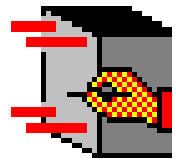


User's manual



ConCreate

© BuildSoft NV

All rights reserved. No part of this document may be reproduced or transmitted in any form or by any means, electronic or manual, for any purpose whatsoever, without the prior written consent from BuildSoft.

The programs described in this manual are subject to copyright by BuildSoft. They may only be used by the licensee and may only be copied for the purpose of creating a security copy. It is prohibited by law to copy them for any other purpose than the licensee's own use.

Although BuildSoft has tested the programs described in this manual and has reviewed this manual, they are delivered 'As Is', without any warranty as to their quality, performance, merchantability or fitness for any particular purpose. The entire risk as to the results and performance of the programs, and as to the information contained in the manual, lies with the end-user.

CONTENT

| | |
|---|-----------|
| CONTENT..... | 3 |
| CHAPTER 1 INTRODUCTION..... | 6 |
| 1.1. Hard- & software requirements | 7 |
| 1.2. Installing ConCrete..... | 7 |
| 1.3. ConCrete, ConCrete Plus and ConCrete List..... | 7 |
| CHAPTER 2 A SIMPLE BEAM..... | 8 |
| 2.1. Introduction..... | 9 |
| 2.2. Creating a new project | 9 |
| 2.3. Entering a new element | 10 |
| 2.4. Interpretation of the calculated results | 12 |
| 2.4.1. The "Structure and loads"-window | 13 |
| 2.4.2. The "Diagrams"-window | 14 |
| 2.4.3. The "Reinforcement sketch"-window..... | 15 |
| 2.5. Modifying entered data | 16 |
| 2.5.1. Direct editing of entered data | 16 |
| 2.5.2. Modifying with the right mouse button..... | 19 |
| CHAPTER 3 A SECOND BEAM..... | 21 |
| 3.1. A beam in the Projectlist-window | 22 |
| 3.2. Concentrated & trapezoidal forces | 22 |
| 3.2.1. Trapezoidal forces..... | 23 |
| 3.2.2. Concentrated forces | 24 |
| 3.2.3. Modifying the amount of zones | 25 |
| 3.2.3.1. Modifying directly | 25 |
| 3.2.3.2. Modification by using the right mouse button | 27 |
| 3.3. Adding concentrated moments..... | 28 |
| 3.4. A beam with a variable section | 29 |
| CHAPTER 4 CALCULATION PARAMETERS | 34 |
| 4.1. Project values | 35 |
| 4.2. Parameters for reinforcement steel..... | 36 |
| 4.3. Parameters for concrete | 38 |
| 4.4. Parameters for the supports | 40 |
| 4.5. Parameters for the redistribution of moments | 41 |
| 4.6. Parameters for the loads | 43 |

| | |
|--|-----------|
| CHAPTER 5 MORE THAN ONE LOAD TYPE | 47 |
| 5.1. Introduction..... | 48 |
| 5.2. A beam with two load types | 48 |
| 5.3. Adding and removing load types..... | 50 |
| 5.3.1. Modify directly | 50 |
| 5.3.2. Modify with right mouse button | 51 |
| CHAPTER 6 AN ENTIRE PROJECT OF BEAMS | 52 |
| 6.1. Introduction..... | 53 |
| 6.2. Defining classes | 54 |
| 6.2.1. Choosing classes | 54 |
| 6.2.2. Input classes | 54 |
| 6.2.3. Project list..... | 56 |
| 6.3. Allocating an element to a certain class | 57 |
| 6.4. Getting an element out of a certain class | 57 |
| 6.5. Moving an element to another class..... | 57 |
| 6.6. Duplicating an element | 58 |
| 6.7. Deleting an element..... | 58 |
| 6.8. Copying elements between projects | 58 |
| CHAPTER 7 SPECIAL ELEMENTS | 59 |
| 7.1. T-, I-, L- en Z-beams..... | 60 |
| 7.2. Beams on an elastic foundation | 63 |
| 7.3. Beams in 2 phases | 65 |
| 7.4 Slabs in 2 phases | 66 |
| 7.5. Hollow core slab library | 66 |
| CHAPTER 8 SAVING, OPENING AND QUITTING | 68 |
| 8.1. Saving a new project..... | 69 |
| 8.2. Opening an existing project | 69 |
| 8.3. Saving an existing project | 69 |
| 8.4. Automatical saving..... | 69 |
| 8.5. Closing a project..... | 71 |
| 8.6. Modifying the project data | 71 |
| 8.7. Quitting the program | 71 |
| CHAPTER 9 PRINTING..... | 72 |
| 9.1. Page setup..... | 73 |
| 9.2. Printing an entire project | 73 |
| 9.3. Printing an entire class | 74 |
| 9.4. Printing of only one element | 74 |

| | |
|---|-----------|
| CHAPTER 10 USEFUL TIPS..... | 75 |
| 10.1. Automatical calculation after input..... | 76 |
| 10.2. Sounds..... | 76 |
| 10.3. Connections with ConCrete Plus..... | 76 |
| 10.4. Showing more diagrams or limit states | 78 |
| 10.5. Sign conventions for moment and shear force | 80 |
| 10.6. Close-up of a diagram in a zone | 81 |
| 10.6.1. Making a close-up | 81 |
| 10.6.2. Changing to another diagram..... | 82 |
| 10.6.3. Withdrawing values in isolated points | 82 |
| 10.7. Use of the "Edit"-menu | 82 |
| 10.8. Calculation method | 83 |
| 10.9. Verification calculation | 83 |
| 10.10. Saving and retrieving preferences in files | 84 |
| 10.11. Adapting ConCrete's environment | 84 |
| | |
| CHAPTER 11 OVERVIEW OF THE MENU'S | 87 |
| 11.1. "File"-menu | 88 |
| 11.2. "Edit"-menu..... | 88 |
| 11.3. "Element"-menu | 88 |
| 11.4. "Window"-menu..... | 89 |
| 11.5. "Preferences"-menu | 89 |
| 11.6. "Help"-menu..... | 90 |

Chapter 1 Introduction

1.1. Hard- & software requirements

ConCrete is a 32-bit software program for the MS Windows operating system. Although it may be possible to run ConCrete on previous versions of MS Windows (without any warranty from BuildSoft, however), it is highly recommended to use it on **MS Windows 98, Me, XP, Vista or 7**.

1.2. Installing ConCrete

The ConCrete software can be installed from CD-ROM or over the internet. In this case, you should have a valid LOGIN and PASSWORD to access the "Customer Care" section of the BuildSoft web site <http://www.buildsoft.eu>. BuildSoft customers receive access to this protected section as part of their maintenance or lease service contract. Other people interested in evaluating ConCrete receive valid access data for a 30- days period provided they register at the BuildSoft web site. In case all defaults are accepted as proposed by the ConCrete installation procedure, the software will be installed on the directory "C:\Program Files\Buildsoft\ConCrete".

1.3. ConCrete, ConCrete Plus and ConCrete List

ConCrete belongs to a package of 3 programs which together manage the design process of reinforced concrete beams.

The ConCrete module automises the static and organic calculation of continuous beams and slabs (in one direction) of reinforced concrete.

The ConCrete Plus module uses these results for generating the reinforcement plans.

The ConCrete List module creates the bar schedules of the used reinforcement steel.

Apart from ConCrete, ConCrete Plus and ConCrete List we can use a CAD program for completing the reinforcement plans. In ConCrete Plus the data can be transferred to AutoCad, VersaCad, Microstation, Speedikon, Architron, Tech2D or any other program that reads DXF format.

Chapter 2 A simple beam

2.1. Introduction

We will now get used to the working method of ConCrete by means of a simple example. Let us consider a beam of one span.



Figure 1 : Sketch of the first example

The length is 5 m and the beam is loaded over its full length with a uniform distributed load $q = 20 \text{ kN/m}$. The width of the beam is set to 190 mm, the height to 450 mm.

Remark: all examples in this manual are calculated with EN 1992 1-1 (B).

2.2. Creating a new project

After having started ConCrete we select in the **File** menu the first command "**New...**".

On the screen appears a dialog box (Figure 2) in which we can enter the main data of our project: a reference code for the project, the coordinates (name, address,...) of the project, of the builder, the architect, the contractor and the engineer.

Figure 2 : Data of a project

The meaning of the right buttons is explained in 4.1.

We leave the dialog box by clicking the "OK"-button (with the mouse or by pressing the ENTER- or RETURN-key).

On the left side of the screen appear two palettes (Figure 3 & Figure 4) and in the bottom left corner a window (Figure 5), with two empty lists in it and with the reference code "CONCRETE" as header in the title bar. From now on we will call this window the **Project list**-window.



Figure 3 :
Palet 1

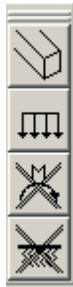


Figure 4 :
Palet 2

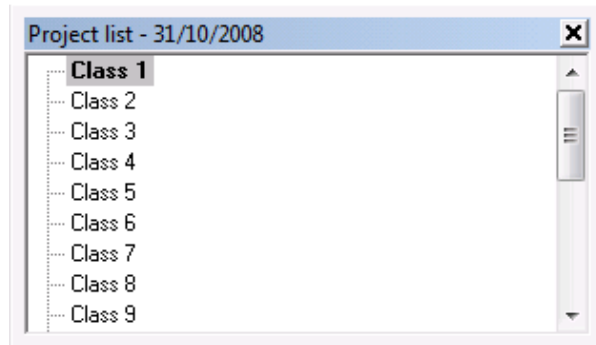


Figure 5 : Project list

2.3. Entering a new element

Now it is possible to select the **Element** menu. We select the first command "New beam" in the **Element** menu or click the icon  .

At the top of the screen an **input panel** appears (with the indication "code beam?") (Figure 6).



Figure 6 : Code beam

Three extra windows are added with resp. heading "**Structure and loads**", "**Diagrams**" and "**Reinforcement sketch**". The two last mentioned windows are empty, the **Structure and loads** window already contains some information.

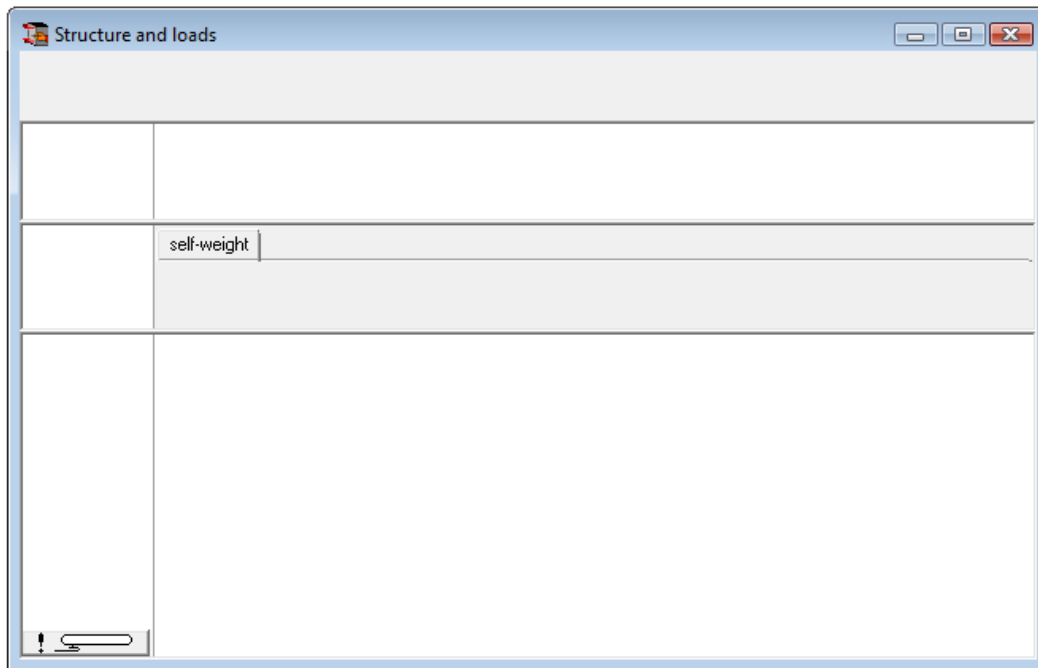


Figure 7 : Structure and loads window

It is possible that the “**Reinforcement sketch**” window is hidden behind the **Diagrams** window. If you wish to bring the first window to the top, you select in the “**Window**” menu the second command “**Reinforcement sketch**”.

We are going to enter all the required data step by step, by answering to the questions asked in the input bar.

The word "code" in the first question "code beam" should not be interpreted too literally; the reference code can contain other characters than figures; the total number of characters is however restricted to 256. Let us enter for the code "1": type these characters by means of the keyboard and press the ENTER- or RETURN-key. The text "1" is now filled out in its destined place in the “**Structure and loads**” window.

ConCrete now asks us to enter the number of spans. We type "1" (don't forget the ENTER- or RETURN-key!). In the same way we enter the width and height the beam (pay attention to the units!).

To the next question "number of load types?" we type "1" again. After entering the length of the first and only span ("5" m) we notice that the left support is yet drawn in the **Structure and loads** window.

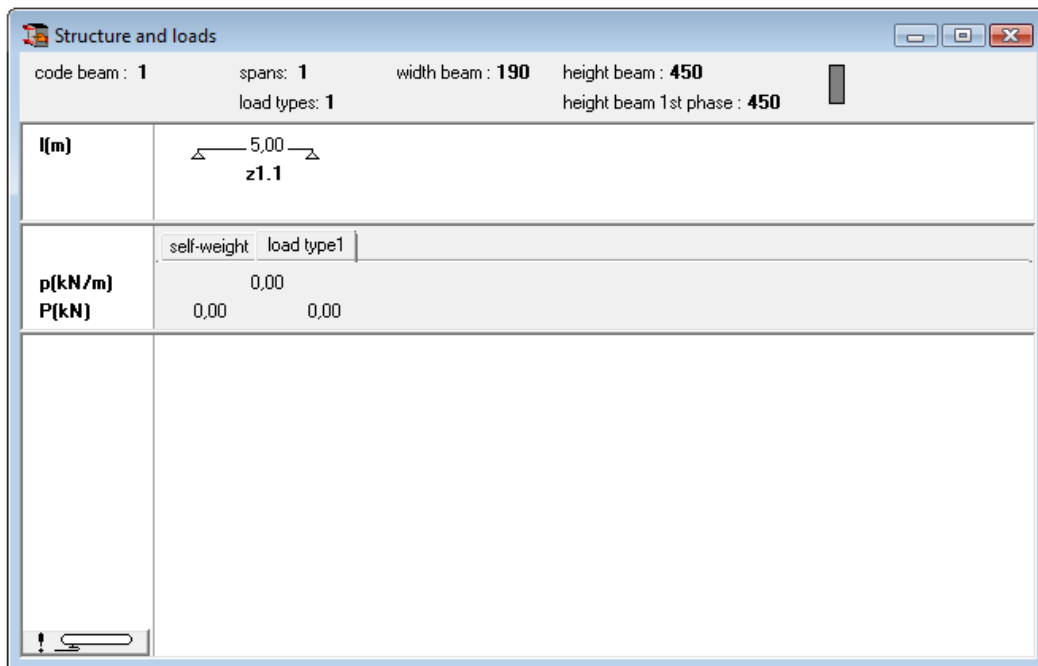


Figure 8 : Structure and loads window - input

To the question "number of zones in 1° span?" the program suggests the answer "1" ; we confirm this by pressing the ENTER- or RETURN-key.

Finally we type the value "20" to the question "value distributed load nr. 1 zone z1.1? (kN/m)". After pressing the ENTER- or RETURN-key, the program automatically calculates the entered beam. The windows "**Structure and loads**", "**Diagrams**" and "**Reinforcement sketch**" are completed with the results of the calculations.

2.4. Interpretation of the calculated results

The results of the calculations are spread over the three windows "**Structure and loads**", "**Diagrams**" and "**Reinforcement sketch**".

2.4.1. The "Structure and loads"-window

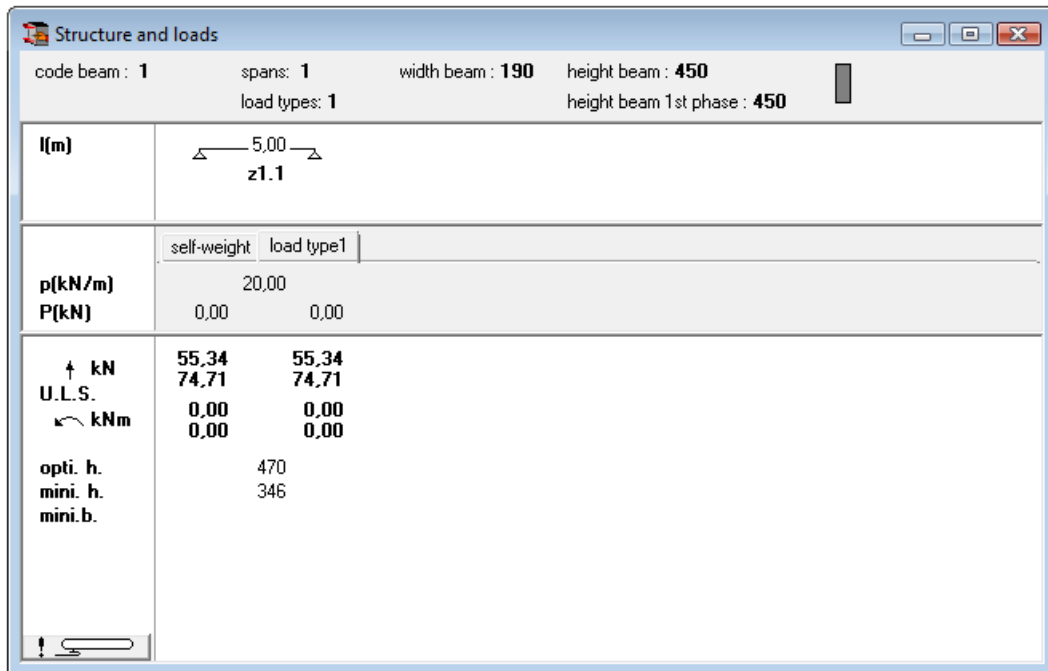


Figure 9 : Structure and loads window - results

Clicking the icon makes the "Structure and loads" window visible. In this window (Figure 9) the minimum and maximum **vertical reactions** in **Ultimate Limit State (U.L.S.)** of each support are indicated. Upward reactions are considered positive. The minimum and maximum value are here 55.34 and 74.71kN. These values are the result of the reaction due to the introduced load and the self weight (25 kN/m³). The minimum value is obtained by considering both loads with load coefficient = 1.0 (load factor for dead loads with favourable effect). The value 55.34 can be explained as follows :

$$(20 + 25 \cdot 0.19 \cdot 0.45) \text{ kN/m} \cdot 5\text{m} / 2 = 55.34 \text{ kN}$$

The maximum value is obtained by considering both loads with load coefficient = 1.35 (load factor for dead loads with unfavourable effect). The value 74.71 can be explained as follows :

$$1.35 \cdot (20 + 25 \cdot 0.19 \cdot 0.45) \text{ kN/m} \cdot 5\text{m} / 2 = 74.71 \text{ kN}$$


The maximum and minimum **moment reactions** in **Ultimate Limit State (U.L.S.)** are indicated under the vertical actions: since we deal with simple supports (without any restraint to rotation) the minimum and maximum value are both equal to 0.00kNm. Moment reactions which work counter clock-wise are considered as positive.

Information on the **dimensioning** of the beam is indicated in the three remaining lines under the moment reactions. On the first line we notice as optimum height the value 470 ; this is the optimum height of the beam in mm. The **optimum height** is the height at which the concrete is subjected to a compressive strain of 3.5‰ and the steel to a tensile strain of 10‰, in other words the height at which the components of the reinforced concrete are used in an optimum way. On the next line the **minimum height** is indicated, here 346 mm. The minimum height is the height which just meets the requirements to resist to the solicitations without needing any compression reinforcement ; in other words, if we should choose a smaller height the beam would collapse by smashing the concrete if we did not foresee compression reinforcement. Only when the entered height is smaller than the optimum height the two higher mentioned values are indicated in the **Structure and loads** window. For beams, these values are only determined by bending.

When the cross section does not meet the requirements to resist the shear force, the **minimum width** at which the concrete compression struts in the classical truss girder analogy of Morsch just fail to get smashed. For slabs, the minimum section of the compression struts is inclined in the calculation of the minimum height.

*Important: the values for optimum height, minimum height and width are related to the **Ultimate Limit State**. How ConCrete handles the **Serviceability Limit State** will be explained further on.*

2.4.2. The "Diagrams"-window

In the menu "**Window**" select the second item "**Diagrams**" or click the icon . In this window (Figure 10) we notice a schematic representation of the entered beam, diagrams for shear forces and for bending moments in ultimate limit state and the required reinforcement quantities. Right of the diagram, the extreme values in negative and positive sense and the units are indicated. Further on in this manual we will learn how we can show the diagram of rotation angles and deflections in this window (10.4) and how we can change the sign convention of the shear force V and the bending moment M (10.5).

The reinforcement diagram takes a shift of the moment-line into account over a distance $a = z \cdot (\cot \theta - \cot \alpha) / 2$. Compression reinforcement is always red and indicated with a negative value, the black dashed line represents the amount of tensile reinforcement for which only the ultimate limit state requirements; the black solid line is the amount of tensile reinforcement required for the serviceability limit state. The

horizontal dotted lines (indicating 117) set the minimum amount of longitudinal reinforcement (here corresponding to a reinforcement percentage $\rho = 0.0015$).

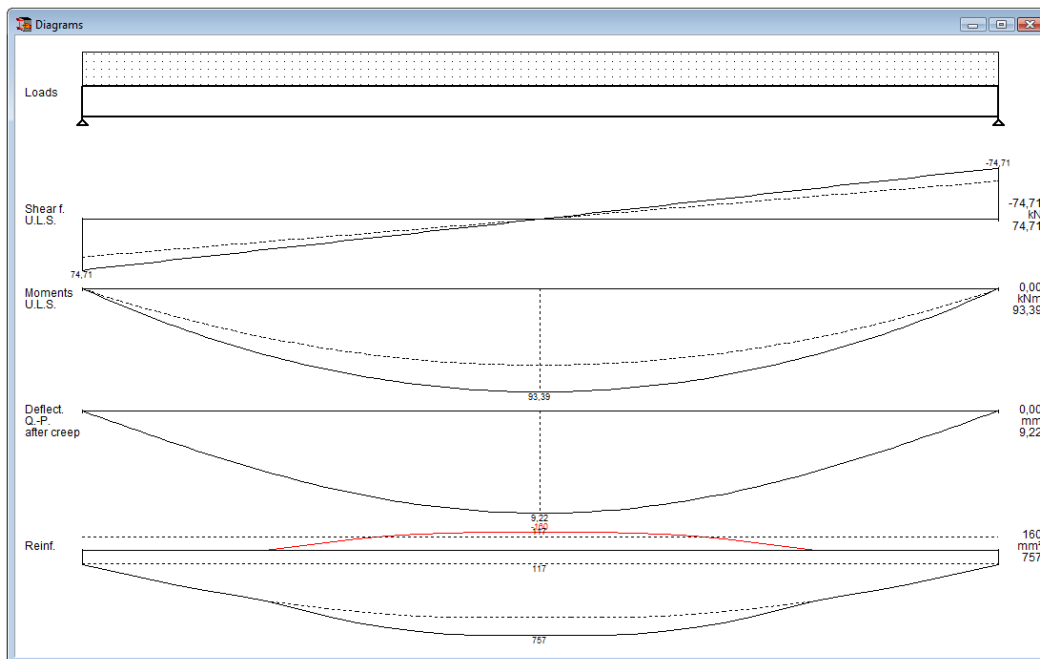



Figure 10 : Diagrams

2.4.3. The "Reinforcement sketch"-window

When this window is still hidden by the "Diagrams"-window, we bring it to top by selecting the second command "Reinforcement sketch" in the **Window** menu or click . In this window (Figure 11) we can see a schematic representation of the front view of the beam with indication of the reinforcement.

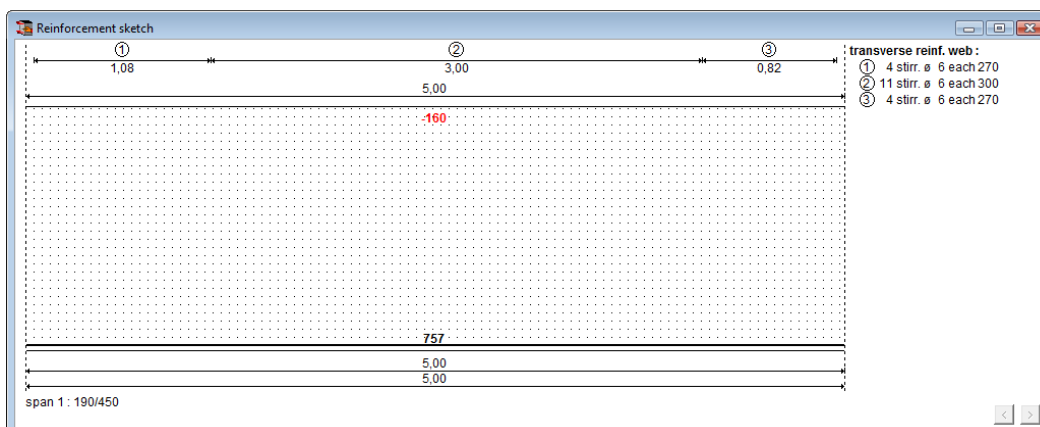


Figure 11 : Reinforcement sketch

Above the beam sketch we see a measuring line with above it the encircled figures "1", "2" and "3" and under it the figures "1.08", "3.00" and "0.82". We should interpret this as follows. The encircled figures are

related to the stirrups and are a reference to the information which is at the right of the beam sketch. The figures under the measuring line indicate the distance in m along which the mentioned stirrups have to be placed. In the information on the right "4 stirrups \varnothing 6 all 270" means that we should provide 4 stirrups with a diameter of 6 millimeter with a theoretical distance of 270 mm over the indicated length.

We always assume in ConCrete that the first, respectively the last stirrup in a span is placed at 5 cm of the left-, respectively the right support of the span. If there are more **stirrup zones** (in our example there are 3) the left arrow of the stirrup zone indicates the place of the first stirrup of that zone when the previous zone is provided with a heavier transverse reinforcement: the right arrow indicates the place of the last stirrup when the next stirrup zone needs a heavier transverse reinforcement. In our example, the arrows between zone 1 and 2 and zone 2 and 3 indicate the place of the first-, respectively the last stirrup of the second zone.

The longitudinal reinforcement is indicated in mm^2 . At places where a bottom reinforcement is required, a bold line is drawn just above the bottom side of the beam: in case of top reinforcement, the line is drawn just below the upper side of the beam. The beginning and ending point of this bold line, is the place where the shifted moment-line doesn't require reinforcement anymore (see 2.4.2). The determination of the diameters of the longitudinal bars and the calculation of the anchorage length is done automatically in Concrete Plus. We can now easily read the required reinforcement from the **Reinforcement sketch** window with this information.

In our example we only need bottom reinforcement over the entire length of the beam. The maximum required section is 757mm^2 . We could place here e.g. two bars of 25 mm diameter. The red-marked negative quantity -160 means that there is 160mm^2 compression reinforcement required on top in order to comply with the limitations for concrete stresses in **Serviceability Limit State**.

2.5. Modifying entered data

2.5.1. Direct editing of entered data

One of the numerous advantages of ConCrete is the simplicity with which already entered data can be modified.

All the data entered, can be modified at any time by clicking the value with the mouse in the **Structure and loads** window, except for 3 data types:

- number of spans, this number can never be less than the number of already present spans. (see 2.5.1.3)
- number of zones, which can only be modified after completely entering the beam. (see 3.2.3)
- number of load types, which can only be modified after completing the current beam or before entering the second load type. (see 5.3)

Let us, for example, change the span length of 5 m into 4.95 m. Select therefore the figure "5.00" in the **Structure and loads** window. When moving with the mouse over the value, the value is highlighted. After selecting this value, the value is framed and appears in the input bar with the following question: "length 1° span? (m)" (Figure 12).

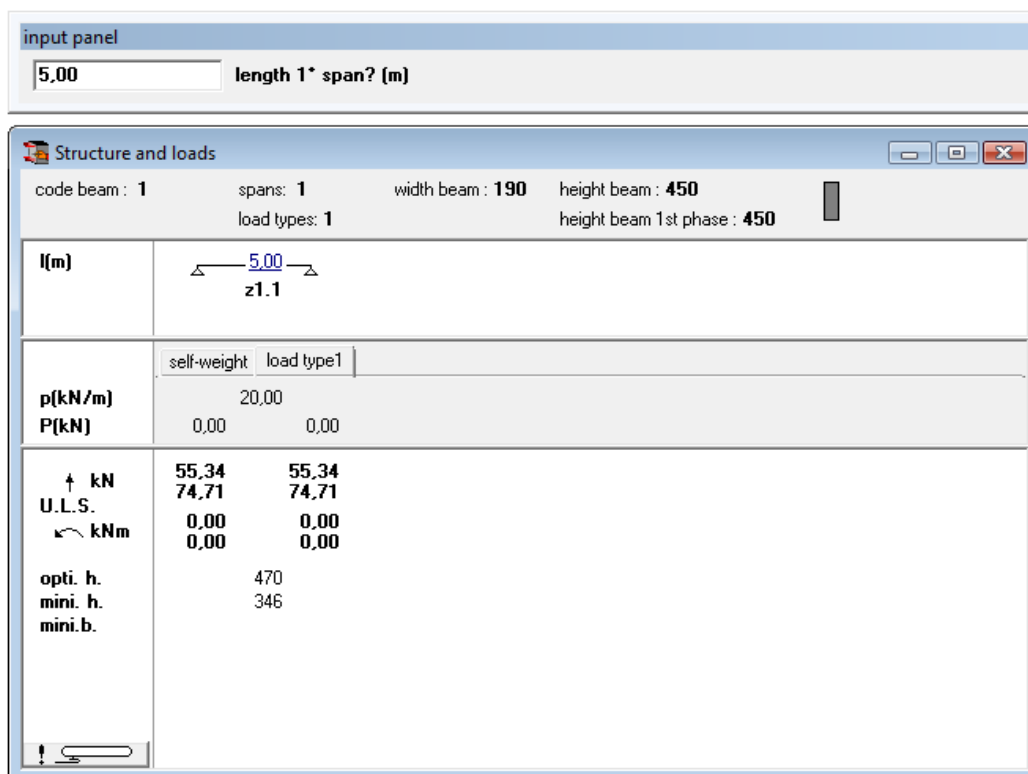


Figure 12 : Adjusting the length of a span

We now type the figure "4.95" and confirm it by pressing the ENTER- or RETURN-key. As soon as we have pressed this last key, the beam is automatically recalculated and all results are adapted in their window. We can notice e.g. that the required section for bottom reinforcement has dropped from 757 mm² to 727 mm² and from 160 mm² to 144 mm² for

the top compression reinforcement. If you change the height of the beam from 450mm to 500mm, the bottom reinforcement will be 516mm² and the compression reinforcement 6mm².

Thanks to the flexible method of modifying the data ConCrete enables you to really **dimension**. When we initially enter an arbitrary low value for the width and height of the beam, the program tells us after calculation how much we need minimally and optimally. In order to obtain a section which is neither too big nor too small, you only have to change the initial low values into higher ones, taking the minimum and optimum values into account.

When the number of spans is altered, the program will ask, depending whether the number of spans is enlarged or reduced, where the spans should be added (Figure 13) or removed (Figure 14).

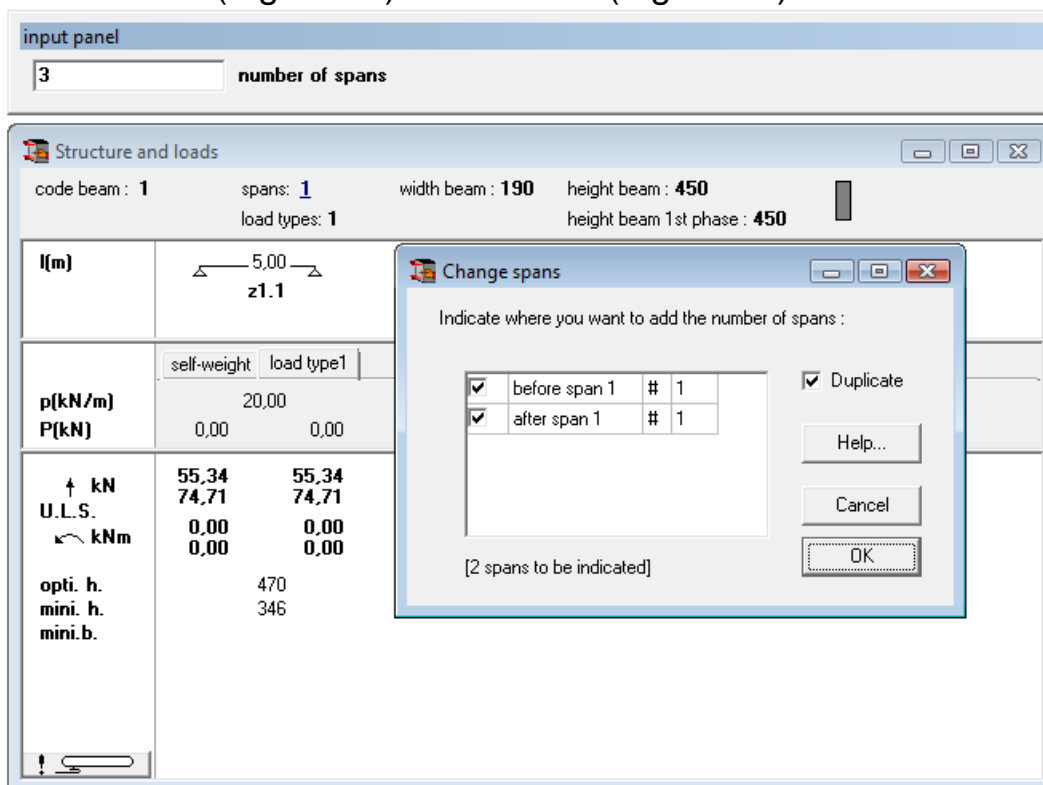


Figure 13 : Enlarging number of spans

Note the choice to duplicate spans. Indicate this checkbox if the added spans must be a copy must be of the previous (when adding a span before the first span, the span added will be a copy of the first span).

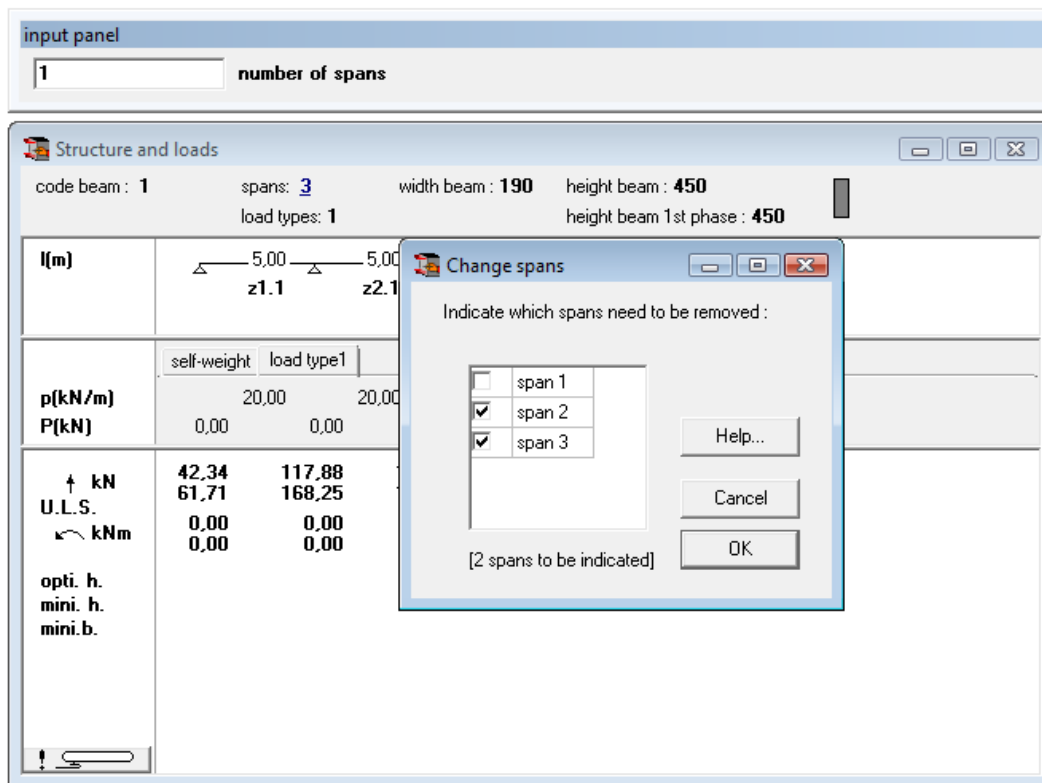


Figure 14 : Reducing number of spans

2.5.2. Modifying with the right mouse button

A second way to modify the entered values is to use the mouse. For example, right-click the name of the length of the span, there appears a so-called popup menu with some items:

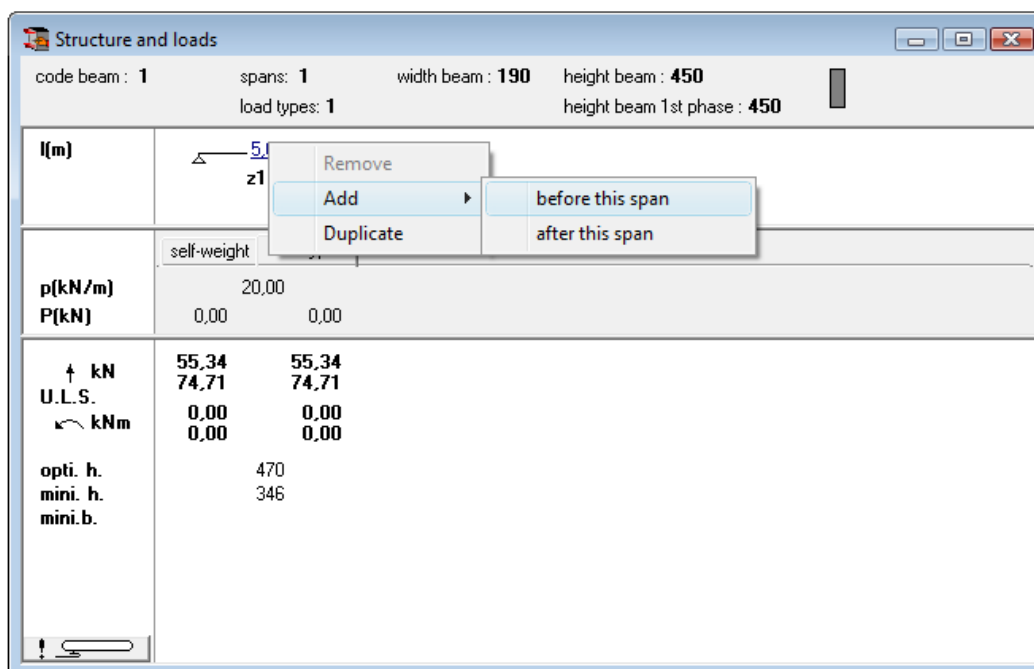


Figure 15 : Pop-up menu – span

The first item "**Remove**" is only available when there is more than one beam span.

The item "**Add**" allows you to add a standard span before or after the appointed span, while "**Duplicate**" copies the current span.

Such a function also exists for editing the loads. Either you select the values of the loads one by one, or you can use the right button of the mouse. With the popup menu that appears, you can quickly copy the value of a loads to other fields.

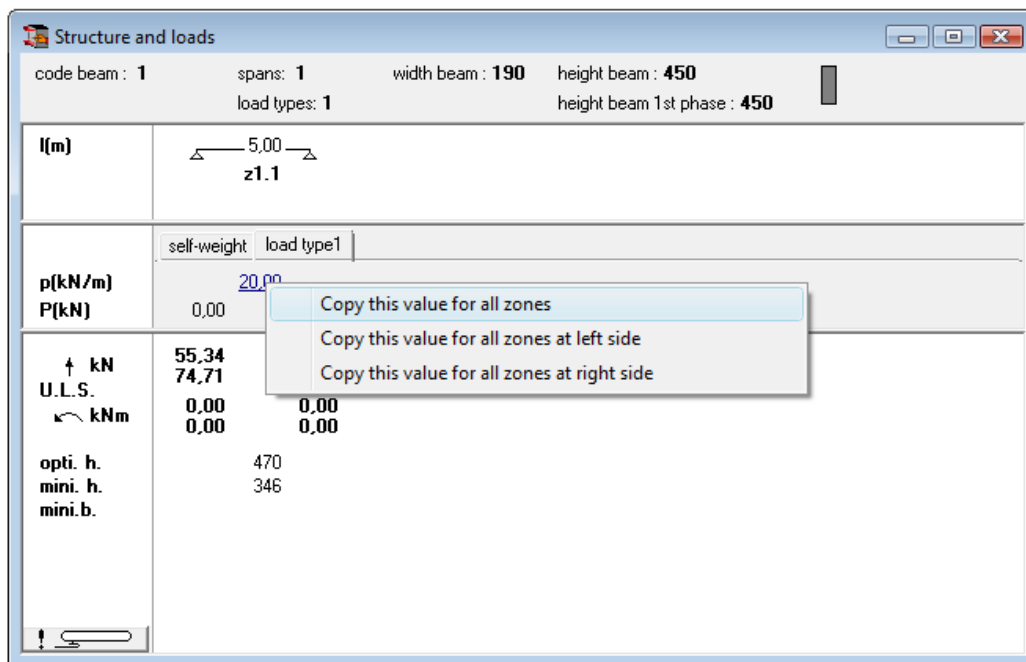


Figure 16 : Pop-up menu – Copying loads

What happens when you changes the amount of load types, will be discussed in 5.3.

The editing of the zones will be handled in 3.2.3.

Chapter 3 A second beam

3.1. A beam in the Projectlist-window


To enter a new beam, we select the first command "New beam" in the **Element** menu or click the icon . The previous beam, named "beam", is now saved in the project.



Figure 17 : Project list

The list is constructed as a tree. The main items are the classes, the subitems are the elements. Currently, only one element is calculated. The second beam has yet to be defined. Both elements are in the first class. The date that the element was last opened is mentioned behind its name. Moreover, the most recent date is listed next to the header of the window itself.

3.2. Concentrated & trapezoidal forces

In chapter 2 we saw how to enter and how to calculate a beam of one span with a uniform distributed load. We now enter a beam of two spans and with a less simple load (Figure 18). The width of the beam is set to 190 mm, the height to 450 mm.

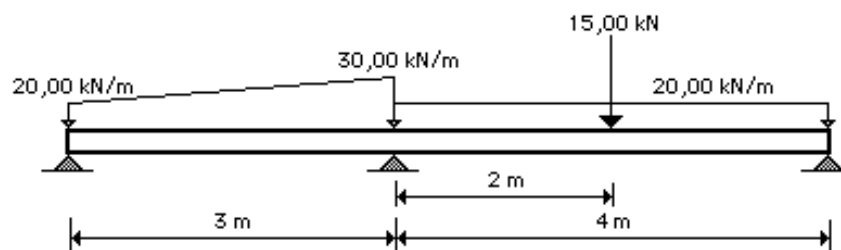


Figure 18 : Sketch of the second example

The input will be completely similar to that described in chapter 2, but take care of the number of spans and consider the remarks 3.2.1 and 3.2.2 before entering the element.

3.2.1. Trapezoidal forces

The **trapezoidal force** is entered as follows. Select with the Mouse in the second pallet (Figure 19, Figure 4) the second icon. When the button is lowered, trapezoidal loads can be entered.

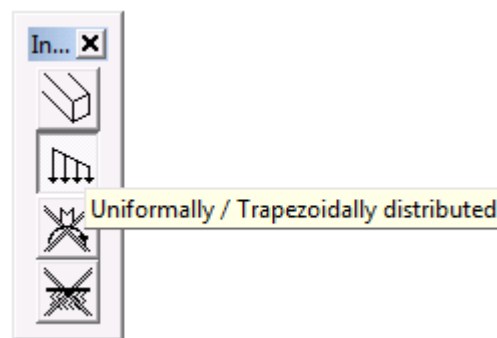


Figure 19 : Palet 2 – Uniformly / Trapezoidally distributed loads

| Structure and loads | | | | |
|---------------------|-------------|---------------|-------|-----------------------------|
| code beam : 2 | | spans: 2 | | width beam : 190 |
| | | load types: 1 | | height beam : 450 |
| | | | | height beam 1st phase : 450 |
| l(m) | | | | |
| | self-weight | load type1 | | |
| p1(kN/m) | 20,00 | 20,00 | 20,00 | |
| p2(kN/m) | 20,00 | 20,00 | 20,00 | |
| P(kN) | 0,00 | 0,00 | 0,00 | 0,00 |
| ↑ kN | 18,26 | 98,47 | 34,35 | |
| U.L.S. | 31,59 | 132,93 | 48,56 | |
| ↶ kNm | 0,00 | 0,00 | 0,00 | |
| opti. h. | | | | |
| mini. h. | | | | |
| mini. b. | | | | |

Figure 20 : Structure and loads window – distributed load

An extra line is added in the **Structure and loads** window (Figure 20). The begin values of the distributed loads will be indicated on the line which is preceded by “p1 (kN/m)” in the left margin; the end values on the line below, preceded by “p2 (kN/m)”.

When entering the data, the program will ask two questions for two values : "begin value distributed load nr. 1 zone z1.1?(kN/m)" and "end value distributed load nr. 1 zone z1.1?(kN/m)".

It is important to know that we can modify this option at any time: this means that even when the beam is completely entered with a uniformal distributed load, we can still choose trapezoidal loads and change possible begin or end values of the distributed loads by simply clicking the value.

3.2.2. Concentrated forces

In order to enter a **concentrated force** we should provide a division between two zones where the concentrated force is situated. A **zone** is a part of a span in which there are no discontinuities neither in the loads nor in the geometry.

This means that we should foresee two zones in the second span, that is to say one left and one right of the concentrated force. When entering data, the program asks, after input of the begin and end values of the distributed force in the first zone of the second span (z2.1), the place of the end point of that zone. The place of the end point of a zone is the distance between this point and the left support of the span to which the concerning zone belongs. In our example this distance is 2 m.

If we now also enter the value of the concentrated force, our **Structure and loads** window will be like in Figure 21.

We then continue to enter the beam in the usual way. The interpretation of the results in the different window is similar to that described in chapter 2. In the bottom right corner of the **Reinforcement sketch** window (Figure 22) we notice a button with a ">"-sign. When we click the mouse on that button, the reinforcement sketch of the second span appears. The button with the ">"-sign has now disappeared. At the same time a button with a "<"-sign has become visible ; by clicking this sign we can return to the previous span. As stated in chapter 2, the begin and end points of the indicated reinforcement are the points of zero of the shifted bending moments diagram.

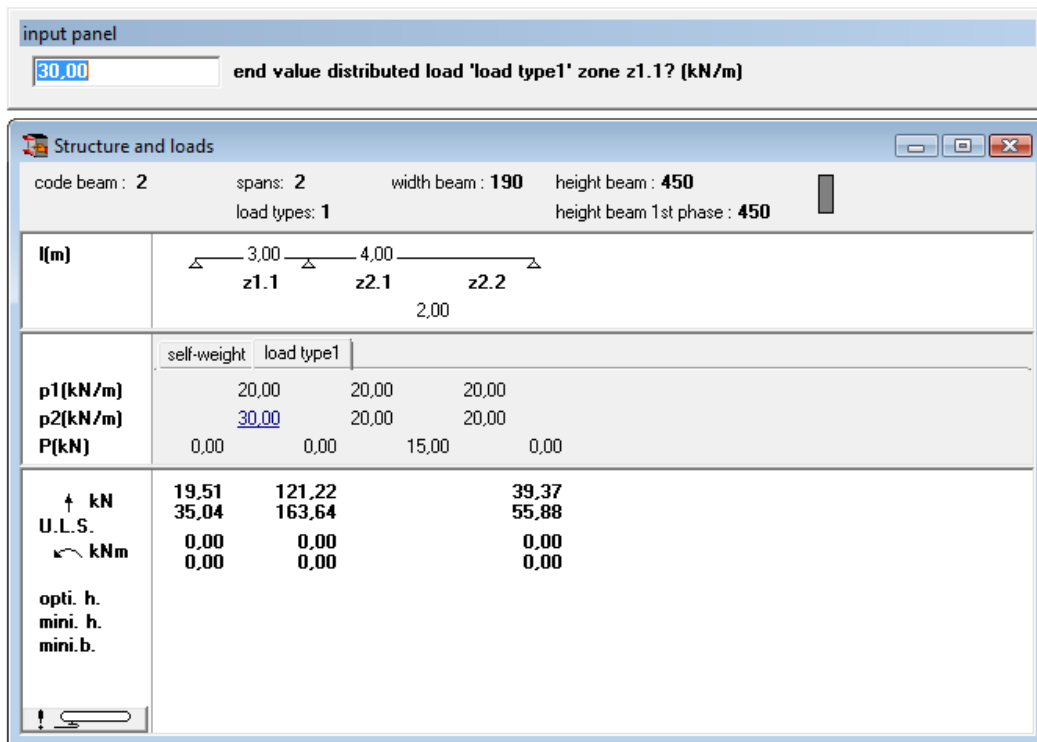


Figure 21 : Structure and loads window – input loads example 2

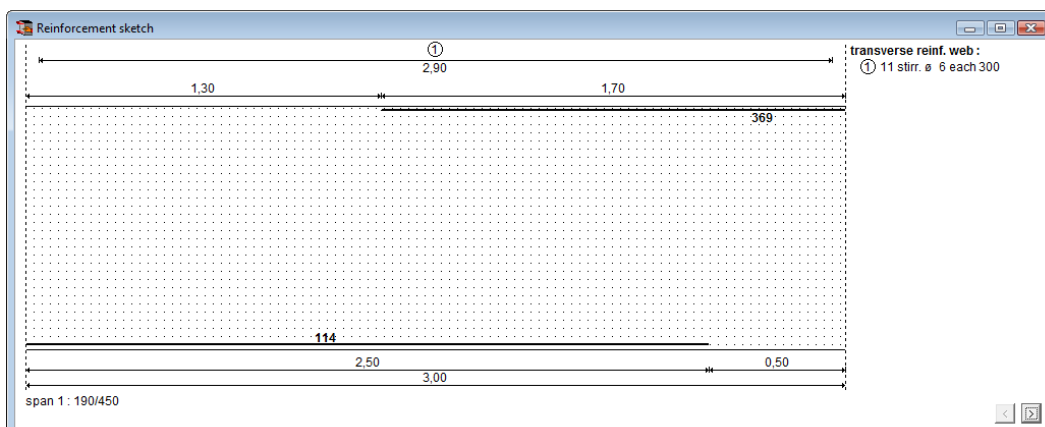


Figure 22 : Reinforcement sketch – example 2

3.2.3. Modifying the amount of zones

3.2.3.1. Modifying directly

If the data must be changed after the entry, you can do this on several ways.

Once the beam is fully implemented, just click the bold text below the sketch of the beam (eg "z2.1" span 2, Zone 1) with the mouse to add the zones. The screen then displays the dialog box shown in Figure 23.

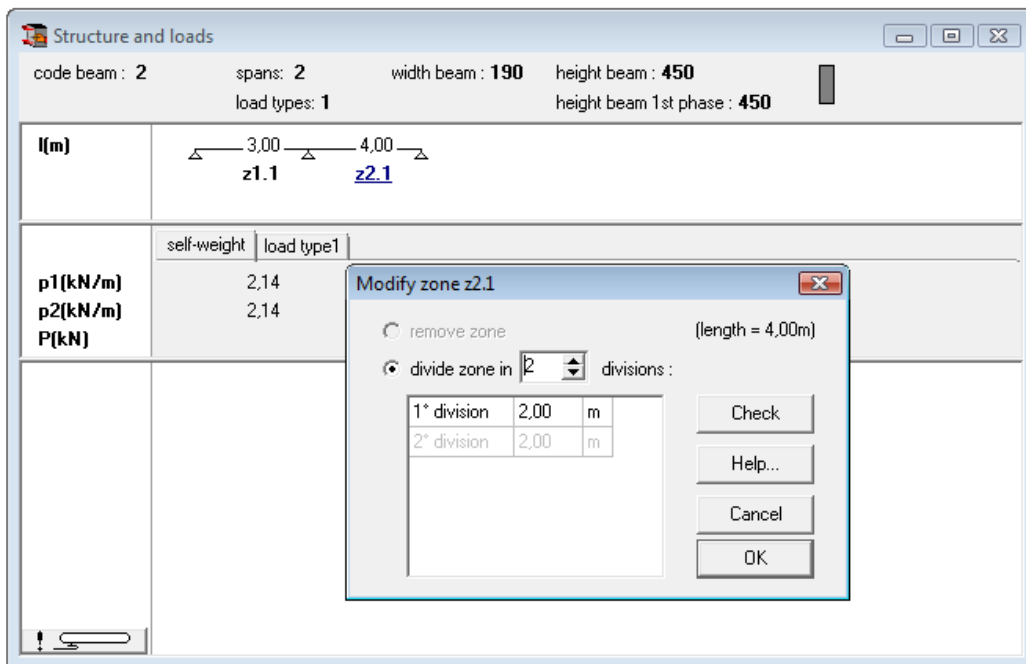


Figure 23 : Modifying a zone

If for example, the zone should be split into 2 parts, then click on the arrows until a 2 appears in the box next to it. The program automatically creates a list of the number of zones and with a uniform distribution of the length (Figure 23). We may wish to make an unequal division, by changing the sections length.

The button '**Check**' allows you to check the entered values.

If the designated area is already part of a span which has multiple zones, we can choose to delete the indicated zone (Figure 24). When the first zone is removed, the second zone will start from the left end of the span. If you remove another zone then the first one, then the previous zone will be extended until the end point of the removed zone.

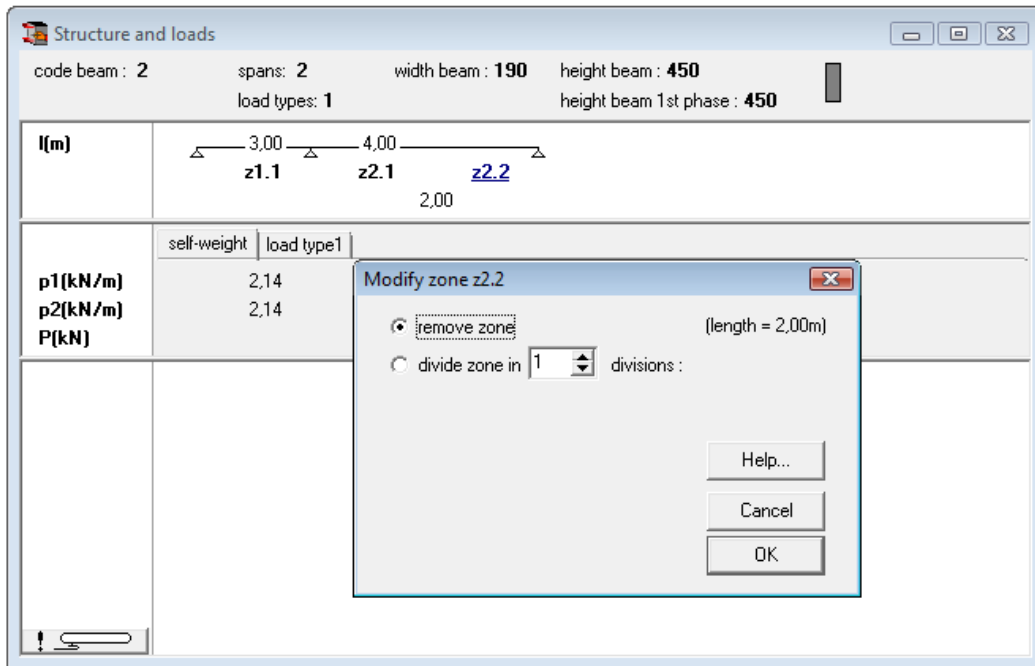


Figure 24 : Modifying a zone 2

3.2.3.2. Modification by using the right mouse button

The zones can also be manipulated using a popup menu. To do so, right-click the name of the zone:

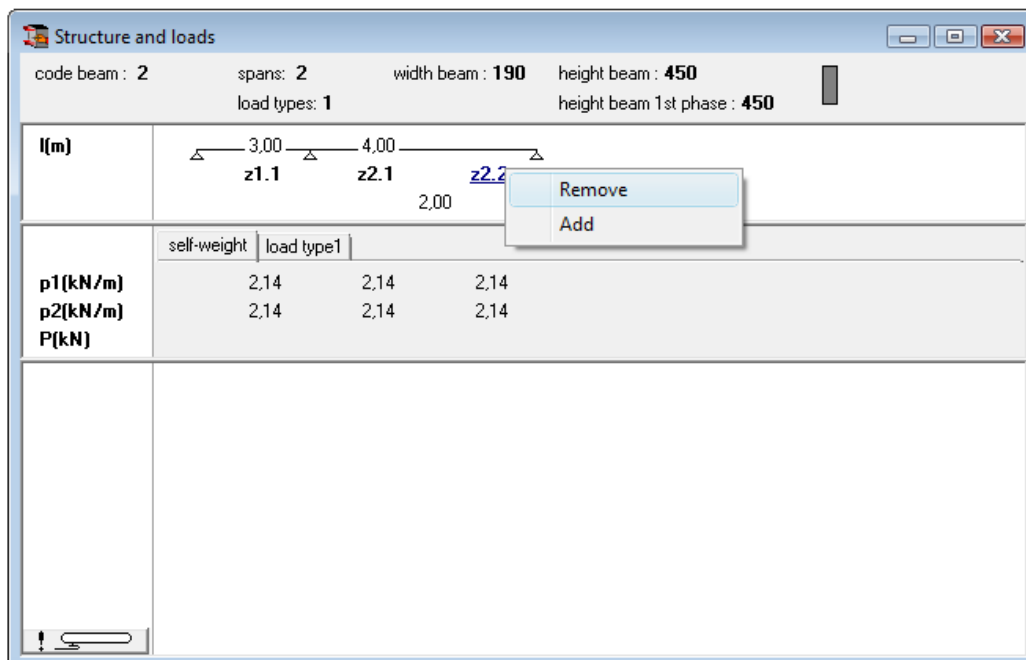


Figure 25 : Popup menu - zone

The first item "**Delete**" will only be selectable if the span consists of more than one zone.

The item "**Add**" allows you to create an additional zone. The function divides the designated area into two equal parts.

3.3. Adding concentrated moments

Suppose that we not only wish to enter a concentrated force but also a concentrated moment with a value of 10 kNm where the concentrated force is situated. We do so by selecting the input choice "**concentrated moments**" via the third icon in the second pallet of Figure 4 (Figure 26).

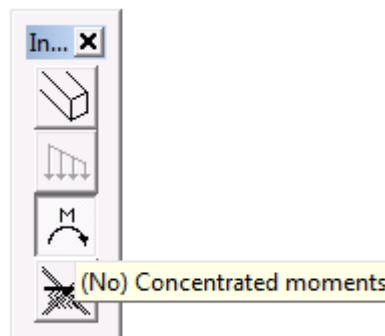


Figure 26 : Pallet 2 – Moment button

A line, preceded by "M(kNm)", is added to the **Structure and loads** window under the line of the concentrated forces (Figure 27). This line serves for entering concentrated moments. We can now enter the value "10.00" for the concentrated moments by selecting the set value "0.00" under the value "15.00" of the concentrated force.

Concentrated moments are considered as positive when working clockwise. This option, just as the option for trapezoidal forces, can be changed at any time. If this option is already selected when entering data, the program asks for the concentrated moments when the entrance takes place.

Please pay attention to the fact that we can also enter concentrated forces and moments at the location of the supports. Concentrated forces on supports can be very important for beams which are supported elastically or when taking the loads of the upper columns into account. The importance of concentrated moments on supports is obvious.

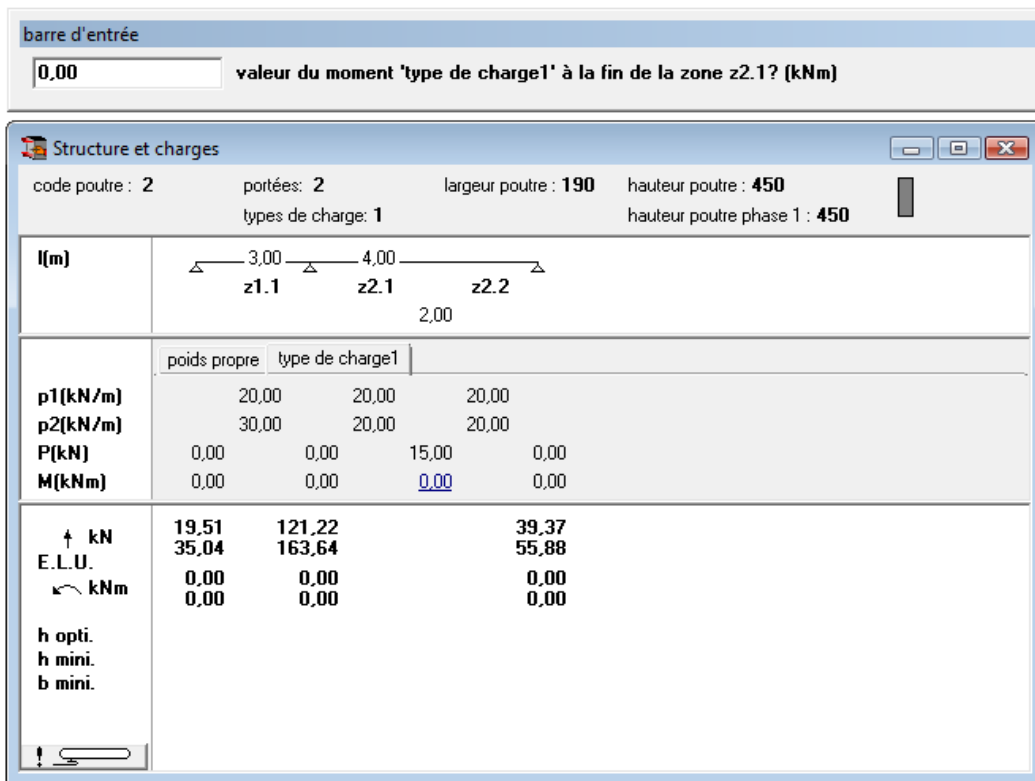


Figure 27 : Structure and loads window – adding a concentrated moment

3.4. A beam with a variable section

So far we have calculated beams with a constant height and width over their entire length. In the practice, cases appear in which e.g. the section is not the same for all spans, or even in which the section within a certain span is not constant. ConCrete enables you to define another section for each zone. We enter the possibility of variable sections by pushing the first button in the second palette of Figure 4 (Figure 28).

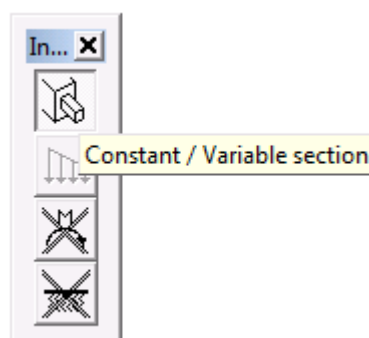


Figure 28 : Pallet 2 – Constant / variable section button

| Structure and loads | | | | |
|---------------------|---------------|------------------|-----------------------------|------|
| code beam : 2 | spans: 2 | width beam : 190 | height beam : 450 | |
| | load types: 1 | | height beam 1st phase : 450 | |
| l(m) | | | | |
| b(mm) | 190 | 190 | 190 | |
| db(mm) | 0 | 0 | 0 | |
| h(mm) | 450 | 450 | 450 | |
| ho1(mm) | 450 | 450 | 450 | |
| dh(mm) | 0 | 0 | 0 | |
| | 2,00 | | | |
| | self-weight | load type1 | | |
| p1(kN/m) | 20,00 | 20,00 | 20,00 | |
| p2(kN/m) | 30,00 | 20,00 | 20,00 | |
| P(kN) | 0,00 | 0,00 | 15,00 | 0,00 |
| ↑ kN | 19,51 | 121,22 | 39,37 | |
| U.L.S. | 35,04 | 163,64 | 55,88 | |
| ↶ kNm | 0,00 | 0,00 | 0,00 | |
| opti. h. | | | | |
| mini. h. | | | | |
| mini. b. | | | | |

Figure 29 : Structure and loads window – Variable sections

Five lines are added in the **Structure and loads** window (Figure 29). The indications in the left margin have the following meaning (Figure 30):

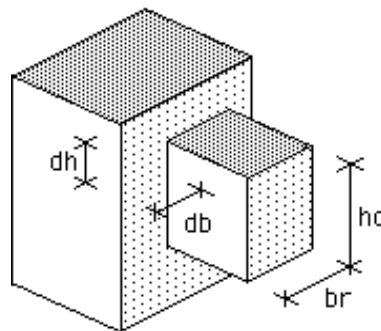


Figure 30 : Distances of a variable section in ConCrete

- b (mm): width of beam zone in mm;
- db (mm): step backwards from the previous front level to this one;
- h (mm): height of beam zone in mm;
- ho1 (mm): height of the beam zone which will be deposited in first phase (e.g. the height of the precast section, see also 7.3). Standard h01 is taken equal to the height ho.
- dh (mm): step downwards from the previous top level to this one.

The values for "b (mm)", "h (mm)" and "dh (mm)" can be changed in the same way as all other data in the **Structure and loads** window. The value of "db (mm)" cannot be changed; it is calculated as being half of the difference in width with the previous zone. The reason is that an asymmetrical jump in the width causes torsion and ConCrete does not include torsion in its calculation.

If we e.g. change the height in zone z2.1 to 350 mm, the word "VAR" is automatically filled out at the place in the **Structure and loads** window where the general valid height of the beam is indicated. This means that the beam has a variable height. At the same time a recalculation of the beam has taken place. We can see the jump in the **Reinforcement sketch** window. In the Diagram window, the development of the required reinforcement is clearly represented (Figure 31).

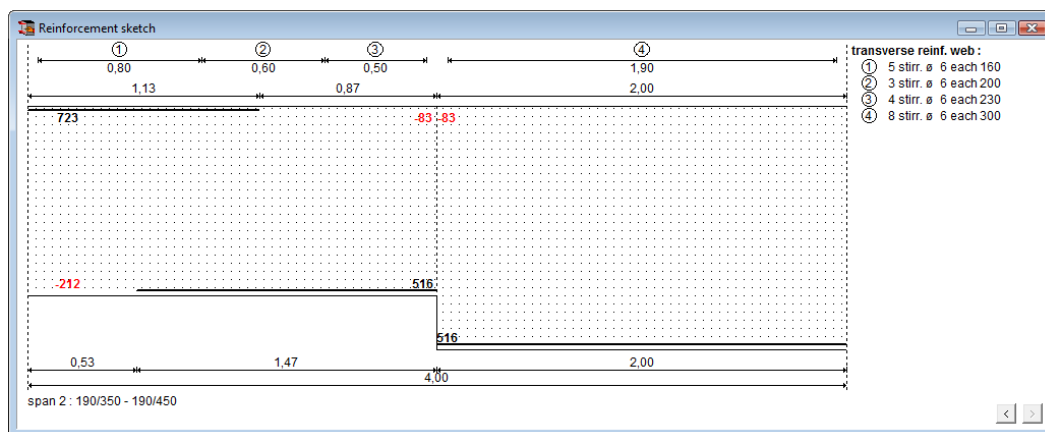


Figure 31 : Reinforcement sketch – variable section

Some considerations :

1) When determining the reinforcement section at the place of the change of section the program takes the fact into consideration that the diagram of moments has to be shifted over a distance δ (Figure 33). This means that the reinforcement section required at a certain place, is the maximum within a zone with length 2δ (centralized around the considered point) of the reinforcement sections calculated from the unshifted diagram of moments.

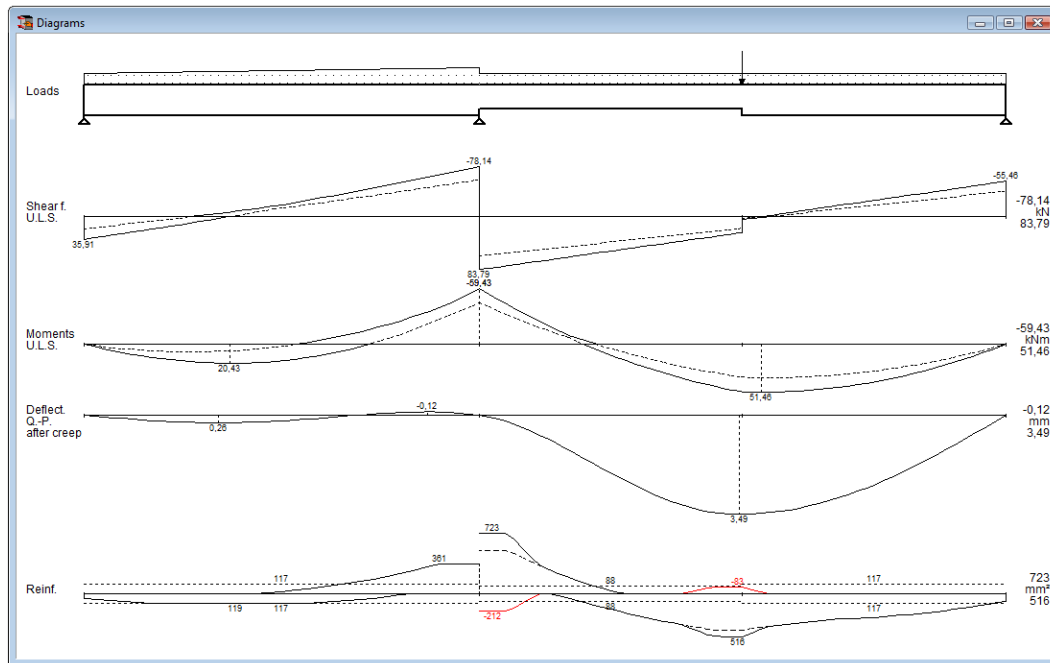


Figure 32 : Diagrams – example 2

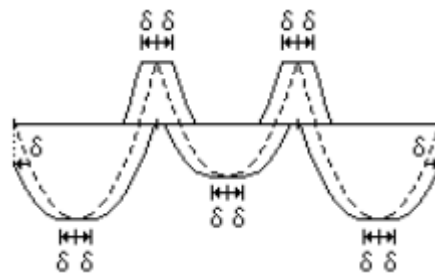


Figure 33 : Shifted moment-line

2) The reinforcement sections are determined by also taking the fact into account that bars, which work at the bottom of the bigger beam section, only become effective after acquiring a certain length. This length is determined by the anchorage length increased with the transfer length needed to transfer forces from one bar level to the other (Figure 34). The program takes as anchorage length a constant value of 60 cm because we have not yet determined bar diameters. In the zone with length equal to the transfer length plus 60 cm, ConCrete assumes for the determination of the required mm^2 reinforcement that the working section varies linear.

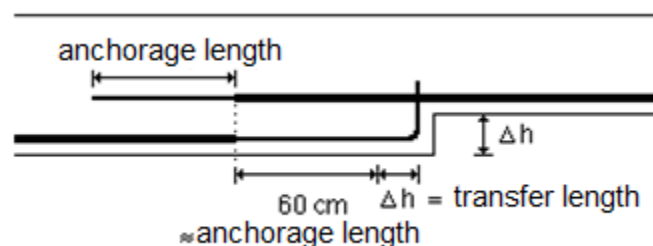


Figure 34 : Reinforcement – anchorage and transfer length

The jump in the required reinforcement diagram is due to the presence of the external moment of 10 kNm.

Chapter 4 Calculation parameters

4.1. Project values

So far we have calculated beams reinforced concrete without e.g. asking ourselves the question, out of which steel quality the reinforcement bars are made. This does not mean that the program does not take that into account ; it is however not necessary to set all calculation parameters when starting the calculation of an element. ConCrete automatically uses the standard set values as basic values. We call these basic values the **project values**. It is however always possible to deviate from these project values at any time and even to set new values as project values. Each time we start ConCrete the calculation parameters are set equal to the project values. If we enter a new element, the calculation parameters remain equal to the current values of at that moment.

We will always find 5 buttons at the bottom of the following dialog boxes (Figure 35).

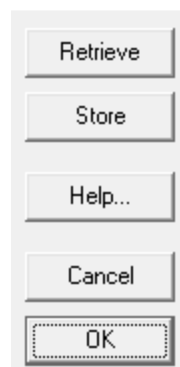


Figure 35 : Dialog boxes – the 5 basic buttons

The current values are set by pressing the **"OK"** button. There appears a dialog box (Figure 35) in which you indicate on which elements you want to apply the changed settings.

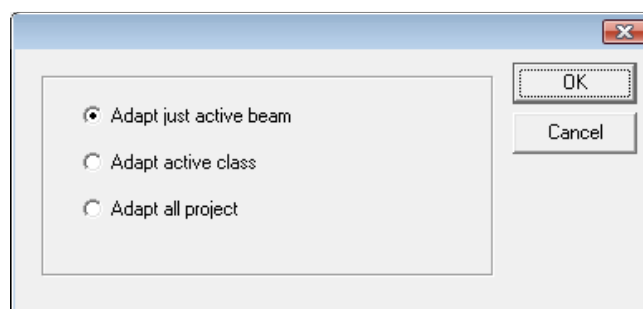


Figure 36 : Adjust current values

Current values can be set as project values by pressing the **"Store"** or **"S"** button. When the current values deviate from the project values they can be set equal by pressing the **"Retrieve"** or **"R"** button. The **"Cancel"** button does not change the current values.

4.2. Parameters for reinforcement steel

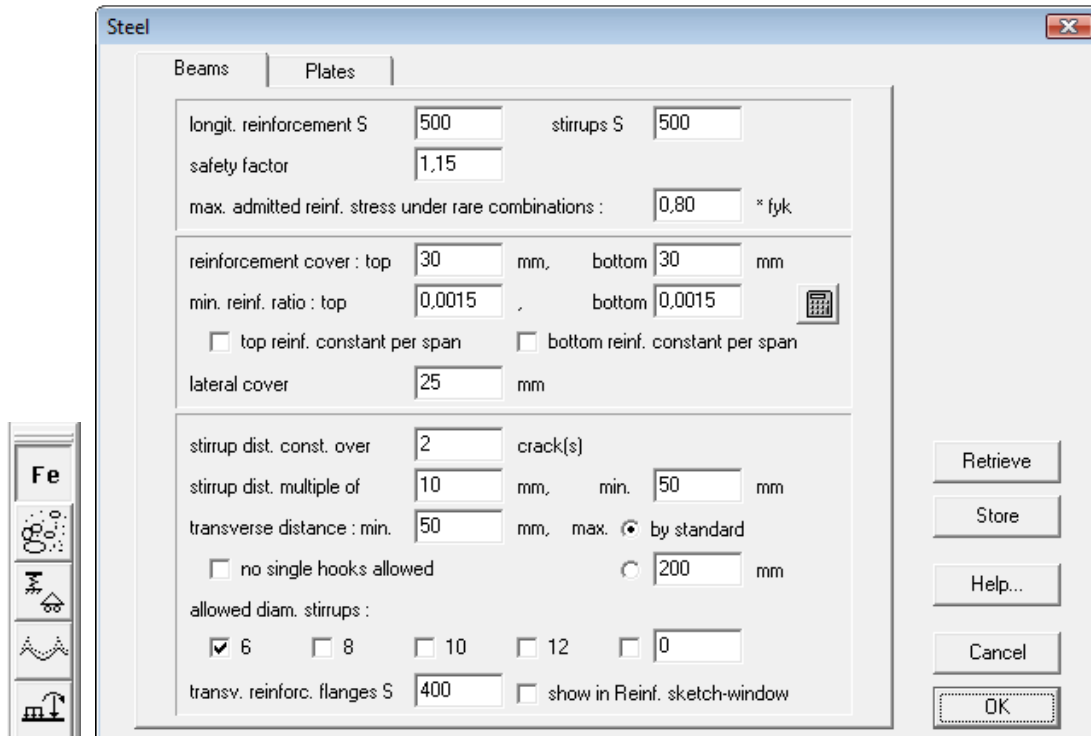


Figure 37 : Parameters for reinforcement steel

We can take a look at and change the parameters of the reinforcement steel by clicking the first icon of the first palette of Figure 3: now a new dialog box appears on the screen (Figure 37). In this dialog box we can change all the parameters of the reinforcement steel freely. The values of the steel quality are characteristic values and are expressed in N/mm². The safety factor is the factor which has to be applied to the characteristic values in order to obtain the design value of the steel quality. This design value will be used in the ultimate limit state. In the serviceability limit state under the rare combinations the maximum stress can be limited to a fraction of the yield stress (80% according to Eurocode). The cover is the net distance between the bar and the outer surface of the concrete, as indicated in Figure 38. When calculating the effective depth, the real height of the beam is decreased with this cover and with 10 mm extra to take the thickness of the reinforcement bars into account (which are unknown at that moment).

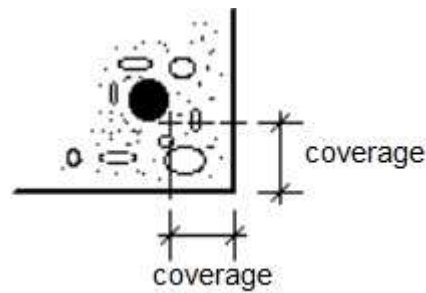


Figure 38 : Sketch of the concrete coverage

The minimum reinforcement ratio is taken into account by the calculation of the crack width and the cracked deformation. Both bottom and top reinforcement can be taken constant over the whole span length. The amount of reinforcement has a favourable effect on the crack width and on the cracked deformation. The meaning of "crack" is related to the truss girder analogy of Mörsh, in which a reinforced concrete beam is visualised as a truss girder with inclined concrete compressive struts, vertical stirrups and horizontal longitudinal reinforcement. The height of the truss girder is 0.9 times the effective depth. The reason for the inclined concrete compressive bars is that shear force cracks always have a slope. The horizontal length of such a tear is function of the slope. By keeping the stirrup distance constant over a certain number of cracks, we obtain a much more practical stirrup distribution.

The minimum distance between stirrups can be set, and this in longitudinal and transverse direction. This enables you to opt for a lower longitudinal distance or for multiple stirrups. If the calculation shows that the distance should be smaller than the minimum set distance, the program keeps the calculated distance, unless it is possible to obtain a larger distance with a bigger stirrup diameter. If so, the program will choose the bigger diameter.

Apart from the minimum transverse distance, we can also set a maximum transverse distance. This is normally defined by the standard. Most standards demand that the maximum transverse distance remains less than the effective height, which for almost square sections leads to multiple stirrups. To avoid this, you can impose a proper maximum (e.g. 200 mm). In case where one stirrups isn't enough, the program will try to solve the problem by adding a single hook in order to have a 3 bars – stirrup. You can however force the program to apply two stirrups by checking the option "**no single hooks allowed**".

The bar diameters that may be used for the stirrups, can be indicated by the user. Apart from the standardised diameters 6, 8, 10 and 12, a free value can be entered.

For beam sections with flanges (T, I-, L- and Z-sections, see also 7.1) the steel quality of the shear reinforcement between the flange and the web can be indicated as well as the fact whether this reinforcement should be represented in 'Reinforcement sketch' window.

4.3. Parameters for concrete

We can set the parameters for concrete with the second icon of the first palette of Figure 3 (Figure 39).

The screenshot shows a dialog box titled "Concrete" with the following parameters and options:

- char. compr. strength (fck): 25 N/mm²
- 2nd ph.: 25 N/mm²
- safety factor: 1,50
- modulus of elasticity: 30472 N/mm²
- creep coefficient ϕ (t, to): 1,29
- Buttons: Ecm,28, n = 15
- include self-weight (25 kN/m²)
- with flanges
- tensile strength for calculating crack width and deflection: 2,56 N/mm²
- Buttons: fctk, fctm
- maximum admitted compressive stress width under the rare combinations: 15,00 N/mm²
- after creep
- Button: 0,6 fck
- maximum admitted compressive stress under the quasi-permanent combinations: 11,25 N/mm²
- after creep
- Button: 0,45 fck
- take concrete shear contribution for: 50 %
- maximum admitted crack width under the quasi-permanent combinations: 0,30 mm
- after creep
- calculate cracking allways under rare combinations
- Buttons: Retrieve, Store, Help..., Cancel, OK

Figure 39 : Parameters for concrete

The characteristic compressive strength f_{ck} is the value of the strength above which we find 95% of the collection of all possible strength measurements exercised on concrete specimen. This compressive strength is determined after 28 days, on cylinders of 150 mm diameter and 300 mm height, kept under water at $20 \pm 2^\circ\text{C}$. As for steel, the safety factor is the factor which should be applied to characteristic values in order to obtain the design value of the concrete quality. This design value will be used for the calculation in the ultimate limit state. The modulus of elasticity can be entered freely or can be calculated in function of the concrete quality by clicking on the button "Ecm,28". The value of the E-modulus is then calculated with the formula:

- Eurocode 2 and NBN B15-002:

$$E_{cm,28} = 9500 \cdot (f_{ck} + 8)^{1/3}$$

- B.A.E.L. 91:

$$E_{cm,28} = 11000 \cdot f_{ck}^{1/3}$$

To take the creep effect into account by the calculation of the deformation and by the limitation of the stress and the crack width, the creep factor $\phi(t, t_0)$ can be introduced. Traditionally the stresses are limited for a ratio between the E-moduli of steel and concrete equal to 15: $n = \frac{E_s}{E_c} = 15$. The value of the creep factor $\phi(t, t_0)$ that corresponds with this ratio can be calculated by clicking on the button "**n = 15**". This value is not the final creep factor. When using this value, the result for the deformation will theoretically be underestimated. However, if we use all recommended values, the calculated deformation exceeds the experimental result.

We can also choose to take the self weight of the beam into account, with or without the weight of the flanges for T-, I-, L- and Z-sections. The specific gravity of concrete is set to 25 kN/m³.

As stated before, the deformation can be calculated taking the cracks into account, i.e. using decreased stiffness. For this calculation, the value of the concrete tensile strength is important. The value can be set free or be calculated as the mean (f_{ctm}) or characteristic ($f_{ctk} = f_{ctk0.05}$) value in function of the compressive strength. These 2 values are calculated with the next formulas:

- Eurocode 2 en NBN B15-002:

$$f_{ctm} = 0.30 \cdot f_{ck}^{2/3}$$

$$f_{ctk0.05} = 0.7 \cdot f_{ctm}$$

- B.A.E.L. 91:

$$f_{ctm} = \frac{f_{ctk0.05}}{0.7}$$

$$f_{ctk0.05} = 0.6 + 0.06 \cdot f_{ck}$$

According to research done at Magnel Laboratory from the Ghent University, both these lead to a higher calculated deformation than the test results. In that research, they conclude that the value of 3.56 N/mm² (for C25/30) which is proposed in the American Standard, is more in comparison with the test results. It's our opinion that without too much risk, the deflection can be calculated with the previous mentioned factor $n = 15$ and with the mean tensile strength $f_{ctm} = 2.56$ N/mm².

Further on, you can impose a limit to the concrete stress in the serviceability limit state under rare combinations and under quasi-permanent combinations. According to Eurocode 2, the limitation under rare combinations is only needed for climate class 3 or 4, and the limit is then $0.6f_{ck}$. You can use the button " $0.6f_{ck}$ " to put this limit in the edit box. Unless the full load is put on the structure almost immediately after it is casted, it is normal to impose this limit after creep. For structures where creep is important, a 45% limit under quasi-permanent combination is recommended, because otherwise the creep effect will be underestimated by the use of a creep factor. If the maximum concrete compressive stress is superseded, than the program will increase the amount of tensile reinforcement and add some compression reinforcement.

4.4. Parameters for the supports

We set the parameters for the supports with the third icon of the first palette of Figure 3. The dialog box of Figure 40 appears on the screen.

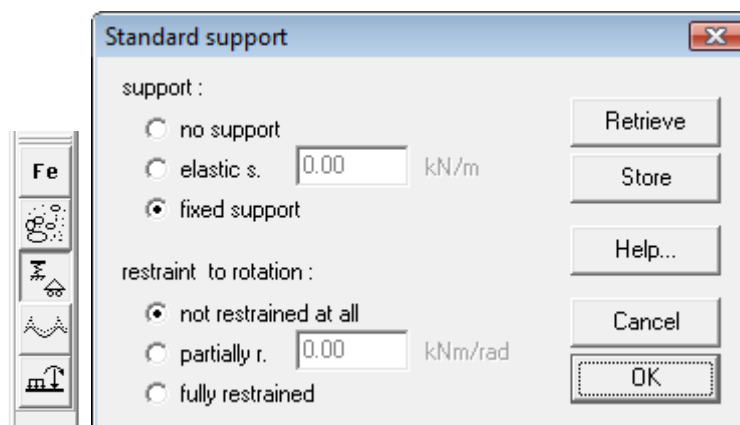


Figure 40 : Standard support

All the introduced changes are related to all the supports. Therefore it is impossible to leave out the support. If we want to change only one support, we click with the mouse in the support sketch in the **Structure and loads** window. We obtain the same dialog box as in Figure 40, valid for this one selected support (Figure 41).

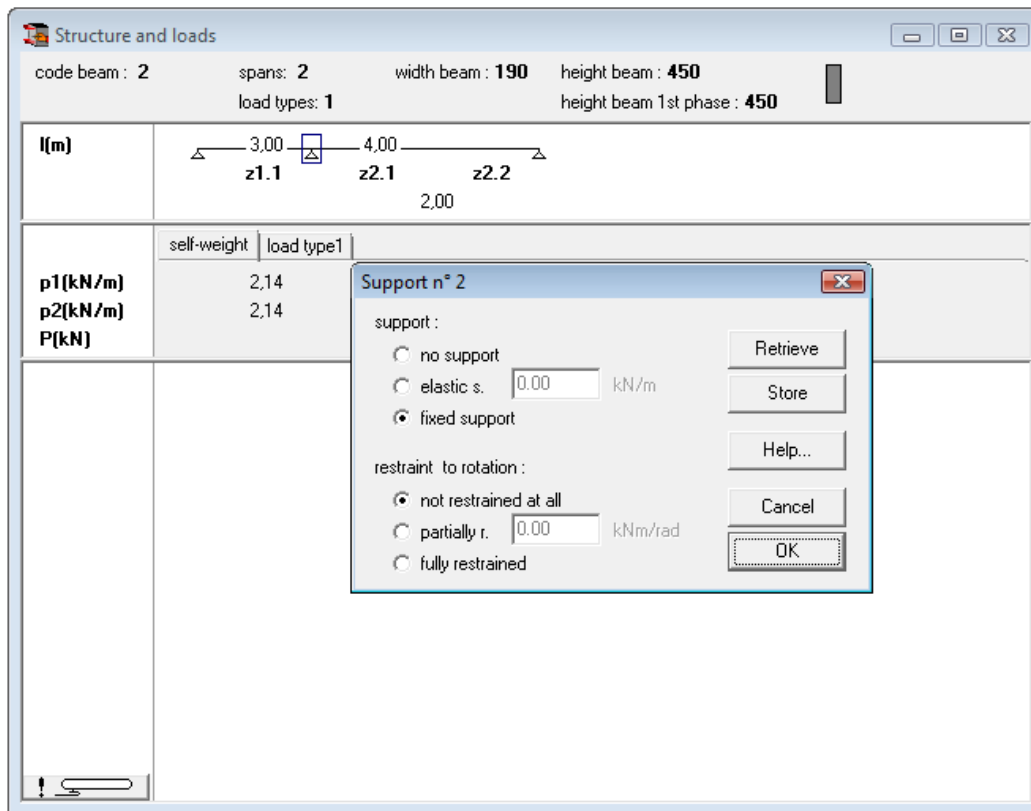


Figure 41 : Adjusting a support

It is however possible to leave out an end support. In that way we can make a cantilever from a simple supported span. Except for **cantilevers** and **simple supports**, we can also enter **translation springs**, **rotation springs** and **fully restrained supports**.

There is however a second possibility to enter a cantilever, namely by entering a negative value for the span length of an end span. ConCrete automatically interprets a negative span length as a cantilever with as length the absolute value of the entered length.

4.5. Parameters for the redistribution of moments

We select the fourth icon of the first palette of Figure 3 if we want to take a look at or change the calculation options of the **redistribution of moments**. Just as for the geometrical parameters of the supports, these options apply to all the supports (or only all the intermediate supports).

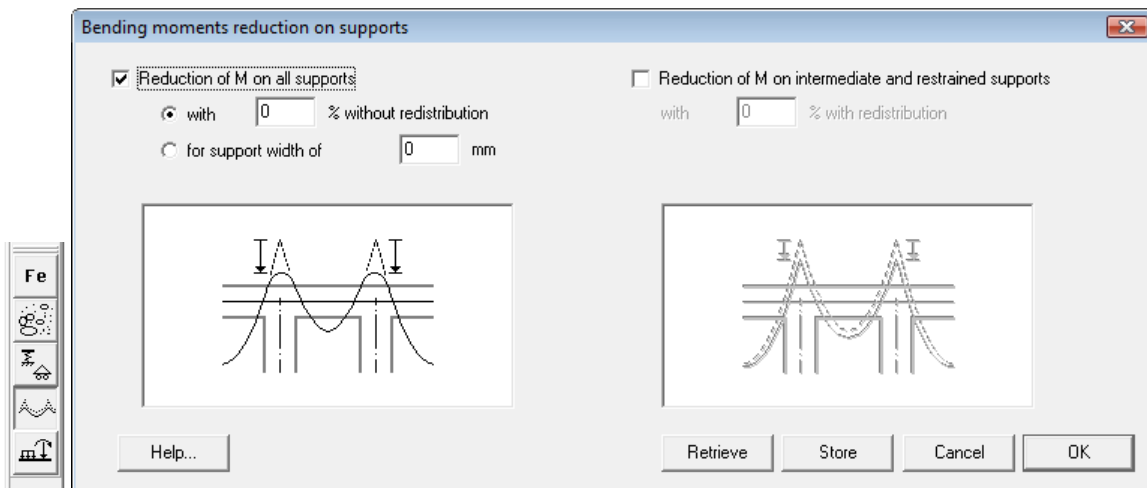


Figure 42 : Parameters for the redistribution of moments

The first option in the dialog box (Figure 42) causes no change in the diagrams, it only produces a reduction of the reinforcement moments. The reason therefore is that with larger supports, the top of the diagram of moments can be cut in order to determine the support reinforcement (Figure 43).

Concrete allows you to reduce the time by either:

- By giving up a percentage
- By giving up the width of the supports.

The reduction of the moment ΔM is calculated as follows:

$$\Delta M = \frac{F_{sup} \cdot b_{sup}}{8}$$

In which F_{sup} is the reaction force in the support and b_{sup} the width of the support.

On the contrary, the second option does change the diagrams by decreasing the moments on the intermediate supports with a percentage entered by the user and by increasing at the same time the moments in the spans so that the obtained diagram of moments is, from a static point of view, a correct solution for the entered loads (Figure 44). This implicates that the shear force line is also changed.

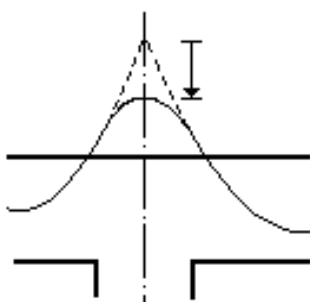


Figure 43 : Moment reduction without redistribution

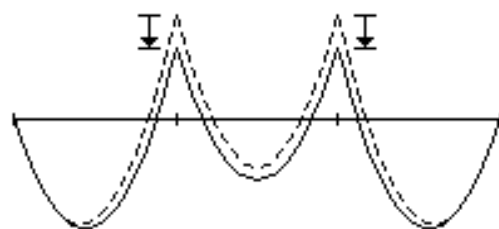


Figure 44 : Moment reduction with redistribution

If we however want to apply a different moment reduction for each support, we keep the Control key pressed when clicking with the mouse on the support sketch in the Structure and loads window. We obtain the same dialog box as in Figure 42, now only valid for this one selected support (Figure 45). The second option has been left out for end supports: there the inner forces are merely determined by the statical balance. Any modification of the diagrams is unacceptable at that location.

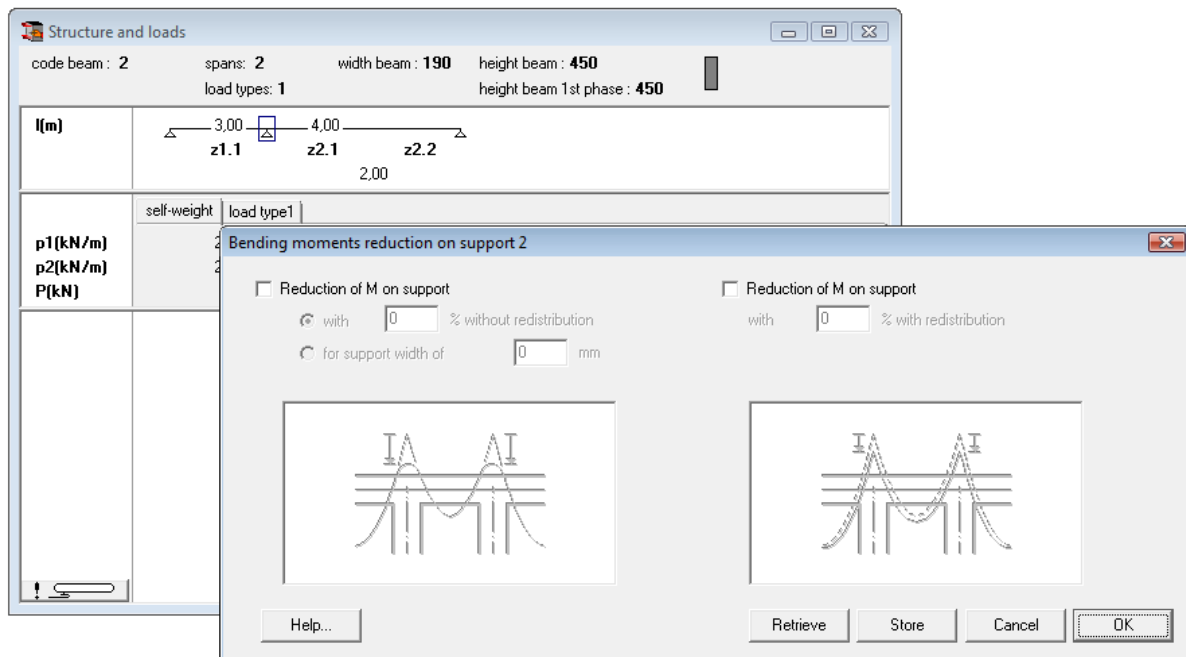


Figure 45 : Moment reduction on a support

4.6. Parameters for the loads

In addition to the self weight we can indicate in ConCrete nine load types. Each load type has its proper **load factors** and can be taken into account with a well determined **combination factor**. To set these factors we click with the mouse in the fifth icon of the first palette of Figure 3. The dialog box in Figure 46 is now visualized on the screen.

In this dialog box we can determine whether the loads will act on the top or bottom level, this is of importance when determining the transverse reinforcement (cfr. Classical truss girder analogy of Mörsch).

| | | γ_{u-} | γ_{u+} | γ_{g-} | γ_{g+} | Ψ_0 | Ψ_1 | Ψ_2 | 1st ph. | 2nd ph. | |
|-------------------------------------|---------------|---------------|---------------|---------------|---------------|----------|----------|----------|--------------------------|-------------------------------------|--|
| <input checked="" type="checkbox"/> | 0 self-weight | 1,35 | 1,00 | 1,00 | 1,00 | 1,00 | 1,00 | 1,00 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 1 load type1 | 1,35 | 1,00 | 1,00 | 1,00 | 1,00 | 1,00 | 1,00 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 2 load type2 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 3 load type3 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 4 load type4 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 5 load type5 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 6 load type6 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 7 load type7 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 8 load type8 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |
| <input checked="" type="checkbox"/> | 9 load type9 | 1,50 | 0,00 | 1,00 | 0,00 | 0,70 | 0,50 | 0,30 | <input type="checkbox"/> | <input checked="" type="checkbox"/> | |

the loads apply on the :

top level
 bottom level

factor for calculating crack width and deflection :
 β_2 :

allways together
 all combinations

Retrieve Store Help... Cancel OK

Figure 46 : Parameters for loads

There are six factors for each load type :

γ_{u-} is the load factor which should be applied to determine the diagrams of the moments and of the shear forces in the ultimate limit state when the load has an unfavourable effect on the concerning diagram;

γ_{u+} has the same functionality as γ_{u-} except that the load has a favourable effect on the concerning diagram;

γ_{g-} is the load factor which should be applied to determine the diagrams of the rotation angles and of the deflections in the serviceability limit state when the load has an unfavourable effect on the concerning diagram;

γ_{g+} has the same functionality as γ_{g-} except that the load has a favourable effect on the concerning diagram;

Ψ_0 is the combination factor which should be applied in the ultimate limit state (fundamental combination) and under the rare combinations in the serviceability limit state when another variable load is the most unfavourable;

Ψ_1 is the combination factor which should be applied under the accidental combinations in the ultimate limit state and under

the frequent combinations in the serviceability state when this variable load is the most unfavourable.

Ψ_2 is the combination factor which should be applied under the quasi-permanent combination in the serviceability limit state. Under the accidental combination in the ultimate limit state and under the frequent combination in the serviceability state this factor is applied when another variable load is the most unfavourable.

The load on a cantilever e.g. is favourable to the moments in the span beside ; for the calculation of these moments the program will apply on this load the factor γ_{u+} ; for the moment in the cantilever however, the program will apply the factor γ_{u-} . For the determination of the application of the favourable or unfavourable factor in order to calculate the moment in a determined point, the program assumes that the considered load within a span, is present either with the favourable or with the unfavourable factor. When determining the moment in an arbitrary point x_1 , the load which continues over several spans can be present in one span with the unfavourable factor and in another span with the favourable factor. In another point x_2 these factors can be the other way round. The program finds out for itself which factors should be applied. It is obvious that what we considered as valid for the diagram of moments is also valid for the other diagrams.

When determining the fundamental combination in the ultimate limit state (U.L.S.) and the rare combinations (R.C.) in the serviceability limit state, the most unfavourable variable load is taken with a combination factor 1, all other variable loads are taken with their combination factor Ψ_0 . Just like for the load factors, the program finds out by itself which variable load is the most unfavourable for the considered point. In the quasi-permanent combination all variable loads are applied with their combination factor Ψ_2 . Permanent loads are always combined without combination factor (i.e. combination factor = 1).

Please notice that when determining the load types we do not make a difference between **permanent** and **variable loads**. This means that we also have a favourable and an unfavourable factor for permanent loads as well as for variable loads. For variable loads we will enter the value "0" as favourable load factor, for permanent loads this value will always be around "1", e.g. "0.9".

The difference between permanent and variable is only made by entering other values for the load- and combination factors.

For each load type it is possible to indicate in which phase(s) the loads of this type are present. The **first phase** is calculated in an *isostatic* way (span by span), possibly with a reduced height (h_{o1} instead of h_o). The required reinforcement for the ultimate limit state is calculated. Then, the beam is calculated in the **second phase**. In this phase, the beam is calculated in its *hyperstatic* condition, with the full height (h_o). Again, the reinforcement for the ultimate limit state is calculated. The final result for the ultimate limit state is the maximum of the first and second phase. Starting from this reinforcement, the stresses in the serviceability limit state in the concrete and in the reinforcement are calculated in the first phase (thus in an isostatic way and with reduced height). Then the increase of stresses in the second phase is determined. If the stresses are too high, the reinforcement is increased.

With the parameter “**always together**” you can force the program to consider the loads of a load type always as one undividable load. Otherwise, ConCrete will automatically load some spans with the favourable load factor γ_+ and other with the unfavourable load factor γ_- to obtain the most damaging load situation. It can be however that the nature of the load makes it impossible to vary the load over different spans between favourable loaded and unfavourable loaded. Think e.g. at a swimming pool that can be filled with water or not. Suppose this pool is supported by continuous beam with 3 spans. The water should be introduced as a variable load with load factors $\gamma_{u-} = 1.50$ and $\gamma_{u+} = 0.00$. Normally, the program will include a load situation where the first and third span is loaded and the second is not. In practice, this is not a possible situation : or the water is in the pool and then, all 3 spans are loaded, or the is not in the pool and then, all 3 spans are unloaded. By indicating that the loads of the load type are always together present or not, the program will not take into account these 2 load conditions (all spans loaded and all spans unloaded).

Chapter 5 More than one load type

5.1. Introduction

The beams which we have entered so far, were indeed submitted to different loads, but they all belonged to the same load type. In practice however, it often occurs that there are loads of different types. Think about permanent loads, service loads, snow loads, wind loads and so on... According to the load type different load and combination factors are appropriate. We saw in 4.6 how to tell the program to which load type a certain load belongs.

5.2. A beam with two load types

We wish to calculate a beam with two spans and with two load types : a permanent load p of 20 kN/m and a variable load q of 10 kN/m. The beam is drawn in Figure 47.

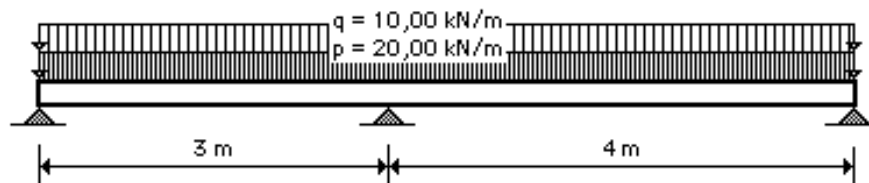


Figure 47 : Sketch of example 3

The set width of the beam is set to 190 mm, the height to 500 mm. Reset the input options of the second pallet (Figure 4) to their initial state : constant section, uniformly distributed loads and no moment loads (no lowered buttons).

The factors are entered as in Figure 48.

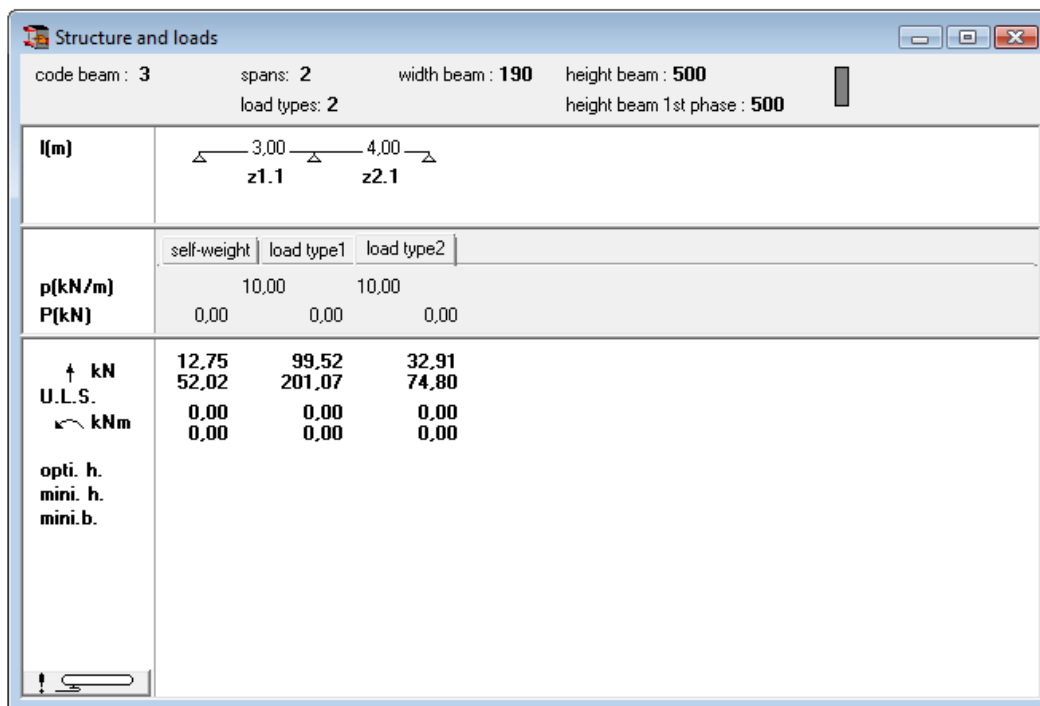


Figure 48 : Structure and loads window - example 3

When entering the data, we indicate that we will enter two load types. The input is done as usual. After entering the value of the first distributed load in the last zone of the last span, the program automatically asks us the value of the second distributed load in the first zone of the first span (Figure 48). Meanwhile the rectangle, in which the loads are visualised, is cleared in the **Structure and loads** window and the counter indicates "2". This tells us that the values which we see in the rectangle in question, are those of the second load type. The former grey arrow above the figure "2" has now become a white arrow with black border. We can go back to the previous load type by clicking with the mouse in this arrow. The arrow beneath "2" is aimed at showing a possible next load type in the rectangle. The functionality of both arrows is analogous to those of the classical scroll bar.

After having entered all data, ConCrete does the calculations as stated in 4.6. As a result we obtain envelope diagrams for the bending moment and the shear forces (Figure 49). We also obtain a coexistence of the top- and bottom reinforcement (Figure 50). This coexistence does not mean that there is no a shiftment of the diagram of moments and no anchorage length requested for the determination of the lengths of the longitudinal bars. As said before, ConCrete Plus takes this automatically into account.

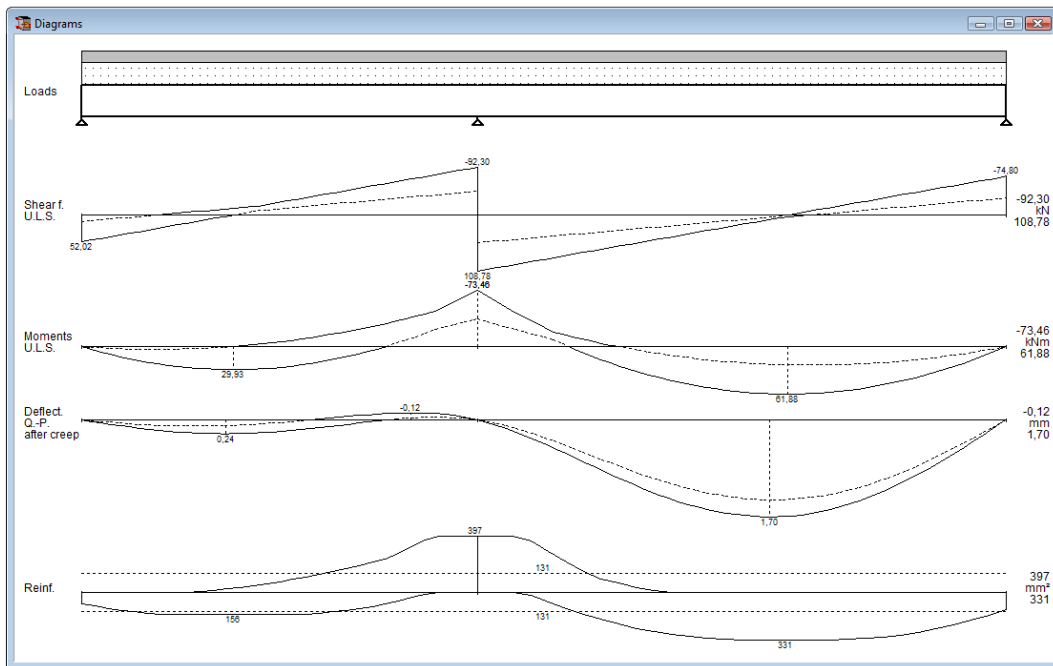


Figure 49 : Diagrams - example 3

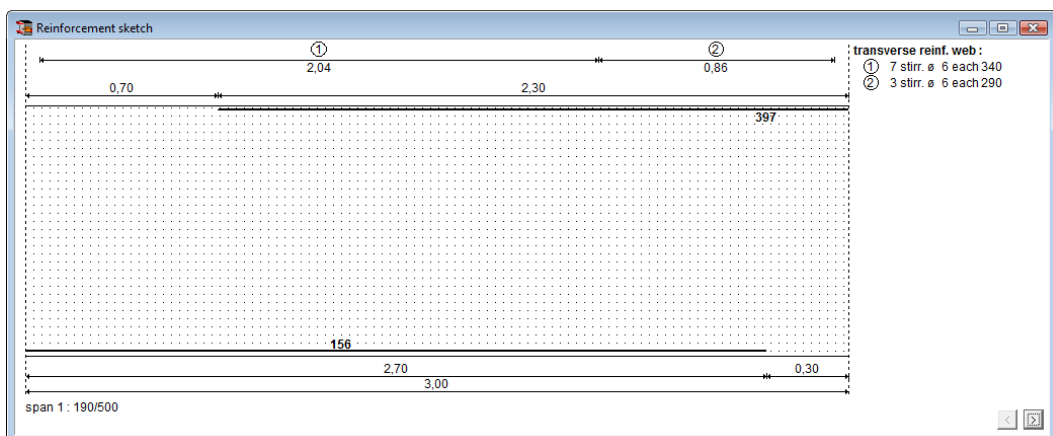


Figure 50 : Reinforcement sketch - example 3

5.3. Adding and removing load types

5.3.1. Modify directly

Click on the number of load types in the "**Structure and loads**"-window and enter a new value. Modification is possible as long as there is no load entered yet, or once the whole input of data is ready. When the number of load types is lower than before, the latest load types are removed; when the number is higher, additional load types are added with all values equal to 0.

5.3.2. Modify with right mouse button

A popup-menu shows the possibilities:

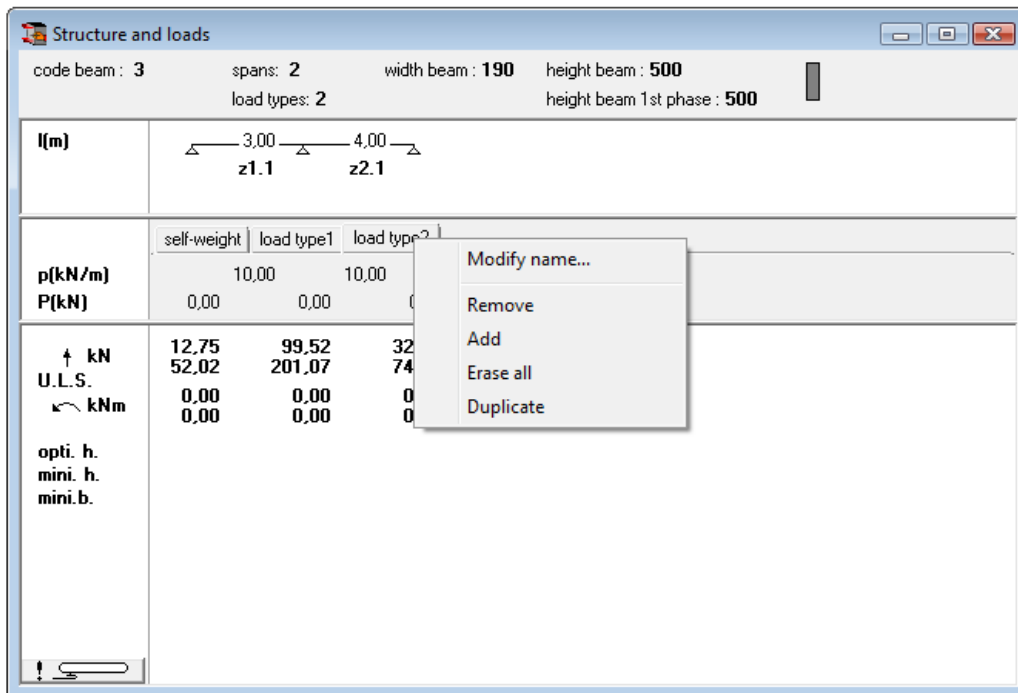


Figure 51 : Popup menu – loadtypes

It is recommended to verify the load and combination factors after changing the load types (see 4.6).

Chapter 6 An entire project of beams

6.1. Introduction

So far we have learned how to calculate a beam with ConCrete. In chapter 3 we noticed (3.1) that when entering a new beam, the previous entered beam is placed in the **Project list**-window. In this chapter we will learn more about the structure and the use of this project list.

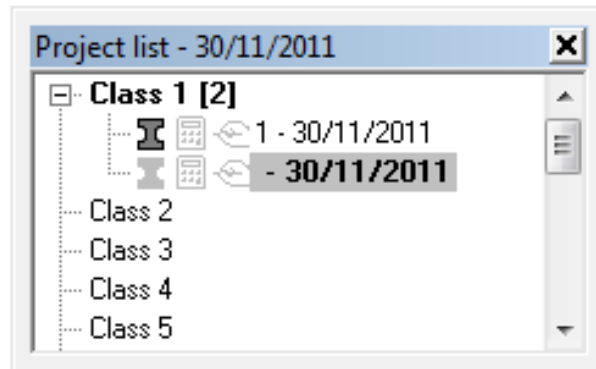





Figure 52 : Project list

The list is constructed as a tree. The main items are the classes, the subitems are the elements. Currently, only one element is calculated. The second bar has yet to be defined. Both elements are in the first class. The date that the element was last opened is mentioned behind its name. Moreover, the most recent date is listed next to the header of the window itself.




The name of the active element has a grey background colour, to distinguish this element from the other ones.

Three icons accompany the code of the element:



The **first icon** indicates what kind of element was introduced:

-  beam
-  slab
-  hollow core slab [only with the hollow core slab module]

The **second icon** indicates if the element is calculated and / or its dimensions are enough:

-  not calculated yet
-  calculated but section is insufficient;
-  calculated and section is sufficient;

The **third icon** indicates whether or not the element is calculated and drawn with the aid of ConCrete Plus:

-  not drawn yet;
-  drawn;

Suppose that we have a project consisting of three blocks A, B and C each containing a basement, a ground floor and ten standard floors. If we consider all the beams of this project we can easily get a hundred different beams.

We can however divide the beams in groups. We can e.g. consider groups according to the block or the floor. We call such a group a **class**. We can define up to 20 classes in ConCrete each consisting of a maximum of 100 beams (and/or slabs).

The elements are automatically ranged according to their name (order: ABC...XYZabc...xyz012...789). The name of a class is limited to 256 characters.

6.2. Defining classes

6.2.1. Choosing classes

We will distribute the beams out of the example described in 6.1 over the following classes:

- KA : all the elements of the basement of block A
- GA : all the elements of the ground floor of block A
- TA : all the elements of the standard floor of block A
- KB : all the elements of the basement of block B
- GB : all the elements of the ground floor of block B
- TB : all the elements of the standard floor of block B
- KC : all the elements of the basement of block C
- GC : all the elements of the ground floor of block C
- TC : all the elements of the standard floor of block C

6.2.2. Input classes

We define the classes in ConCrete as follows : we select the last command "**List of classes...**" in the **Edit** menu. The dialog box of Figure 53 is now visualised on the screen.

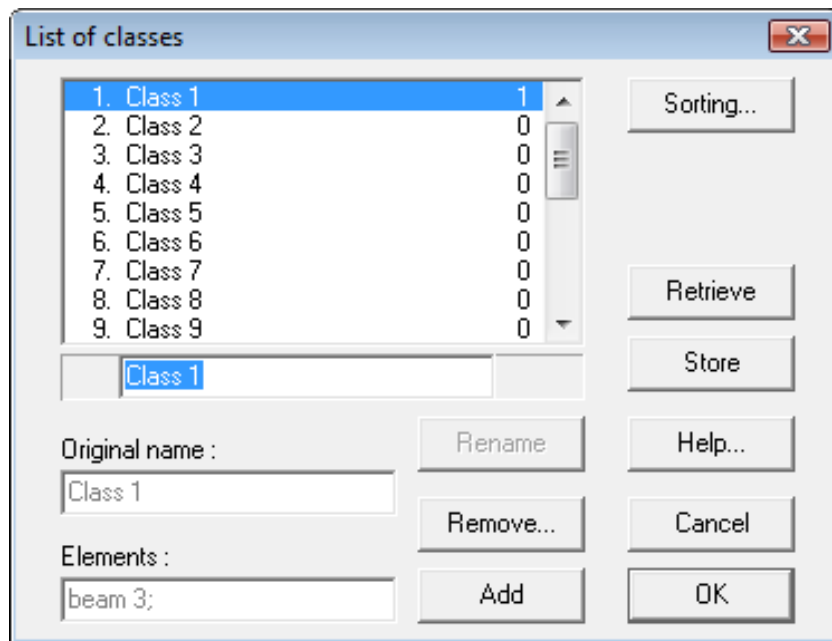


Figure 53 : Editing the list of classes

The names of the 20 classes are in a cadre (for the moment all blank). The meaning of the buttons "**OK**", "**Store**" and "**Retrieve**" at the bottom of the dialog box is the same as in 4.1.

By typing the word "KA" in the cadre right on top, the button "**Add**" can be selected. After clicking this button the name "KA" is added in the class list.

In a new project, where no elements have been entered, "KA" will be situated on the first line.

It is also possible to rename a selected class from the list. Just click the button "**Rename**" after selecting instead of the "**Insert**". Note that the original name of the class (the name that the class was when opening the dialog) in the box under "Original Name" is written. This may include the elements listed in the box under "Elements".

Now enter the other classes (Figure 54).

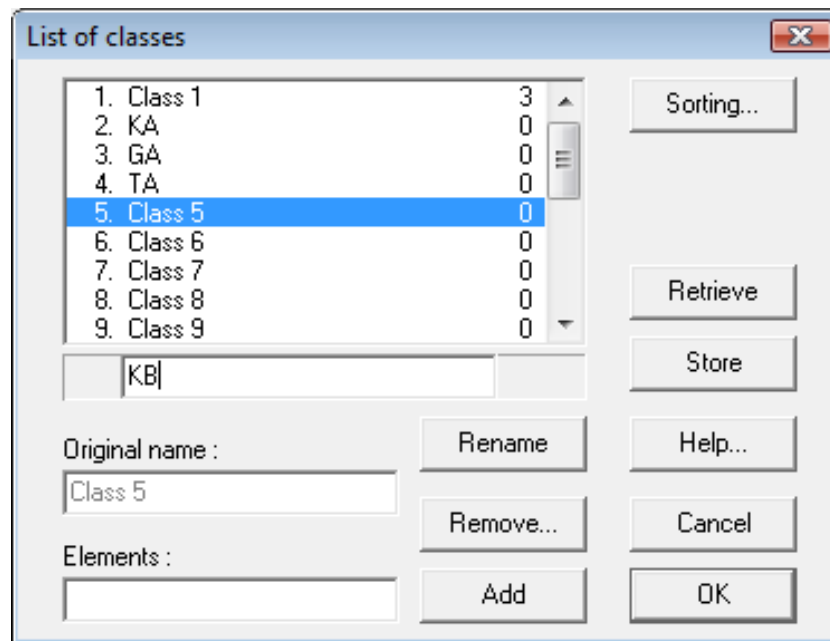


Figure 54: Editing the list of classes

The button "**Remove**" can only be selected when a certain class is selected in the list. With this button not only the name of the class is cleared out of the list but also all the elements (beams and/or slabs) that the class possibly contains.

6.2.3. Project list

After confirming the set classes, the **Project list**-window is visualised as in Figure 55.

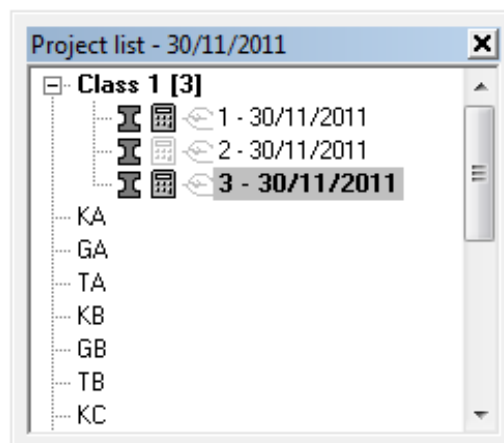


Figure 55: Project list – inserted classes

The list under "Class 1" contains the elements already entered.

Whether a class consist elements or not, is indicated by the mark in front of the class name. If there is a sign ('+' or '-'), then the class consists at least one element. Clicking the '+' will show all elements of that class,

the '+' becomes a '-'. Clicking the '-' hides the elements of the class. There is one exception: if the active element (the element currently shown in the window) is in the class.

If all elements should or shouldn't be visible, you can use the right mouse button (Figure 56).

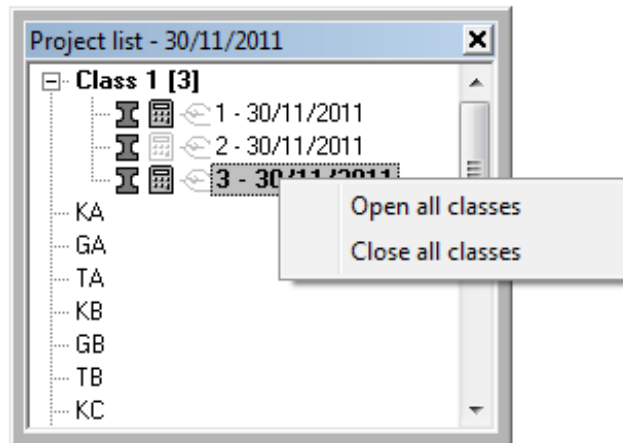


Figure 56: Pop-up menu – project list

6.3. Allocating an element to a certain class

When you create a new element, this will automatically be put in the selected class. If there are no classes defined, a standard class "class 1" will be automatically created and selected.

You can move an element from one class to another (see 6.5).

6.4. Getting an element out of a certain class

Are the elements belonging to the class not visible, double click then first on the class name or click on the '+' sign at the left.

To show and possibly modify the beam, we have two possible methods. First select the beam in the list and then use the command "**Element**" -

"**Edit**" or click on the icon , or just double click the beam in the list.


The selected beam is shown in the **Structure and loads** window, the **Diagrams** window and the **Reinforcement sketch** window.

6.5. Moving an element to another class

To move an element to another class, just select the element in the list and drag it to another class. In other words : select it with the mouse, keep the mouse button pressed, move the mouse to another class and release the mouse button.

6.6. Duplicating an element


Suppose that we have to calculate two elements which are nearly identical. In that case we could first calculate the first element, and then afterwards make a copy of it and modify this copy in order to obtain the second element. We can do so in ConCrete in a very easy way.

We first enter the first element and add it to the desired class as described in 6.3. Afterwards we select this element in the **Project list**-window. We then select the fourth command "**Duplicate**" in the **Element** menu or click on the icon . The selected beam is visualised in the windows "**Structure and loads**", "**Diagrams**" and "**Reinforcement sketch**"; nevertheless it is still present in the **Project list**-window. In fact we made an autonomous copy of the beam. This copy does not differ in any way from the traditionally entered beam. This means that we can make arbitrary modifications without influencing the first beam.

It is obvious that the time of duplicating the beam can be chosen freely : immediately after entering the beam or later.

In 6.9 is explained how elements can be copied from one project to another or from ConCrete to ConCrete Plus (and vice versa).

6.7. Deleting an element

To delete an element, we first select it in the **Project list**-window. We then select the fifth command "**Remove**" in the **Element** menu or click on the button . The element is now cleared out of the class.

6.8. Copying elements between projects

To copy an element from one project to another, first get the element as described in 8.3 and select then the "**Copy**" command in the "**Edit**" menu. Then, open the project to which you want to copy it. Select there the desired class and click then on the "Paste" command in the "Edit" menu.

Chapter 7 Special elements

7.1. T-, I-, L- en Z-beams

With concrete, it is not only to calculate rectangular cross-sections, but also T, I, L and Z-sections, and they may vary by zone. In case a beam with constant section, it can be entered in the "**Structure en Loads window**" with the mouse clicking in the top right rectangle (Figure 57). This rectangle is the outline of the beam cross section.

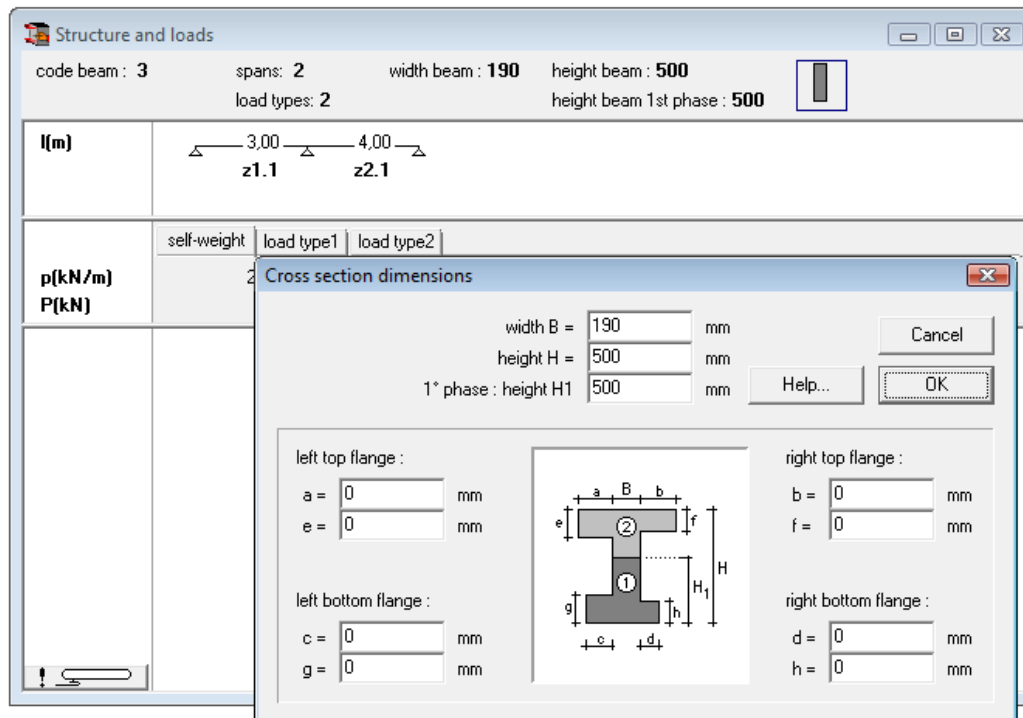


Figure 57 : Details cross section dimensions

In case of a beam with variable sections, the section can also be added per zone by clicking on the section sketch in "z_._" of the changing area (z2.1 in Figure 58), again the dialog Figure 57 appears, but only valid for the one zone.

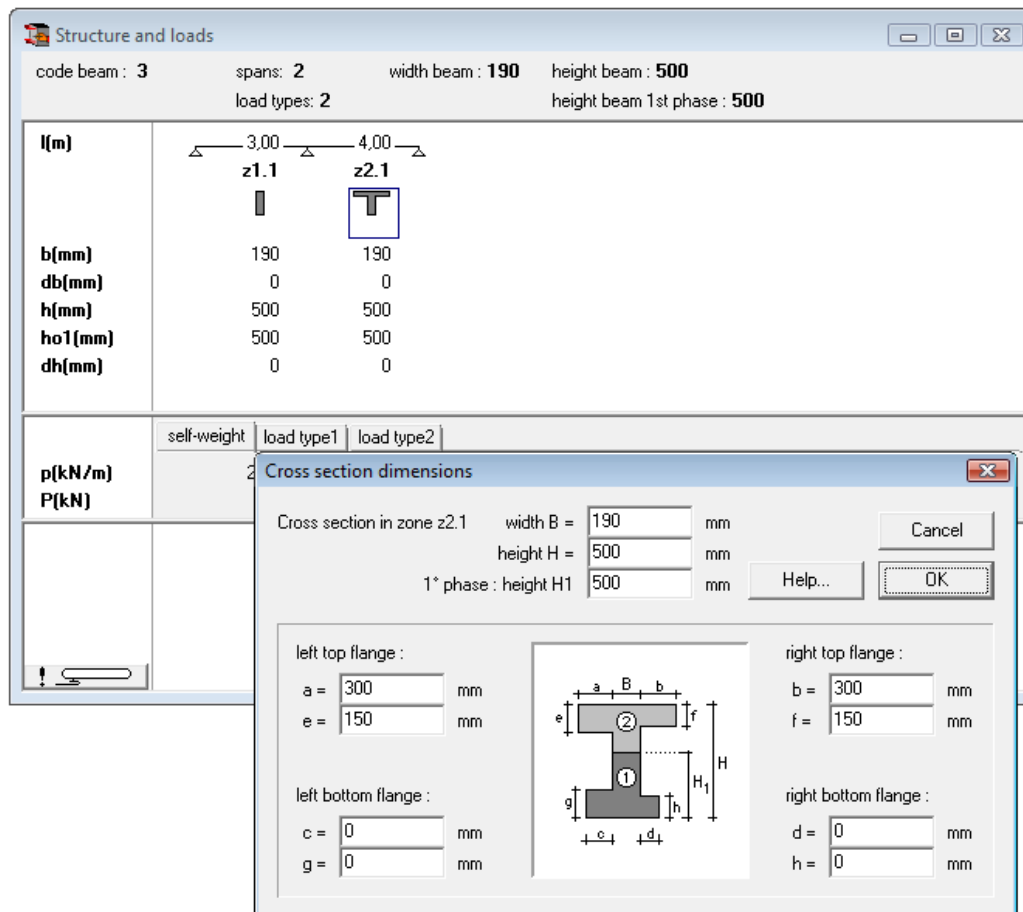


Figure 58 : Structure en loads window – variable sections

In this dialog box the dimensions of the flanges can be adapted.

Four flanges are possible: two upper flanges and two bottom flanges, one each left and right flange. The width of the flanges (the dimensions a, b, c and d) can be both a positive and a negative value, negative values are used to cut a rectangle from the web of the beam (a "bite" from the web, Figure 59), positive values indicate a flange on the body (Figure 60). The flange thickness (the dimensions e, f, g and h) have always positive values (or zero).

Note that the width, height and height 1° phase (see 7.3) can be changed.



Figure 59 : 'Negative' flange



Figure 60 : 'Positive' flange

As already noted in 4.3, in the dialog box "Parameters of concrete" you can indicate whether the weight of the flanges will be taken into account or not.

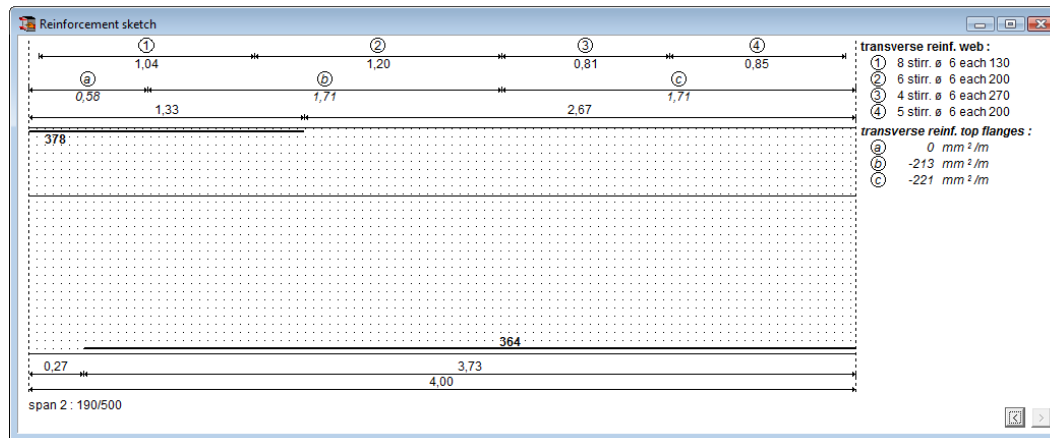


Figure 61 : Reinforcement sketch – detail upperflange

If in the dialog box for the parameters of the reinforcing steel (Figure 37) the option to display the transverse reinforcement in the flanges is checked, you will see the "Reinforcement Sketch" dialog example from above.

As for the transverse reinforcement in the body, there is a dimension line with the above circled letter "a", "b" and below the respective figures, all in italics. The circled letters are again referring to the information on the right, the numbers below are the distances over which these zones are valid. The right information includes the number of required reinforcing steel mm² (perpendicular to the plane of the drawing) per m length of the beam. Negative values give an indication of the amount of reserve. The designation of the reserve can sometimes be important: the standard requires only a weighted average of the distance between a zero point and a time limit. ConCrete, however, a more precise indication: areas where the moment increases faster, get more transverse reinforcement allocation than areas where less time is rapidly increasing. It is entirely possible that ConCrete indicates that a certain zone no transverse reinforcement is required (reserve) and in the zone next to it, while the averaging according to the standard no transverse reinforcement is required, hence the program indicates how much the reserve, so we these values can optionally averaging over all time zones located between zero and maximum. In case of unequal left and right flange, the shear reinforcement ConCrete proposes is valid for both flanges.

7.2. Beams on an elastic foundation

In previous examples the beams were calculated with only a few supports (solid or elastic). However, if a beam resting on land must be calculated, the discrete points will not simulate the continued support. Such beams are calculated classically known as beams on elastic foundation. The resisting character of the foundation (ground) is determined by giving the elastic constant k .

ConCrete goes a lot further than simply calculating a beam on elastic foundation. In ConCrete, it is even possible to give for each zone a different elastic constant k , and on the other hand, you can add discrete supports (fixed or elastic and/or fixed or elastic restraint to rotation). The importance to have a different elastic constant per zone is shown in the following problems:

- A beam which is posed on a homogenous ground, except that the ground is crossed by a little brook;
- a beam supported by a number of concrete columns and by a large wall in bricks;

In the first example, the presence of the brook can be incorporated by a lower value (or even a zero value) of the elastic constant. In the second example, the concrete columns can be entered as normal supports (elastic or fixed), while the large wall can be simulated by zone with a large elastic constant. In both examples, it would be very hard to obtain an even adequate result, if we could only calculate beams supported by either only discrete supports or a unique elastic foundation over the whole length.

Click in the second pallet on the forth icon (Figure 4, Figure 62) to be able to enter elastic foundations.



Figure 62 : Pallet 2 – Elastic foundation

In the "**Structure and loads**"-windows an extra line, proceeded with "k(N/cm³)" (Figure 63), is added under the line with the indication of the different zones. On this line you can enter the value of the elastic constant in the zone. The **elastic constant** gives the relation between the settlement and the reaction of the foundation:

$$q(x) = k \cdot v(x)$$

with

- q the reaction of the foundation (force / surface)
- k the elastic constant
- v the vertical displacement (settlement)

| |
|-----------------------|
| l(m) |
| br(mm) |
| db(mm) |
| ho(mm) |
| ho1(mm) |
| dh(mm) |
| k(N/cm ³) |
| x(m) |

Figure 63 : Structure and loads window – List of units

In lack of more accurate data, the value of the elastic constant k can be determined as follows:

- for clayish grounds:

$$k = \frac{0.5 \cdot \frac{q_r}{b} \cdot (l + 0.5 \cdot b)}{1.5l}$$

with

- q_r compressive strength of the ground in N/cm²
- b width of the beam in m
- l length of the beam in m

- for sandy grounds:

$$k = k_1 \cdot \left(\frac{b + 0.3048}{2 \cdot b} \right)^2$$

where the value of k_1 is given table 1.

| sand | low packed | medium packed | high packed |
|-------------------------------|------------|---------------|-------------|
| k_1 (in N/cm ³) | 2.7 – 8.1 | 8.1 – 40 | 40 – 136 |

table 1: elastic constant - value k_1 for sand

Higher stated values are taken from Campus F., "Contribution à l'étude des pièces fléchies dans le sol. Application aux pieux et palplanches", Mémoire du Centre d'Etudes, de Recherches et d'Essais Scientifiques du Génie Civil de l'Université de Liège, N°39, Avr il 1972.

7.3. Beams in 2 phases

Today more and more beams are partially precast. Once they are placed at their position, a second phase is casted in which the individual beams (per span) are casted together to form a continuous beam (possibly the are also cast together with the slabs).

In some cases, the precast beams are used to bear a part of the loads, without the help of additional struts. Special attention should be paid at the impositions ! The beams are prefabricated per span and will be loaded in an isostatic way by their selfweight and e.g. also the selfweight of the slabs posed on those beams and the selfweight of the second phase concrete. Once the beam is completely casted and hardened, the full section is effective and the spans are completely joined to form a hyperstatic construction (continuous beam).

In the program two phases can be taken into account. Therefore you have to enter next things:

- Phase 1: indicate for each section till which height the section will be casted in the first phase, in other words. this is the height of precast section. The program indicates this height as "ho1" and sets this height by default equal to the total height "ho". You can change these values by clicking in the "**Structure and loads**"-window on the value or on the cross section sketch (see 7.1) and then change it in the dialog box.
- Phase 2: In the dialog box for the load factors (see 4.6) you indicate which load types are acting on the precast section in the first phase, and which on the total section (section 2nd phase). Load types can act in both phases or only in one of them. Of course it is possible to have load types that act only in the second phase.

The phases are taken into account by the calculation of the reinforcement and of the stresses in the concrete and in the reinforcement, as explained in 4.6.

For the sake of completeness, we want to mention that the calculation of the crack width and the deformation (neither uncracked nor cracked) does not take these phases into account.

7.4 Slabs in 2 phases

ConCrete can, besides beam, can also calculate plates in reinforced concrete.

The plates are carrying in one direction and can also be executed in two phases (Figure 64).

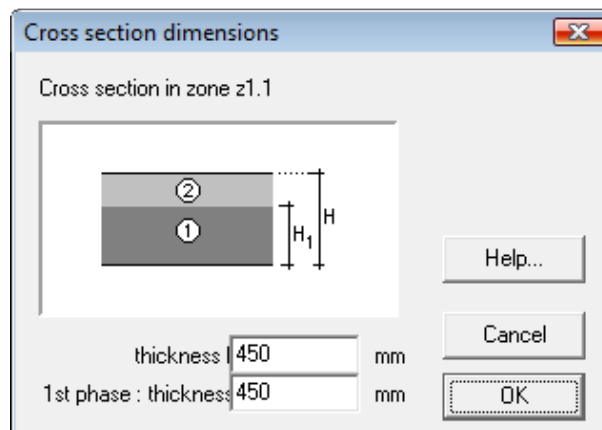


Figure 64 : Slab in 2 phases

7.5. Hollow core slab library

Remark: This functionality is not available in the standard version of Concrete. Only ConCrete Master allows you to calculate hollow core slabs.

Finally, ConCrete has the possibility to calculate hollow core slabs. These hollow core slabs are stored in the so-called hollow core slabs library. To do this, select the menu "**Edit**" command "**Hollow core slabs library**". A window appears (Figure 65) which you can define and manage the slabs.

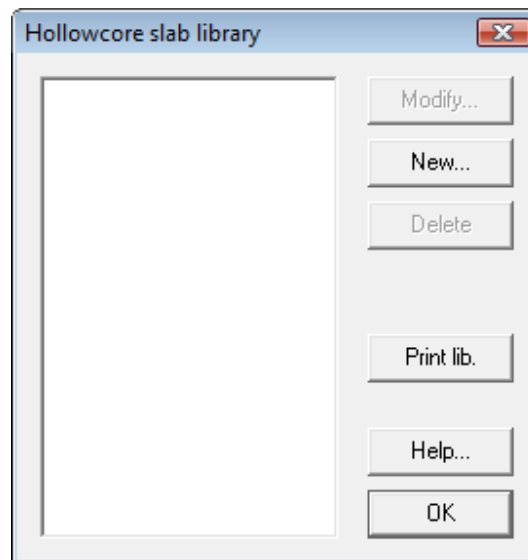


Figure 65 : Hollowcore slab library

With the button "**New**" you can make a new hollowcore slab. A dialog box (Figure 66) then prompts you to different dimensions. Click "**OK**" to confirm the geometry data and the hollowcore slab will be placed in the library.

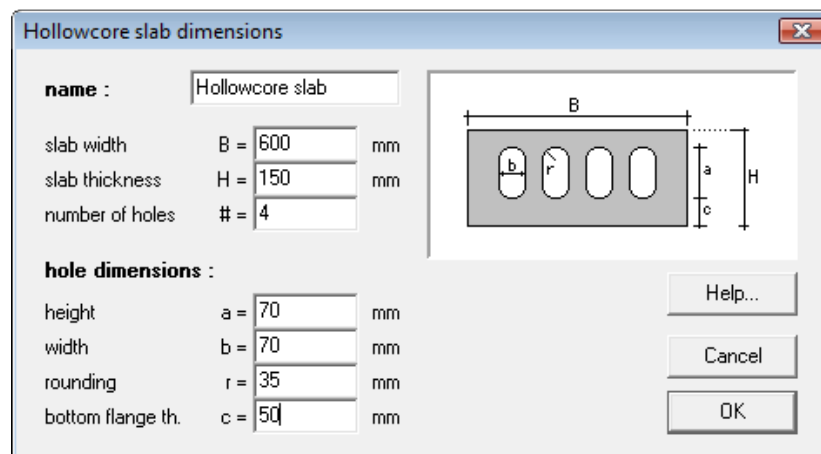




Figure 66 : Hollowcore slab library

Chapter 8 Saving, opening and quitting

8.1. Saving a new project

If we now wish to save the already accomplished work as a document on the hard disk or on another saving unit, we select the fifth command "**Save as...**" in the **File** menu or we click on the icon . The program suggests to save the project in a document with the same name as the set reference code of the project (see 2.2). We can however enter an arbitrary name if we want to. If we save the project under another name, this new name becomes the new title of the **Project list**-window.

8.2. Opening an existing project

To open an existing project in ConCrete, we select the second command "**Open...**" in the **File** menu or we click on the icon . The **Project list**-window has got the same name as the opened file. Besides all the documents created with ConCrete we can also open documents made with ConCrete and adapted with ConCrete Plus. If we e.g. open a project of 10 beams which are all of them already drawn in ConCrete Plus, we can modify the calculation of these beams or add another beam to this project. Beams which are already drawn in ConCrete Plus and of which the calculation is modified in ConCrete, do not however keep their corresponding reinforcement plan. The other beams of the project keep of course their plan.

8.3. Saving an existing project

There are two possibilities to save an existing project in ConCrete.

The first possibility consists of resaving the document under the same name by selecting the fourth command "**Save**" in the **File** menu. This command can only be selected when changes have been made in the earlier saved project. When saving, all data which were joined to the project in ConCrete Plus are also saved, with the exception of the in 8.2 discussed data. With the "**Save**" instruction files are always saved in binary format (type ConCrete).

The second possibility is saving the project under another name by selecting the fifth command "**Save as...**" in the **File** menu. This command can be selected at any time, in contrast to the previous.

8.4. Automatical saving

In order to protect ourselves against the loss of data by a breakdown or any other cause, we can activate the option "**Save automatically**" in the

Preferences – Save automatically menu. This option is marked by a "☑" – sign if its active; in the other case, the option is not marked. When the option is activated, the program will automatically save after certain actions.

To define after which actions the automatically save procedure should be performed, click on the "**Preferences...**" command in the **Preferences – Save automatically** menu. The following dialog box appears:

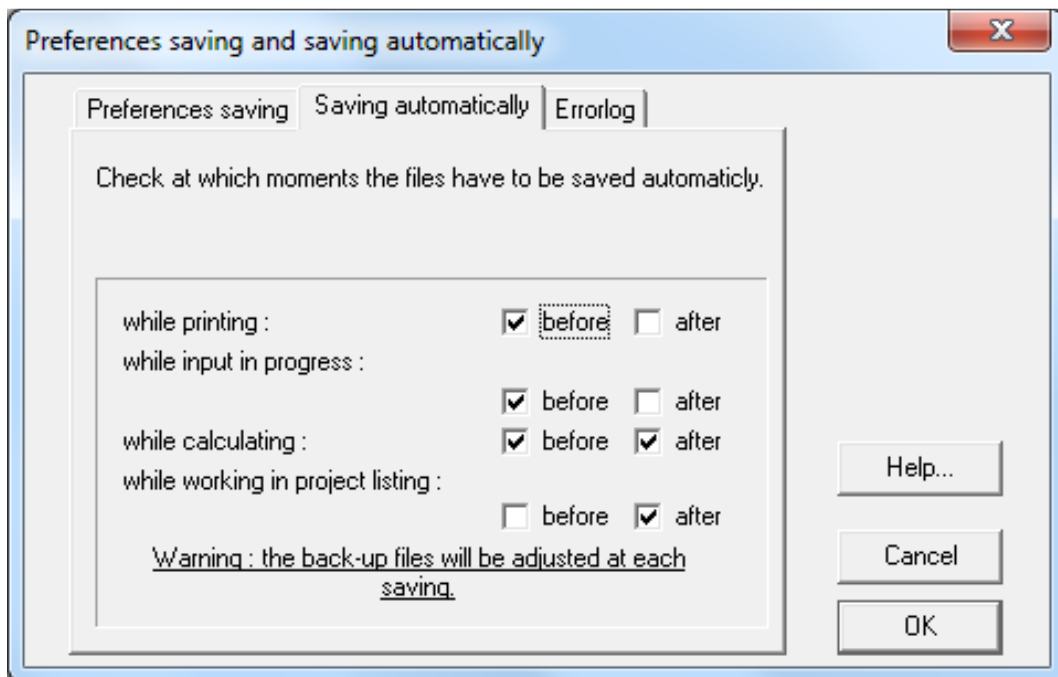


Figure 67 : settings for saving automatically

Besides the fact that you can decide when a saving procedure should be performed, you can also decide to keep a number of back-up files. Each time a save action is performed, the previous version of the file is kept. The number of back-up files that should be kept, can be indicated in the tab "**Preferences saving**" :

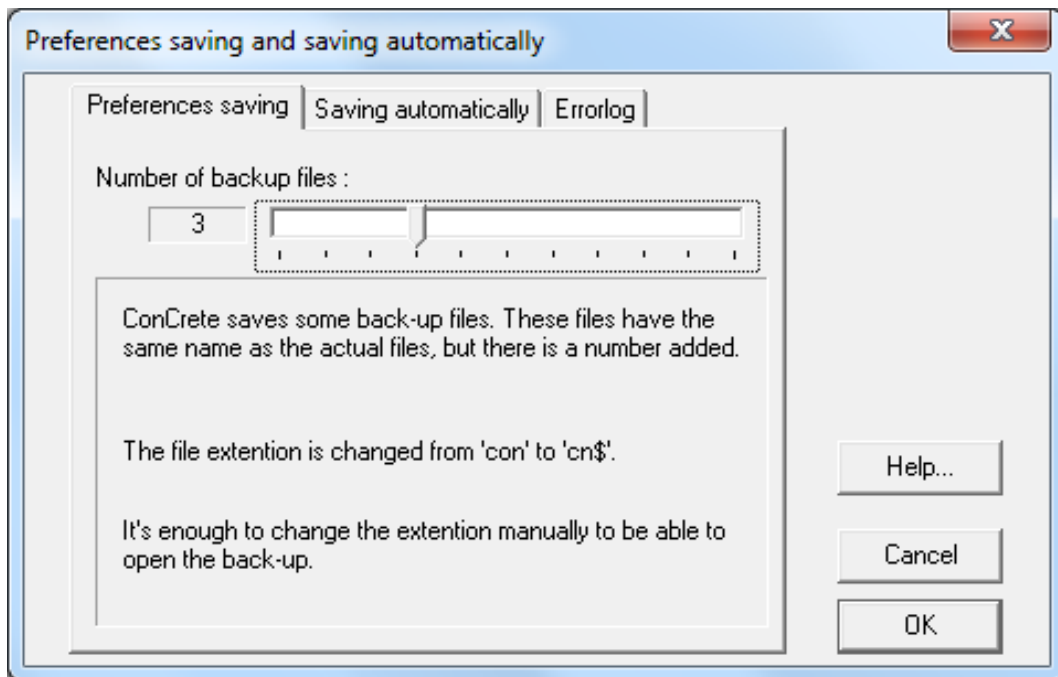


Figure 68 : Settings for back-up

Higher stated protection is relative. You can always make back-ups of the entered project manually.

8.5. Closing a project

To close a project we select the third command "**Close**" in the **File** menu. If modifications have been made and not yet saved, the program asks us automatically whether we want to save this project or not.

8.6. Modifying the project data

To modify project data we select the last but one command "**Modify...**" in the **File** menu. The dialog box of Figure 2 is again visualized on the screen. We can now freely manipulate the data.

8.7. Quitting the program

To quit the program we select the last command "**Quit**" in the **File** menu. If we are working in a project and the modifications have not yet been saved, the program asks us automatically whether we want to save our project or not.

Chapter 9 Printing

9.1. Page setup

To set the print options we select the command "**Page setup**" in the **File** menu.

The dialog box of Figure 69 is now visualized on the screen. In this dialog box we can select the printer and set its options as well as four print options specific for ConCrete (Figure 69).

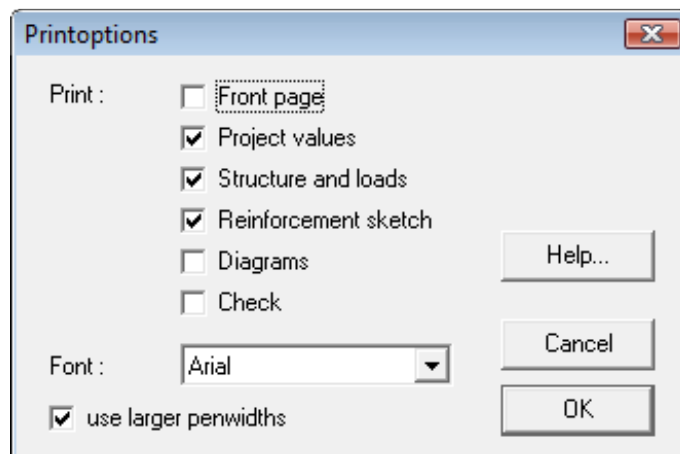


Figure 69 : Printoptions

When selecting the first option "**Front page**" a front page, with all project data of Figure 2 mentioned in it, is proceeding the whole printing. The following three options are used to print the content of the windows which carry these titles. With the second option "**Project values**", all calculation parameters are printed : one page with all the standard values and for each element the differences with those standard values. The next 3 options serve to print or not the content of the windows that have the same title.

9.2. Printing an entire project

To print an entire project we select the command "**Print project...**" in the **File** menu. If there is still an element present in the windows "**Structure and loads**", "**Reinforcement sketch**" and "**Diagrams**" and this element is already completely entered, the program puts this element in the first selected class of the **Project list**-window. The program then prints the entire project, class after class and element after element. Only one **front page** is printed per project (if the option is selected) and only one page with the **project values**. The data of the **Structure and loads** window are printed per beam, followed by the **deviations** of the present values of the calculation parameters versus the file values. The data of the **Reinforcement sketch** window and/or the **Diagrams** window are printed according to the case. The program starts a new page for each

beam. A new page is also started when printing the content of the **Diagrams** window. The program sees to it that the content of this last window is neatly flattened to the paper format. The reference code of the project and the date are mentioned on each page. The number of the element, preceded by the name of the class, is mentioned right on top of the pages, specific to the element.

9.3. Printing an entire class

If we wish to print all the elements of only one specific class, we select this class in the **Project list**-window. Afterwards we select the command "**Print class...**" in the **File** menu. This command works in a similar way to that described in 8.2.

9.4. Printing of only one element

To print only one element, we have to get it from the **Project list**-window and select the command "**Print element...**" in the **File** menu. This command works analogous to 8.2.

Chapter 10 Useful tips

10.1. Automatical calculation after input

So far the program has always calculated the element after complete input and after every modification. If we however duplicate a beam to modify it, it can be disturbing that the program interrupts us after every modification for a calculation which after all does not interest us. We can avoid this by disactivating the third option "**Calculate automatically**" in the **Preference** menu. Now the program only gives the notification "end of the beam" after the end of every input and every modification, without doing any calculation. If all the data have been entered and/or modified and we finally want the beam to be calculated, we can do so by selecting the last command "**Calculate**" in the **Element** menu.

10.2. Sounds

ConCrete enables you to accompany the entry of data by specific sound. Each quantity (like width, height, length of a span, value of a concentrated force, of a distributed force, and so on) has a specific sound which can serve for quick entry and modification of the data. If we activate the fifth option "**Sounds**" in the **Preferences menu** (= marked with a ""), a specific sound can be heard for each question that appears in the entry beam. Also when selecting the already entered data in the Structure and loads window the question is accompanied by a sound. In the other case, the entry and modification is done purely on a visual basis by a question in the entry beam.

10.3. Connections with ConCrete Plus

The connections between ConCrete and ConCrete Plus are situated on four levels.

First of all, the format of the files is identical for ConCrete and ConCrete Plus. This makes it possible to complete a project, made in ConCrete, with ConCrete Plus, and then afterwards extend it with ConCrete, to step over again to ConCrete Plus.
(see 7.2).

Secondly ConCrete offers the possibility to transmit a message to ConCrete Plus. This can be very important when the person who calculated the project with ConCrete is not the same as the one who will draw it with ConCrete Plus. We can enter such a message by clicking in the text balloon left at the bottom in the **Structure and loads** window. A dialog box, in which we can enter the desired message, is now visualised on the screen (Figure 70). When printing this beam, the message will be printed along with the entry data. When this beam is opened in ConCrete

Plus, a dialog box with this message automatically appears on the screen.

Thirdly, we can indicate in ConCrete the maximum tolerated variation in the dimensions between the calculation in ConCrete and the drawing in ConCrete Plus. If this variation is within the tolerance, there is no new calculation of the beam required and the calculated values are applied for drawing the beam. Setting these tolerances is done by selecting the last command "**Tolerances...**" in the **Preferences** menu. The dialog box of Figure 71 is now visualised on the screen.

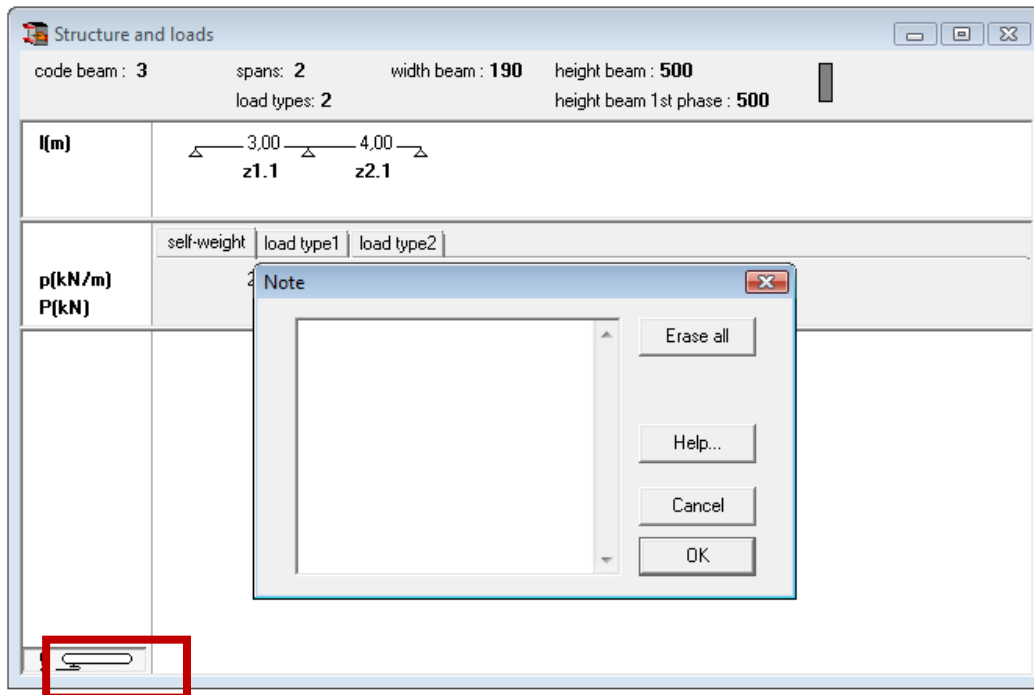


Figure 70 : Note

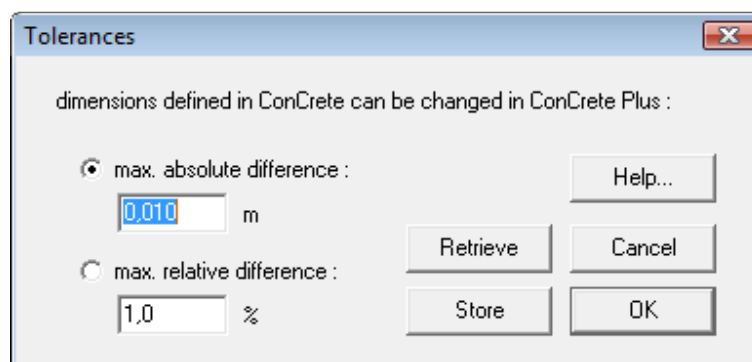



Figure 71 : Tolerances

This tolerance is not applied on the width of the supports. The width of the support for that matter is not indicated in ConCrete but in ConCrete Plus. The tolerance of the span lengths is valid for the centre-to-centre distance of the supports.


Finally, we can exchange elements between ConCrete and ConCrete Plkus by the commands "**Edit**" "**Copy**" and "**Edit**" "**Paste**" (see also 6.9).

ConCrete Plus exists as a separate program or can be integrated as a DLL in ConCrete. If you use the separate program, then you switch from ConCrete to ConCrete Plus by the command "File – Transfert ConCrete Plus". Of course, it is also possible to simply open a ConCrete file within ConCrete Plus".

If ConCrete Plus integrated in Concrete, then you open the module by the button  in the icon bar. We refer to the ConCrete Plus user's manual for more information over the functionalities of van ConCrete Plus.

10.4. Showing more diagrams or limit states

So far the **Diagrams** window only contained the diagrams of the bending moments and of the shear forces in U.L.S, the deflection under a Q.P. limit state and the reinforcement.

Select in the menu "**Preferences**" the first command "Show diagrams" or click on this  button to show more or less diagrams in this window. A dialog box (Figure 73) will appear on the screen. In this window you can check the diagrams that should appear in the window '**Diagrams**'. The height scale of the graphs is always determined such that all desired graphs nicely distributed evenly across the window.

The diagrams of the rotation angles and of the deflections are calculated according to the linear-elastic method. If we select the option "**with cracked sections**", the program does not only calculate the elastic deflection, but also a more sophisticated deflection which takes the cracked state into account. This last one is represented in red in the **Diagrams** window.

The deflection in the serviceability limit state under the quasi-permanent combinations should be limited to 1/250 of the span length, where the deflection can cause damage to other elements even 1/500. Is the option "**after creep**" designated, then the creep will be charges in the elastic analysis and in the calculation of the cracked state.

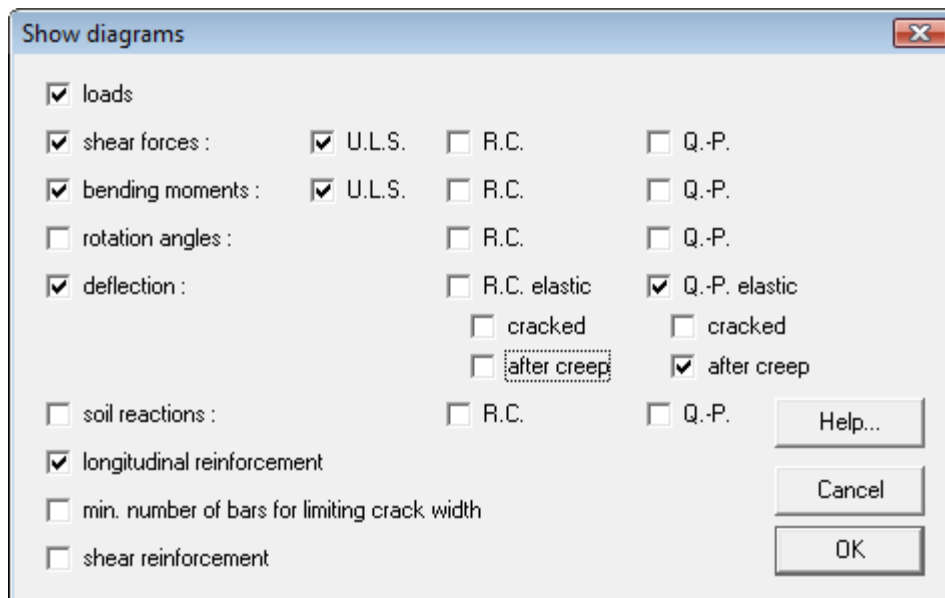



Figure 72 : Show diagrams

Since there are no rod diameters chosen yet, the calculation of crack width is only based on the theoretically required reinforcement sections. The crack width depends on the diameter and number of the rods, such that a calculation of crack width in itself can not be approached from theoretical reinforcement sections. The limitation of crack width in Concrete leads to a somewhat reverse calculation: ConCrete calculates how many bars at least should be placed so that the maximum allowable crack width is not exceeded, assuming all the very minimum reinforcement section is placed and that all bars will have the same diameter.

So far the "Structure and loads" window contained the vertical reactions and, should this be the case, moment reactions in the ultimate limit state (ULS). Should the reactions in the serviceability limit state rare combinations (ZC) or the serviceability limit state quasi-permanent combinations (Q.-P.) be visible, then select in the menu "**Preferences**" the second command "**Show results**" or click . The screen displays the following dialog:

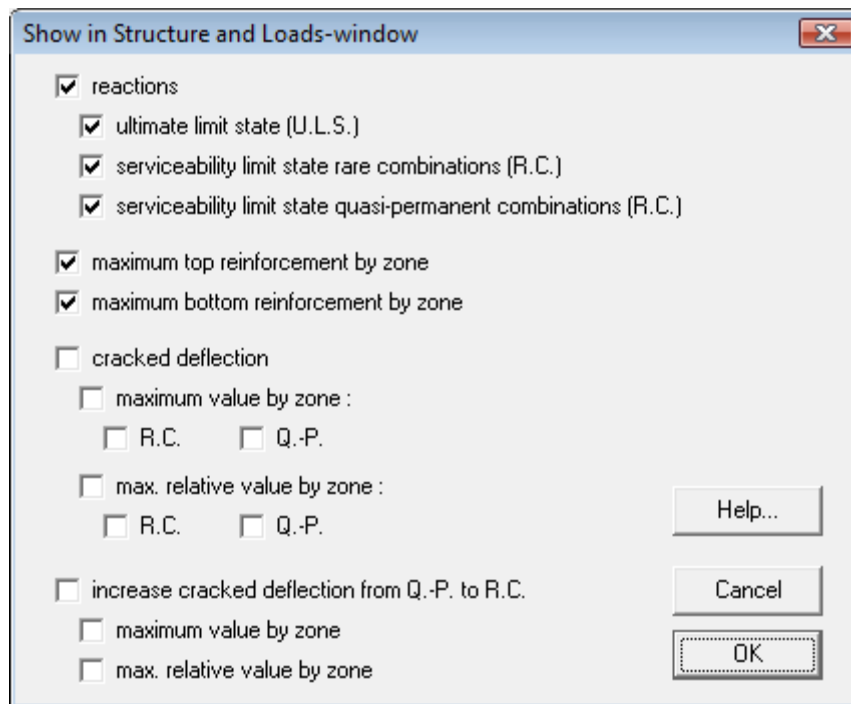


Figure 73 : Show results

In this window you can indicate of which limit state the reaction should be displayed in the 'Structure and loads'-Window. Furthermore, you can also

- indicate that the 'Structure and loads'-window should show the maximum required upper and lower reinforcement per zone
- show the maximum (in cracked condition computed) deflection among the rare-permanent and / or quasi-permanent combination
- show how the deflections relate to the span length
- show the maximum increase of the deflections under the rare combinations compared to the deflections under the quasi-permanent combination (possibly also in relation to the span length).

10.5. Sign conventions for moment and shear force

Not everybody uses the same sign convention for the bending moment M and the shear force V . We can modify or overview this sign convention by selecting the third command "**Sign**" in the **Preferences** menu. We can specify our usual sign convention in the thus created dialog box (Figure 74).

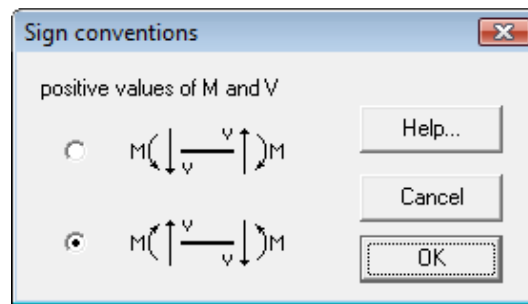


Figure 74 : Sign conventions

10.6. Close-up of a diagram in a zone

The diagrams in the **Diagrams** window are in some cases too little to determine the exact value of one or other statical quantity at a well determined place. ConCrete enables you to ask for a detail of a diagram per zone.

10.6.1. Making a close-up

If we want to get more details about a statical quantity in a certain zone, we double click with the mouse in the Diagrams window where that diagram in that determined zone is located. A dialog box, with a detail of the desired line, is now visualised on the screen (Figure 75).

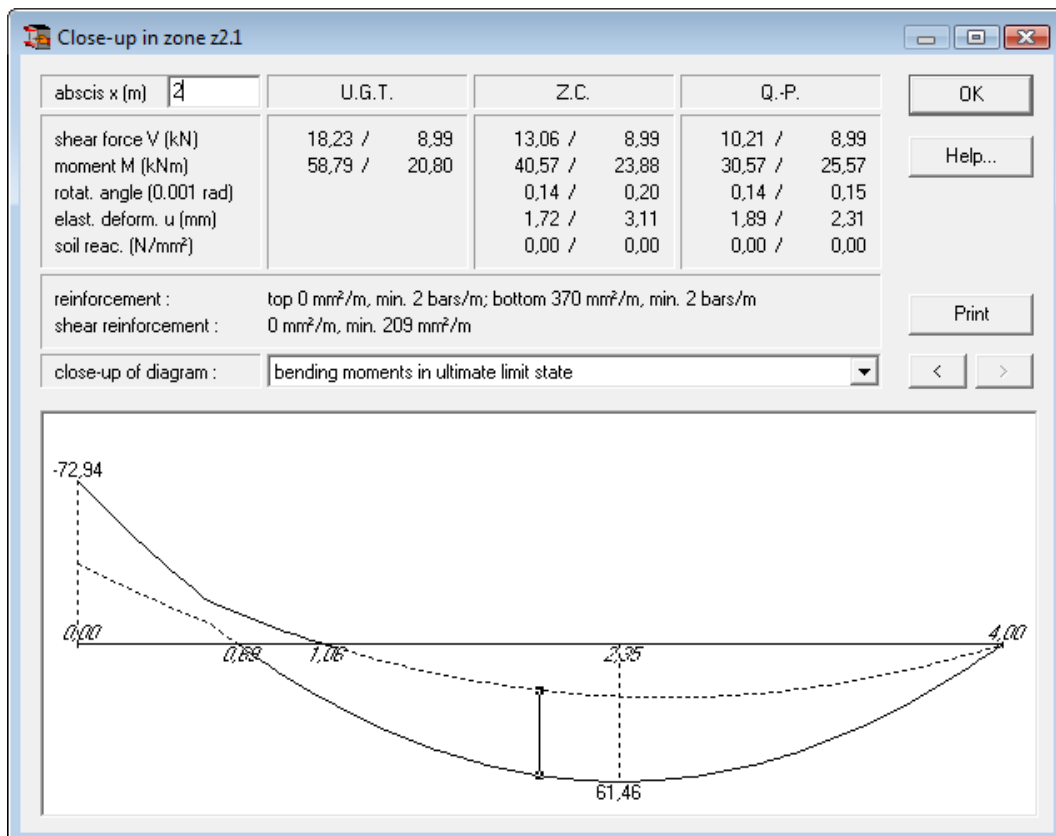


Figure 75 : Sign conventions

In the lowest half of this dialog box we see the requested detail of the diagram in the considered zone with indication of the characteristic values like the extremes and the O-points. In the upper half we see the minimum and maximum values of the shear force, the bending moment, diagram of rotation angles and deflection in one well determined point. The values for the deflection are the elastic calculated ones. Right on top there is a button with which we can print the content of this dialog box.

10.6.2. Changing to another diagram

Use the mouse in the menu next to the "**close-up of chart**": the desired diagram.

10.6.3. Withdrawing values in isolated points

The minimum and maximum values of all static quantities at a specific point can be requested by entering the x-value (abscissa) of that point in the edit box. Only abscissa values within the zone taken into close-up, give results. The abscissa value is determined from the left support of the span to which that zone (and also that point) belongs.

In Figure 75 the values are shown in the point that is situated at 2 m from the left support of the first span of 'beam 3'.

In the diagram, the abscissa value is indicated by two small squares, respectively the minimum and maximum values, connected by a vertical line.

Instead of typing the abscissa value with the keyboard, you can also use the mouse to indicate the desired location in the close-up chart. The abscissa value is automatically put in the edit box. Hold the mouse button down to see the position continuously changing.

10.7. Use of the "Edit"-menu

One of the most known functions of Windows and other GUI's is the mutual use of the commands "**Undo**", "**Cut**", "**Copy**", "**Paste**" and "**Clear**". While the bar is not fully implemented, these functions can not be used. We can e.g. use these functions to calculate determined loads with the help of a calculation program and copy these values to the edit box of the input bar.

Once the bar is completely entered, we can use the commands "**Copy**" and "**Paste**", to copy elements from a ConCrete-file into another ConCrete-file or to a file in Concrete Plus and vice versa (see also 6.8).

10.8. Calculation method

In ConCrete, the organic calculations are done by using the method of limit states. The calculation can be carried out according to different standards. The desired standard is set by selecting "**Preferences**" command "**Calculation Method**" option. In the dialog box (Figure 76) you can indicate your choice.

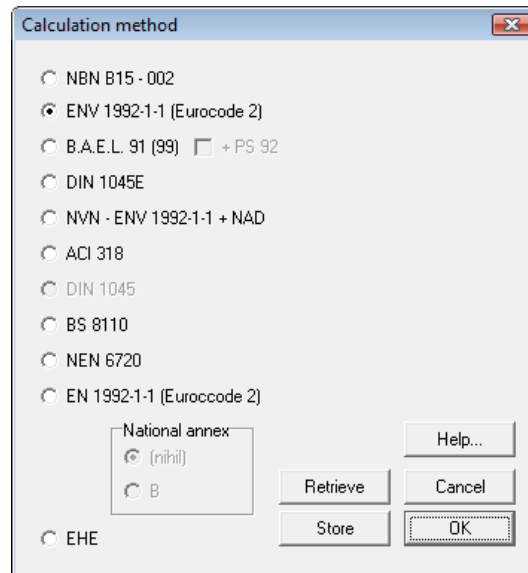



Figure 76 : Sign conventions

10.9. Verification calculation

In ConCrete the calculation of the cracked deformation and the crack width is done by using the theoretically required reinforcement sections. The results of this calculation gives a good indication of the expected results of a calculation that takes into account the actual planned reinforcement. In some cases the user is not satisfied with the "rough" calculation based on the theoretical reinforcement sections.

Once the beam is drawn in Concrete Plus, you can recalculate the structure in ConCrete, but taking into account the actual planned reinforcement. This verification calculation involves the calculation of the cracked bending under rare combinations and under the quasi-permanent combination, and the crack width in the quasi-permanent combination. If the effects of creep were taken into account in the initial "rough" calculation, then they will also be used in the verification calculation.

It goes without saying that any change in the calculation parameters, except whether or not taking creep into account, will influence the reinforcement choice and therefore the verification calculation will be destroyed.

To execute the verification calculation, select with the mouse in the menu "**Element**" the command "**Check**". The resulting diagrams for the cracked deflection and crack width are presented in the "**Control**"-window. You can access this window through "**Window – Control**" command or with the  button.

10.10. Saving and retrieving preferences in files

In chapter 4 we explained how to change, set as default and retrieve the various parameters used in the calculation.

Every time the program is restarted, these default values are loaded. Each new set of the default values, will erase the previous ones. It may however be useful to have different sets of default values in place: for example, to distinguish the calculation of beams from slabs, prefabricated from in-situ beams, projects with or without external control, foreign from domestic projects, etc.

Therefore ConCrete has the ability to store the default values in a separated file with the comment "**Preferences**" "**Save Defaults**". In the dialog box that appears on the screen, you can identify where and under what name the default values should be saved. To use previously saved default values, select the command "**Preferences**" "**Open default values**". The default values read, stay the default values when ConCrete is restarted.

10.11. Adapting ConCrete's environment

You can customize your work environment with the concept of "dockable toolbars". This means that you can drag the icons over the screen, and place them wherever you want. Moreover, you can include the palettes "**Input Data**" and "**Calculation Parameters**" into the toolbar. To do so drag the palettes on the screen or you double-click on the pallets. If you no longer want these icons on the screen, you can simply hide them by closing the palettes. You can recall them at any time through the command "**Window - Toolbars**".

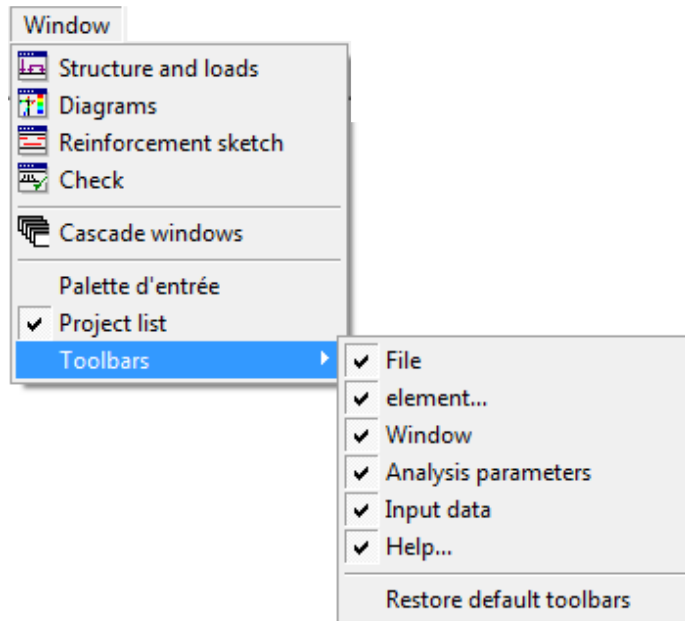


Figure 77 : Show or hide toolbars

Choosing the option “**Restore Toolbars**”, will activate the original configuration again.

In addition, the project list can also be given a permanent place in the Concrete user interface. By double-clicking the title bar of the “**Project List**”-window or by dragging the window, you can put it e.g. on the left.

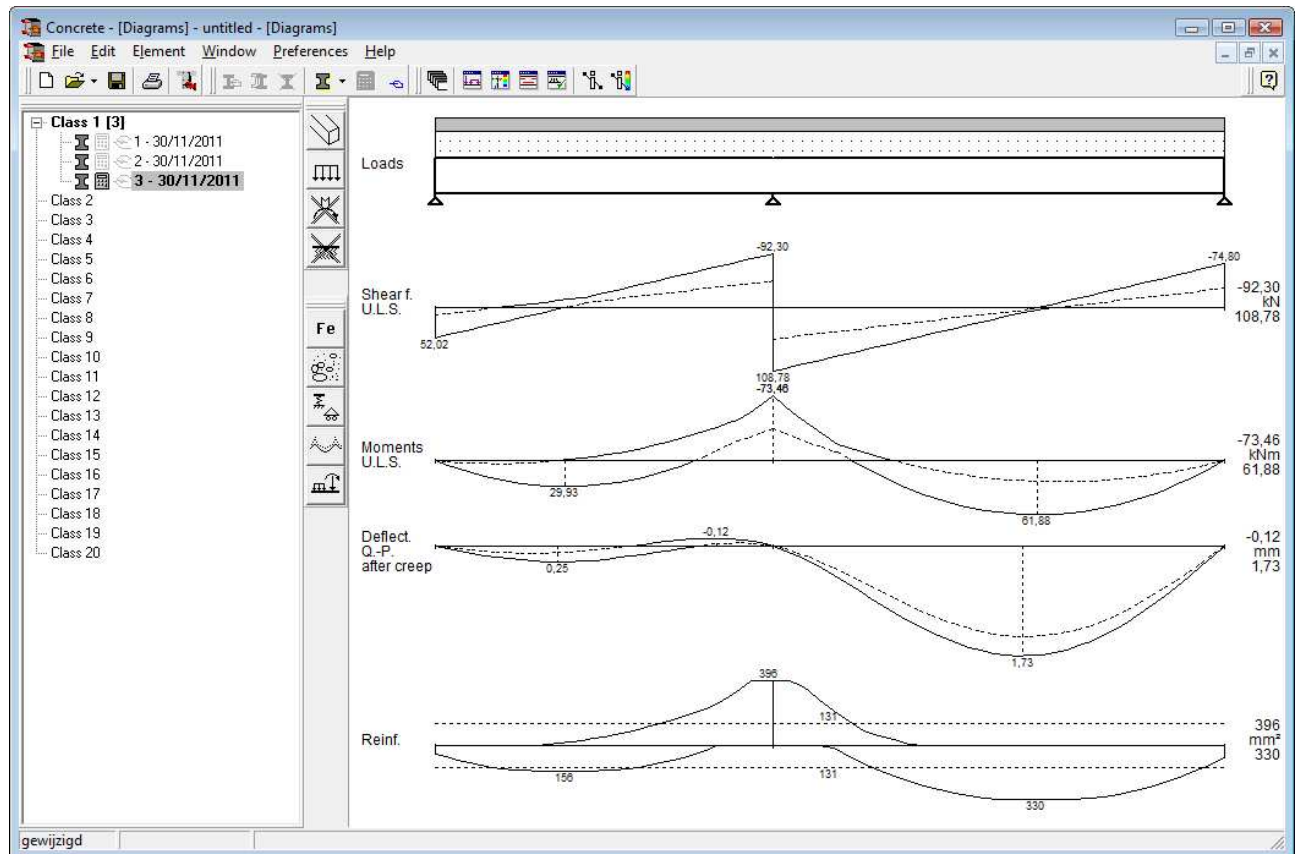


Figure 78 : ConCrete's environment

Note: Your personal settings are automatically saved and will be applied the next time you reboot ConCrete.

Chapter 11 Overview of the menu's

11.1. "File"-menu

| | |
|----------------------------|--|
| "New..." | create a new project (see 2.2) |
| "Open..." | open an existing file (see 7.2) |
| "Close" | close the actual project (see 7.5) |
| "Save" | resave the current project (see 7.1 and 7.3) |
| "Save as..." | save the current project (see 7.3) |
| "Preferences saving..." | backup and save automatically (zie ook 8.4) |
| "Page Setup..." | printer and page settings (see 8.1) |
| "Printoptions..." | set the print options (see 8.1) |
| "Print -> project..." | print the current project (see 8.2) |
| "Print -> class..." | print the selected class (see 8.3) |
| "Print -> element..." | print the selected element (see 8.3) |
| "Modify..." | change the file data (see 7.6) |
| "Transfer ConCrete Plus" : | make ConCrete Plus link (see 10.3) |
| "Quit" | quit the program (see 7.7) |

11.2. "Edit"-menu

| | |
|------------------------------|---|
| "Undo" | Windows functionality (see 9.7) |
| "Cut" | Windows functionality (see 9.7) |
| "Copy" | Windows functionality (see 9.7) |
| "Paste" | Windows functionality (see 9.7) |
| "Clear" | Windows functionality (see 9.7) |
| "List of classes..." | define the classes (see 6.2) |
| "Hollowcore slab library..." | Viewing the hollowcore slab library (see 7.5) |

11.3. "Element"-menu

| | |
|-----------------------|--|
| "New beam" | enter a new beam (see 2.3) |
| "New slab" | enter a new beam (identical) |
| "New hollowcore slab" | Enter a new hollowcore slab (identical) |
| "Modify" | modify the selected element (see 6.4) |
| "Duplicate" | duplicate the selected element (see 6.6) |
| "Remove" | remove the selected element (see 6.7) |
| "Calculate" | calculate the current element (see 9.1) |
| "Check" | check the current element (see 10.9) |
| "Redraw" | redraw projectlist |

11.4. "Window"-menu

| | |
|------------------------|--|
| "Structure and loads" | bring this window to front (see 2.3 and 2.4.1) |
| "Diagrams" | identical (see 2.3 and 2.4.2) |
| "Reinforcement sketch" | identical (see 2.3 and 2.4.2) |
| "Check" | identical (see 10.9) |
| "Cascade windows" | cascade windows |
| "Input panel" | pop up input panel |
| "Project list" | bring the project list window to front (see 2.2 and 2.3) |
| "Toolbars" | customize ConCrete workspace (see 10.11) |

11.5. "Preferences"-menu

| | |
|---------------------------|---|
| "Show diagrams..." | show different diagrams (see 9.4) |
| "Show results..." | show different results in several limite states, reinforcement, cracked deformation (see 9.4) |
| "Signs..." | set sign convention for M and V (see 9.5) |
| "Calculate automatically" | immediate calculation after entry (see 9.1) |
| "Save automatically" | save after every five new elements (see 7.4) |
| "Sounds" | specific sounds when entering and changing see (9.2) |
| "Tolerances..." | measurement deviations of ConCrete vs. ConCrete Plus (see 9.3) |
| "Calculation method..." | set calculation code (see 10.8) |
| "Language" | choose language |
| "Load settings..." | Loads a set of saved parameters for calculation and sets them as default (see 10.10) |
| "Save settings..." | Saves the parameters for calculation (see 10.10) |
| "General settings..." | restore standard settings |

11.6. "Help"-menu

| | |
|---------------------|---|
| "Manual" | allows a dynamic use of the manual to the screen |
| "Mail to BuildSoft" | automatically opens the default mail program and enters the email address |
| "Network... " | look for network licence dongle |
| "About ConCrete..." | information about ConCrete |