ПРОЕКТСОФТ

Software Package

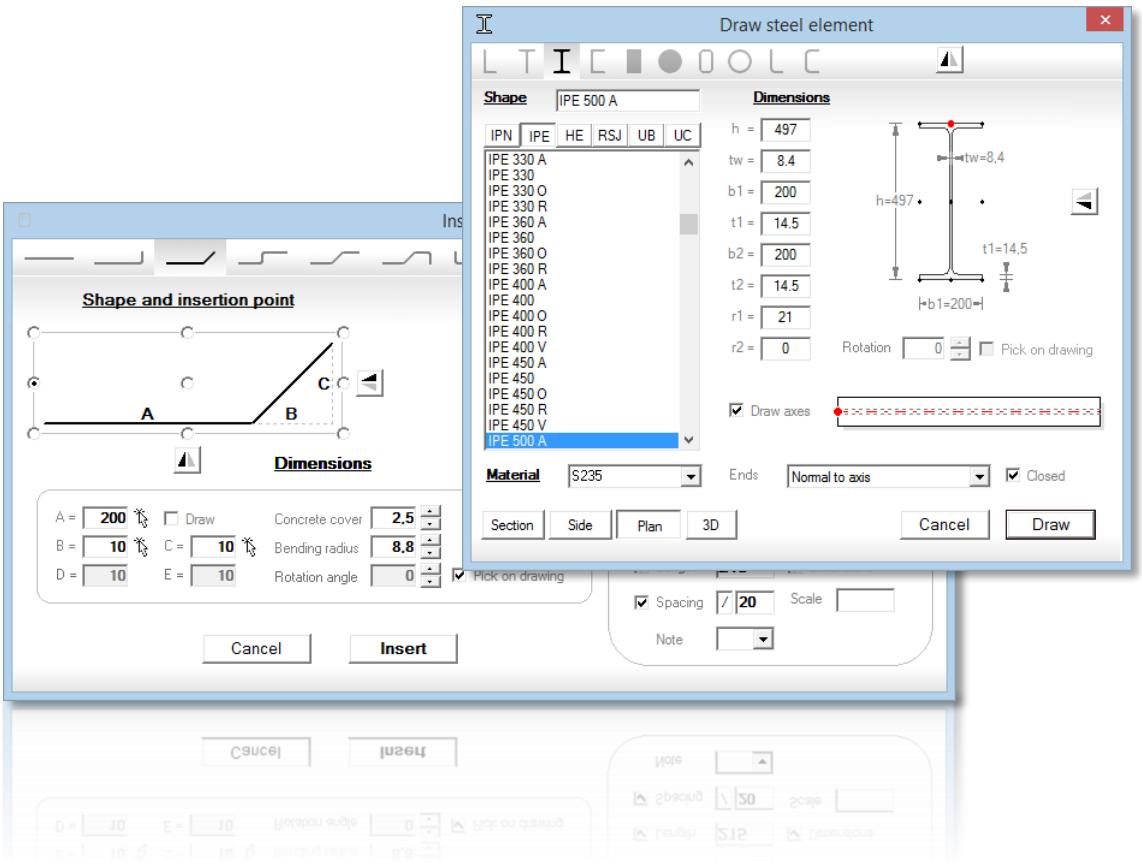
Design Expert version 2.7

Structural Design and Detailing to Eurocode

Plug-in for AutoCAD and ZWCAD+

Drawing of RC and steel structures

User manual



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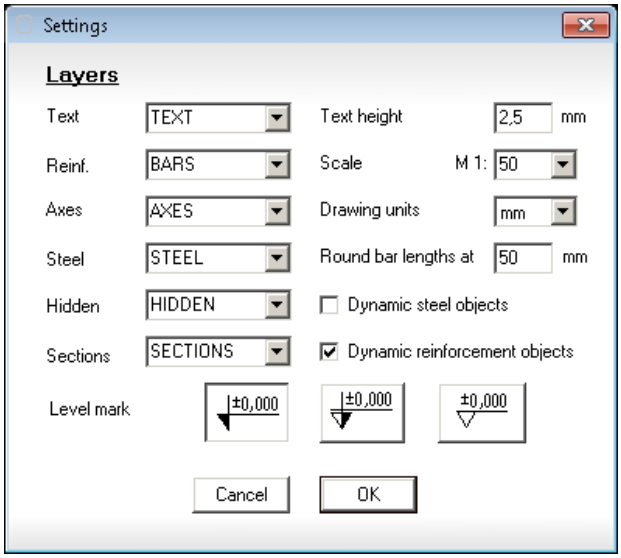
About the program

Design Expert Plug-in for ZWCAD and AutoCAD is created for detailing and drawing of RC and steel structures with ZWCAD, ZWCAD+ and AutoCAD. Commands can be started using the „Design Expert" menu, toolbar or the command line. This plug-in is compatible with the reinforcement output from other Design Expert modules for beams, columns, shear walls, foundations and stairs. Command aliases are not loaded automatically with ZWCAD but can be found in **c:\DesignExpert_EN\ZWCAD\Menu\DesignExpert.ica** and can be loaded manually into ZWCAD.

Settings

Command: DEXSET, DXS



Displays a dialog with the program settings. Always start with this command when you are going to work in a new drawing. You can select or enter preferred names for all layers, used by the program. Enter final text size in [mm] in "Text height" box as it should appear after printing the drawing to scale. Actual text size in the drawing is calculated automatically depending on specified units and drawing scale. For example, actual text size for displayed settings on right will be 2.5*50= 125 mm.

For reinforcement drawings select units to be [mm] or [cm] and for steel drawings always select [mm].

Select the "Dynamic steel objects" option when you are going to draw steel structures. Then the steel elements will be updated automatically in 2D or 3D each time when element axis is modified (with MOVE, STRETCH, GRIP, COPY etc.). Otherwise they should be updated manually with the "Steel Update" command.

Select the "Dynamic steel objects" option when you are going to draw RC structures. Then reinforcement data will be updated automatically each time when bars or zones are modified. For example, if you stretch the bar, bar length will be updated and if you stretch the arrangement zone, the corresponding bar count will be updated. Otherwise they should be updated manually using the "Bar label" command.

General rules for using commands

Most of the program commands require data input either by typing into the command line or by picking points or objects from the drawing. Commands follow the same general rules used for the native ZWCAD/AutoCAD commands.

Some commands have **parameters**, which can be used for changing command settings or switching to different path of execution. Parameters are displayed in square brackets **[]**, divided by slashes **/**. You need to type the capital letters of the parameter name and press **Enter** in order to select it from the list.

When you are prompted to enter a value in the command line, there is often a **default value** that you can select by pressing Enter. That makes the input faster and saves a lot of writing. Default values are displayed in brackets <>. For example, the command for drawing series of bars in section requires you

to enter bar spacing (which is used for calculation of total number). You can also enter the total number, alternatively, and bar spacing will be calculated using the total length and number. You will get the following prompt when you start the command:

Command: Bar spacing <200, 0 mm> or [Count]:

The default value in this case is ***<200, 0 mm*** >. If you press **Enter**, you will accept the default value. If you write a new value and press **Enter**, the new value will be used instead. In this command, ***[Count]*** is a parameter. In order to use the parameter, press ***C*** and then **Enter**. You will be prompted to enter the total bar number instead of bar spacing:

Command: Bar count <1>:

Dialog with program commands will be marked with ***BoldItallic*** further in this text.

Reinforced concrete detailing (ZwRC)

Bends and hooks

Command: BARROUND, RN

Appends bends and hooks to the selected bars. Bars should be defined as polylines.

You can select objects before or after you start the command. Then enter the following data in the command line:

Command: Bending radius <Use standard>:

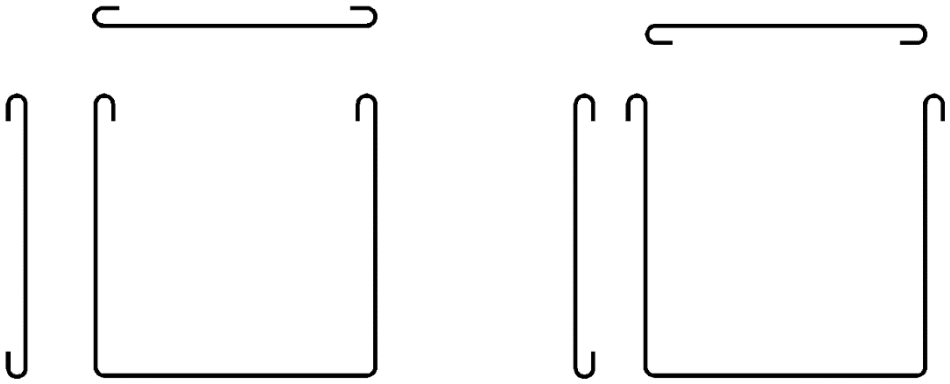
For precise drawing you should enter the radius at centerline r_o which is greater with 0.5·Ø than mandrel radius r_m. Recommended radiuses to Eurocode 2 are given in the following table:

Ø	≤ 16	> 16
r_m	2.0 Ø	3.5 Ø
r_o	2.5 Ø	4.0 Ø

Press ***Enter*** in order to use the standard radiuses. They will be calculated according to the above table. Bar diameter Ø is taken from the polyline global width even if it is not converted to bar yet.

Command: Hook length (0 – no hooks) <>:

If you enter 0, only bar corners will be rounded and no hooks will be added. If you enter a positive value, the hook direction will be top and left (as for bottom reinforcement) for straight bars and inside for bended bars. If you enter a negative value, the hook direction will be bottom and right (as for top reinforcement) for straight bars and outside for bended bars. The result from using the command with positive and negative values is shown on the picture bellow.



Command: Hook angle (0°, 45°, 90°) <>:

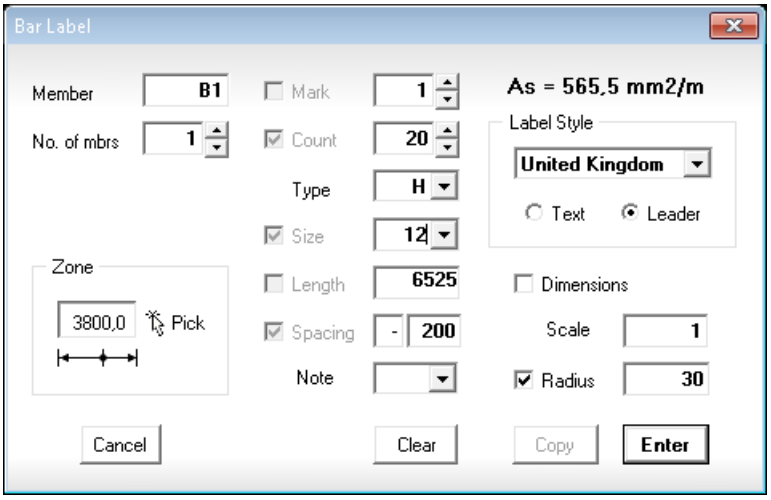
Default value is displayed in brackets <>. Press ***Enter*** to use the default value.

If you have selected bars preliminary, hooks are added to both ends of each polyline and the command is completed. If you have not, you are prompted to

Command: **Select Object:**

Click a polyline closer to the end where you would want to have a hook. That is how you can add a hook only at one side. You can continue with other polylines until you press ***Esc*** or click in empty space.

Bar label



Command: BARTEXT, BTE

With this command you can input bar data such as bar mark, count, type, diameter and spacing for selected bars. Dimensions can be optionally drawn as well. Bar length is calculated automatically. Reinforcement area „As" is calculated and displayed for information.

Bar data is stored into the selected polylines. When a data field is empty, current data from the polyline is used. The "***Clear***" button clears all fields. You can use this option to change only certain fields and keep the other. If you want to refresh the labels without changing data, clear all fields and click "***Enter***". If you want to copy bars, copy polylines only and use the command to create the labels.

Bar count is calculated automatically when you specify arrangement zone and spacing. Click the "Pick" button to specify the zone. You are prompted to select two points. Zone length is taken to be the distance between both points. Then you are prompted to confirm if the zone object is created. With „Object" option you can attach an existing zone to the bar (see „Bar arrangement zones"). You are prompted to select an object. If you select a line or a polyline, only length is taken. Zones are actually connected to the respective bars and count will be updated when you modify the zone object. Count is equal to zone length/spacing + 1. Count will be updated automatically if you modify the spacing. If you do not want that, switch the zone button off.

You can write number of members ***m*** and it will appear in the label as ***mxc*** (2x20). You can write additional ***nx***'s inside the count field. Then the label will be ***mxnxc*** (2x2x20). If bar zone is modified only the last portion of the count will be updated (2x2x25).

You can select what data will be included in labels by the respective checkboxes. With the „☑ ***Dimensions***" checkbox you can select if bar dimensions are drawn or not. You can specify a scale factor for dimensions and lengths. For example, if a bar is scaled twice, you should specify a scale factor „**x0.5**" in order to get the actual dimensions.

After you have entered all data, click the "***Enter***" button to draw or update labels. With the "***Copy***" button you can export bar and label to another location. This is used when you draw bars in sections (e.g. for columns, beams etc.).

Labels and zones are attached to the corresponding polylines to create complex objects "**bars**". If the "**Dynamic reinforcement objects**" option is selected in the "**Settings**" dialog, bars react dynamically to ZWCAD/AutoCAD commands (GRIP, STRETCH) and updates as necessary. For example, if you stretch the bar, length is updated and if you stretch the zone, count is updated and changed in the label. If you copy bars using COPY, ARRAY or MIRROR commands, new complex objects are created with the same behavior.

Link label

Command: LINKTEXT, LTE

Command is similar to previous, but only general dimensions for width and height are drawn instead of all straight segments.

Clear bar data

Command: BARCLEAR, BCL

Clears all reinforcement data from the bar and deletes the label. Bar is converted to a simple polyline which will not be included into the schedule.

Draw bars

Command: BARDRAW, BD

Draws a bar with arbitrary shape as a polyline, adds hooks and bends and displays the "***Bar label***" dialog.

When you start the command you are prompted to enter a point:

Command: Start point [Offset <>]:

With the "***Offset***" option (***O***) you can specify offset distance. Current value is displayed in brackets <>.

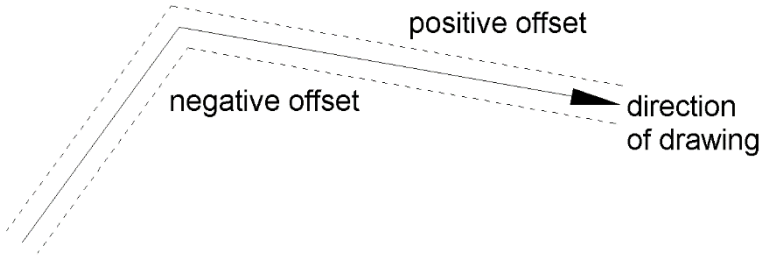
Command: Enter offset distance [Pick] <>:

Enter distance or type "***Pick***" (***P***) to pick the distance on screen.

Command: Distance first point:

Command: Distance second point:

When the offset is positive, the bar is drawn on the left side if you look towards the direction of drawing. When the offset is negative, the bar is on the right side according to the following picture:



After you enter the offset, the command returns to the beginning.

Command: Start point [Offset <>]:

Enter a point.

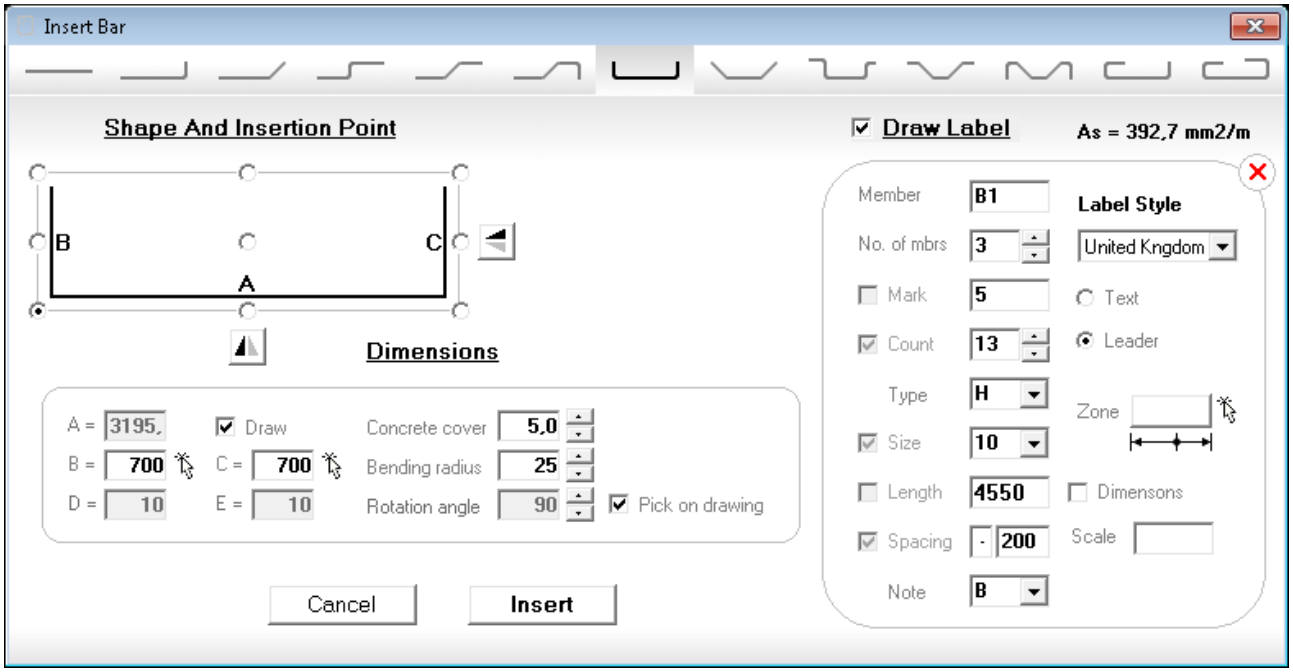
Command: Next point [Undo/Close]:

Enter next point and continue until you draw the polyline. Press "***Enter***" or right click to stop. Polyline is drawn parallel to the picked sequence of points at the specified offset distance. If you specify the offset to be the concrete cover, you can pick your points along the concrete edge and get your bar drawn inside the concrete.

Insert bars

Command: BARINSERT, BI

Inserts a standard bar into the drawing. The following dialog is displayed when the command is started:



Select bar shape from the toolbar on the top. Below is a bar scheme where dimensions "***A – E***" are indicated. Select an insertion point using option buttons on the scheme. Enter values for "***A – E***" in "***Dimensions***" box as well as concrete cover, bending radius and rotation in plan. You can pick a dimension directly from the drawing with the respective button. Concrete cover is automatically extracted so you can pick on concrete edge for your convenience.

Select the "***Draw Label***" checkbox to add a label to the bar. Data is entered in the same way as the „***Bar label***" command.

When you are ready with the required data, click the "***Insert***" button and select insertion point. If "***Draw***" is checked for dimension "***A***", you should specify second point to define bar right end. Dimension "***A***" is calculated from the specified points. If "***Pick on drawing***" is checked for rotation angle then rotation is defined by the specified points as well.

Bar arrangement zones

Command: BARZONE, BZ

Draws a bar arrangement zone. Enter first and second point and select bar(s). Zone is represented by dimension line and circle at intersections with bars. Bar count is calculated automatically. The zone is attached to the bars and their count is updated automatically each time when the zone is modified.

Draw single open link

Command: LINKDRAW0, SD

Draws single shear links. When the command is started you are prompted to enter:

Command: First point [Cover<>/Radius<>/Hooks<>/Angle<>]:

Type „***C***" (Cover), „***R***" (Radius), „***H***" (Hooks) or „***A***" (Angle) to change concrete cover, bending radius, hooks length or hooks angle, respectively. Current values are given in brackets <>. Enter first and second points on concrete edge. Concrete cover is extracted automatically from total length. Then link is drawn and the labeling dialog appears. Click "***Copy***" instead of "***Enter***" if you want to copy the link to another location.

Draw single closed link

Command: LINKDRAW1, SD

Command works exactly as previous, but draws a closed link.

Draw open link

Command: LINKDRAW2, LDO

Command is similar to previous, but you should specify the opposite corners of the section.

Draw closed link

Command: LINKDRAW3, LDC

Command is similar to previous.

Draw double closed link

Command: LINKDRAW4, LDD

Command is similar to previous.

Draw bars in section

Command: BARSECTION, BS

Draws a sequence of circles with specified diameter and spacing along path defined by the user. Command requires input of diameter, spacing and sequence of points as follows:

Command: Bar size <>:

Command: Bar spacing < > or [Count]:

If you type „***C***" (Count), you can enter total count for each straight segment and spacing is calculated from segment length and count. Then proceed with:

Command: Start point:

Command: Next point:

Circles are drawn in the specified segment. You can continue drawing segments until you press „***Esc***" or right click to finish.

Bars in section along object

Command: BARDIVIDE, BD

Draws a sequence of circles with specified diameter and spacing along path defined by selected objects. This command works as follows:

Command: Select Objects

Select single or multiple objects and press „***Enter***" or right click to continue. You can select different types of objects such as lines, polylines, arcs and circles.

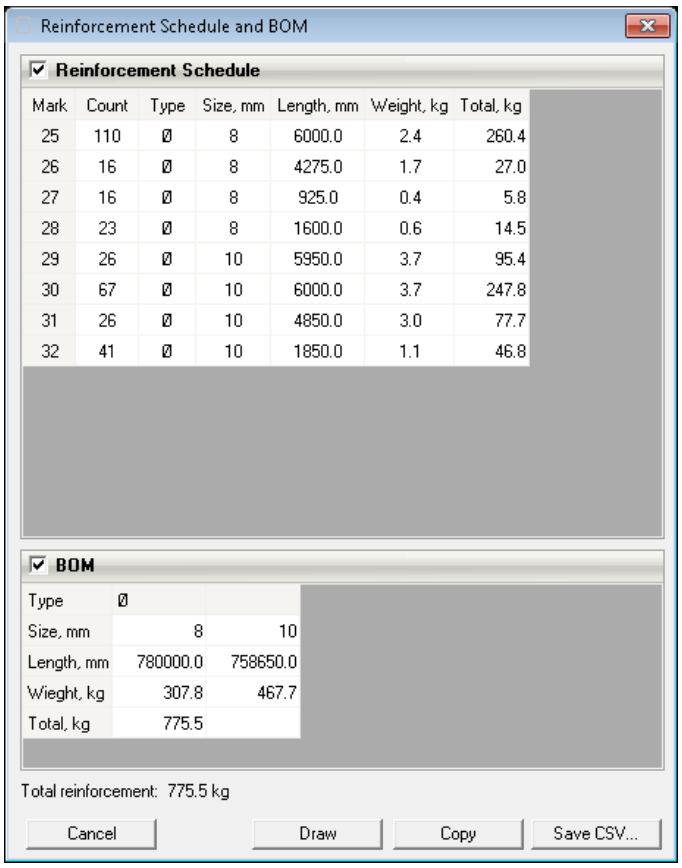
Command: Bar size <>:

Command: Bar spacing <> or [Count]:

Bending schedule

Command: BARSCHED, BSC

Automatically calculates and draws reinforcement bending schedule and bill of materials (BOM). All reinforcement should be drawn as polylines in the respective layer as defined in "***Settings***". Bar data should be assigned to all polylines using the "***Bar label***" command. If UK settings are used for labeling of bars, the schedule is generated according to BS 8666:2005, otherwise, a general format is used.



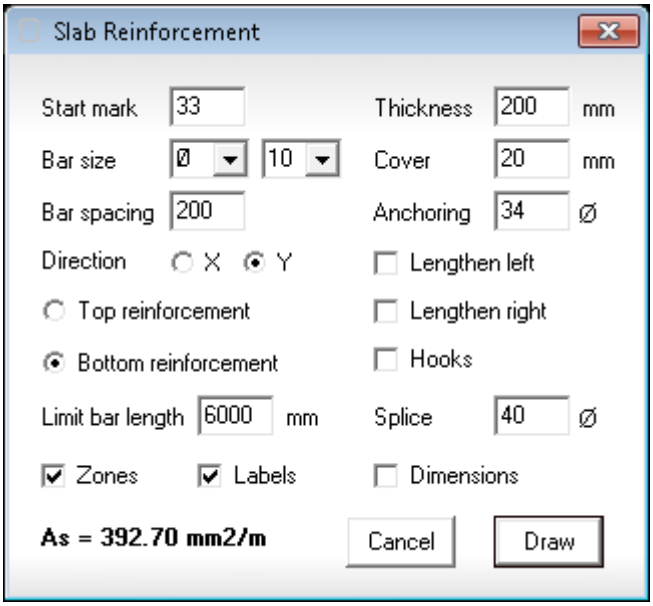
When the command is started you are prompted to select objects. All objects that do not satisfy the above criteria are filtered out. Objects are highlighted so you can check if your selection is correct. Then press "***Enter***" or right click.

The program collects bar data and builds the schedule. It compares data and when it finds different bars with same marks it fixes them automatically after user confirmation. You can miss bar marks completely when drawing and the program will number them for you in the end.

Bending schedule and BOM are presented in tables. Click the "***Draw***" button to insert the tables into the drawing at selected location. The "***Copy***" button sends table contents to system clipboard and you can paste it into other programs. The "***Save CSV…***" button writes a ****.csv*** file and opens it with the default application (e.g.MS Excel).

Slab reinforcement

Command: PLATE, PLA



Automatically generates uniform reinforcement with the selected bar size and spacing inside outlines with arbitrary shapes. Outlines should be defined as closed polylines. They could represent separate spans of ribbed slabs, edges of flat or foundation slabs or partial zones with additional top or bottom bars. Start the command and select the outlines. Input dialog appears on screen.

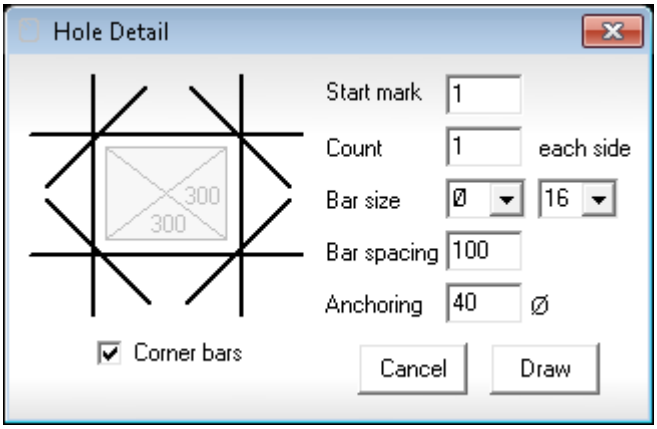
You should enter start bar number, bar size and spacing, direction X/Y, position top/bottom, concrete cover and anchoring length. If „☑ **Lengthen left/right**" is checked, bars are lengthened outside the selected outline by anchoring length. This is used to provide additional anchorage at supports or beyond the zone where reinforcement is theoretically required.

You can also specify limit bar length which is possible to be delivered/produced. If outline dimensions are greater than limit length then several bars are lapped to fill the outline.

If you want labels immediately, you can check „☑ ***Labels***". If you are going to modify the reinforcement, uncheck this setting. You can always add labels with "***Bar Labels***" command. Check „☑ ***Zones***" setting to add arrangement zones.

Holes

Command: HOLE, HO



Draws reinforcement details around holes. You are prompted to select two opposite corners of the opening:

First corner [Object]:

Second corner:

Press „***O***" (Object) to select hole outline from the drawing. It should be defined as closed polyline.

A settings dialog appears on screen. Anchorage should be measured outside the hole edge.

Design and draw beams

Command: BEAM, BE

Starts **BeamExpert.exe** application. For more information see Beam Expert.pdf.

Draw columns

Command: COLUMN, CL

Starts **ColumnExpert.exe** application. For more information see Column Expert.pdf.

Draw shear walls

Command: SHEARWALL, SW

Starts **ShearWallExpert.exe** application.

Design and draw foundation pads

Command: PAD

Starts **PadExpert.exe** application. For more information see Pad Expert.pdf.

Design and draw stairs

Command: STAIR, STA

Starts **Stair Expert.exe** application.

Groups on

PICKSTYLE 1, Ctrl+Shift+A

Objects that are used to draw a reinforcement bar (polyline, text, dimensions etc.) are grouped. If grouping is on you can select the whole group by clicking just one of the included objects. That makes selection of bars easier in crowded drawings when bars or text overlap. You can use this option when you want to move or copy the entire bar with all of its components.

Groups off

PICKSTYLE 0, Ctrl+Shift+A

You can turn grouping off, when you want to select and modify some components of bars separately. You can move only the text or the zone without moving the polyline.

Steel structures (ZwSteel)

Draw gridlines

Command: GRIDLINE, GRL

Draws sequence of structural gridlines. Numbers (letters) for the names increase automatically. Gridlines can be orthogonal or inclined. Spacing can be variable, but if you have equal spacing, you can draw multiple gridlines at a time by specifying their number. The command performs in the following sequence:

Command: First point:

Command: Second point:

Enter the first and second point of the first gridline. It is not necessary to be orthogonal and all the remaining gridlines created within the current command will be parallel to the first one. If you have several sequences of gridlines with different directions, you will have to run this command several times, one for each sequence. You can draw a single gridline as well.

Command: Extend at ends <>:

Enter a distance to extend the line beyond the points entered. Default distance is displayed in brackets <>. Press ***Enter***, without writing anything to accept the default distance.

Command: Gridline name or "0" to finish <1>:

Enter number, letter or both for gridline name. Special symbols are also allowed. If the gridline direction is closer to Y, then number is proposed by default, starting from 1 and automatically increasing for the next gridlines. If the gridline direction is closer to X, then starting letter “A” is proposed by default, also increasing, following the English alphabet. If you want to change the automatic sequence, you can type a new value and press ***Enter***.

Command: Distance to the next gridline (L or n*L) <>:

Enter the distance to the next gridline. The default value in brackets <> shows the last distance entered. If you have several gridlines with consecutive numbers and equal spacing, you can draw all of them at once. Enter the number and the distance as **n*L**. For example, if you have 4 gridlines at 6000 mm spacing, enter **4*6000**.

Command: Gridline name or "0" to finish <2>:

Enter the name of the next gridline or press ***Enter*** to accept the default value. You will continue drawing gridlines in the same way until you enter 0 as a gridline name or ***Esc***.

Draw levels

Command: LEVEL, LVL

Draws sequence of level marks in elevation. Level values are calculated automatically according to their relative position to the specified zero level. You can select different styled of level using the settings command (DEXSET). The command is performed in the following sequence:

Command: Direction [X/Y] <Y>:

It is possible that an elevation or section is rotated along X direction. Then you should select X, otherwise select Y. Level values will increase from bottom to top and right to left on the drawing.

Command: Enter base point at level zero:

Enter a point with vertical position at level 0.00. The horizontal position of the point will be used to align all level marks one above the other.

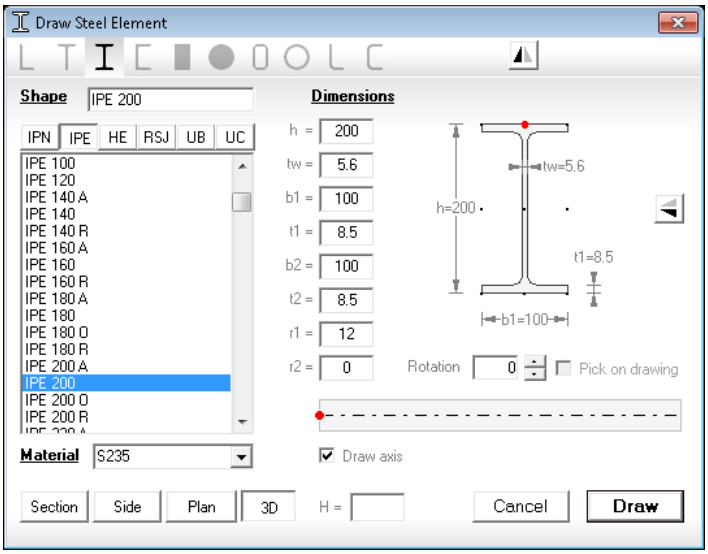
Command: Enter point on the section/elevation:

Pick points on the section/elevation drawing where you want to add a level mark. Level marks are inserted as blocks with attributes and values are calculated automatically.

Draw steel elements

Command: STEELDRAW, STL

This command is created for drawing of steel elements in 2D and 3D.



Select cross section type from the toolbar on top. List of standard sections are loaded on left and you can select one. Dimensions are loaded from corresponding text files, stored in „.\ZWCAD\SteelShapes\" subfolder. You can open these files with Excel and add or modify sections following the same format. Section name is filled in the "***Shape***" field. You can change it or you can add more information like identification number, mark, etc. If section is not standard you can enter custom dimensions in the corresponding fields.

You can specify materials as well. Materials list is stored in file „.\ZWCAD\Materials.txt".

You can draw elements in different 2D views: section, side, plan or 3D. You need to specify one point to insert a section and two points to define element axis for other views. If you have a vertical column in 3D you can input column height into the "***H=***" field. Then you need to specify only one point to insert the column. Red point indicates the alignment point which you will specify in the drawing. Section will be positioned relative to this point as shown. Click on scheme to change the alignment point. Use the and buttons to mirror the section about the horizontal and vertical axis, respectively. For sections and 3D columns you may specify a rotation angle. If „☑ ***Pick on drawing***" is checked then you will be prompted for second point that will define the rotation angle. Click "***Draw***" to close the dialog and proceed to drawing. Sections are represented by closed polylines, side and plan views are represented by parallel dashed and solid lines for the corresponding visible and hidden edges and 3D elements are drawn as 3D solids. Axis line is added and it has an important function – it contains element data used by all other commands. If you copy an axis alone, data is copied as well. Then element edges are drawn automatically.

Input steel sections

Command: STEELINPUT, STI

Assigns sections and materials to existing axes and draws elements in side view, plan or 3D. You can use this commands to modify elements sections and redraw them with the new section. Axes must be line objects in the respective layer (see “Settings”). Start the command, select objects and press “***Enter***” or right click. Data is entered into the same dialog as the “Draw steel elements” command.

You can draw a frame in 3D, export geometry to structural analysis software and calculate the required sections. Then you can assign them to the elements and generate automatic bill of materials and 2D structural plans and elevations using the commands bellow.

Draw steel sections

Command: STEELSECTION, STS

Draws sections of selected elements in a 2D drawing. You are prompted to enter two points. The line defined by these points should intersect the elements we need. Sections are drawn along the line at intersections with the respective elements.

Draw main sections and elevations

Command: STEELVIEW, STV

Automatically generates vertical sections and elevations out from a 3D drawing. You are prompted to enter two points that defines a window. Elements that will be shown in view should fit entirely into the window. Then enter an insertion point on left or right side. Elements that cross the window on the opposite side are drawn as sections.

Update drawing 2D/3D

Command: STEELUPDATE, SU

Automatically redraws selected elements that contain steel data. A 3D drawing can be converted to 2D one and all z coordinates will be set to zero. That is how you can generate 2D plans out of 3D models. You can create a 2D plan of a separate floor in multistory building. Just copy all elements from the floor including columns outside, select them and start the command.

Labels of steel elements

Command: STEELTEXT, STT

Automatically draws a label for each element that contains section name.

Steel bill of materials

Command: STEELBOM, STB

Automatically composes and draws bill of materials (BOM) for the steel elements created by the program. Drawing should be in [mm]. First, you are prompted to select elements axes. Then calculated BOM is displayed in a table. Click the "***Draw***" button to insert the table into the drawing at selected location. The "***Copy***" button sends table contents to system clipboard and you can paste it into other programs. The "***Save CSV…***" button writes a ****.csv*** file and opens it with the default application (e.g. MS Excel).

You can generate BOM out of 2D and 3D drawings. Sections in 2D drawing will not be included.