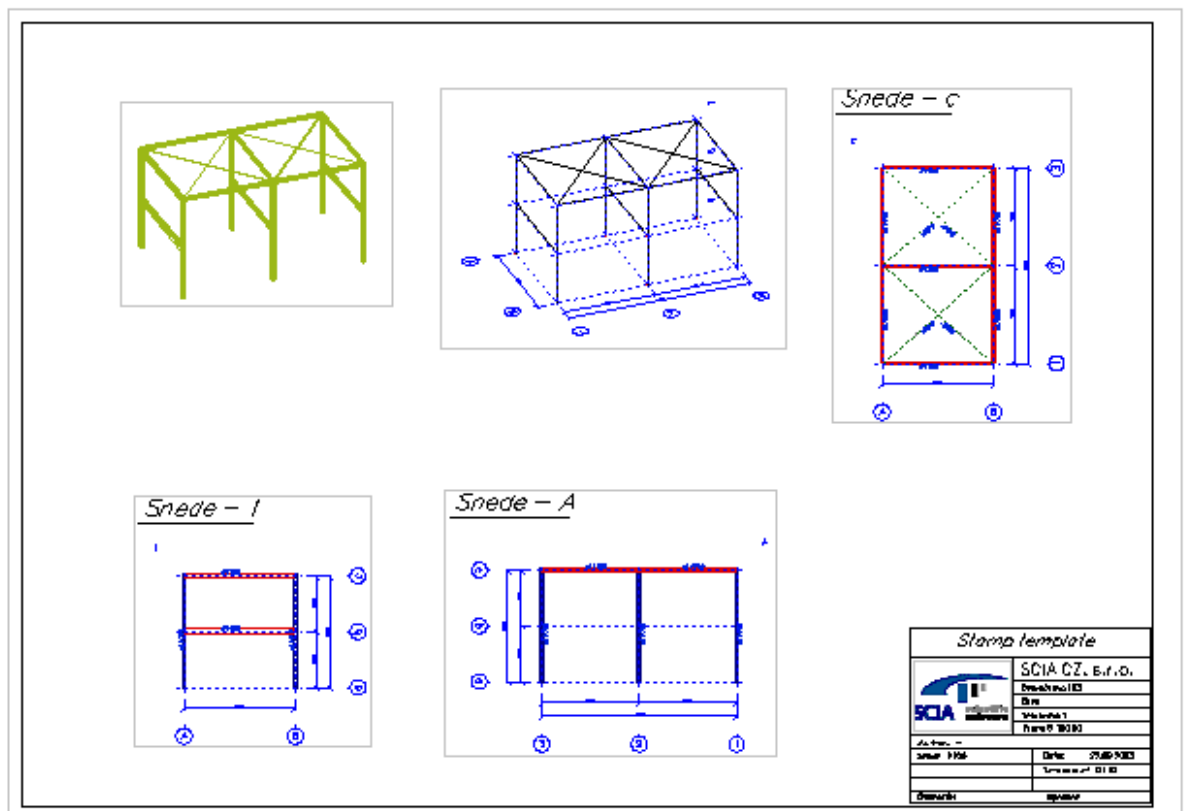
1. Double-click  Paperspace galerij in the tree menu
2. Create an empty drawing note in the gallery using the  button
3. Click  and choose **Import image from gallery**.
4. Select a drawing from the image gallery. Click on the name of the desired drawing: the line is highlighted in blue.



5. Then, click **[Close]**.
6. You will return to the drawing note of the paperspace gallery. Drag the mouse to move the drawing to the desired position. Then click with the left mouse button to fix the drawing.



Finally, a composition can look like the image below.






# Cut box and Activities

## Cut box

Cut box is a powerful tool facilitating adaptations, especially for excessive and complex constructions. The Cut box defines a range or area (3D-area) that is visible on the screen. The remainder of the construction, located behind this area, is temporarily hidden for the user.


---

### Defining the cut box

1. Click the button **Set Cut box for active image**  in the **Image** toolbar and select the **Cut box – new** function.
2. Define the origin (i.e. the centre of the Cut box.)
3. The settings dialogue appears on the screen.
4. Complete the table. You can set the origin, the size and the angle.
5. Confirm with **OK**

---

### Adapting the cut box with the mouse

1. Turn the Cut box ON (click the button **Set Cut box for active image**  in the **Image** toolbar and select the function **Cut box (on/off)**.
2. Position the mouse cursor above one of the borders of the Cut box.
3. Click the left mouse button to select the Cut box
4. In the middle of all Cut box surfaces, special edit symbols are shown for the box. The **ball** symbol provides for the modification of the box size, the **cylinder** symbol enables the user rotating the box.
5. Select the symbol corresponding to the desired adaptation
6. Click the left mouse button and keep it down
7. Drag the mouse to adapt the Cut box according to your wishes
8. Release the mouse button.
9. Press the Esc key to close the adaptation function.

Other possibilities for the cut box set-up can be found in the reference manual.

# Activities

The activity property provides for a selection of only those members, which are indispensable for a certain adaptation or action. The remainder of the construction is temporarily hidden for the user.

---


## Activity types

The user can define the active construction part in different ways.

- (i)      Layers      The activity is entirely controlled by layers
- (ii)     Work plane   Only members located in the current plane become active.
- (iii)    Selection     Only currently selected members become active.
- (iv)    Cut box       Only members located within the current Cut box become active


---

## Activate or deactivate Activity

- (i)      Open the function **Tools > Activity > Activity on**
- (ii)     Or click the button **Activity on** (or Activity off) in the **Activity** toolbar (  )

---


## Activity through the work plane

- (i)      Open the function **Tools > Activity > Activity through work plane**
- (ii)     Or click the button **Activity through work plane** in the **Activity** toolbar (  )

---

## Activity through Cut box

If this type of activity is selected, all members located within the currently adapted Cut box will become active. All other members become inactive

- (i)      Activate the Cut box and adapt it in the desired way
  - (ii)     Or open the function **Tools > Activity > Activity through Cut box** or click the button **Activity through Cut box** in the **Activity** toolbar (  )
-

# Index

## A

|  |    |
|--|----|
| Activate or deactivate Activity .....        | 82 |
| Activating a label for member names .....    | 23 |
| Activating a label for node names .....      | 23 |
| Activity through Cut box .....               | 82 |
| Activity through the work plane .....        | 82 |
| Activity types.....                          | 82 |
| Adapting a series of concentrated loads..... | 37 |
| Adapting the cut box with the mouse .....    | 81 |
| Adapting the loads .....                     | 37 |
| Adding a profile .....                       | 15 |
| Adding an image to the document .....        | 48 |
| Adding buckling supports.....                | 59 |
| Adding hinges.....                           | 25 |
| Adding supports.....                         | 26 |

## C

|  |    |
|--|----|
| Carrying out an optimisation .....           | 60 |
| Carrying out the linear calculation.....     | 43 |
| Closing a project.....                       | 12 |
| Configuration of the columns .....           | 53 |
| Configuring the graphic output.....          | 56 |
| Copying bolts from the system database ..... | 66 |
| Copying loads .....                          | 40 |
| Creating a filter selection.....             | 26 |
| Creating a window selection.....             | 27 |
| Creating documentation.....                  | 47 |
| Creating the grid .....                      | 73 |

## D

|   |    |
|---|----|
| Default configuration.....                                | 53 |
| Defining a connection.....                                | 67 |
| Defining a permanent load case.....                       | 33 |
| Defining a variable exclusive load case.....              | 34 |
| Defining combinations.....                                | 41 |
| Defining the cut box .....                                | 81 |
| Defining the fire resistance.....                         | 63 |
| Defining the self-weight through a distributed load ..... | 35 |
| Drawing a member .....                                    | 17 |
| Drawing bracings.....                                     | 21 |

## E

|  |    |
|--|----|
| Editing a drawing of the gallery.....                    | 76 |
| Entering a column.....                                   | 17 |
| Entering a distributed load .....                        | 39 |
| Entering a series of concentrated loads on members ..... | 36 |

## G

|   |    |
|---|----|
| Generating the Cad form from a connection.....      | 72 |
| Generating the Cad model .....                      | 49 |
| Graphically representing the result of a check..... | 57 |

## Ch

|                                    |    |
|------------------------------------|----|
| Changing the active load case..... | 35 |
| Checking a construction.....       | 22 |

|   |    |
|---|----|
| Checking the fire resistance .....  | 64 |
| <b>I</b>  |    |
| Image parameters – Entities .....   | 30 |
| Image parameters – Labels and Description .....   | 31 |
| Image parameters – Shortcuts .....  | 32 |
| Isolating the connection using the cut box .....  | 71 |
| <b>L</b>  |    |
| Load groups determine the generated combinations .....                                  | 34 |
| Load rules .....  | 42 |
| <b>M</b>  |    |
| Making multiple copies .....  | 19 |
| Modifying the active profile .....  | 19 |
| Modifying the image parameters using the .....  | 30 |
| Modifying the properties of a table .....   | 48 |
| Modifying the table of contents .....   | 48 |
| Modifying the viewpoint onto the construction .....                                     | 29 |
| <b>O</b>  |    |
| Opening a project .....   | 12 |
| <b>P</b>  |    |
| Parameters to set a Cad form .....  | 49 |
| Print preview of a code check .....   | 61 |
| Print preview with the report of the check of the connection .....                      | 71 |
| <b>R</b>  |    |
| Relative and absolute nodes .....   | 23 |
| Representing the system lengths on the structure .....                                  | 52 |
| <b>S</b>  |    |
| Saving a project .....  | 12 |
| Section check .....   | 58 |
| Sending a drawing to the gallery .....  | 76 |
| Setting a default priority type .....   | 49 |
| Setting a priority value .....  | 49 |
| Setting a view direction with regard to the global coordinate system .....              | 29 |
| Setting the buckling parameters for an element .....                                    | 54 |
| Setting the properties of a load group .....  | 34 |
| Setting up the snap mode to snap on a line grid .....                                   | 74 |
| Showing .....   | 74 |
| Starting a new project .....  | 10 |
| Starting the program .....  | 10 |
| <b>T</b>  |    |
| The Connection Mono-drawings Wizard .....   | 78 |
| The Construction service .....  | 16 |
| The magnifying glass .....  | 29 |
| <b>U</b>  |    |
| Using the Layout Wizard .....   | 77 |
| <b>V</b>  |    |
| Viewing the result of forces on members, graphically represented on the structure ..... | 44 |
| Viewing the results of forces on members in tabular form .....                          | 45 |

