ПРОЕКТСОФТ

Software Package

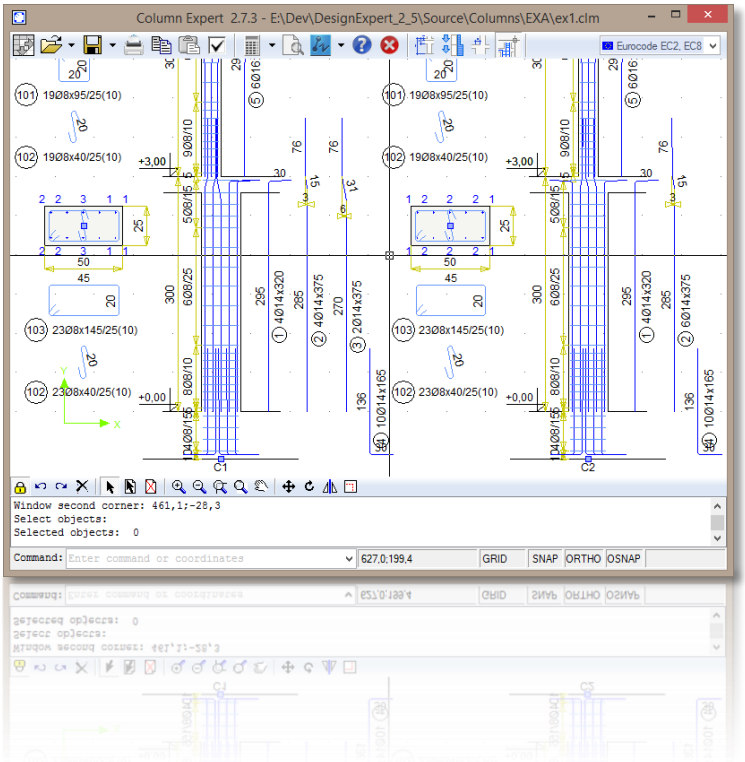
Design Expert version 2.7

Structural Design and Detailing to Eurocode

Column Expert

Design and detailing of reinforced concrete columns

User Manual



 2014

TABLE OF CONTENTS

About the program

Column Expert is a software product for design, detailing and drafting of reinforced concrete columns according to Eurocode (EC2, EC8 etc.). It is part of Design Expert software package.

Column dimensions and loads for each storey are easily defined in tables. You can draw cross sections using powerful graphical editor. All sections are stored in a library and can be reused multiple times. You can easily add or remove sections from the library. Main features of the program are:

Design

Columns are designed for axial load with nominal eccentricity, second order effects and initial imperfections. You can assign cross sections preliminary and verify column capacities. Alternatively, the program can automatically select the required cross sections and reinforcement bars based on the defined loads.

Detailing

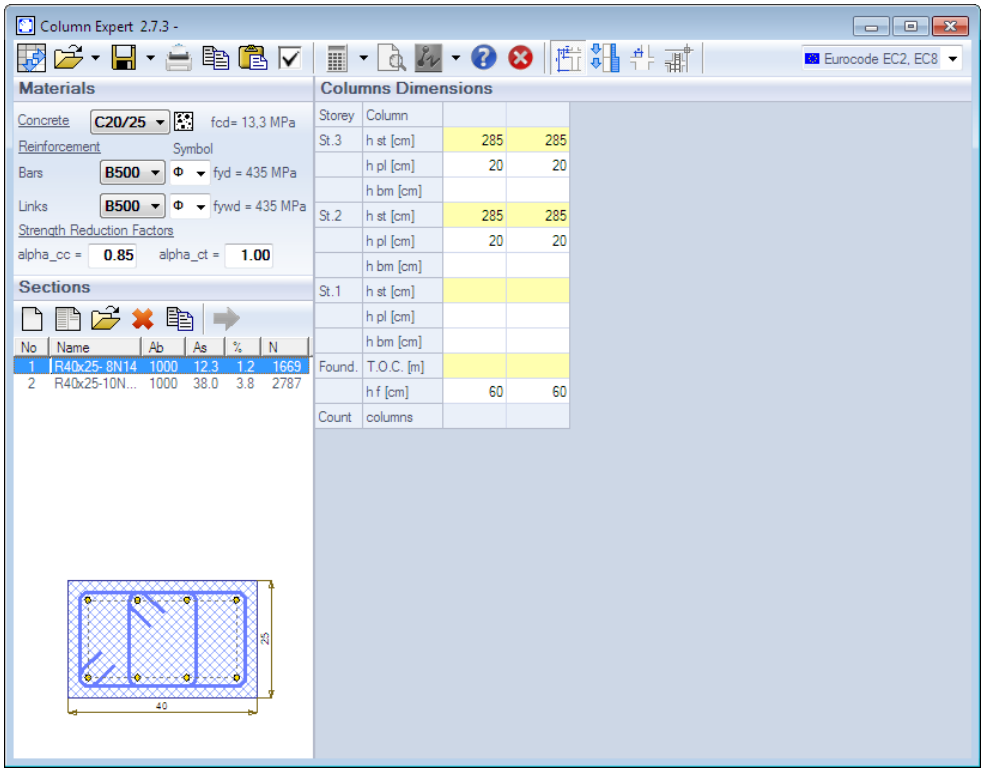
All reinforcement bars and shear links are generated automatically for the defined column dimensions, bar diameters and counts. Most of Eurocode detailing provisions like reinforcement ratios, anchoring, splice lengths, bar and link spacing, critical zone lengths etc. are included in this software. They are applied automatically during the detailing process.

Drafting

The drawing is created firstly in Design Expert internal graphical editor where you can review and modify. Then you can export it directly to ZWCAD+ or AutoCAD or save a script file for AutoCAD LT. The software generates bill of materials for both steel and concrete and reinforcement bending schedule. The reinforcement output is compatible to Design Expert Plug-in module for reinforcement detailing and scheduling with AutoCAD or ZWCAD+.

How it works?

The software includes standard graphical user interface for Windows. You can enter commands either by clicking buttons on the main toolbar or by typing commands in the command line located at bottom. Detailed descriptions of all commands are provided further in this manual. If you hold the mouse over a button, a tooltip appears with short description for the respective command.



Working with files

Column Expert has its own file format which is used to save program data permanently on a disk. Input file extension is ***.col**. Results are stored into ***.col.html** files.

Open a file

Click the button to open a file from the disk. A standard file selection dialog appears on screen. Browse for the file using the mouse or type file path and name and click **"Open".**

Save a file

Click the button to save a file to the disk. A standard file selection dialog appears on screen. Select the destination folder and file name. If file already exists you will be prompted to overwrite or change the name. You can save with a different name directly by clicking the small arrow next to the button. Then select **"Save as..."** from the drop-down menu.

Input data

Input data is divided into several pages for convenience. You can switch between pages by clicking the respective buttons on the main toolbar  . Use the text fields and tables inside each page to enter data.

Working with tables

Most of the input data is entered in tables. You can use the following commands to work with all tables inside the program:

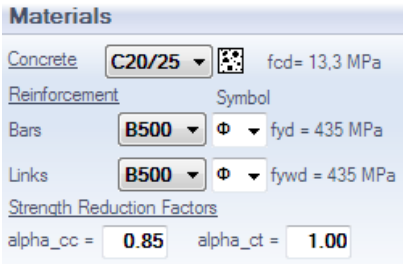
- Insert new row – press **Ins** key or "**+**" button. When you go to the end of the last row and press **Enter**, a new row opens automatically;
- Delete last row – press **Backspace** or "**–**" button. Some tables are with fixed dimensions and you cannot add or remove rows;
- Move the current focus with one cell – press arrow keys ←, ↑, ↓, →;
- Move the focus to the first or the last row – press **Page Up**, **Page Down**, **Home**, **End**;
- Edit current cell contents – press **F2** or just start typing – an input box is opened automatically;
- Finish cell edit – press **Enter** or arrow key – the input box is closed and changes are stored into the cell;
- Cancel cell edit – press **Esc** – the input box is closed and changes are discarded. The original contents remains in the current cell;
- Delete cell contents – press **Del** – the contents of all selected cells is cleared;
- Select a range of cells – the first method is to use the keyboard – select the first cell, hold **Shift** and press arrow keys or **Page Up**, **Page Down**, **Home**, **End** to move to the cell at the other corner of the area. Alternatively, you can click with the mouse at the first corner, hold **Shift** and click at the opposite corner;
- Copy the contents of the selected cells – press Ctrl+C;
- Paste into the selected cells – press Ctrl+V;

You can copy from and paste to the same or other tables as well as external programs like Word, Excel, etc. If you try to paste a range of cells which area is greater than the area of the destination cells, you will receive a warning. This is necessary to avoid unwanted data overwriting.

Design code

Design Expert is compatible to Eurocodes, mainly EN 1992-1-1 and EN 1998-1-1. It is applicable to most countries as far as you can define your own material properties, partial safety factors, loads and some other important parameters. Detailed description of all design methods and formulas used in this program is provided further in this manual.

Materials



Material and section properties are entered in the first page of the program. It is active at startup by default. If you have moved to another page, you can always go back with the button.

You have to enter concrete grade C20/25 and steel grades for main B500 and shear B500 reinforcement. Also, you can specify a symbol (N, Ø, ect.) to be used for bar labelling in the drawings. Characteristic and design values for material properties are predefined in tables. Concrete compressive and tensile strengths are additionally multiplied by the sustained loading factors α_{cc} and α_{ct}. They should be defined separately in the respective fields since they are not included in the table values.

Material tables

You can open the material tables by clicking the button. A dialog containing both concrete and reinforcement tables appears on screen. You can modify values, add and remove rows by clicking the „**+**" and „**-**" buttons, respectively. Finally you should press "**Save**" to save changes and close the dialog. If you want to discard changes, press "**Exit**" and you will return to the main window.

Material tables are common for the whole computer. Any changes you make will reflect all Design Expert modules and input files.

Material data to Eurocode

Concrete

Design Expert includes the following concrete grades according to EN 1992-1-1, Table 3.1:

Name	E_{cm} GPa	$f_{ck,cube}$ MPa	f_{cd} MPa	f_{ctd} MPa	f_{ck} MPa	$f_{ctk,0.05}$ MPa	ε_{c2}	ε_{cu2}
C12/15	27.0	15.00	8.00	0.73	12.00	1.10	0.002	0.0035
C16/20	29.0	20.00	10.67	0.87	16.00	1.30	0.002	0.0035
C20/25	30.0	25.00	13.33	1.00	20.00	1.50	0.002	0.0035
C25/30	31.5	30.00	16.67	1.20	25.00	1.80	0.002	0.0035
C30/37	33.0	37.00	20.33	1.33	30.50	2.00	0.002	0.0035
C35/45	34.0	45.00	23.33	1.47	35.00	2.20	0.002	0.0035
C40/50	35.0	50.00	26.67	1.67	40.00	2.50	0.002	0.0035
C45/55	36.0	55.00	30.00	1.80	45.00	2.70	0.002	0.0035
C50/60	37.0	60.00	33.67	1.93	50.50	2.90	0.002	0.0035

The following symbols are used in the above table:

E_{cm} – concrete secant modulus of elasticity;

$f_{ck,cube}$ – characteristic cube strength;

f_{ck} – characteristic cylinder strength;

$f_{ctk,0.05}$ – characteristic tensile strength with 5% probability of failure;

$f_{cd} = \alpha_{cc} f_{ck}/\gamma_c$ – design compressive strength;

$f_{ctd} = \alpha_{ct} f_{ctk,0.05}/\gamma_c$ – design tensile strength;

ε_{c2} – compressive strain at maximum stress for parabolic-linear stress-strain;

ε_{cu2} – ultimate compressive strain at concrete edge.

Design values for compressive and tensile strengths in the table are determined for partial safety factor γ_c = 1.5. They still do not include α_{cc} and α_{ct} factors which should be defined additionally. Some countries use α_{cc} = 0.85 and α_{ct} is usually equal to 1.0. You should look for these values in your national annex document.

Reinforcement

Design Expert includes the following reinforcement steel grades:

Name	E_s GPa	f_{yd} MPa	f_{yk} MPa	ε_{yd}
B220	200	191	220	0.01
B250	200	217	250	0.01
B420	200	365	420	0.01
B460	200	400	460	0.01
B500	200	435	500	0.01

The following symbols are used in the table:

E_s – design modulus of elasticity;

f_{yd} – design yield strength;

f_{yk} – characteristic yield strength;

ε_{yd} – design ultimate strain.

Number of columns and storeys

With Column Expert you can draw multiple columns at multiple storeys at a time. All input data is defined in tables. Before you start, you should define number of columns and number of storeys in order to size the tables. Press the  button in the main toolbar. A dialog appears where you can enter both numbers in the respective fields. Then press "**Save**" to finish and return to the main window. The necessary columns and rows are added to or removed from the input tables. Existing data is reordered if necessary to fit the new table sizes.

Geometrty data

Geometry data table is activated by pressing the  button in the main toolbar. Column labels should be defined in the first row, e.g. (C1, C2, etc.). Then, three rows have to be be entered for each storey:

- $\boldsymbol{h_{st}}$ – storey height measured between T.O.C. of the lower slab and T.O.C. of upper slab [cm];

- $\boldsymbol{h_{pl}}$ – upper slab thickness [cm];

- $\boldsymbol{h_{bm}}$ – beam depth [cm]. If you have several beams with different depths, enter the larger one. If there no beams (e.g. flat slab) leave the cell empty or enter "0".

Foundation data is entered at the bottom of the table: **T.O.C.** – top of concrete [m] and $\boldsymbol{h_f}$ – foundation height [cm]. If height is greater than zero, the program will automatically add starter bars for each column. Sometimes you can have equal columns in the drawing. In this case, you can draw only one of them and

specify the total count of the identical columns in the last row. You can use copy – paste to quickly distribute the information to other columns in the table and speed up the input. For more information, see "**Working with tables**".

Loads and sections

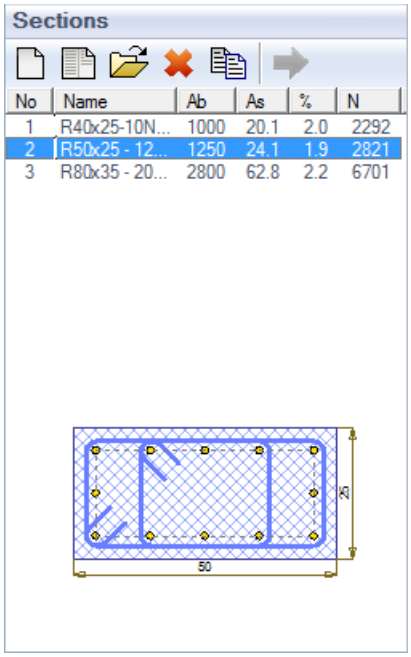
Loads and sections table is activated with the button. Column labels are defined in the first row, e.g. (C1, C2, etc.). Then you should fill two values for each column and each storey:

- **Load** – design axial force in the column in the respective storey [kN];
- **Section** – the number of the selected section from the section list - see "**Cross Sections**".

Column capacity is calculated and filled automatically by pressing the button. You can find detailed information in the "**Reinforcement Design**" chapter further in this manual.

Cross sections

Loading cross sections into the current project



Before drawing columns, you should create or load all cross sections that you are going to use. Then you have to assign section numbers to each column and each storey.

Sections are loaded into the "**Sections**" list located in the left panel of the main window. You can insert existing sections from the "**RC Sections Library**" by clicking the button. If you cannot find the necessary section in the library, you can create a new one. Click the button to open the section editor and draw the new section. Click the button to open the selected section for editing. Use the button to remove the selected sections from the list. It does not actually delete the sections from the disk. They remain in the library and you can find and insert them again later. The numbers of the removed sections are also removed from the column table. All other sections are automatically renumbered. You can quickly create different versions of existing sections using the button. The current section is opened for editing first. Make the necessary changes and save the section with a different name. The new section is added to the list unlike the button which replaces the current section. That is how you can quickly make several sections with equal shape and dimensions but different reinforcement.

You can select single or multiple sections in the list by clicking with the left mouse button or pressing the arrow keys while holding **Shift** and **Ctrl**. When you click a section, you can see a small drawing bellow the list. Section numbers are automatically assigned by the program. Section names are entered by user. You can see additional data for each section in the list as follows:

- A_b – area of concrete [cm^2];
- A_s – area of main reinforcement [cm^2];
- $\%$ – reinforcement ratio [%];
- N – section capacity for axial force [kN]: $N = A_c \cdot f_{cd} + A_s \cdot f_{yd}$.

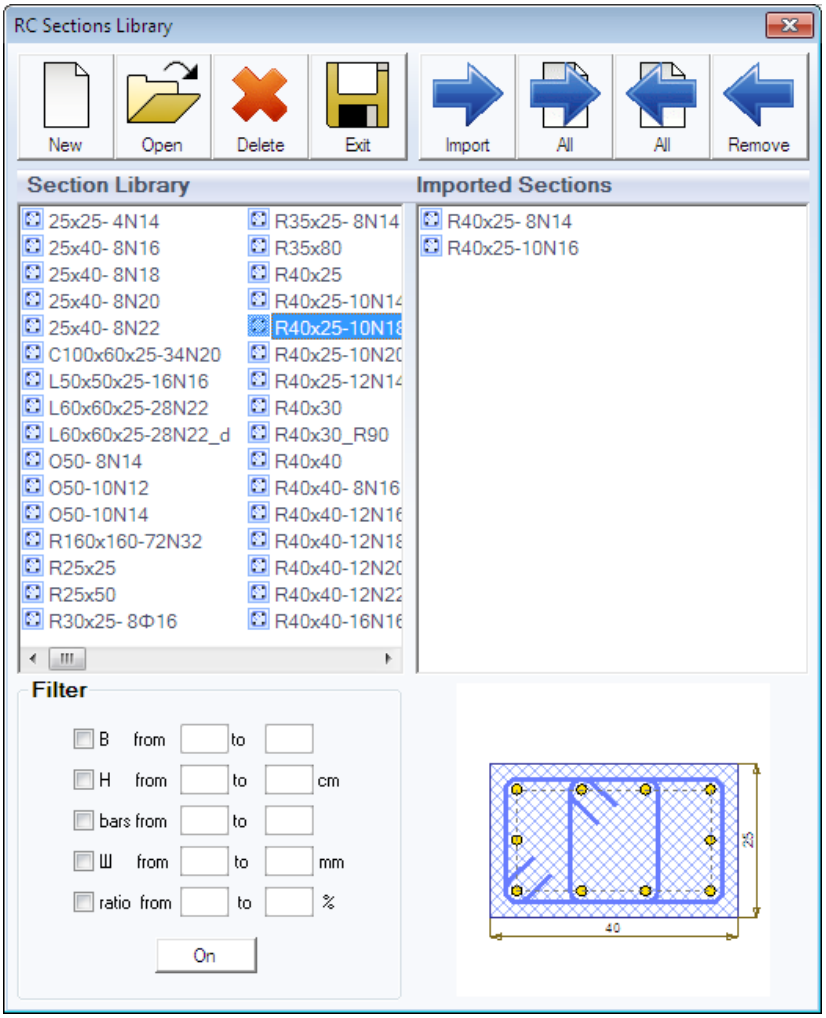
Section capacity is provided for information only. It does not include initial imperfections, nominal eccentricities and second order effects. It is greater than the respective column capacity and cannot be used for direct verification. It can only help you to find the appropriate sections more easily. Column capacities are calculated when you start the design.

Assign sections to columns

Sections are assigned in the "**Column Loads and Sections**" table. You can open the table with the button from the main toolbar. You should fill the section number in the row "Section" for each column and each storey. The number should correspond to one of the sections in the list on the left. You can quickly assign a section to multiple columns and storeys at a time as follows: Select a range of cells in the table. Double click the required section in the list or select the section and click the button.

RC Sections Library

Design Expert allows you to build a library of sections with different dimensions and reinforcement and use them multiple times. Each time you create a new section, it is automatically stored in the library. You can open the library with the button above the section list.



All available sections in the library are loaded in the left panel. The sections to be used for the current project are listed in the right panel. If you click a section with the mouse, you will see a picture at the right bottom side. You can use the following commands to manage the library:

Load and unload sections

You can have hundreds of sections in the library. The program allows you to preliminary select only those sections that you are going to use in the current project. That makes your further work easier. In order to load sections into the current project, select them in the left panel named "**Section Library**" and move them to the right panel "**Imported Sections**". You can do this in several ways: Drag and drop the sections from left to right using the mouse. Double click a section on the left. Select the sections and press the „**Import**" button. Press „**All**" to import all available sections into the current project.

You can unload sections by transferring them from right to left in a similar way. Use the „**Remove**" и „**All**" buttons, respectively.

Add new sections

Click the „**New**" button. The section editor dialog appears. You can use it to enter or draw section dimensions and reinforcement bars. Finally, you should save the section to a file in the library.

Edit

Select the section you are going to edit and press „**Open**". It is loaded into the section editor. You can modify dimensions and reinforcement and save the changes.

Delete

Select the sections to be removed and click „**Delete**". You will be prompted for confirmation and after that, the selected sections will be permanently erased from the disk.

Filter

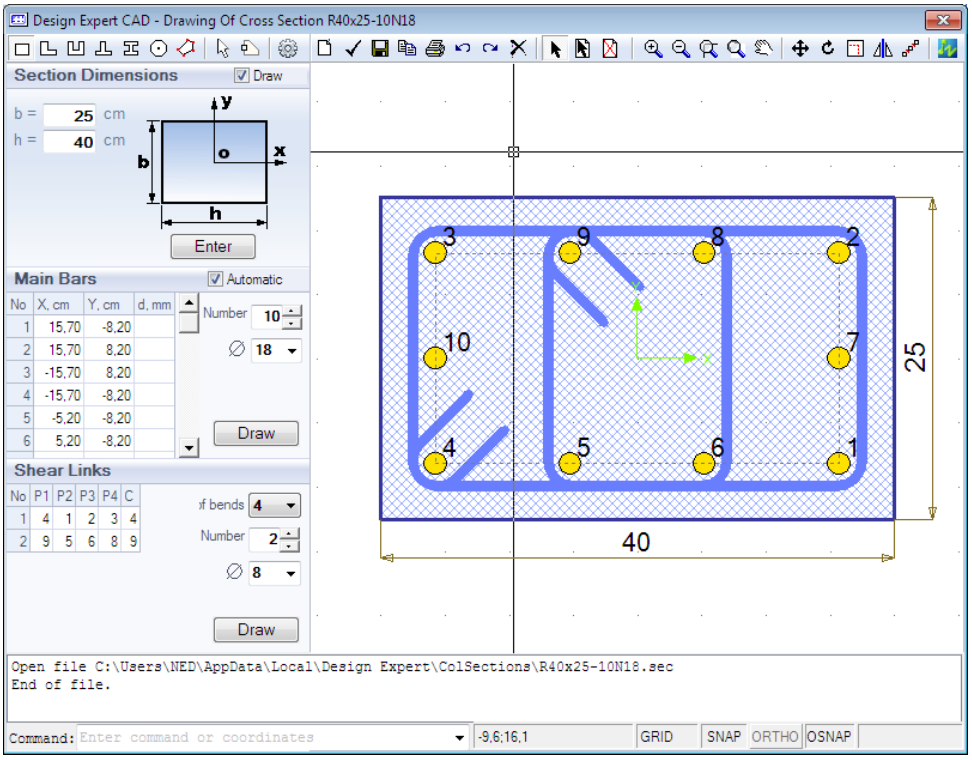
If you have too many sections in the library, it is difficult to find the necessary ones. Then you can use filtering by one or several of the following criteria:

- **B** – section width [cm];
- **H** – section height [cm];
- **bars** – total count of bars;
- **Ø** – bar diameter [mm];
- **ratio** – reinforcement ratio [%].

Select the required criteria, specify the "**from-to**" margins and press "**On**" to activate the filter. To deactivate press "**Off**". If you change some of the criteria you should press "On" and "Off" again to reactivate the filter.

Drawing new sections

You can create new sections and modify existing ones using the internal section editor as described below.

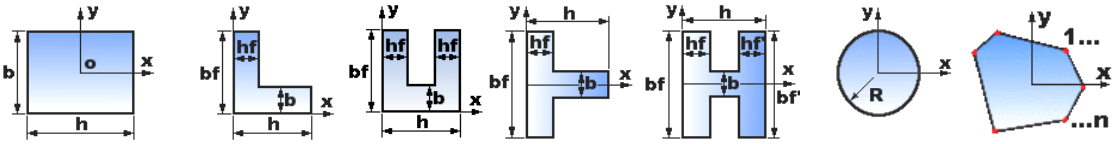


Settings

Before you start, you can check and modify the reinforcement detailing settings by clicking the button. See the "**Settings**" chapter further in this manual.

Section shape and dimensions

Select the shape from the toolbar . Enter dimensions as shown on the respective pictures and click the "**Enter**" button. The following dimensions are required for each shape:



If you have a general section, you should enter the coordinates of the outline points. You have two possible input methods – tabular and graphical:

- Tabular input: Enter the coordinates X and Y for each point in the table and click "**Enter**". If the "Automatic" option is selected, main reinforcement and shear links are also created;
- Graphical input: Click the button. Draw the section by clicking with the left mouse button on the drawing (see "Working with Design Expert CAD Environment")

You can also import sections directly from a ZWCAD+/AutoCAD drawing by using the button. Sections should be closed polylines. After clicking the button, you will be prompted to pick a polyline in the current ZWCAD+/AutoCAD drawing.

Main bars

You have to enter coordinates **X**, **Y** and diameter **d** (optional) for each bar. There are two ways to enter bar data:

- Tabular input: Enter the coordinates in the table. Before that, you can enter total bar count in the text field on the right. If the “Automatic” option is selected, bars are distributed evenly along the section perimeter each time you change the number of bars.
- Graphical input: Press the “**Draw**” button. Then click bar positions in the drawing with the left mouse button. Click the right button or press “**Enter**” to finish. The program automatically maintains the concrete cover and moves the bars inside if necessary. That allows you to pick points at concrete edge and get the bars inside at the right positions.

All bars are created with the default diameter Ø. It is defined in the respective field on the right. If there are bars with different diameters, you can enter the values in the ”**d, mm**” column in the table.

Shear links

Each shear link is defined by the corresponding bar numbers at the bends (P1 – P4). By default, you can specify 2 to 4 bars. If you need more complex links, you can increase the maximum number of bends up to 6 or 8 in the respective field on the right.

You can fill the numbers of bars in the “**Shear Links**” table. During the input, the program shows the current link shape on the drawing. Links can be closed or opened. For closed links, you should fill the number of the first bar **P1** once again in the “**C**” column.

Alternatively, you can draw shear links graphically. Press the “**Draw**” button and click consequently near each bar located at a bend. Click the right mouse button or press Enter to finish. You will receive a question “Close link?”. If you answer “Yes”, the link will be closed, otherwise it will remain open. The link is drawn automatically around the specified bars with the required bends and hooks.

Check section

You can verify whether the section complies with code requirements for the specified ductility class. There are three possible ductility classes in Eurocode 8 – Low (DCL), Medium (DCM) and High (DCH). The required ductility class can be selected in the “**Settings**” dialog. If your element is not intended to take seismic loads, you can select DCL.

The program checks section dimensions, reinforcement ratio (min and max) bar spacing (min and max), link spacing, distance between link bends, minimal diameters for bars and links, concrete cover etc. If some of these criteria is not satisfied, you will see a warning message on the screen. Detailed list of Eurocode requirements included into the program is provided bellow.

Detailing requirements to Eurocode 2 and Eurocode 8

All parameters, used for automated detailing of columns are listed in the table below. References to the corresponding sections in Eurocode are provided in brackets.

Section dimensions		**DCL Non-seismic element EC2**	**DCM Seismic element EC8**	**DCH Seismic element EC8**
Minimum section dimensions	min b_C	200 mm	200 mm	250 mm (**5.5.1.2.2 (1)**)
Maximum dimensions ratio	max h_C/b_C	4 (**EN 1992-1-1**, **9.5.1 (1)**)		

Longitudinal reinforcement		**DCL**	**DCM**	**DCH**
Minimum diameter	$d_{bL,min}$	12 mm (**9.5.2 (1)**, **NA.2.84**)		
Minimum reinforcement ratio	ρ_{min}	0.002 (**9.5.2(2)**, **NA.2.85**)	0.01 (**5.4.3.2.2 (1)**)	
Maximum reinforcement ratio	ρ_{max}	0.04		
		(**9.5.2(3)**, **NA.2.86**)	(**5.4.3.2.2 (1)**)	(**5.5.3.2.2 (1)**)
Minimum clear spacing between bars	a_{min}	50 mm		
Maximum spacing between bars centers	$a_{L,max}$	300 mm	200 mm	150 mm
Maximum spacing between bars at corners of links	$a_{h,max}$	300 mm[1)] (**9.5.3 (6)**)	200 mm (**5.4.3.2.2 (11)b**)	150 mm (**5.5.3.2.2 (12) c**)
Mandrel diameter for bending	d_m	For Ø ≤16 mm - d_m = 4Ø For Ø >16 mm - d_m = 7Ø		
Anchorage length	l_{bd}	$f_{bd} = 2.25\eta_1\eta_2 f_{ctd}$, $l_{b,rqd} = d_L/4\cdot\sigma_{sd}/f_{bd}$ $l_{bd} = \alpha_1\alpha_2\alpha_3\alpha_4\alpha_5 l_{b,rqd} > l_{b.min}$ $l_{b.min}$ = max{0.3$l_{b,rqd}$,10d_L, 10 cm}		
Lap length	l_0	$l_0 = \alpha_1\alpha_2\alpha_3\alpha_6 l_{b,rqd}$, α_6 = 1.5		

Shear reinforcement		**DCL**	**DCM**	**DCH**
Minimum diameter	$d_{bw,min}$	6 mm, 0.25·d_{bL} (**9.5.3 (1)**)	6 mm, (**5.4.3.2.2 (10)**) 0.25·d_{bL}	6 mm, 0.4·d_{bL}·(f_{ydL}/f_{ydw})$^{1/2}$ (**5.5.3.2.2 (12) a**)
Maximum spacing between shear links centers along the column	s_{max}	b_C 20·d_{bL} 400 mm (**9.5.3 (3)** and **NA.2.87**)		

Maximum spacing between shear links centers in critical zones	$s_{cr,max}$	0.6·s_{max} **(9.5.3 (4))**	$b_o/2$ $8d_{bL}$ 175 mm **(5.4.3.2.2 (11) a)**	$b_o/3$ $6d_{bL}$ 125 mm **(5.5.3.2.2 (12) b)**
Maximum spacing between shear links centers along bar laps	$s_{l,max}$	$b_c/4$, 100 mm **(5.6.3 (3))**		
Area of one shear reinforcement leg in the zone of bar lapping	A_{st}	$s \cdot d_{bL}/50 \cdot f_{ydL}/f_{ydw}$ **(5.6.3 (4))**		
Anchorage length inside concrete	l_{bw}	$10d_{bw}$ **(5.6.1 (2))**		
Critical zone length	l_{cr}	h_C **(9.5.3 (4))** $(l_o + 4d_{bL})$ [2)]	h_C $l_{cl}/6$ 450 mm **(5.4.3.2.2(4))** $(l_o + 4d_{bL})$ [2)]	$1.5h_C$ $l_{cl}/6$ 600 mm **(5.5.3.2.2 (4))** $(l_o + 4d_{bL})$ [2]

[1)] According to 9.5.3 (6), the maximum distance from free bar to bar with shear link is 150 mm. This will be always provided if spacing between legs of shear links is 300 mm. This is more conservative than the Eurocode requirement.

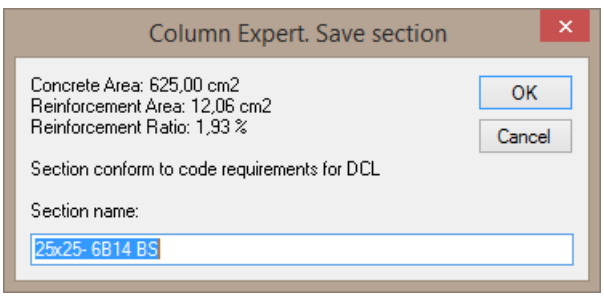
[2)] In order to simplify the detailing, critical zone and lap zone are merged at the bottom of the column and relevant length and link spacing is used. The additional $4d_{bL}$ is according to EN 1992-1-1, 8.7.4.2 (1) for shear reinforcement at distance of $4d_{bL}$ after the end of compression bars lapping.

List of symbols:

- d_{bL} – main reinforcement diameter;
- f_{ydL} – design yield strength for main reinforcement;
- f_{ydw} – design yield strength for shear reinforcement;
- b_C – column section width (the smaller dimension);
- h_C – column section height (the larger dimension);
- b_o – width of confined core, inside shear reinforcement;
- d_{bw} – shear reinforcement diameter;
- l_{cl} – clear storey height;
- l_o – lap length for main bars.

Save

You should save the section to the disk in order to use it further. Click the button. You will see a dialog showing general information about the section and results from code compliance checks. Enter section name. Do not include any file path or file extension. A good section name should provide information about shape, dimensions and reinforcement. That will make easier to use it further.



Results

Reinforcement design

Eurocode design basis

Column Expert uses the PMM Expert module internally to design columns. It takes into account the axial load, initial imperfections, nominal eccentricity and second order effects. You can find detailed description of the design procedure in the PMM Expert manual.

The design performed inside Column Expert is only valid for axially loaded columns. If there are considerable moments or axial shift of columns between storeys, you should make additional checks using PMM Expert.

There are three ways to use the reinforcement design procedure inside Column Expert:

Check the selected sections

Before you proceed to drawing, you can check if the selected sections provide sufficient capacity to take the column loads. First, you should go to the "**Column Loads and Sections**" table. Assign section numbers to all columns. Then press the button or the arrow▼ on the right and "**Check Using Current Sections**" submenu. The program calculates the capacities of all columns and fills them into the table. If the column load is greater than the respective capacity, the cell is painted in red, otherwise it is painted in green. That provides a clear view of all columns that fail and you can assign "stronger" sections to them.

Automatic reinforcement selection

Click the arrow next to the ▼ button and select the "**Automatic Reinforcement Selection**" submenu. Columns are first checked for the selected sections. For those that fail, new sections with the same dimensions but different reinforcement are automatically assigned. That requires several versions of each section to be preliminary loaded in the sections list. You can keep dimensions and bar positions equal for all sections and change only the diameters. If several sections are suitable for a single column, the one with minimum reinforcement is selected.

Automatic section selection

Click the arrow next to the ▼ button and select the "**Automatic Section Selection**" submenu. Columns are first checked for the selected sections. For those that fail, new sections with both different dimensions and different reinforcement are selected.

Calculation report

You can print a detailed calculation report including results and input data for materials, cross sections and dimensions. Click the button and you will open the report for preview. It is enabled when you activate the "**Columns Loads and Sections**" table. Also, you should have already performed design calculations. The program generates an html file and opens it with Internet Explorer.

View and Align Columns

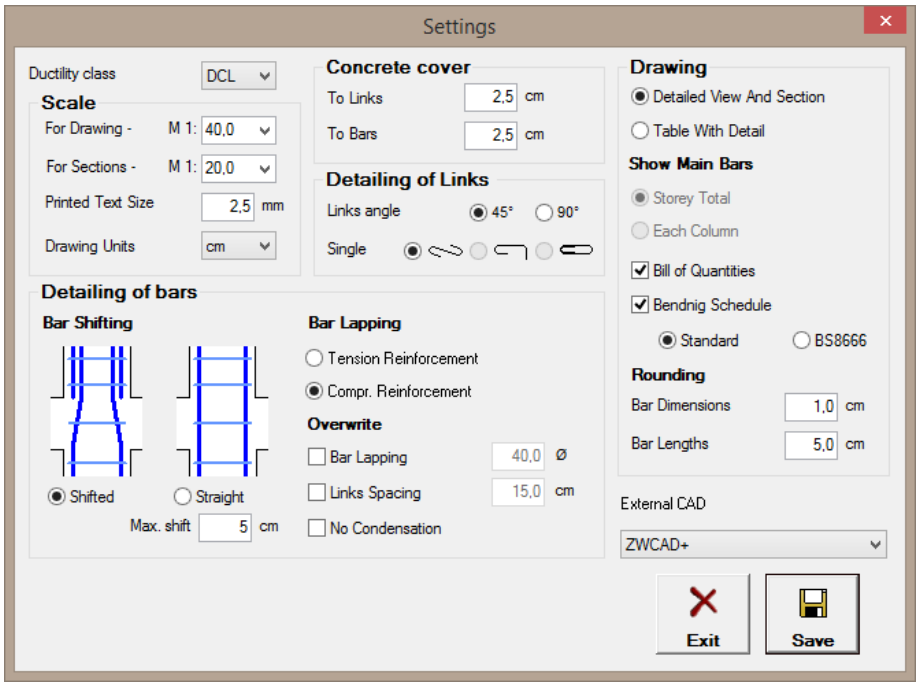
The next step after design is to review column geometry and align column centers. Click the button. The program loads the graphical environment and draws side views of all columns including dimensions and levels. By default, column centers on each storey are aligned in height to a vertical line.

You can move the column on each storey to left and right using the respective grip. Select the grip with the mouse first. Then use grip stretch or move commands (see "**Working with Design Expert graphical Environment**"). The value of the eccentricity "**e**" is also displayed in the drawing.

The program do not include these eccentricities in the design calculations automatically. You should check these columns additionally using PMM Expert.

Settings

Before proceeding further to reinforcement drawing, you should define the detailing settings. You can select different options in respect to reinforcement detailing, drawing scale and drawing layout. Click the button to view the settings dialog. You can specify the following settings:



Ductility class

Detailing requirements for different ductility classes are included. The program automatically applies the requirements for the selected ductility class during the detailing process. You should select one of the following possible values from the combo box: DCL, DCM or DCH. For more information, see "**Detailing requirements to Eurocode 2 and Eurocode 8**" above. If you need to design to Eurocode 2 only, select DCL. Otherwise, Eurocode 8 is applied.

Drawing scale

You can set different scales for elevations and sections. Specify text size in millimetres as it should appear in the printouts. Actual text size on the screen is automatically calculated according to the scale. You can also select different drawing units (mm, cm or m).

Concrete cover

This setting will apply only to new sections. Existing sections are not automatically modified each time you change the concrete cover. You should enter the distance between concrete surface and surface of reinforcement. It is possible to specify different values for main bars and shear links. In this case, bar positions are determined using the greater value of: cover to bars and cover to links + link diameter. In any case, the covers should not be less than bar diameter Ø + 10 mm. If this is not satisfied, the program automatically applies the necessary corrections.

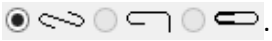
Detailing of bars

You can select whether bars should continue **straight** ——— or **shifted** ——‾— to allow easier installation of the upper reinforcement. The shift is equal to twice of bar diameter. If a bar goes outside the upper column it is shifted to fit inside, regardless this option. Sometimes it is more appropriate to stop such bars at the current floor and provide additional starter bars. This is controlled by the "Max. shift" distance. Bars that require greater shift than the max value are stopped and the others are shifted. Additional starters are provided for all bars in the upper column that cannot be lapped with bars in the lower column.

Bar lapping is calculated automatically to Eurocode, depending on concrete grade, reinforcement grade, diameter, etc. You can overwrite the default lapping by specifying your own value. Click the checkbox and set your own value relative to the diameter Ø.

Link spacing is calculated automatically according to Eurocode max. spacing requirements. Links are automatically condensed along the lapping zone and critical zone. Critical zone height is determined automatically. See "Detailing requirements to Eurocode 2 and Eurocode 8" for details. You can overwrite link spacing outside the lapping and critical zones. Click the checkbox and enter your own value in [cm]. If you select the "**No condensation**" option, all links will have equal spacing except for the lapping zone.

Detailing of links

You can select angle of hooks to be 45° or 90° and shape of single links  .

Drawing

There are two ways to draw columns – detailed views with elevations and sections or table view. Detailed views are suitable in all cases especially when you have complicated geometry with different sections at different storeys or axis shift. Table view is more compact and it is convenient when you have straight columns with simple geometry. Each column on each storey is represented by a separate cell in the table containing cross section drawing, main bars and shear links. Main bars can be shown separately for each column or total for the whole storey.

Bill of materials

Bill of materials (**BOM**) includes weight of reinforcement (kg), total and by bar size, as well as concrete volume (m^3) and formwork area (m^2).

Bending schedule

Bending schedule includes information about diameter, length, count, shape and dimensions for each bar mark. You can select between two styles of scheduling: "Standard" and „BS8666". The standard style includes drawings with dimensions for for each bar mark. BS8666 style is according to British Standard BS8666:2005.

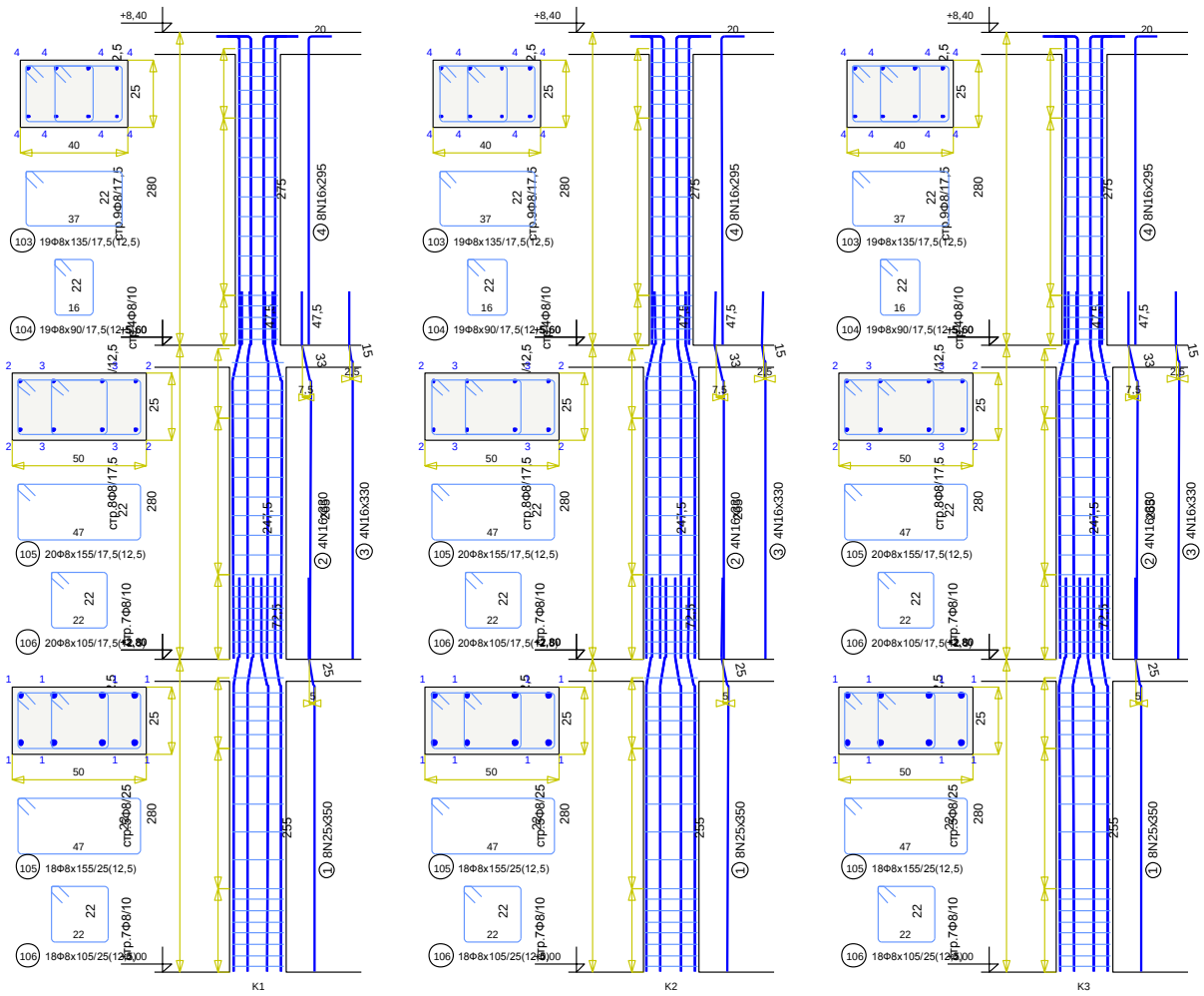
Each bar shape is represented by shape code and all dimensions (A, B, C etc.) are filled in a table. Bars are not drawn except for shape code 99.

External CAD

You can export the drawing to different CAD systems. You have to select the preferred system (ZWCAD+ or AutoCAD) in the combo box. See “Export to AutoCAD and ZWCAD” further in this manual.

Reinforcement Drawing

You can draw the reinforcement automatically by clicking the button. The button is enabled only at **“Columns Side View and Align”** page. After that, you will see the settings dialog. Make the necessary changes as decribed above and click „**Save**” or „**Exit**”, if no changes. The entire drawing is generated automatically based on data in the input tables. All bars and shear links are numbered automatically and their shapes, dimensions, lengths and counts are calculated as well. This is performed according to Eurocode detailing requirements as described above in this manual. You can view and modify the drawing inside Design Expert and then export it directly to ZWCAD+ or AutoCAD.



Working with Design Expert graphical environment

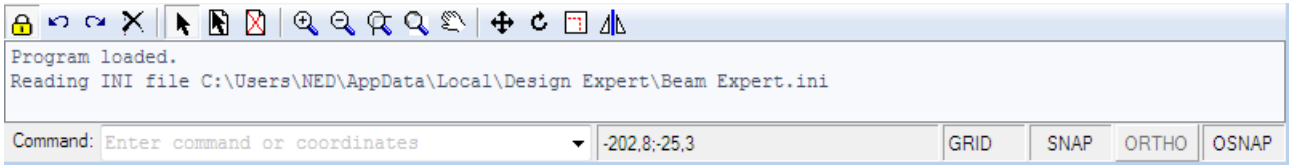
All drawings are generated in the internal Design Expert graphical environment first. There you can view, modify and align objects before exporting them to ZWCAD+ or AutoCAD. The graphical environment includes a basic set of commands for drawing and editing.

Commands

How to enter commands?

You can use several ways to enter a command in this program:

- Type it into the command line;
- Type the short version (command alias);
- Press a button on the toolbar;



Alternatively, instead of typing you can select the command from a drop down list by clicking the small arrow right to the command line. Some commands may require you to select objects or enter coordinates. You should watch the prompt on the left side of the command line. Press enter or right mouse button to complete a command that is running. You can cancel a command prematurely by pressing Esc or right mouse button. Commands generate various output including error or warning messages, results and general information intended for the user. You can find it in the output window just above the command line. You can start the previous command by pressing Enter or Space key instead of typing it again or pressing a button.

List of commands

A list of all available commands including icons, aliases and short descriptions is provided in the table below. You can find detailed descriptions of all commands further in this manual.

	Command	Alias	Description
	ZWCAD+ AUTOCAD	CAD	Export the current drawing to AutoCAD/ZWCAD+.
	COPY	CP, CO	Replicate the selected objects by moving, rotating, mirroring or scaling.
	COPYBITMAP	CB, COPYBMP	Copy the current drawing as Bitmap to system clipboard where it is available to paste in other programs.
	COPYMETAFILE	CM, COPYWMF	Copy the current drawing as Metafile.
	DELETE	E, D, DEL, ERASE	Delete selected objects from both screen and memory.
	DESELECTALL	DE, DESEL, DESELECT	Deselect all objects.
	DISTANCE	DI, DIST	Measure distance and angle between points.
	EXIT	QUIT	Close the program and exit.

	GRID	GR	Turn grid on and off.
	HELP		Display user manual.
	MIRROR	MI	Mirror the selected objects about a line defined by two points.
	MOVE	M, MO	Move the selected objects along a vector defined by two points.
	NEW	N	Create a new file.
	OPEN	O	Open an existing file from the disk
	ORTHO	OR	Turn orthogonal drafting mode on and off.
	OSNAP	OS	Turn object snap mode on and off.
	PRINT	PR, PRN	Send the current drawing to the printer.
	REDO	RE	Restore the last command after UNDO.
	REDRAW	RD	Redraw the screen view.
	ROTATE	RO	Rotate the selected objects about a specified center point and angle.
	RTPAN	PA, PAN	Move the screen view to other part of the drawing.
	SAVE	S	Save the current data to a file on the disk.
	SCALE	SC	Scale the selected object with specified center point and scale factor.
	SCRIPT		Save a script file (*.scr) with AutoCAD commands needed to create the current drawing in AutoCAD.
	SELECTALL	A, ALL, SELALL	Select all objects in the drawing that are not hidden or locked.
	SNAP	SN	Turn snap to grid mode on and off.
	UNDO	U	Undo the last command.
	ZOOMIN	ZI, Z+	Zoom in the screen view by factor of 1.5.
	ZOOMLIMITS	ZL, ZA, ZE	Zoom the screen view in order to fit all objects inside program window.
	ZOOMOUT	ZO, Z-	Zoom out the screen view by factor of 0.5.
	ZOOMWINDOW	ZW	Zoom the screen view in order to fit inside the specified rectangle.

The following commands are available only in the cross section editor:

	Command	**Alias**	**Description**
	BAR	B	Draw main bars.
	CHECK		Check design code requirements for the section.
	LINK	L	Draw shear links.
	OPTIONS	OP, OPT	Display settings dialog.
	SECTION	SE, SEC	Draw section with the mouse.

Undo wrong action or command

Click the button or type the UNDO command.

It cancels the results from the last command and recovers the previous drawing state. You can undo only one step back. If you need to go back further, use the other commands to recover the original drawing state.

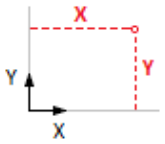
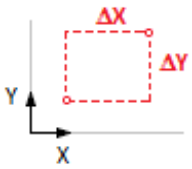
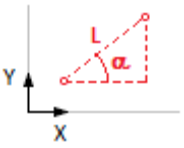
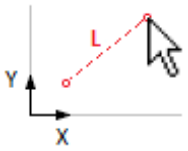
Redo a command that has been undone

Click the button or type the REDO command.

It repeats the last command in case it has been accidently undone. REDO must follow the UNDO command immediately before any other command. Otherwise, the command cannot be recovered.

Points and coordinates input

Design Expert has its own CAD environment where you can create and modify drawings. Some commands require the user to enter coordinates of points. You can do this by clicking with the mouse in the drawing window or by typing the coordinates in the command line. Typing input should follow some standard formats as described below. Coordinates can be absolute or relative to the previous point.

Type	Input format	Example	Description	Picture
Absolute	$X;Y$	10,5;15	Values are defined in global coordinate system Oxy.	
Relative	**_**$\Delta X;\Delta Y$ **@**$\Delta X;\Delta Y$	@25;35	Relative distances "25" и "35" to the previous point along X and Y, respectively.	
Polar	**<**$\alpha°;L$	<45;100	Distance of "100" is measured to the previous point at 45° angle from X axis.	
Distance	L	50	Distance of "50" to the previous point measured towards mouse cursor.	

Press Enter or Space after you enter the coordinates in the command line. If you want to enter points with the mouse, you have to move the cursor to the required location and click with the left mouse button. You can see the current coordinates of the cursor in the status bar located at the bottom of the main window. You can use several precision tools that can help you to get the exact coordinates when clicking:

- **GRID** – shows a uniform grid of dots over the working area of the drawing;
- **SNAP** – rounds the coordinates to a specified step along X and Y;
- **ORTHO** – orthogonal drawing mode. Current point is aligned to horizontal or vertical line with the previous point depending on the mouse position;
- **OSNAP** – gets the coordinates of an existing point in the drawing, when you move the mouse or click over it closer than a specified range. If several points are located within the range, the closest one is returned. When a point is snapped, an "×" mark appears on the screen. It is always the same symbol regardless the point type.

You can switch on and off the precision tools using the respective buttons on the status bar or by typing the respective commands in the command line.

Manage the screen view

The drawing is located in the model space and it is defined in global coordinate system Oxy. Then it is projected to the screen to certain scale. You can see only a part of the model space that is visible within the program window. We will call this “screen view”. You can scale and move the screen view over the drawing using ZOOM and PAN commands. That is how you can work with different parts of the drawing as necessary.

Zoom in and out

If you have a wheel mouse, you can zoom in and out by rotating the wheel forward and backward. The center of the transformation (the point that does not move) is assumed to be the current position of the cursor. You can move quickly to different parts of the drawing by positioning the cursor at different locations and zooming in and out. Also, you can use some additional commands as follows:

 *ZOOM IN* – zooms in the screen view with one step;

 *ZOOM LIMITS* – zooms the screen view so that all visible objects fit inside the screen;

 *ZOOM WINDOW* – zooms the screen view in a user defined window. When you start the command, you have to enter two points, at the opposite corners of the window;

 *ZOOM OUT* – zooms the screen view out with one step.

Pan

You can move the screen view at preferred direction in order to see other parts of the drawing. If you have a three-button mouse, you can use the middle button to pan. Press and hold the middle button, drag it to the new location and release the button. When you press the button, the cursor changes to  and when you release it, the old cursor is restored back.

Alternatively, you can use the  *RTPAN* command. It requires two points to define the length and the direction of movement (towards the second point). Since *RTPAN* is a command like any other, you have to finish the previous command before that. Unlike *RTPAN*, the middle button method can be used transparently inside any command without interrupting it.

Copy screen

You can copy the screen view to the clipboard any time and insert it into other programs using Paste command or *Ctrl+V.* You can use the following commands for coping:

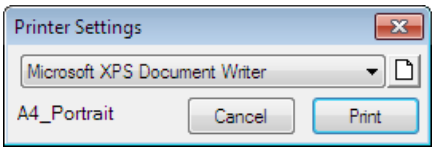
 *COPYBITMAP* – copies the screen image as Bitmap;

COPYMETAFILE – copies the screen image as Metafile.

Bitmap is a raster format file that stores information about colors of separate pixels. Metafile is a vector format file that stores coordinates of graphical objects. The boundaries of the copied image match the boundaries of the program window. Only objects that are visible on the screen will appear in the image. For best results, you can stretch the program window beforehand in order to fit the drawing tightly in the window without white spaces.

Print screen

You can send the screen view directly to the printer by pressing the button or typing the *PRINT* command. A setup dialog appears on screen. Select the required printer device from the list. You can change paper size and orientation as well as other options by clicking the button. Press the “Print” button to finish.



Modify objects

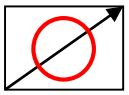
Block editing mode vs free mode

Graphics in Design Expert are represented by basic objects like lines, polylines, circles, texts, dimensions etc. They are grouped in blocks in order to form more complex objects like reinforcement bars, sections or entire elements. Each block is attached to one or more grips that are displayed as small blue boxes. By default, the drawing is locked and you can move only entire blocks using the respective grips. This is called “block mode”. You cannot modify separate objects within blocks. If you want to do that, you have to unlock the drawing first. Locking and unlocking is performed by clicking the respective buttons .

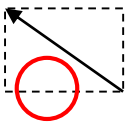
Select

Selection is a way to determine which objects should be affected by a certain command. You can select objects either before or after the command. There are several ways to select objects:

- Single – click on the object outline with the left mouse button. The outline should intersect the cursor selection box . If there are no object at the specified point, the program automatically continues to window selection mode.



- Window – you have to enter two points at the opposite corners of a window. If you draw the window from left to right, all objects that fit entirely inside are selected. If you draw the window from right to left, all objects that intersect or fit inside the window are selected. The window is displayed with solid line in the first case and dashed line in the second.



- All – selects all visible and unlocked objects. Click the button or type *SELECTALL* to start the command.

Selected objects are redrawn in red.

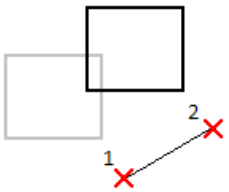
Deselect

Deselection is performed in the same way as selection but additionally you should hold the Shift button. Alternatively, you can click an object with the right mouse button. In order to deselect all objects, press Esc, click the button or type *DESELECTALL*.

Delete

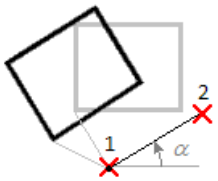
Click the button or type *DELETE*. All selected objects are erased both from screen and memory.

Move



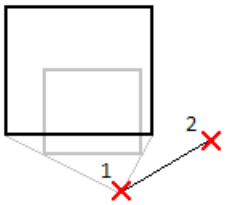
Moves the selected objects along a vector defined by two points. Click the button or type *MOVE*. Then enter first and second point and press Enter or click the right mouse button.

Rotate



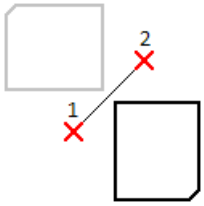
Rotates the selected objects around a center and with angle defined by user. Click the button or type *ROTATE*. Then enter first and second point and press Enter or click the right mouse button. The first point defines the center of rotation and the second is for the angle. The angle is measured between the line and the +X axis counterclockwise. You can also enter the exact value of the angle using polar coordinate input format. Type "<*α*;1" in the command line instead of clicking the second point, where *α* should be the rotation angle in degrees.

Scale



Scales the selected objects with a center and scale factor defined by user. Click the button or type *SCALE*. Then enter first and second point and press Enter or click the right mouse button. The first point represents the center of transformation. Scale factor is defined as the distance between the first and the second point. Alternatively, you can type the scale factor in the command line instead of entering a second point.

Mirror



Mirrors the selected objects about a line defined by user. Click the button or type *MIRROR*. Then enter first and second point and press Enter or click the right mouse button.

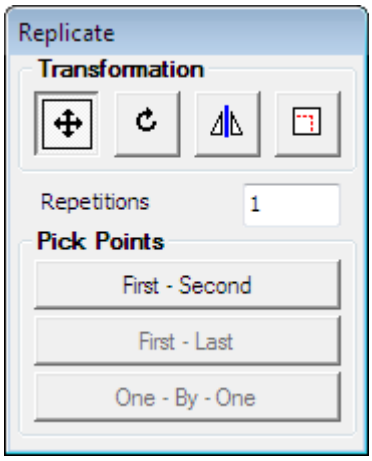
Stretch

When the drawing is unlocked, you can stretch separate objects like points, lines, polylines, dimensions, circles, polygons and texts by “dragging” with the mouse. Select the object and click on a point (end, middle or center point) to “catch” it. Then move the cursor to a new location and second click to “release” it. Texts are selected and moved using their base points displayed as small circles. If you stretch a line, polyline or polygon and you hold shift before the second click you will insert a new vertex.

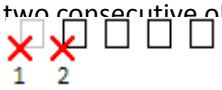
When the drawing is locked then you work in block editing mode. You can move entire blocks by stretching the respective grips. First, you have to select a grip by clicking with the mouse. Then, click again on the grip to “catch” it, move it to the new location and click to “release” it.

Copy

Creates one or multiple copies of the selected objects using one of the available transformations (move, rotate, scale or mirror). Click the button or type the *COPY* command. Select objects and press Enter or click the right mouse button. A settings dialog appears on screen. Select method of transformation using the icons on the top, number of repetitions and method of pointing:

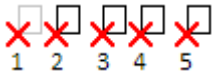


- **First – second** – click two points that define the distance between two consecutive objects;



- **First – last** – click two points that define the distance between the first and the last object. All other objects will be distributed evenly between them;

- **One - by - one** – click a base point first. Then you have to enter separate points to define the location of each object independently.



Coping is not available for some objects in some modules.

Export to ZWCAD+ or AutoCAD

You can export the drawing directly to ZWCAD+ or AutoCAD by clicking the / button from the main toolbar. It is always the same button, but the icon is different depending on the selected “External CAD” option in the settings dialog. If you click the arrow ▾ next to the button you can select other CAD system from the drop down menu. Supported versions are ZWCAD+ 2012 to 2015 and AutoCAD 2004 to 2015. If there is an instance of ZWCAD+ or AutoCAD already running, the drawing is sent to the active document. Otherwise, a new session is opened. Alternatively, you can type one of the following commands: *ZWCAD+*, *AutoCAD* or just *CAD*.

The drawing is exported as simple polylines, texts, dimensions, lines, circles and hatches. There are no blocks or any other complex objects, so it is easy to be modified with the standard AutoCAD commands. Current text and dimension styles are used. If you use templates, the drawing will look as any of your other drawings. For best results you have to define "Text Placement" to be "Over the Dimension Line, Without a Leader" in the dimension style settings. Objects are distributed in separate layers. If the required layers do not exist, they are created automatically. Reinforcement output is compatible to Design Expert Plug-in module. You can use it to additionally modify and schedule the reinforcement bars.

For versions not supported by the direct output, you can create AutoCAD command script files. Click the arrow next to the button and select “Save script file *.scr”. Enter file path and name and click “Save”. Then you can load the saved script into ZWCAD+ and AutoCAD using the *SCRIPT* command or menu "Tools\Run Script...“.