User Guide

Autodesk®
March 2009

TABLE OF CONTENTS

1. PROGRAM DESCRIPTION

1.1. General description of the program

AutoCAD Structural Detailing has been developed in order to facilitate preparing final drawings of component parts of designed structures. After completing the stage of architectonic structure design and generating a structure calculation model (this stage comprises calculations and verification of structure elements), the stage of preparing project technical documentation with required final drawings takes place.

AutoCAD Structural Detailing - Reinforcement allows creating detailed drawings of RC structure reinforcement; it is equipped with a complete set of options for drawing details of an RC structure drawing, which are adjusted to the engineer-designer's needs and make it possible to draw objects in an intuitive manner.

AutoCAD Structural Detailing - Reinforcement is a program enabling generation of detailing drawings of RC structure elements; it is divided into a few parts which are responsible for:

1. edition of a drawing or part of a drawing (projections, intersections, etc.) - including additional drawing elements, correction of existing structure elements, adding structure element dimensions
2. generation of final drawings
3. printout management.

AutoCAD Structural Detailing combined with ***Autodesk Robot Structural Analysis*** allows the user to make a complete project of a structure:

- generation of a structure model and structure calculations
- structure design (e.g. calculation of reinforcement required in RC structure elements)
- generation of final drawings allowing edition of prepared drawings.

The following objects have been distinguished in the program:

- View - a single drawing; it is always a document's component, if it has been added to a printout, it is simultaneously a printout's element. NOTE: only a view (drawing) contained in a printout may be printed
- Printout - ready-to-use printout composed of views; its equivalent in the AutoCAD ® program is a layout together with AutoCAD views provided on it; for each printout there is exactly one layout corresponding to it.

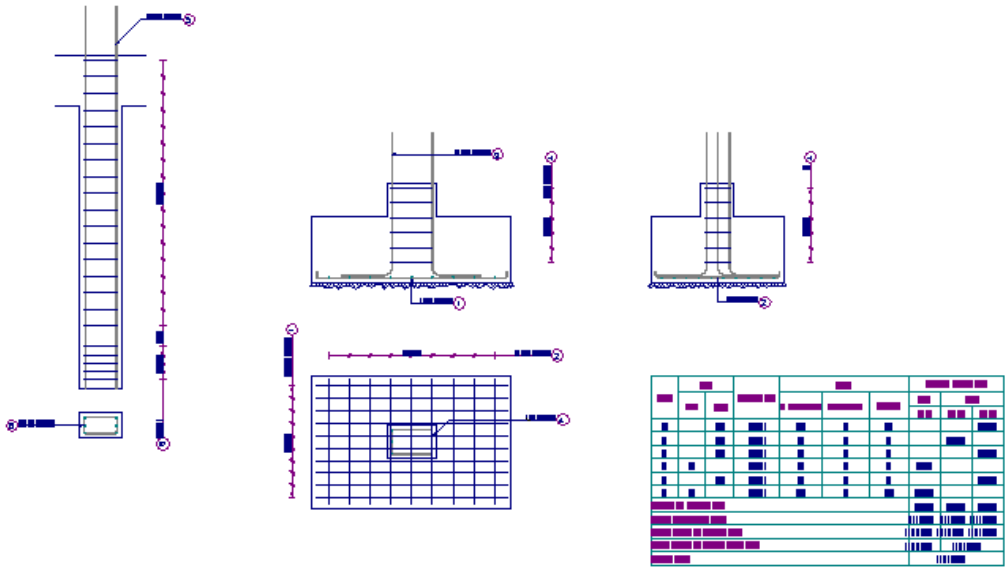
The elements listed concern the stages of work on a project.

1.2. Stages of work on structure project drawings

The work concerned with creating the documentation of a designed structure (drawings) may be divided into the following stages:

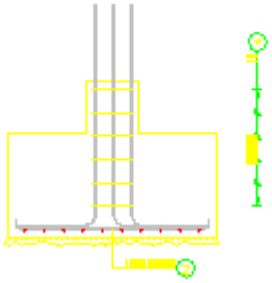
1. Definition of documents

A document - a set of drawings (views) of RC structure elements; a document consists of views (a document cannot be printed).



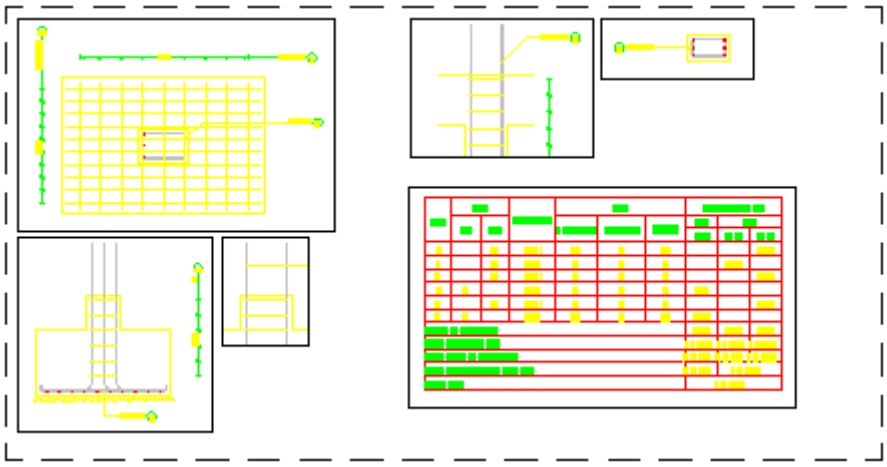
2. Definition (edition) of views

A view - a single drawing which always constitutes a document component; if it has been added to a printout, then it is simultaneously a printout element - only a view (drawing) contained in a printout may be printed.



3. Printout generation

A printout - prepared, ready-to-use printout; it consists of views (for each Printout there is exactly one layout corresponding to it).

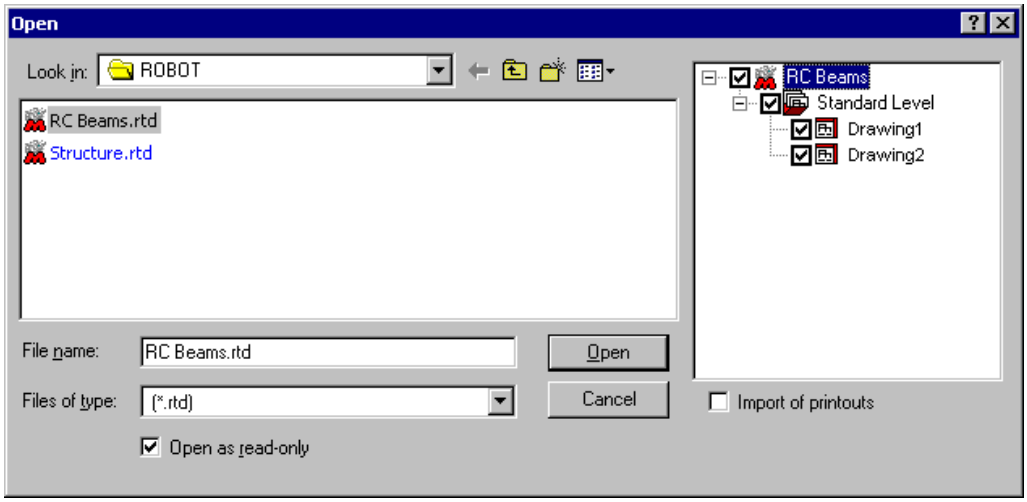


All operations are performed in printout layouts.
A printout layout is an object of the ***AutoCAD*** ® program. It is used for composition of a final printout. For each printout layout there is one printout.

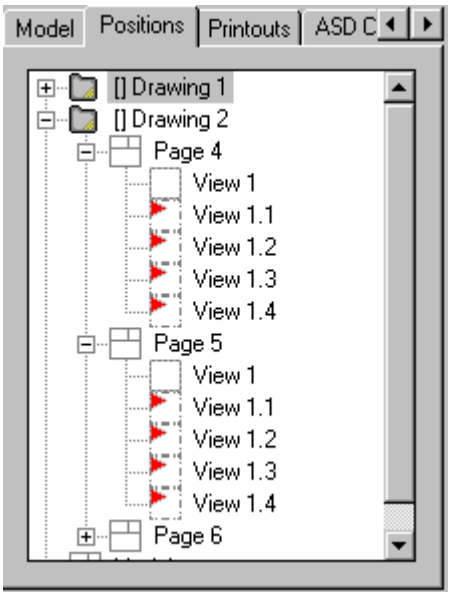
1.3. Example of loading a drawing created in the Robot program

To load a drawing made in an ***Autodesk Robot Structural Analysis*** RC module (beams, columns, spread footings, etc.), follow the steps below:

- run the menu option *Reinforcement / Insert drawing from Robot* or press the icon 
- in the ***Open*** dialog box shown below, in the tree located in the right-hand side of the dialog box indicate a drawing to be opened in ***AutoCAD Structural Detailing - Reinforcement*** (in this case these are *Drawing1* and *Drawing2* on the standard level, belonging to the ***RC Beams*** project)
  NOTE: If the *Import of printouts* option is switched on, then the program will load drawings prepared for printing in the ***Robot*** file.



- press the **Open** button in the above dialog box; the selected drawings of an RC beam will be loaded to ***AutoCAD Structural Detailing - Reinforcement***, in the ***Object Inspector*** dialog box, on the *Positions* tab for each drawing the program will create positions including appropriate views of the RC beam elements (see the drawing below).



In ***AutoCAD Structural Detailing - Reinforcement*** there is a possibility to edit the loaded drawings and to prepare final drawings of RC structure elements.

1.4. Options available in the menu

All the options available in ***AutoCAD Structural Detailing - Reinforcement*** (menu of the RC part) are presented below. The following information is included: position of the option in the text menu, icon symbolizing this option, command activating the option from the command line and short description of the option.

Reinforcement elevation  - Opens the ***Reinforcement - elevation*** dialog box; the option allows definition of reinforcing bars (longitudinal reinforcement) in an element of an RC structure.
Menu: the *Reinforcement / Reinforcement - elevation* command
Command line: RBCR_DEF_BAR_BV

Reinforcement cross-section  - Opens the ***Reinforcement - cross-section*** dialog box; the option allows definition of reinforcing bars (transversal reinforcement) in a cross-section of an RC structure element.
Menu: *Reinforcement / Reinforcement - section* command
Command line: RBCR_DEF_BAR_BS

Special stirrups  Opens the ***Special stirrups*** dialog; use this option to define special stirrups (transversal reinforcement) in the cross-section of an RC structure element.
Menu: *Reinforcement > Special stirrups* command
Command line: RBCR_DEF_STIRRUP_SPEC

Reinforcement- point  Opens the **Reinforcement- point** dialog box; the option enables defining distribution of reinforcement which is presented as points (reinforcement n cross-section).
Menu: *Reinforcement / Reinforcement- point* command
Command line: RBCR_DISTRIBUTION_POINT

Special reinforcement


Opens the **Special reinforcement** dialog box; the option enables definition of particular reinforcing bars used in different elements of RC structures (e.g. crest-shaped reinforcement, corbel reinforcement, transport handles, etc.).
Menu: *Reinforcement / Special reinforcement* command
Command line: RBCR_DEF_BARLIBSPECIAL

Wire fabrics in cross section


Opens the **Wire Fabric Shape** dialog box; the option enables definition of a wire fabric in the cross section of an RC structure element.
Menu: *Reinforcement / Wire fabrics in cross section* command
Command line: RBCR_DEF_NET_SIDE

Wire fabrics in cross section - symbol


The option is used to present an indicated wire fabric in cross section outside the formwork contour to show a full shape and geometry of a reinforcement (this is a detailed presentation of a wire fabric needed for a bar bender to shape the reinforcement properly).
Menu: *Reinforcement / Wire fabrics in cross section - symbol* command
Command line: RBCR_DEF_NET_PULL

Reinforcement - bar legend


The option allows presenting an indicated bar outside the formwork contour to show the whole reinforcement shape and geometry (it is a detailed bar presentation needed for a bar bender to shape reinforcement properly).
Menu: *Reinforcement / Reinforcement - bar legend* command
Command line: RBCR_DEF_PULL

Reinforcement detailing


The option permits definition of reinforcement distribution.
Menu: *Reinforcement / Reinforcement detailing* command
Command line: RBCR_DEF_BAR_DISTRIBUTION

Surface reinforcement – wire fabrics


Opens the ***Surface reinforcement distribution - wire fabrics*** dialog box; the option enables defining regions of wire fabric distribution (e.g. reinforcement of RC plates).
Menu: *Reinforcement / Surface distribution – wire fabrics* command
Command line: RBCR_NETD_RECT

Surface reinforcement - bars


Opens the ***Surface reinforcement - bars*** dialog box; the option enables defining regions of bar distribution (e.g. reinforcement of RC slabs).
Menu: *Reinforcement / Surface distribution - bars* command
Command line: RBCR_DEF_BAR_SURF

Radial reinforcement - bars


Opens the ***Radial surface reinforcement*** dialog box; the option enables defining radial reinforcement (e.g. reinforcement of round RC slabs).
Menu: *Reinforcement / Radial distribution* command
Command line: RBCR_CREATE_RADIAL

Surface reinforcement - wire fabrics


Opens the ***Surface reinforcement distribution - wire fabrics*** dialog box; the option enables defining regions of wire fabric distribution (e.g. reinforcement of RC slabs).
Menu: *Reinforcement / Surface reinforcement – wire fabrics* command
Command line: RBCR_NETD_RECT

Steel profiles

Steel profiles


Opens the ***Steel profiles*** dialog box; the option enables defining steel profiles.
Menu: *Reinforcement / Definition - steel profiles / Steel profiles* command
Command line: RBCR_CREATE_STEEL_VIEW

Steel profiles - description

Opens the ***Profile description*** dialog box; the option enables description of an indicated steel profile.
Menu: *Reinforcement / Definition - steel profiles / Steel profiles - description* command
Command line: RBCR_CREATE_STEEL_DESC

Steel profiles - section

Selecting this option results in generation of a steel profile section; to create a vertical section, the user should indicate a cutting line (two points defining a segment) and a section 'depth'.
Menu: *Reinforcement / Definition - steel profiles / Steel profiles - section* command
Command line: RBCR_CREATE_STEEL_SECTION

Cut profile to line

The option enables cutting a steel profile so that it fits a plane defined by the line.
Menu: *Reinforcement / Definition - steel profiles / Cut profile to line* command
Command line: RBCR_CREATE_STEEL_CUT

Delete cut

The option enables deleting a cut of the steel profile made earlier.
Menu: *Reinforcement / Definition - steel profiles / Delete cut* command
Command line: RBCR_CREATE_STEEL_CUTS

Bar description

The option allows defining a description of a reinforcing bar.
Menu: *Reinforcement / Reinforcement description / Bar description* command
Command line: RBCR_BARBVFORMDESC

Reinforcement distribution description

The option enables defining a description of reinforcement distribution
Menu: *Reinforcement / Reinforcement description / Reinforcement distribution description* command
Command line: RBCR_DISTRIB_DESC

Bar ends

The option allows defining a symbol of bar ends.
Menu: *Reinforcement / Reinforcement description / Bar ends* command
Command line: RBCR_BAR_END

Description of wire fabric in cross section

The option enables defining a description of a wire fabric in the crosss section.
Menu: *Reinforcement / Reinforcement description / Description of wire fabric in cross section* command
Command line: RBCR_DEF_NET_SIDE_DESC

Styles of reinforcement description
Opens the ***Description of reinforcement shape*** dialog box; the option allows defining description styles (format) for individual elements of reinforcement.
Menu: *Reinforcement / Reinforcement description / Styles of reinforcement description* command
Command line: RBCR_SHAPE_DESC

Bars - Main table

The option enables adding a main table for reinforcing bars presented in a drawing at a point in the drawing indicated by the user.
Menu: *Reinforcement / Reinforcement table / Bars - Main table* command
Command line: RBCR_LIST_BAR_MAIN

Bars - Element table


The option enables creating a reinforcement table which presents reinforcing bars divided into structural elements (beams, columns, etc.).
Menu: *Reinforcement / Reinforcement table / Bars – Element table* command
Command line: RBCR_LIST_BAR_ELEM

Bars - Detailed table


The option enables adding a detailed table for a reinforcement position with variable distribution or with bar surface distribution which is provided in a drawing at the point indicated by the user.
Menu: *Reinforcement / Reinforcement table / Bars - Detailed table* command
Command line: RBCR_LIST_BAR_DETA

Bars - Summary table


The option allows adding a summary table for reinforcing bars presented in a drawing at a point indicated by the user.
Menu: *Reinforcement / Reinforcement table / Bars - Summary table* command
Command line: RBCR_LIST_BAR_SUM

Wire fabrics – Main table


The option enables adding a main table for wire fabrics presented in a drawing at a point in the drawing indicated by the user.
Menu: *Reinforcement / Reinforcement table / Wire fabrics - Main table* command
Command line: RBCR_LIST_NET_MAIN

Wire fabrics – Summary table


The option allows adding a summary table for wire fabrics presented in a drawing at a point in the drawing indicated by the user.
Menu: *Reinforcement / Reinforcement table / Wire fabrics - Summary table* command
Command line: RBCR_LIST_NET_SUM

Update - reinforcement tables


If this option is activated, it results in updating a selected table after making changes in reinforcement geometry/parameters.
Menu: *Reinforcement / Reinforcement table / Update - reinforcement tables* command
Command line: RBCR_LIST_ACT

Table Printout/Export/Edit


The option permits printing a table or exporting a table of the indicated reinforcement to an *.xls or *.csv format file.
Menu: *Reinforcement / Reinforcement table / Table Printout/Export/Edit* command
Command line: RBCR_LIST_EXP

Styles - reinforcement tables

Opens the ***Reinforcement tables - style manager*** dialog box; the option enables defining/modifying styles of the reinforcement tables applied to prepare a reinforcement table for RC structure elements.
Menu: *Reinforcement / Reinforcement table / Styles - reinforcement tables* command
Command line: RBCR_LIST_PAR

Insert drawing from *Autodesk Robot Structural Analysis*


The option allows inserting a drawing prepared in ***Autodesk Robot Structural Analysis***.
Menu: *Reinforcement / Insert drawing from Robot* command
Command line: RBCR_TOOL_IMPORT_RM

Reinforcement areas from *Autodesk Robot Structural Analysis*


The option allows reading values of required (theoretical) reinforcement areas calculated for a plate in ***Autodesk Robot Structural Analysis***. A ***Robot*** file (*.rtd) is read, which provides values of top and bottom reinforcement for every finite element on the plate. After opening the file in ***AutoCAD Structural Detailing*** plate reinforcement is presented in a form of crosses which disappear once the appropriate plate reinforcement is generated in ***AutoCAD Structural Detailing***.
Menu: *Reinforcement / Reinforcement areas from* ***Robot*** command
Command line: RBCR_TOOL_IMPORT_PL

Typical structures - reinforcement

Submenu with commands which enable defining reinforcement of typical elements of RC structures.

Spread footing


The option permits defining reinforcement of a typical spread footing once several characteristic parameters are determined in the dialog box (spread footing geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Spread footing* command

Sleeve footing


The option enables defining reinforcement of a typical sleeve footing once several characteristic parameters are determined in the dialog box (footing geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Sleeve footing* command

Continuous footing


The option permits defining reinforcement of a typical continuous footing once several characteristic parameters are determined in the dialog box (continuous footing geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Continuous footing* command

Column


The option permits defining reinforcement of a typical column - circular or rectangular, once several characteristic parameters are determined in the dialog box (column geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Column* command

Opening


The option permits defining reinforcement of a typical opening - circular or rectangular, once several characteristic parameters are determined in the dialog box (opening geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Opening* command

Corner


The option permits defining reinforcement of a typical corner once several characteristic parameters are determined in the dialog box (corner geometry/reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Corner* command

Slab corner


The option enables defining reinforcement of a typical RC slab corner after specifying a few characteristic parameters in the dialog box (slab corner geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Slab corner* command

Distribution of prefabricated slabs

The option enables defining distribution of prefabricated slabs and generating reinforcement for these slabs after specifying several characteristic parameters in the dialog box (geometry of the slab distribution region/ reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Distribution of prefabricated slabs* command

Beam

The option enables definition of typical beam reinforcement after specifying a few characteristic parameters in the dialog box (beam geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Beam* command

Stairs

The option enables definition of typical stair reinforcement after specifying a few characteristic parameters in the dialog box (stair geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Stairs* command

Pile cap

The option enables definition of typical reinforcement of a pile cap (pile foundation) after specifying a few characteristic parameters in the dialog box (pile cap geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Pile cap* command

Pile

The option enables defining reinforcement of a typical pile of pile foundation after specifying a few characteristic parameters in the dialog box (pile geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Pile* command

Ground beam

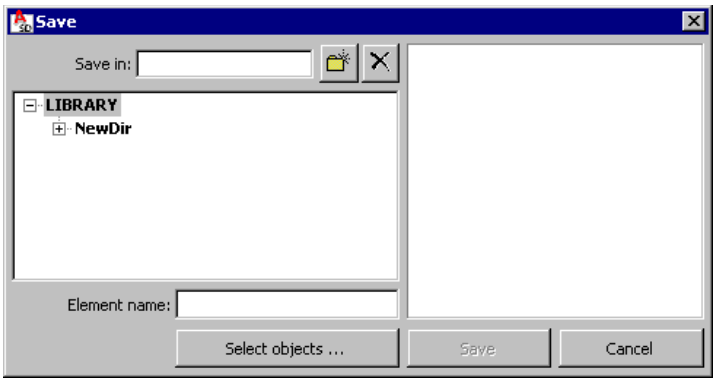
The option enables definition of typical reinforcement of a ground beam after specifying a few characteristic parameters in the dialog box (beam geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Ground beam* command

Parapet

The option enables definition of typical reinforcement of a parapet after specifying a few characteristic parameters in the dialog box (parapet geometry / reinforcement parameters).
Menu: *Reinforcement / Typical structures - reinforcement / Parapet* command

Create linear element

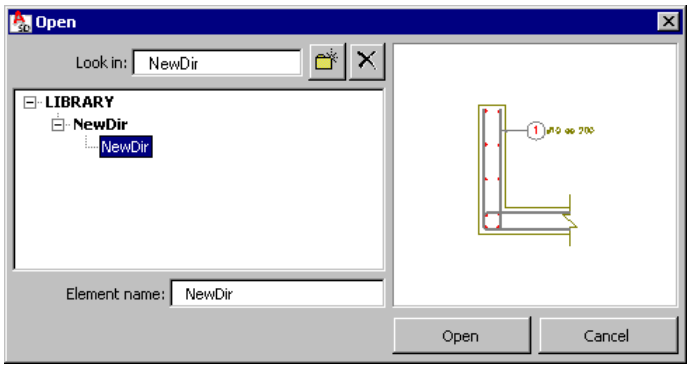
The option enables definition of an RC structure element (section of an RC element) that will be assigned one characteristic: element length. Once the option is selected, the screen shows the dialog box presented below:



After pressing the **Select objects** button, an element (a block of drawing elements) should pointed on the screen. Such an element may be saved to the database.
Menu: *Reinforcement / Typical structures - reinforcement / Create linear element* command

Insert linear element

The option enables insertion of an RC structure element (section of an RC element) that has been assigned length. Once the option is selected, the screen shows the dialog box presented below:



After selecting a linear element saved earlier on the disk, a block of drawing elements is placed in a drawing. The linear element chosen is assigned name and length; the spacing of elements over the length (e.g. 3 meters of length with spacing every 20 cm) is also specified.
Menu: *Reinforcement / Typical structures - reinforcement / Insert linear element* command

Formworks

Submenu with commands which enable defining the formwork of typical elements of RC structures. .

Spread footing

The option enables defining the formwork of a typical spread footing after providing several characteristic parameters in the dialog box (spread footing geometry).
Menu: *Reinforcement / Formworks / Spread footing* command

Continuous footing

The option enables defining the formwork of a typical continuous footing after providing several characteristic parameters in the dialog box (continuous footing geometry).
Menu: *Reinforcement / Formworks / Continuous footing* command

Column

The option enables defining the formwork of a typical column after providing several characteristic parameters in the dialog box (column geometry).
Menu: *Reinforcement* / *Formworks* / *Column* command

Beam

The option enables defining the formwork of a typical beam after providing several characteristic parameters in the dialog box (beam geometry).
Menu: *Reinforcement* / *Formworks* / *Beam* command

Stairs

The option enables defining the formwork of typical stairs after providing several characteristic parameters in the dialog box (geometry of stairs).
Menu: *Reinforcement* / *Formworks* / *Stairs* command

Pile cap

The option enables defining the formwork of a typical pile cap (pile foundation) after providing several characteristic parameters in the dialog box (pile cap geometry).
Menu: *Reinforcement* / *Formworks* / *Pile cap* command

Ground beam

The option enables defining the formwork of a typical ground beam after providing several characteristic parameters in the dialog box (beam geometry).
Menu: *Reinforcement* / *Formworks* / *Ground beam* command

Parapet

The option enables defining the formwork of a typical parapet after providing several characteristic parameters in the dialog box (parapet geometry).
Menu: *Reinforcement* / *Formworks* / *Parapet* command

Set scale of reinforcement description

The option enables changing scale of reinforcement description; once this option has been applied, the drawing displayed on the *Model* tab shows no changes, because modification of the scale is presented while generating a printout (a final drawing) on printout layouts.
Menu: *Reinforcement* / *Tools* / *Set scale of reinforcement description* command
Command line: RBCR_DESC_SCALE

Create projection plane

The option allows defining a view used during generation of a final drawing; while generating a projection plane, the user should specify its scale.
Menu: *Reinforcement* / *Tools* / *Create projection plane* command
Command line: RBCT_ADDVIEW

Element manager

The option enables division of reinforcing bars into structural elements (beams, columns, etc.).
Menu: *Reinforcement* / *Tools* / *Create element* command
Command line: RBCR_CREATE_ELEMENT

Create cross-section

The option allows creating a cross-section; to create a cross-section, select an object, cutting line (two points defining a segment) and cross-section 'depth'. Use this option for surface and linear distributions.
Menu: *Reinforcement* / *Tools* / *Create cross-section* command
Command line: RBCR_CREATE_ELSECTION

Copy view

The option allows copying a selected view. You can use this option for views (of cross-sections, for example) in which top / bottom reinforcement of a slab is displayed. Copied views are mutually linked.
Menu: *Reinforcement* / *Tools* / *Copy view* command
Command line: RBCR_COPY_VIEW

Edit bar / wire fabric database

The option allows editing databases of reinforcing bars and wire fabrics (modifying parameters of bars or wire fabrics, adding new ones).
Menu: *Reinforcement* / *Tools* / *Edit bar* \ *wire fabric database* command
Command line: RBCR_TOOL_DBEDIT

Multiple reinforcements

The option allows defining a multiple of a selected reinforcement (the reinforcement quantity in a structure); the value specified is provided in the reinforcement summary table (in the *Number of elements* column).
Menu: *Reinforcement* / *Tools* / *Multiple reinforcements* command
Command line: RBCR_TOOL_QUANTN

Reinforcement - information

The option enables displaying basic information concerning indicated reinforcing bar or reinforcement distribution.
Menu: *Reinforcement* / *Tools* / *Reinforcement – information* command
Command line: RBCR_TOOL_INFO

Renumbering of reinforcement position

The option enables changing the reinforcement numbering; the following elements (assigned to the bar shape) are considered during renumbering: spacing description, bar description placed outside the formwork contour, reinforcement tables, etc.
Menu: *Reinforcement* / *Tools* / *Renumbering of reinforcement position*
Command line: RBCR_TOOL_RENUM

Find reinforcement

The option allows finding a reinforcement position in a generated drawing.
Menu: *Reinforcement* / *Tools* / *Find reinforcement* command
Command line: RBCR_TOOL_FINDR

Show reinforcement without description
The option enables marking (highlighting on the screen) the reinforcement for which a description has not been generated yet.
Menu: *Reinforcement* / *Tools* / *Show reinforcement without description* command
Command line: RBCR_TOOL_SELNDSC

Add lap splices
The option allows generating automatically lap splices for point reinforcement bars that are not assigned a shape (only their length is defined) and whose length exceeds the commercial length of reinforcing bars (for example 12 m). The software generates bar lap splices in accordance with options for lap splices of reinforcing bars on the *Codes* / *Materials* tab in the Job Preferences dialog.
Menu: *Reinforcement* / *Tools* / *Add lap splices* command
Command line: RBCR_DISTRIBUTION_POINT_ADD_LAP

Explode

The option allows exploding some of the composed objects into individual elements.
Menu: *Reinforcement* / *Tools* / *Explode* command
Command line: RBCR_EXPLODE

Reinforcement calculator

The option opens the Reinforcement calculator dialog box. Bar diameters and reinforcement areas are given in units that have been selected in Preferences. The calculator enables calculation of the following quantities:

- reinforcement areas, for example:
  7*ϕ 12 = 7.92 cm2
  7*ϕ 12 + 5*ϕ 16 = 17.97 cm2
  7*ϕ 12 + 5*ϕ 16 + 8*ϕ 10 = 24.25 cm2

- required number of reinforcing bars (for example 44/ϕ14 = 29 bars)
- required number of reinforcing bars with the assumed diameter (e.g. 18 and 12 mm) with additional assumption that numbers of bars with each diameter are approximately equal (for example: 44 /ϕ 18 /ϕ 12 = 12*ϕ 18.0 + 12*ϕ 12.0)
- required number of reinforcing bars with the assumed diameter (e.g. 18 and 12 mm) in such a way so that bars with 12 mm diameter constitute a certain percent of all bars (for example: 44 /ϕ 18 /ϕ 12 %25 = 16 * ϕ 18.0 + 5 * ϕ 12.0)
- difference between the area given (e.g. 44 cm2) and the total area of indicated reinforcing bars (for example: 44 - 5* ϕ 12 = 38.35 cm2).

Menu: *Reinforcement / Tools / Reinforcement calculator* command
Command line: RBCR_TOOL_CALCULATOR

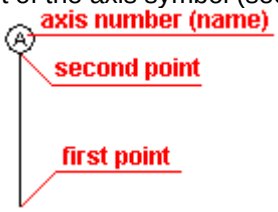
Save model in DWG format

The option enables saving a model of a structure element in a DWG format file. It allows opening a file in the AutoCAD ® program and carrying out further operations on a generated drawing.
NOTE: If a drawing is saved in a *.DWG format file, and next opened in AutoCAD ® on a computer where ***AutoCAD Structural Detailing - Reinforcement*** is **NOT** installed, then diameter symbols will not be displayed. For diameter symbols to be displayed on such a computer, the user should additionally copy the diam.sex file to the appropriate AutoCAD folder (the file has to be copied to the folder to which the path is set in the AutoCAD ® program).
Menu: *Reinforcement / Tools / Save model in DWG format* command
Command line: RBCT_MODELEXPORT

Graphic elements

Insert axis

The option enables inserting an axis to a selected place in a drawing. Symbols are drawn according to the default style set in the ***Job Preferences*** dialog box. To insert a symbol of a structural axis in a drawing, do as follows:
1 select the command *Reinforcement / Graphic elements /Insert axis*
2 enter a number (name) of the structural axis
3 indicate the first point of the axis symbol
4 indicate the second point of the axis symbol (see the drawing below).



The axis number is proposed according to the settings in the default style; while inserting the axis any number may be typed (every next one will be inserted according to the recently-applied numbering). The axis number may be modified using the context menu option.

Menu: *Reinforcement / Graphic elements /Insert axis* command
Command line: RBCT_DEF_SYMBOL_AXIS

Insert section symbol

The option enables inserting a section symbol to a selected place in a drawing. Symbols are drawn according to the default style set in the ***Job Preferences*** dialog box. To insert a section symbol in a drawing, do as follows:

1 select the command *Reinforcement / Graphic elements /Insert section symbol*
2 enter a number (name) of the section
3 indicate the first point of the section
4 indicate the second point of the section.
Profile numbering is proposed according to the settings in the default style; while inserting section symbols any number may be typed (every next one will be inserted according to the recently-applied numbering). The section symbol may be modified using the context menu option.

Menu: *Reinforcement / Graphic elements /Insert section symbol* command
Command line: RBCT_DEF_SYMBOL_SECTION

Insert elevation mark

The option enables inserting an elevation mark to a selected place in a drawing. Symbols are drawn according to the default style set in the ***Job Preferences*** dialog box. To insert an elevation mark in a drawing, do as follows:
1 select the command *Reinforcement / Graphic elements /Insert elevation mark*
2 determine the reference (zero) level
3 indicate a point on a selected level.
Levels inserted during one session are linked with each other; when several levels are defined a value of the level inserted first must be specified, whereas the remaining ones are entered depending on the place where the symbol is inserted. Modification of an elevation mark may cause changes in values of individual levels (the remaining ones are recalculated), deleting designations, adding new symbols to an already-existing group.

Menu: *Reinforcement / Graphic elements /Insert elevation mark* command
Command line: RBCT_DEF_SYMBOL_LEVEL

Styles – graphic elements

Opens the ***Styles of symbols*** dialog; use this option to define styles (format) of symbols presented in structure drawings (elevation mark, section symbol or structural axis symbol).
Menu: *Reinforcement / Graphic elements / Styles – Graphic elements* command
Command line: RBCT_DEF_SYMBOL_STYLE

Modify

Reinforcement

The option allows modifying parameters of a selected reinforcement (reinforcing steel grade, diameter, etc.). The dialog box appears, in which reinforcement parameters may be changed.
Menu: *Reinforcement / Modify / Reinforcement* command
Command line: RBCR_MOD_REINF

Graphical parameters of reinforcement

The option allows modification of graphical parameters of a selected reinforcement (filling, color, etc.) and of switching on/off descriptions of values for reinforcement crosses imported from ***Autodesk Robot Structural Analysis***. The dialog box appears, in which reinforcement parameters may be changed.
Menu: *Reinforcement / Modify / Graphical parameters of reinforcement* command
Command line: RBCR_MOD_PROP

Lap splices


The option enables modifying lap splice parameters in bars. The program displays the dialog box where the user may modify lengths of lap splices in bars.
Menu: *Reinforcement / Modify / Lap splices* command
Command line: RBCR_MOD_BAR_LAP

Reinforcement description


The option enables modifying description parameters of a selected reinforcement and description of automatic distribution of wire fabrics. The dialog box appears, in which parameters of reinforcement description may be changed.
Menu: *Reinforcement / Modify / Reinforcement description* command
Command line: RBCR_MOD_DESC

Cover


The option enables changing a cover value for the existing reinforcement; this parameter refers to a cover of bar segments, cover of bar ends (bars are mainly ended with hooks), to region for distribution varying linearly and wire fabrics in cross section.
Menu: *Reinforcement / Modify / Cover* command
Command line: RBCR_TOOL_EDCOV

Bent diameters


The option allows modifying values of bend diameters of reinforcing bars and wire fabrics in cross section.
Menu: *Reinforcement / Modify / Bent diameters* command
Command line: RBCR_TOOL_EDBEND

Length of bar segment


The option enables modifying lengths of reinforcing bar segments and wire fabrics in cross section. A value of lengthening or shortening of a bar segment is entered directly from the keyboard.
Menu: *Reinforcement / Modify /Length of bar segment* command
Command line: RBCR_TOOL_EDSEGM

Delete first/last bar segment


The option enables deleting the first or the last element of reinforcement.
Menu: *Reinforcement / Modify / Delete first/last bar segment* command
Command line: RBCR_TOOL_BAR_DEL

Add first/last bar segment


The option allows adding the first or last reinforcement element.
Menu: *Reinforcement / Modify / Add first\last bar segment* command
Command line: RBCR_TOOL_BAR_ADD

Modification of surface distribution region


The option enables modifying a cover value or values of support width of the region for surface distribution regions (bar distribution or wire fabric distribution).
Menu: *Reinforcement / Modify / Modification of surface distribution region* command
Command line: RBCR_MOD_CHBOUNDARY

Job Preferences


Opens the ***Job Preferences*** dialog box; the option enables assuming basic parameters applied in ***AutoCAD Structural Detailing*** (codes, units, materials, etc.).
Menu: *Reinforcement / Job Preferences* command
Command line: RBCR_JOB_PREF

Preferences

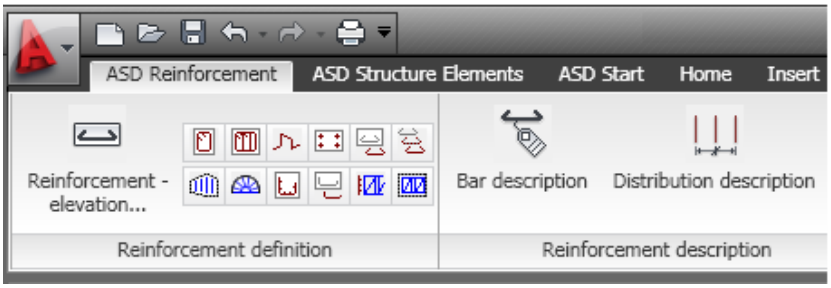

Opens the ***Options*** dialog box; the option enables setting parameters of work environment in ***AutoCAD Structural Detailing***.
Menu: *Reinforcement / Preferences* command

Check for updates The option, which when run, opens the Internet page of the software producer or dealer; from this page the user will be able to download program updates (Service Pack, information about a new version, etc.).
Menu: *Reinforcement / Check for updates* command
Command line: RBCT_UPDATESERVICE

Object Inspector – Show / Hide 
The option enables switching on / off the presentation of the ***Object Inspector*** dialog box provided by standard in the left part of the screen.
Menu: *Reinforcement / Object inspector – Show / Hide* command
Command line: RBCTOI

1.5. Ribbon

The ribbon is an element of the user interface which replaces the traditional menu and toolbars and allows easy managing and adjusting the workspace.



The ribbon consists of several panels, grouped on tabs that are designated by task or subject. The ribbon panels include many commands that have been on toolbars and in dialogs so far, such as icons, drop-down lists, sliders, text fields and other elements characteristic of a given tab. Using the ribbon, you do not have to display many toolbars; thus the application displays fewer functions and increases the allowable workspace placing the whole interface on a small area that can be anytime shown or hidden.
The ribbon displays automatically when a drawing is created or opened using the *2D Drafting* or *3D Modeling* workspace. You can display the ribbon manually using either of the following methods:

- select the main menu Tools > Palettes > Ribbon
- type *RIBBON* in the command line to show the ribbon or *RIBBONCLOSE* to hide it.

You can customize the ribbon, that is you can add, delete and modify positions of panel elements, in the Customize User Interface (CUI) editor window. Open this editor using either of the following methods:

- click 

on the Manage tab > Customization > User Interface
- type *CUI* in the command line.

NOTE:
You can display the ribbon horizontally, vertically or as a floating palette.
Using the editor you can also switch between workspaces (such as the classic workspace without the ribbon). To do it, select the *Customize* tab > *Workspaces* and select *Set current* from the context menu.

To change between workspaces, you can also use the  Workspace Switching icon at the bottom right corner of the screen.

2. CONFIGURATION

2.1. Job preferences

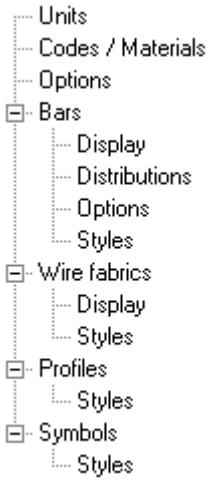
2.1.1. Job preferences

The option allows adopting basic parameters used in ***AutoCAD Structural Detailing***. The option is available:

- from the menu by selecting the *Reinforcement* / *Job preferences* option
- by pressing the *Job preferences* icon
- from the command line: RBCR_JOB_PREF.

The ***Job preferences*** dialog box can be split into two main parts:

- the left part of the dialog box contains a selection tree (see the drawing below) from which the user selects - by means of the mouse - one of the options of the program job preferences.



- to the right of the selection tree there is a part of the dialog box including parameters appropriate for an option selected by the user from the selection tree; the dialog box is updated after selecting an option by the user.

The right part of the dialog box provides the standard buttons (**OK**, **Cancel**, **Help**) and the following ones:

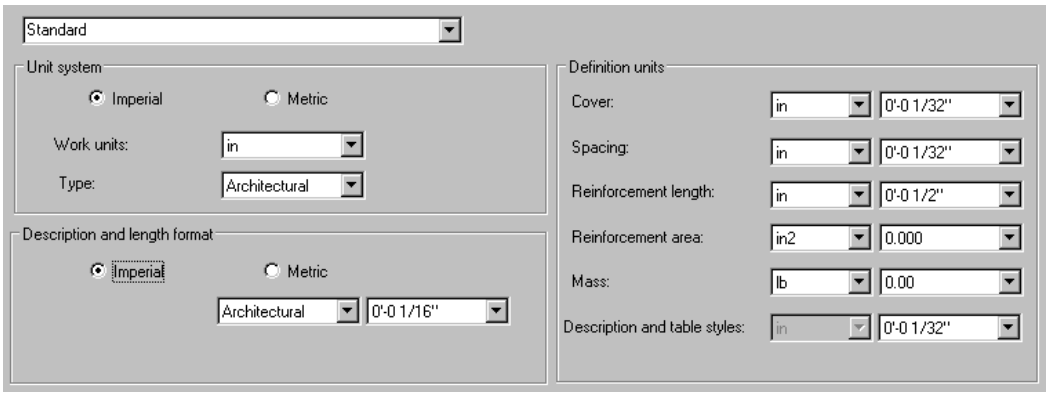
Default - pressing this button saves values of the job preferences parameters as the default values

Save - pressing this button saves the current status of preference parameters under a name located in the dialog box on the *Units* tab

Delete - pressing this button deletes the set of job preferences saved under the currently-presented name, located in the dialog box on the *Units* tab.

2.1.2. Units

After selecting the *Units* option from the selection tree located in the left part of the Job Preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.



The top part of the dialog box contains the field for selection of the preference option set.

Below, the user may determine the work units in ***AutoCAD Structural Detailing***. Contours of RC structure elements, bar lengths, etc. will be defined in the selected units on the screen.

You can select one of the following unit systems:

- imperial
  type:
  - architectural (0'-0)
  - engineering (0'-0'')
- metric.

There is also the *Description and length format* option provided in the dialog box; it is used to parameterize the length unit. The option is applied only for dimensions and description of reinforcement length (reinforcement presented outside the drawing contour). You can select imperial or metric system for description. If the imperial format is selected, choose the engineering or architectural type. For metric format, specify how to present length values less than 1m and length values over 1m. For example, if centimeters are selected in the format *for < 1.0 m* field, then the dimension 0.33 m will be displayed as 33 cm.

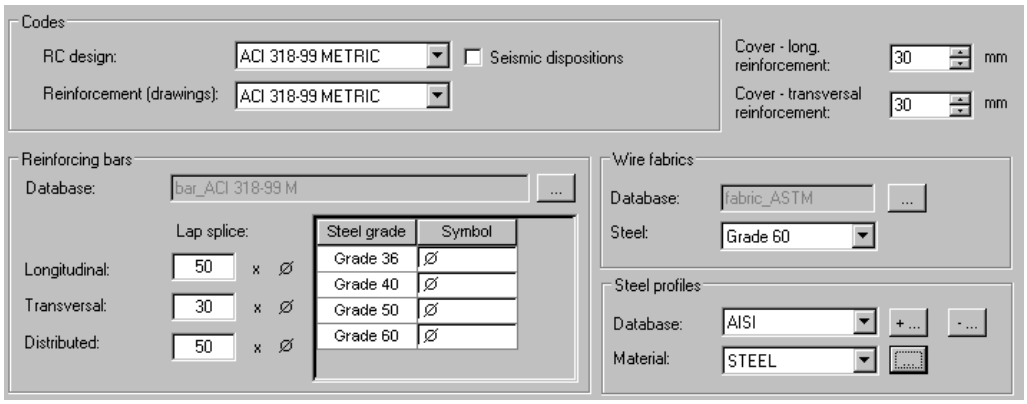
In the bottom part of the dialog box the user may select units applied in ***AutoCAD Structural Detailing - Reinforcement*** dialog boxes. Units have been divided into the following categories:

- length unit, including the following components: cover, spacing, reinforcement length, formwork dimensions, etc.
- unit of reinforcement diameter
- unit of reinforcement area
- mass unit
- number of decimal places for description and table styles.

Units are selected from the drop-down list available for each of the categories. For all the units it is possible to change the manner of presenting the format of numbers of the quantities listed. These fields enable defining the number of decimal places for each of the quantities. To change the number of decimal places, select the relevant item on the 2nd selection list to the right of the unit. The unit precision will be reflected in descriptions of reinforcement, dimensions, etc. Take note that reinforcement tables have their own precision settings.

2.1.3. Codes / Materials

After selecting the *Codes / Materials* option from the selection tree located in the left part of the Job Preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.



NOTE:
Materials are read in automatically after selecting an RC code.

In the top part of the dialog box, in the *Codes* field the user may select codes used in ***AutoCAD Structural Detailing***: code for RC structure design and drawing code (drawings of reinforcement). A selected drawing code imposes the appropriate symbols of designation, hatching, etc. valid in a given country whose code is currently in use. The following codes are available in the current version of the program:

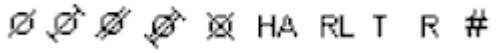
- RC codes:
  - American code ACI 318-08/M
  - British code BS 8110
  - Eurocode 2
  - Eurocode 2 (Italian NAD)
  - French code BAEL 99
  - Belgian code NBN B 15-002
  - Polish codes: PN-84/B-03264 and PN-B-03264:1999
  - Romanian code STAS 10107/0-90
  - Russian code SNiP 2.03.01-84
  - Ukrainian code DSTU 3760-98
  - Spanish code EHE 98
  - Norwegian code NS 3473E: 1999
  - Italian code DM 9/96
  - South African code SABS 82: 1997
  - German code DIN 1045
  - Swedish code BBK 04
  - Danish code DS 411
  - Austrian code ONORM B 4700
  - Indian code IS 456: 2000
  - Singaporean code CP65

- drawing codes: (corresponding to appropriate RC code):
  - American code ACI 318-08/M
  - British code BS 8666: 2000
  - Eurocode 2
  - Eurocode 2 (Italian NAD).

- French code NF P 02-016
- Belgian code NBN B 15-002
- Polish code PN-ISO 4066:1194
- Romanian code STAS
- Russian code GOST 21.501-93
- Spanish code EHE 98
- Norwegian code NS 3473
- Italian code DM 9/96
- South African code SABS 82: 1997
- German code DIN 1045
- Swedish code BBK 04
- Singaporean code CP65.

The *Codes* field also contains the *Seismic dispositions* option. It has effect on length of reinforcing bar hooks and length of lap splices in longitudinal bars; in the reinforcing bar databases, for French and Romanian codes there are additional columns containing hook lengths available if seismic dispositions are active. If the *Seismic dispositions* option is switched on in the above dialog box, then hook lengths are taken from this additional column in which seismic effects are considered (if seismic dispositions are active, hook lengths are increased approximately twice depending on the code). Values of lap splices are increased by 30% in relation to those defined in the preferences.

In the *Reinforcing bars* field the user may select steel classes for the relevant reinforcement type from the selected database of reinforcing bars; a steel class may be ascribed a symbol corresponding to it. The drop-down lists contain steel symbols (they depend on a selected code).
Steel classes available on the selection lists correspond to the chosen database of reinforcing bars; to change the reinforcing bar database (the field in which the file name is presented in inaccessible), the user should press the (**...**) button located next to the *Database* field and in the ***Open*** dialog box indicate appropriate database file (*.mdb). It results in adapting the reinforcement parameters to the user needs.
On the unfolding list graphical symbols will be displayed; the user will be able to enter any character string from the keyboard.
Below the list of standard steel designations is presented:



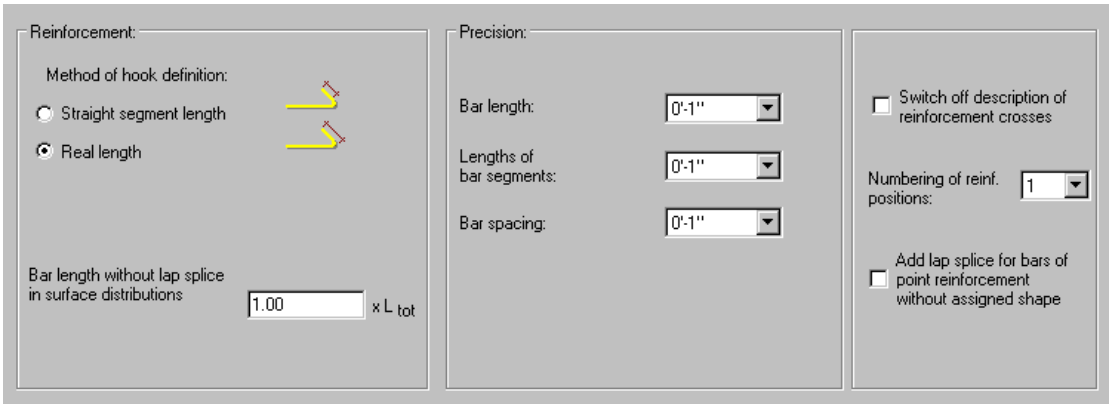
.

Apart from that, for reinforcing bars the user may select a lap splice for longitudinal, transversal and distributed reinforcement; a lap splice length is adopted as a multiple of a reinforcing bar diameter.
The *Wire fabrics* field enables selection of steel classes from the chosen wire fabric database. Steel classes available on the selection lists correspond to the selected wire fabric database; to change the wire fabric database (the field in which the file name is presented in inaccessible), the user should press the (**...**) button located next to the *Database* field and in the ***Open*** dialog box indicate appropriate database file (*.mdb). It results in adapting the wire fabric parameters to the user needs.
In the upper part of the dialog box a value of a reinforcement cover may be specified: separately for longitudinal bars (reinforcement – elevation), and separately for transversal bars (reinforcement – section). A cover value for longitudinal reinforcement given in the edit fields is used while defining reinforcement - elevation and special reinforcement, whereas a value of the transversal reinforcement cover is used when defining reinforcement - section.
The *Steel profiles* field allows selection of steel profile databases. To add a new profile database to the list of active profile databases, the user should press the '**+...**' button. It opens the dialog box where a steel profile database should be selected. A profile database can be deleted from the list of active profile databases; to do it, the user should select a profile database from the list and press the '**-...**' button.

The *Steel profiles* field also allows selection of materials for steel profiles. Materials found on the selection list correspond to selected materials from a material database. To add a material to the list, the user should press the (**...**) button located next to the *Material* field and select an appropriate material in the ***Material database*** dialog box.

2.1.4. Options

After choosing the *Options* option from the selection tree in the left part of the Job preferences dialog box, the options as shown in the drawing below appear in the right part of the dialog box.



In the top right part of the dialog box is the option: *Switch off description of reinforcement crosses*. If it is activated, values loaded with reinforcement crosses for slabs will not be presented on the screen (drawing).
Use the *Numbering of reinforcement positions* option to specify the way numbers of reinforcement positions are presented (it is particularly important for the British RC code); you can present the numbering as 1, 2, 3, ... or 01, 02, 03, ...

Below is the *Add lap splice for bars of point reinforcement without assigned shape* option. If selected, lap splices are added automatically for point reinforcement bars that are not assigned a shape (only their length is defined) and whose length exceeds the commercial length of reinforcing bars (12 m, for example). it is presented in the reinforcement table. Lap splices for bars are generated based on the options for lap splices of reinforcing bars on the *Codes* > *Materials* tab in the ***Job Preferences*** dialog.

Use the *Reinforcement* field to specify the method of defining a hook length:

-  as a length of the straight segment of a hook
-  as a real length of a hook.

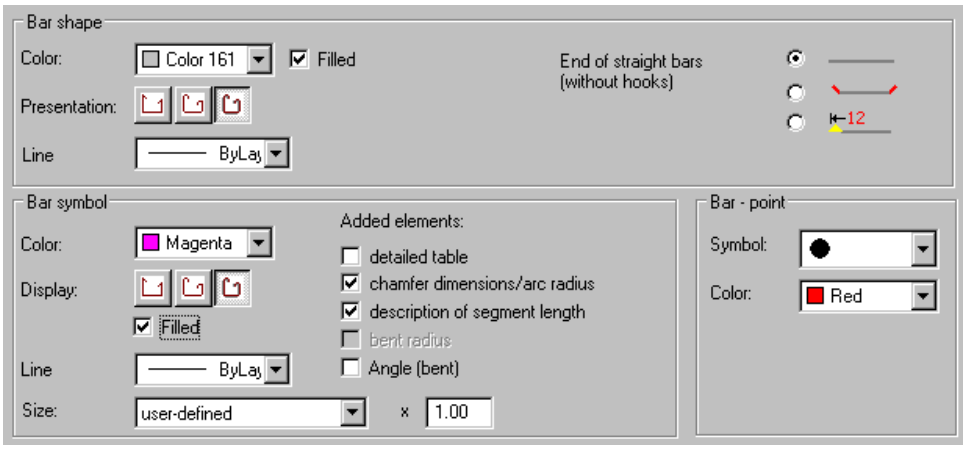
Use the *Bar length without lap splice in surface distributions* field to specify the length of bars for which reinforcement lap splices will not be used in reinforcement distribution (see: Distribution/definition of reinforcement - lap splices). Bars are distributed so that they fit the distribution region, but without taking account of the maximum length of bars (single bars in the distribution can be longer than Lmax) and without laps.

Use the lists in the *Precision* field to select precision for:

- length of reinforcing bars
- length of reinforcing bar segments
- bar spacing.

2.1.5. Display (bars)

After selecting the *Bars / Display* option in the selection tree located in the left part of the Job preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.

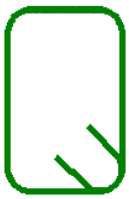


The options included in the *Bar shape* field:

- *Color* - selects color used to draw reinforcement; thickness of reinforcing bars will always be drawn in proportion to their diameter
- if the *Filled* option is switched on, the reinforcing bar contour that is being drawn will be filled in completely with a selected color.

The *End of straight bars* option enables the user to set the manner of presenting bar ends in a drawing (presentation: without ends, with ends, with ends and description); the option pertains only to straight bars without hooks.
Below the shapes of reinforcing bars are presented:

- contour - filled



- contour - not filled

The user may also choose a color and thickness of lines with which a bar shape is drawn. Three icons allow determining the method of presenting reinforcement:

-  first of them provides rough (schematic) reinforcement presentation - in the form of a broken line
-  the second one presents reinforcement together with bend curvatures
-  the third one shows reinforcement presenting its real diameter and real dimensions.

The line reflecting simplified shapes is a bar axis (with arc elements) or the outer line of a bar (without arc elements), while its location with respect to the formwork (an RC element contour) depends on a cover value.

NOTE:
The bar length is the same irrespective of the selected presentation method.

The options in the *Bar - point* field allow selecting the type of reinforcing bar presentation in a section (point reinforcement). The following symbols used to designate bars in a section are provided on the drop-down list:

Moreover, a color can be chosen for the indicated symbol.
The options from the *Bar symbol* field pertain to reinforcement whose description is provided outside the formwork contour.
The user may select color and line thickness to be applied while drawing reinforcement. Three icons allow determining the manner of presenting reinforcement:

-  - first of them provides rough (schematic) reinforcement presentation - in the form of a broken line (polyline)
-  - the second one presents reinforcement together with bend curvatures
-  - the third one shows reinforcement presenting its real diameter and real dimensions.

If the third option is selected, then also the *Filled* option becomes accessible which, when switched on, allows the user to fill in the reinforcing bar shape that is being drawn. In case of rough presentation and presentation showing bent curvatures, the list of line thickness selection is available.
This field also includes the *Added elements* option; this is a list of elements to be added to a bar whose description is provided outside the formwork contour:

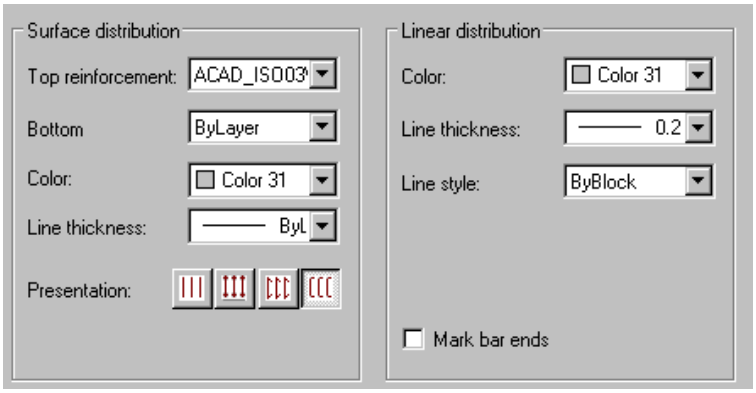
- *detailed table* - in the case of a bar (whose description is provided outside the contour), whose length is linearly variable (the result of the linearly-varying distribution), there appears a table containing a detailed list with a separate description of each bar; for a bar of constant length, there appears a table consisting of one line that contains description of bar dimensions
- *chamfer dimensions* / arc radius - (horizontal and vertical) dimension lines describing chamfered segments of reinforcement and arcs
- *description of segment length* - dimensions determining total length (with hooks included) of each bar segment
- *bent radius* - in some cases information about the size of radiuses of roller mandrels that form bents, is needed – the option is not available in the current program version
- *angle (bent)* – an angle between the neighboring bar segments is specified.

The options included in the drop-down *Size* list are used to determine the size of reinforcement symbols. The following sizes are available:

- *1 : 1* - it indicates that a symbol size equals the size of reinforcement in an element formwork
- *user-defined* - once this option is selected, the user needs to indicate (graphically) the contour in which the bar symbol is to be contained
- *scale factor* - once this option is selected, there appears an edit field in which the user may determine a scale factor that will decrease or increase the symbol with respect to the real size of a bar included in a formwork; for example, entering the coefficient value equal to 0.5 causes the drawing to be twice smaller, whereas entering the value 2 indicates that the drawing will be increased twice.

2.1.6. Distributions (bars)

After selecting the *Bars / Distributions* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



The options located in the *Surface distribution* field concern reinforcing bars belonging to surface distributions.

The *Surface distribution* field allows the user to:

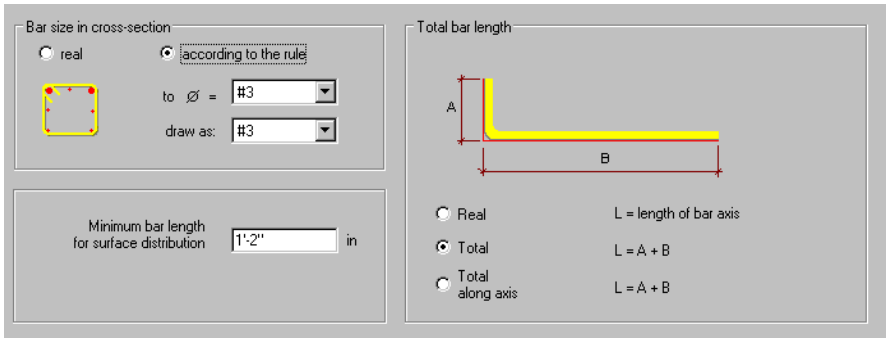
- determine a line style used for drawing the top / bottom reinforcement
- select a color and thickness of the line for distributed reinforcement
-  select (the *Presentation* option) the method of presenting the reinforcement (significant for bars with hook ends).

The options in the *Linear distribution* field concern the distribution of existing (with the shape already defined) reinforcement. The user may choose a color for presenting reinforcement in the linear distribution and line thicknesses and style.

If the *Mark bar ends* option is selected, ends of reinforcing bars in the linear distribution are marked according to the description style for ends of distributed bars.

2.1.7. Options (bars)

After selecting the *Bars / Options* option in the selection tree located in the left part of the Job preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.

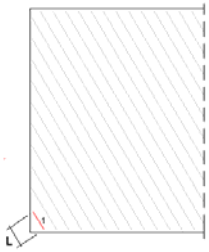


This dialog box contains options that have effect on the method of presenting the reinforcing bar in cross-section of an RC structure element:

Bar size in cross-section:

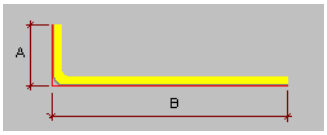
- *real* – if this option is selected, then the real size (in scale) of a bar in cross-section (reinforcing bar diameter) will be presented in a drawing
- *according to the rule* - if this option is selected, then the user may choose a size of the bar in cross-section, whose diameter is not greater than the reinforcing bar diameter selected by the user; for example, if the following size values are chosen: to ϕ = 18 draw as 25 mm, then all reinforcing bars in cross-section whose diameters are not greater than 18 mm will be presented as bars of 25 mm diameter.

Below is the *Minimum bar length for surface distribution* option; it allows the user to determine the minimal bar length; a bar whose length is less than this value will not be generated during definition of surface distribution. The example situation is illustrated in the drawing below. Bar no. 1 (presented in red in the drawing), whose length is less than the minimal bar length defined in the dialog box above, will not be generated (considered in tables).



The right part of the dialog box contains the *Total bar length* field; there are the following options:

Real – if selected, then in tables the reinforcing bar length will be determined as exact length of bar axis

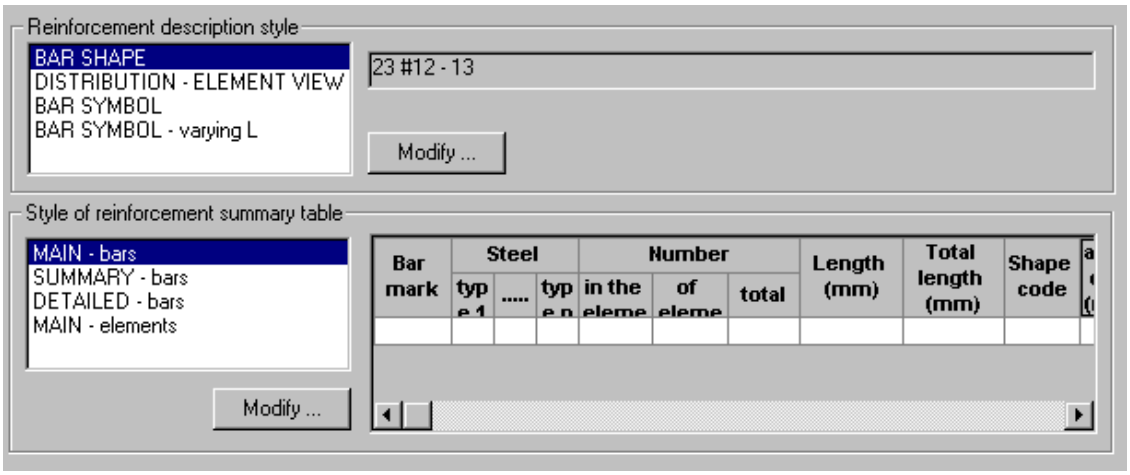


Total – if selected, then in tables the reinforcing bar length will be determined as the total of lengths of bar segments with hooks; bar lengths are given in compliance with total outer length

Total along axis - if selected, the length of a reinforcing bar is calculated - in reinforcement tables - as a sum of lengths of segments of the bar with hooks; lengths of bars along their axes are specified.

2.1.8. Styles (bars)

After selecting the *Bars / Styles* option in the selection tree located in the left part of the Job preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.



The dialog box provides basic information about the styles currently defined and relevant reinforcement or table to which these styles apply.
The top part of the dialog box includes styles of reinforcement descriptions with reinforcement division into categories considered. Once an appropriate reinforcement category is selected on the list (the category is highlighted), the program displays an example description based on the settings of the reinforcement description style. Pressing the **Modify** button opens the dialog box used for defining styles of reinforcement description; it enables direct modification of a selected type of reinforcement.
In the bottom part of the dialog box the table styles applied to prepare a reinforcement table, are presented. Once an appropriate table style is selected, an example reinforcement table ascribed to an element indicated on the list, is displayed. Pressing the **Modify** button opens the dialog box used for defining table styles.

2.1.9. Display (wire fabrics)

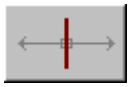
After selecting the *Wire fabrics / Display* option located in the selection tree in the left part of the Job preferences dialog box, the right part of the dialog box provides the options shown in the drawing below.

The options located in the *Wire fabric shape* field:

- *Color* – selection of a color that will be used to draw a wire fabric
- *Line thickness* – thickness of a line representing a wire fabric in a drawing
- if the *Filled* option is activated, it means that the wire fabric contour being drawn will be completely filled with a selected color.

The *End of straight bars* option allows the user to set the method of presenting bar ends in the drawing (presentation: without ends, with ends, with ends and description); the option concerns only straight bars without hooks.

Use the *Wire fabric distribution* field to specify the style of lines used to draw top / bottom reinforcement.
You can also select a type of wire mesh presentation in drawings (see reinforcement description styles – wire fabrics):

-  exact
-  group
-  simplified.

The options in the *Wire fabric symbol* field refer to reinforcement with a description placed outside the formwork contour.
Here it is possible to select a color and thickness of a line that will be used to draw a wire fabric. Three buttons enable the user to determine the method of reinforcement presentation:

- the first of them is responsible for schematic representation of a wire fabric - in the form of a broken line
- the second presents a wire fabric with bend curvatures
- the third presents a wire fabric with a real diameter and real dimensions.

If the third option is chosen, the *Filled* option is accessible then. Switching it on makes it possible to fill, the drawn shape of a wire fabric with a color. For the schematic presentation and the presentation including bend curvatures the list for selection of line thickness is available.
This field holds also the *Added elements* option; this a list of elements that will be appended to a wire fabric presented outside the formwork contour:

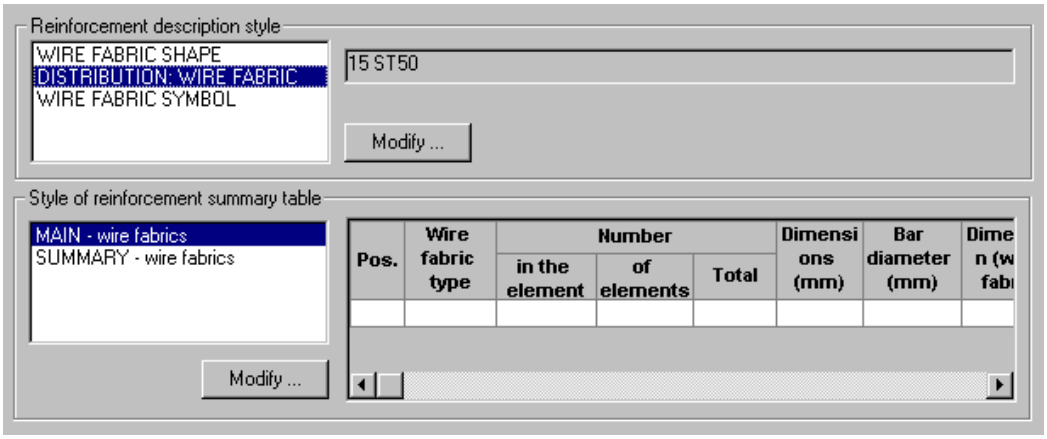
- *detailed table* – a table including a detailed description of a wire fabric is displayed
- *chamfer dimensions / arc radius* – dimension lines (horizontal and vertical) describing chamfered reinforcement segments and arcs
- *description of segment length* – dimensions determining total length of every wire fabric segment
- *bent radius* – in some cases it is necessary to provide information about the size of roller mandrels forming bends – the option is not accessible in the current version
- *angle (bent)* – an angle between the neighboring wire fabric segments is specified.

The options included in the drop-down *Size* list are used to determine the size of reinforcement symbols. The following sizes are available:

- *1 : 1* - it indicates that a symbol size equals the size of a wire fabric in an element formwork
- *user-defined* - once this option is selected, the user needs to indicate (graphically) the region in which the wire fabric symbol is to be contained
- *scale factor* - once this option is selected, there appears an edit field in which the user may determine a scale factor that will decrease or increase the symbol with respect to the real size of a wire fabric included in a formwork; for example, entering the factor value equal to 0.5 results in the drawing being twice smaller, whereas entering the value 2 - in the drawing being twice larger.

2.1.10. Styles (wire fabrics)

After selecting the *Wire fabrics / Styles* option located in the selection tree in the left part of the Job preferences dialog box, the right part of the dialog box provides the options shown in the drawing below.



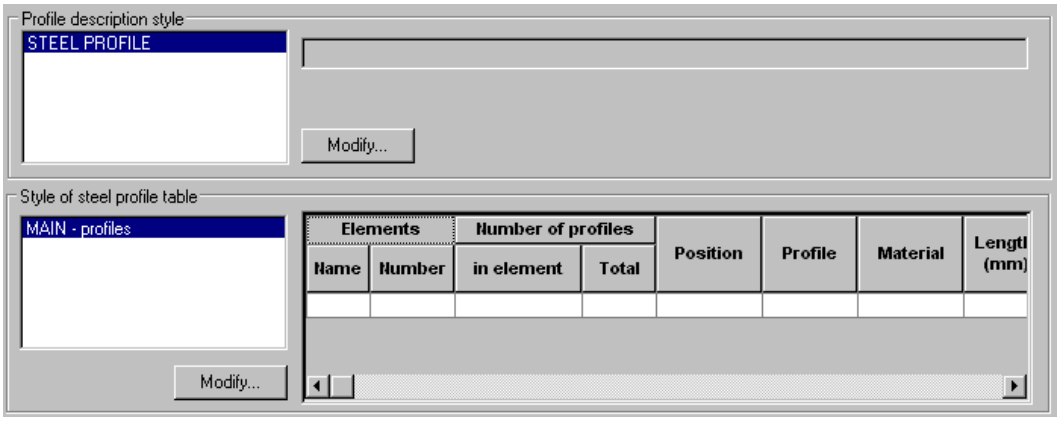
The dialog box provides basic information about the styles currently defined and the relevant wire fabric reinforcement or table where they belong.

The top part of the dialog box includes styles of reinforcement descriptions with division of wire fabric reinforcement into categories considered. Once an appropriate reinforcement category is selected on the list (the category is highlighted), the program displays an example description based on the settings of the reinforcement description style. Pressing the **Modify** button opens the dialog box used for defining styles of reinforcement description; it enables direct modification of a selected type of reinforcement.

In the bottom part of the dialog box, table styles applied to prepare a wire fabric reinforcement table, are presented. Once an appropriate table style is selected, an example reinforcement table ascribed to the element indicated on the list, is displayed. Pressing the **Modify** button opens the dialog box used for defining table styles.

2.1.11. Styles (steel profiles)

After selecting the *Profiles / Styles* option in the selection tree in the left part of the Job preferences dialog box, the right part of the dialog box includes the options shown in the drawing below.



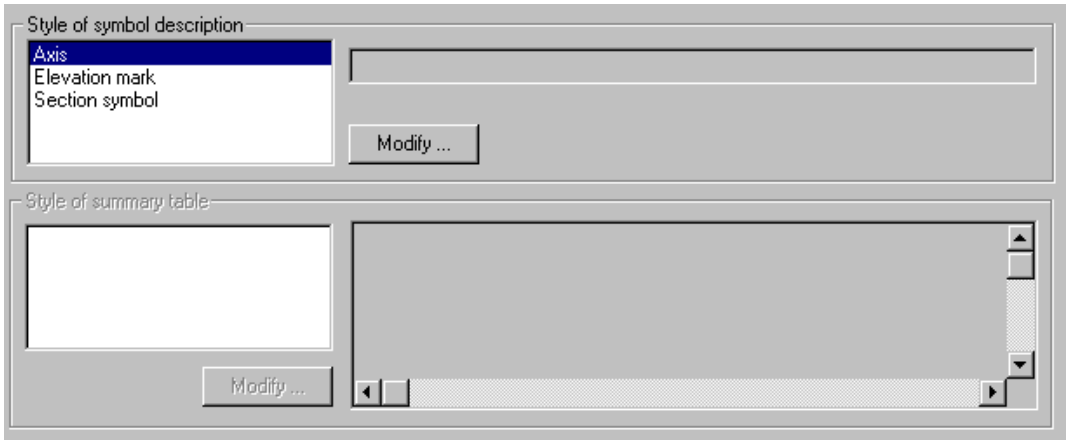
The dialog box provides basic information about the styles currently defined and the relevant reinforcement or table that these styles apply to.

The upper part of the dialog box includes styles of steel profile descriptions. Once an appropriate category is selected on the list (the category is highlighted), the program displays an example description based on the settings of a description style of a steel profile. Pressing the **Modify** button opens the dialog box used for defining styles of steel profile description; it enables direct modification of a selected steel profile.

The lower part of the dialog box presents table styles applied to prepare steel profile tables. After selecting an appropriate table style, the program displays an example of a steel profile table ascribed to the element indicated on the list. Pressing the **Modify** button opens the dialog box used for defining table styles.

2.1.12. Styles (symbols)

After selecting the *Symbols* / *Styles* option located in the selection tree in the left part of the Job preferences dialog box, the right part of the dialog box provides the options shown in the drawing below.



The dialog box holds basic information about the currently-defined styles of symbol description (axis, level and section).

The upper part of the dialog box includes description styles for the following symbols: axis symbol, elevation mark symbol and section symbol. After selecting an appropriate style on the list (e.g. the elevation mark symbol – the name is highlighted), the program displays a description example based on the settings of the description style. Pressing the **Modify** button opens the dialog box used for defining description styles for axes, elevation marks or sections.

In the bottom part of the dialog box styles of a summary table are presented. After selecting an appropriate table style, the program displays an example of a table assigned to an element indicated on the list. Pressing the **Modify** button opens the dialog box used for defining table styles.

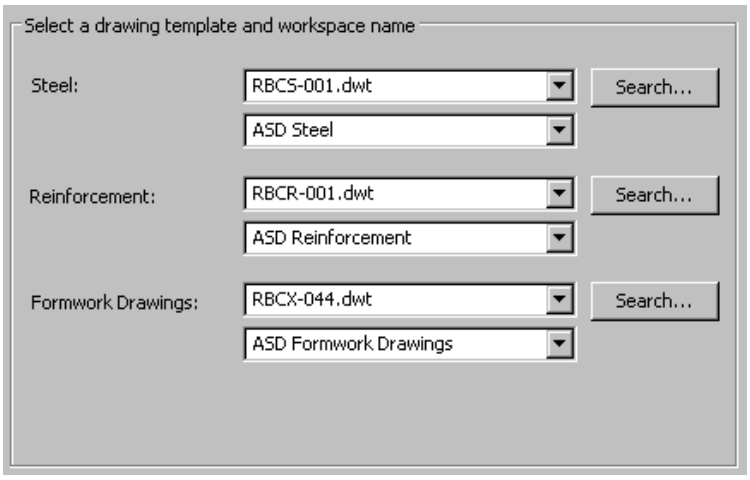
2.2. Preferences

2.2.1. Preferences

The option allows assuming basic parameters used in ***AutoCAD Structural Detailing***. The option is available:

- from the menu selecting the option *Reinforcement / Preferences*
- by pressing the *Preferences* icon

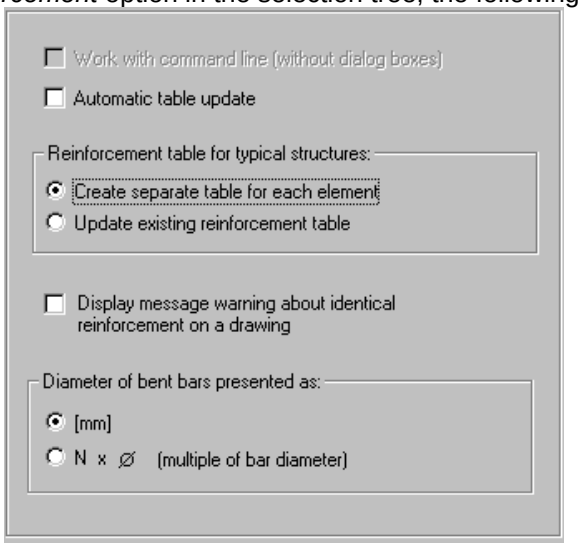
In the AutoCAD ® Options dialog, on the *Structural Detailing / General Settings* tab you can select a work template for ***AutoCAD Structural Detailing*** modules and workspace names (such as ASD or ASD Classic). Templates are located in the CFG folder and contain settings for a given country, for example the RBCR-001.dwt file is the template for the USA.



The *Structural Detailing* tab in the ***Options*** dialog box of the AutoCAD ® program can be divided into two main parts:

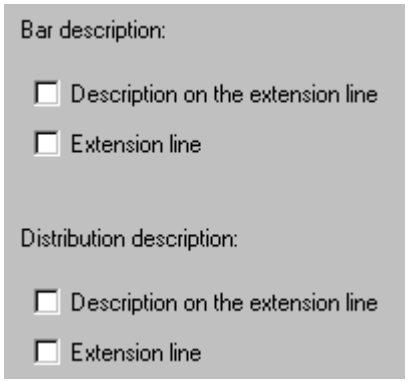
- in the left-hand part of the dialog box is a selection tree from which the user selects one of the program preference options using the mouse cursor
- the part of the dialog box to the right of the selection tree includes parameters corresponding to the option selected by the user from the selection tree; the dialog box is updated after selecting an option.

After selecting the *Reinforcement* option in the selection tree, the following dialog displays:



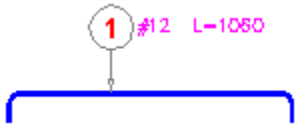
- *Automatic table update* - if this option is switched on, then while working in the program reinforcement tables will be updated automatically after changes are made in a drawing
- *Reinforcement table for typical structures* – enables the user to select the method of generating a table for typical structures (column, beam, spread footing, pile cap, pile, etc.); the table may be generated for every element of an RC structure separately (separately for a beam, separately for a column, etc.) or an existing reinforcement table may be updated after adding another element of an RC structure
- *Display message warning about identical reinforcement in a drawing* – if this option is switched on, then while working with the program, the user is informed about the identical reinforcement being present
- *Diameter of bent bars presented as* – enables the user to choose the method of presenting diameters of bent bars: they are expressed either in selected units (e.g. in mm) or as a multiple of a reinforcing bar diameter.

After selecting the *Reinforcement* > *Descriptions parametrization* option from the selection tree, the dialog includes options to set default settings for descriptions of bars and bar distributions:



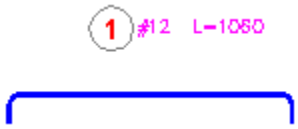
- bar description:

  the *Description on the extension line* option switched off - a bar description is inserted as shown in the drawing below

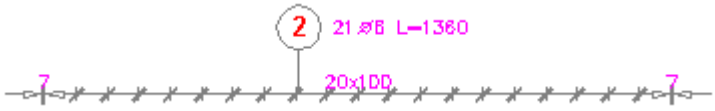


  the *Description on the extension line* option switched on - a bar description is inserted as shown in the drawing below

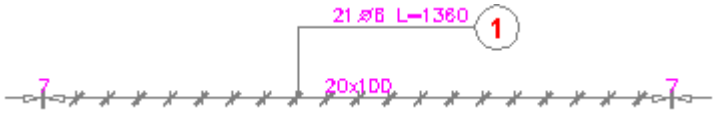
  the *Extension line* option switched off - a bar description is inserted as shown in the drawing below (without the line connecting the bar with the label of the position number)



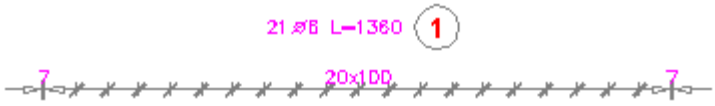
- distribution description:
  the *Description on the extension line* option switched off



the *Description on the extension line* option switched on



the *Extension line* option switched off



NOTE:
A change of default settings in the above dialog box has effect on the way new descriptions are inserted; descriptions which were defined earlier do not change.

AutoCAD Structural Detailing has the following language versions:

 USA

 UK

 Italy

 Russia.

 Poland

 France

 Spain

3. OBJECT INSPECTOR

3.1. Object Inspector Description

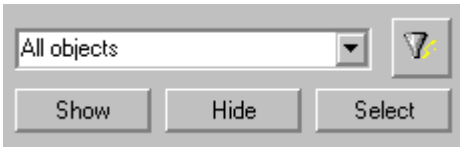
Inspector is a tool enabling management of elements (objects) included in a project created in ***AutoCAD Structural Detailing***. By standard, the ***Inspector*** dialog box is presented in the left-hand side of the program window, beside the viewer of graphic model definition. The width of the dialog box may be freely adjusted to leave as much space as possible for the field of graphic model definition.
The most important tasks carried out in the **Inspector** include:

- presentation of project contents
- presentation of generated views out of which printouts can be composed
- change of scale of the general view in which objects will be created
- change of view scales
- change of name
- deletion of views
- filtering of elements (objects) in drawings
- generation and management of the project drawing documentation.

The ***Object Inspector*** dialog box, shown in the drawing below, may be divided into three parts:

- options for object filtering
- four tabs containing lists (sets) of project elements depending on the design stage (modeling / positions / printouts)
- the table presenting properties of selected objects.

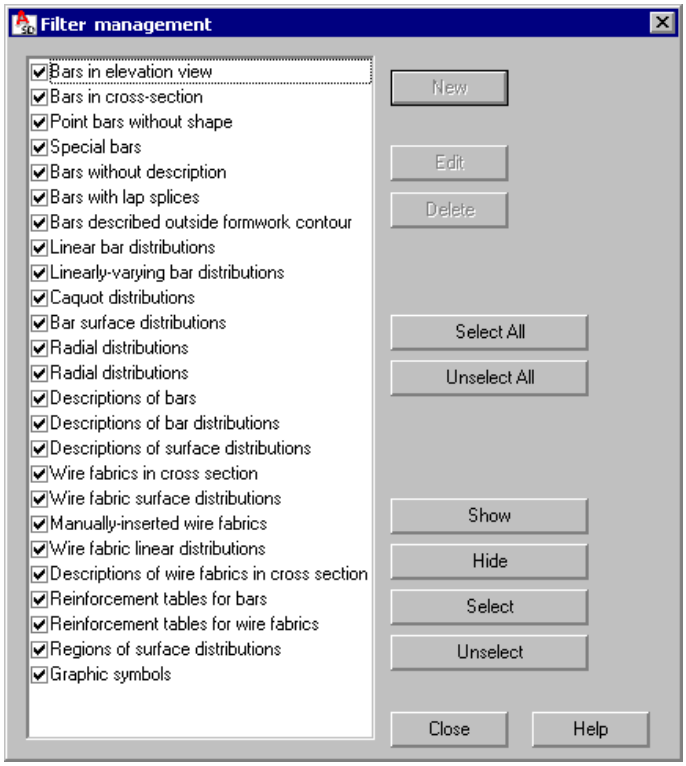
At the top of the dialog box there are options for filtering objects in drawings. A defined filter is identified by the name presented in the filter selection list in the ***Object Inspector*** dialog box.



The following buttons are provided under the selection list:

- **Show** - pressing this button displays selected elements (e.g. radial distributions or bar descriptions) in the drawing
- **Hide** - pressing this button hides selected elements (e.g. radial distributions or bar descriptions) in the drawing
- **Select** - pressing this button selects chosen elements (e.g. radial distributions or bar descriptions) in the drawing.

Pressing the  icon in the ***Object Inspector*** dialog box opens the ***Filter management*** dialog box shown in the drawing below.

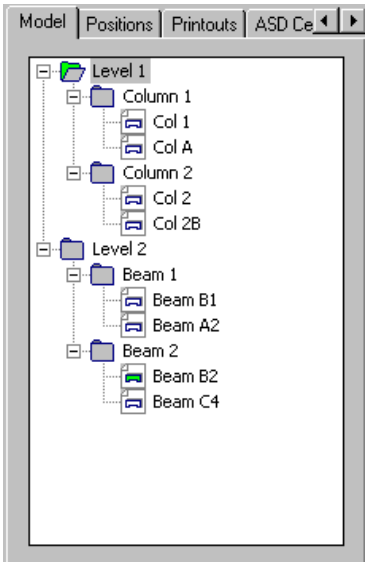


The dialog box above includes all defined filters and the following buttons:

- **Show, Hide, Select** - they work the same as in the ***Object Inspector*** dialog box (see the description above)
- **Unselect** - pressing this button switches off selection of chosen elements (e.g. radial distributions or bar descriptions) in the drawing.
- **Select All / Unselect All** - pressing this buttons selects / switches off selection of all elements (filters) provided on the list
- **New, Edit, Delete** - the options are not available in the current program version.

3.2. Model

This tab presents a list of defined elements (levels, groups of elements as well as elements belonging to levels and groups) that describe division of reinforcing bars into structural elements such as a beam, a column, a spread footing, etc.



Individual components included on the list presented on the *Model* tab may be freely composed: deleted, shifted between folders - using the mouse; all these operations will result in updating the data contained in the ***Element manager*** dialog box.
A structure of user-defined levels, groups and elements is shown in the form of a tree as illustrated in the drawing above.
The context menu available on the *Model* tab holds the following commands:

- for a level and a group:

1. *Add element* – enables adding a new element to a selected group (the ***New element*** dialog box opens on the screen then)
2. *Add group* (the option is accessible only for a level) – enables adding a new group to a selected level
3. *Steel table* – generates a reinforcement table with division into elements (the table works for multiselection)
4. *Delete* – deletes a level or a group; if a level containing components is deleted, then the components will remain (as if the level had never been created); if a group containing elements is deleted, then these elements can be assigned to a level or be left without being assigned.

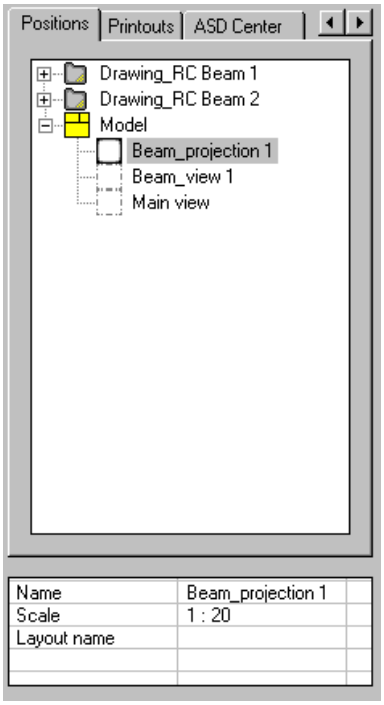
- for an element:

1. *Show element* – zooms in a drawing to show all the components of the drawing
2. *Steel table* - generates a reinforcement table with division into elements (the table works for multiselection)
3. *Add to element* – switches to the selection mode that enables choosing objects to be added to a selected element (once they are selected the ***Element manager*** dialog box opens on the screen)
4. *Delete* – deletes an element from the list (assignment of objects to an element is deleted).

In ***AutoCAD Structural Detailing - Reinforcement*** there is not any strictly determined order of creating 'elements'. Which operation is performed first in the program depends on the user's choice and habits; elements can be created after drawing reinforcement of all elements of a structure (all at a time), or created one by one while drawing.
If additional reinforcement needs to be added to an existing element, then it is necessary to use the context menu command *Add to element* available for the element.

3.3. Positions

This tab presents a list of defined views containing name, scale and name of the printout layout including a given view.



The icon of a document included on the list may be presented as follows:

- - it means that this document has been read from ***Autodesk Robot Structural Analysis***
- in yellow - it means that this document is active on the edition layout; moreover, there may exist views for a document, presented as:
  – active view
  – inactive view.

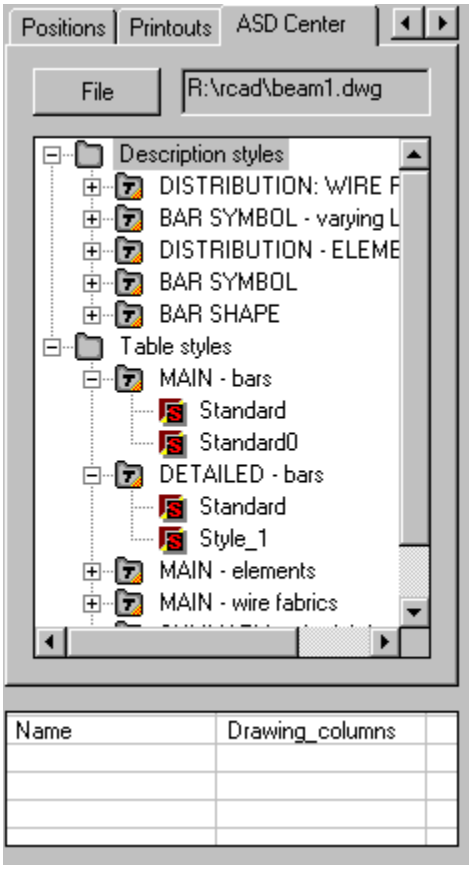
If the icon of a view provided on the list is shown in bold line, it means that it is an active view.

The context menu, that appears after pressing the right mouse button at the moment when the cursor is located on the *Positions* tab, contains several options which allow performing operations on selected positions:

- *Change name* - choosing this command enables changing the name of a highlighted view (the name is entered to the command line)
- *Activate* – choosing this command causes the highlighted view to become active (visible on the whole screen); it also means that the model takes over the properties of an active view (e.g. scale)
- *Delete view* - choosing this command enables deletion of the created view.

3.4. Structural Detailing Center

This tab enables copying settings (styles) between user's projects.



After pressing the **File** button and selecting a file with an earlier-saved project, the ***Inspector*** dialog box shows all the defined styles that may be used in the current project.
After highlighting a selected style, pressing the right mouse button and choosing the *Add* command, the selected style is added to the styles available in the current project.

3.5. Printouts

This tab enables management of printouts in ***AutoCAD Structural Detailing***; it presents the list of all printouts defined in the ***AutoCAD Structural Detailing*** project.

Printouts are presented together with a set of views. The printout list contains all the printouts, even those which do not include any views. The structure of user-defined printouts and views is shown in a form of a tree. Due to logical reasons, views are placed in a printout, however, for the user's convenience, the tree also includes an intermediate level, so that it is obvious to which document given views belong. If the printout layout is active, then the icon of a printout corresponding to the active printout layout is presented in red color.

Printouts provided on the list may be selected by indicating them with the mouse cursor (take note that only elements of one printout may be selected at a time - it is impossible to select elements of two different printouts). Selection of the printout in the dialog box is synchronized with the graphic editor - an appropriate drawing is displayed on the screen.

The context menu, that appears after pressing the right mouse button at the moment when the cursor is located on the *Printouts* tab, contains several options which allow performing operations on selected printouts:

- *Change name* - choosing this command enables changing the name of a highlighted printout (the name is entered to the command line)
- *Delete* - choosing this command results in removal of selected printouts from a project
- *Activate* - choosing this command causes the highlighted printout to become active (visible)
- *Unload printout* - choosing this command causes the selected printout to be excluded from the list of available printouts
- *Save printout* - choosing this command enables saving the selected printout as a *.dwg file
- *Add printout* - selecting this command adds an empty printout to the project (the name is entered to the command line).

4. TYPICAL STRUCTURES

4.1. Reinforcement / formwork of typical RC structure elements

The option enables definition of typical RC structure elements and their reinforcement. The option is accessible:

- from the menu by selecting one of the options available in the submenu: *Reinforcement / Typical structures – reinforcement*
- by pressing the relevant icon on the *Typical structures – reinforcement* toolbar
- from the command line: RBCT_MACRO.

Typical structures are grouped into certain categories. When choosing a structure, the user should first find an appropriate category. The current version of ***AutoCAD Structural Detailing - Reinforcement*** offers access to the following databases (macros) used to define geometry / reinforcement of RC structure elements:

- Spread footing
- Sleeve footing
- Continuous footing
- Column
- Beam
- Opening
- Corner
- Slab corner
- Distribution of prefabricated slabs
- Stairs
- Pile cap (pile foundation)
- Pile
- Ground beam
- Parapet
- Retaining wall
- Additional connecting elements
- Linear element.

The macros listed are available from the menu (*Reinforcement / Typical structures - reinforcement)* and the *Typical structures - reinforcement* toolbar.

* Spread footing - 
* Sleeve footing - 
* Continuous footing - 
* Column - 
* Beam - 
* Opening - 
* Corner - 
* Slab corner - 
* Distribution of prefabricated slabs - 
* Stairs - 
* Pile cap (pile foundation) - 
* Pile - 
* Ground beam - 
* Parapet - 
* Retaining wall - 

* Linear element consisting of two options:
   Create linear element – the option enables definition of an RC structure element (section of an RC element) that will be assigned element length; such an element may be saved to the database
   Insert linear element - the option enables insertion of an RC structure element (section of an RC element) that has been assigned length; after selecting a linear element saved earlier on the disk, a block of drawing elements is placed in a drawing – a selected linear element is assigned name and length.

The enumerated categories of typical structures may be configured quite freely within a given model. After selecting a category, an additional dialog box is displayed on the screen in which the user may determine parameters of a selected element of an RC structure. The shape of this dialog box depends on a structure category selected.

Moreover, in ***AutoCAD Structural Detailing – Reinforcement,*** in the *Formworks* toolbar as well as in the (*Reinforcement / Formworks*) menu there are the following macros for formworks of RC structure elements available:

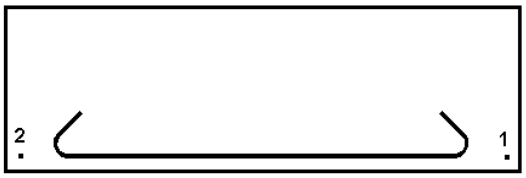
- Spread footing - 
- Sleeve footing - 
- Continuous footing - 
- Column - 
- Beam - 
- Stairs - 
- Pile cap (pile foundation) - 
- Ground beam - 
- Retaining wall - 
- Parapet - 
- Additional connecting elements -  .

5. RULES APPLIED WHILE DEFINING REINFORCEMENT

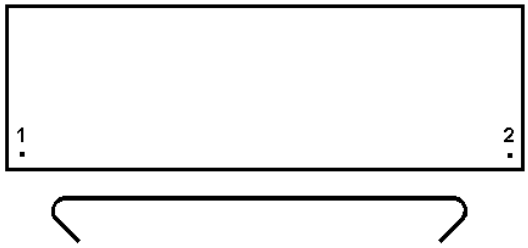
5.1. Location of a reinforcing bar in a drawing

The following principle holds in the program while determining the position of a reinforcing bar in a drawing:
Bar location depends on a direction of point definition. The principle that holds when defining a reinforcing bar consists in determining the order of points clockwise (along the EXTERNAL part of an object). It means that the defined bar will always be positioned in the inner part of an object. In the case of a bar with hooks of a bending angle greater than zero degrees, the hooks will be located on the opposite side with respect to the side where points defining the bar length are placed, thus they will be turned towards the middle of an object.
The above principle is illustrated schematically in the drawings below.

Bar defined 'from the right to the left':
1 - **beginning of a reinforcing bar**
2 - **end of a reinforcing bar**



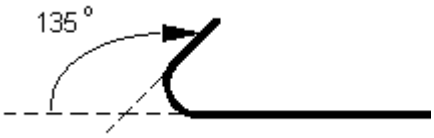
Bar defined 'from the left to the right':
1 - **beginning of a reinforcing bar**
2 - **end of a reinforcing bar**



5.2. Angle of hook bending

The following principle holds in the program while determining a value of the angle of hook bending:
The angle of hook bending is an angle by which a reinforcing angle should be bent to obtain a hook.

The above principle is presented in the drawing below (the angle of hook bending equals 135 degrees).



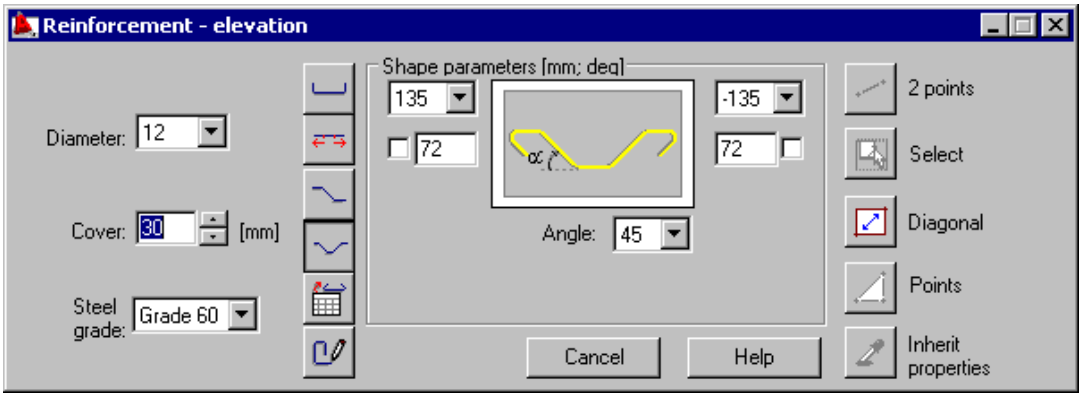
6. DEFINITION OF REINFORCEMENT - LONGITUDINAL REINFORCEMENT

6.1. DEFINITION OF BAR REINFORCEMENT - bar elevation (longitudinal reinforcement)

The option enables definition of reinforcing bars (longitudinal reinforcement) in an element of RC structure. The option is available from:

- the menu by selecting the *Reinforcement / Reinforcement - elevation* option
- the toolbar by pressing the icon
- the command line: RBCR_DEF_BAR_BV.

Once the *Reinforcement - elevation* option is activated, first the dialog box used to select a shape of longitudinal reinforcement appears on the screen (NOTE: while reinforcement is being defined this dialog box remains visible).



The ***Reinforcement - elevation*** dialog box can be split into three parts:

- the left part of the dialog box contains basic information concerning reinforcement: diameter, cover and steel grade adopted from preference settings
- in the middle part of the dialog box the icons are provided that symbolize basic shapes of longitudinal reinforcement and allow selection of the reinforcement type; once a longitudinal reinforcement type is selected, contents of the field with parameters of a reinforcement shape changes
- the right part of the dialog box includes several icons which are used to select the mode of graphical definition of reinforcement; if they are pressed, the mode of graphic interface is selected according to the specific manner of contour definition. NOTE: only after one of these icons is pressed, bars can be defined in the program graphical window.

The dialog box opens showing the longitudinal reinforcement type recently defined and parameters adopted for it.

The icon located in the bottom right corner is used to inherit (adopt) parameters from the reinforcement already defined. It is the standard tool provided in most dialog boxes.

The following types of longitudinal reinforcement are available within the program:

- straight bar
- straight bar with bar anchor generation
- bent bar - type 1

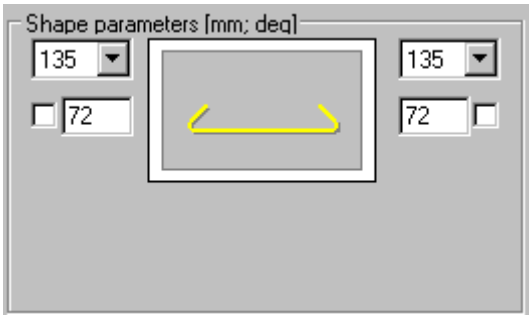
- bent bar - type 2
- bar from database
- bar of any shape.

Once definition of reinforcement shape is completed, the Reinforcement description dialog box appears in which the user may select elements of reinforcement description.

Reinforcing bars may also be defined by means of the commands available in the program.

6.2. Straight bar

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- cover of a reinforcing bar (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - parameters of bar ending with hooks, i.e., a hook angle and length (the hook length defined in the dialog box denotes a length of the straight segment of a hook or its real length - see *Options* in the ***Job preferences*** dialog box); you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The list of standard hook angle values is presented below (see: method of measuring the angle of bar bending):

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

While a straight bar is being defined, in the right part of the dialog box only the 2 Points icon is available (the remaining icons are unavailable), which when pressed, starts definition (by indicating on the graphical screen) of two points.

NOTE:
Bar location depends on the direction of point definition. The principle that applies while defining a reinforcing bar consists in determining the order of points clockwise (along the EXTERNAL part of an object). It means that the defined bar will always be positioned in the inner part of an object. In the case of bar with hooks with a bending angle greater than zero degrees, the hooks will always

be positioned on the side opposite to the side where points defining the bar length are placed, thus they will be turned towards the middle of an object.

After defining the first point determining the bar position, the bar length changes depending on the cursor position.

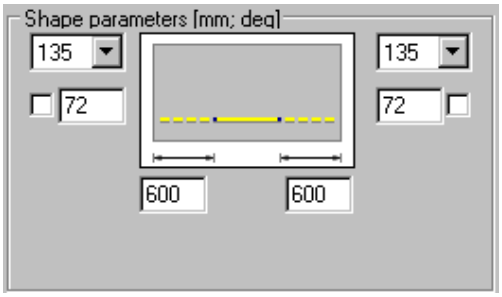
6.3. Definition of a straight bar (longitudinal reinforcement)

To define a straight bar, in the ***Reinforcement - elevation*** dialog box the user should:

- determine information concerning the reinforcing bar defined: diameter, cover as well as parameters of bar ending with hooks, in other words, hook angle and length
- press the *2 Points* icon included in the right part of the dialog box
- move to the graphical viewer and determine a position of the beginning and end bar points.

6.4. Straight bar with anchor element

Once the icon is pressed, the dialog box assumes the form shown in the figure below.

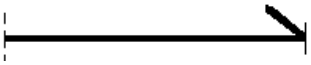


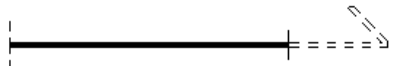
To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently for this reinforcement type is adopted by default)
- cover of reinforcing bars (the cover defined recently for this reinforcement type is adopted by default)
- in the *Parameters of reinforcing bar shape* field - parameters of anchors for reinforcing bar ends (anchor length or an anchor ended with a hook); you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

For an anchored bar, the program suggests by default, definition of an anchor for a straight segment of a bar (hook angle equals zero). The user may also define an anchor for a bar with hook of a specified bending angle. Thus three choices are available:

1. 
   anchor:
   $\alpha = 0$
   $l \neq 0$

2. 
   hook:
   $\alpha \neq 0$ (by default, the value of 135 degrees is adopted)
   $l = 0$

3.  anchor ended with a hook:
$\alpha \neq 0$
$l \neq 0$

The list of standard hook angle values is presented below (see: method of measuring the angle of bar bending):

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

While a straight bar is being defined, in the right part of the dialog box only the 2 Points icon is available (the remaining icons are unavailable), which when pressed, starts definition (by indicating on the graphical screen) of two points.

NOTE:
Bar location depends on the direction of point definition. The principle that applies while defining a reinforcing bar consists in determining the order of points clockwise (along the EXTERNAL part of an object). It means that the defined bar will always be positioned in the inner part of an object. In the case of bar with hooks with bending angle greater than zero degrees, the hooks will always be positioned on the side opposite to the side where points defining the bar length are placed, thus they will be turned towards the middle of an object.

After defining the first point determining the bar position, the bar length changes depending on the cursor position.

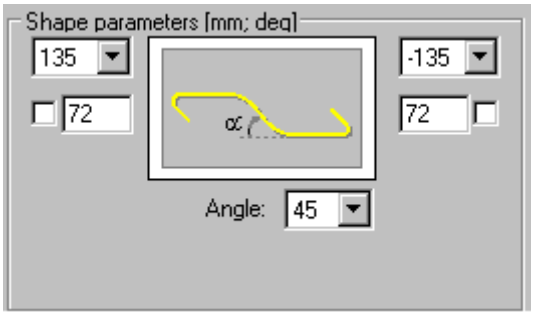
6.5. Definition of a straight bar with anchors (longitudinal reinforcement)

To define a straight bar with an anchor, in the ***Reinforcement - elevation*** dialog box the user should:

- determine information concerning the reinforcing bar defined: diameter, cover as well as parameters of bar ending with hooks, i.e., hook angle and length
- select the anchor type (only anchor or anchor and/or hook)
- press the *2 Points* icon included in the right part of the dialog box
- move to the graphical viewer and determine a position of the beginning and end bar points.

6.6. Bent bar - type 1

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently for this reinforcement type is adopted by default)
- cover of a reinforcing bar (the cover defined recently for this reinforcement type is adopted by default)
- in the *Parameters of reinforcing bar shape* field - parameters of anchors for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining a bar, it is important to determine the bar orientation. The default value of the inclination angle for a bent part in a longitudinal bar is 45 degrees. On the selection list other values of inclination angle are also available: 30, 45, 60 degrees.
Bar orientation (numbers shown in the drawing denote points defining a bar: **1** - **beginning of a reinforcing bar**, **2** - **end of a reinforcing bar**):

1. beginning point of a bar is located on the right side

2. beginning point of a bar is located on the left side

For a definition of a bent bar, only the *Diagonal* icon is available (the remaining icons are unavailable).
After defining a first point determining the bar position, bar length and height change depending on the cursor position. Once a second point is indicated, then the mode changes to definition of a position of the oblique reinforcement segment; the user should determine the position of the oblique branch point with respect to which the position of the oblique bar part will be determined. While the user is moving the cursor on the screen, the program displays information about the distance between the indicated point and the bar end.

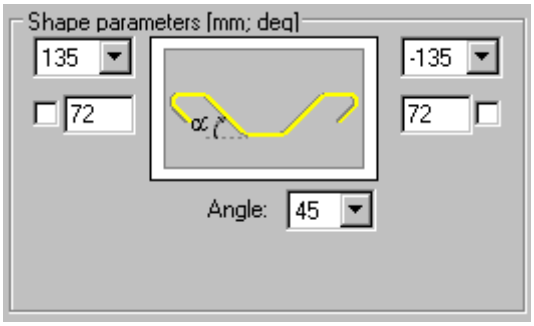
6.7. Definition of a bent bar (type 1) - longitudinal reinforcement

To define a bent bar, in the ***Reinforcement - elevation*** dialog box the user should:

- determine information concerning a reinforcing bar being defined: diameter, cover as well as parameters of a bent bar ending with hooks, i.e. hook angle and length at the beginning and end of a reinforcing bar
- determine the angle of the bar bending (the following values of a bending angle are available: 30, 45 and 60 degrees)
- press the *Diagonal* icon included in the right part of the dialog box
- move to the graphical viewer and determine the position of the beginning and end bar points
- determine length (position) of the bent part of a reinforcing bar.

6.8. Bent bar - type 2

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently for this reinforcement type is adopted by default)
- cover of a reinforcing bar (the cover defined recently for this reinforcement type is adopted by default)
- in the *Parameters of reinforcing bar shape* field - parameters of anchors for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining a bar it is important to determine the bar orientation. The default value of the inclination angle for a bent part in a longitudinal bar is 45 degrees. On the selection list other values of inclination angle are also available: 30, 45, 60 degrees.
Bar orientation (numbers shown in the drawing denote points defining a bar: **1** - **beginning of a reinforcing bar**, **2** - **end of a reinforcing bar**):

1. beginning point of a bar is located on the right side

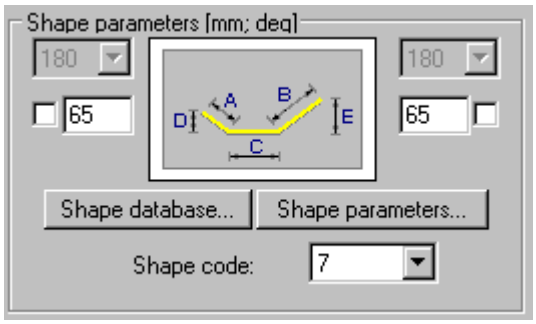
2. beginning point of a bar is located on the left side

For a definition of a bent bar only the *Diagonal* icon is available (the remaining icons are unavailable).
After defining a first point determining the bar position, bar length and height change depending on the cursor position. Once a second point is indicated, then the mode changes to definition of a position of the oblique reinforcement segment; the user should determine the position of the oblique branch point with respect to which the position of the oblique bar part will be determined. While the user is moving the cursor on the screen, the program displays information about the distance between the indicated point and the bar end.

The above operation is performed for a second oblique segment of reinforcement.

6.9. Bar from database

After pressing the icon, the dialog box assumes the form shown in the figure below.



NOTE:
Bar shapes are used to define a bar and to assign an appropriate identification code to it.

In the *Shape parameters* field the following options are provided:

- schematic drawing of the reinforcement shape
- edit fields where hook parameters are determined (hooks are ascribed permanently to the shape from database); you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.
- selection list containing codes - they affect bending diameters (French code)
- two buttons: **Shape database** and **Shape parameters**.

In the right part of the dialog box only the *Points* icon is available, which when pressed starts definition of a bar from database The remaining icons are unavailable.

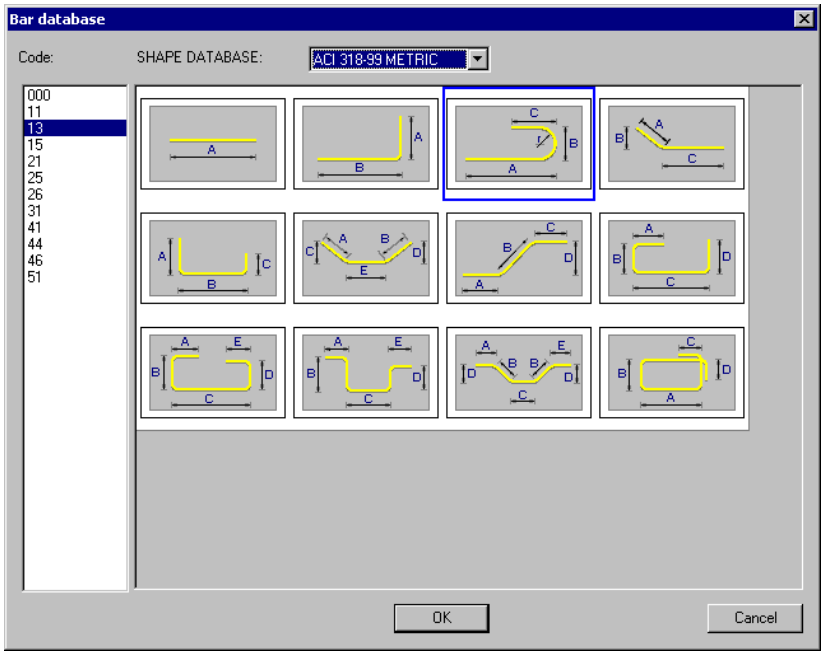
After pressing the **Shape database…** button, the program opens the ***Bar database*** dialog box.

NOTE:
The database of reinforcing bar shapes and corresponding shape codes depend on a selected code of RC structure design.

Structure of the shape database:

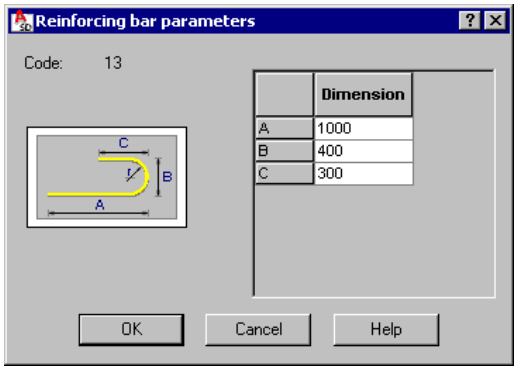
- a shape from the database may be selected both by indicating directly a schematic drawing from the list of available shapes or by selecting a shape code number

- the database contains shapes only, thus allowing the user to complete the fields with the code (the code will be displayed in the ***Shape definition*** dialog box)
- the database contains the field with reference to the active code
- the database comprises the basic shapes repeated which can be selected directly from the ***Shape definition*** dialog box.



Definition of bars from the database consists in selecting one of them from the available shape list, and afterwards, defining successive bar segments by indicating next bar characteristic points presented in a schematic drawing (the user may also enter dimensions of individual segments from the keyboard).
Once a bar is selected from the database, bar segments are defined one by one based on the indicated points (there apply rigid rules of defining successive segments so that the bar shape is maintained). While defining, the bar length is calculated automatically. The bar dimensions proposed by default in the dialog box are the dimensions of the recently-defined bar.
A bar defined in the bar shape database remembers the shape of a selected bar type. The user may replace such a bar with a regular bar which does not remember the bar geometrical shape - to do that the ***AutoCAD Structural Detailing*** EXPLODE option should be applied.

There is also another manner of bar definition. Once the **Shape parameters…** button is pressed, the additional dialog box appears on the screen in which (see the drawing below) a table containing dimensions of individual bar segments is available. After determining bar dimensions, a completed bar may be added to the formwork.

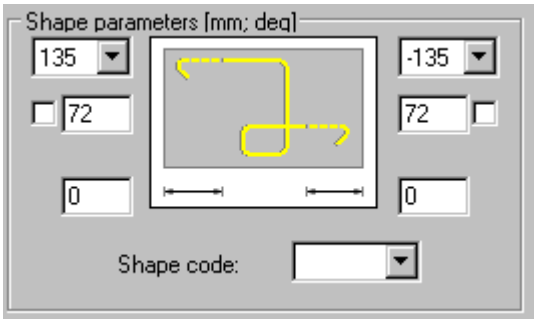


The above dialog box contains a table that displays dimensions of bar segments. If the dialog box is opened while a bar is being defined, then dimensions that are not defined yet will be assigned zero value. The user will be able then to change the bar segment lengths determined and to complete the dimensions missing.
The field with a shape code is filled out automatically depending on a typical shape selected.

Both modes of reinforcement definition are synchronized with each other. It means that both definition modes can be used alternately.

6.10. Arbitrary shape of a bar

After pressing the icon, the dialog box assumes the form shown in the figure below.



This dialog box allows defining any shape of reinforcement.
You can select a type of hook at the beginning and end of the defined reinforcement and lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.
Definition consists in indicating the successive characteristic points with the cursor. On these points the transversal reinforcement of any shape will be based. When defining a bar graphically, it is possible to change its location with respect to the insertion points (formwork), a value of the bar cover and a location of hooks.
The field with a shape code is filled out by the user (a unique code of the bar shape should be entered there). This code will be presented in the reinforcement table; it will be also included in the information about the bar.

While generating reinforcement, bent radiuses will be drawn automatically on the basis of the conditions determined in a selected code. This information is provided in the bar database.

The following icons are active for bar definition:

Points

Select - it enables changing indicated arcs and open polylines to a reinforcing bar.

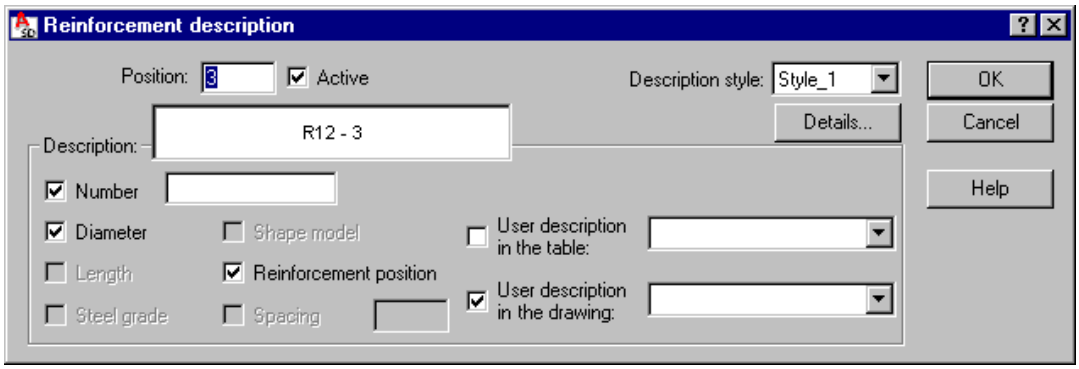
The remaining icons are unavailable.

6.11. Reinforcement description

Once a reinforcement shape is defined, the dialog box shown in the drawing below, used to describe reinforcement, appears automatically on the screen.

NOTE:

The dialog box below opens after selecting the Modify / Reinforcement description option provided in the menu or pressing the icon; in this case two additional options are available in the dialog box: Description style (used to modify a description style of the reinforcement chosen) and the ***Details*** *button, which when pressed opens the dialog box used for modifying a style of reinforcement description (shape).*



The options provided in this dialog box enable final selection of a reinforcement description. It can be performed by switching off active variables initialized based on the syntax defined.

REMARKS:

- a reinforcement shape can be described only for a single position, in other words, two or more bars cannot be described simultaneously
- the *Number* edit field allows the user to enter - from the keyboard - an ultimate number of reinforcing bars that will be used directly in the reinforcement summary tables. The amount of reinforcement specified during bar definition is a superior quantity with respect to the amount resulting from the subsequent distribution of this bar and that reinforcement amount is provided in the table; the number of bars may be represented as an equation, e.g. 2*(8+4);
- in the *Spacing* edit field the user may enter spacing values, despite reinforcement distribution not being defined yet. This field is editable on condition that the "%spa" variable is contained in the style of reinforcement description. It is only a static parameter that serves informative purposes and may be applied in reinforcement tables which include the spacing parameter. However, it should be remembered that if the user has entered a spacing value in this field, then regardless of real values of bar spacing in the structure element, the value provided in the dialog box will be assumed in the table.
- the *Active* option. If the option is turned on, it results in including the reinforcement being described in the reinforcement table. It means that for reinforcement that is described for the first time this option will be active. When the same reinforcement is described twice, the option will be switched off on its own. In this manner it is possible to prevent (when

describing the same reinforcement twice) the number of reinforcing bars calculated when preparing a bar table to be doubled; description of active and not active reinforcement may differ in a generated drawing – the options used for this purpose are provided in the Description of reinforcement shape dialog box.

- the fields: *User description in the drawing* and *User description in the table* are used to add any text to a reinforcement description (by typing it from the keyboard); the description will be presented correspondingly in drawings (included in bar descriptions in the drawing) and in the table (included only in the reinforcement table); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; a text taken from the library of standard descriptions may also be applied. These descriptions assume the style of the text describing the reinforcement. The user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are applied. A user description together with an extension line and label make up one object. Such an object may be edited (translation, rotation); to do that, first the ***AutoCAD Structural Detailing*** EXPLODE option should be used.
- a style of reinforcement description is chosen from the *Description style* selection list; the list contains all description styles defined for that type of reinforcement (the first on the selection list is a default description style chosen in the ***Description of reinforcement shape*** dialog box); before a reinforcement description is inserted in a drawing, parameters of the description style can be changed; pressing the **Details...** button opens the ***Reinforcement description*** dialog box where modifications of the style can be made (NOTE: modifications of the description style refer only to that one reinforcement description).


NOTE:
That whether any of the options listed above is turned on depends on the defined description syntax that is available in styles of reinforcement description. If for example, a bar symbol is to be included in the bar description, the user should open the ***Description of reinforcement shape*** *dialog box, select reinforcement description (e.g. Bar shape), press the* ***Modify*** *button, switch on the Reinforcement symbol option provided on the Description syntax tab, press the button with the arrow; the variable containing a bar symbol will be added in the Description edit field; to end the operation, the* ***Add*** *button should be pressed.*

6.12. User description - AutoCAD program mechanisms

While defining a user description, the user may apply formatting by introducing format codes, i.e. the mechanisms available in the AutoCAD ® program. To apply formatting, format codes presented in the table below should be used.

Format codes for paragraphs

Format code	*Purpose*
\0...\o	Turns overline on and off
\L...\l	Turns underline on and off
\~	Inserts a nonbreaking space
\\	Inserts a backslash
\{...\}	Inserts an opening and closing brace
\Cvalue;	Changes to the specified color
\File name;	Changes to the specified font file
\Hvalue;	Changes to the text height specified in drawing units
\Hvaluex;	Changes the text height to a multiple of the current text height
\S...^...;	Stacks the subsequent text at the \, # or ^ symbol
\Tvalue;	Adjusts the space between characters, from .75 to 4 times
\Qangle;	Changes obliquing angle
\Wvalue;	Changes width factor to produce wide text
\A	Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)
\P	Ends paragraph

Multiline text objects use word wrap to break long lines into paragraphs. For AutoCAD to break lines automatically and not to create a new paragraph, the line should end with either a backslash (\) or a space character.

6.13. Commands from the command line - definition of longitudinal reinforcement

The following parameters may be determined in the command line while defining a longitudinal bar:

Bar diameter or [Cover / Bar type / Define] <12>: 16

Select option [Diameter / Cover / Bar type / Define] <Define>:

Reinforcement diameter <12>:

Cover <5.5> : 6

Bar type: [Straight / Anchored / 1bent / 2bent / Base / Any] <Straight>:

For straight and anchored bar the following parameters may be defined:

Beginning point

End point or [Side]

where:

Side - it determines a change of a bar position with respect to the formwork line (with hooks included)

For bent bars the following parameters may be defined:

First corner

Second corner

Segment location or [Back]:

For bars from database the following parameters may be defined:

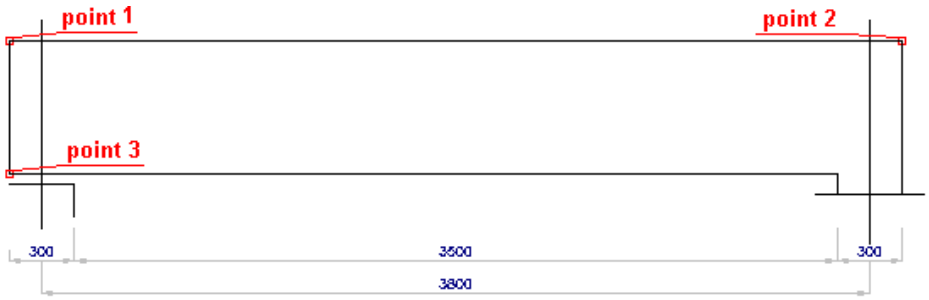
Bar code or [Select from database]

Beginning point
Next point or [Side / Cover / Mirror / Back]
where:
beginning point - first point determining shape of a bar (bar beginning)
next point - next points determining shape of a bar
side - determines on which side of a cross section contour of an RC element the current segment of reinforcement is to be located (change of a bar position with respect to the line - points of definition to the opposite one)
cover - value of a cover for the current segment of reinforcement
mirror - determines the mirror reflection of the current reinforcement segment
back - cancels the last command.

For bar of arbitrary shape the following parameters may be defined:
Beginning point
Next point or [Side / Cover / 1hook / 2hook / Back]
Cover <6>: 7
where:
beginning point - first point determining shape of a bar (bar beginning)
next point - next points determining shape of a bar
side - determines on which side of a cross section contour of an RC element the current segment of reinforcement is to be located
cover - value of a cover for the current segment of reinforcement
1 hook - direction of hook bending
2 hook - direction of hook bending (opposite to hook 1)
back - cancels the last command.

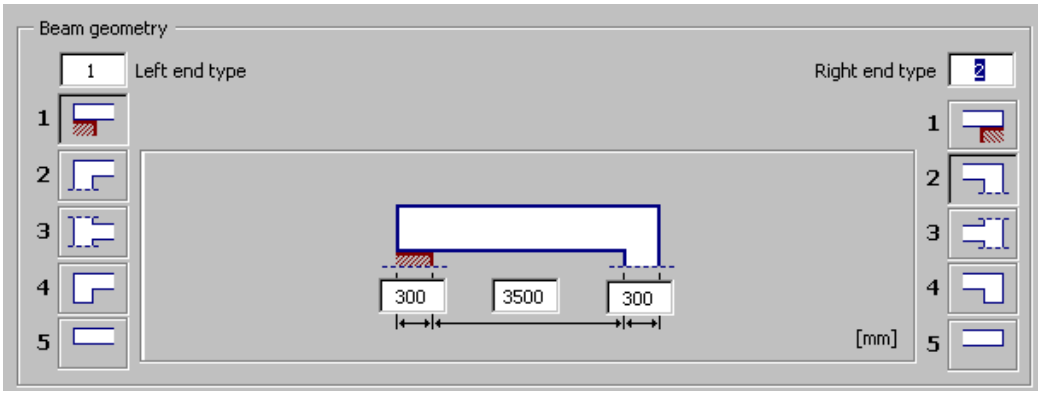
6.14. Example of definition of longitudinal reinforcement

The example below illustrates definition of longitudinal reinforcement of the beam presented in the following drawing. For definition of the beam contours the ***Formworks - Beam*** macro has been used.



To define longitudinal reinforcement of the RC beam, follow the steps below:
DEFINITION OF THE BEAM FORMWORK

- run the ***Formworks – Beam*** macro selecting the menu option: *Reinforcement / Formworks / Beam* or pressing the icon
- in the ***Formworks – Beam*** dialog box adopt the following parameters:
  - section type: 1 (rectangular)
  - beam type: 1 (single-span beam)
  - dimensions of the beam cross section: height = 600 mm, width = 300 mm
  - beam geometry as shown in the drawing below

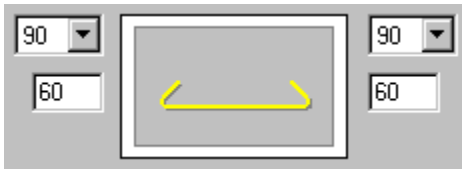


- press the **Insert** button in the ***Formworks – Beam*** dialog box and indicate the location of the beam formwork in the drawing

DEFINITION OF THE BEAM REINFORCEMENT

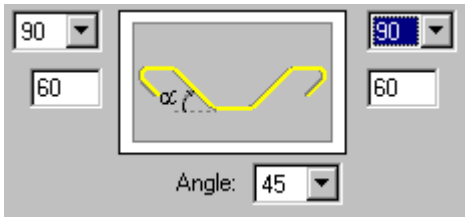
- run the menu option *Reinforcement / Reinforcement - elevation* or press the icon
- in the ***Reinforcement – elevation*** dialog box determine the following parameters:

  - type of reinforcing bars: straight bar
  - bar diameter: 12 mm; cover of reinforcing bars: 30 mm
  - steel grade: R
  - shape parameters as shown in the drawing below



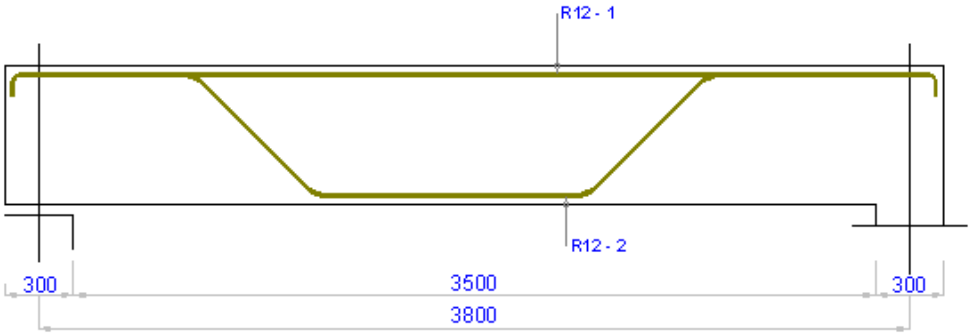
- press the *2 points* icon located in the right-hand side of the dialog box
- in the drawing of the beam formwork indicate point 1 and point 2 (see the drawing at the beginning of the example)
- accept the deafult reinforcement description proposed in the ***Reinforcement description*** dialog box by pressing the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu
- run again the menu option *Reinforcement / Reinforcement - elevation* or press the icon
- in the ***Reinforcement – elevation*** dialog box determine the following parameters:

  - type of reinforcing bars: bent bar
  - bar diameter: 12 mm; cover of reinforcing bars: 30 mm
  - steel grade: R
  - shape parameters as shown in the drawing below



- press the *Diagonal*  icon located in the right-hand side of the dialog box
- in the drawing of the beam formwork indicate point 2 – the first corner and point 3 – the second corner (see the drawing at the beginning of the example)
- in the drawing of the beam formwork indicate points that determine location of straight segments of the longitudinal reinforcement
- accept the deafult reinforcement description proposed in the ***Reinforcement description*** dialog box by pressing the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu.

The defined longitudinal reinforcement is illustrated below.



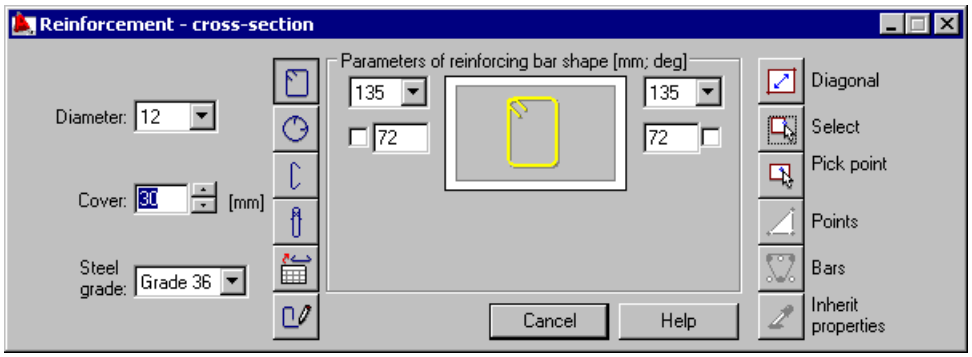
7. DEFINITION OF REINFORCEMENT - TRANSVERSAL REINFORCEMENT

7.1. DEFINITION OF BAR REINFORCEMENT - bar section (transversal reinforcement)

The option allows defining reinforcing bars (transversal reinforcement) in a cross section of an RC structure element. The option is available from:

- the menu by selecting the option *Reinforcement / Reinforcement - section*
- the toolbar by pressing the icon
- the command line: RBCR_DEF_BAR_BS.

Once the *Reinforcement - section* option is activated, first the dialog box used to select a shape of reinforcement in a cross section appears on the screen (NOTE: while reinforcement is being defined this dialog box remains visible all the time).



The ***Reinforcement - cross-section*** dialog box can be split into three parts:

- the left part of the dialog box contains basic information concerning reinforcement: diameter, cover and reinforcing steel grade adopted from preference settings
- in the middle part of the dialog box the icons are provided that symbolize basic shapes of transversal reinforcement and allow selection of the transversal reinforcement type; once transversal reinforcement type is selected, contents of the field with parameters of reinforcement shape changes
- the right part of the dialog box includes several icons which are used to select the mode of graphical definition of reinforcement; if they are pressed, the graphic interface mode is selected according to the specific manner of contour definition. NOTE: only after one of these icons is pressed, bars can be defined in the graphical window of the program.

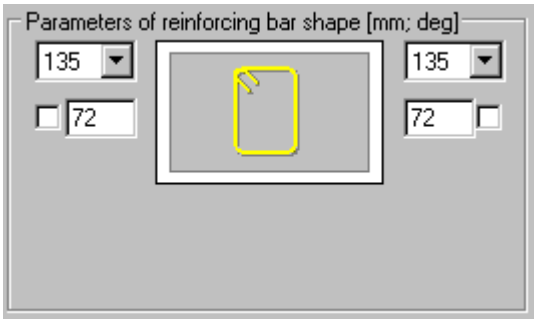
The dialog box opens showing the transversal reinforcement type recently defined and parameters adopted for it.
The following types of transversal reinforcement are available within the program:

- rectangular stirrup - closed
- round stirrup
- pin
- shackle
- bars from database
- any shape of a bar belonging to transversal reinforcement.

Once definition of reinforcement shape is completed, the Reinforcement description dialog box appears in which the user may select elements of reinforcement description.

7.2. Rectangular (closed) stirrup

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- cover of a reinforcing bar (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The shape of a bar presented in a schematic drawing is adjusted dynamically to the specified values of a hook bending angle. The list of standard hook angle values is presented below:

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

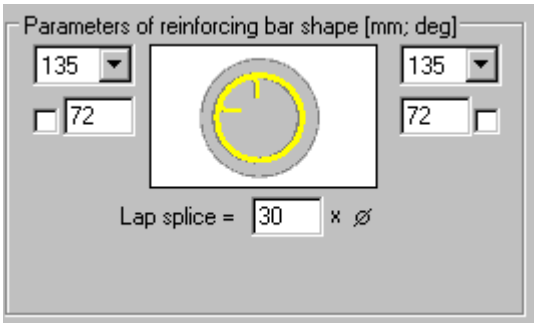
The following icons are available in the right part of the dialog box while a rectangular stirrup is defined (the remaining icons are unavailable):

- ***Select*** - is used to indicate directly the contour formed from a polyline. Once this button is selected, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' - phase of selecting the object to be copied). After selecting any point on the screen, the ACAD object is detected and transversal reinforcement (stirrup) is drawn within the indicated object.
- ***Diagonal*** - the function is used to create a rectangular contour by defining a diagonal. The dialog box closes and while the cursor is being moved a defined rectangular stirrup is presented dynamically.
- ***Pick point*** - function that allows searching a closed contour by clicking inside the contour. The dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'hatch'). After selecting any point on the screen, the minimum closed contour is detected. The detected contour is changed to a stirrup shaped like the detected contour, but decreased by the cover value.

The ***Select*** and ***Pick point*** options operate for all figure types.
Once the stirrup shape is defined initially, the program displays a question about the places where hooks are to be located. They may be selected by indicating a stirrup corner.

7.3. Round stirrup

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- cover of reinforcing bars (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for reinforcing bar ends, i.e. hook angle, length and lap splice length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The list of standard hook angle values is presented below:

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

The following icons are available in the right part of the dialog box while a round stirrup is being defined (the remaining icons are unavailable):

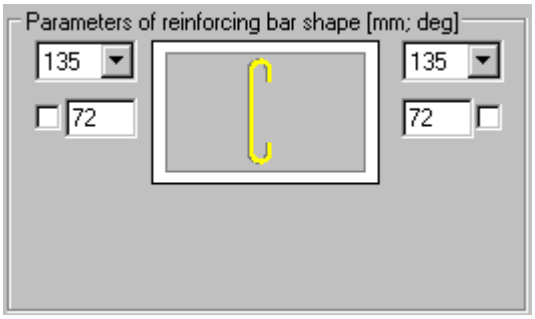
- ***Select*** - is used to indicate directly the contour formed from a circle or polyline. Once this button is selected, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' - the phase of selecting the object to be copied). After selecting any point on the screen, the ACAD object is detected and the stirrup is drawn within the indicated object.
- ***Pick point*** - function that allows searching a closed contour by clicking inside the contour. The dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'copy' - phase of indicating the beginning and end points). After selecting any point on the screen, the program detects the minimum concave round contour that results from inscribing it in the detected contour. NOTE: round stirrups may be defined by indicating the following geometrical figures: a regular polygon and a circle - other cases are not supported in the program

- *Points* - function that allows defining a closed contour by specifying a circle center and radius (or diameter).

The definition **ALWAYS** consists in creating a ROUND stirrup which is inscribed in the indicated contour.
Once the stirrup shape is defined initially, the program displays a question about the points where hooks are to be located.

7.4. Pin

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- cover of reinforcing bars (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The list of standard hook angle values is presented below:

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

The following icons are available in the right part of the dialog box while a pin is being defined (the remaining icons are unavailable):

- ***Points*** - the function is used to indicate directly two points located on a contour edge. After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'line'). After defining the first point determining a bar position, bar length changes dynamically on the screen depending on the cursor position. Bar location depends on a direction of point definition. The principle that holds when defining reinforcement consists in determining the order of points clockwise (along the EXTERNAL part of an object). It means that the defined bar will always be positioned in the inner part of an object. In the case of bar with hooks of a bending angle greater than zero degrees, the

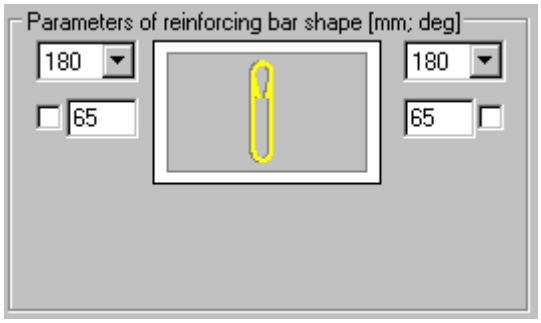
hooks will always be located on the side opposite to the side where points defining the bar length are placed, thus they will be turned towards the middle of an object.
Command line:
Side - d
First / second hook
Cover

- ***Bars*** - the function that allows bar definition on the existing point reinforcement (point reinforcement, i.e. presentation of longitudinal reinforcement in a cross section). After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'select obj.'). A bar definition is completed when another point reinforcement bar is indicated. In this case, a cover value is not considered.
  Command line:
  Side
  First / second hook.

A definition is **ALWAYS** completed at the moment a second point in the form of a point reinforcement bar, is indicated.

7.5. Shackle

After pressing the icon, the dialog box assumes the form shown in the figure below.



To define a bar, the following should be determined in the above dialog box:
- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- cover of reinforcing bars (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The list of standard hook angle values is presented below:
⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

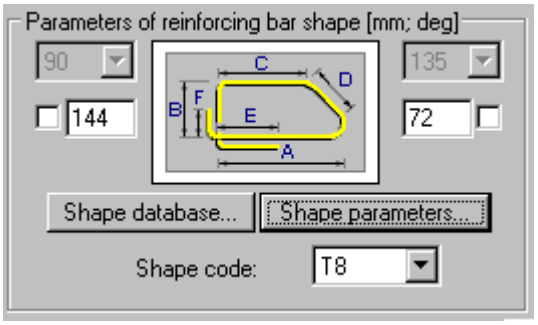
The following icons are available in the right part of the dialog box while a shackle is being defined (the remaining icons are unavailable):

- ***Points*** - the function is used to indicate directly two points located on a contour edge. After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'line'). After defining the first point determining the bar position, bar length changes dynamically on the screen depending on the cursor position.
  Bar location depends on a direction of point definition. The principle that holds when defining reinforcement consists in determining the order of points clockwise (along the EXTERNAL part of an object). It means that the defined bar will always be positioned in the inner part of an object. In the case of bar with hooks of a bending angle greater than zero degrees, the hooks will always be located on the side opposite to the side where points defining the bar length are placed, thus they will be turned towards the middle of an object.
  Command line identical as for pin
- ***Bars*** - the function that allows bar definition on the existing point reinforcement (point reinforcement, i.e. presentation of longitudinal reinforcement within a cross section). After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'polyline'). Once a bar is indicated, the position of the bar with hooks is shown dynamically on the screen. A bar definition is completed when another point reinforcement bar is indicated. In this case, a cover value is not considered.
  Command line:
  Side.

A definition of a bar shape is **ALWAYS** completed at the moment a second point in the form of a point reinforcement bar, is indicated.

7.6. Bar from database

After pressing the icon, the dialog box assumes the form shown in the figure below.



NOTE:
Bar shapes are used to define a bar and assign an appropriate identification code to it.

The *Parameters of reinforcing bar shape* field includes the following options:

- schematic drawing of a reinforcement shape
- edit fields where hook parameters are specified; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change
- selection list containing codes - they affect bending diameters (French code)
- two buttons: **Shape database** and **Shape parameters**.

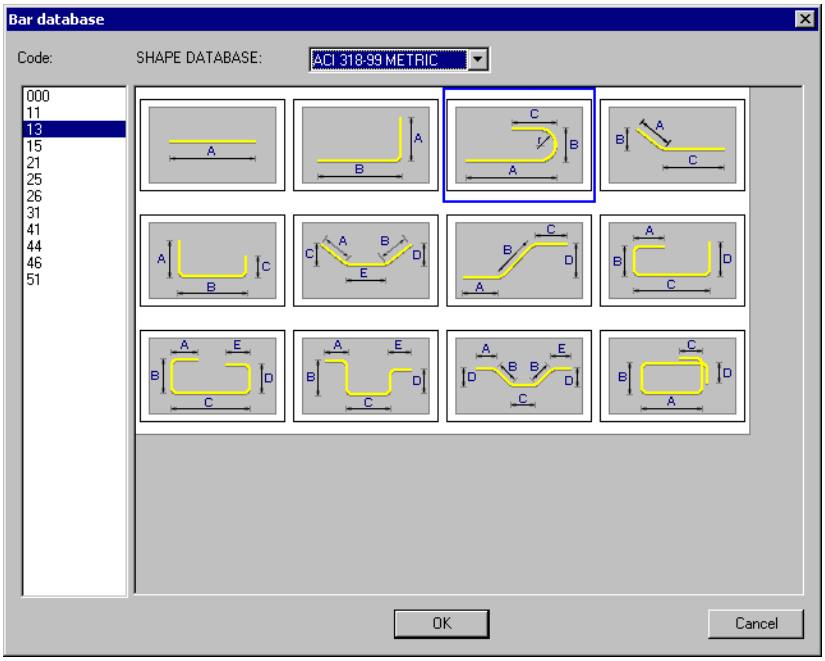
In the right part of the dialog box only the ***Insert*** and ***Points*** icons are available which when pressed, start definition of a bar from database. The remaining icons are unavailable.

After pressing the **Shape database…** button, the ***Bar database*** dialog box is opened.

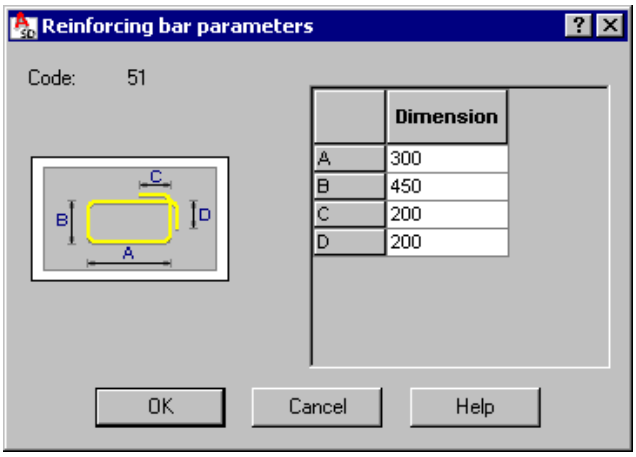
NOTE:
The databases of reinforcing bar shapes together with corresponding shape codes depend on a selected code of RC structure design.

Structure of the shape database:

- a shape can be selected from the database both by indicating directly a schematic drawing from the list of available shapes or by selecting a shape code number
- the database includes the field with a reference to the active code
- the database comprises the basic shapes repeated which can be selected directly from the ***Shape definition*** dialog box (closed and open stirrups, pin, shackle - it allows a different manner of definition by means of dimensions).



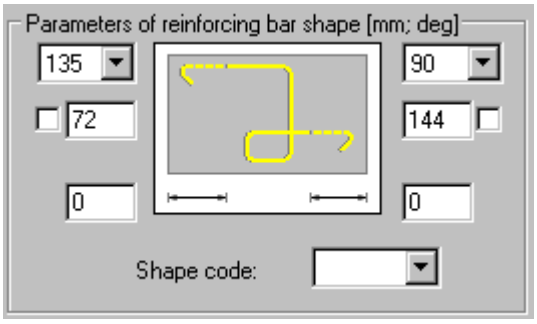
Definition of bars from the database consists in selecting one of them from the available shape list, and afterwards, defining next bar segments by indicating next bar characteristic points presented in a schematic drawing (the user may also enter dimensions of individual segments from the keyboard).
Once a bar is selected from the database, the bar segments are defined one by one based on the indicated points (there apply rigid rules of defining successive segments so that the bar shape is maintained). While defining, the bar length is calculated automatically. The bar dimensions proposed by default in the dialog box are the dimensions of the recently-defined bar.
A bar defined in the bar shape database remembers the shape of a selected bar type. The user may replace such a bar with a regular bar which does not remember the bar geometrical shape - to do that the ***AutoCAD Structural Detailing*** EXPLODE option should be used.
There is also another manner of bar definition. Once the **Shape parameters…** button is pressed, the additional dialog box appears on the screen in which (see the drawing below) a table containing dimensions of individual bar segments is available. After determining bar dimensions, a completed bar may be added to a formwork.



The above dialog box contains a table that displays dimensions of bar segments.
The field with a shape code is filled out automatically depending on a typical shape selected.
Both modes of reinforcement definition are synchronized with each other. It means that both definition modes can be used alternately.

7.7. Arbitrary bar shape

After pressing the icon, the dialog box assumes the form shown in the figure below.



This dialog box allows definition of any reinforcement shape.
The user may select a hook type at the beginning and end of a defined reinforcement and define bar anchors on its both ends; a default value of the anchor for an arbitrary bar equals zero. You can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.
The definition consists in indicating with the cursor, the successive characteristic points being the bars of point reinforcement or characteristic points.
Transversal reinforcement of arbitrary shape will be based on these points.
The field with a shape code is filled out by the user (a unique code of the bar shape should be entered there). This code will be presented in the reinforcement table; it will be also included in the information about the bar.

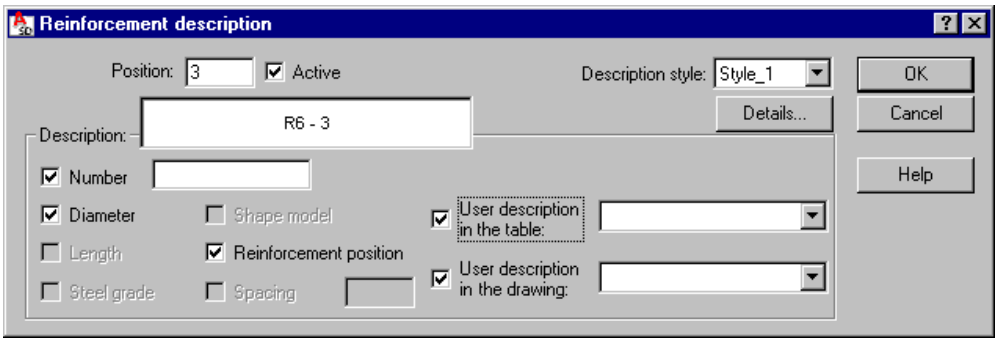
Four icons are active when defining this reinforcement type (the remaining icons are unavailable):

- ***Select*** - the function is applied to define reinforcement on an open polyline (arcs may be included there)
- ***Pick point*** - the function is applied to detect a contour (identically as for a polygon-shaped stirrup); a contour may include arc elements

- ***Points*** - the function is used to indicate directly any points. After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'line'). After defining the first point determining a bar position, on the screen bar length changes dynamically depending on the cursor position. While indicating next bars, the defined bar is 'wound' around these points, whereas the condition of a bending diameter is maintained automatically.
- ***Bars*** - the function that allows bar definition on the existing point reinforcement (point reinforcement, i.e. presentation of longitudinal reinforcement within a cross section). After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'select'). A bar definition is completed when another point reinforcement bar is indicated.

7.8. Description of reinforcement

Once definition of reinforcement shape is completed, the dialog box shown in the figure below, used to describe reinforcement appears on the screen.



The options provided in this dialog box enable final selection of the reinforcement description. It can be performed by switching off active variables initialized based on the syntax defined.

REMARKS:

- reinforcement shape can be described only for a single position, in other words, two or more bars cannot be described simultaneously
- the *Number* edit field allows the user to enter - from the keyboard - an ultimate number of reinforcing bars that will be used directly in the reinforcement summary tables
  the number of bars may be represented as an equation, e.g. 5*(3+7)
- in the *Spacing* edit field the user may enter spacing values, despite reinforcement distribution not being defined yet. This field is editable on condition that the "%spa" variable is contained in the style of reinforcement description. It is only a static parameter that serves informative purposes and may be applied in reinforcement tables which include the spacing parameter. However, it should be remembered that if the user has entered a spacing value in this field, then regardless of real values of bar spacing in the structure element, the value provided in the dialog box will be assumed in the table.
- the *Active* option. If the option is switched on, it results in including the reinforcement being described in the reinforcement table. It means that for reinforcement that is described for the first time this option will be active by default. When the same reinforcement is described twice, the option will be switched off on its own. In this manner it is possible to prevent (when describing the same reinforcement twice) the number of reinforcing bars calculated when preparing a bar table to be doubled; description of active and not active reinforcement may differ in a generated drawing – the options used for that purpose are located in the Description of reinforcement shape dialog box.

- the fields: *User description in the drawing* and *User description in the table* enable adding any text to a reinforcement description (by entering it from the keyboard); the description will be presented correspondingly in drawings (included in bar descriptions in the drawing) and in the table (included only in the reinforcement table); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; a text taken from the library of standard descriptions may also be applied. These descriptions assume the style of the text describing the reinforcement. The user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are applied. A user description together with an extension line and label make up one object. Such an object may be edited (translation, rotation) using available grips – these are small squares that appear at characteristic points of indicated objects; by means of the ***AutoCAD Structural Detailing*** *Explode* option a number of grips in descriptions may be increased
- a style of reinforcement description is chosen from the *Description style* selection list; the list contains all description styles defined for that type of reinforcement (the first on the selection list is a default description style chosen in the ***Description of reinforcement shape*** dialog box); before a reinforcement description is inserted in a drawing, parameters of the description style can be changed; pressing the **Details...** button opens the ***Reinforcement description*** dialog box where modifications of the style can be made (NOTE: modifications of the description style refer only to that one reinforcement description).

NOTE:
That whether any of the options listed above is turned on, depends on the defined description syntax available in styles of reinforcement description. If for example, a bar symbol is to be included in the bar description, the user should open the ***Description of reinforcement shape*** *dialog box, select reinforcement description (e.g. Bar shape), press the* ***Modify*** *button, switch on the Reinforcement symbol option provided on the Description syntax tab, press the button with the arrow; the variable containing a bar symbol will be added in the Description edit field; to end the operation, the* ***Add*** *button should be pressed.*

7.9. Commands from the command line - definition of transversal reinforcement

The following parameters may be determined in the command line while defining a transversal bar:
Bar diameter or [Cover / Stirrup type / Define] <6>: 8
Select option [Diameter / Cover / Stirrup type / Define] <Define>:
Reinforcement diameter <8>:
Cover <5.5> : 6
Stirrup type: [Polygon / Round / Pin / Shackle / Base / Any] <Polygon>:

For a polygon-shaped stirrup the following parameters may be defined:

Select option [DIagonal / Selection / INternal point] <Diagonal>:
First corner
Second corner or [Side / Back]
Select object
Hook location or [Side / Back]
Internal point
Hook location or [Side / Back]

For a round stirrup the following parameters may be defined:
Select option [2points / Selection / Internal point] <Selection>:
Circle center;

Circle diameter or [Diameter / Side / Back]<33>:
Hook location
Select object
Hook location or [Side / Back]
Internal point
Hook location or [Side / Back]

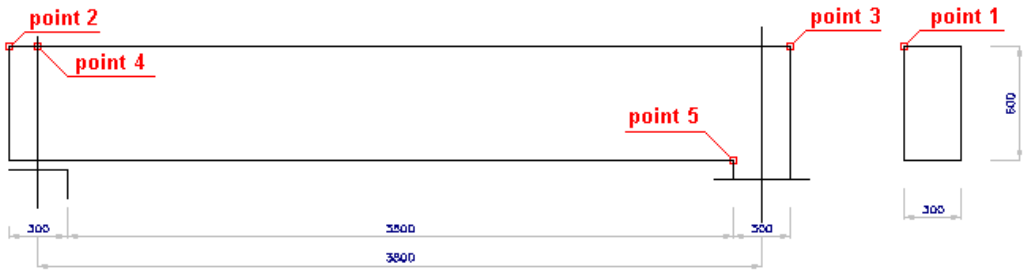
For a pin and shackle the following parameters may be defined:
Select option [Points / BArs] <BArs>:
Beginning point
End point or [Side / Back]:
First point
Second point or [Side / Back]:}
where:
beginning point - the first point determining the shape of a bar (bar beginning)
end point - the last point determining the shape of a bar
side - determines on which side of a cross section contour of an RC element the current segment of reinforcement is to be located
back - cancels the last command.

For bars from database the following parameters may be defined:
Bar code or [Select from database]
Beginning point
Next point or [Side / Cover / 1hook / 2hook / Back]
where:
beginning point - the first point determining the shape of a bar (bar beginning)
next point - next points determining the shape of a bar
side - determines on which side of a cross section contour of an RC element the current segment of reinforcement is to be located
cover - value of a cover for the current segment of reinforcement
1 hook - direction of the hook bending
2 hook - direction of the hook bending (opposite to hook 1)
back - cancels the last command.

For a bar of arbitrary shape the following parameters may be defined:
Beginning point
Next point or [Side / Cover / 1hook / 2hook / Back]
where:
beginning point - the first point determining the shape of a bar (bar beginning)
next point - next points determining the shape of a bar
side - determines on which side of a cross section contour of an RC element the current segment of reinforcement is to be located (change of a bar position with respect to the line - points of definition to the opposite one)
cover - value of a cover for the current segment of reinforcement
1 hook – changes the direction of hook bending to the opposite
2 hook - direction of hook bending
back - cancels the last command.

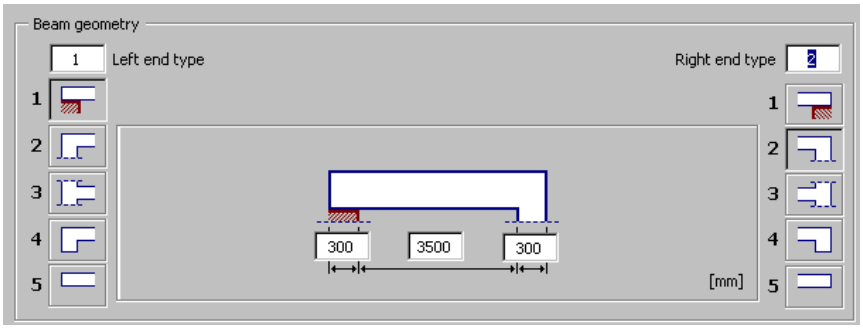
7.10. Example of definition of transversal reinforcement

The example below illustrates definition of transversal reinforcement in the cross section of the beam presented in the following drawing. For definition of the beam contours the ***Formworks - Beam*** macro has been used.



To define transversal reinforcement of the RC beam, follow the steps below:
DEFINITION OF THE BEAM FORMWORK

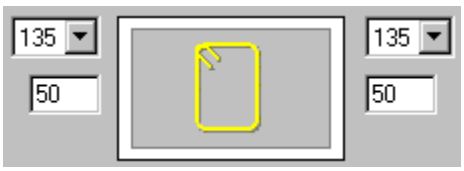
- run the ***Formworks – Beam*** macro selecting the menu option: *Reinforcement / Formworks / Beam* or pressing the icon
- in the ***Formworks – Beam*** dialog box adopt the following parameters:
  - section type: 1 (rectangular)
  - beam type: 1 (single-span beam)
  - dimensions of the beam cross section: height = 600 mm, width = 300 mm
  - beam geometry as shown in the drawing below



- press the **Insert** button in the ***Formworks – Beam*** dialog box and indicate the location of the beam formwork in the drawing

DEFINITION OF THE BEAM REINFORCEMENT

- run the menu option *Reinforcement / Reinforcement - section* or press the icon
- in the ***Reinforcement – cross-section*** dialog box determine the following parameters:
  - type of reinforcing bars: polygon-shaped (closed) stirrup
  - bar diameter: 8 mm; cover of reinforcing bars: 30 mm
  - steel grade: R
  - shape parameters as shown in the drawing below



- press the *Pick point* icon located in the right-hand side of the dialog box
- in the drawing of the beam formwork indicate a point within the beam cross section and point 1 that determines the position of stirrup hooks (see the drawing at the beginning of the example)
- accept the deafult reinforcement description proposed in the ***Reinforcement description*** dialog box pressing the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu.

The reinforcement in the beam cross section has been defined; below is presented definition of transversal reinforcement distribution along the beam length. To do it, follow the steps below:

- press the *Reinforcement distribution* icon
- indicate the previously-defined stirrup and press the ENTER key; the ***Reinforcement detailing*** dialog box appears on the screen then; in this dialog box select the following options:

  - distribution TYPE: linear (press the icon)

  - distribution METHOD: zone (press the icon)

  - viewing DIRECTION: press the icon

  - press the **OK** button
- indicate the start distribution point (point 2 in the drawing presented at the beginning of the example) and the end distribution point (point 3 in the drawing presented at the beginning of the example)
- indicate the points of the zone beginning (point 4 in the drawing presented at the beginning of the example) and the zone end (point 5 in the drawing presented at the beginning of the example); press the **Enter** key
- accept the default reinforcement distribution proposed in the the ***Reinforcement distribution*** dialog box by pressing the **OK** button
- adopt the following parameters of description of the reinforcement distribution

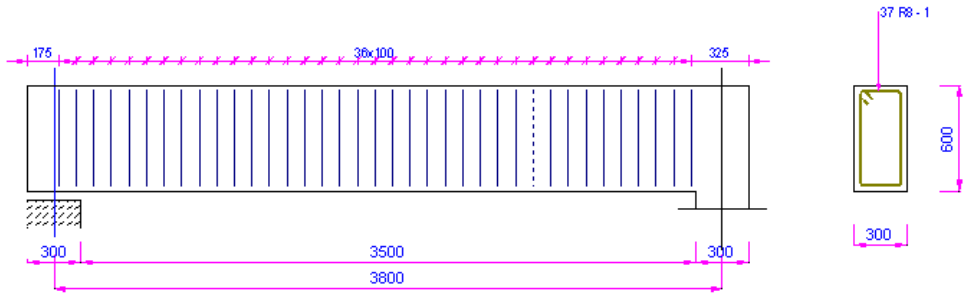
  - type of reinforcing bar presentation: all

  - position: the *Active* option switched on

  - type of distribution description:

  - press the **OK** button
- indicate the location of the reinforcement description in the drawing pressing the **Enter** key or clicking the right mouse button and choosing the *Enter* option from the context menu.

The defined transversal reinforcement with its distribution along the beam length is illustrated below.



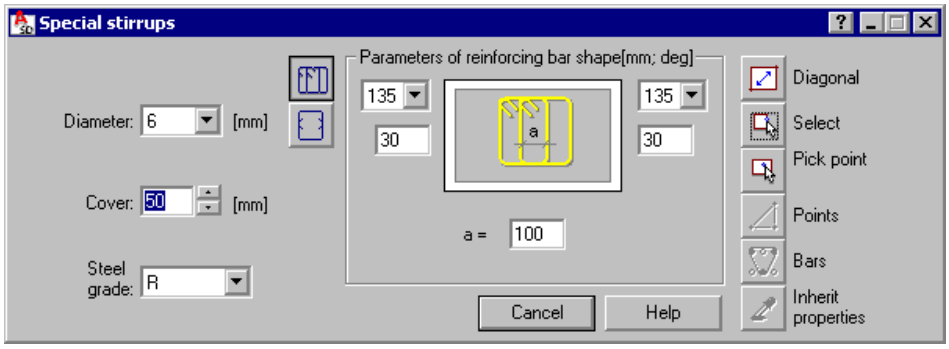
8. DEFINITION OF REINFORCEMENT - SPECIAL STIRRUPS

8.1. DEFINITION OF BAR REINFORCEMENT - bar section (special stirrups)

The option is used to define stirrups (transversal reinforcement) in the cross-section of an RC structure element. The option is available from:

- the menu by selecting the *Reinforcement / Special stirrups* option
- the toolbar by pressing the icon
- the command line: RBCR_DEF_STIRRUP_SPEC.

After activating the *Special stirrups* option, first the dialog box for selecting a shape of the reinforcement in the cross-section appears on the screen.



The ***Special stirrups*** dialog box can be split into three parts:

- the left part of the dialog box contains basic information concerning the reinforcement: diameter, cover and reinforcing steel grade adopted from preference settings
- the middle part of the dialog box includes icons that represent basic shapes of the transversal reinforcement and allow selection of the stirrup type; once the transversal reinforcement type is selected, contents of the field with parameters of the reinforcement shape changes
- the right part of the dialog box includes several icons which are used to select a mode of graphical reinforcement definition; if they are pressed, the graphic interface mode is selected according to a specific way of contour definition. NOTE: only after one of these icons is pressed, bars can be defined in the graphical window of the program.

The dialog box opens showing the transversal reinforcement type recently defined and parameters adopted for it.

The following types of special stirrups are available in the program:

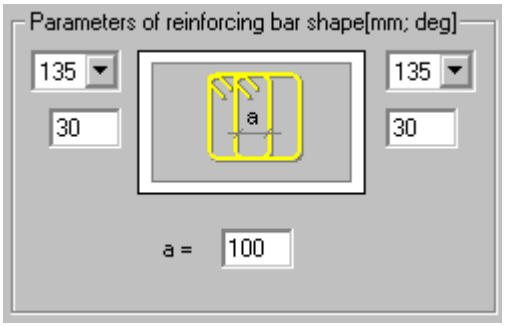
- four-leg stirrups
- stirrups made of bars in the shape of the double U-letter.

Once definition of the reinforcement shape is completed, the Reinforcement description dialog box appears on the screen where the user may select elements of a reinforcement description.

NOTE:
After completing definition of four-leg stirrups, the description command is run automatically. The command works for a selection and describes bars one by one moving from the first one to the second without the necessity to reactivate the command. It is possible to interrupt bar description and describe only one bar.

8.2. Special stirrups - four-leg stirrups

After pressing the icon the dialog box looks as shown in the drawing below.



To define a bar, the user should define the following parameters in the above dialog box:

- a bar diameter (the diameter defined recently in the dialog box is adopted by default)
- a cover of reinforcing bars (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for bar ends, i.e. hook angle, length and value of the a parameter which defines a clear distance between middle stirrup legs.

The shape of a bar presented in a schematic drawing is adjusted dynamically to the specified values of a hook bending angle. The list of standard values of hook angles is presented below:

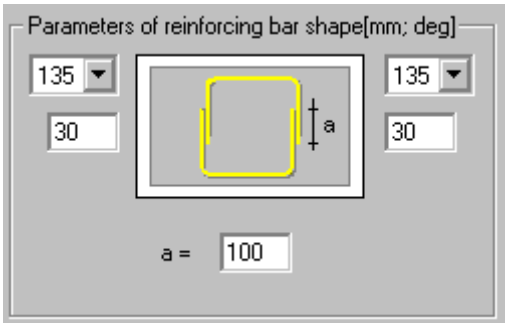
⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

The following icons are available in the right part of the dialog box while a rectangular stirrup is defined (the remaining icons are unavailable):

- *Diagonal* - the function is used to create a rectangular contour by defining a diagonal. The dialog box closes; while moving the cursor, a defined rectangular stirrup is presented dynamically
- *Select* - is used to indicate directly a rectangular contour. Once this button is pressed, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' - the phase of selecting the object to be copied). After selecting any point on the screen, the ACAD object is detected and a transversal reinforcement (stirrup) is drawn within the indicated object
- *Pick point* - the function that allows searching a closed contour (rectangle) by clicking inside the contour. The dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'hatch'). The detected rectangle is changed to a stirrup shaped like a detected contour, but decreased by the cover value.

8.3. Special stirrups - bars in the shape of the double U-letter

After pressing the icon the dialog box looks as shown in the drawing below.



To define a bar, the user should define the following parameters in the above dialog box:

- a bar diameter (the diameter defined recently in the dialog box is adopted by default)
- a cover of reinforcing bars (the cover defined recently in the dialog box is adopted by default)
- in the *Parameters of reinforcing bar shape* field - anchor parameters for bar ends, i.e. hook angle, length and value of the a parameter which defines a length of overlapping bars.

A shape of a bar presented in the schematic drawing is adjusted dynamically to the specified values of a hook bending angle. The list of standard values of hook angles is presented below:

⇒ 0°
⇒ 90°
⇒ 135°
⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

The following icons are available in the right part of the dialog box while a rectangular stirrup is defined (the remaining icons are unavailable):

- *Diagonal* - the function is used to create a rectangular contour by defining a diagonal. The dialog box closes; while moving the cursor, a defined rectangular stirrup is presented dynamically
- *Select* - is used to indicate directly a rectangular contour. Once this button is pressed, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' - the phase of selecting the object to be copied). After selecting any point on the screen, the ACAD object is detected and a transversal reinforcement (stirrup) is drawn within the indicated object
- *Pick point* - the function that allows searching a closed contour (rectangle) by clicking inside the contour. The dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'hatch'). The detected rectangle is changed to a stirrup shaped like a detected contour, but decreased by the cover value.

9. DEFINITION OF REINFORCEMENT - POINT REINFORCEMENT

9.1. Point reinforcement

The option allows the user to define distribution of reinforcing bars in an element of an RC structure. The option is available from:

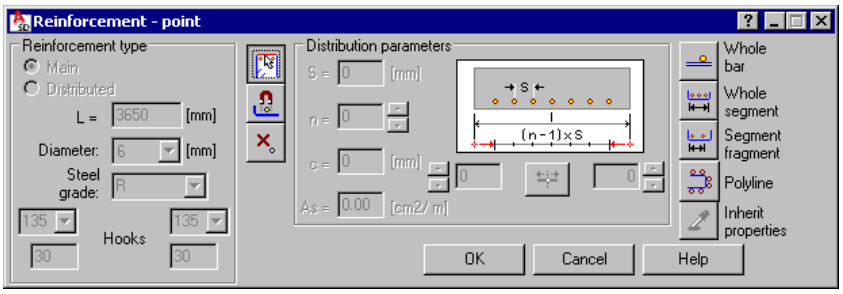
- the menu by selecting the *Reinforcement / Reinforcement - point* option
- the toolbar by pressing the icon
- the command line: RBCR_DISTRIBUTION_POINT.

The dialog box is displayed if reinforcement has been selected or if a number of a reinforcement position has been specified in the command line (by default, the number of the recently-defined bar is suggested):

- if main reinforcement has been selected, then the fields in the dialog box are filled out with data (diameter, position number, appropriate bar length)
- if distributed reinforcement has been selected, then the fields in the dialog box are filled out with the following data: diameter, position number (both parameters are not accessible); the edit field used for definition of the reinforcement length is available.

NOTE:
When defining the point reinforcement which does not refer to the existing reinforcement (e.g. when defining distributed reinforcement), the dialog box may be opened after pressing the ESC button.

The dialog box may also be opened automatically from the level of the dialog box for reinforcement distribution. Then the dialog box is opened in the mode of main reinforcement definition with an appropriate diameter, position number, and appropriate bar length (the parameters are not accessible).
After selecting the option, the dialog box shown below is displayed on the screen.



The above dialog box is divided into three principal parts:

- the left part of the dialog box (the *Reinforcement type* field) contains basic information concerning the reinforcement:)
  - reinforcement type: main, distributed
  - edit field where reinforcement length can be defined
  - reinforcement diameter

  This part of the dialog box undergoes no changes.
  Moreover, there is the information concerning hooks on ends of a reinforcing bar (beginning and end of a bar)
- the middle part of the dialog box includes three icons, which when pressed, change the content of the *Distribution parameters* field with parameters of reinforcement distribution:

- regular distribution of reinforcement
- automatic distribution of reinforcement
- any distribution of reinforcement

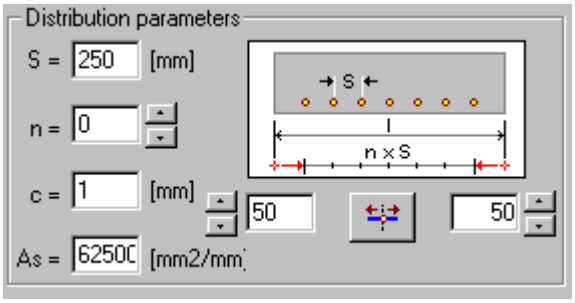
Note: Automatic distribution and any distribution of reinforcement are coupled, as regards the regular distribution of reinforcement, it can be combined neither with automatic nor any distribution of reinforcement

- in the right part of the dialog box a group of icons is provided which are used for selecting the graphic definition mode; pressing these icons activate different modes of graphic interface.

The dialog box remains visible throughout the process of defining reinforcement distribution, which enables the user to change the distribution parameters of reinforcement while defining it. All the methods of bar distribution can be applied with respect to one reinforcement position.
The distribution of a given reinforcement position is completed by pressing the **OK** button. Afterwards, distribution description is defined.

9.2. Regular distribution of reinforcement

After pressing the icon, the Reinforcement - point dialog box assumes the form shown in the figure below.



The left part of the above dialog box contains the parameters controlling reinforcement distribution within one zone:

- S - reinforcement spacing
- n - number of spacings between bars in a given zone
- c - distance (cover) between bars and reinforcement or polylines along which reinforcement is distributed (the default value equals zero). The cover is always interpreted as perpendicular to a bar or formwork line.
- As - area of reinforcement per length unit.

Next to them a drawing of reinforcement distribution is provided, whereas directly underneath, the options that allow positioning can be found.

The following methods of reinforcement distribution may be adopted:

1. - along the whole indicated bar
2. - along the whole indicated segment of a bar
3. - along the fragment of the indicated bar segment
4. - along the indicated polyline (arc, circle, line i.e. between 2 points).

When defining the distribution applying the methods 1 or 2, the user should indicate a bar or segment of reinforcement; the cursor assumes the shape of a square. After selection and initial positioning of reinforcement by the program according to the distribution parameters defined in the dialog box, the user may determine (by means of the options available in the command line) some of the parameters such as:

- change of the side on which the distributed reinforcement is positioned - with respect to the bar (the Side command)
- change of the parameter spacing / number (the Spacing / Number command)
- Justify - positioning the distribution centrally or justified to the left or right side of the distribution
- Adjust - uniform distribution of reinforcement along the whole distribution length
- Cover - change of a cover value = offset.

If distribution is defined by means of method 3, the extreme points are determined by the user. In the case of distribution defined with the use of method 4, reinforcement is distributed on the whole selected objects such as **line, polyline, circle**.
The user may change values of the above-discussed parameters using the options available in the dialog box , however, it should be remembered, that in the dialog box the side on which reinforcement is positioned cannot be changed.

The method of reinforcement distribution consists in specifying the number of reinforcing bars/spacing or reinforcement area per one running meter. Hooks are not considered in case of distribution along the bar. For distribution along the segment, the length between fillets is taken into account.

Command line (regular distribution)

Distribution type: [Regular / Automatic / Any] **Regular**

Regular
Select Bar or [Segment / Segment fragment / Polyline]

Bars
Select bar or [Side / Back]

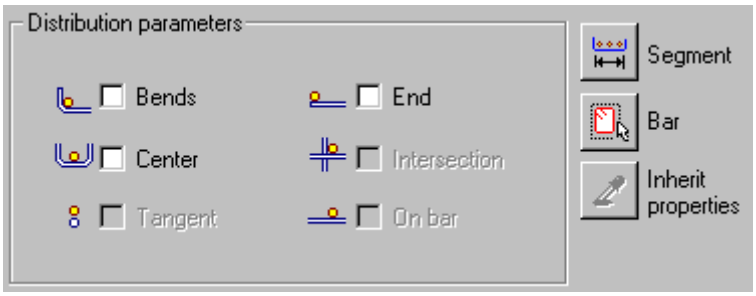
Segment
Select bar segment or [Side / Back]

Segment fragment
Select bar segment
Select beginning point or [Back]
Select end point or [Side / Back]

Polyline
Select object or [Side / Back]

9.3. Automatic distribution of reinforcement

After pressing the  icon, the Reinforcement - point dialog box assumes the form shown in the figure below.



NOTE: The options: bar intersection, tangent to bar and on bar are not accessible in the current program version.

The method is intended only for defining bar distribution for the existing reinforcement. It has been designed in such a manner so that in typical cases the user can easily arrange bars based on the following characteristic points of reinforcement (not all the options are available):

- hook bending
- end of bar or segment
- center of bar or segment
- intersection of bars - the option is not available in the current program version
- tangentially to bar - the option is not available in the current program version
- on bar - the option is not available in the current program version.

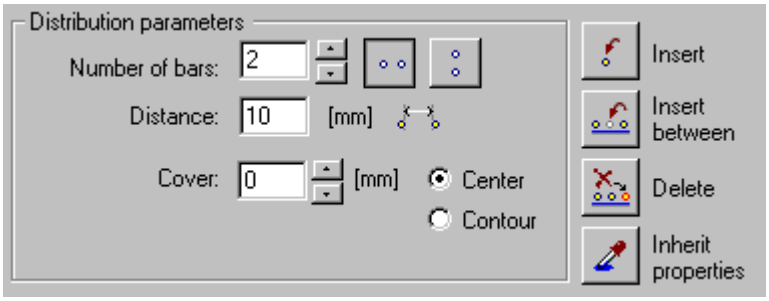
Bars may be distributed along the following:

- a selected bar segment -
- a defined bar shape (whole bar) -.

Commands of the command line are set in such a manner so that the user may select one or several bars/segments simultaneously.
Apart from that, the user may change position of the reinforcement being distributed with respect to a bar / segment axis.

9.4. Any reinforcement distribution

After pressing the icon, the Reinforcement - point dialog box assumes the form shown in the figure below.



The distribution method is intended for all these cases where standard methods of reinforcement distribution prove ineffective. This method will allow the user to position a bar or a bar group freely, based on the distribution carried out earlier; the distribution may also be defined independently.

The method described enables distribution of one or more bars simultaneously. For at least two bars that are inserted concurrently, the user should determine the distance between bars and their orientation (horizontally or vertically) while inserting.
Precise arrangement of bars may be facilitated by the use of characteristic points (characteristic points of the ***AutoCAD*** ® program).
For bars defined as vertical ones and when applying points characteristic for reinforcement, it should be remembered that bar orientation will always be perpendicular to the bar / segment axis.
The right part of the dialog box contains the icons defining the following distribution methods:

Insert

The options allows starting definition of distribution; the definition consists in indicating - with the mouse cursor successive positions of reinforcement. The only assistance available while defining distribution is offered the characteristic points - snap points (of the ***AutoCAD*** ® program). Distribution may also be defined from the level of the command line.

Insert between

The option allows the user to indicate two point bars and insert 1 bar or a group of bars between them. When inserting a bar, the order between the indicated bars changes; if the two selected bars had been inserted using the characteristic points, then the bars inserted should behave identically as base bars; if it was not so, then the bars added are inserted along a straight line between the existing bars.

Delete

The option permits deleting an indicated bar.

Command line (any distribution)

Distribution type: [Regular / Automatic / Any]: **Any**

Select attachment point or [Modify properties / Insert between / Delete / Snap / Back]

Modify parameters [Number of bars / Spacing / Orientation]
Number of bars <2> : 1
Spacing <0.12> : 0.18
Orientation <V> : H

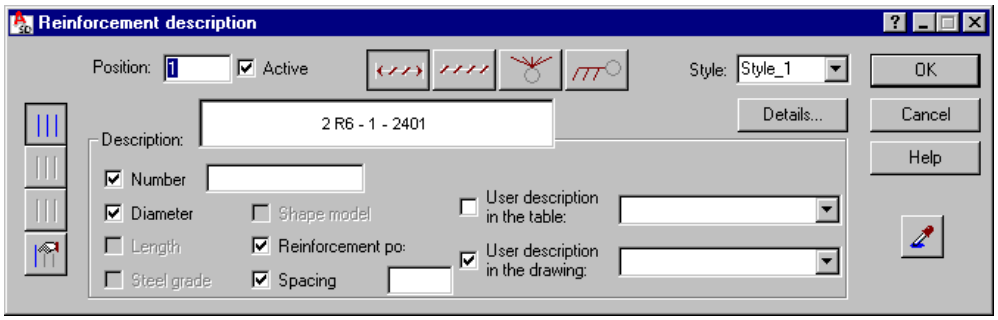
Insert Between
Select initial bar
Select end bar or [Back]

Delete
Select bar or [Back]

9.5. Reinforcement description

Once definition of reinforcement and its distribution is completed, the below-presented dialog box used for describing reinforcement distribution appears on the screen.

NOTE:
The dialog box below opens after selecting the Modify / Reinforcement description option provided in the menu or pressing the icon; in this case two additional options are available in the dialog box: Description style (used to modify a description style of the reinforcement chosen) and the ***Details*** *button, which when pressed opens the dialog box used for modifying a style of reinforcement description (shape).*



The options provided in this dialog box enable final selection of reinforcement description and distribution. It can be performed by switching off active variables initialized based on the syntax defined.
The dialog box contains the following options:

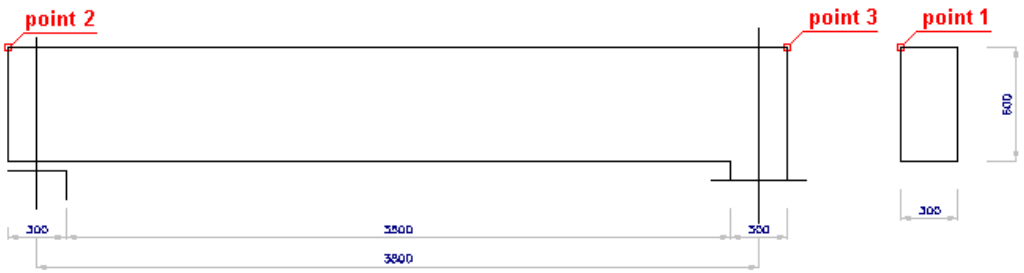
- the field informing about the current position of reinforcement; the *Active* option allows the user to assign a description to the reinforcement which already has been given the description defined earlier - it enables avoiding the situation when the same reinforcement is included in calculations twice or many times when preparing a reinforcement table
- the icons used to define the manner of presenting the distribution:
  - all elements are presented in a drawing showing given reinforcement distribution
  - only the middle representative of the distribution in a given zone is presented (the remaining elements are not visible)
  - only the extreme elements of the distribution in a given zone are presented (the remaining elements are not visible)
  - it allows the user to indicate graphically distribution elements to be presented
- the fields: *User description in the drawing* and *User description in the table* enable adding any text to a reinforcement description (by entering it from the keyboard); the description will be presented correspondingly in drawings (included in bar descriptions in the drawing) and in the table (included only in the reinforcement table); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; a form of the displayed description is closely connected with the appropriate and default style of reinforcement description. The user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are applied. A user description together with an extension line and label make up one object. Such an object may be edited (translation, rotation) using available grips – these are small squares that appear at characteristic points of indicated objects; by means of the ***AutoCAD Structural Detailing*** *Explode* option it is possible to increase a number of grips in descriptions
- a style of reinforcement description is chosen from the *Style* selection list; the list contains all description styles defined for that type of reinforcement (the first on the selection list is a default description style chosen in the ***Description of reinforcement shape*** dialog box); before a reinforcement description is inserted in a drawing, parameters of the description style can be changed; pressing the **Details...** button opens the ***Reinforcement description*** dialog box where modifications of the style can be made (NOTE: modifications of the description style refer only to that one reinforcement description).

NOTE:
If for example, a bar symbol is to be included in the bar description, the user should open the ***Description of reinforcement shape*** *dialog box, select reinforcement description (e.g. Bar shape), press the* ***Modify*** *button, switch on the Reinforcement symbol option provided on the Description syntax tab, press the button with the arrow; the variable containing a bar symbol will be added in the Description edit field; to end the operation, the* ***Add*** *button should be pressed.*

Thanks to the options (description components) provided in the dialog box, the user may switch off active description components. The additional description allows entering an additional text. The comments entered are remembered and may be reused later on.

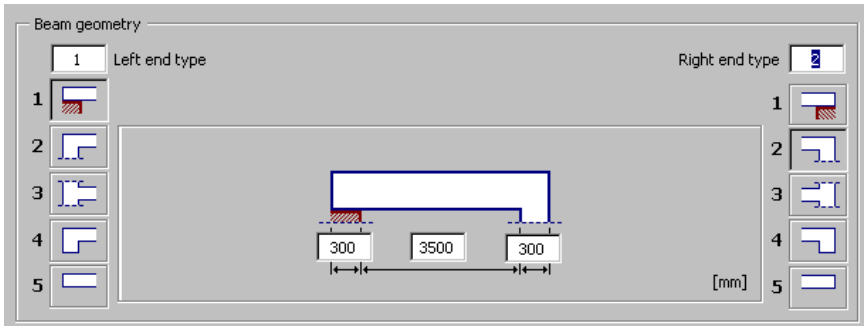
9.6. Example of definition of point reinforcement

The example below illustrates definition of point reinforcement in the cross section of the beam presented in the following drawing. For definition of the beam contours the ***Formworks - Beam*** macro has been used.



To define point reinforcement in the beam, first define transversal and longitudinal reinforcement:
DEFINITION OF THE BEAM FORMWORK

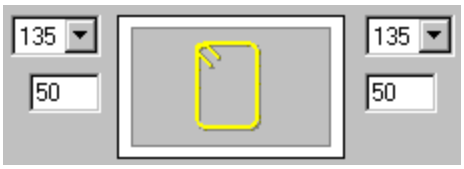
- run the ***Formworks – Beam*** macro selecting the menu option: *Reinforcement / Formworks / Beam* or pressing the icon 
- in the ***Formworks – Beam*** dialog box adopt the following parameters:
  - section type: 1 (rectangular)
  - beam type: 1 (single-span beam)
  - dimensions of the beam cross section: height = 600 mm, width = 300 mm
  - beam geometry as shown in the drawing below



- press the **Insert** button in the ***Formworks – Beam*** dialog box and indicate the location of the beam formwork in the drawing

DEFINITION OF THE BEAM REINFORCEMENT

- run the menu option *Reinforcement / Reinforcement - section* or press the icon 
- in the ***Reinforcement – cross-section*** dialog box determine the following parameters:
  - type of reinforcing bars: polygon-shaped (closed) stirrup 
  - bar diameter: 8 mm; cover of reinforcing bars: 30 mm
  - steel grade: R
  - shape parameters as shown in the drawing below

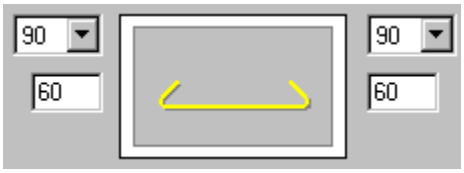


- press the *Pick point* icon located in the right-hand side of the dialog box
- in the drawing of the beam formwork indicate a point within the beam cross section and point 1 that determines the position of stirrup hooks (see the drawing at the beginning of the example)
- accept the deafult reinforcement description proposed in the ***Reinforcement description*** dialog box pressing the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu.

The reinforcement in the beam cross section has been defined; below is presented definition of straight longitudinal reinforcement. It comprises the following steps:

- run the menu option *Reinforcement / Reinforcement - elevation* or press the icon
- in the ***Reinforcement – elevation*** dialog box determine the following parameters:

  - type of reinforcing bars: straight bar
  - bar diameter: 12 mm; cover of reinforcing bars: 30 mm
  - steel grade: R
  - shape parameters as shown in the drawing below

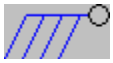


- press the *2 points* icon located in the right-hand side of the dialog box
- in the drawing of the beam formwork indicate point 2 and point 3 (see the drawing at the beginning of the example)
- accept the deafult reinforcement description proposed in the ***Reinforcement description*** dialog box by pressing the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu.

The next step, once the transversal and longitudinal reinforcement is defined, is definition of point reinforcement; to do it, follow the steps below:

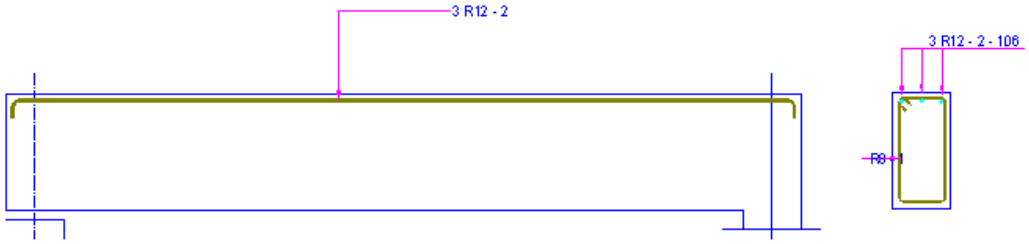
- run the menu option *Reinforcement / Reinforcement - point* or press the icon
- indicate the defined longitudinal bar
- in the ***Reinforcement – point*** dialog box determine the following parameters:

  - bar distribution: regular
- press the *Whole segment* icon
- indicate the upper part of the defined stirrup (see the drawing at the beginning of the example) and press the **Enter** key
- in the ***Reinforcement – point*** dialog box determine the following parameters:
  - number of bars n = 3
  - leave default values of the remaining parameters
- press the **OK** button in the ***Reinforcement – point*** dialog box
- adopt the following parameters of the description of reinforcement distribution:

- type of reinforcing bar representation: all 
- position: the *Active* option switched on
- type of distribution description: 

- press the **OK** button
- indicate the position of the reinforcement description in the drawing by pressing the **Enter** key or pressing the right mouse button and choosing the *Enter* option from the context menu.

The defined point reinforcement is presented in red in the drawing below.



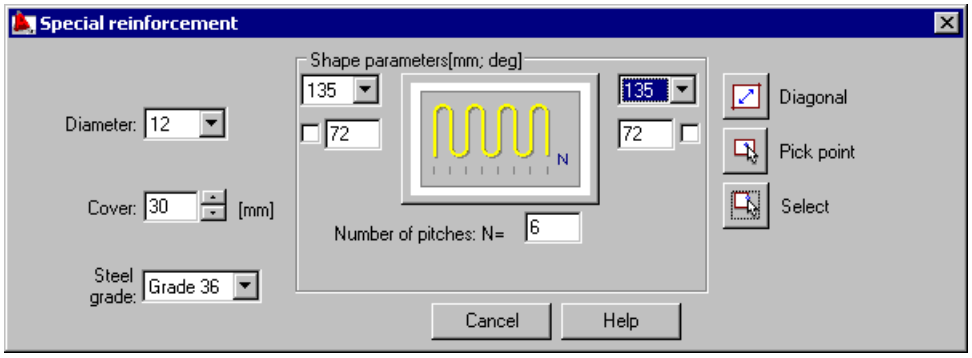
10. DEFINITION OF REINFORCEMENT - SPECIAL BARS

10.1.Special reinforcement

The option enables definition of particular reinforcing bars used in different elements of RC structures (e.g. crest-shaped reinforcement, corbel reinforcement, transport handles). The option is available from:

- the menu by choosing the *Reinforcement / Special reinforcement* option
- the toolbar by pressing the icon
- the command line: RBCR_DEF_BARLIBSPECIAL.

Once the *Special reinforcement* option is selected, the dialog box shown in the figure below is displayed on the screen.



Three parts may be distinguished in the ***Special reinforcement*** dialog box:

- the left part of the dialog box contains basic information concerning reinforcement: diameter, cover and steel grade adopted from the preference settings
- the middle part of the dialog box includes parameters of a selected type of the special reinforcement; clicking the left mouse button on the field containing the drawing of the special reinforcement opens an additional dialog box in which reinforcement type (shape) may be chosen; selection of special reinforcement type changes the contents of the field with parameters of reinforcement shape
- the right part of the dialog box comprises a few icons that are used for selecting a mode of graphical reinforcement definition (the number of icons depends on a chosen type of special reinforcement); pressing any of them indicates selection of the graphical interface mode according to the specific character of region definition. NOTE: only if one of these icons is pressed, bar definition is enabled in the program graphical window.

The dialog box opens showing the recently-defined type of special reinforcement and parameters assumed for it.

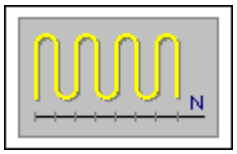
The following types of special reinforcement are available in the program:

- 3D reinforcement (defined in the drawing plane): corbel reinforcement, goalposts reinforcement and helix of RC structure elements with round cross-section
- transport handles
- the remaining reinforcement types: crest-shaped reinforcement, arc-shaped reinforcement, vertical loop.

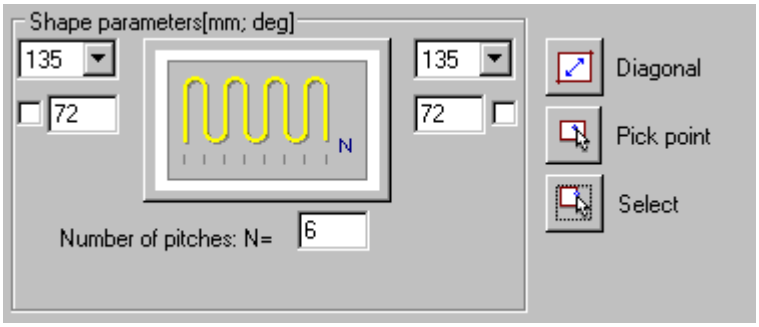
Once definition of reinforcement shape is completed, the program opens the Reinforcement description dialog box used to select elements of reinforcement description.

Reinforcing bars may also be defined by means of the commands available in the program.

10.2. Crest-shaped reinforcement



Once crest-shaped reinforcement is chosen, the Special reinforcement dialog box assumes the form shown in the figure below.



NOTE:
Crest-shaped reinforcement can be defined only within a rectangular region.

To define a bar, the following should be determined in the dialog box:

- bar diameter (the diameter defined recently in the dialog box is assumed by default)
- reinforcing bar cover (the cover defined recently in the dialog box is assumed by default)
- in the *Shape parameters* field – anchorage parameters of reinforcing bar ends, i.e. hook angle and length as well as number of legs in crest-shaped reinforcement; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining the crest-shaped reinforcement, the following icons are available in the right part of the dialog box:

- *Diagonal* – the function is used to determine a rectangular region by defining a diagonal. The dialog box closes; while moving with the cursor, the reinforcement being defined is presented dynamically

  Command line:
  First corner
  Second corner or [Orientation / First hook / Second hook / Cover]

  where:
  First corner – indicates the initial apex of a rectangle within which crest-shaped reinforcement is to be defined
  Second corner - indicates the second apex of a rectangle (along the rectangle diagonal) within which crest-shaped reinforcement is to be defined
  Orientation – rotates (changes position) of reinforcement within the contour
  First hook – changes the hook bend at the beginning of a reinforcing bar (first corner) to the opposite
  Second hook - changes hook bend at the end of a reinforcing bar to the opposite
  Cover – defines a cover value globally for the entire region of a rectangle

- *Pick point* – function which enables searching a closed region in the shape of a rectangle by clicking inside the region. The dialog box closes, whereas the cursor assumes the form of a cross (see the ACAD command 'copy' - the stage of indicating the beginning and end points). Once any point is selected, the minimal contour is detected on the screen
- *Select* – is used to indicate directly a rectangle-shaped contour. Once this option is chosen, the dialog box closes, whereas the cursor assumes the form of a square (see the ACAD command 'copy' - the stage of selecting an object for copying). Once any point is selected, an ACAD object is detected on the screen and reinforcement is drawn within the indicated object.

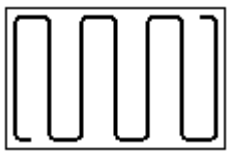
  Command line (for the options: *Pick point* and *Select*)
  Select internal point of the object (for the *Pick point* option)
  Select object (for the *Select* option)
  Orientation / First hook / Second hook / Cover

  where:
  orientation - rotation of reinforcement within the contour by indicating a contour side
  first hook - change of the hook bend at the beginning of a reinforcing bar (first corner) to the opposite
  second hook - change of hook bend at the end of a reinforcing bar to the opposite
  cover - definition of a cover value globally for the entire region of a rectangle

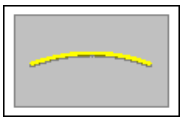
10.3.Example definition of crest-shaped reinforcement

To define the crest-shaped reinforcement within the rectangle shown in the figure below, the user should follow the steps below, for example:

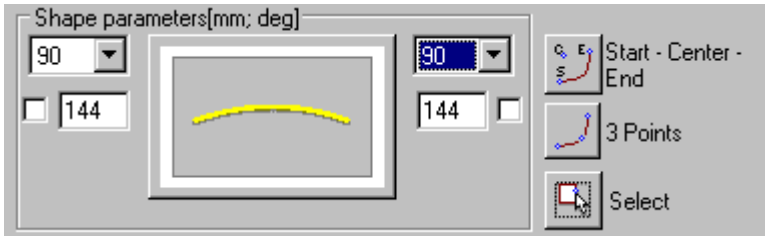
- in the ***Special Reinforcement*** dialog box choose the reinforcement type – crest-shaped reinforcement
- define the following values in the dialog box (the *Shape parameters* field): hook bend angles at the bar beginning and end 90 degrees, hook lengths 0.06 m, number of legs n = 7
- press the *Diagonal* icon
- indicate a first rectangle apex (see the above drawing)
- in the command line choose the Second hook command (it may be done by entering the capital letter) – it changes a hook position on the bar end
- in the command line choose the Orientation command – it rotates reinforcement within the contour
- indicate a second rectangle apex (see the above drawing) – the reinforcement obtained is shown in the drawing below.



10.4. Arc-shaped reinforcement



Once the arc-shaped reinforcement is chosen, the Special reinforcement dialog box assumes the form shown below.



To define a bar, the user should determine the following in the dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)
- in the *Shape parameters* field – anchorage parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining the arc-shaped reinforcement, the following icons are available in the right part of the dialog box:

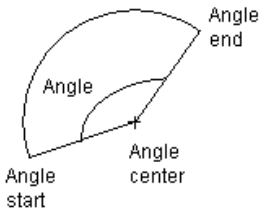
- *Start - Center - End* – function is used to define an arc by specifying three snap points of the arc: arc beginning point, arc center and arc end point. The dialog box closes; while moving with the cursor, the reinforcement being defined is presented dynamically

  Command line:
  Select start point
  Select arc center
  Select arc end

  or

  Select arc center
  Select arc start
  Select arc end or [Angle / Arc length]

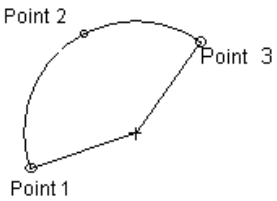
  where:
  start point – arc beginning
  arc end - end point
  arc center – point being arc center
  angle – value of arc angle
  length – value of arc length



- *3 Points* – the function which enables arc definition by specifying three points belonging to the arc. The dialog box closes; while moving with the cursor the reinforcement being defined is presented dynamically

  Command line:
  Select first point
  Select second point
  Select third point

  where:
  first, second and third points denote successive points belonging to an arc

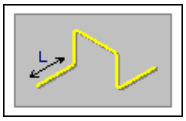


- *Select* – is used to indicate directly an arc-shaped object. Once this option is chosen, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' – the stage of selecting an object for copying). Once any element is selected, an ACAD object is detected on the screen and reinforcement is drawn within the indicated object.

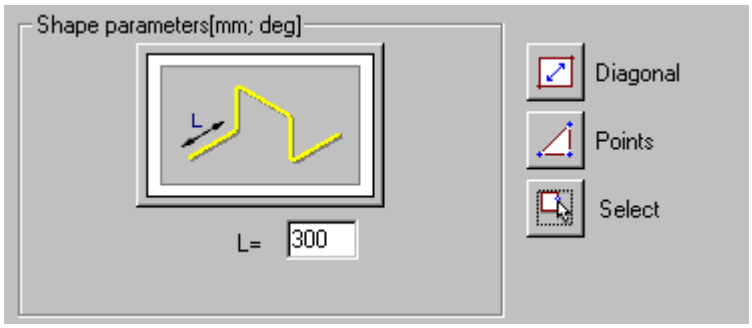
  Command line:
  Select object
  Side
  1 Hook
  2 Hook

  where:
  object – arc or polyline segment
  side – modifies reinforcement position with respect to an object (formwork)
  1 hook – direction of hook bend
  2 hook - direction of hook bend (opposite in relation to hook 1)

10.5.'Goalposts' reinforcement



After the ‘goalposts’ reinforcement is selected, the Special reinforcement dialog box assumes the form presented in the figure below.



To define a bar, the user should determine the following in the dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)
- in the *Shape parameters* field – reinforcement leg length (dimension perpendicular to the drawing plane).

While defining the ’goalposts’ reinforcement, the following icons are available in the right part of the dialog box:

- *Diagonal* – function is used to determine a rectangular region by defining a diagonal. The dialog box closes; while moving with the cursor, the reinforcement being defined is presented dynamically

  Command line:
  First corner
  Second corner or [Orientation / Cover]

  where:
  first corner - indicates the initial apex of a rectangle within which reinforcement is to be defined
  second corner - indicates the second apex of a rectangle (along the rectangle diagonal) within which reinforcement is to be defined
  orientation - rotates (changes position) of reinforcement within the contour; indicates location of the bar open segment
  side - modification of reinforcement position with respect to an object (formwork)

- *Points* – function which enables defining reinforcement by specifying positions of reinforcement snap points

  Command line:
  Select first point
  Select second point or [Cover]
  Select third point or [Cover]

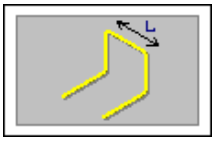
where:
first, second and third points – successive snap points of reinforcement (see the drawing below)
cover – determines cover value for individual segments of reinforcement

-  *Select* – is used to indicate directly the object of reinforcement shape. Once this option is chosen, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' – the stage of selecting an object for copying). Once any point is selected, the ACAD object is detected on the screen and a stirrup is drawn within an indicated object

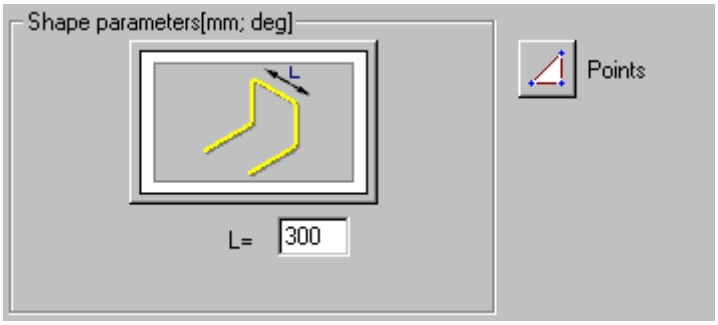
  Command line:
  Select object
  Orientation

  where:
  object – rectangle-shaped contour
  orientation – rotates (changes position) of reinforcement inside the contour; indicates location of bar open segment

10.6.Corbel reinforcement



After the corbel reinforcement (e.g. short cantilevers to be placed under a crane girder) is selected, the Special reinforcement dialog box assumes the form presented in the figure below.



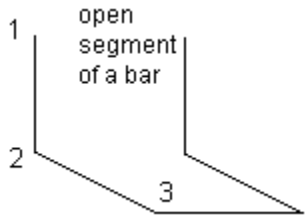
To define a bar, the user should determine the following in the dialog box:
- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)

- in the *Shape parameters* field – reinforcement leg length ('depth') (dimension perpendicular to the drawing plane).

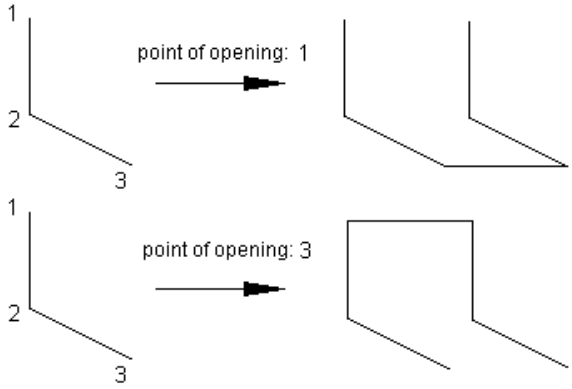
While defining the corbel reinforcement, the  *Points* icon which enables defining successive snap points for this type of reinforcement, is available in the right part of the dialog box.

Command line:
Select first point
Select second point or [Side / Cover]
Select third point or [Side / Cover]
Indicate location of open segment of a bar

where:
first, second and third points – successive reinforcement snap points (see the drawing below)

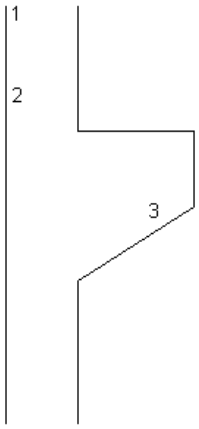


cover – determines a cover value for individual segments of reinforcement
side – changes position of reinforcement with respect to an object (formwork)
location of open segment of a bar – indicates a point (in the plane) at which bar opening is located – see the drawing below.

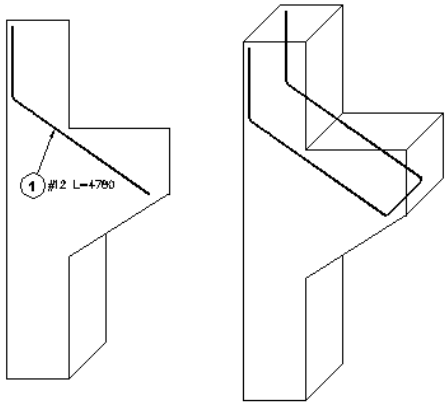


10.7. Example definition of corbel reinforcement

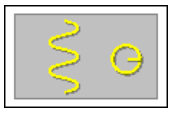
To define reinforcement of a cantilever to be provided under a crane girder, shown in the figure below, the user should follow the steps below, for example:



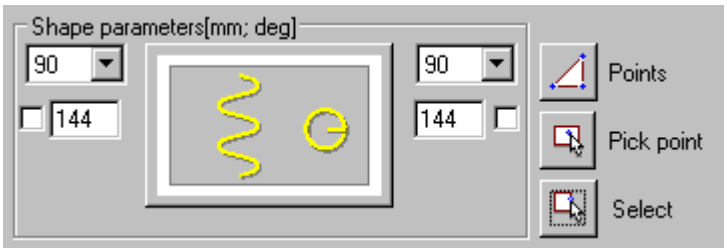
- in the ***Special reinforcement*** dialog box select the reinforcement type – corbel reinforcement
- define the value (in the *Shape parameters* field) of leg length equal to 30 cm
- press the  *Points* icon
- indicate point 1
- in the command line select the Side command – it modifies position of reinforcement with respect to the object (formwork)
- indicate successive points: 2 and 3 (see the figure above)
- indicate point 1 as a point of reinforcement opening; the defined reinforcement is presented in the figure below.



10.8.Helix



Once reinforcement in the form of helix is chosen, the Special reinforcement dialog box assumes the form shown in the figure below.



NOTE:
The helix may be defined only for two types of cross-section of an RC structure element (contour): circle or regular polygon.

To define a bar, the user should determine the following in the dialog box:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)
- in the *Shape parameters* field – anchorage parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

The helix is a three-dimensional reinforcement. In the program this reinforcement type is defined in the drawing plane. The helix is defined in two stages:

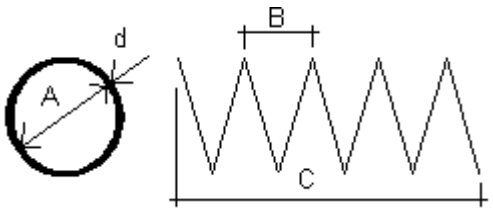
- definition of reinforcement within the cross-section of an RC structure element (e.g. column)
- definition of reinforcement along the length of RC structure element.

NOTE:
The program provides the possibility to modify a helix pitch during definition of this reinforcement type.

Total length of this reinforcement type is calculated according to the following formula:

$L = C / B * \pi * (A + d) +$ hook lengths

where (see the drawing below):
L – total helix length
C – helix length in plane
B – helix pitch
A – helix internal diameter
d – bar diameter.



While defining the helix, the following icons are available in the right part of the dialog box:

- *Points* – function which enables definition of a closed region by specifying circle center and circle radius (or diameter) as well as indicating a region of reinforcement distribution - beginning and end of the region within which reinforcement is distributed (e.g. along the column length)

Command line:
Definition of reinforcement within a cross-section
Circle center {2Points / 3Points]
Circle radius [Diameter]

Definition of reinforcement along the length of reinforced element
Select first point
Select second point [Spiral pitch]

where:
a circle is defined within the cross-section by specifying: circle center and circle radius/diameter or by determining 2 or 3 points belonging to a circle
first point – beginning point of the helix along the length of an RC element
second point – end point of the helix along the length of an RC element
spiral pitch – value of the helix pitch

-  *Pick point* – function which enables searching a closed region by clicking inside the region. The dialog box closes, whereas the cursor assumes the form of a cross (see the ACAD command 'copy' - the stage of indicating the beginning and end points). Once any point is selected, the minimal contour is detected on the screen; the helix can be defined by indicating the following geometric figures: regular polygon and circle; to conclude the operation, the user has to indicate the region of reinforcement distribution – beginning and end of the region within which reinforcement is distributed (e.g. along the column length)

  Command line
  Definition of reinforcement within a cross-section
  Select internal point of the object
  Orientation

  Definition of reinforcement along the length of reinforced element
  Select first point
  Select second point [Spiral pitch]

  where:
  a circle is defined within the cross-section by specifying an internal point of a circle or regular polygon and indicating positions of hooks (orientation)
  first point – beginning point of the helix along the length of an RC element
  second point – end point of the helix along the length of an RC element
  spiral pitch – value of the helix pitch

-  *Select* – is used to indicate a contour directly. Once this option is chosen, the dialog box closes, whereas the cursor assumes the shape of a square (see the ACAD command 'copy' – the stage of selecting an object for copying). Once any point is selected, the ACAD object is detected on the screen and reinforcement is drawn within an indicated object.

  Command line:
  Definition of reinforcement within a cross-section
  Select object
  Orientation

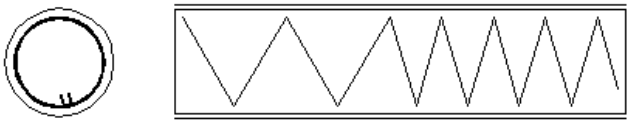
  Definition of reinforcement along the length of reinforced element
  Select first point
  Select second point [Spiral pitch]

where:
a circle is defined within the cross-section by selecting a contour (of a circle or regular polygon) and indicating positions of hooks (orientation)
first point – beginning point of the helix along the length of an RC element
second point – end point of the helix along the length of an RC element
spiral pitch – value of the helix pitch

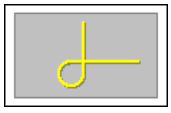
10.9. Example of helix definition

To define a helix (with varying pitch) in a column of round cross-section shown in the figure below, the user should follow the steps below, for example:

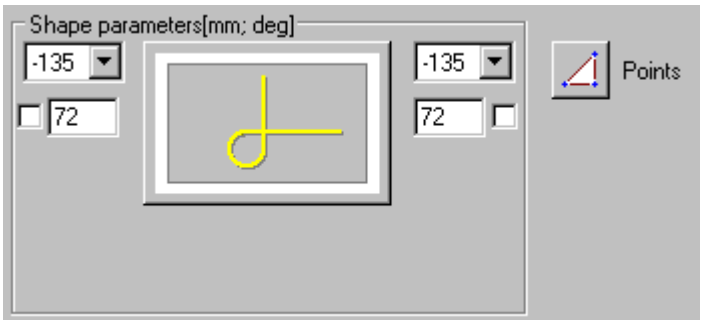
- in the ***Special reinforcement*** dialog box select the reinforcement type - helix
- define the following values in the dialog box (the *Shape parameters* field): angles of hook bend at the bar beginning and end: 90 degrees, hook length 0.06 m
- press the  *Points* icon
- in the drawing indicate circle center and radius (definition of a helix within the column cross-section)
- indicate point 1 presented in the drawing above
- indicate point 2 presented in the drawing above (it completes definition of the helix with default helix pitch over the segment 1-2)
- select the Spiral pitch option (in can be done by entering the letter S into the command line)
- specify a new value of the spiral pitch, e.g.: 200 mm
- indicate point 3 shown in the drawing above; the reinforcement defined is illustrated in the drawing below.



10.10. Vertical loop



Once this type of reinforcement is chosen, the Special reinforcement dialog box assumes the form shown in the figure below. This type of reinforcement may be applied in retaining walls or tanks.



To define a bar, the user should determine the following in the dialog box:

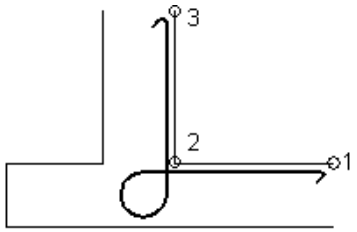
- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)
- in the *Shape parameters* field – anchorage parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining this reinforcement type, only one icon *Points* is available in the right part of the dialog box. After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'line'). During reinforcement definition, the user should specify three snap points of this reinforcement type.

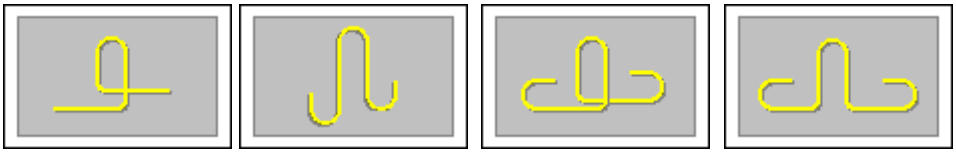
NOTE:
Once definition of this reinforcement type is completed, reinforcing bar parameters can be modified (in particular, a radius of bar bend) by means of the options provided in the Reinforcement / Modify menu (a radius of bar bend can be changed by activating the Reinforcement / Modify / Bent diameters option).

Command line:
Select bar attachment point
Select second bar point
Select bar node
Cover / Side / Mirror

where:
first (attachment point), second and third point as shown in the drawing
cover – determines a cover value for each segment of reinforcement separately
side – changes reinforcement position with respect to an object (formwork)
mirror – mirror reflection of the defined reinforcement with respect to the first segment of reinforcement



10.11. Transport handles



Once any of the transport reinforcement types is chosen, the Special reinforcement dialog box assumes the form shown in the figure below.

To define a bar, the user should determine the following:

- bar diameter (the diameter defined recently in the dialog box is adopted by default)
- reinforcing bar cover (the cover defined recently in the dialog box is adopted by default)
- in the *Shape parameters* field – anchorage parameters for reinforcing bar ends, i.e. hook angle and length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

While defining this reinforcement type, only one icon  *Points* is available in the right part of the dialog box. After pressing this icon, the dialog box closes, whereas the cursor assumes the shape of a cross (see the ACAD command 'line'). During definition of transport reinforcement, the user should specify three or four snap points of this reinforcement type.

Command line:
Select bar attachment point
Select second bar point
Select bar node
If need be, fourth point
Cover / Side / Mirror

where:
first (attachment point), second, third and if need be, fourth reinforcement snap point
cover – determines a cover value for each segment of reinforcement separately
side – changes reinforcement position with respect to an object (formwork)
mirror – mirror reflection of the defined reinforcement with respect to the first segment of reinforcement

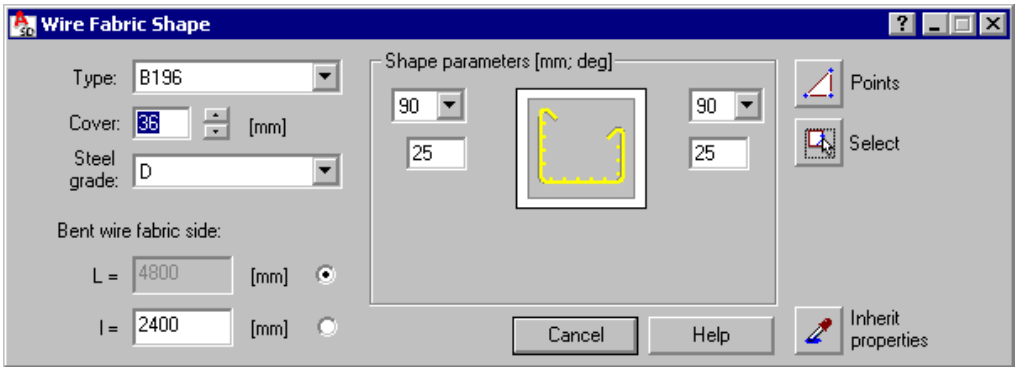
11. WIRE FABRICS IN CROSS SECTION

11.1.Wire fabrics in cross section – wire fabric shape

The option enables definition of a wire fabric in the cross section (of a wire fabric shape) of an RC structure element. The option is available from:

- the menu by selecting the option: *Reinforcement / Wire fabrics in cross section*
- the toolbar by pressing the icon
- the command line: RBCR_DEF_NET_SIDE.

After calling up the *Wire fabrics in cross section* option, first the dialog box used to define a wire fabric shape in the cross section of an RC structure element appears on the screen.



The ***Wire Fabric Shape*** dialog box can be divided into three parts:

- the left-hand side part of the dialog box contains basic information about a wire fabric: wire fabric type, cover and reinforcing steel grade adopted from the settings in the preferences
- the central part of the dialog box holds options for definition of geometrical parameters of a wire fabric
- the right-hand side part of the dialog box includes icons that are used to select a mode of graphic definition of a wire fabric; NOTE: wire fabrics may be defined in the program graphic viewer only after pressing these icons.

The dialog box opens showing the last-defined type of a wire fabric and parameters adopted for it.

The icon located in the bottom right corner is used to assume parameters of the earlier-defined wire fabric.

To define a wire fabric in the cross section it is necessary to determine the following parameters in the above dialog box:

- a wire fabric type (by default the type defined last in the dialog box is adopted)
- a wire fabric cover (by default the cover defined last in the dialog box is adopted)
- a reinforcing steel grade
- a bent wire fabric side (a shorter or longer side of the wire fabric should be chosen)
- in the *Shape parameters* field – parameters of the hook ending of a wire fabric, i.e. a hook angle and length (the hook length defined in the dialog box is a length of the straight segment of the hook).

Below is presented the list of standard values of hook angles (see: the method of measuring the angle of bar bending):

⇒ 0°
⇒ 90°
⇒ 135°

⇒ 180°
⇒ -90°
⇒ -135°
⇒ -180°.

While defining a wire fabric shape in the cross section, the following icons are available in the right-hand part of the dialog box:

Points - pressing this icon starts definition (selection in the graphical viewer) of points that determine the wire fabric shape

Select - pressing this icon allows selection of an object (line, polyline, arc, etc.) whose shape will determine the wire fabric shape.

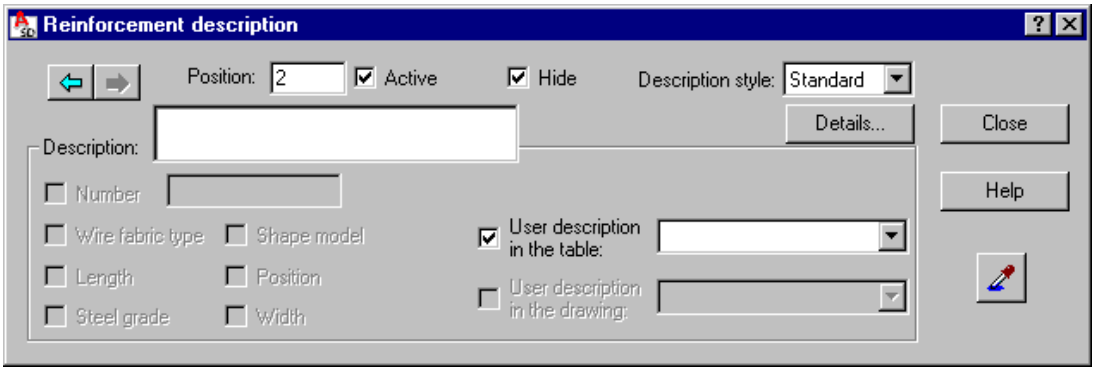
NOTE:
Wire fabric location depends on the direction of point definition. The rule that holds while defining a wire fabric is identical as in definition of a reinforcing bar: it consists in determining the order of points clockwise (along the EXTERNAL part of an object).

Once definition of a reinforcement shape is complete, the Reinforcement description dialog box, used to select elements of reinforcement description, is displayed on the screen.

11.2.Reinforcement description - wire fabric shape

Once definition of a wire fabric shape is complete, the dialog box shown below, used for reinforcement description, is displayed automatically on the screen.

NOTE:
The dialog box below opens after selecting the Modify / Reinforcement description option provided in the menu or pressing the icon ; in this case there are two additional options accessible in the dialog box: Description style (used to change a description style for selected reinforcement) and the ***Details*** *button, pressing which opens the dialog box for modification of the style of the reinforcement description (shape).*

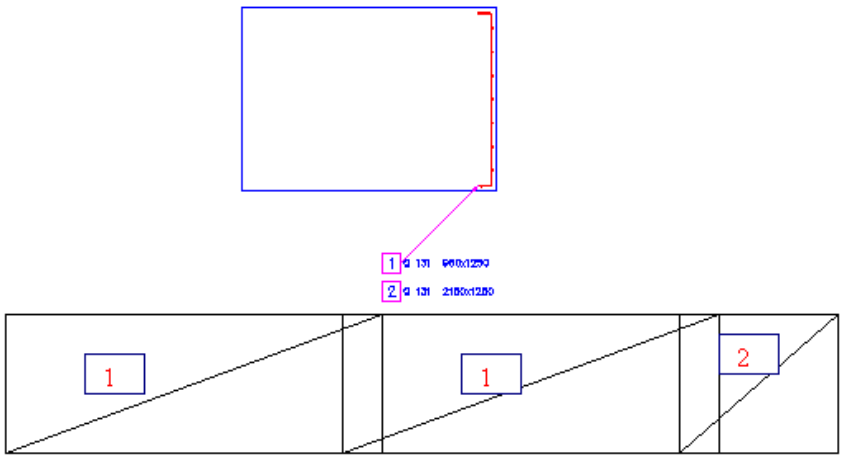


The options in this dialog box enable final selection of a reinforcement description. It can be achieved by switching off active variables initialized on the basis of a defined syntax.

REMARKS:

- description of a reinforcement shape is possible only for a single position, i.e. two or more wire fabrics cannot be described simultaneously

- the *Number* edit field makes it possible to enter from the keybord the final number of wire fabrics that will be included directly in a table
- the *Active* option. If the option is switched on, then the described reinforcement should be considered in the reinforcement table. It means that for reinforcement that is described for the first time this option will be active by default. When the same reinforcement is described for the second time, the option will be switched off on its own. In this way it is possible to prevent (when describing the same reinforcement twice) doubling the number of reinforcement elements calculated when preparing a reinforcement table; a description of active and not active reinforcement may differ in a generated drawing – the options used for this purpose are provided in the ***Description of reinforcement shape*** dialog box
- in the case of the linear distribution, there may be 2 different wire fabrics (with two different numbers) in a region; a description of the distributed wire fabric in cross-section, after its distibution is performed, shows a number of wire fabrics (see the drawing below); while modifying such a description of the wire fabric in cross section, it is possible to hide one of the wire fabrics (the *Hide* option should be used); the arrows allow switching between two wire fabrics included in the distibution



- the fields: *User description in the drawing* and *User description in the table* enable adding any text to a reinforcement description (by entering it from the keyboard); the description will be presented correspondingly in drawings (included in wire fabric descriptions in the drawing) and in the table (included only in the reinforcement table); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; a text taken from the library of standard descriptions may also be applied. These descriptions assume the style of the text describing the reinforcement. The user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are applied. The user description together with an extension line and a label make up one object. Such an object may be edited (translated, rotated); to do that, first use the ***AutoCAD Structural Detailing*** EXPLODE option.

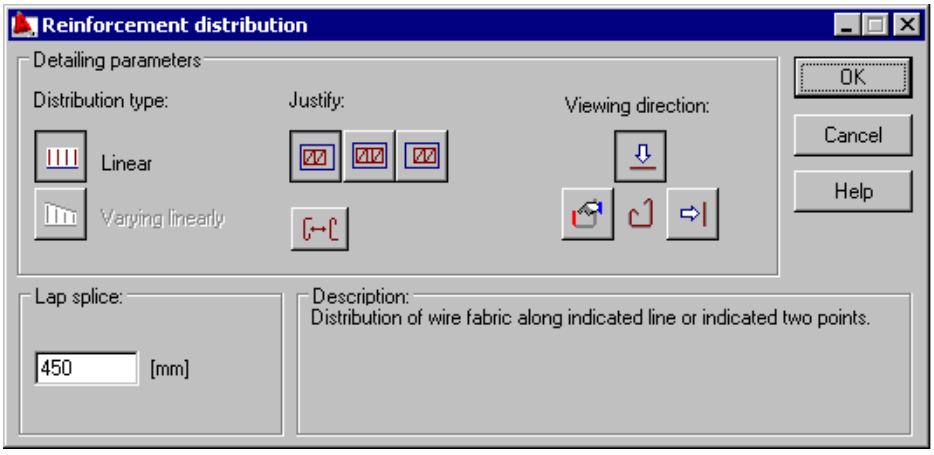
12. WIRE FABRIC DISTRIBUTION

12.1.Wire fabric distribution

The dialog box below is used to define wire fabric distribution. The option is available from:

- menu by selecting the option *Reinforcement / Wire fabric distribution*
- toolbar by pressing the icon
- command line: RBCR_NS_DISTRIBUTION.

Once the *Wire fabric distribution* option is selected, it is necessary to indicate an object whose distribution should be defined.



The options in the left part of the dialog box, in the *Detailing parameters* field, allow determining a type of reinforcement distribution:
- linear (bar distribution along a selected line or indicated two points)
- varying linearly (this option is not available in the current program version).

The following parameters should be defined for the selected distribution type:

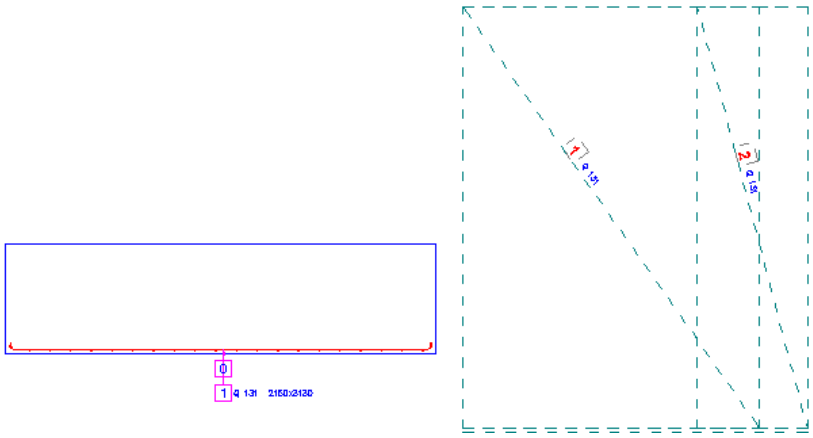
- *Justify* – it determines to which edge created wire fabrics will be justified:
  – justify to 1st edge
  – after selecting this option, one edit field for definition of a lap splice of wire fabrics is inaccessible; it means that automatic lap splice lengths will be increased in such a way so that wire fabrics are justified to both edges without trimming wire fabric sheets
  – justify to 2nd (opposite) edge
- - refers to created 3D cross-sections; using it, you can position wire fabric bars in the cross-section in accordance with the principles of projecting and making cross-sections; this way you can determine the side of the bar (beginning or end) on which the selected bar end types are provided.

Apart from that, it should be determined how a wire fabric is to be presented in distribution:

- top view
- side view
- as an indicated segment; it should be added here, that once this option is selected, a selected bar is distributed, whereas the drawing of the distribution will present a selected segment of the bar in its total dimension (in its current geometry).

The option at the bottom of the dialog box allows defining a value of the lap splice of wire fabrics.

On pressing the **OK** button the dialog box closes and after indicating a line or two points the program perfoms the defined distribution of the wire fabric. The linear distribution will include at the most 2 different wire fabrics (two different wire fabric numbers).
In a description of the distributed wire fabric in cross section, once its distribution is performed, a number of wire fabrics is updated and another – additional label is provided (see the drawing below).

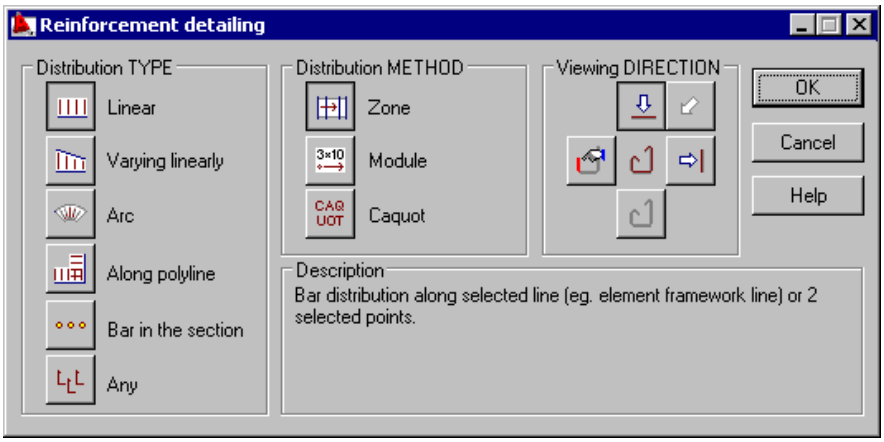


13. REINFORCEMENT DISTRIBUTION - REINFORCING BARS

13.1.Reinforcement distribution

The dialog box below is used to define distribution of reinforcement. The option is available from:

- the menu by selecting the *Reinforcement / Reinforcement distribution* option
- the toolbar by pressing the icon
- the command line: RBCR_DEF_BAR_DISTRIBUTION.



The dialog box may be divided into three primary parts:
In the left part of the dialog box, in the *Distribution type* field the user may determine a type of reinforcement distribution.

- linear - bar distribution along a selected line (e.g. the line of an element formwork) or along two indicated points
- varying linearly - bar distribution within the region indicated (e.g. a formwork element of variable height); the height of a bar being distributed will be adjusted automatically to the formwork shape (prior to distribution, the user should indicate reinforcement segments whose length changes during distribution)
- arc - bar distribution along the indicated arc
- along polyline - bar distribution along the indicated broken line (polyline)
- distribution: reinforcement as a point (bar in a section) - graphical representation of a bar in a section
- any (all options in the right part of the dialog are not available); you can distribute a selected bar or bars; in the latter case, bars are distributed as a group.

Once the distribution type is selected, the method of reinforcement distribution should be defined. The only exception constitutes the point reinforcement which requires neither defining a distribution method nor the direction of viewing the reinforcement. There are the following methods of reinforcement distribution available:

- zone method - a formwork element is divided into several regions (zones); in each of the designated zones, the user will be able to define reinforcement spacing independently

- module distribution - reinforcement spacing is defined by specifying the order of spacing in the following form, e.g. 3x25, 5, 6x15, etc. with respect to the base point, and next, with respect to the bar defined recently in the distribution
- Caquot distribution - a simplified method of reinforcement distribution which consists in specifying the length of distribution as well as initial spacing and maximum spacing; spacing values are calculated based on a simple method maintaining the spacing values obtained as a result of applying the method.

Finally, a manner of presenting the reinforcement distribution should be determined:

- top view
- side view
- as an indicated segment; it should be added here, that once this option is activated, the selected bar is distributed and the distribution drawing shows the actual geometry of the chosen bar segment
- projection
- any.

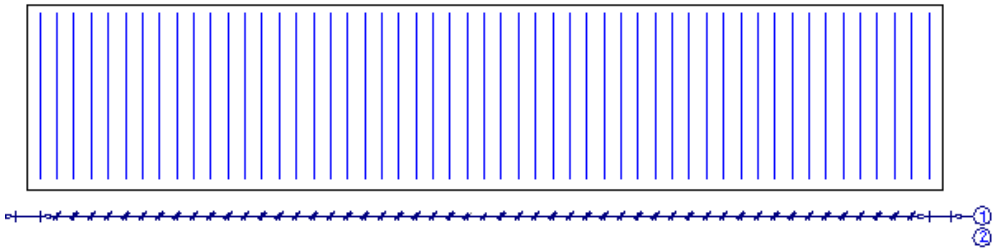
Once the **OK** button is pressed, the dialog box closes and reinforcement is distributed according to the definition.

For the distribution types: linear and varying linearly all the options provided in the dialog box are available. For the distribution type: any, the distribution methods are inaccessible, whereas the viewing direction is available identically as in the case of the linear distribution.
For the bar distribution as a point, both the methods and view direction are unavailable. Once the **OK** button is pressed, the program opens the Reinforcement - point dialog box.
The bottom part of the dialog box will show the description of an option selected in the dialog box.

NOTE:
The current program version offers the possibility of distribution of several reinforcing bars simultaneously. After running the Reinforcement distribution option the user should indicate several bars; the program recognizes automatically a number of bars and distributes them according to the adopted parameters defined by the user (similarly as for a single bar) - see: Distribution of several bars simultaneously.

13.2. Distribution of several bars simultaneously

Distribution of several bars simultaneously is possible only for the linear distribution.



Two bars distributed simultaneously are represented by one line whose length equals the height of the larger stirrup.

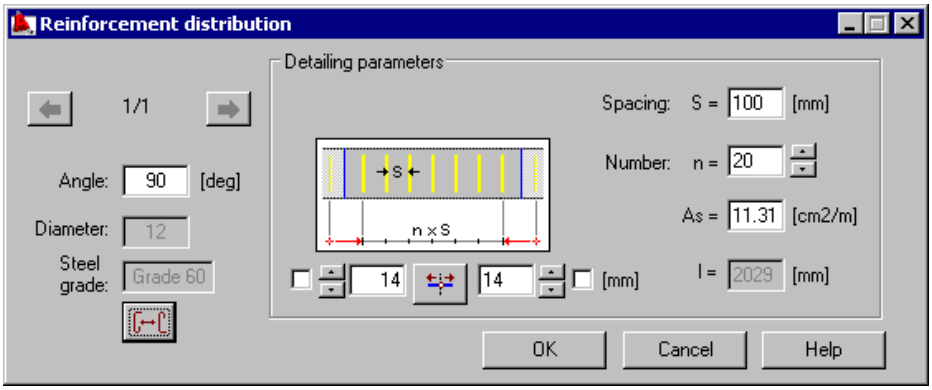
Distribution parameters given by the user are defined as for a single bar. If distributed bars differ in diameter or steel grade, then the fields for selection of these parameters will be left not filled out.
A description of a distribution of several bars simultaneously looks similar to that for a distribution of one bar. If different bars are distributed, then a description of the distribution will include all bar numbers (a number of descriptions will equal a number of distributed bars).

13.3.Linear distribution

After selecting the linear distribution  , three distribution methods are accessible in the Reinforcement distribution dialog box:
zone
module
Caqout.

13.4.Linear distribution (zone)

After beginning definition of linear distribution, the user should determine 2 points or line with respect to which the reinforcement will be distributed. That, on which side of a distribution segment the reinforcement is positioned, depends on the segment sense. The rule of clockwise movement applies here (as during reinforcement definition).
The distance between the bars being distributed and the distribution line is assumed by default to be the cover adopted from the bar shape. A direction of distribution is parallel to the defined distribution line.
If definition of zones is completed, then the program opens the dialog box which is used to manage distribution within each of the defined zones.
The  icon refers to created 3D cross-sections; using it, you can position reinforcing bars in the cross-section in accordance with the principles of projecting and making cross-sections. This way you can determine the side of the bar (beginning or end) on which the selected bar end types are provided.



Distribution parameters may be defined by:

- specifying the value of reinforcement spacing in a given zone
- determining the value of parameter n - it corresponds to the number of reinforcement spacings (NOTE: this value should nor be confused with a number of reinforcement bars which equals n+1)
- determining reinforcement area.

If any of the parameters listed changes, it causes values of the remaining parameters to be updated.

The parameters enumerated above are illustrated by a schematic drawing provided in the dialog box. The options located next to the drawing enable the user to position precisely the distribution of reinforcement in a given zone:

- two edit fields allow defining value of the distance between the extreme reinforcement bars and the zone limit
- the button placed between the edit fields enables centering the distribution
- the options located on both sides of the drawing (selection options) enable determining the element from which the distance to the extreme reinforcement bars is measured (this may be: zone limit or adjacent reinforcement from the previous zone).

Command line

Select distribution type : [Linear / varying Linearly / Arc / Point] <Linear>

Distribution method : [Zone / Module / Caquot] **< Zone >**
Direction of projection - viewing a bar [X / Y / Any / Segment] <Y>
Any – Determine direction of viewing a bar
Segment – Select bar segment
Dialog box closing

Start distribution point or [Line]
End distribution point or [Back]

Beginning of 1 zone or [Side / Cover / Angle / N-zones / Back]
Cover <5.5> : 6

Angle
Reinforcement inclination angle [Select / Points] <90>

Number of zones <1> : 3

End of 1 zone or [Back]

End of 2 zone or [Back]

Next zone or change [Spacing / Number / Alignment]
Spacing <0.12> : 0.18
Number <7> : 6

Alignment: [Center / Left / Right] <Center>

13.5.Linear distribution (module)

After beginning definition of linear distribution, the user should determine 2 points or line with respect to which the reinforcement will be distributed. That, on which side of a distribution segment the reinforcement is positioned, depends on the segment sense. The rule of clockwise movement applies here (as during reinforcement definition).
The distribution may be carried out only along the length of a segment defined. The distribution start point corresponds to the first point indicated while defining a segment or to the closest point during definition with the use of a line.
All operations are carried out ONLY along a selected line within the distribution region. The modular distribution starts with defining a position of a first bar to be distributed (the position may be determined graphically or by entering the value from the keyboard). A unit of the value entered is the unit of the ***AutoCAD Structural Detailing*** program set in preferences. While defining distribution, the following options are available:

- Direction - changes the direction of distribution to the opposite
- Mirror - provides mirror reflection of the distribution already carried out
- Insert between - when distribution is carried out on one side and then, the direction changes and the reinforcement is distributed on the other side, the user may introduce the reinforcement in the area between these both distributions defining maximal spacing or number of bars.

Command line

Select distribution type: [Linear / varying Linearly / Arc / Point] <Linear> :

Distribution method: [Zone / Module / Caquot] **< Module>**
Direction of projection - viewing a bar [X /Y/ Any/ Segment] <Y>
Any – Determine direction of viewing a bar
Segment – Select bar segment
Dialog box closing

Start distribution point or [Line]
End distribution point or [Back]
Position of first bar or [Side / Direction / Cover / Attachment / Cover] <5>:
Cover <5.5> : 6
Angle
Reinforcement inclination angle [Select / Points] <90>

<Number> x <Spacing> or [Direction / Insert between / Mirror / Back]
Insert between
Emax or [Number]
Defined spacing = 22.23 cm
End or [+1 / -1 / Back]

13.6.Linear distribution (Caquot)

Distribution is ALWAYS symmetrical with respect to the length of a segment in which distribution is performed. The designations:
L - length of a segment of distribution
s1- initial spacing
Smax - maximum spacing
n = L/2 - spacing module rounded upwards to a whole value
so =s1/2 - position of the first point at the beginning and end of a distribution line.

The set of available CAQUOT spacing values includes: 7, 9, 11, 13, 16, 20, 25, 35, 60 cm

Command line

Select distribution type : [Linear / varying Linearly / Arc / Point] <Linear>

Distribution method : [Zone / Module / Caquot] < Caquot>
Direction of projection - viewing a bar [X / Y / Any / Segment] <Y>
Any – Determine direction of viewing a bar
Segment – Select bar segment

Dialog box closing

Start distribution point or [select Line]

End distribution point or [Back]
Initial spacing or [Side / Cover / Back] <9>
Maximum spacing <35>

13.7. Distribution varying linearly

After selecting the linear distribution  , three distribution methods are accessible in the Reinforcement distribution dialog box:
zone
module
Caqout.

13.8. Distribution varying linearly (zone)

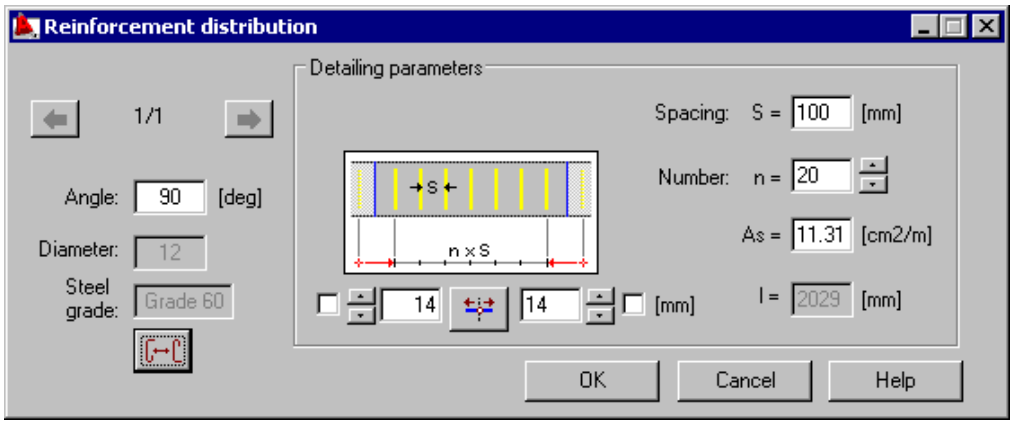
After starting definition of the linear distribution, the user **should determine bar segments whose length changes in the course of distribution**. To do that, the user should cut with a line through bar segments, thus indicating that their length is to be variable in the course of distribution. Next, a region of reinforcement distribution and a cover value (by default, it is adopted from a bar shape) have to be defined.
A direction of reinforcement distribution may be defined by indicating:

- 2 points
- contour edge.

By default, the distribution direction is parallel to 1 defined distribution line (region edge) - it is presented by means of two arrows.
In the next stage, the distribution region is divided into zones. If definition of zones is completed, then the program opens the dialog box used to manage distribution within each of the defined zones.

When the distribution varying linearly is applied, it is necessary to define a region. There are three methods available: Select, Pick point and X-points.
For the X-points method, the region is defined by means of a closed broken line. By default, first two indicated points will determine the distribution line.
When applying Selection (select), a selected edge will be the distribution line (the closest edge, if it is selected by means of a window); it works similarly when the Pick point option is selected: the closest edge from the indicated point is the distribution line.
Once the region is pointed out, the program draws a CONTOUR of the distribution region with a cover included. Points (zone boundaries) may be indicated only within the distribution region. A cover is ALWAYS defined as the distance measured from reinforcement, and perpendicular to the distribution line. The zone length is ALWAYS measured along the distribution line.

The  icon refers to created 3D cross-sections; using it, you can position reinforcing bars in the cross-section in accordance with the principles of projecting and making cross-sections. This way you can determine the side of the bar (beginning or end) on which the selected bar end types are provided.



Distribution parameters may be defined by:

- specifying the value of reinforcement spacing in a given zone
- determining the value of parameter n - it corresponds to the number of reinforcement spacings (NOTE: this value should nor be confused with a number of reinforcement bars which equals n+1)
- determining reinforcement area.

If any of the parameters listed changes, it causes values of the remaining parameters to be updated.

The parameters enumerated above are illustrated by a schematic drawing provided in the dialog box. The options located next to the drawing enable the user to position precisely the distribution of reinforcement in a given zone:

- two edit fields allow defining value of the distance between the extreme reinforcement and the zone limit; for extreme zones the distance value indicates a distance of the extreme bar in the distribution to the zone limit, whereas for intermediate zones this is a distance between the extreme bars of the neighboring zones
- the button placed between the edit fields enables centering the distribution.

Command line

Select distribution type: [Linear / varying Linearly / Arc / Point] **<varying linearly>**
Distribution method: [Zone / Module / Caquot] **< Zone >**
Direction of projection - viewing a bar [X / Y / Segment] <Y>
Segment – Select bar segment

Dialog box closing

Select segment(s) of varying length : *(this text appears ONLY for X and Y)*
Region of reinforcement distribution [Select / Pick point / Define] : <Select>
Define
Start point
Next point or [Cover / Back]
Cover <5>
Next point or [Close / Cover / Back]
Reinforcement distribution direction or [Select / Points / Angle]
Select object

Start point
End point

Angle <90>

Beginning of 1 zone or [N-zones / Back]
Number of zones <1>
End of 1 zone or [Back]
End of 2 zone or [Back]

Next zone or change [Spacing / Number / / Alignment]
Spacing <0.12>
Number <7>

Alignment: [Center / Left / Right] <Center>

13.9.Distribution varying linearly (module)

After starting definition of the linear distribution, the user **should determine bar segments whose length changes in the course of distribution**. To do that, the user should cut with a line through bar segments, thus indicating that their length is to be variable in the course of distribution. Next, a region of reinforcement distribution and a cover value (by default, it is adopted from a bar shape) have to be defined.
A direction of reinforcement distribution may be defined by indicating:

- 2 points
- contour edge.

By default, the distribution direction is parallel to 1 distribution line defined (region edge) - it is presented by means of two arrows.

Afterwards, the user should determine a bar attachment point (by default, the bar is attached in the distance equal to the cover value defined for a bar shape). The distribution start point corresponds to the first point indicated while defining a segment or to the closest point if the line is selected.
A region is defined and the distribution line is assumed in the identical manner as in case of the distribution varying linearly (zone distribution).
Once the region is defined, the program draws a contour with a representative bar. This bar is attached to the cursor and responds to changes in its position. The cursor may move only along the distribution line and determines the position of a newly-defined bar.
The modular distribution starts with defining a position of a first bar to be distributed (the position may be determined graphically or by entering the value from the keyboard). A unit of the value entered is the unit of the ***AutoCAD Structural Detailing*** program set in preferences. While defining distribution, the following options are available:

- Direction - changes the direction of distribution to the opposite
- Mirror - provides mirror reflection of the distribution already carried out
- Insert between - when distribution is carried out on one side and then, the direction changes and the reinforcement is distributed on the other side, the user may introduce the reinforcement in the area between these both distributions defining maximal spacing or number of bars.

Command line

Select distribution type: [Linear / varying Linearly / Arc / Point] **<varying Linearly>**

Distribution method : [Zone / Module / Caquot] **< Module>**
Direction of projection - viewing a bar [X / Y / Segment] <Y>
Segment - Select bar segment

Dialog box closing

Select segment(s) of varying length : *(this text appears ONLY for X and Y)*
Region of reinforcement distribution [Select / Pick point / Define] : <Select>
Define
Start point
Next point or [Cover / Back]
Cover <5>
Next point or [Close / Cover / Back]
Reinforcement distribution direction or [Select / Points / Rotation]

select object

Start point
End point

Angle <90>

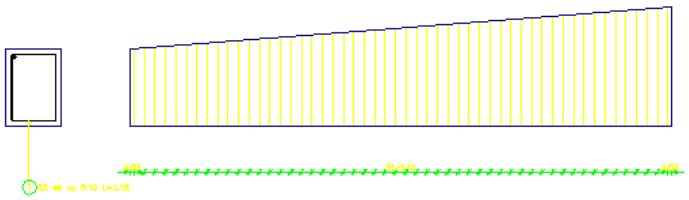
Position of first bar or [Direction] <5>
<Number> x <Spacing> or [Direction / Mirror / Insert between / Back]
Insert between
Emax or [Number]
Defined spacing
End or [+1 / -1 / Back]

13.10. Distribution varying linearly (Caquot)

The distribution type is defined identically as the linear distribution. Only a contour with a distribution line are drawn on the screen.

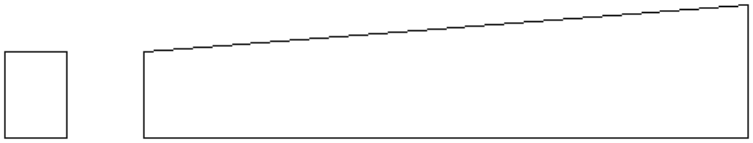
13.11. Generation of distribution varying linearly and detailed table

Definition of distribution varying linearly of transversal reinforcement will be presented below for a tapered beam shown in the figure underneath. For such reinforcement distribution a detailed table will be prepared.

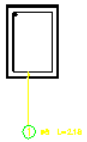


To obtain reinforcement distribution shown in the drawing, the user should:

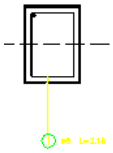
- define transversal and longitudinal beam section (as in the figure below) using the options available in the AutoCAD ® program (rectangle, line)



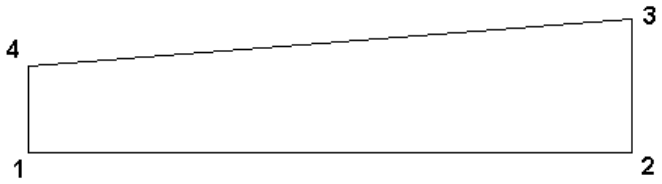
- define transversal reinforcement in the beam cross section
  - press the *Reinforcement – cross-section* icon
  - in the dialog box select the option and press the *Diagonal* icon
  - indicate the top left and bottom right apices of the rectangle designating the cross section
  - in the ***Reinforcement description*** dialog box press the **OK** button (acceptance of the default description)
  - point out the description position in the drawing (see the figure below)



- press the *Reinforcement distribution* icon (it starts definition of reinforcement distribution varying linearly)
- indicate the stirrup defined earlier and press the ENTER key; the ***Reinforcement detailing*** dialog box opens on the screen; the following options may be selected in the dialog box:
  - Distribution TYPE: varying linearly (press the icon)
  - Distribution METHOD: zone (press the icon)
  - Viewing DIRECTION: press the icon
  - press the **OK** button
- determine successive reinforcement segments (in this case these are vertical legs of the stirrup) whose length changes in the course of distribution - see the figure underneath

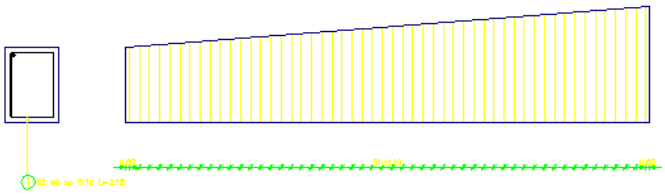


- point out the beginning point and successive points within which the varying distribution is to be contained; indicate one by one the points: 1, 2, 3, 4 and finally indicate 1 again; press the ENTER key



- determine direction of distribution; press the ENTER key
- point out the beginning and end of the zone (begining and end of the beam, if the distribution concerns the whole beam); press the ENTER key
- press the **OK** button in the ***Reinforcement detailing*** dialog box – default values will be adopted
- press the **OK** button in the ***Reinforcement description*** dialog box – default values will be adopted

- indicate the position of reinforcement description and press the ENTER key (see the figure below).



To create a detailed table for the reinforcement distribution varying linearly defined above, the user should:

- press the *Bars –Detailed table* icon
- indicate the generated reinforcement distribution (varying linearly)
- point out the location of the detailed table.

13.12. Arc distribution

This is the linear distribution along a defined / indicated part of an arc; the distribution along an arc is an extension of the linear distribution - this is the reason why a distribution angle is not defined.

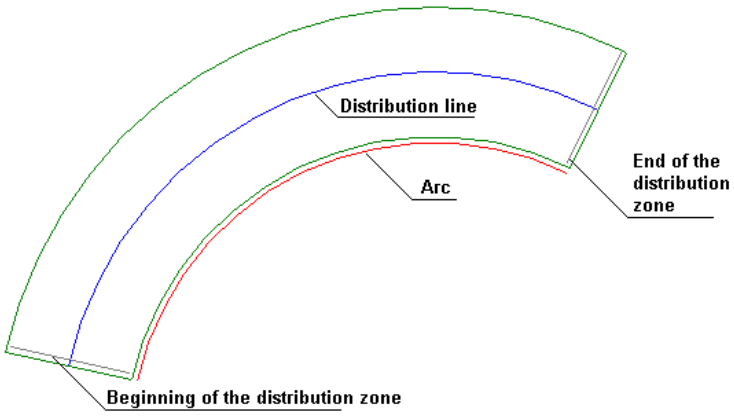
Reinforcing bars in distribution are perpendicular to the distribution line

To define a distribution region along an arc, the user should (see the drawing below):

1. indicate a curve defined using AutoCAD options or indicate the position of three successive points belonging to an arc
2. indicate the distribution line; there are three locations of the distribution line possible: inner, middle and outer.

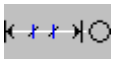
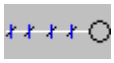
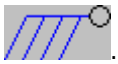
For the indicated distribution line bars in the distribution are arranged according to the parameters adopted in the dialog box.

NOTE:
The location of the distribution line may be changed.



A description of the distribution is an arc-shaped line as for the radial distribution. This line **always** shows the spacing from the distribution line.

Four types of the line describing this distribution are possible:

- 
- 
- 
-  .

First two descriptions are arc-shaped lines, the latter two ones look as for straight distributions.

Command line

Select distribution type: [Linear / varying Linearly / Arc / Point] **<Arc>** :

First point of distribution or [Circle / Select] :

Circle - a circle is selected as a basis for the arc distribution

Select - successive three points belonging to an arc are indicated

Second point of distribution [Side]

Third point of distribution [Side]

Select location of distribution line

Beginning of zone [Side / Cover / N-zones]

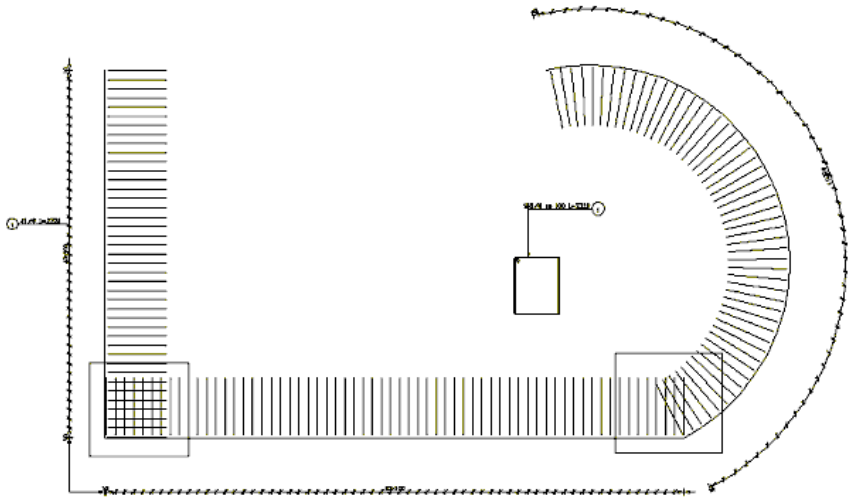
End of zone [Back]

13.13. Distribution along the polyline

Linear distribution along a defined / selected broken line (polyline). Distributed reinforcing bars are perpendicular to the distribution line.

The software generates one reinforcement distribution zone for each segment of a broken line. The distribution of reinforcement is described by one description line.

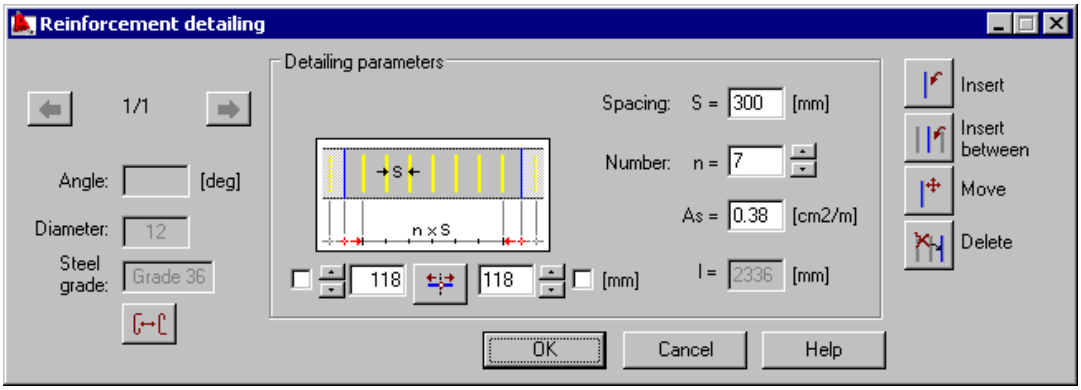
You can modify this reinforcement distribution type exactly as the zone distribution.



13.14. Reinforcement distribution - along polyline

After you define basic parameters of reinforcement distribution the Reinforcement detailing dialog displays. A cover ALWAYS denotes the distance measured from the reinforcement and perpendicular to the distribution line. The zone length is ALWAYS measured along the distribution line.

Using the  icon, you can position reinforcing bars in the cross-section in accordance with the principles of projecting and making cross-sections; this way you can determine the side of the bar (beginning or end) on which the selected bar end types are provided.



You can define the distribution parameters by:

- defining the reinforcement spacing in a given zone
- specifying a value of the n parameter - it corresponds to the number of reinforcement spacings (NOTE: be careful not to confuse this value with the quantity of reinforcement elements which is n+1)
- specifying the reinforcement area.

Changing any of the parameters above results in updating the values of the remaining parameters.

The schematic drawing in the dialog box illustrates the above-mentioned parameters. The options next to the drawing let you precisely position the reinforcement distribution in a given zone:

- use the two edit fields to specify the distance between the extreme reinforcement and the zone border; for extreme zones the distance value denotes the distance from the extreme bar in the distribution to the zone border, while for middle zones it is the distance between the extreme bars of adjacent zones
- use the button between the edit fields to center the distribution.

There are the following icons in the right part of the dialog:

 Insert - inserts a reinforcing bar at the selected location

 Insert between - inserts a reinforcing bar between 2 selected reinforcing bars

 Move - moves a reinforcing bar; you should specify a translation vector

 Delete - deletes a selected reinforcing bar.

13.15. Any distribution

After selecting the any distribution of bars, the ***Reinforcement detailing*** dialog closes.

To define the any distribution of bar:

- select the base point of distribution (this point does not have to be associated with the selected bar)
- select the attachment (insertion) point for the distribution.

Click Enter to finish defining the distribution.

After completing a definition of the distribution, the dialog displays where you can specify parameters of the distribution description. The options for the distribution description are the same as for the description of point reinforcement.

14. SURFACE REINFORCEMENT DISTRIBUTION - WIRE FABRICS

14.1. Surface distribution - wire fabrics

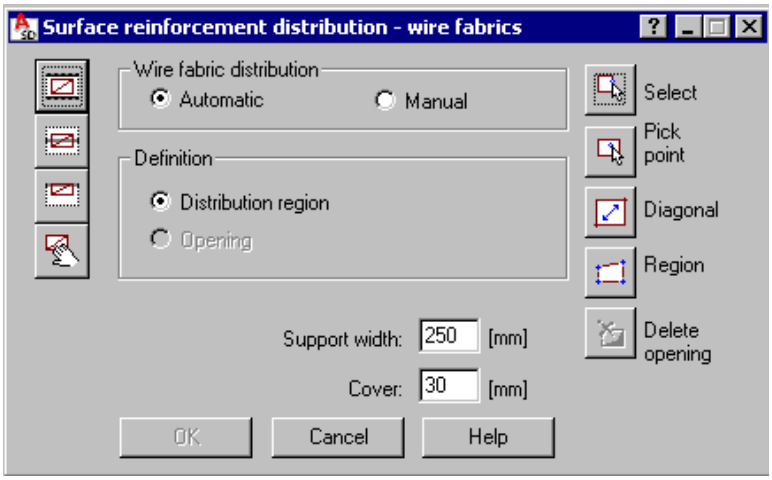
The option allows defining distribution of wire fabrics for 2D contours (e.g. RC slab). The option is available from:

- the menu by selecting the *Reinforcement / Surface reinforcement – wire fabrics* option
- the toolbar by pressing the icon
- the command line: RBCR_NETD_RECT.

NOTE:
In the program a total number of wire fabrics (panels) is calculated; cuttings obtained while trimming wire fabrics are not considered in reinforcement tables.

NOTE:
In the program databases of wire fabrics (.mdb files) used in the surface distribution are made available. The file name, for example for the British code, is fabric_BS.mdb (after the part 'fabric', the name of the code chosen in the Job preferences is added). Database files are available in the DATA folder in* ***AutoCAD Structural Detailing****. Wire fabric databases may be freely modified in order to adapt wire fabric parameters to the user needs.*

Once the *Surface reinforcement - wire fabrics* option is selected, the dialog box shown in the figure below is displayed on the screen.



NOTE:
A defined contour is a common contour for generation of reinforcing bar distribution and wire fabric distribution (bar distribution may be generated in the contour for wire fabrics and vice versa).

The dialog box may be divided into three main parts:
The left part of the dialog box contains four icons which determine the type of distribution:

- **Distribution A** - wire fabric distribution within a span (within the defined contour e.g. plate span or wall)

 - **Distribution B** - wire fabric distribution above the intermediate support (with respect to the support axis)

 - **Distribution C** - wire fabric distribution above the extreme support (along the support edge)

 - **Distribution D** - any manner of wire fabric distribution (without the necessity to define a contour) without trimming wire fabrics.

There are two modes of defining surface distribution available in the program:

- automatic mode – an indicated contour is detected automatically and wire fabrics are trimmed to fit the contour
- manual mode – wire fabrics are distributed manually and timmed automatically to fit the contour.

For both modes identical methods of wire fabric definition - as described below – are available in the program.

In the middle part of the dialog box the parameters are provided which are indispensable while defining a selected distribution type:

Distribution A

The top part of the dialog box includes options used for definition of a distribution region or opening(s); moreover, there are edit fields available which enables defining values of cover for wire fabrics and support width.

When starting definition of a distribution contour, the *Opening* option is inaccessible, whereas the *Distribution region* option is active and selected. Once the contour definition is completed, the *Distribution region* option is no longer active, whereas the *Opening* option becomes active and selected. Then the user may define an opening contour within the earlier-defined contour; a number of openings may be freely determined by the user.

Distribution B

The following options are provided in the middle part of the dialog box:

- edit field used for defining the support width
- edit fields which allow defining the value informing how far the wire fabric extends outside the support face (in both directions).

Distribution C

The following options are provided in the middle part of the dialog box:

- edit field enabling definition of cover value for wire fabrics
- edit field used for defining the support width
- edit field which allows defining the value informing how far the wire fabric extends outside the support face.

Distribution D

The options in the middle part of the dialog box are inaccessible.

There are two reasons for defining support width:

- contour within which wire fabric is to be distributed is an external contour of a slab or wall; definition of a support width value models the support (wall) – within the support region a wire fabric will not be generated
- after determining the support width, the program defines automatically overhangs for it – when defining reinforcement above supports (distribution B or C), they are recognized automatically.

The right part of the dialog box contains several icons which allow the user to determine a region of wire fabric distribution:

for **Distribution A**

- *Select* (selects directly a region defined as rectangle, polygon, circle or indicates the existing defined distribution region)
- *Pick point* (indicates an internal point for a closed region; as a result of the operation, the region contour is detected)
- *Diagonal* (determines rectangular region by means of defining its diagonal)
- *Region* (defines closed region by means of a broken line)
- *Delete opening* (allows deleting an earlier-defined opening contour)

for **Distribution B** and **C**

- *Select* (selects directly one edge of a region defined as line or support)
- *2 Points* (indicates two points determining the axis of a rectilinear support)

In the dialog box, the user defines dimensions (apart from a cover value and width support) which inform how far a wire fabric extends outside the support face.

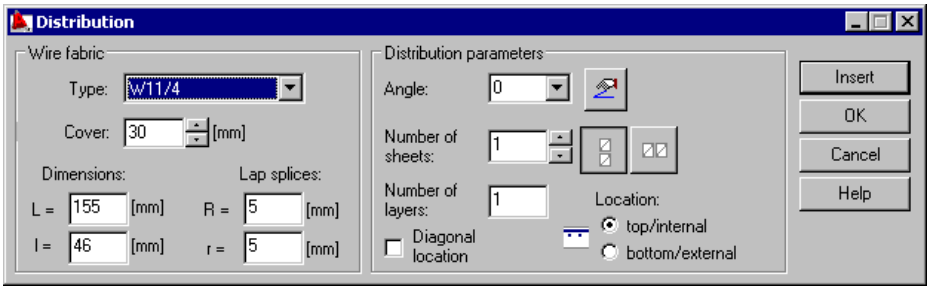
for **Distribution D**
Icons are inaccessible. Pressing the **OK** button results in switching to distribution definition (without definition of a region).

NOTE:
*If **distribution A, B** or **C** is selected, then it is required to define a region in which wire fabrics will be distributed; therefore, the **OK** button is unavailable until the region is defined. The necessity of defining the region is imposed by the subsequent mode of wire fabric definition; wire fabrics distributed one by one are moved to the edges of a defined region or trimmed and adjusted to the region contour.*

Once the **OK** button is pressed, the program opens the Wire fabric distribution – manual mode or the Wire fabric distribution – automatic mode dialog box.
The bottom part of the dialog box will present a description of the option selected in the dialog box.

14.2.Wire fabric distribution - manual mode

The dialog box shown in the figure below is displayed on the screen after choosing the manual mode of wire fabric definition and pressing the **OK** button in the Surface reinforcement distribution – wire fabrics dialog box.



The options contained in the dialog box are used to parametrize distribution in the region defined:

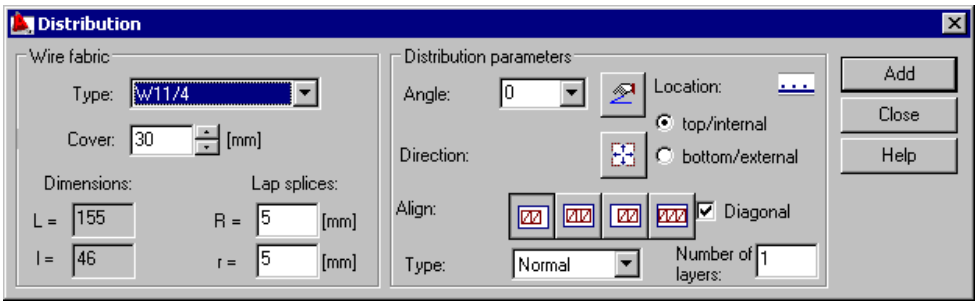
- in the *Wire fabric* field the user may select a wire fabric type from the list; contents of the list depend on the RC code selected in the Job preferences dialog box (similarly as for reinforcing bars); once the wire fabric type is selected, the fields are filled automatically with

data from the fabric_xxxx.mdb file (where xxxx stands for a name of wire fabric database) located in the DATA folder in ***AutoCAD Structural Detailing*** (NOTE: **there is a possibility of modifying the wire fabric database or adding user-defined types of wire fabrics in the file of wire fabric database - new records must must be given identical descriptions as records existing in the database**):
wire fabric cover
L and l - total dimensions of a wire fabric sheet (the dimensions may only be decreased - they cannot be increased)
R and r - lap slices; these dimensions are ascribed to a given wire fabric and saved in the wire fabric database

- in the *Distribution parameters* field the following options may be defined:
  Angle - the selection list containing angle values (the angle value may be changed); pressing the  icon enables reading the inclination angle of the indicated edge of a region contour directly from a drawing
  Number of sheets- this option enables the user to insert simultaneously several wire fabric sheets; if a number of sheets is greater than 1, then the following become accessible: two icons  and  (they allow determining how the sheets are to be positioned with respect to each other) *Number of layers* - information only for the needs of the subsequent wire fabric table
  Location (*top/internal, bottom/external*) - parameter describing location of wire fabrics; it affects graphical representation of wire fabrics (see Job preferences).

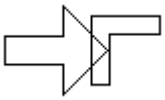
14.3.Wire fabric distribution - automatic mode

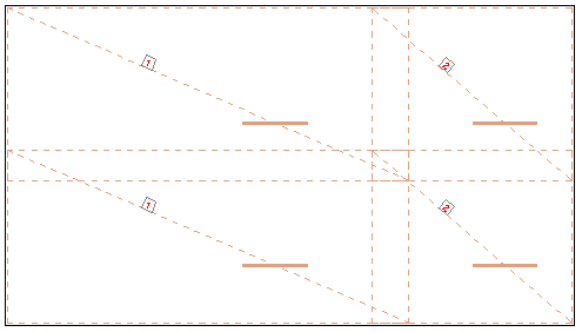
The dialog box shown in the figure below is displayed on the screen after choosing the automatic mode of wire fabric definition and pressing the **OK** button in the Surface reinforcement distribution – wire fabrics dialog box.



The options contained in the dialog box are used to parametrize distribution in the region defined:

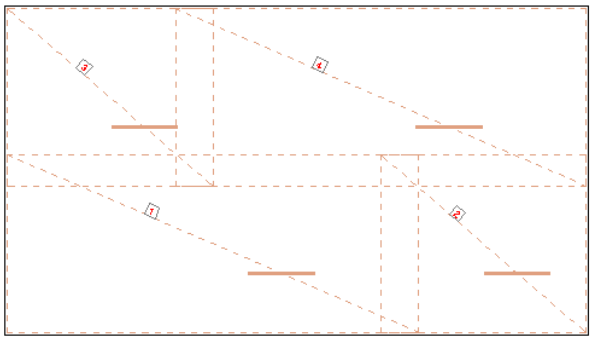
- in the *Wire fabric* field the user may select a wire fabric type from the list; contents of the list depend on the RC code selected in the Job preferences dialog box (similarly as for reinforcing bars); once the wire fabric type is selected, the fields are filled automatically with data from the fabric_xxxx.mdb file (where xxxx stands for a name of wire fabric database) located in the DATA folder in ***AutoCAD Structural Detailing*** (NOTE: **there is a possibility to modify the wire fabric database or add user-defined types of wire fabrics in the file of wire fabric database - new records have to be given identical descriptions as records existing in the database**):
  wire fabric cover
  L and l - total dimensions of a wire fabric sheet (the dimensions may only be decreased - they cannot be increased)
  R and r - lap slices; these dimensions are ascribed to a given wire fabric and saved in the wire fabric database
- in the *Distribution parameters* field the following options may be defined:

Angle - the selection list containing angle values (the angle value may be changed); pressing the  icon enables reading the inclination angle of the indicated edge of a region contour directly from a drawing
Direction – indicates the contour side from which wire fabric distribution will start; pressing the  icon enables changing the direction wire fabric distribution; the side from which wire fabric distribution will begin, is identified in a drawing by means of the  symbol
Align – determines the side with respect to which the generated wire fabrics will be aligned:
 – aligns to 1st edge
 – after selecting this option, one edit field for defining a lap splice of wire fabrics becomes inaccesible; it means that lap splice lengths will be increased automatically in such a manner so that wire fabrics are aligned on both sides without the necessity to trim wire fabric sheets
 - aligns to the 2nd (opposite) edge
 - aligns to both edges; after you select this option, the edit fields for defining the lap splice of wire fabrics become inaccessible
after selecting the *Diagonal* option, the location of the diagonal changes in the wire fabric distribution (if the diagonal is displayed in the distribution)
Number of layers - information only for the needs of the subsequent wire fabric table
Location (*top/internal, bottom/external*) - parameter describing location of wire fabrics; it affects graphical representation of wire fabrics.
Grouping – if this option is switched on, then wire fabrics of identical parameters positioned next to each other (i.e. with their edges touching) are presented as one object; it is an operation aimed at improving transparency and readability of a whole drawing
Type – allows selecting one out of three possibilities:
normal type – shown in the drawing below

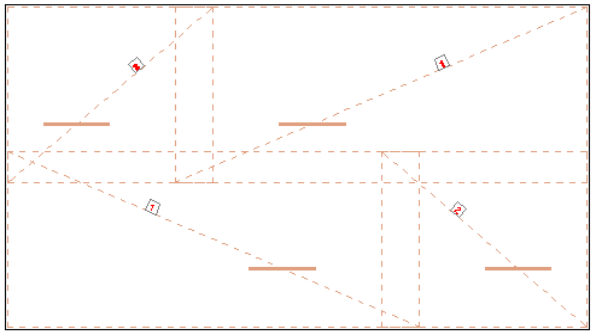


passing and half-way types - wire fabrics are shifted in relation to each other by half the length (to avoid concentration of lap splices of several wire fabrics at the same point) – see the drawings below

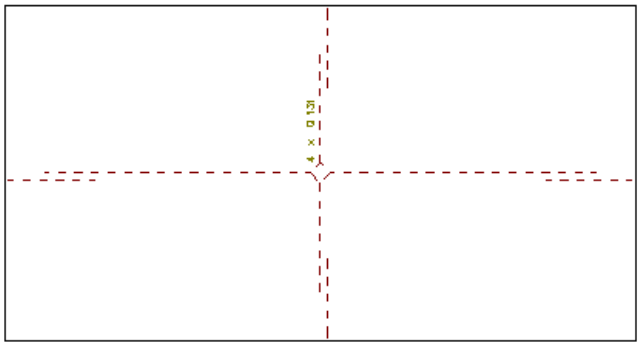
half-way type



passing type

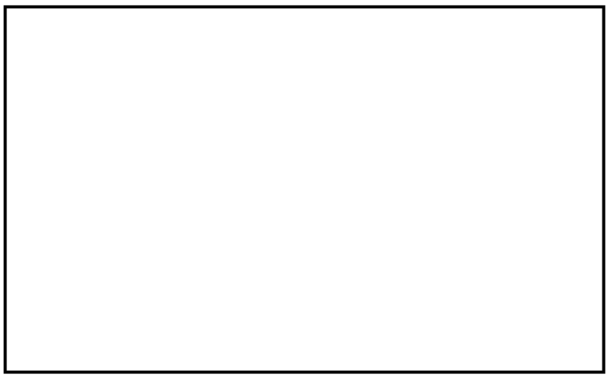


There is also a possibility of simplified presentation of wire fabrics; the drawing below shows the simplified method of presenting a generated wire fabric.



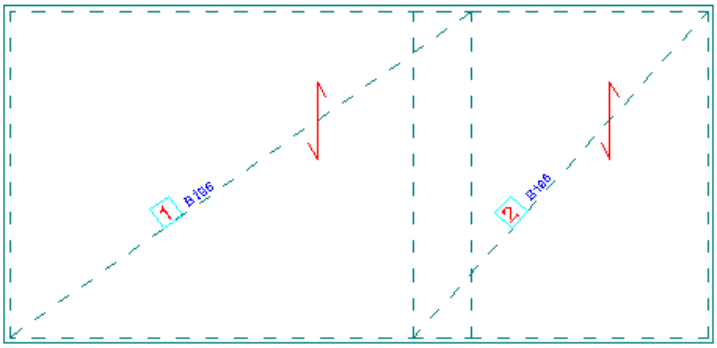
14.4. Example of definition of wire fabric surface distribution

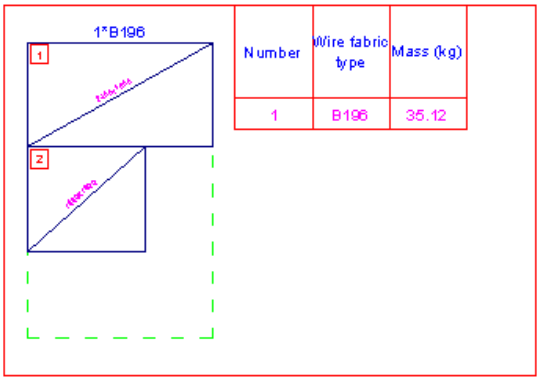
The example steps needed to define the surface distribution of wire fabrics in the slab illustrated in the drawing below are as follows:



- run the menu option: *Reinforcement / Surface reinforcement - wire fabrics* or press the icon
- in the ***Surface reinforcement distribution – wire fabrics*** dialog box determine the parameters listed below:
  - wire fabric distribution: automatic
  - distribution type:
  - support width = 250 mm
  - cover = 30 mm
- press the *Pick point* icon in the right-hand part of the dialog box and indicate a point located within the slab contour
- in the ***Surface reinforcement distribution – wire fabrics*** dialog box select the *Distribution region* option and press the **OK** button
- in the ***Distribution*** dialog box determine the parameters listed below:
  in the *Wire fabric* field:
  - type: B196
  - lap splices: R = 450 mm, r = 300 mm

  in the *Distribution parameters* field:
  - angle = 90
  - location: top/internal
  - align: to 1st edge
  - type: normal
  - *Grouping* option: switched off
  - number of layers: 1
- press the **Add** button; the generated surface distribution of wire fabrics is shown in the drawing below (a summary table for wire fabrics has been added: the menu option *Reinforcement / Reinforcement table / Wire fabrics – summary table* or the icon).





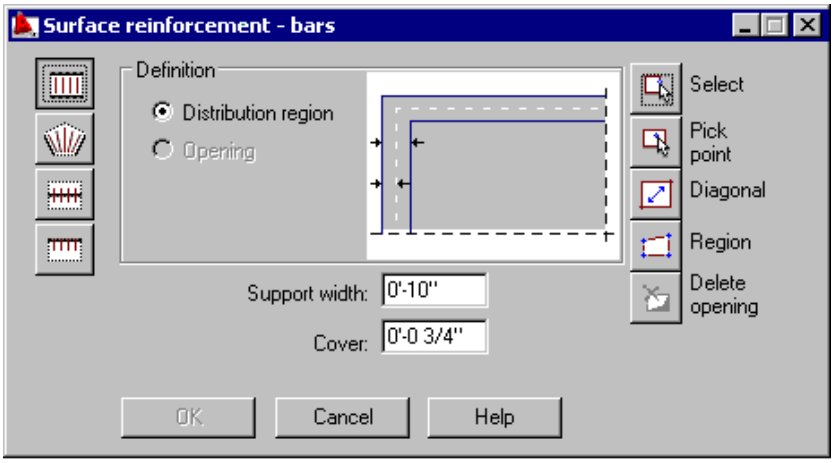
15. SURFACE REINFORCEMENT DISTRIBUTION - BARS

15.1. Surface distribution - bars

The option enables defining reinforcement distribution for reinforcing bars for 2D contours (e.g. wall or RC slab). The option is available from:

- the menu by selecting the *Reinforcement / Surface reinforcement – bars* option
- the toolbar by pressing the icon
- the command line: RBCR_DEF_BAR_SURF.

Once the *Surface reinforcement - bars* option is selected, the dialog box shown in the figure below is displayed on the screen.



NOTE:
In the program there are available databases of reinforcing bars (.mdb files) used in the surface distribution. The file name, for example for the British code, is - bar_BS 8666_2000.mdb (after the part 'bar', the name of the code chosen in the Job preferences is added). Database files are available in the folder ROBOT Office Common / Data / Reinf. Bar databases may be freely modified in order to adapt bar parameters to the user needs (for modification use the MS Access 97 © program).*

While defining distribution the program generates an object contour (e.g. plate, wall with supports); it is determined as a closed external contour. Into the defined region objects such as supports or unsupported edges may be introduced. Apart from that, the user may define (or remove) an opening in a defined region.

NOTE:
A defined contour is a common contour for generation of reinforcing bar distribution and wire fabric distribution (bar distribution may be generated in the contour for wire fabrics and vice versa).

There are two reasons for defining support width:

- contour within which reinforcement is to be distributed is an external contour of a slab or wall; definition of a support width value models the support (wall) – within the support region reinforcement distribution will not be generated

- after determining the support width, the program defines automatically overhangs for it – when defining reinforcement above supports (distribution B or C), they are recognized automatically.

The dialog box may be divided into three main parts:
The left part of the dialog box contains four icons which determine the type of distribution (type of region):
 - **Distribution A** – surface distribution of bars within a defined contour (e.g. span of slab or wall)
 - **Distribution B** – surface distribution of bars above the intermediate support (with respect to the support axis)
 - **Distribution C** - surface distribution of bars above the extreme support (along the support edge).
 - **Distribution D** - surface distribution of bars within a defined contour (a distribution region is defined as distribution A, whereas bars are distributed radially ('fan-shaped' arrangement) between both edges).
The middle part of the dialog box includes parameters which are indispensable while defining a selected distribution type:

Distribution A
The top part of the dialog box contains options used for definition of a distribution region or opening(s); moreover, additionally, there is edit fields available which enable defining support width values and reinforcement cover.
When starting definition of a contour of reinforcement distribution, the *Opening* option is inaccessible, whereas the *Distribution region* option is active and selected. Once definition of a contour is completed, the *Distribution region* option is no longer active, whereas the *Opening* option becomes active and selected. Then the user may define an opening contour within the earlier-defined contour; a number of openings may be freely determined by the user.

Distribution B
The following options are provided in the middle part of the dialog box:
- edit field used for defining support width
- edit fields allowing definition of a value indicating how far the reinforcement extends outside the support face (in both directions).

Distribution C
The following options are provided in the middle part of the dialog box:
- edit field enabling definition of a reinforcement cover value
- edit field used for defining support width
- edit field allowing definition of a value indicating how far the reinforcement extends outside the support face.

Distribution D
Definition of a distribution region proceeds as for distribution A, assuming that the radial distribution is distribution in one zone; bars are distributed radially ('fan-shaped' arrangement) between two edges. The bar spacing is constant only in one line (referred to as a distribution line).

The right part of the dialog box contains several icons which allow the user to choose the mode of graphical definition of distribution. The number of icons depends on a selected type of surface distribution.

for **Distribution A**
 - *Select* (direct selection of a region defined as rectangle, polygon, circle or indication of an existing region)

Command line:
Select object
Distribution direction
Select / 2Points

where:
select object – indicates an object defining the contour of distribution region
select – indicates the contour side determining distribution direction
2Points – indicates 2 points that determine distribution direction

 - *Pick point* (indicates an internal point of a closed region; as a result of the operation, the region contour is detected)

Command line:
Pick object internal point
Distribution direction
Select / 2Points

where:
object internal point – indicates a point positioned within the object defining a contour
select – indicates the contour side determining distribution direction
2Points – indicates 2 points that determine distribution direction

 - *Diagonal* (defines a rectangular region by determining its diagonal)

Command line:
First corner
Second corner
Distribution direction
Select / 2Points

where:
first, second corner – defines the opposite apexes of a rectangle defining the contour
select – indicates the contour side determining distribution direction
2Points – indicates 2 points that determine distribution direction

 - *Region* (defines a closed region by means of a broken line) – in this definition mode supports with different width values on contour edges may be defined

Command line:
First point
Next points [Cover / Support width]
Distribution direction
Select / 2Points

where:
first, second, …, next point – defines apexes of a polygon defining the contour
cover – determines cover values for each contour edge separately
support width – determines support width values for each contour edge separately
select – indicates the contour side determining distribution direction
2Points – indicates 2 points that determine distribution direction

 - *Delete opening* (allows deleting an earlier-defined opening contour)

for **Distribution B** and **C**

 - *Select* (selects directly one edge of a contour defined as line or support)

Command line:
Select support
Continue or change [Side / Support width / First overhang / Second overhang]

where:
select support – indicates a line denoting support of a slab or wall
side – determines on which support side reinforcement is to be located (refers to distribution C)
support width – determines a support width value
first and second overhang – determines an overhang value on both sides of the support (for distribution C – on one side of the support)

 - *2 Points* (indicates two points determining the axis of a rectilinear support)

Command line:
Select first point
Select second point or [Side / Support width / First overhang / Second overhang]

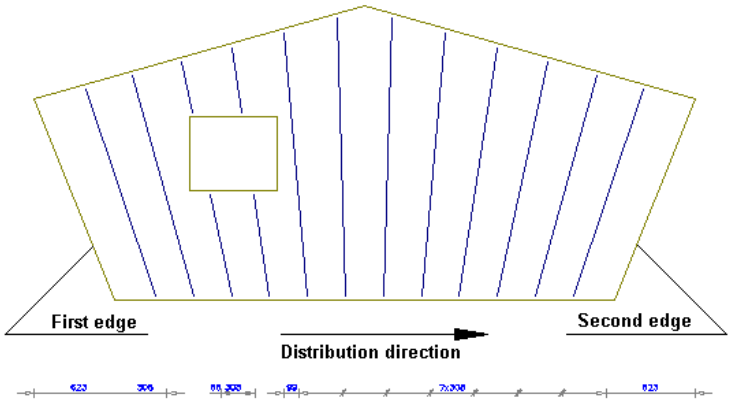
where:
first and second point – indicates first and second point of a line denoting the support of a slab or wall
side – determines on which support side reinforcement is to be located (refers to distribution C)
support width – determines a support width value
first and second overhang – determines an overhang value on both sides of the support (for distribution C – on one side of the support)

Furthermore, for distribution C it should be determined on which side of a support the reinforcement is to be located.
In the dialog box, the user defines dimensions (apart from a cover value and support width) informing how far the reinforcement extends outside the support face. One or more supports may be selected.

for **Distribution D**
The list of icons located in the right-hand side of the dialog box is identical as for distribution A.

Command line:
Distribution direction [Select / 2Points]
first edge
second edge
location of a distribution line (when determining the location of a description)

where:
distribution direction:
2 points – after indicating the points the program draws a line joining these points, which at the same time is the line of constant distribution; bars are distributed radially (‘fan-shaped’ arrangement) between two edges intersected by this line
select – indicate two edges; once they are selected, the program draws a line joining the centers of these edges, which at the same time is the line of constant distribution of bars.
first, second edge – lines that limit the radial (‘fan-shaped") distribution
location of a distribution line – a dimension line of the distribution description which shows a bar spacing in the distribution line.
Below are presented elements needed to define the radial (‘fan-shaped") distribution for an example contour with the distribution.



After pressing the **OK** button, the Reinforcement: definition and detailing dialog box opens on the screen.

NOTE:
For surface distribution of a bar a DETAILED table is also available: table with total numbers of reinforcing bar consumption split into individual bar diameters.

NOTE:
Description of each reinforcing bar of the surface bar distribution may be provided outside the formwork contour (the menu option: RCBR / Reinforcement – bar legend or the Reinforcement – bar legend icon).

15.2. Reinforcement - definition and detailing

The dialog box consisting of the following tabs opens on the screen after pressing the **OK** button in the Surface reinforcement - bars dialog box:
Reinforcement
Shape
Openings
Reinforcement detailing
Detailing options
Reinforcement lap.

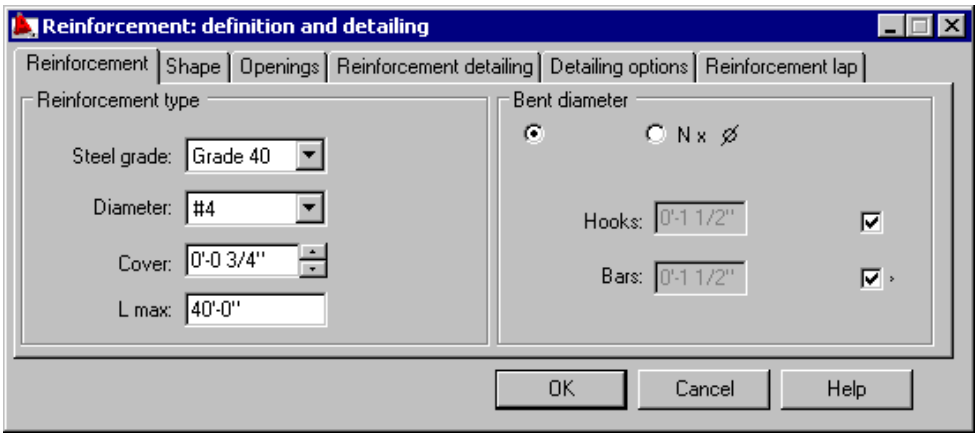
Once definition of a region is completed, the program suggests automatically a direction of distribution; the direction may be subject to modifications (by selecting an edge or by defining two points).
Reinforcement distribution is always perpendicular with respect to the distribution direction. Length of bars in the distribution is adjusted to the shape of a region contour considering the cover and occurrence of openings, if need be.

The distribution is divided automatically into sectors (in case of a fairly complicated contour shape). Reinforcement in each of the sectors is described separately.

After definition of reinforcement and its distribution within the defined region, the Reinforcement description dialog box opens on the screen.

15.3.Reinforcement

The dialog box assumes the form shown in the drawing below after selecting the *Reinforcement* tab in the Reinforcement: definition and detailing dialog box.

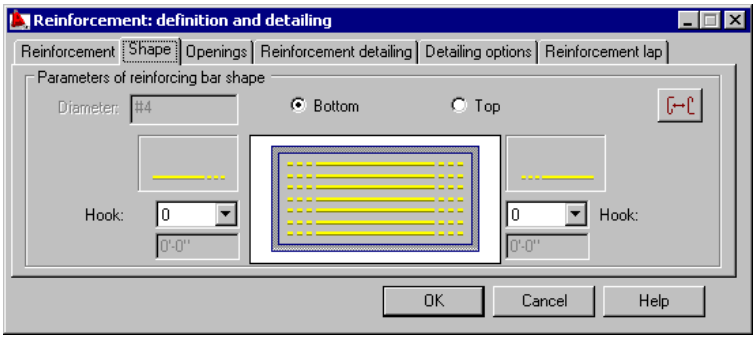


Specify the following parameters in this dialog:

- *Reinforcement type* field
  - steel grade
  - reinforcement diameter
  - reinforcement cover
  - maximum length of reinforcing bars
- *Bent diameter* field
  use this field to define bent diameters for bars and hooks: specify the diameter as a value (in units) or as the multiple of a bar diameter
  to ensure that all generated bars or hooks in the distribution have the same bent diameters and equal values are entered in the *Bars* and/or *Hooks* fields, select the option (√) next to one and/or both these fields; NOTE: after selecting these options the *Bars* and *Hooks* fields are not available.

15.4.Shape

Select the *Shape* tab in the Reinforcement: definition and detailing dialog to display the following options.



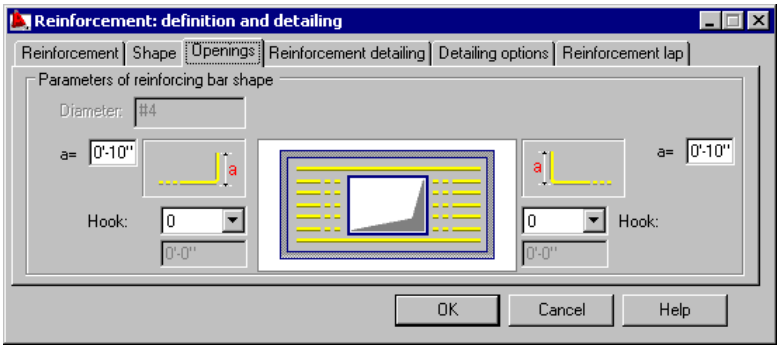
Specify the following parameters in this dialog:

- *Parameters of reinforcing bar shape* – use this field to define a reinforcement shape
  select the *Position* option to place the defined reinforcement in the top or bottom part of a slab

click  to change the orientation of asymmetrical reinforcement; it allows you to specify on which side of the bar (beginning or end) the selected types of bar ends are located
you can also select overall dimensions
use the options at the bottom of the dialog to select geometry of bar ends (note that bar ends always have dimensions as defined in the dialog, whereas a length of the middle part of bars can be changed while defining a distribution); click the icon representing a bar end type to open the ***Shape definition*** dialog where you can select a bar end; using this dialog you can specify values of dimensions for the bar end; options in this dialog allow you to assign compatible bar end types (top and bottom); moreover, you can define hook ends with specified angles for bars.

15.5.Openings

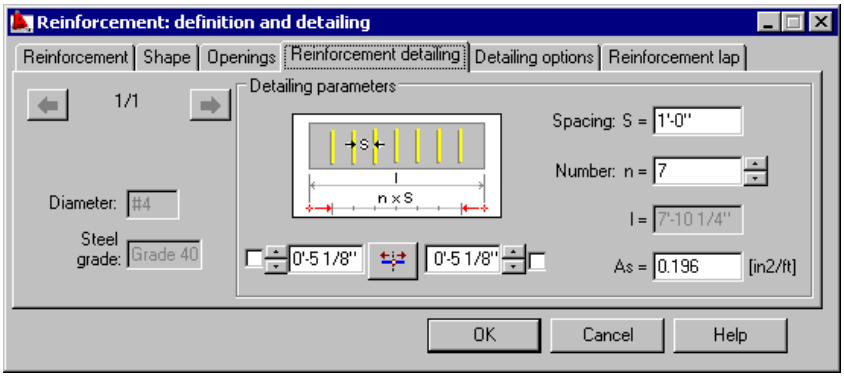
Select the *Openings* tab in the Reinforcement: definition and detailing dialog to display the following options.



Use this dialog to specify parameters for bar ends near openings. The same end types as for other bars are available for bars that adjoin to openings (see the Shape tab).
If reinforcing bars pass through an opening, they are divided into 2 parts (on 2 sides of the opening). You can define an end type for each part of a divided bar; thus bar ends on both sides of the opening may differ.
Note that a bar may adjoin to the opening with one end and with the other - to the slab edge; ends of such bar are defined on 2 tabs (*Openings* and *Shape*) then.

15.6.Reinforcement detailing

The dialog box assumes the form shown in the drawing below after selecting the *Reinforcement detailing* tab in the Reinforcement: definition and detailing dialog box.



The options contained in the above dialog box are used for defining parameters of reinforcement distribution.

The distance between the distributed bars and distribution line is assumed by default to be the cover adopted from the bar shape. The direction of distribution is parallel to the defined distribution line.

Distribution parameters may be defined by:

- defining a value of reinforcement spacing (this is the basic quantity and the basis for calculation of the remaining quantities)
- determining a value of n parameter – it corresponds to the number of reinforcement spacings (NOTE: this value should not be confused with the reinforcement number which equals n+1)
- determining reinforcement area.

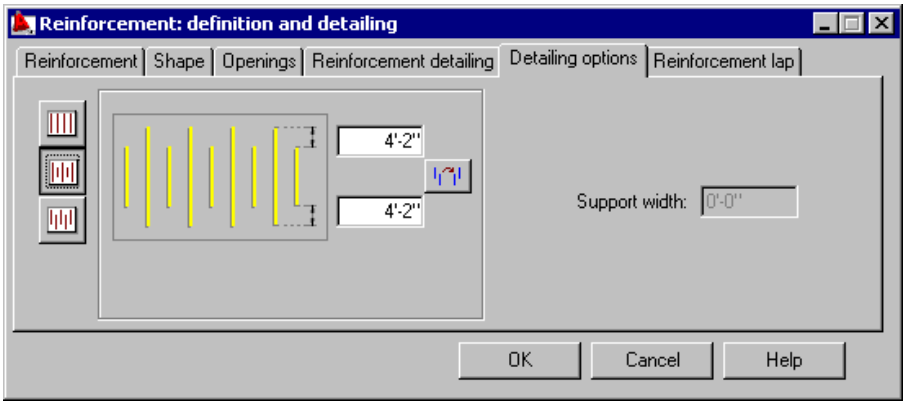
Change in any of the parameters listed results in updating the values of the remaining parameters.

A schematic drawing provided in the dialog box illustrates the parameters enumerated above. The options located next to the drawing enable the user to place accurately the reinforcement distribution in a given zone:

- two edit fields allow determining a value of the distance between the extreme reinforcement and the zone border; there is a possibility to block the distance value for each zone – once the distance on one side is blocked, the distance value on the other side will be automatically adjusted to a specified number of bars and a spacing value; after blocking the distance on both sides, the spacing will be adjusted automatically to a given number of bars
- the button located between the edit fields enables centering the distribution.

15.7.Detailing options

The dialog box assumes the form shown in the drawing below after selecting the *Detailing options* tab in the Reinforcement: definition and detailing dialog box.



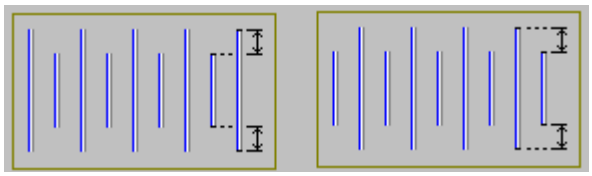
The options located on this tab enable more exact definition of the method of reinforcement distribution; here the array valid for the entire distribution on the surface is defined. In the left part of the dialog box three icons are provided:

– for this type no parameters are determined: the whole surface is covered evenly with bars

– defines passing distribution: every second bar extends from edge to edge, as regards the remaining ones, they end in a proper distance from the edge; middle bars are always straight bars; two edit fields enable defining the length of the middle bars and the additional icon located between the edit fields is used for changing the array (see the drawing below)

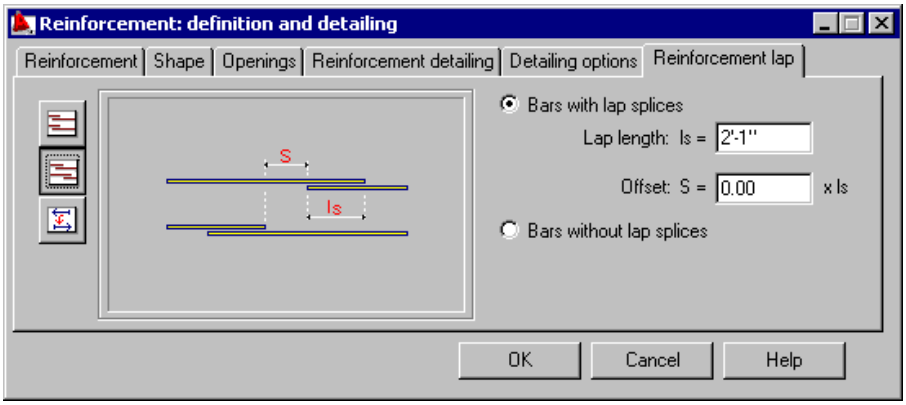
- defines passing distribution: bars in the distribution are of the same shape, however, they are shifted with respect to each other; if there is an opening (in the zone near support), application of this distribution type is not justified, then the uniform distribution is applied.

The support width option is inaccessible; it is used when defining distribution above supports.



15.8.Reinforcement lap

The dialog box assumes the form shown in the drawing below after selecting the *Reinforcement lap* tab in the Reinforcement: definition and detailing dialog box.



When main reinforcement is being distributed, it may prove necessary to use reinforcement laps. If reinforcement length exceeds the value defined in the preferences (e.g. 12 000 mm), then the options provided on this tab enable defining reinforcement laps (connections of reinforcing bars).
The options provided on this tab allow the user to:
1. make a decision if bars should be with or without laps
2. specify parameters of bars with laps.
If the *Bars without lap splices* option is selected, then reinforcement laps will not be used in a reinforcement distribution. Bars will be distributed so that they fit the distribution region, but without considering the maximum length of bars (single bars in the distribution may be longer than Lmax) and without laps. The reinforcement table will include information about the total length of bars in a distribution multiplied by a factor defined in the ***Job preferences*** dialog box (the Options tab).
'LM' bars are bars presented in the table in running meters as a total sum of all segments formed as a result of distribution.
If the *Bars with lap splices* option is selected, then reinforcement laps will be created during reinforcement distribution. The following parameters of reinforcement laps may be defined on the tab above:

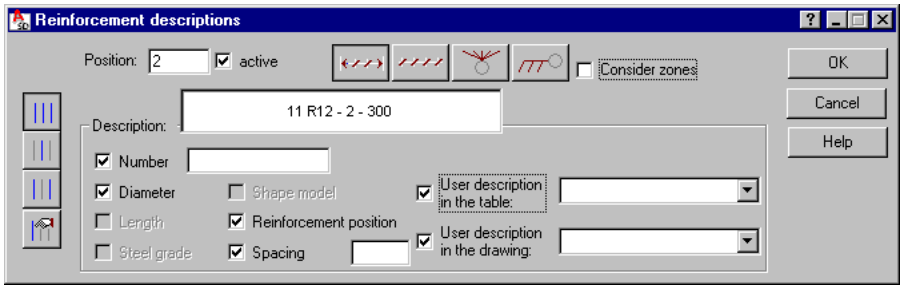
- a lap length (proposed value is a multiple of reinforcement diameter)
- a value of an offset when laps are arranged in the passing manner.

Definition of positions of Z laps may also involve changing the lap position (mirror reflection with respect to the bar center).

NOTE:
Reinforcement laps do not refer to the distributed reinforcement.

15.9.Description of reinforcement distribution

After completing reinforcement definition and determining its distribution within the defined region , the program opens the dialog box used for definition of reinforcement descriptions and reinforcement spacing descriptions.



The options included in this dialog box enable final selection of reinforcement description and distribution. It may be done by switching off active variables initialized based on the defined syntax.
The following options are provided in the dialog box:

- the field informing about the current reinforcement position; the *Active* option allows the user to avoid double or multiple calculation of the same reinforcement while preparing the reinforcement table
  this part of the dialog box also contains the *Consider zones* options; if this option is switched on, then individual zones of reinforcement distribution will be described separately, however, if this option is switched off, all zones of reinforcement distribution will be described jointly
- icons used for defining the method of presenting distribution:
  – all elements are presented in a drawing of given reinforcement distribution
  – only the middle representative of distribution in a given zone is presented (the remaining elements are not visible)
  – only extreme elements of distribution in a given zone are presented (the remaining elements are not visible)
  – it allows indicating graphically distribution elements to be presented
- the fields: *User description in the drawing* and *User description in the table* enable adding any text to a reinforcement description (by entering it from the keyboard); the description will be presented correspondingly in drawings (included in bar descriptions in the drawing) and in the table (included only in the reinforcement table); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; a text taken from the library of standard descriptions may also be applied. These descriptions assume the style of the text describing the reinforcement. The user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are applied. A user description together with an extension line and a label make up one object. Such an object may be edited (translation, rotation); to do it, the user should first use the *Explode* option available in ***AutoCAD Structural Detailing***.

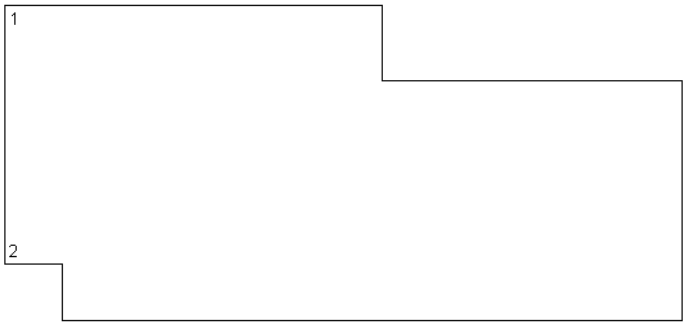
A description is assigned to the defined reinforcing bar distribution sector by sector (a sector is a contour part automatically recognized by the program as a part with similar distribution).

NOTE:
For surface distribution of a bar a DETAILED table is also available: table with total numbers of reinforcing bar consumption split into individual bar diameters.

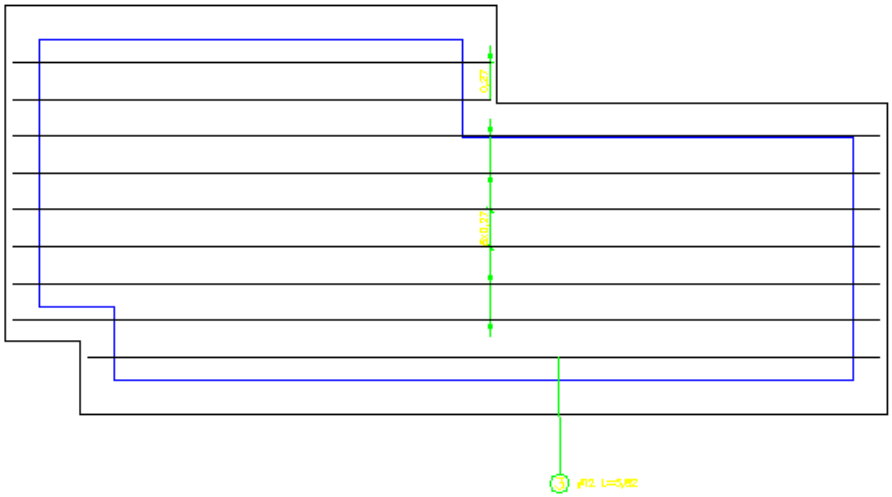
NOTE:
Description of each reinforcing bar of the surface bar distribution may be provided outside the formwork contour (the menu option: RCBR / Reinforcement – bar legend or the Reinforcement – bar legend icon).

15.10. Example definition of surface bar distribution

To define surface bar distribution within the slab shown in the figure below, the user should follow the steps below, for example:



- in the ***Surface reinforcement – bars*** dialog box select distribution type (method A) – distribution within a defined contour
- define values of support width (25 cm) and cover (5 cm) – there will be identical values assumed for each contour edge
- press the *Pick point* icon
- indicate any point within the contour shown in the figure above
- press the **OK** button in the ***Surface reinforcement – bars*** dialog box
- in the command line choose the 2Points command and indicate graphically point 1 and 2 as presented in the figure above
- press **Enter** key
- the ***Reinforcement: definition and detailing*** dialog box appears on the screen
- on the *Reinforcement detailing* tab change the number of bars in the *Number n* field (the remaining values are recalculated automatically)
- press the **OK** button in the ***Reinforcement: definition and detailing*** dialog box
- the ***Reinforcement descriptions*** dialog box appears on the screen
- press the **OK** button in the ***Reinforcement descriptions*** dialog box (adoption of default reinforcement descriptions)
- define descriptions of distribution sector by sector; the generated reinforcement is illustrated in the figure below.



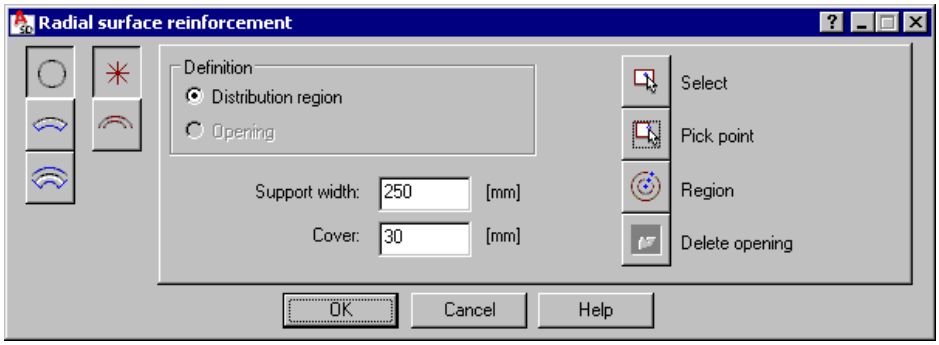
16. SURFACE REINFORCEMENT DISTRIBUTION - BARS (RADIAL REINFORCEMENT)

16.1. Radial reinforcement - bars

The option is used to define a distribution of reinforcing bars for 2D contours (e.g. an RC slab) in the shape of a circle or circle sector. The option is available from:

- the menu by selecting the *Reinforcement / Radial reinforcement – bars* option
- the toolbar by pressing the icon
- the command line: RBCR_CREATE_RADIAL.

Once the *Radial reinforcement - bars* option is activated, the dialog box shown in the drawing below appears on the screen.



The distribution allows defining two types of bars:

- *radial distribution* - bars distributed perpendicularly to the circle along radiuses, with the assigned distribution angle (bars are spaced regularly along the circle circumference or circle sector)
- *polar distribution* - bars of the circumferential reinforcement distributed uniformly with the assigned spacing, perpendicularly to the radius.

The dialog box may be divided into three main parts:

The left part of the dialog box contains icons defining a distribution type / contour type:
SELECTION OF A CONTOUR TYPE

- a contour of surface distribution of bars (this may only be a contour in the shape of a circle, circle sector, ring or ring sector) - radial or polar distribution

- a contour of surface distribution of bars above an intermediate support (with respect to the support axis) – radial and polar distribution (NOTE: such a distribution may be defined only above an arc or circle)

- a contour of surface distribution of bars above the outermost support (along the support edge) – radial and polar distribution (NOTE: such a distribution may be defined only above an arc or circle)

SELECTION OF A DISTRIBUTION TYPE

- a surface distribution of bars within a defined contour (this may only be a contour in the shape of a circle, circle sector, ring or ring sector) - radial distribution

- a surface distribution of bars within a defined contour (this may only be a contour in the shape of a circle, circle sector, ring or ring sector) – polar distribution

The central part of the dialog box includes parameters which are indispensable while defining a selected type of the reinforcement distribution:

Radial distribution (within a contour)

The upper part of the dialog box contains options for definition of a distribution region or opening(s): additionally, there are edit fields which enable defining values of support widths and a reinforcement cover.

When starting definition of a reinforcement distribution contour, the *Opening* option is inaccessible, whereas the *Distribution region* option is active and selected. Once definition of a contour is completed, the *Distribution region* option is no longer active, whereas the *Opening* option is active and selected. Then the user may define an opening contour within the earlier-defined contour; a number of openings may be freely determined by the user.

Once the beginning point of a distribution is indicated, reinforcing bars are distributed perpendicularly to the arc of the radius length reduced by twice the value of the cover (at the beginning and end of the bar). If a bar length does not exceed the allowable maximum bar length, then single bars are drawn. If it is longer, bars are automatically divided into segments with lap splices.

Radial distribution (with respect to the support axis)

The following options are provided in the central part of the dialog box (reinforcing bars are distributed perpendicularly to the arc):

- the edit field for defining a support width
- edit fields for defining values of the reinforcement overhang outside the support face (in both directions).

Radial distribution (above the outermost support)

The following options are provided in the central part of the dialog box (reinforcing bars are distributed perpendicularly to the arc):

- the edit field for defining a value of the reinforcement cover
- the edit field for defining a support width
- the edit field for defining a value of the reinforcement overhang outside the support face.

Polar distribution (within a contour)

Circle-shaped bars are distributed parallelly to the circle or arc, with the defined spacing. There may be more than one zone in a distribution of circle-shaped bars. If a length of ring bars exceeds the allowable maximum bar length, then they are divided into segments considering the lap splice. Division into segments is performed according to the following rule:

1. a number of bars has to be such so that the length of each of them does not exceed the allowable maximum bar length (considering the lap splice)
2. lengths of all bars are identical.

Ring bars are distributed perpendicularly to the radius of a distribution region.

Polar distribution (above the support)

Reinforcing bars are distributed parallelly to the circle or arc (above the support).

The right part includes icons for selecting a mode of graphical distribution definition. The number of icons depends on a selected type of the surface distribution.

Radial and polar distributions (within a contour)

 - *Select* (to directly select a contour defined as a circle, circle sector, ring or ring sector)

 - *Pick point* (to indicate an internal point of a closed contour); the operation results in detecting a contour)

 - Region (to define a ring of two radiuses: inner and outer) - for this mode it is possible to define supports of different widths on the contour edges

 - *Delete opening* (to delete an earlier-defined opening contour; available while modifying reinforcement in a defined region with an opening or while defining a greater number of openings)

Radial distribution (with respect to the support axis and above the outermost support) and polar distribution (above the support)

 - *Select* (to directly select one edge of a contour defined as a line or support)

 - *Start - Center - End* (to indicate three characteristic points of an arc: arc beginning, arc end and center of a circle to which the arc belongs)

 - *3 Points* (to indicate three points which lie on the arc)

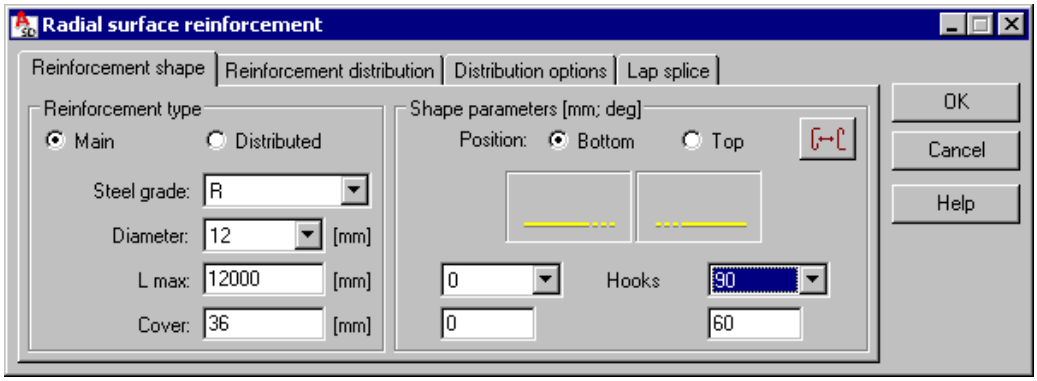
Pressing the **OK** button opens the Reinforcement: definition and detailing – radial reinforcement dialog box.

16.2. Radial/polar reinforcement - definition and detailing

Pressing the **OK** button in the Radial surface reinforcement dialog box opens the dialog box composed of the following tabs:
Reinforcement shape
Reinforcement distribution
Distribution options
Lap splice.
After defining a reinforcement and its distribution in a defined region, the Reinforcement description (radial) dialog box opens on the screen.

16.3. Reinforcement shape

Once the *Reinforcement shape* tab is selected, the Reinforcement: definition and detailing – radial reinforcement dialog box looks as shown in the drawing below.



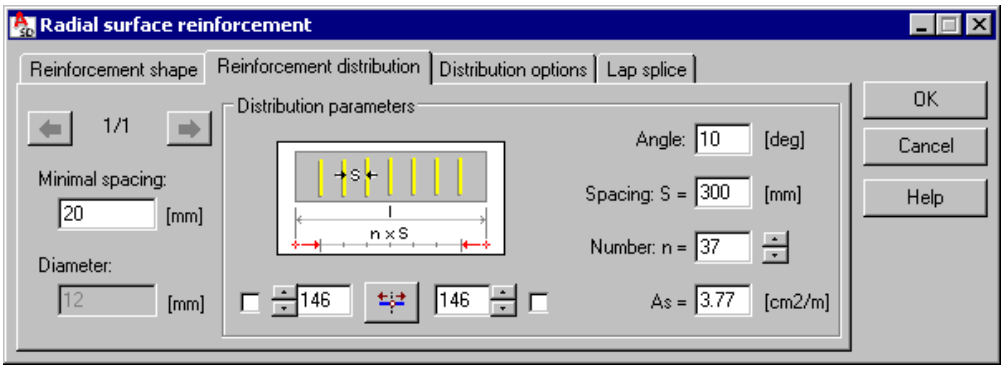
The above dialog box allows defining parameters as follows:

- in the *Reinforcement type* field
  selection of reinforcement type: main or distributed reinforcement; the following parameters may be defined for the reinforcement:
  steel grade
  diameter of reinforcing bars
  maximum length of a reinforcing bar
  reinforcement cover;

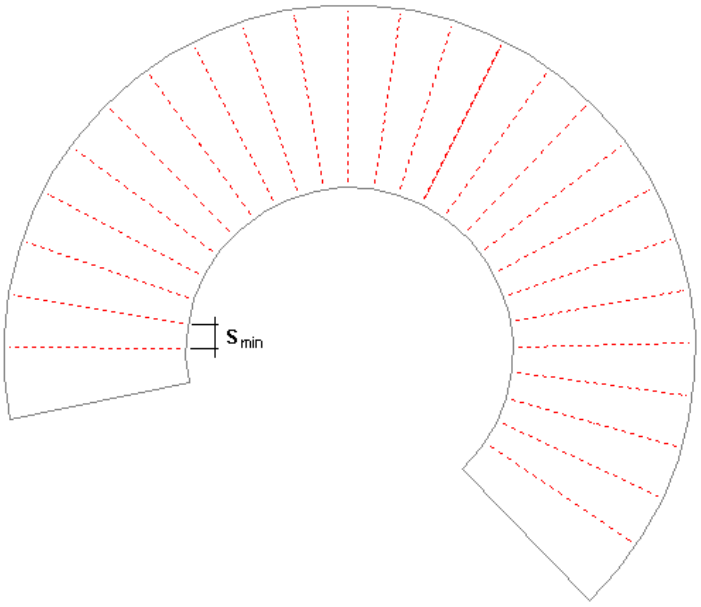
- in the *Shape parameters* field
  definition of a reinforcement shape; the *Position* option enables placing the defined reinforcement in the upper or lower part of a slab
  pressing the  icon changes the orientation of an asymmetrical reinforcement allowing the user to determine on which side of the bar (beginning or end) the chosen types of bar ends are located
  in the lower part of the dialog box the user may choose geometry of bar ends (take note that dimensions of bar ends always remain the same as defined in the dialog box, whereas a length of the middle part of bars may be modified while defining a distribution); pressing the icon which represents a bar end type opens the additional dialog box - ***Shape definition*** where an appropriate bar end can be chosen; in the above dialog box the user may determine values of appropriate dimensions of the bar end; the possibility to choose the bar end type for both bar ends (top and bottom) is worth mentioning here; moreover, the user may define bar ends in the form of hooks (the following information needs to be given: a value of the bend angle and a hook length).

16.4. Reinforcement distribution

Once the *Reinforcement distribution* tab is selected, the Reinforcement: definition and detailing – radial reinforcement dialog box looks as shown in the drawing below.



The options provided in the above dialog box are used for defining parameters of a reinforcement distribution.
The distance between distributed bars and the distribution line is assumed by default to be the cover adopted from the bar shape.
In the upper part of the dialog box there is the *Minimal spacing* edit field used to define the minimal spacing for bars distributed along the radius (a default value equals 50 mm). The minimal spacing concerns the distance between ends of distributed reinforcing bars positioned closer to the center of a circle/arc and must be greater or equal to a given value.



Distribution parameters can be defined by:

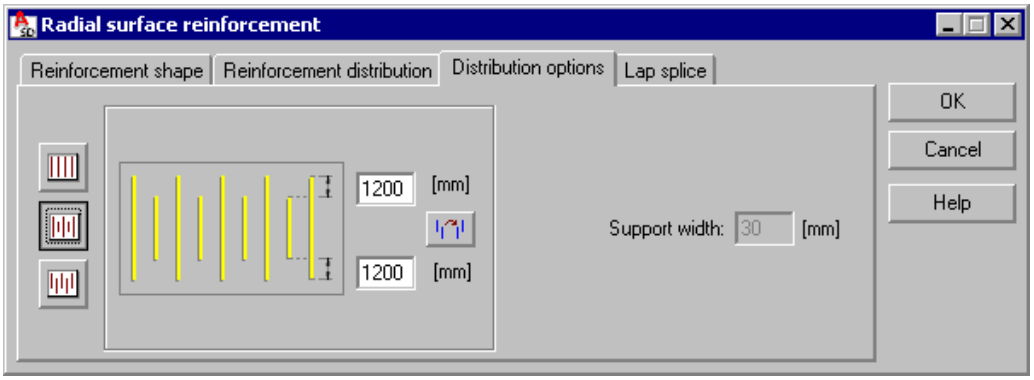
- giving a value of the distribution angle (this is the angle between bars in a defined distribution)
- giving a value of the reinforcement spacing (this is the distance between ends of distributed bars measured along the arc on the circle circumference); it is always a greater value of the spacing values for both ends)
- giving a value of the n parameter - it corresponds to the number of reinforcement spacings
- determining a reinforcement area (presented on the region circumference).

Changing any of the listed parameters results in updating the values of the remaining ones.
The schematic drawing provided in the dialog box illustrates the above-mentioned parameters. The options next to the drawing enable the user to place the reinforcement distribution with precision in a given zone:

- two edit fields allow defining a value of the distance between the extreme reinforcement and the zone border; there is a possibility to block the distance value for each zone - once the distance on one side is blocked, the distance value on the other side will be automatically adjusted to a specified number of bars and a spacing value; after blocking the distance on both sides, the spacing will be adjusted automatically to a given number of bars
  - for a circle and ring distances are summed and make up the spacing S between the first and the last bar in the distribution
  - for a circle sector or ring, these are distances between extreme bars and straight edges of the region specified on the greater arc of the region circumference
- the button located between the edit fields enables centering the distribution.

16.5. Distribution options

Once the *Distribution options* tab is selected, the Reinforcement: definition and detailing – radial reinforcement dialog box looks as shown in the drawing below.



The options located on this tab enable more exact definition of the reinforcement distribution method; an arrangement for the entire distribution on the surface is defined here. The left part of the dialog box holds three icons:

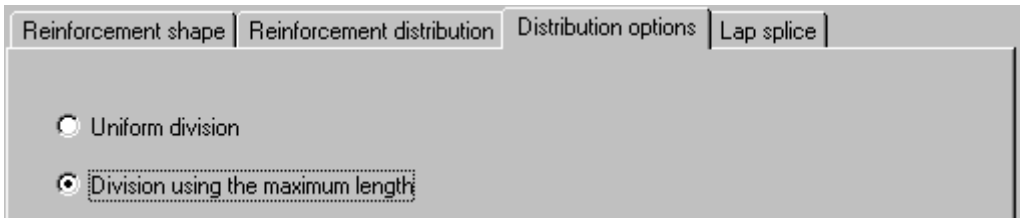
 - for this type no parameters are determined: a whole distribution surface is covered evenly with bars

 - defines a passing distribution: every second bar extends from edge to edge, while the remaining ones end in an appropriate distance from the edge; middle bars are always straight bars; two edit fields allow defining the length of middle bars and the additional icon between the edit fields is used for exchanging the arrangement

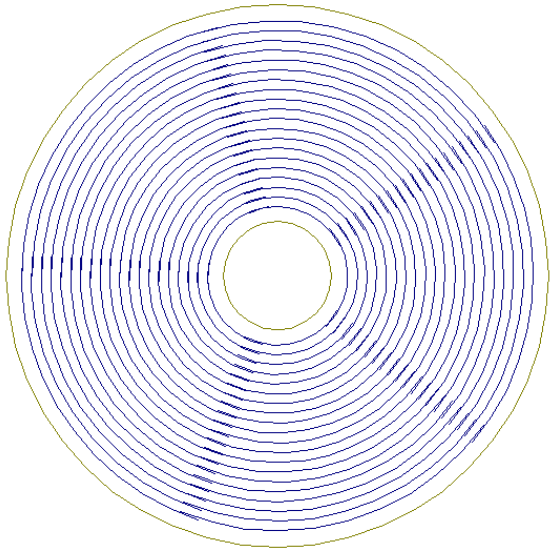
 - defines a passing distribution: distributed bars are of the same shape, however, they are shifted with respect to each other; if there is an opening (in the zone near the support), use of this distribution type is not justified, the uniform distribution is applied then.

The support width option is inaccessible; it is used when defining distributions above supports.

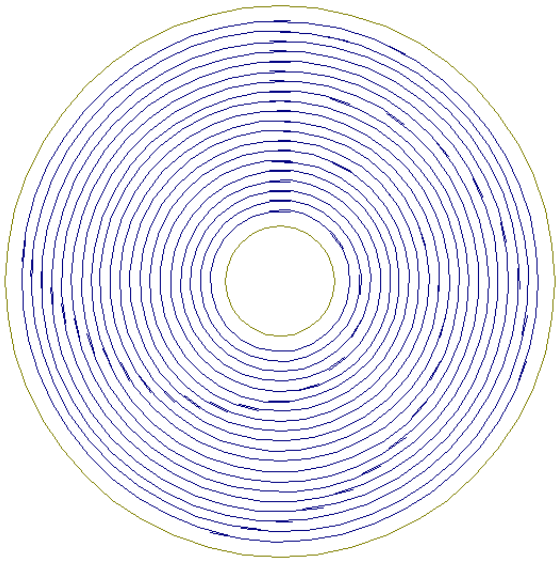
For definition of the polar distribution (ring bars) there are two ways available of dividing bars (distributing lap splices of bars in the polar distribution):



1. uniform division (see the drawing below)



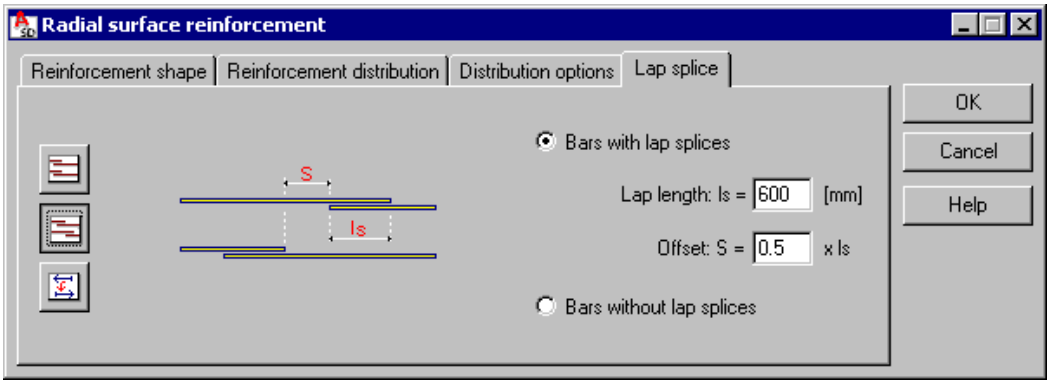
2. division using the maximum length of a bar (see the drawing below).



Each bar in the polar distribution is divided separately following the rule: bar of the maximum length + bar of the maximum length + the rest resulting from the remaining length of a bar to be distributed. Division of every bar in the polar distribution starts at the same point.

16.6. Lap splice

Once the *Lap splice* tab is selected, the Reinforcement: definition and detailing – radial reinforcement dialog box looks as shown in the drawing below.



When distributing the main reinforcement, it may prove necessary to use reinforcement laps. If a reinforcement length exceeds the value defined in the preferences (e.g. 12 000 mm), then the options provided on this tab enable defining reinforcement laps (connections of reinforcing bars).
The options provided on this tab allow the user to:
1. make a decision if bars should be with or without laps
2. specify parameters of bars with laps.

If the *Bars without lap splices* option is selected, then reinforcement laps will not be used in a reinforcement distribution. Bars will be distributed in such a way so that they fit the distribution region, but without considering the maximum length of bars (single bars in the distribution may be longer than Lmax) and without lap splices. The reinforcement table includes information about the total length of distributed bars multiplied by a factor defined in the ***Job preferences*** dialog box (the *Options* tab).
If the *Bars with lap splices* option is selected, then reinforcement lap splices will be created during reinforcement distribution. The following parameters of reinforcing bar laps may be defined on the tab above:

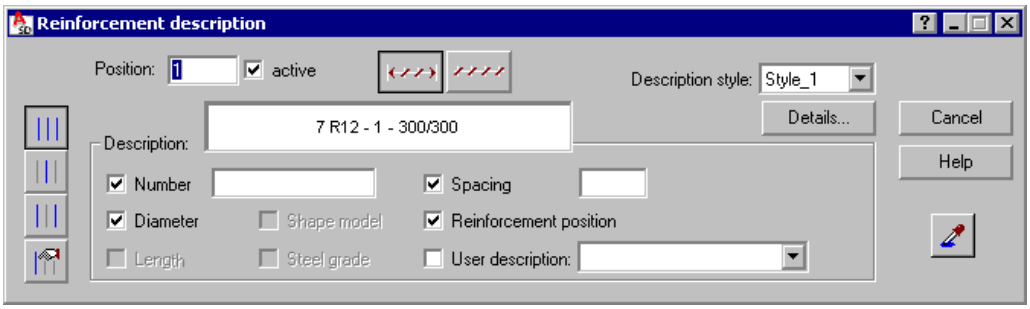
- a lap length (the proposed value is a multiple of a reinforcement diameter)
- an offset value when laps are arranged in the passing manner.

Definition of positions of Z laps may also involve changing the lap position (a mirror reflection with respect to the bar center).

NOTE:
Reinforcement laps do not refer to the distributed reinforcement.

16.7. Reinforcement description - radial/polar reinforcement

After completing definition of a distribution and its parameters, the program opens the dialog box for defining descriptions of the reinforcement and the reinforcement spacing.



Radial distributions
A description of the radial distribution is always an arc-shaped line presenting an angular or linear spacing between bar ends. A dimension line may be complete (describes a distribution along its entire circumference) or partial (describes indicated bars in a part of the region and has arrowheads on its ends pointing where it continues along the circumference).
A description syntax for radial bars contains an additional variable (%deg) describing an angular spacing between bars.

Polar distributions
Polar distributions are defined with division into zones. A description of distributed bars is a (straight) dimension line.

The options in this dialog box allow final selection of the reinforcement description and distribution. It can be performed by switching off active variables initialized on the basis of a defined syntax.
The dialog box holds the options as follows:

- the field informing about the current position of a reinforcement; the *Active* option allows avoiding the situation when the same reinforcement is included in calculations twice or more times when preparing a reinforcement table
- icons used to define the way of presenting a distribution:
  - all elements are presented in a drawing of a given reinforcement distribution
  - only the middle representative of a distribution in a given zone is presented (the remaining elements are not visible)
  - only the extreme elements of a distribution in a given zone are presented (the remaining elements are not visible)
  - it allows the user to indicate graphically distribution elements to be presented
- the *User description* field enables adding any text to a reinforcement description (by entering it from the keyboard); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on; it is also possible to use a text from the library of standard descriptions. These descriptions assume a text style of the reinforcement description. A user description may be presented on the screen in several lines; then the mechanisms accessible in the AutoCAD ® program are used. The user description together with an extension line and label make up one object. Such an object may be edited (translation, rotation); to do it, first the EXPLODE option available in ***AutoCAD Structural Detailing*** should be used.

A description is assigned to a defined distribution of reinforcing bars sector by sector (a sector is a part of a contour, automatically recognized by the program, with a similar distribution).

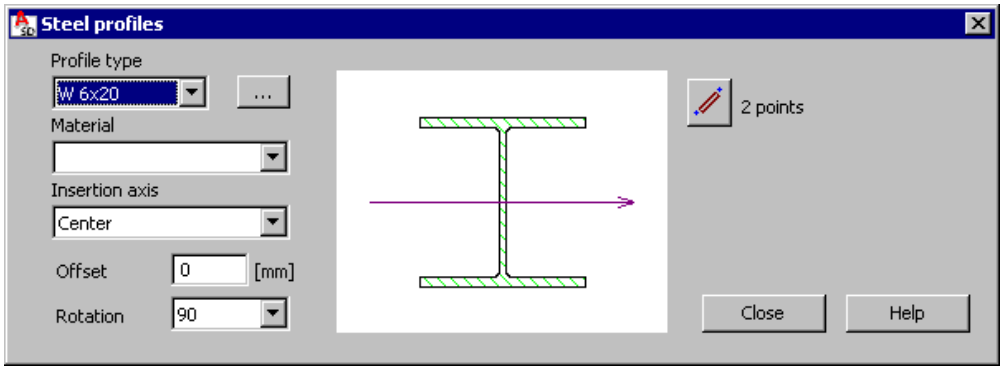
17. STEEL PROFILES

17.1.Steel profiles

The option enables defining a steel bar with a selected section. The option is available from:

- the menu, selecting the option *Reinforcement / Definition - steel profiles / Steel profiles*
- the toolbar by pressing the icon
- the command line: RBCR_CREATE_STEEL_VIEW.

After activating the *Steel profiles* option, the dialog box shown in the drawing below appears on the screen.



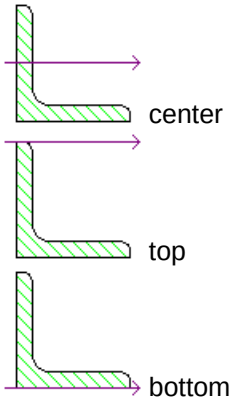
The ***Steel profiles*** dialog box can be divided into three parts:

- the left part of the dialog box allows definition of basic information concerning the section of a defined steel bar:

  a steel profile type is selected from the *Profile type* list (the list proposes the last-defined profile of a steel bar); presing the (**...**) button opens the ***Profile list*** dialog box where profiles from databases available in the program can be added to the list of available profiles (the profile database that profiles will be selected from can be set in the ***Job preferences*** dialog box)

  the *Material* list enables selection of a material type assigned to the defined steel profile (the material list can be defined in the ***Job preferences*** dialog box)

  the *Insertion axis* field is used to determine the axis of definition of a steel profile



  it is also possible to define an offset (shift) of the point of insertion of a steel profile; the offset denotes a shift of the profile center with respect to the insertion axis. an offset value may be positive (offset upwards) or negative (offset downwards)

the *Rotation* list allows selecting an angle of rotation of the cross-section of a steel profile; the following typical values of the rotation angle are available: 0, 90, 180 and 270 degrees

- the central part of the dialog box holds a graphic field presenting a selected steel profile
- in the right-hand part of the dialog box there is the icon used for definition of a steel profile:

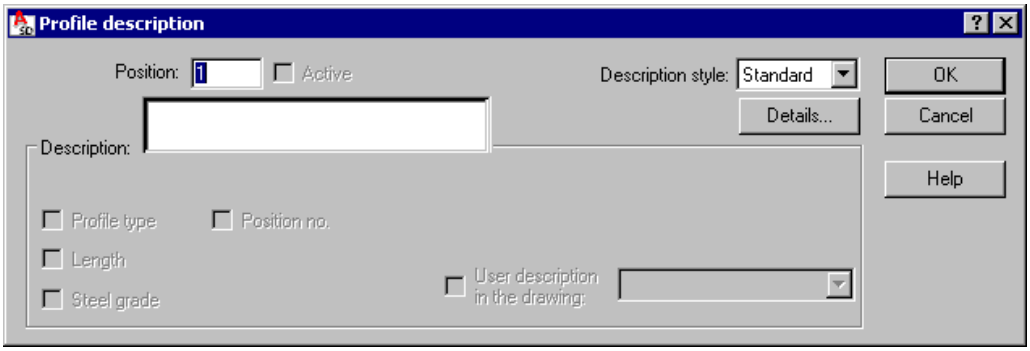
  - by means of 2 points (the beginning and end of a steel profile)

The dialog box opens showing bar profile parameters defined recently.

17.2. Description of a steel profile

After finishing definition of a steel profile, it can be described by:

- selecting the menu option *Reinforcement / Definition - steel profiles / Steel profiles - description*
- pressing the icon .



Options in this dialog box allow selecting a profile description. It is possible by switching off active variables initialized on the basis of a defined syntax (e.g. profile type, profile length, steel grade).

The *Position* field is used to specify a number of a described element. If the the *Active* option is switched on, the profile described will be included in the table. It means that for a profile described for the first time the option will be active by default. When the same profile is described twice, the option will be switched off on its own. Thus it is possible to prevent (when the same profile is described twice) taking steel profiles into account twice in the table.

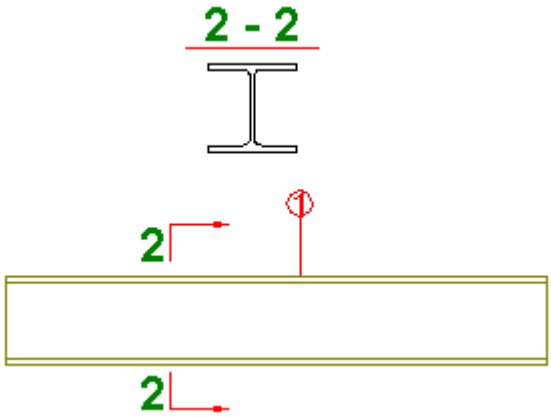
The *User description in the drawing* field enables adding any text to a steel profile description (by entering it from the keyboard); the description will be presented appropriately in drawings (included in profile descriptions in the drawing); since that moment the added text will be remembered (on the selection list) and the user will be able to use it later on.

A style of steel profile description can be selected from the *Description style* list; the list contains all description styles defined for steel profiles (the first item on the selection list is a default description style chosen in the ***Description of reinforcement shape*** dialog box); parameters of a description style can be modified before a steel profile description is inserted into the drawing; pressing the **Details...** button opens the ***Reinforcement description*** dialog box where modifications of the style of steel profile description can be made.

17.3. Operations performed on steel profiles

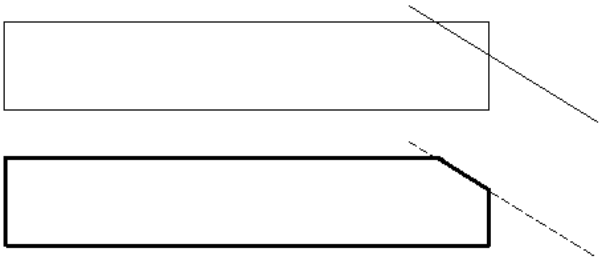
The following operations are possible on defined steel profiles:

1. a section through an indicated steel profile



To make a section through a steel section, follow the steps below:
- indicate the beginning and end point of the section
- define section depth / direction
- select location of a drawing of the section through a steel profile.

2. cutting the steel profile so that it fits the defined line



Cutting the profile is possible by specifying positions of two points that define a line. Additionally, a direction is indicated (a point on one side of the cutting line): it determines on object part to be cut off.

18. ELEMENT MANAGER

18.1.Element manager

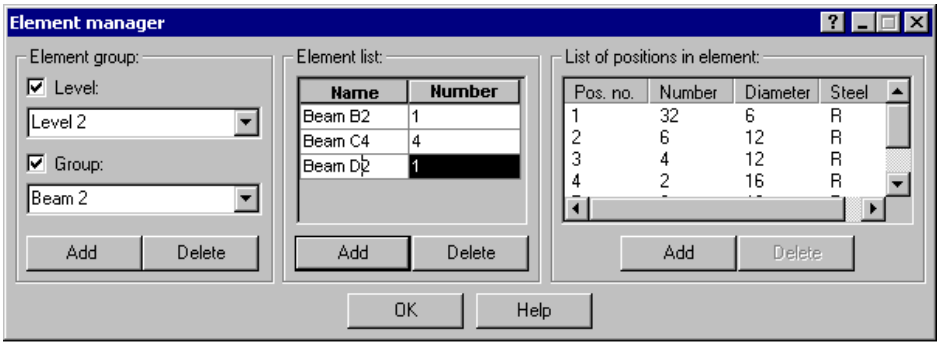
This option enables creating a reinforcement table (the table *Bars – Element table*) with reinforcing bars divided into structural elements such as a beam, a column, a spread footing, etc. The option is available:

- from the menu by selecting the command *Reinforcement / Tools / Create element*
- by pressing the icon 
- from the command line: RBCR_CREATE_ELEMENT.

The option is a convenient tool for grouping reinforcing bars into elements (groups) that may be created in the following ways:

- in the ***AutoCAD*** ® program or by means of macros for generation of formworks of RC structure elements - countours of structural elements are created and reinforcement is drawn into them; next, using the *Create element* command, reinforcing bars are assigned to appropriate elements
- in the ***AutoCAD*** ® program or by means of macros for generation of formworks of RC structure elements - countours of structural elements are created; next, using the *Create element* command 'empty' elements are defined, and finally, the program draws bars which should be added to an element
- by means of macros for generation of reinforcement of RC structure elements – contours of structural elements with their reinforcement are created; reinforcing bars are automatically assigned to appropriate elements.

After calling up the option, the dialog box presented in the drawing below appears on the screen.



The options provided in the *Element group* field allow definition of hierarchy; the following rules apply while defining the hierarchy:

- in the hierarchy a level is a superior element with respect to a group
- within a level several different groups may be defined
- every group may contain many elements.

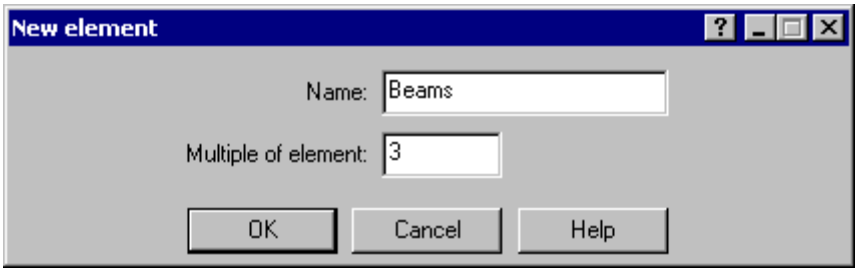
To define a level / group follow the steps below:

- switch on the *Level / Group* option (the symbol √ appears then)
- enter a name of a level and a group in the relevant field
- press the **Add** button located in the left part of the dialog box.

Defined levels / groups can be deleted from the lists of available levels / groups. To delete a group, choose the name of the group, set the mouse cursor in the *Group* field and press the **Delete** button.

The central part of the dialog box, i.e. the *Element list* field, holds options that make possible creation of elements in a selected group (and level). After choosing – from the drop-down lists –

the names of the level and the group in which an element will be created, and after pressing the **Add** button, the ***New element*** dialog box appears on the screen.



In the above dialog box the user should specify a name of the element and a number of these elements (multiple of element); After pressing the **OK** button, indicate in the drawing the objects that should belong to the element being created; it is possible to select contours (e.g. beams along with axes, section symbols, an elevation mark) and reinforcing bars. If reinforcing bars have not been drawn in a formwork yet, then only formwork parts will be included in the defined element.

Selected reinforcing bars together with the elements defined in the manner as described above are presented in the ***Element manager*** dialog box (selected reinforcement is entered into the right panel of the dialog box, in the *List of positions in element* field); names of created elements will be added automatically to the hierarchy tree located in the ***Object Inspector*** dialog box on the *Model* tab.

If while creating an element contours and reinforcing bars are chosen, then reinforcing bars will be entered into the *List of positions in element* field in the ***Element manager*** dialog box. The *List of positions in element* field is used to display reinforcing bars included in the element. If the element contains both reinforcing bars and wire fabrics, then bars are shown first on the list, whereas wire fabrics are listed after them (NOTE: for wire fabrics, in place of a diameter, a wire fabric type is presented).
The *List of positions in element* field holds two buttons:
- **Add** – enables adding a reinforcing bar / a wire fabric to a selected element
- **Delete** – enables deleting a reinforcing bar (a wire fabric) indicated on the list from an element.

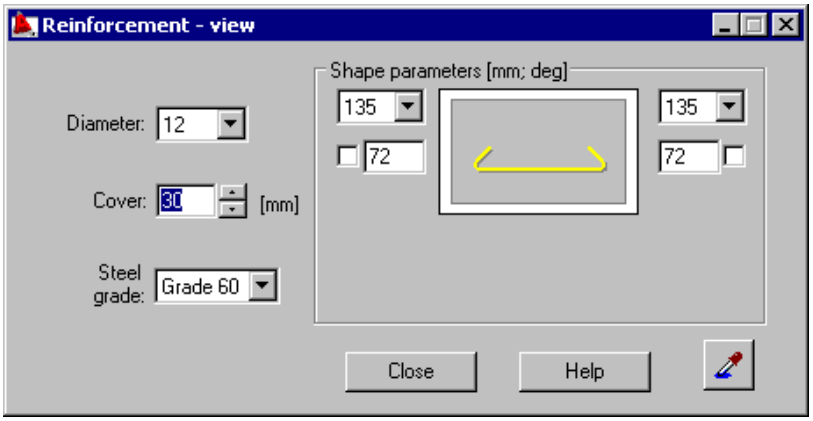
19. REINFORCEMENT MODIFICATION

19.1. Longitudinal reinforcement (reinforcement - elevation) - modification

The option enables modification of reinforcing bars (longitudinal reinforcement) in the longitudinal section of an RC structure element. The option is available from:

- the menu by selecting the *Reinforcement / Modify / Reinforcement* option
- the toolbar by pressing the icon
- the command line: RBCR_MOD_REINF.

After activating the option and selecting longitudinal reinforcement, the dialog box shown in the drawing below is displayed on the screen.



The following parameters of a reinforcing bar (longitudinal reinforcement) may be modified:

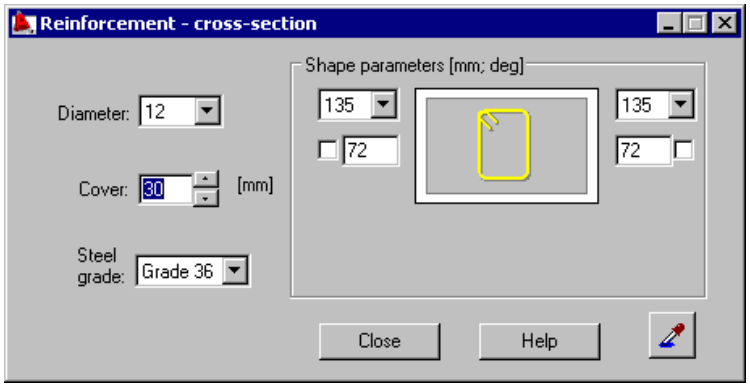
- bar diameter (the current diameter of a reinforcing bar is presented in the edit field)
- reinforcing bar cover (the current cover of a reinforcing bar is presented in the edit field)
- steel grade (the current steel grade is chosen from the selection list)
- shape parameters, that is parameters of anchors of reinforcement ends: hook angle and hook length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

19.2. Transversal reinforcement (reinforcement - section) - modification

The option enables modification of reinforcing bars (transversal reinforcement) in the cross-section of an RC structure element. The option is available from:

- the menu by selecting the *Reinforcement / Modify / Reinforcement* option
- the toolbar by pressing the icon
- the command line: RBCR_MOD_REINF.

After activating the option and selecting transversal reinforcement, the dialog box shown in the drawing below is displayed on the screen.



The following parameters of a reinforcing bar (transversal reinforcement) may be modified:

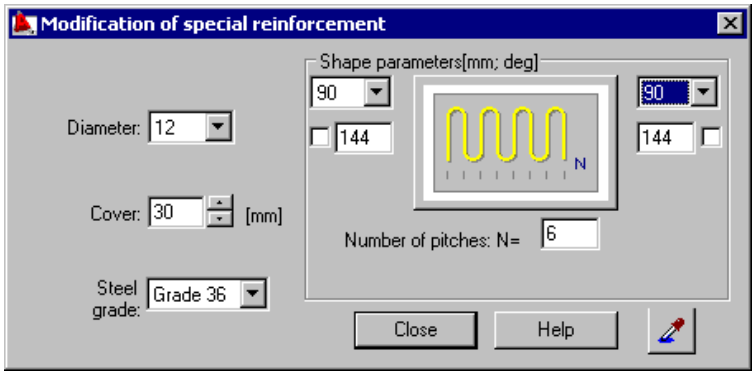
- bar diameter (the current diameter of a reinforcing bar is presented in the edit field)
- reinforcing bar cover (the current cover of a reinforcing bar is presented in the edit field)
- steel grade (the current steel grade is chosen from the selection list)
- shape parameters, i.e. parameters of anchors of reinforcement ends: hook angle and hook length; you can lock a hook length by selecting the option next to the edit field for defining a hook length (√ appears): the edit field becomes inaccessible then; if you modify parameters (length or diameter) of a bar with locked hook length, the hook length does not change.

19.3. Special reinforcement - modification

The option enables modification of reinforcing bars (of special reinforcement) in an RC structure element. The option is available from:

- the menu by selecting the option *Reinforcement / Modify / Reinforcement*
- the toolbar by pressing the icon 
- the command line: RBCR_MOD_REINF.

After calling up the option and selecting a special reinforcement, the dialog box shown below appears on the screen.

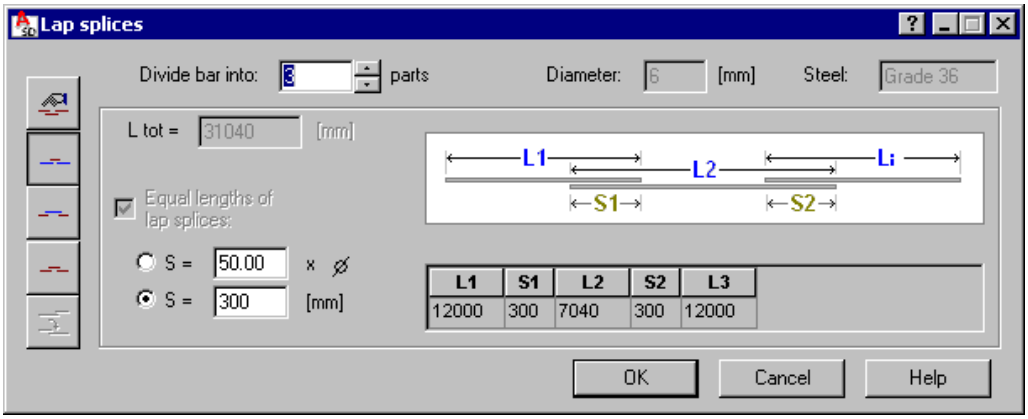


The following parameters of a reinforcing bar (special reinforcement) can be modified:

- a bar diameter (the edit field shows the current reinforcing bar diameter)
- a reinforcing bar cover (the edit field shows the current reinforcing bar cover)
- reinforcement grade (the current steel grade is selected on the selection list)
- shape parameters depending on a selected type of the special reinforcement.

19.4.Lap splices

The option enables definition / modification of lap splices in reinforcing bars if the bar length exceeds the bar length defined in the database. The dialog box shown in the drawing below is displayed on the screen after defining the bar whose length is greater than a value of the maximal bar length and accepting the information about division of the reinforcing bar.



The left part of the dialog box comprises the following icons determining methods of lap splice definition:

- – manual definition (a lap splice is defined in the table provided in the bottom part of the dialog box by specifying lengths of successive bar segments L1, L2, ... and lengths of lap splices S1, S2, etc.)
- – bar is divided in the direction ‘from outside’
- - bar is divided in the direction ‘from inside’
- – lengths of all bars are equal
- – lengths of the first and last bars are identical.

Information concerned with the bar being divided is given in the top part of the dialog box:

- number of bar divisions (this value may be modified in the dialog box – the value is available only if all segments are equal and if bar is divided into segments manually)
- bar diameter and steel grade (neither of these values can be changed in this dialog box).

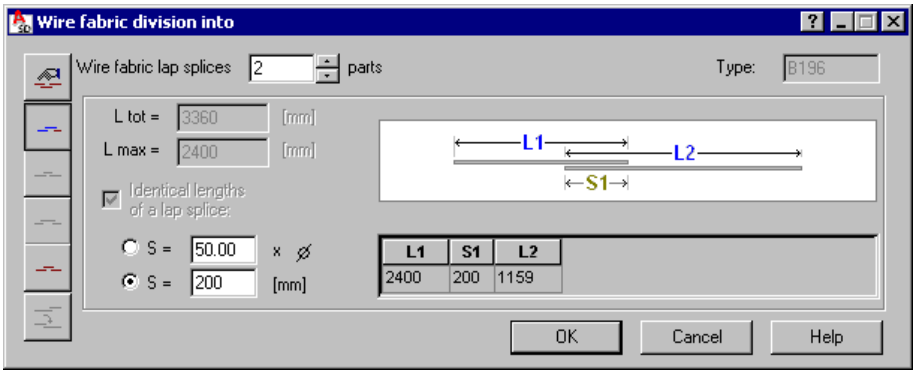
Below is presented the following data:

- bar total length (with bar lap splices included – it is the total of all bar segments with lap splices)
- if the *Equal lengths of lap splices* option is switched on, then all the lap slices in bar will be of identical length; lap splice length can be determined by giving the value of lap splice length or as a multiplier of bar diameter.

The bottom part of the dialog box includes a table presenting the defined bar division into successive segments:
- L1, L2, L3, etc. denote lengths of successive bar segments
- S1, S2, S3, etc. denote lengths of successive lap splices in bar.

19.5.Wire fabric lap splices

The option allows defining / modifying lap splices of wire fabrics defined in the cross section when the length of a wire fabric exceeds the wire fabric length defined in the database. After defining a wire fabric whose length exceeds the value of the maximum wire fabric length and accepting the information about division of the wire fabric, the dialog box shown in the drawing below appears on the screen.



The left part of the dialog box contains the following icons determining methods of definition of a lap splice of a wire fabric:

-  - manual definition (a lap splice is defined in the table provided in the lower part of the dialog box by specifying lengths of successive wire fabric segments L1, L2, ... and lengths of lap splices S1, S2, etc.)
-  - a wire fabric is divided in the direction ‘from outside’
-  - a wire fabric is divided in the direction ‘from inside’
-  - lengths of all wire fabrics are equal
-  - lengths of the first and last wire fabric are identical.

The upper part of the dialog box shows information concerning the divided wire fabric:

- number of wire fabric divisions (this value may be modified in the dialog box - the value is available only if all segments are equal and if a wire fabric is divided into segments manually)
- wire fabric type - may not be modified in this dialog box.

Below the following information is presented:

- total length of a wire fabric (including lap splices of the wire fabric – this is the total of all wire fabric segments with lap splices)
- if the *Identical lengths of a lap splice* option is switched on, then all lap splices in a wire fabric will be of identical length; a lap splice length can be determined by giving a value of the lap splice length or as a multiplier of the wire fabric diameter.

In the lower part of the dialog box there is a table presenting the defined division of a wire fabric into successive segments:

- L1, L2, L3, etc. denote lengths of successive segments of a wire fabric
- S1, S2, S3, etc. denote lengths of successive lap splices of a wire fabric.

19.6. Modification of reinforcement graphical parameters

The option enables modifying graphical parameters of reinforcing bars. The option is available from:

- the menu by selecting the *Reinforcement / Modify / Graphical parameters of reinforcement* option
- the toolbar by pressing the  icon
- the command line: RBCR_MOD_PROP.

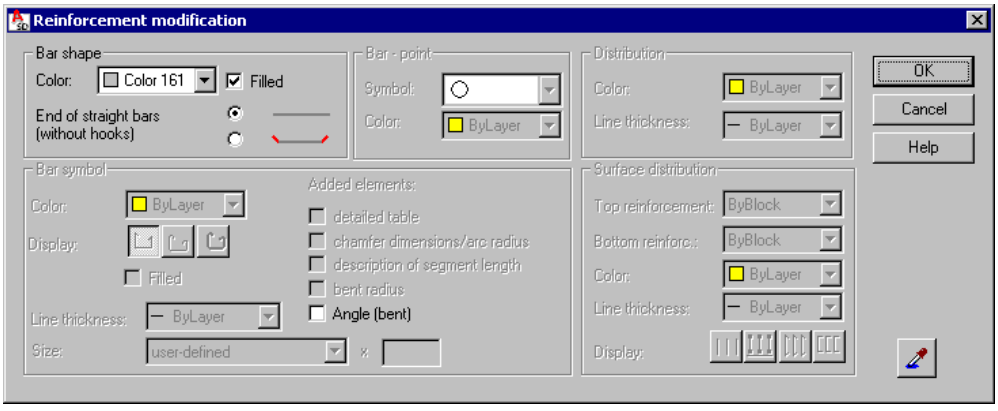
After choosing the option allowing modification of graphical parameters of reinforcement, the dialog box appears on the screen which contains options enabling modification of a selected reinforcement type.
The dialog box appearance depends on a selected reinforcement type:

- reinforcing bars
- wire fabrics.

Selection of the type of bar reinforcement (e.g. stirrup reinforcement in cross section or reinforcement distribution) causes this that in the ***Reinforcement modification*** dialog box only these options are available that enable modification of parameters of a selected reinforcement type.

19.7. Modification of graphical parameters of reinforcement (bars)

After activating the *Modify: graphical parameters of reinforcement* option and selecting bar reinforcement, the dialog box shown in the figure below is presented on the screen.



NOTE:
Selection of the type of bar reinforcement (e.g. stirrup reinforcement in cross section or reinforcement distribution) causes this that in the ***Reinforcement modification*** *dialog box only these options are available that enable modification of parameters of a selected reinforcement type.*

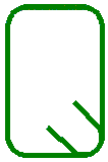
The following options provided in the *Bar shape* field may be modified:

- *Color* - selection of a color with which reinforcement will be drawn; thickness of reinforcing bars is always drawn proportionally to their diameter
- if the *Filled* option is switched on, then a reinforcing bar contour being drawn will be filled in completely with a selected color.

The *End of straight bars* option allows the user to set the method of presenting bar ends in a drawing; the option concerns only straight bars without hooks.

Reinforcing bar shapes are presented below:

- contour - filled



- contour - not filled

The options in the *Bar - point* field allow modifying the type of presentation of a reinforcing bar in cross section. The following symbols used to designate bars in cross section are provided on the drop-down list:

Moreover, a color can be chosen for the indicated symbol.

The options provided in the *Distribution* field are concerned with distribution of the existing (with shape already defined) reinforcement. The user may modify a color used to present distribution of reinforcement elements and line thicknesses.

The options from the *Bar symbol* field refer to reinforcement whose description is provided outside the formwork contour.
The user may modify color and line thickness applied while drawing reinforcement. First three buttons allow determining the manner of reinforcement presentation:

- first of them provides rough (schematic) reinforcement presentation - in the form of a broken line
- the second one presents reinforcement together with bend curvatures
- the third one presents reinforcement showing real diameters and real dimensions.

If the third option is selected, then the program also makes the *Filled* option accessible which, when switched on, allows the user to fill in the drawn reinforcing bar shape. For rough presentation and presentation showing bend curvatures, the selection list of line thicknesses is available.

This field also includes the *Added elements* option; this is a list of elements to be added to a bar whose description is provided outside the formwork contour:

- *detailed table* - in the case of a bar (whose description is provided outside the contour), whose length is linearly variable (the result of the linearly-varying distribution), there appears a table containing a detailed list with a separate description of each bar; for a bar of constant length, there appears a table consisting of one line that contains description of bar dimensions
- *chamfer dimensions* - (horizontal and vertical) dimension lines describing chamfered segments of reinforcement

- *description of segment length* - dimensions determining total length (with hooks included) of each bar segment
- *bent radius* - in some cases information about the size of radiuses of roller mandrels that form bends, is needed.

The options included in the drop-down *Size* list are used to determine the size of reinforcement symbols. The following sizes are available:

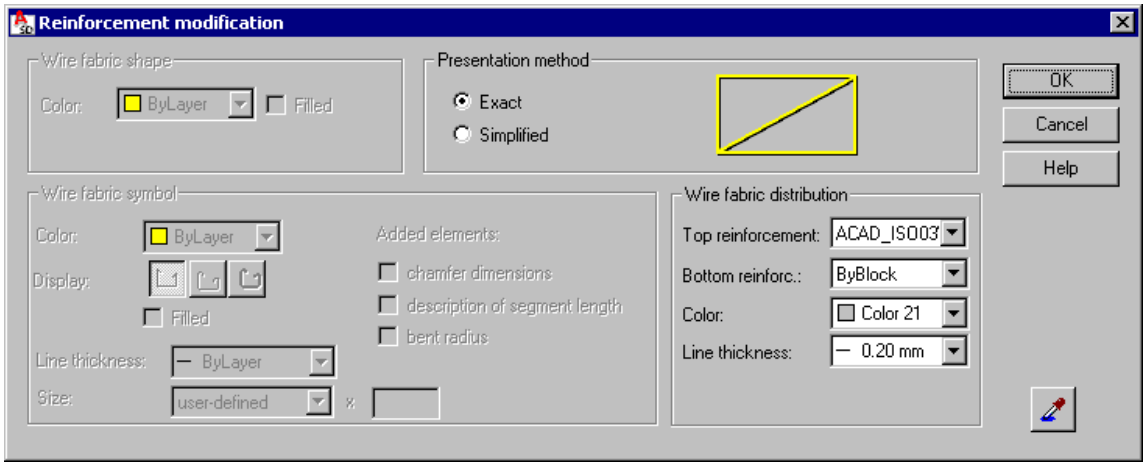
- *1 : 1* - it indicates that a symbol size equals the size of reinforcement in an element formwork
- *user-defined* - once this option is selected, the user needs to indicate (graphically) the region in which the bar symbol is to be contained
- *scale coefficient* - once this option is selected, there appears an edit field in which the user may determine a scale coefficient that will decrease or increase the symbol with respect to the real size of a bar included in a formwork; for example, entering the coefficient value equaling 0.5 causes the drawing to be twice smaller, whereas entering the value 2 means that the drawing will be increased twice.

The *Surface distribution* field allows the user to:

- modify the style of lines applied to draw top / bottom reinforcement
- modify color and line thickness for reinforcement distribution
- set (the *Display* option) the method of presenting reinforcement (significant for bars ended with hooks).

19.8. Modification of graphical parameters of reinforcement (wire fabrics)

After activating the option *Modify: Graphical parameters of reinforcement* and selecting wire fabric reinforcement, the dialog box presented in the figure below is displayed on the screen.



The *Wire fabric distribution* field allows the user to:

- modify the style of lines applied to draw top / bottom reinforcement
- modify color and line thickness for elements of reinforcement distribution.

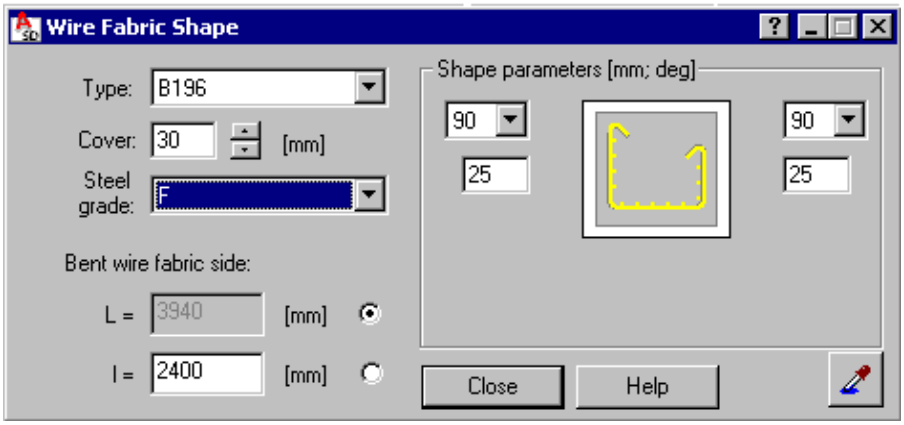
The remaining options are not available in the current program version.

19.9. Wire fabrics in cross section - wire fabric shape - modification

The option enables modification of wire fabrics in the cross section of an RC structure element. The option is available from:

- the menu by selecting the option: *Reinforcement / Modify / Reinforcement*
- the toolbar by pressing the icon
- the command line: RBCR_MOD_REINF.

After calling up the option and selecting a wire fabric in the cross section, the dialog box shown in the drawing below appears on the screen.



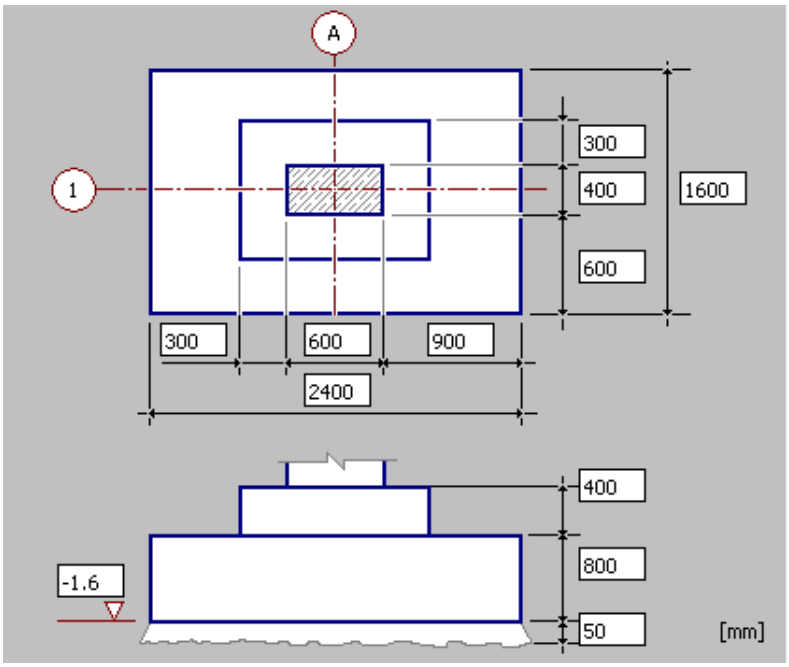
The following parameters of a wire fabric in the cross section can be modified:

- a wire fabric type (the current wire fabric type is presented in the edit field)
- a wire fabric cover (the current wire fabric cover is presented in the edit field)
- a reinforcing steel grade (the current steel grade is selected from the selection list)
- a bent wire fabric side (a shorter or longer wire fabric side should be selected)
- in the *Shape parameters* field – parameters of the hook ending of a wire fabric, i.e. a hook angle and length.

19.10. Example of reinforcement modification

Modification of reinforcement will be illustrated based on changes in parameters of reinforcement generated for a spread footing with the use of the ***Spread Footing*** macro. To define the spread footing reinforcement, follow the steps below:

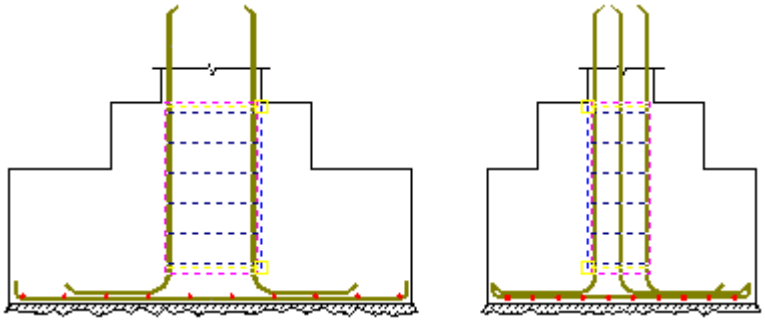
- select the menu command: *Reinforcement / Typical structures - reinforcement / Spread footing* or press the icon
- in the ***Spread footing - GEOMETRY*** dialog box specify the following parameters:
  Element name: spread footing S1
  Number of elements = 1
  Foundation shape - 1 rectangular spread footing
  Column shape - 1 column with rectangular cross-section
- adopt the dimensions of the column, the spread footing and the column pier as shown in the drawing below



- press the **Next >** button; it opens the ***Spread footing – REINFORCEMENT*** tab
- switch off the *Top reinforcement of spread footing* option; this type of reinforcement will not be generated in the defined spread footing
- determine the following parameters of the bottom reinforcement of the spread footing:
  lower layer: ϕ = 12 mm, spacing s = 150 mm
  upper layer: ϕ = 10 mm, spacing s = 250 mm
  steel: R, cover 30 mm
- determine the following parameters of dowel bars of the column base-column connection:
  reinforcement type: 1
  reinforcement: cover = 30 mm, number of bars: side A = 3, side B = 2
  ϕ = 12 mm, steel: R
  anchorage length: in column = 50*ϕ, in foundation = 50*ϕ
  stirrups: ϕ = 6 mm, spacing s = 180 mm, steel: R
- press the **Insert** button
- accept the default reinforcement number (given in the command line) pressing the **Enter** key
- indicate the location of the generated drawing of the spread footing and its reinforcement in the graphic viewer.

In the case reinforcement parameters or a reinforcement description needs to be modified once the spread footing reinforcement has been defined, the option *Modify – Reinforcement* or *Modify – Reinforcement description* should be used.
For example, to modify parameters of the stirrups generated in the spread footing, do as follows:

- select the stirrup reinforcement in both sections of the spread footing (see the drawing below)



- click the right mouse button and choose the *Modify* option from the context menu
- in the ***Reinforcement distribution*** dialog box change the stirrup spacing entering n = 6 (instead of n = 5); the stirrup spacing is adjusted automatically (the change from s=180 mm to s=150mm)
- press the **OK** button; stirrups will be modified in the drawing
- press the *Reinforcement table update* icon
- select the reinforcemet table range: *All* and press the **Enter** key; the reinforcement table will be modified and it will include new reinforcement parameters.

Similarly, there is a possibility to modify a description of a reinforcement position; after indicating a number of the reinforcement position (label), pressing the right mouse button and choosing the *Modify* option from the context menu, the ***Reinforcement description*** option appears on the screen. In this dialog box the user may select a new reinforcement description style or modify the currently used reinforcement style (after pressing the **Details** button).

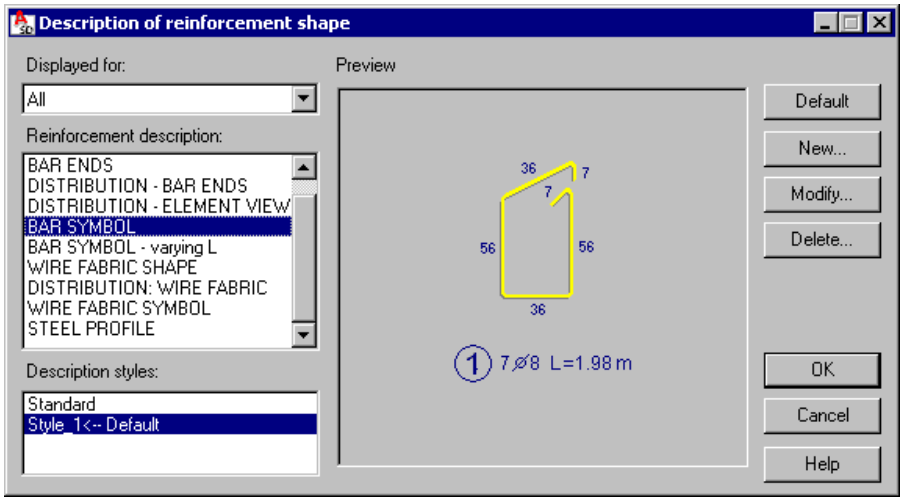
20. DESCRIPTION OF REINFORCEMENT

20.1. Description of reinforcement shape

After defining reinforcement (bar shape, reinforcement distribution, wire fabric distribution, etc.), the program runs automatically the option that suggests assigning a description appropriate for the determined reinforcement type. The *Description of reinforcement bar shape* option serves this purpose; the option is available from:

- the menu by selecting the *Reinforcement / Reinforcement description / Styles of reinforcement description* option
- the command line: RBCR_DEF_BAR_BV.

Once this option is selected, the dialog box shown in the drawing below is displayed on the screen.



For purposes of description, reinforcement (bars) has been divided into four categories:

- **Reinforcement shape** – a bar in the elevation view and a bar in the section of an element (e.g. main bar of a beam, stirrup)
- **Bar ends** – a selected type of ends in the form of additional lines drawn at bar ends
- **Reinforcement distribution** - stirrup distribution along the beam length, bar distribution over the plate surface
- **Bar symbol** - a bar with a description provided outside the formwork contour.

For purposes of description, reinforcement (wire fabrics) has been divided into three categories:

- **Wire fabric shape** – a wire fabric in the elevation view and a wire fabric in the section of an element
- **Wire fabric distribution -** wire fabric(s) in an element projection or view
- **Wire fabric symbol** - a wire fabric with a description provided outside the formwork contour.

There is also one more category **Steel profile**.

The left part of the dialog box contains three options:

- the *Displayed for* selection list - the user may select the description list for bars, for wire fabrics, or for both bars and wire fabrics
- two fields: *Reinforcement description* and *Description styles*.

The *Reinforcement description* field presents the list of reinforcement descriptions:

- the following descriptions are available for bars: Bar shape, Bar ends, Distribution - element view, Bar symbol, Bar symbol - varying length of a bar

- the following descriptions are available for wire fabrics: Wire fabric shape, Distribution: wire fabric, Wire fabric symbol.
- for steel profiles one description is available: Steel profile.

The *Description styles* field comprises the styles defined for a selected description of reinforcement.
The *Preview* field presents parameters set graphically for an indicated style of reinforcement description.

In the right part of the dialog box (apart from the standard buttons: **OK**, **Cancel** and **Help**) the following buttons are located:

- **Default** – pressing this button sets a selected style as a default style of reinforcement description
- **New** - pressing this button opens the Reinforcement description dialog box which enables defining a new style of reinforcement description
- **Modify** - pressing this button opens the Reinforcement description dialog box where changes for a selected type of reinforcement description can be made
- **Delete** - if this button is pressed, the highlighted style of reinforcement description is deleted from the list of styles available in the *Description styles* field.

20.2.Reinforcement description

The dialog box is used to define elements that are to be included in a description of a reinforcement shape.
The ***Reinforcement description*** dialog box may be opened after pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

The dialog box appearance depends on a type of the selected reinforcement description:
BARS

- Bar Shape - the dialog box consists of two tabs: Description elements, Description syntax
- Bar Ends – the Reinforcement description dialog box
- Distribution – Bar Ends – Reinforcement description dialog
- Distribution - Element View - the dialog box consists of two tabs: Description elements, Description syntax
- Bar Symbol - the dialog box consists of two tabs: Description elements, Description syntax
- Bar Symbol - Varying Length - the dialog box consists of two tabs: Description elements, Description syntax

WIRE FABRICS

- Wire Fabric Fhape - the dialog box consists of two tabs: Description elements, Description syntax
- Distribution: Wire Fabric - the dialog box consists of two tabs: Description elements, Description syntax
- Wire Fabric Symbol - the dialog box consists of two tabs: Description elements, Description syntax

STEEL PROFILES

- Steel Profile - the dialog box consists of the two tabs: Description elements, Description syntax

20.3.Bars

20.3.1. Bar shape

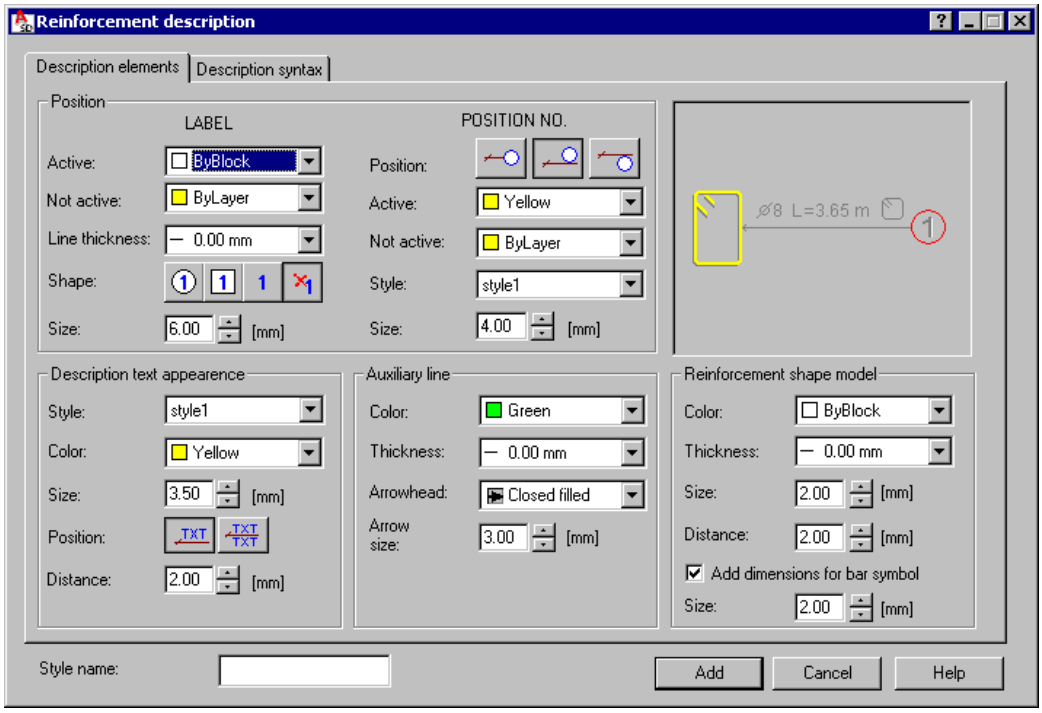
The ***Reinforcement description*** dialog box opened when describing a bar shape consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box may be opened after selecting the description of reinforcement shape and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

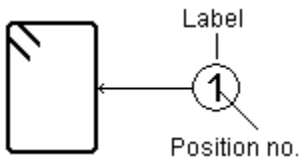
To add a new description of a bar shape, the user should press the **Add** button once parameters of the bar shape description are defined.

20.3.2. Description elements

The Reinforcement description (bar shape) dialog box assumes the form show in the figure below once the *Description elements* tab is selected therein.



On the first tab main settings (colors, line thickness, size and shape of a font, etc.) are defined. The elements of reinforcement descriptions that can be defined by the user are discussed below.
The options included in the *Position* field refer to a label together with a number the label contains (see the drawing below).

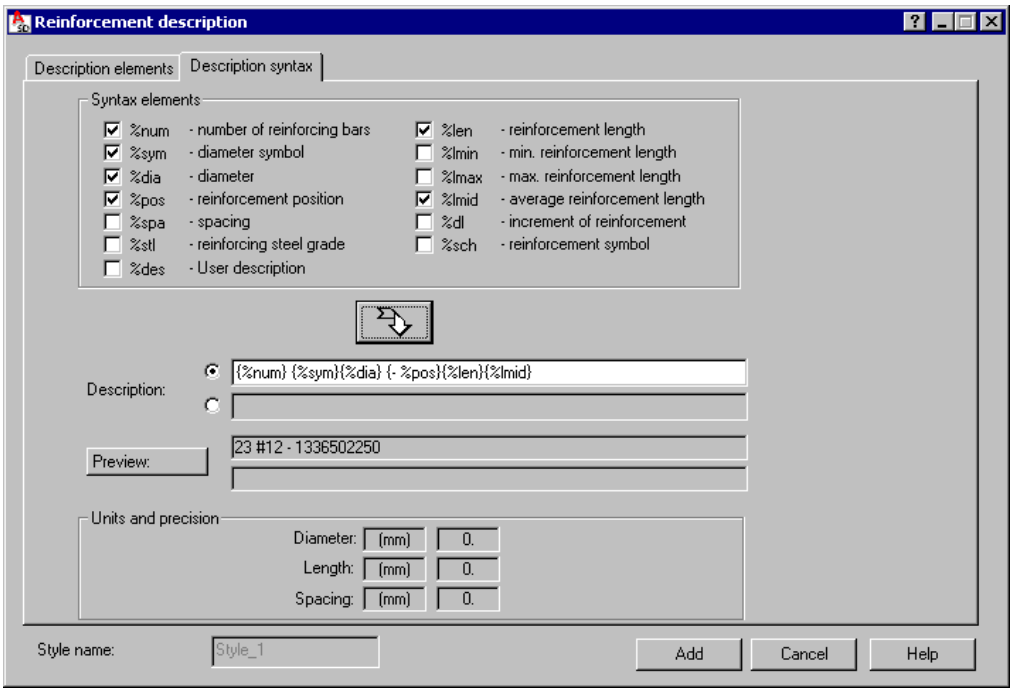


The first column of parameters pertains to a label and allows selecting color of active and not active position, line thickness, label shape and size. In the second column the user may define parameters of a position number contained in a label: default position on the extension line, color of active and not active position, font style and size. Options used for color selection for active and not active position are related to the *Active* option (see the drawing below) contained in the Reinforcement description dialog box. If the *Active* option is switched on in the ***Reinforcement description*** dialog box, then a color of this position is chosen from the *Active* list; if it is switched off in the dialog box below, then a color of this position is selected from the *Not active* list.

The options provided in the *Description text appearance* field are concerned with a description placed between a position (label) and the described reinforcement. The following parameters can be set: font style, color, size and position: above or above and under the line and vertical translation with respect to the auxiliary line.
The options contained in the *Auxiliary line* field enable the user to determine the following parameters of a line connecting a position with reinforcement: color, thickness and type of arrowheads. Moreover, there is a parameter provided that determines a size of the arrow.
The options located in the *Reinforcement shape model* field allow the user to define parameters of a symbolic shape of the described reinforcement. The following parameters can be defined: color and thickness of a line forming a symbol, size of a symbol and its position - vertical distance to a description text.
The bottom part of this field includes the option *Add dimensions for bar symbol*. When this option is switched on, then if the bar description includes a bar symbol, dimensions of individual segments of a reinforcing bar are additionally presented in the bar symbol in the description (the *Size* field enables definition of dimensions).

20.3.3. Description syntax

The Reinforcement description (bar shape) dialog box assumes the form show in the figure below once the *Description syntax* tab is selected therein.



The options provided on this tab are used to define syntax of a reinforcement description; The above dialog box provides access to both the defined general style of reinforcement description and to the mechanism allowing free composition of syntax and contents of reinforcement description.

By selecting appropriate description components contained in the *Syntax elements* field, the user may move them to the *Description* fields (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. provided by the user. An example description is as follows: ϕ10 **every** 15 **cm length =** 2.0 **m.**

The description may consists of two parts, therefore, two edit fields are available. Selected description components are moved to the active edit field after pressing the arrow. The upper edit field contains desciption components provided above the line, whereas the lower edit field – the components presented under the line (see the *Position* option on the Description elements tab).

The *Preview* field presents description of reinforcement that results from the defined syntax. This description is based on number values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with a description is the information concerning units and precision applied while displaying a given value.

The list of variables included in a reinforcement description is as follows:

%num	- number of reinforcing bars
%sym	- diameter symbol
%dia	- diameter
%pos	- number of reinforcement position
%spa	- spacing
%stl	- reinforcement class
%len	- reinforcement length
%lmin	- minimum reinforcement length
%lmax	- maximum reinforcement length
%lmid	- average reinforcement length
%dl	- increment of reinforcement length equal to a constant value
%sch	- reinforcement symbol.
%des	- user description.

The *User description* option enables adding a text to the reinforcement description determined in the Reinforcement description dialog box (for longitudinal and transversal reinforcement, point reinforcement, distribution of reinforcement, etc.).

NOTE:
The manner of reinforcement description depends on a type of reinforcement being described and country whose code is applied, e.g. the description may take the following form:

Bar shape %sym%dia L= %len m ϕ10 L= 2.0m

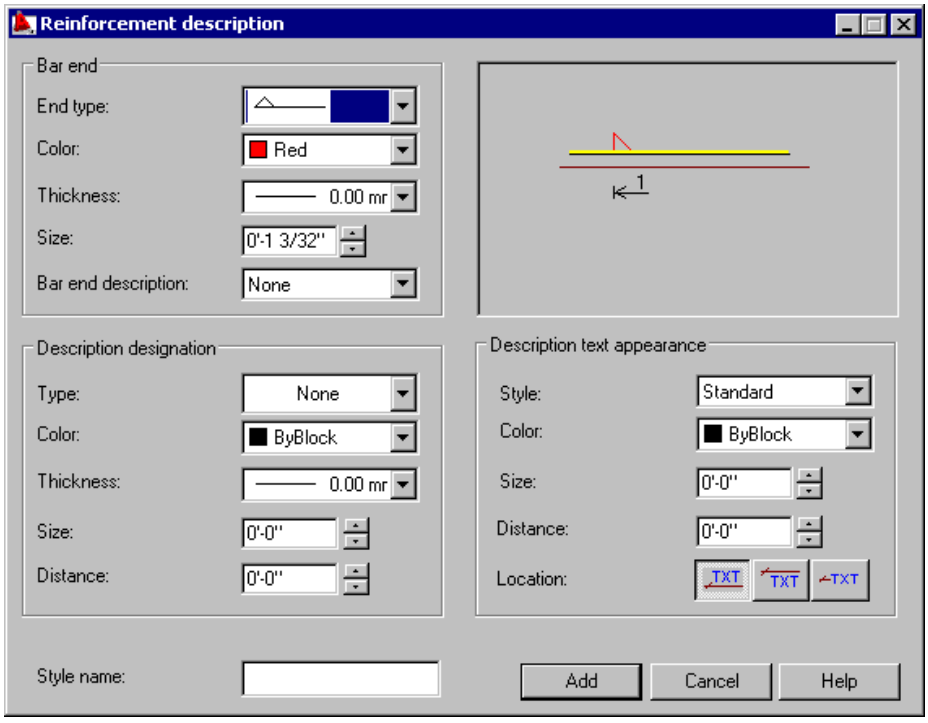
Characters added should be put in parentheses. Thanks to that, if any data in a description is missing (e.g. spacing), the entire description text with a variable will not be displayed.

Variables that may be applied in a syntax, must be put in the parentheses {}; the user may add any texts in between the successive variables put in these parentheses.

NOTE:
If for example, a bar symbol is to be included in the description of a defined reinforcing bar, the user should switch on the Reinforcement symbol option and press the button with an arrow; the variable containing a bar symbol will be added to the Description edit field; in the end the ***Add*** *button should be pressed.*

20.3.4. Bar ends - reinforcement description

The ***Reinforcement description*** dialog box is activated for description of bar ends. The ***Reinforcement description*** dialog box opens after selecting the description of bar ends and pressing the ***New*** or ***Modify*** button in the Description of reinforcement shape dialog box.



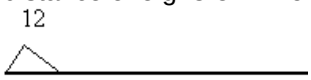
The options provided in the *Bar end* field refer to the way ends of reinforcing bars are presented (additional lines at bar ends). These lines are additional objects linked with the bar. Apart from a graphical designation, it is also possible to insert a description of a bar end in the form of an arrowhead with the bar number.

A description of the bar end can be deleted; the description is not linked with the designation of the bar end. Deletion of the designation results smultaneously in deleting the description of the bar end.
The *Bar end* field contains the options as follows:
- end type; the following end types are available:

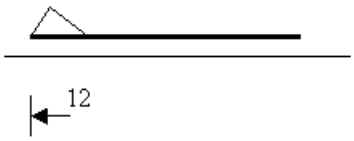
- color and thickness of the line used to draw bar ends
- symbol size
- description of the bar end (none or description type: 1, 01).

The options included in the *Description designation* field allow determining the following parameters of the line representing a bar end description: type, color, thickness, size and distance of the line end. The following types of designation are available:
- none
- 
-  .

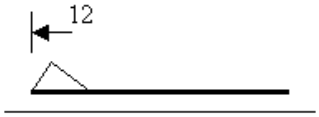
If the *none* designation type is chosen, then the description of a bar end will be inserted in the distance of e.g. 3-5 mm over the bar end.



If the designation type in the form of an arrowhead is selected, then the bar end description is inserted together with the arrowhead:
1. under the bar in the distance of: a value of the *Distance* specified in the edit field + a cover value



2. over the bar, if a value of the *Distance* has a negative sign (then the cover is subtracted)

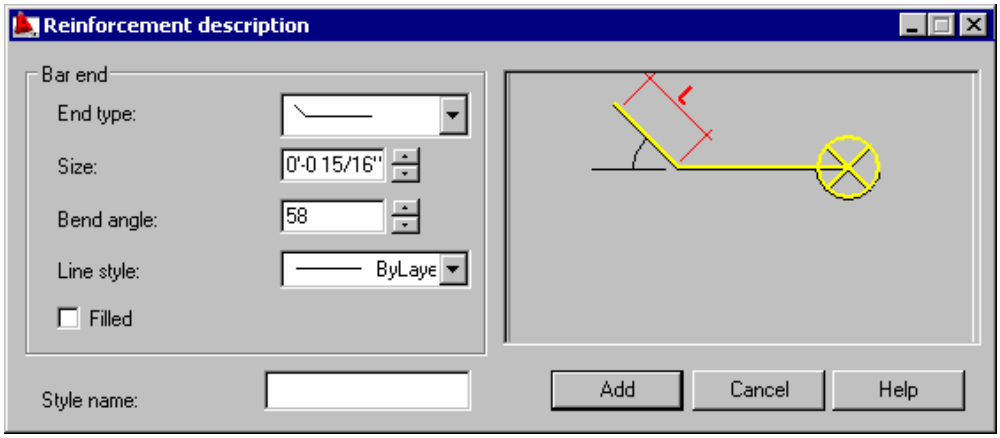


The options in the *Description text appearance* field refer to the description of bar ends. The following parameters can be set here: font style, color, size and location: over, under or next to the line.

NOTE:
If in the ***Job Preferences*** *dialog box (the Bars / Display tab) the option that allows drawing bars without hooks - with bar ends is set, then bar ends will be drawn automatically according to the default style.*

20.3.5. Distribution - Bar ends - Reinforcement description

This is the ***Reinforcement description*** dialog for descriptions of bar ends for distributions. To open this dialog, select the *Distribution – Bar ends* option and click **New** or **Modify** in the Description of reinforcement shape dialog.



The *Bar end* field has options for presenting reinforcing bar ends (additional lines at bar ends) in distributions. These lines are additional objects connected with the bar.
Specify the following parameters in the *Bar end* field:

- end type; you can select the following end types:

- size (length l)
- bend angle
- line style.

After selecting the *Filled* option, a symbol is filled  .

20.3.6. Distribution - element view

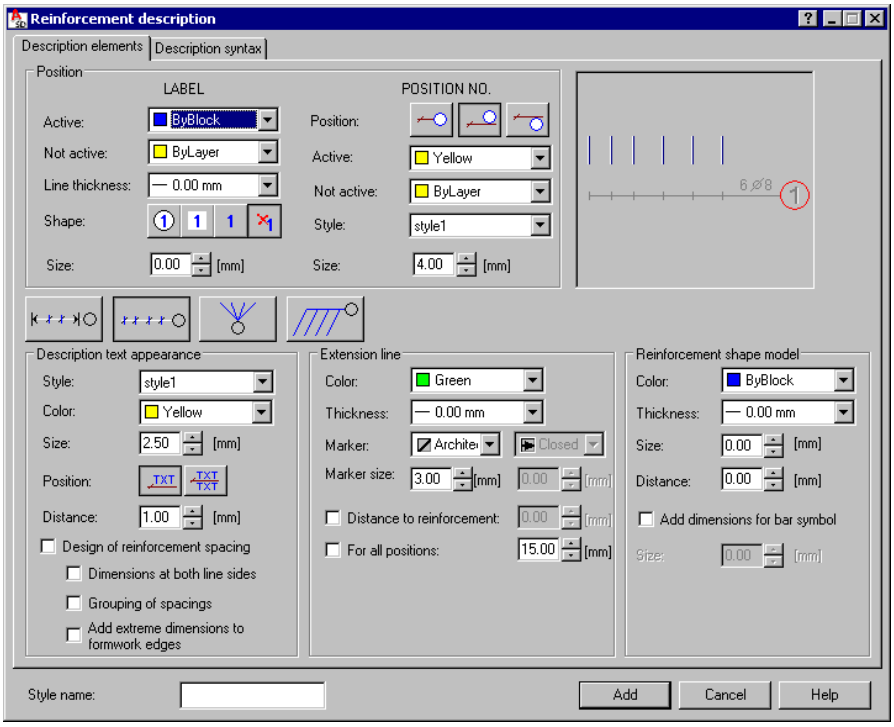
The ***Reinforcement description*** dialog box opened when describing bar distribution (element view) consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box may be opened after selecting the description of reinforcement distribution (element view) and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

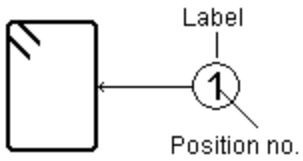
To add a new description of distribution (element view), the user should press the **Add** button once parameters of distribution description are defined.

20.3.7. Description elements

The Reinforcement description (distribution - element view) dialog box assumes the form show in the figure below, once the *Description elements* tab is selected therein.



The options included in the *Position* field refer to a label together with a number the label contains (see the drawing below).

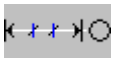


The first column of parameters pertains to a label and allows selecting color of active and not active position, line thickness, label shape and size. In the second column the user may define parameters of a position number contained in a label: default position on the extension line, color of active and not active position, font style and size. Options used for color selection for active and not active position are related to the *Active* option (see the drawing below) contained in the Reinforcement description dialog box. If the *Active* option is switched on in the ***Reinforcement description*** dialog box, then a color of this position is chosen from the *Active* list; if it is switched off in the dialog box below, then a color of this position is selected from the *Not active* list.

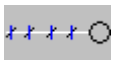
The options provided in the *Description text appearance* field are concerned with a description placed between a position and the described reinforcement. The following parameters can be set: font style, color, its size and position above or above and under the line as well as horizontal distance to a label.

The options provided below pertain to first two types of distribution description (line linking a position with described reinforcement). Switching on the first option (*Design of reinforcement spacing*) results in adding a dimension of reinforcement spacing to the line describing distribution; for this dimension the user may activate additional options applied to determine the position of dimensions - the position is presented in the drawing located in the top right part of the dialog box.

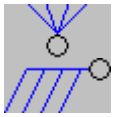
The options contained in the *Extension line* field pertain to the line connecting a position with the described reinforcement. 4 proposed styles of distribution description can be parametrized. For each of them separately appropriate characteristic parameters can be defined. Parameters of line color and thickness are invariable and they do not depend on a selected style.
The remaining components are activated and set properly depending on a selected style:



the first style allows definition of two independent types of markers (on ends of an extension line and in between them) and their sizes; the *Distance to reinforcement* option is accessible

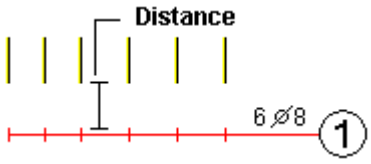


the second style allows definition of one marker type and size; the *Distance to reinforcement* option is accessible



for the third and fourth styles one field is provided for selecting a marker: arrowhead and size; the *Distance to reinforcement* option is accessible

If the *Distance to reinforcement* option is activated, it enables the user to determine the distance between reinforcement description and reinforcement distribution (see the drawing below). If the distance equals zero, then reinforcing bars in the distribution are connected with the description.

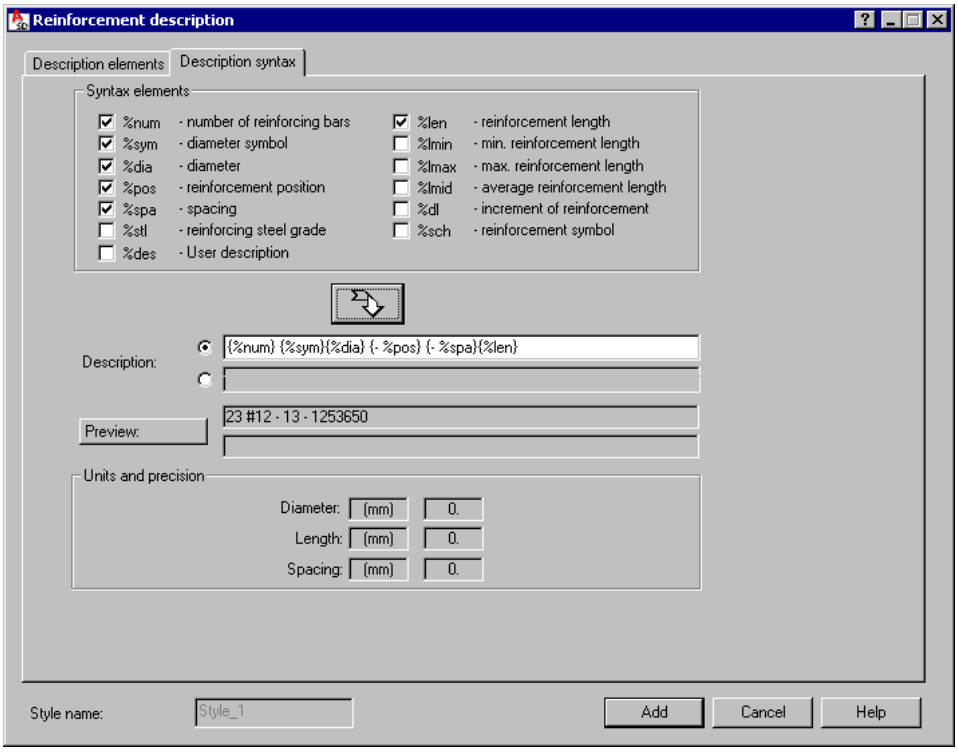


If the *For all positions* option is switched on, it means that the distance between the reinforcement description and the reinforcement distribution will equal the value defined for all positions.

The options provided in the *Reinforcement shape model* field are used to define parameters of a symbolic shape of described reinforcement. The following parameters can be defined: color and thickness of the line making up a symbol, symbol size and location – a horizontal distance to the description text. In the lower part of this field is the *Add dimensions for bar symbol* option. After switching on this option, if a bar description includes the bar symbol, then additionally, the bar symbol in the description will present dimensions of individual segments of a reinforcing bar (the *Size* field allows defining dimensions).

20.3.8. Description syntax

The Reinforcement description (distribution - element view) dialog box assumes the form show in the figure below, once the *Description syntax* tab is selected therein.



The options provided on this tab are used to define description syntax of reinforcement distribution.

By selecting appropriate components of distribution description contained in the *Syntax elements* field, the user may move them to the *Description* fields (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. specified by the user. An example description is as follows: ϕ10 **every** 15 **cm length =** 2.0 **m**.

The description may consists of two parts, therefore, two edit fields are available. Selected description components are moved to the active edit field after pressing the arrow. The upper edit field contains desciption components provided above the line, whereas the lower edit field – the components presented under the line (see the *Position* option on the Description elements tab).

The *Preview* field presents description of reinforcement distribution that results from the defined syntax. This description is based on number values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with a description is the information concerning units and precision applied while displaying a given value.

The list of variables included in description of reinforcement distribution is as follows:

%num	- number of reinforcing bars
%sym	- diameter symbol
%dia	- diameter
%pos	- number of reinforcement position
%spa	- spacing
%stl	- reinforcement class
%len	- reinforcement length
%lmin	- minimum reinforcement length

%lmax	- maximum reinforcement length
%lmid	- average reinforcement length
%dl	- increment of reinforcement length equal to a constant value
%sch	- reinforcement symbol.
%des	- user description.

The *User description* option enables adding a text to the reinforcement description determined in the Reinforcement description dialog box (for longitudinal and transversal reinforcement, point reinforcement, distribution of reinforcement, etc.).

NOTE:
The manner of describing reinforcement distribution depends on a type of the reinforcement being described and country whose code is applied.
Characters added should be put in parentheses. Thanks to that, if any data in a description is missing (e.g. spacing), the entire description text with a variable will not be displayed

NOTE:
If for example, a bar symbol is to be included in the description of a defined reinforcing bar, the user should switch on the Reinforcement symbol option and press the button with an arrow; the variable containing a bar symbol will be added to the Description edit field; in the end the ***Add*** *button should be pressed.*

20.3.9. Bar symbol

The ***Reinforcement description*** dialog box opened when describing a bar symbol or a symbol of a bar with variable length consists of two tabs:
Description elements
Description syntax.

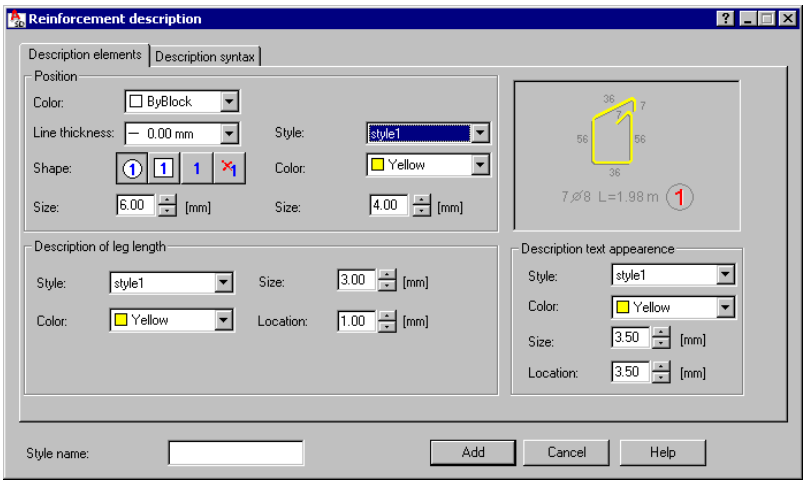
The options contained in this dialog box allow defining components that are provided when a reinforcement description is placed outside the contour of an element formwork.

The ***Reinforcement description*** dialog box may be opened after selecting the description of reinforcement distribution (element view or element projection) and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

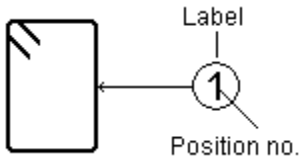
To add a new description of bar symbol, the user should press the **Add** button once parameters of a bar symbol description are defined.

20.3.10. Description elements

The Reinforcement description (bar symbol/with variable L) dialog box
assumes the form shown in the figure below once the *Description elements* tab is selected therein.



The options included in the *Position* field refer to a label together with a number the label contains (see the drawing below).



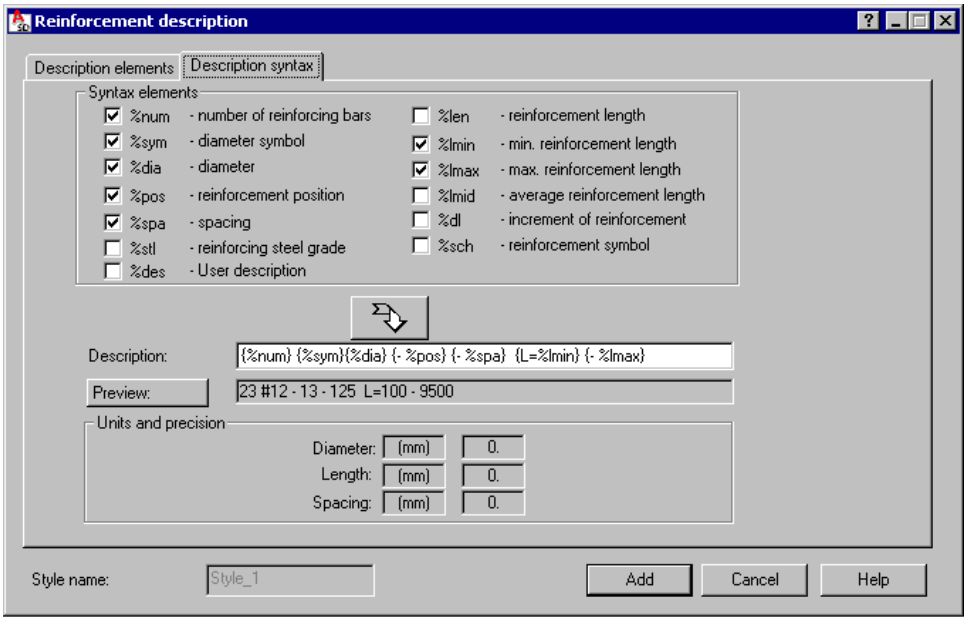
The first column of parameters pertains to a label and allows selecting color, line thickness, label shape and size. In the second column the user may define parameters of a number contained in a label: font style, color and size.

The options included in the *Description of leg length* field allow defining style, color, size as well as position of the text with respect to the reinforcement being described; such a text contains a description of the bar leg length.

The options provided in the *Description text appearance* field are concerned with a description placed directly behind a position. The following parameters can be set: font style, color, size and position with respect to a label (horizontal distance).

20.3.11. Description syntax

The Reinforcement description (bar symbol/with variable L) dialog box
assumes the form shown in the figure below once the *Description syntax* tab is selected therein.



The options provided on this tab are used to define description syntax for a symbol of reinforcement whose description is provided outside the formwork contour.
By selecting appropriate components of a symbol description contained in the *Syntax elements* field, the user may move them to the *Description* field (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. specified by the user. An example description is as follows: f10 **every** 15 **cm length =** 2.0 **m**.

The *Preview* field presents description of a reinforcement symbol that results from a defined syntax. This description is based on number values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with a description is the information concerning units and precision applied while displaying a given value.
The list of variables included in description of a reinforcement symbol is as follows:

%num	- number of reinforcing bars
%sym	- diameter symbol
%dia	- diameter
%pos	- number of reinforcement position
%spa	- spacing
%stl	- reinforcement class
%len	- reinforcement length
%lmin	- minimum reinforcement length
%lmax	- maximum reinforcement length
%lmid	- average reinforcement length
%dl	- increment of reinforcement length equal to a constant value
%sch	- reinforcement symbol
%des	- user description.

The *User description* option enables adding a text to the reinforcement description determined in the Reinforcement description dialog box (for longitudinal and transversal reinforcement, point reinforcement, distribution of reinforcement, etc.).

NOTE:
The manner of describing reinforcement symbol depends on the type of reinforcement being described and country whose code is applied.
Characters added should be put in parentheses. Thanks to that, if any data in a description is missing (e.g. spacing), the entire description text with a variable will not be displayed

NOTE:
If for example, a bar symbol to be included in the description of a defined reinforcing bar, the user should switch on the Reinforcement symbol option and press the button with an arrow; the variable containing a bar symbol will be added to the Description edit field; in the end the ***Add*** *button should be pressed.*

20.4.Wire fabrics

20.4.1. Wire fabric shape

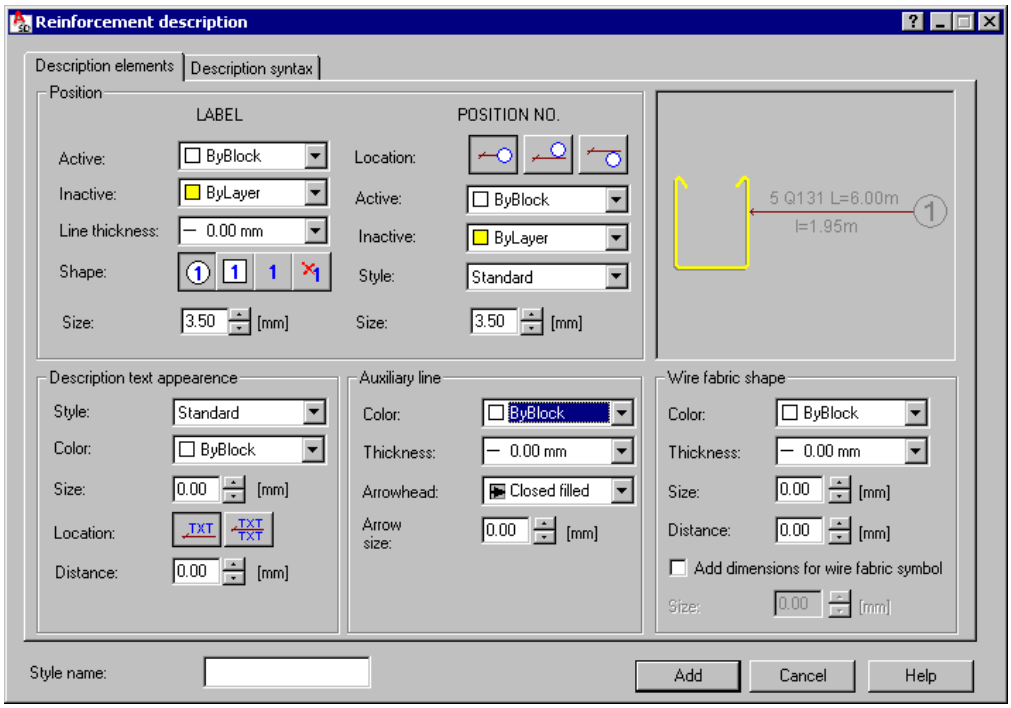
The ***Reinforcement description*** dialog box opened when describing a wire fabric shape consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box opens after selecting a description of the wire fabric shape and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

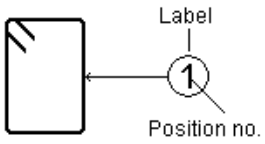
To add a new description of the wire fabric shape, press the **Add** button after defining parameters of the description of the wire fabric shape.

20.4.2. Description elements

The Reinforcement description (wire fabric shape) dialog box assumes the form shown in the drawing below once the *Description elements* tab is selected.



The first tab is used to define main settings (colors, line thickness, font size and style, etc.). Below are presented elements of a wire fabric description that can be defined by the user.
The options included in the *Position* field refer to a label and to a number it contains (see the drawing below).



The first column of parameters refers to the label and enables selecting a color of an active and inactive position, line thickness, the label shape and size. The options in the second column allow defining parameters of a position number in the label: default position on a leader, color of active and inactive positions, font style and size. Options used to select a color for active and inactive positions are related to the *Active* option (see the drawing below) in the Reinforcement description dialog box. If the *Active* option is switched on in the ***Reinforcement description*** dialog box, then the color of this position is chosen from the *Active* list; if it is switched off in the dialog box below, then the color of this position is selected from the *Inactive* list.

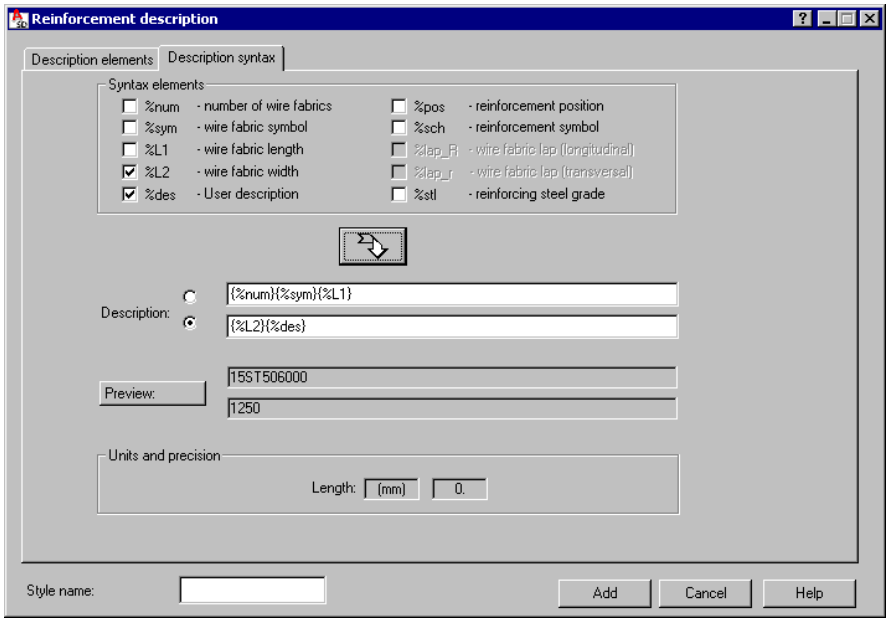
The options provided in the *Description text appearance* field are concerned with a description placed between a position (label) and a described wire fabric. The following parameters can be set here: font style, color, size and position: above or above and under the line and vertical translation with respect to the auxiliary line.

The options in the *Auxiliary line* field make it possible to define the following parameters of the line connecting a position with reinforcement: color, thickness and type of arrowheads. Moreover, there is a parameter that determines the arrow size.

The options located in the *Wire fabric shape* field enable definition of parameters of a symbolic shape of a described wire fabric. The following parameters can be defined: color and thickness of the line forming a symbol, symbol size and position - horizontal distance to a description text. At the bottom of this field is the option *Add dimensions for wire fabric symbol*. When this option is switched on, then if the wire fabric description includes the wire fabric symbol, dimensions of individual segments of the wire fabric bar are additionally presented in the description in the wire fabric symbol (the *Size* field enables definition of dimensions).

20.4.3. Descrption syntax

The Reinforcement description (wire fabric shape) dialog box assumes the form shown in the drawing below once the *Description syntax* tab is selected.



The options provided on this tab are used to define a syntax of a description of a wire fabric shape.
By selecting appropriate components of a shape description from the *Syntax elements* field, the user may move them to the *Description* field (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. provided by the user.
The *Preview* field presents a description of the wire fabric shape that results from a defined syntax. This description is based on number values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with description is the information about the units and the precision applied while displaying a given value.
The list of variables included in the wire fabric description is as follows:

%num - number of wire fabrics
%sym - wire fabric symbol
%L1 - wire fabric length
%L2 - wire fabric width
%des - user description
%pos - reinforcement position
%sch - reinforcement symbol
%lap_R - wire fabric lap (longitudinal)

%lap_r - wire fabric lap (transversal)
%stl - reinforcing steel grade.

NOTE:
The method of wire fabric description depends on the type of described reinforcement and the country whose code is applied.

Variables that may occur in a syntax, must be put in the parentheses {}; the user may add any texts in between the successive variables put in these parentheses.

NOTE:
For example, for a wire fabric symbol to be included in the description of a defined wire fabric, switch on the Reinforcement symbol option and press the button with the arrow; the variable containing the wire fabric symbol will be added to the Description edit field; to finish, press the ***Add*** *button.*

20.4.4. Wire fabric distribution

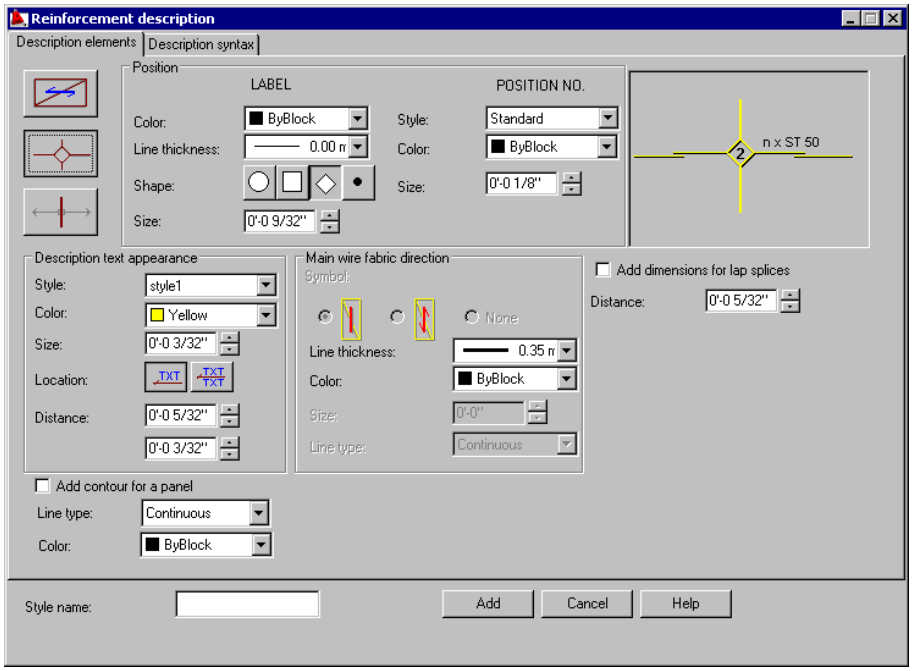
The ***Reinforcement description*** dialog box opened when describing wire fabric distribution consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box may be opened after selecting description of wire fabric distribution and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

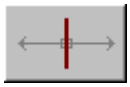
To add a new description of a wire fabric shape, the user should press the **Add** button once parameters of a description of wire fabric distribution are defined.

20.4.5. Description elements

The Reinforcement description (wire fabric distribution) dialog box
assumes the form shown in the figure below once the *Description elements* tab is selected therein.



There are the following options for presenting a wire fabric in drawings:

-  exact
-  simplified
-  group.
-

Options in the dialog are specific to a selected type of presenting a wire fabric.

The options included in the *Position* field refer to a leader and concern a leader (label) together with a number the label contains. The options provided allow changing color, line thickness, label shape and size. Moreover, the user may define parameters of a number contained in a label: font style, color and size.

The options provided in the *Description text appearance* field are concerned with a description placed behind a reinforcement position. The following parameters can be set: font style, color, size and position with respect to a label and bottom level of a label.

The options included in the *Main wire fabric direction* field refer to that how a symbol denoting the main wire fabric direction should be presented in a drawing. There are three options available:

- 
- 
- absence of the symbol indicating the main wire fabric direction.

For the symbols denoting the main wire fabric direction in the first two options the following parameters may be defined: symbol line thickness, symbol color and size.

If you select the *Add dimensions for lap splices* option, descriptions include dimensions of lap splices.

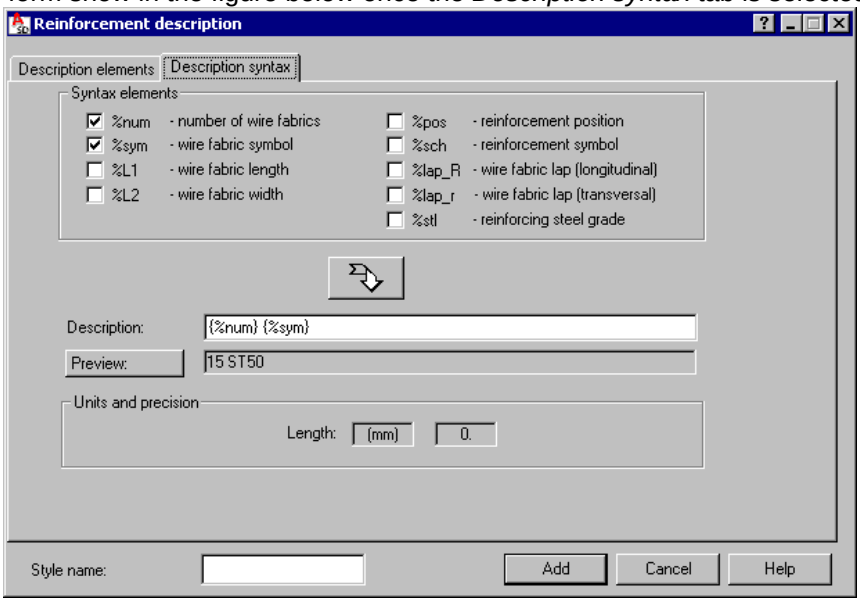
At the bottom of the dialog is the option for simplified presentation of wire fabrics - *Add contour for a panel.* After you select this option, additionally, a contour is drawn for a panel (slab). Specify a line type and color for this contour.

The *Distribution line* field has options for the group presentation of wire fabrics:

- type and color of the distribution line
- marker type
- size
- thickness
- shape.

20.4.6. Description syntax

The Reinforcement description (wire fabric distribution) dialog box
assumes the form show in the figure below once the *Description syntax* tab is selected therein.



The options provided on this tab are used to define the description syntax of wire fabric distribution.
By selecting appropriate components of a distribution description contained in the *Syntax elements* field, the user may move them to the *Description* field (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. specified by the user.

The *Preview* field presents a description of reinforcement distribution that results from a defined syntax. This description is based on number values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with a description is the information concerning units and precision applied while displaying a given value.
The list of variables included in a description of wire fabrics is as follows:

%num - number of wire fabrics
%sym - wire fabric symbol
%L1 - wire fabric length
%L2 - wire fabric width
%pos - reinforcement position

%sch - reinforcement symbol
%lap_R - wire fabric lap (longitudinal)
%lap_r - wire fabric lap (transversal)
%stl - reinforcing steel grade.

NOTE:
The manner of describing wire fabrics depends on the type of reinforcement being described and country whose code is applied.

Variables that may be applied in a syntax, must be put in the parentheses {}; the user may add any texts between the successive variables put in these parentheses.

NOTE:
If for example, a wire fabric symbol is to be included in the description of a defined wire fabric, the user should switch on the Reinforcement symbol option and press the button with the arrow; the variable containing the wire fabric symbol will be added to the Description edit field; in the end the ***Add*** *button should be pressed.*

20.4.7. Wire fabric symbol

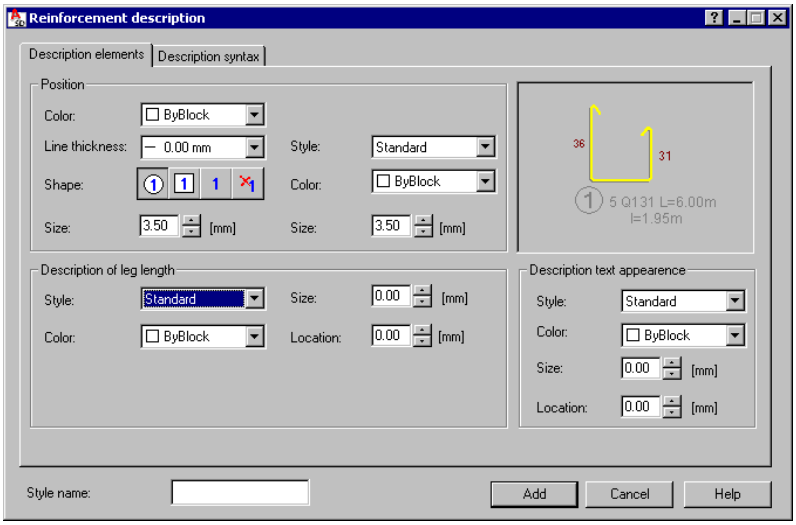
The ***Reinforcement description*** dialog box opened when describing a wire fabric symbol consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box opens after selecting a description of the wire fabric symbol and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

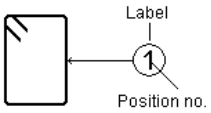
To add a new description of the wire fabric symbol, press the **Add** button after defining parameters of the description of the wire fabric symbol.

20.4.8. Description elements

The Reinforcement description (wire fabric symbol) dialog box assumes the form shown in the drawing below once the *Description elements* tab is selected there.



The options included in the *Position* field refer to a label and a number the label contains (see the drawing below).



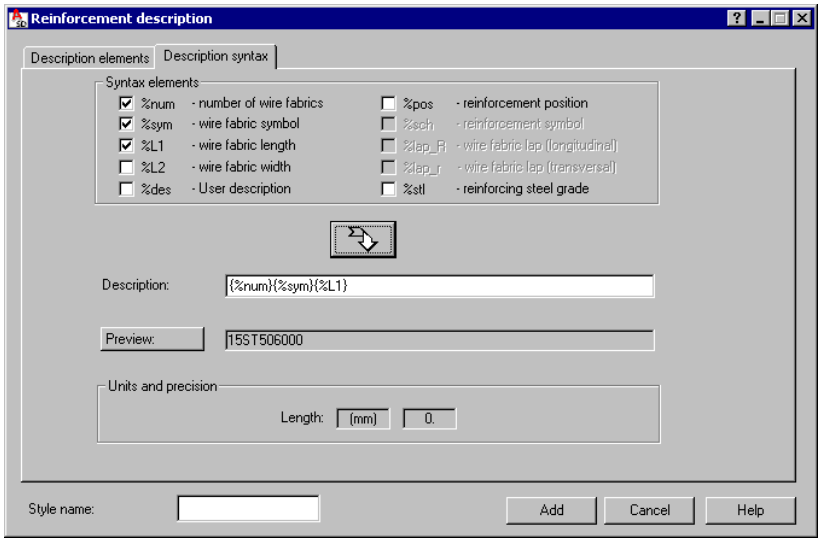
The first column of parameters refers to a label and allows selecting color, line thickness, label shape and size. In the second column the user may define parameters of a number in the label: font style, color and size.

The options included in the *Description of leg length* field enable defining style, color, size as well as position of a text with respect to the described wire fabric; this text is a description of the length of wire fabric legs.

The options in the *Description text appearance* field are concerned with a description placed directly behind a position. The following parameters can be set here: font style, color, size and position with respect to a label (horizontal distance).

20.4.9. Description syntax

The Reinforcement description (wire fabric symbol) dialog box assumes the form shown in the drawing below once the *Description syntax* tab is selected.



The options on this tab are used to define a description syntax for a wire fabric symbol.
By selecting appropriate components of a symbol description from the *Syntax elements* field, the user may move them to the *Description* field (by pressing the arrow located in the middle part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. specified by the user.
The *Preview* field presents a description of the wire fabric symbol that results from a defined syntax. This description is based on number values saved as fixed ones, and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.
Under the field with description is the information about the units and the precision applied while displaying a given value.

The list of variables included in the wire fabric description is as follows:

%num	- number of wire fabrics
%sym	- wire fabric symbol
%L1	- wire fabric length
%L2	- wire fabric width
%des	- user description
%pos	- reinforcement position
%sch	- reinforcement symbol
%lap_R	- wire fabric lap (longitudinal)
%lap_r	- wire fabric lap (transversal)
%stl	- reinforcing steel grade.

NOTE:
The method of wire fabric description depends on the type of described reinforcement and the country whose code is applied.

Variables that may occur in a syntax, must be put in the parentheses {}; the user may add any texts in between the successive variables put in these parentheses.

20.5.Steel profiles

20.5.1. Steel profile

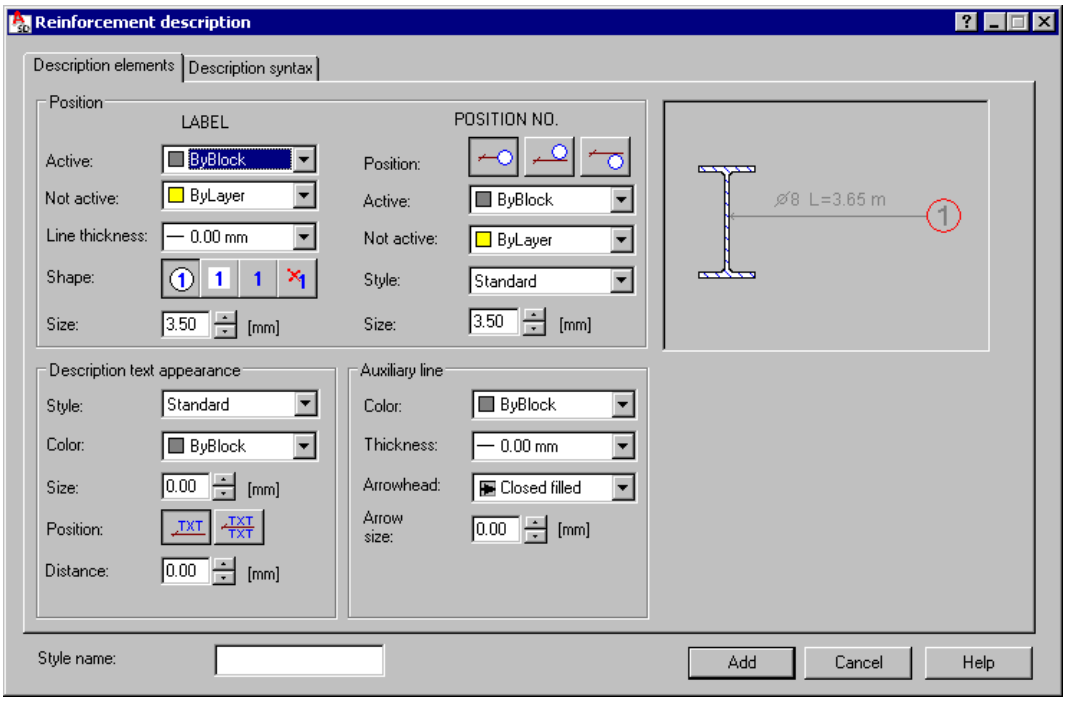
The ***Reinforcement description*** dialog box opened when describing steel profiles consists of two tabs:
Description elements
Description syntax.

The ***Reinforcement description*** dialog box opens after selecting a steel profile description and pressing the **New** or **Modify** button in the Description of reinforcement shape dialog box.

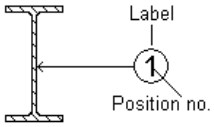
To add a new description of a steel profile, press the **Add** button after defining parameters of the steel profile description.

20.5.2. Description elements

The Reinforcement description (steel profiles) dialog box looks as presented in the drawing below after selecting the *Description elements* tab in it.



The first tab is used to define main settings (colors, line thickness, size and style of fonts, etc.). Below are presented elements of the steel profile description that can be defined by the user.
Options in the *Position* field refer to a label and a number in the label (see the drawing below).



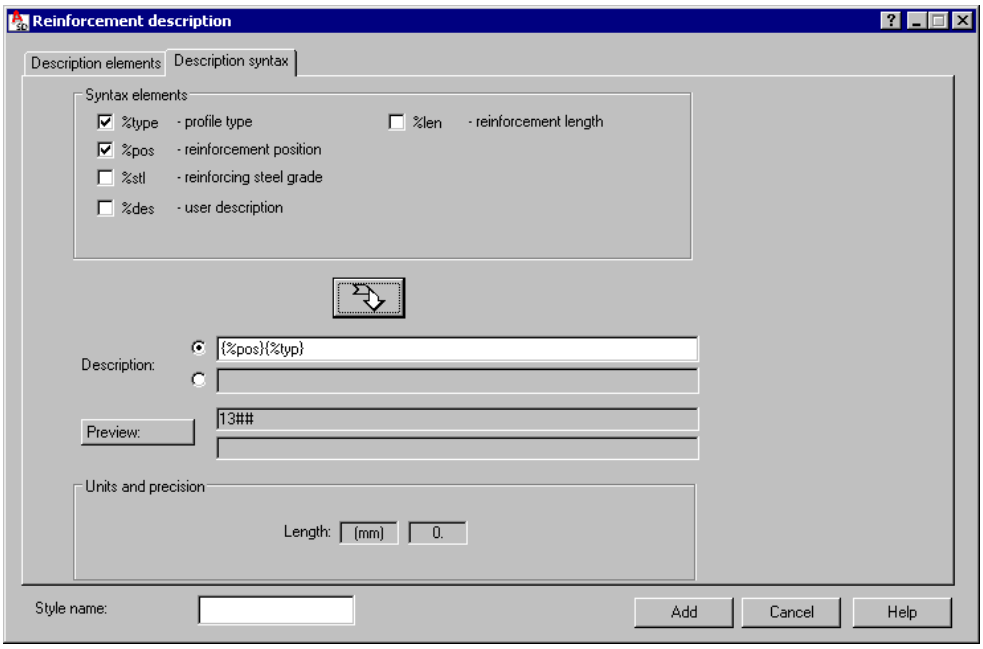
The first column of parameters concerns a label and allows selecting a color of an active and not active position, line thickness, label shape and size. In the second column parameters of a position number in the label are defined: default position on the extension line, color of an active and not active position, font style and size. Options used to select a color for an active and not active position are linked with the *Active* option (see the drawing below) in the Reinforcement description dialog box. If the *Active* option is switched on in the ***Reinforcement description*** dialog box, then a color of this position is chosen from the *Active* list; if it is switched off in the dialog box below, then a color of this position is selected from the *Not active* list.

The options in the *Description text appearance* field refer to a description placed between the position (label) and the described steel profile. The following parameters can be set here: font style, color, size and location: above or above and under the line as well as a vertical translation with respect to the auxiliary line.

Options in the *Auxiliary line* field allow determining the following parameters of a line connecting the position with the steel profile: color, thickness and type of arrowheads. Additionally, there is a parameter that specifies a size of the arrow.

20.5.3. Description syntax

The Reinforcement description (steel profiles) dialog box looks as presented in the drawing below after selecting the *Description syntax* tab in it.



Options on this tab are used to define a syntax of the steel profile description.

By selecting appropriate description components in the *Syntax elements* field, the user may move them to the *Description* fields (by pressing the arrow located in the central part of the dialog box), where variables can be freely arranged together with texts, symbols, etc. given by the user.

The *Preview* field shows the steel profile description resulting from a defined syntax. This description is based on numerical values saved as fixed and responds to changes in the preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

Under the field with a description is the information concerning units and precision used while displaying a given value.

The list of variables included in a steel profile description is as follows:

%type – profile type

%len - reinforcement (profile) length

%pos - reinforcement position

%stl - reinforcing steel grade

%des - user description.

NOTE:
The manner of reinforcement description depends on a type of described reinforcement and a country whose code is applied.

Variables that may appear in a syntax, must be put in parentheses {}; the user may add any texts between the successive variables put in these parentheses.

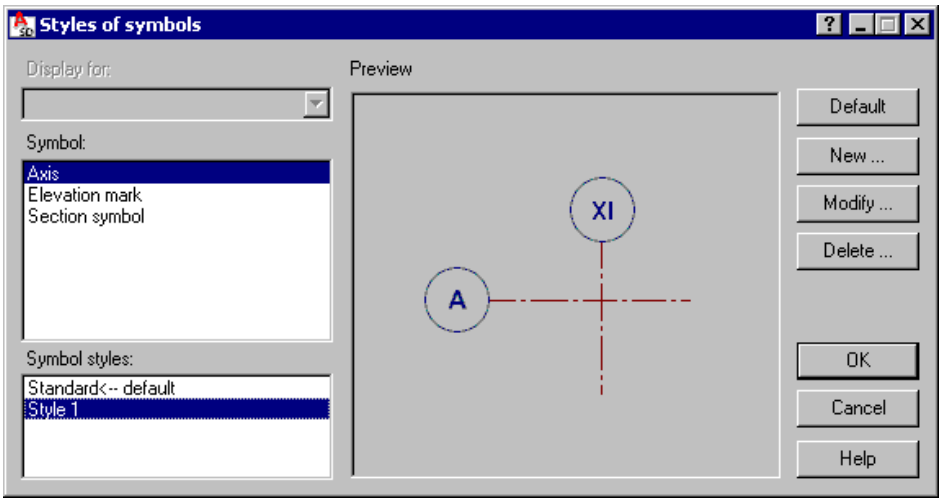
21. STYLES OF SYMBOLS

21.1.Styles of symbols

The option enables definition/modification of symbols of axes, levels and sections used in elements of an RC structure. The option is available from:

- the menu by selecting the option *Reinforcement / Graphic elements / Styles – graphic elements*
- the command line: RBCT_DEF_SYMBOL_STYLE.

After activating the option the ***Styles of symbols*** dialog box, shown in the drawing below, appears on the screen.



The program offers the following types of symbols used for elements of RC structures (presented in the *Symbol* field in the above dialog box):

- axis symbol
- elevation mark symbol
- section symbol.

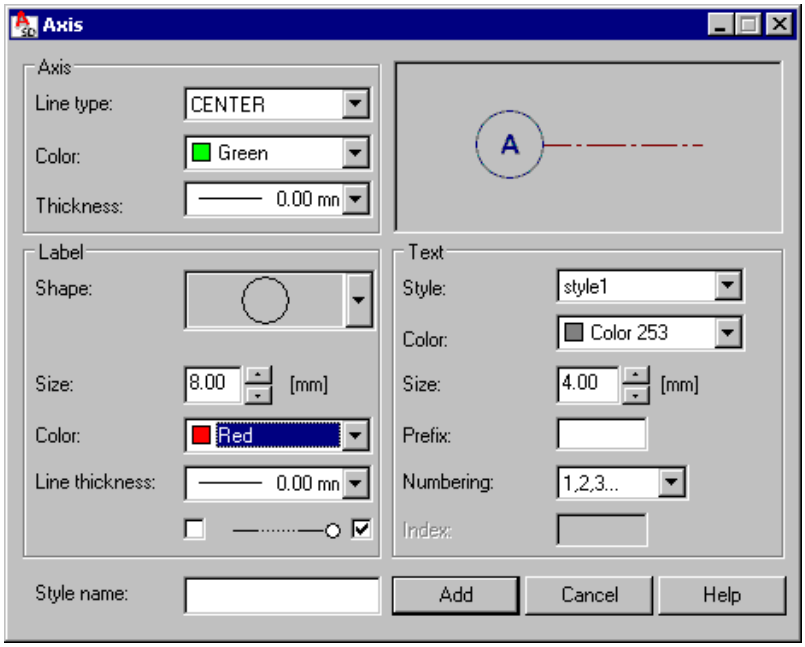
For every symbol type a standard style has been defined in the program. After highlighting a symbol type and a symbol style, the current view of the symbol of an axis, level or section is presented in the central part of the dialog box (in the *Preview* field).

The right-hand part of the dialog box holds the following buttons (apart from the standard buttons **OK, Cancel** and **Help**):

- **Default** – pressing this button sets a selected style as a default symbol style for an axis, level or section
- **New** - pressing this button opens one of the dialog boxes: Axis, Elevation mark or Section symbol, where a new style of the selected symbol type can be defined (based on the existing style)
- **Modify** - pressing this button opens one of the dialog boxes: Axis, Elevation mark or Section symbol, where a selected symbol type can be modified
- **Delete** - pressing this button deletes a highlighted style from the list of styles available in the *Symbol styles* field.

21.2. Axis

The dialog box is used to define a new style or to modify an existing style of an axis. It opens on pressing the buttons **New** or **Modify** in the Styles of symbols dialog box if an axis style is chosen in the *Symbol* field.

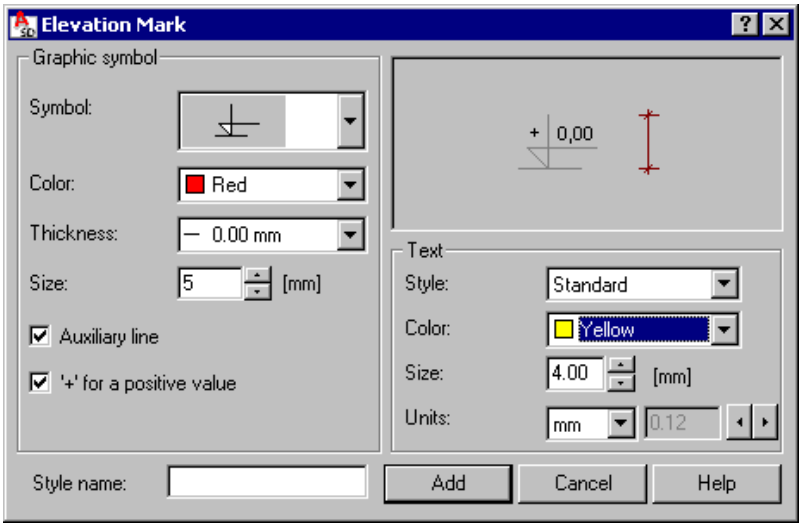


The *Axis* field holds parameters of a line presenting the axis in a drawing: line type, color and thickness.
In the *Label* field there is a possibility to determine parameters of a label of the axis description: label shape, size, color and thickness of the label line; the following label shapes are available in the program: circle, ellipse, square, octagon. At the bottom of this field there are options that allow switching on / off the display of a label at ends of the axis.
The options in the *Text* field refer to the axis description provided in the label. Settings of the following parameters can be defined here: font style, color and size; additionally, it is possible to choose a structure axis prefix and a numbering type: with letters: A, B, C, with numerals: 1, 2, 3, with numerals: I, II, III or by determining any other designations (after selecting the *Define* option).

To add a new axis description, specify a name of the axis description style (in the *Style name* field) and press the **Add** button.

21.3. Elevation mark

The dialog box is used to define a new style or to modify an existing style of an elevation mark. It opens on pressing the buttons **New** or **Modify** in the Styles of symbols dialog box if an elevation mark style is chosen in the *Symbol* field.



The *Graphic symbol* field holds parameters of a symbol representing a level in the drawing: symbol type, color, size and thickness of the line in the symbol. Below are two options:

- *Auxiliary line* – if this option is switched on, then apart from the elevation mark symbol, the drawing will present an auxiliary line connecting the elevation mark symbol with the level of an RC element; when the option is switched off, the elevation mark symbol will be shown without any additional lines
- *'+' for a positive value* - if this option is switched on, then the '+' symbol will appear additionally when a value of a number presented in the elevation mark symbol is greater than zero; if this option is switched off, a positive value will be presented in the elevation mark symbol without any additional symbol.

The options in the *Text* field refer to the elevation mark description provided in the elevation mark symbol. Settings of the following parameters can be defined here: font style, color and size; additionally, it is possible to select a unit used to present a level value and number precision (a number of decimal places).

To add a new elevation mark description, specify a name of the elevation mark description style (in the *Style name* field) and press the **Add** button.

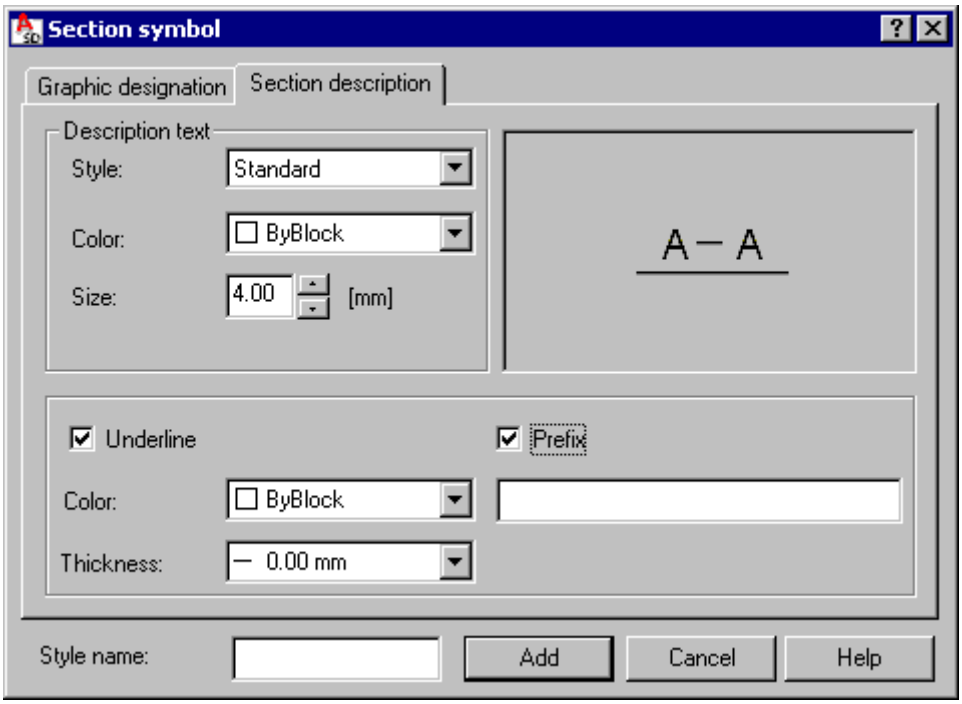
21.4.Section symbol

The dialog box is used to define a new style or to modify an existing style of a section of an RC structure element. It opens on pressing the buttons **New** or **Modify** in the Styles of symbols dialog box if a section symbol style is chosen in the *Symbol* field.
The dialog box consists of two tabs:
Graphic designation
Section description.

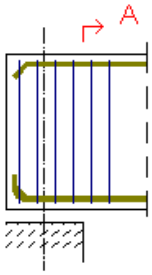
To add a new description of a section symbol, specify a name of the description style of the section symbol (in the *Style name* field) and press the **Add** button.

21.5. Graphic designation

The dialog box as shown in the drawing below appears on the screen once the *Graphic designation* tab is selected in the Section symbol dialog box.



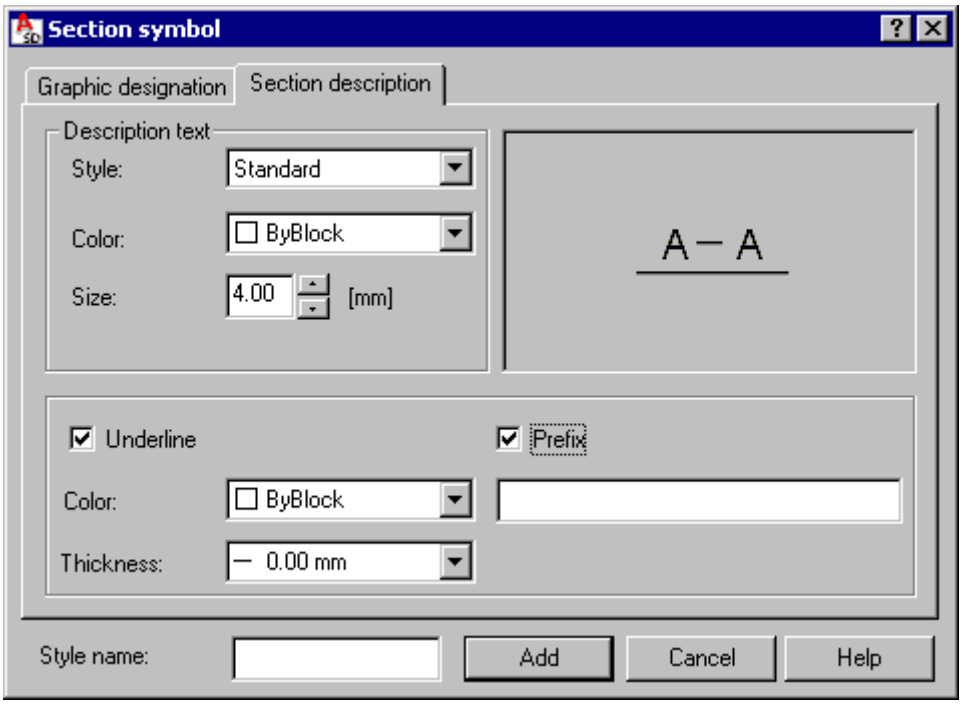
The *Position* field holds parameters of a symbol representing the section of an RC structure element in a drawing: graphic symbol type, arrowhead of the section symbol, symbol color, size and thickness of a line in the symbol; besides, it is possible to select numbering of the section symbol: with letters: A, B, C, with numerals: 1, 2, 3, with numerals: I, II, III or by determining any other designations (after the *Define* option is selected). An example of a section designation is illustrated in the drawing below.



The options provided in the *Number* field refer to a section description. The following parameters can be set: font style, color and size.

21.6. Section description

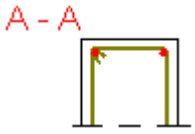
The dialog box as shown in the drawing below appears on the screen once the *Section description* tab is selected in the Section symbol dialog box.



The options located in the *Description text* field concern the description of an RC element section. The following parameters can be set: font style, color and size. Below are two options:

- *Underline* – if this option is activated, then the section description (name) will be additionally underlined with a line of a selected color and thickness; if this option is switched off, the section name will not be underlined
- *Prefix* - if this option is activated, then an additional edit field is accessible in which it is possible to define any character string being a prefix of the section name.

An example of a designation of the section name is illustrated in the drawing below.



22. REINFORCEMENT TABLES

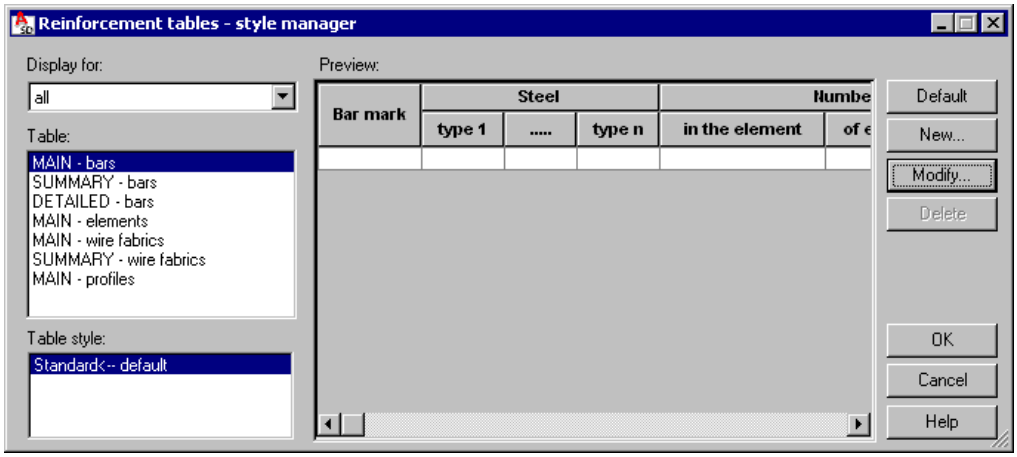
22.1.Reinforcement tables (style manager)

The option enables definition/modification of tables used to prepare the tables of reinforcement in RC structure elements. The option is available from:

- the menu by selecting the *Reinforcement / Reinforcement table / Styles - reinforcement tables* option
- the command line: RBCR_LIST_PAR.

NOTE:
Reinforcement tables in ***AutoCAD Structural Detailing - Reinforcement*** *are updated automatically after modifying a drawing, if in the* ***Options*** *dialog box (the Structural Detailing tab) the Automatic table update option is switched on.*

After selecting the option, the ***Reinforcement tables - style manager*** dialog box shown in the figure below appears on the screen.



The following types of the reinforcement table are available in the program (they are presented in the *Table* field of the above dialog box):

- MAIN - reinforcing bars
- SUMMARY - reinforcing bars
- DETAILED - reinforcing bars
- MAIN - elements (see Element manager)
- MAIN - wire fabrics
- SUMMARY - wire fabrics
- MAIN - (steel) profiles.

Individual types of reinforcement tables present the following information:

- MAIN table – it is a global reinforcement table
- SUMMARY table – it is a reinforcement table intended only for the distribution varying linearly (e.g. for stirrups distributed within a tapered beam), to obtain such a table, the user should choose reinforcement distribution varying linearly and select the option *Reinforcement table / Summary table*
- DETAILED table – it is a summary table concerning the reinforcement steel consumption broken down by bar diameters (see also: Generation of distribution varying linearly and detailed table) – available for distribution varying linearly and bar surface distribution.

For each table type the standard table style (it is presented in the *Table style* field) has been defined in the program. Once the table type and table style are highlighted, the view of a selected table style is presented in the middle part of the dialog box (the *Preview* field).
The right part of the dialog box (apart from the standard buttons **OK**, **Cancel** and **Help**) contains the following buttons:

- **Default** - pressing this button sets a selected style as a default table layout (description style)
- **New** - pressing this button opens the Definition of new reinforcement table style dialog box where a new style of a selected table type can be defined (based on the style that already exists)
- **Modify** - pressing this button opens the Modification of reinforcement table style dialog box where changes can be made in a selected table type and table style
- **Delete** - pressing this button deletes the highlighted table style from the list of styles available in the *Table style* field.

22.2.Style definition/modification

22.2.1. Definition/modification of reinforcement table style

The dialog box is used to define a new style or to modify an existing style of a table type.

The ***Definition of new reinforcement table style*** dialog box can be opened after pressing the **New** button in the Reinforcement tables (style manager) dialog box (the ***Modification of reinforcement table style*** dialog box can be opened after pressing the **Modify** button).

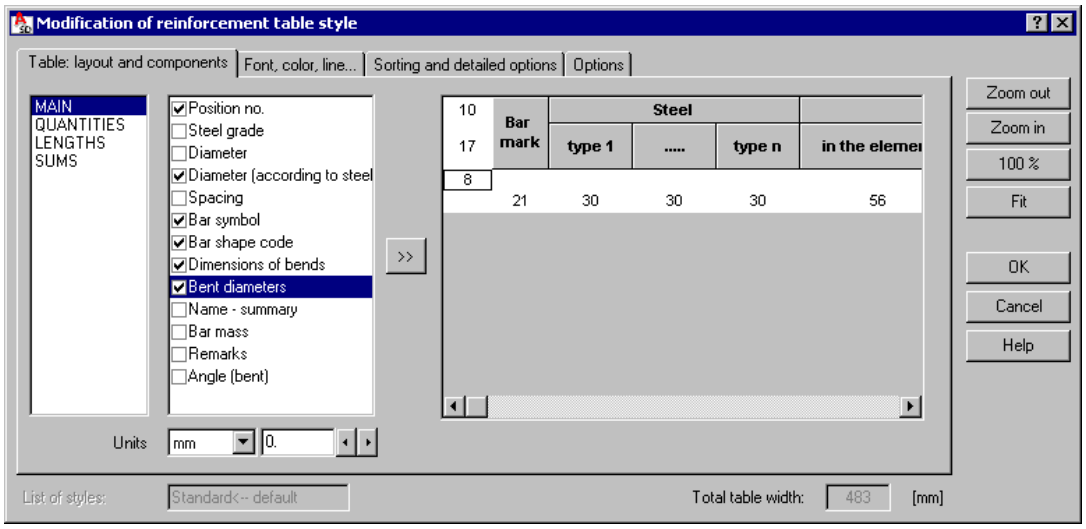
The dialog box consists of three tabs:
Table: layout and components
Font, color, line
Options.

NOTE:
If one of the reinforcement table types for bars is chosen (MAIN - bars, MAIN - elements, SUMMARY - bars, DETAILED - bars, MAIN - profiles), there is also another tab available: Sorting and detailed options.

NOTE:
If the type SUMMARY – wire fabrics is chosen, then another tab Display options (wire fabric cuts) is also available for wire fabrics.

22.2.2. Table layout and components

The drawing below presents the dialog box that appears once the *Table: layout and components* tab is selected in the Definition/modification of reinforcement table style dialog box.



NOTE:
The options located on this tab depend on a table type selected in the ***Reinforcement tables - style manager*** *dialog box. The drawing above shows the options available after selecting the main table for reinforcing bars.*

The bottom part of the dialog box contains the *Style name* edit field; a name of the defined table style should be entered there (in case of modifying the table style, the *Style name* field is inaccessible).
To define/modify a table style, the user should:

- select a set of table components (e.g. in the dialog box shown above they include: *Main, Quantities, Lengths, Sums*)
- in the next field switch on the components to be included in the table (the option is switched on when √ symbol appears)
- press the **>>** button.

The right part of the dialog box presents the table layout defined.

For example, for a bar symbol (shape) to be presented in the MAIN reinforcement table, the user should:

- in the Reinforcement tables – style manager dialog box select the MAIN table
- press the **Modify** button
- on the *Table: layout and components* tab in the Modification of reinforcement table style dialog box switch on the *Bar symbol* option
- press the **>>** button which causes a new column to be added to the main reinforcement table
- set the added column with the bar symbol in the appropriate place in the table
- press the **OK** button.

The remaining components of a reinforcement table can be added in the similar manner.

NOTE:
Column positions and column names may be modified freely for each reinforcement table style.

The following options are also available in the above dialog box:

- *total table width* - inaccessible edit field which displays the defined table width determined in the program
- buttons: **Zoom in**, **Zoom out**, **100%**, which allow zooming in/zooming out the table presented.

- the **Fit** button, which when pressed, enables fitting the width of table columns to the length of texts in column headings.

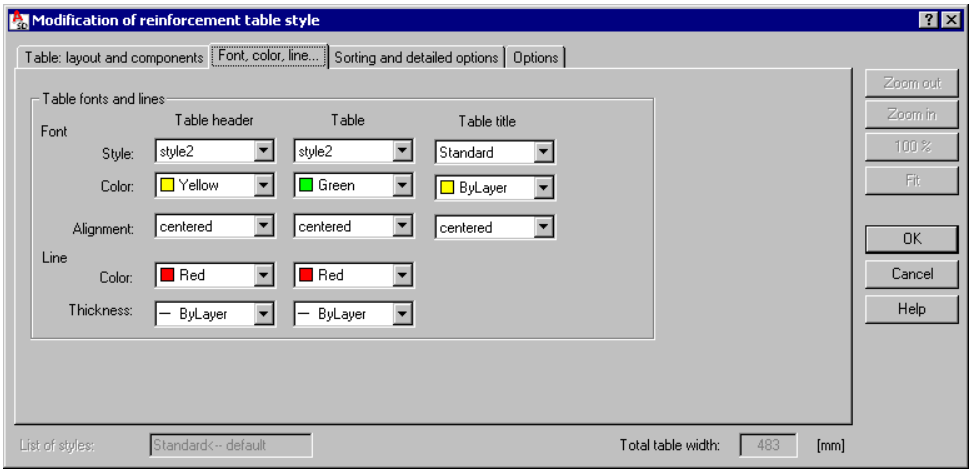
The order of individual table columns may be freely arranged. To do that, the user should select the entire column and next, holding the left mouse button pressed, move the column to the selected position. Moreover, in a table the cell height may be increased and in the table heading additional user-defined descriptions or changed names of the existing columns may be entered.

The tables enable the user to:

- add blank lines at the beginning and end of a table
- add blank columns in tables
- define exactly table dimensions (column width and line height) – dimensions of the column width and the line height are presented under the table and to the left of the table, respectively.

22.2.3. Font, color, line

The drawing below presents the dialog box that appears once the *Font, color, line* tab is selected in the Definition/modification of reinforcement table style dialog box.

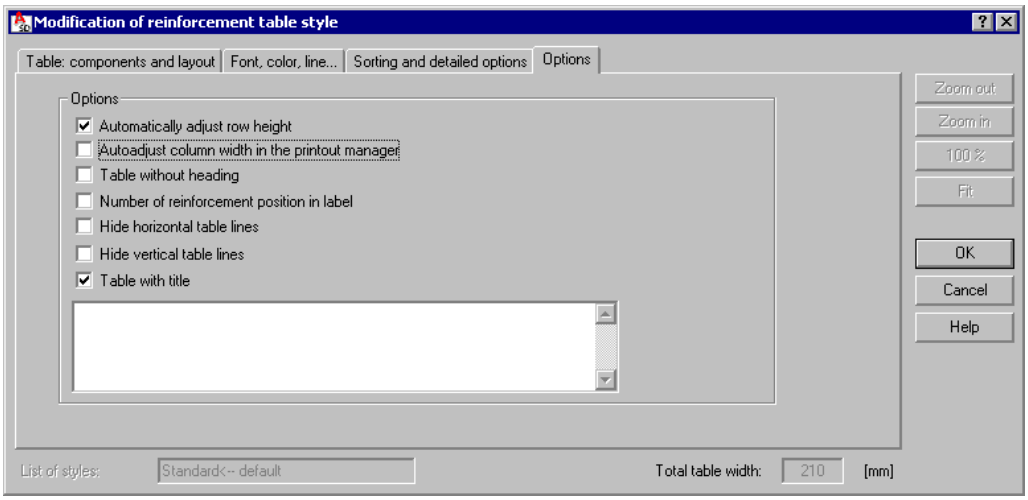


The options available in the above dialog box enable the user to:

- determine a font used in the table (in the table header and all table cells): style, color and alignment of table header and texts in table cells
- select table lines: thickness and color
- define a font used in the table title (if the *Table with title* option is activated on the *Options* tab): style, color and alignment of table title.

22.2.4. Options

The drawing below presents the dialog box that appears once the *Options* tab is selected in the Definition/modification of reinforcement table style dialog box.

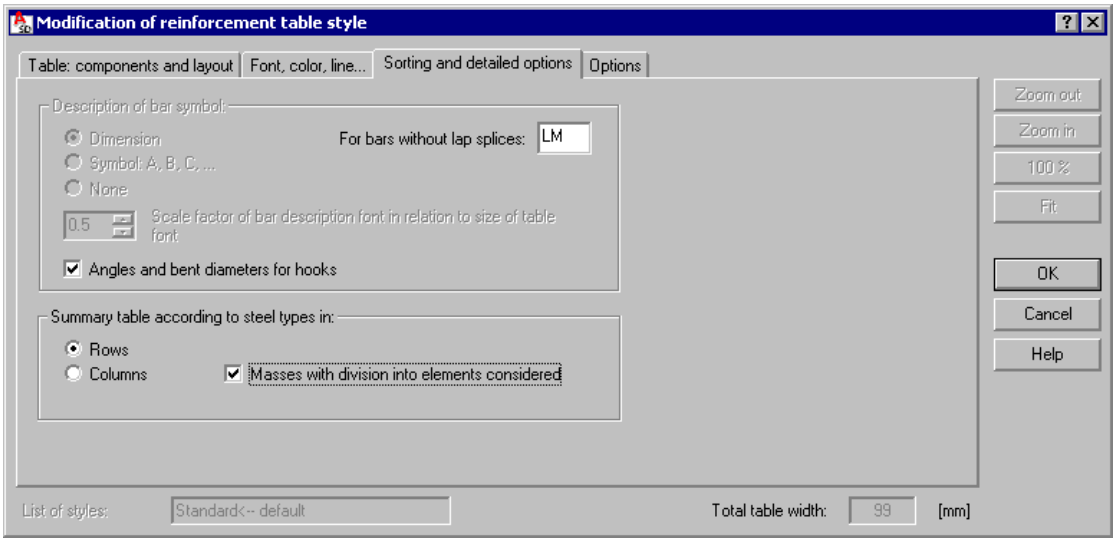


The following options are available in the *Options* field:

- *automatically adjust row height* - if this option is on (√ symbol appears), then height of table rows will be automatically adjusted to the size of symbols of element shapes presented in the table
- *autoadjust column width in the printout manager* - if this option is on (√ symbol appears), then widths of table columns will be automatically adjusted to fit lengths of titles of table columns (see the description of the Table printout manager dialog box)
- *table without heading* - if this option is on (√ symbol appears), then the table generated will not have a heading
- *number of reinforcement position in label* - if this option is on (√ symbol appears), then a number of reinforcement position in the table will be presented in a circular label
- *hide horizontal table lines* - if this option is on (√ symbol appears), then horizontal lines will not be shown in the table
- *hide vertical table lines* - if this option is on (√ symbol appears), then vertical lines will not be shown in the table.
- *table with title* - if this option is on (√ symbol appears), then the edit field in the lower part of the dialog box becomes accessible and a table title may be typed there; a font used in the table title may be defined on the *Font, color, line* tab.

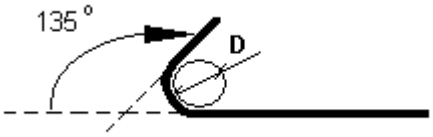
22.2.5. Sorting and detailed options

The drawing below presents the dialog box that appears once the *Sorting and detailed options* tab is selected in the Definition/modification of reinforcement table style dialog box.



NOTE:
The tab is available only in case of reinforcement table types for bars (MAIN, SUMMARY, DETAILED).

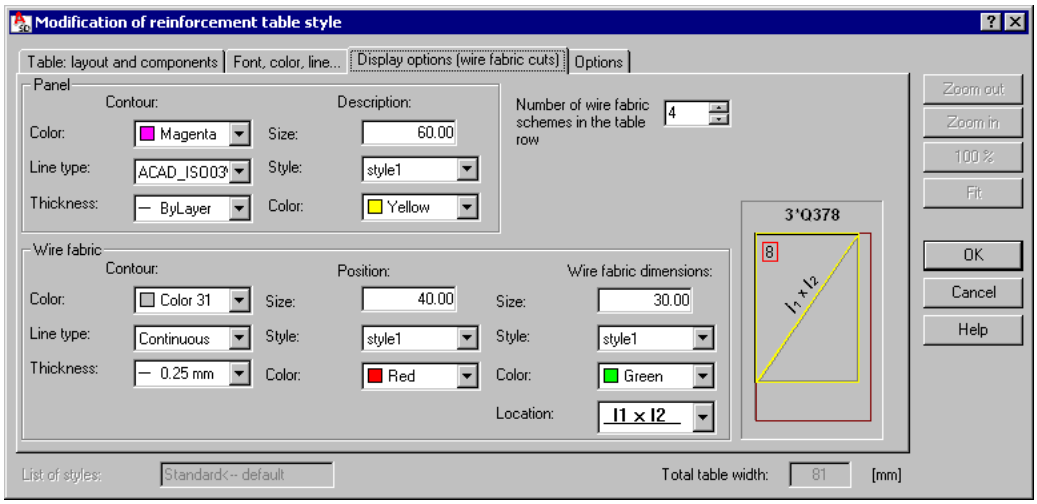
In the top part of the dialog box the description of bar symbol may be chosen; there are three possibilities of bar symbol description: by specifying a dimension, by presenting symbol of a bar or without any description. The user may also select a scale factor which determines the ratio of the size of bar description font to the size of font in the table. These options are available for the following table types: MAIN and DETAILED.
The *For bars without lap splices* field is used to specify a length designation in a bar symbol in the table; 'LM' bars are bars presented in running meters as a total sum of all segments formed as a result of distribution (it refers only to bars in surface distributions) - see Reinforcement lap.
If the *Angles and bent diameters for hooks* option is switched on, then the table will include information concerning hooks at bar ends: a hook angle and a diameter of hook bending (see the drawing below).



The option *Summary table according to steel types in:* rows or columns is accessible only for the SUMMARY table. Moreover, in the SUMMARY table it is possible to add a row with bar masses (with division into individual elements).

22.2.6. Display options (wire fabric cuts)

The drawing below presents the dialog box that appears once the *Display options (wire fabric cuts)* tab is selected in the Definition/modification of reinforcement table style dialog box.



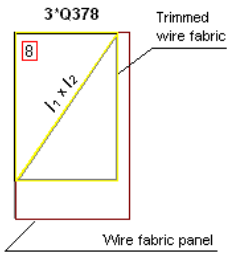
NOTE:
The tab is available only for one type of reinforcement table: SUMMARY for wire fabrics.

In the top part of the dialog box the user may choose a number of schemes to be included in a row of the reinforcement table.

NOTE:
For the wire fabric cuts to be displayed in the table, the Wire fabric cuts option has to be switched on on the Table: layout and components tab.

The options located on the above tab enable defining display parameters for wire fabric panels and generated (trimmed) wire fabrics:

- for wire fabric panels:
  - contour parameters (color, line type and thickness)
  - description parameters (font size, style and color)
- for trimmed wire fabrics:
  - contour parameters (color, line type and thickness)
  - description parameters (font size, style and color)
  - dimension parameters (font size, style, color and location).



23. PRINTOUT

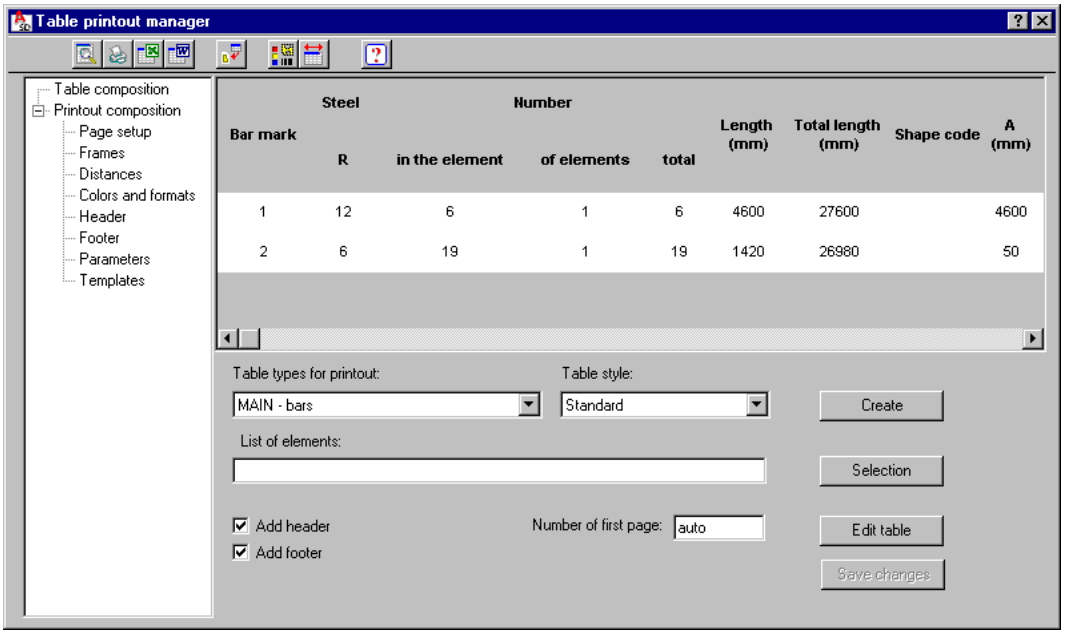
23.1. Table printout manager

The option allows defining/modifying the printout layout for tables used to prepare steel summary tables. The option is available from:

- the menu by selecting the option: *Steel / Reports / Printout* (steel part) or *Reinforcement / Reinforcement table / Table Printout/Export /Edit* (reinforcement part)
- the toolbar by pressing the icon (reinforcement part) or the icon (steel part)
- the command line: RBCS_LISTPRINT (steel part) or RBCR_LIST_EXP (reinforcement part).

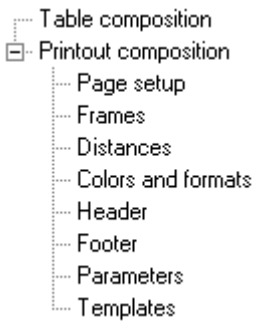
NOTE:
*Reinforcement tables in **AutoCAD Structural Detailing - Reinforcement** will be updated automatically after changes are made in a drawing, if in the **Options** dialog box (Structural Detailing tab) the Automatic table update option is switched on.*

Once the option is selected, the ***Table printout manager*** dialog box, presented in the drawing below, appears on the screen (the dialog box is shown for RC structure elements).



The ***Table printout manager*** dialog box may be divided into two primary parts:

- the left part of the dialog box contains the selection tree (see the drawing below) from which the user, by means of the mouse, selects one of the printout manager options



- to the right of the selection tree there is the part of the dialog box which contains appropriate parameters for the option selected by the user in the selection tree; the dialog box is updated after selecting the option by the user. The top part of the dialog box shows the layout of a selected table type.
- The top part of the dialog box contains a few icons:
  - *Printout preview* - pressing this icon opens a preview of a table printout; the user may return to the dialog box by pressing the **Close** button
  - *Print table* - pressing this icon starts printing a table
  - *Save table* - pressing this icon opens the dialog box in which a table may be saved in the format of MS Excel © program. A table may be saved to:
    ⇒ *.CSV (Comma Separated Values) format files which are text files
    ⇒ *.XLS format files – this saving method fully reflects table settings that can be seen in a preview window
  - *Save table (MS Word)* - pressing this icon opens the ***Save As*** dialog box which enables saving a table in an MS Word file with the specified name
  - *Save graphical settings* - pressing this icon enables saving current settings of the printout manager
  - *Automatic adjust of column width to header text* - pressing this icon results in adjusting the width of table columns to the length of table column names.
  - *Help* - pressing this icon opens Help.

NOTE:

*For **AutoCAD Structural Detailing - Reinforcement** there is another icon*  *available; when pressed, it enables inserting a table into an edited drawing.*

It should be mentioned here that, although a table shape (cell height, column width) depends directly on a defined table style, the user may freely determine it (i.e. cell height or column width).

In addition, there is a context menu available in the table; it contains the following options:

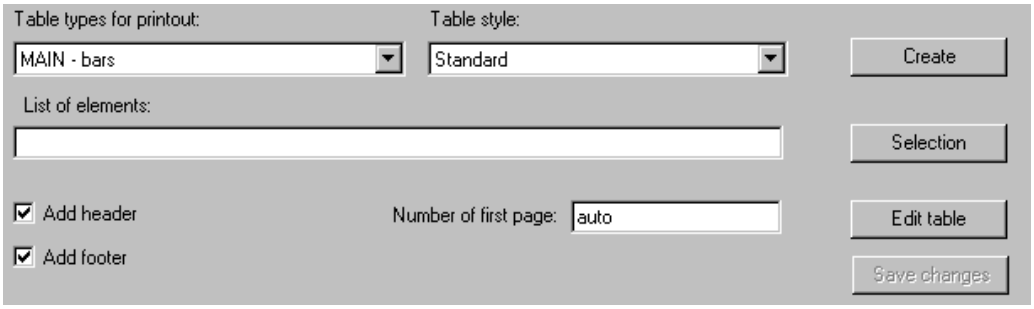
- table cells – the options: group (merges several table cells into one cell) and ungroup
- text orientation – vertical, horizontal
- adding or deleting a column in the table
- adding or deleting a row in the table.

An example of table layout after adding a new column at the beginning of the table, merging several table cells and changing text orientation to vertical, is illustrated in the figure below; the table presents data for a few positions of structure element reinforcement.

Element	Bar mark	Steel		Number			Length (mm)	Total length (mm)
		Grade 36	Grade 60	in the element	of elements	total		
beam 1	1		12	1	1	1	2680	2680
	2		12	1	1	1	2330	2330
column 1	3	6		1	1	1	3190	3190
	4	6		1	1	1	3390	3390

23.2.Table composition

Once the *Table composition* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



For steel structure elements the user may choose one of the following table types:

- Material summary
- Plate summary
- Profile summary
- User parts
- Element list
- Assembly list
- Bolt (rivet) list.
- Cut list
- Bolt assign.

For RC structure elements the user may choose one of three table types:

- Main (reinforcing bars)
- Summary (reinforcing bars)
- Detailed (reinforcing bars) - it is necessary to select graphically the distribution varying linearly / surface bar distribution or to enter a number of reinforcement position
- Main (elements) – see the description of the ***Element manager*** dialog box
- Main (wire fabrics)
- Summary (wire fabrics).

For each table type the user may choose a table style defined previously for the selected table type.

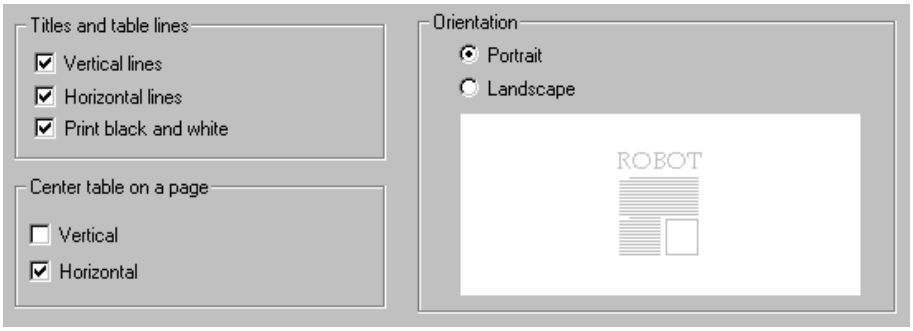
Pressing the **Create** button generates a table of a given type based on the current element list (if the list is empty, then a printout of a whole structure is made) in the format of the active table style (see: Styles applied in tables).

In the *List of elements* field numbers of the elements included in a table are displayed. Pressing the **Selection** button closes the dialog boxes and enables the user to select graphically elements to be included in the table.
Pressing the **Edit table** button enables the user to indicate graphically the table to be edited. Changes made in a table may be saved after pressing the **Save changes** button.
The bottom part of the dialog box contains the following options:

- *Number of first page* - the field in which the user may specify a number of the first page to be printed
- *Add header* - if this option is turned on, then a defined header will appear on a printout
- *Add footer* - if this option is turned on, then a defined footer will appear on a printout.

23.3.Page setup

Once the *Page setup* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

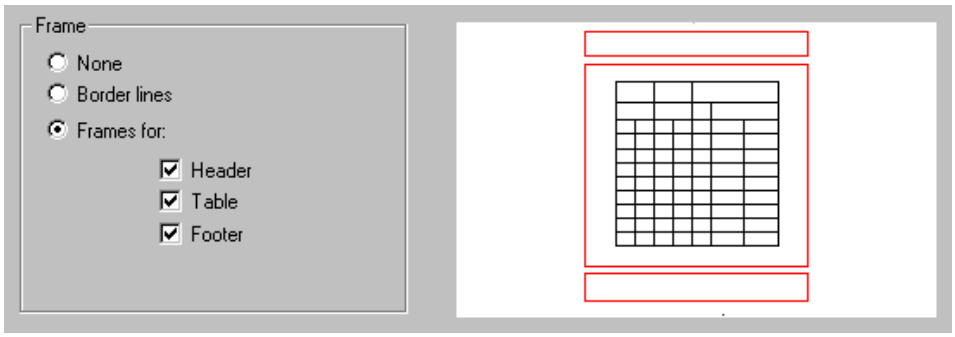


In the above dialog box the user may determine the manner of table presentation:

- in the *Titles and table lines* field:
  - if the *Vertical lines* option is turned off, then no vertical lines are displayed in a table
  - if the *Horizontal lines* option is turned off, then no horizontal lines are displayed in a table
  - if the *Print black and white* option is turned off, then a table is printed with the defined colors applied
- the options in the *Center table on a page* field determine how a table is to be centered (horizontally, vertically or both vertically and horizontally)
- in the *Orientation* field the user may define paper orientation (horizontal - the longer side of a paper sheet is horizontal, vertical - the longer side of a paper sheet is vertical).

23.4.Frames

Once the *Frames* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

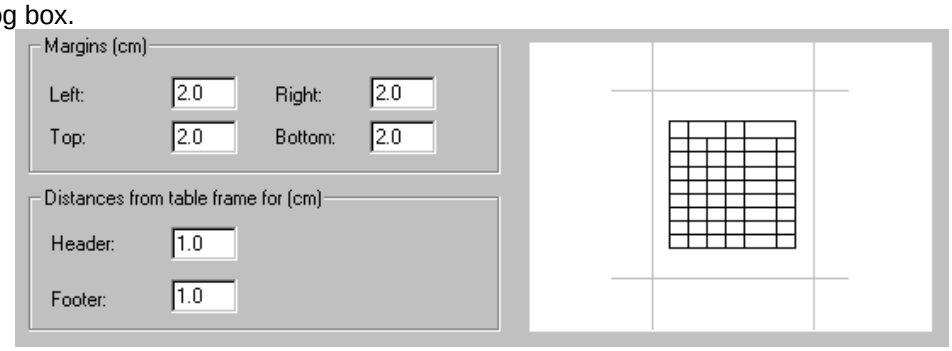


In the above dialog box the user may determine the manner of table presentation on the page:

- no separation between a header/footer and table
- with a border line that separates header and footer from the table
- headers and footers presented in frames (the user may select a frame only for a footer, only for a header, only for a table or combine frames of the elements listed).

23.5.Distances

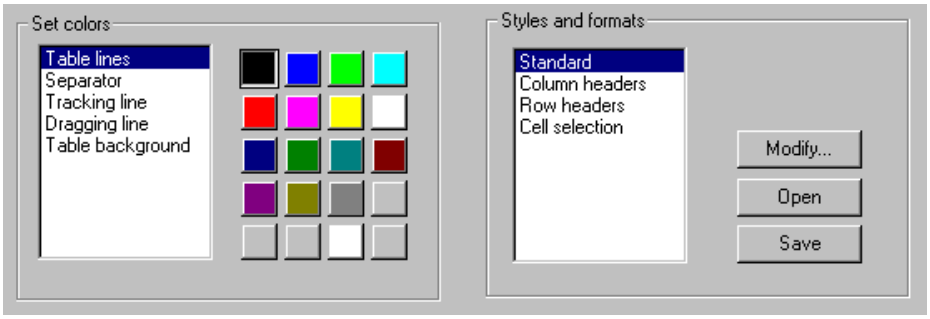
Once the *Distances* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box allows determining (identically as in each text editor) page margins: left, right, top and bottom. Moreover, the user may define distances between the table frame and header or footer.
The size of a header and footer is calculated automatically in the program; the parameters mentioned depend on a size of the applied font, size of a drawing with the company logo and number of lines required in a header or footer.

23.6.Colors and formats

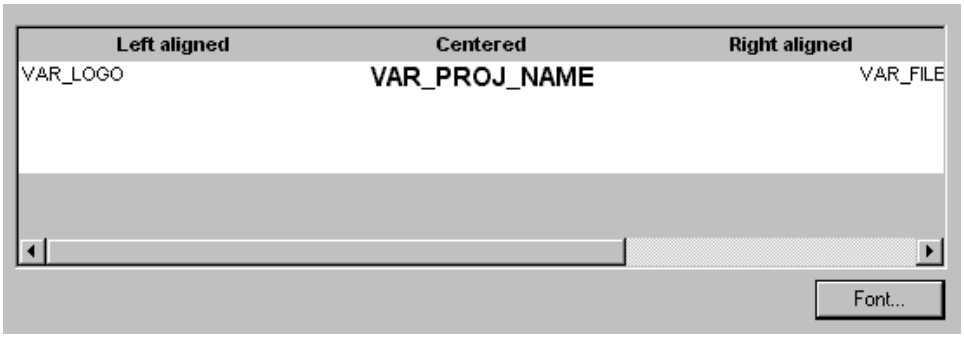
Once the *Colors and formats* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



In the *Set colors* field the color of the following table elements may be chosen: table lines, separators, tracking lines, dragging lines and table background.
The *Styles and formats* field allows selection of formats and styles applied in the following table elements: table column headers, table row headers and text contained within a table. Pressing the **Modify** button opens the dialog box where the format (font, font color, alignment method) for the enumerated table elements may be chosen.

23.7.Header

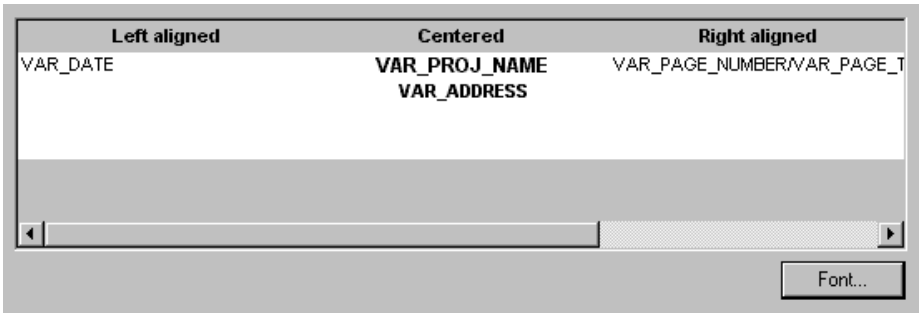
Once the *Header* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box shows the layout of the printout header.
In order to change the header layout, the user should press the table field presenting the header layout; then the list of available variables unfolds, on which an appropriate variable may be selected. When the cursor is positioned in a field of the table presenting the header layout, then pressing the **Font** button opens the dialog box where the user may choose the font to be applied in a selected field.

23.8.Footer

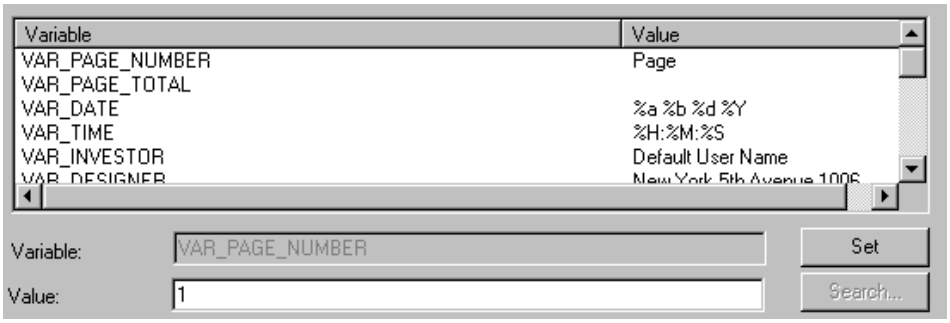
Once the *Footer* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box shows the layout of the printout footer.
In order to change the footer layout, the user should press a field of the table presenting the footer layout; then the list of available variables unfolds, on which an appropriate variable may be selected. When the cursor is positioned in a field of the table presenting the footer layout, then pressing the **Font** button opens the dialog box where the user may choose the font to be applied in a selected field.

23.9.Parameters

Once the *Parameters* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box contains all the variables defined in the system and their names.

NOTE:
*Setting of a variable and next, its modifying **must** be confirmed by pressing the **Set** button.*

The variables are used for formatting header and footer of the printout:
VAR_PAGE_NUMBER - variable that allows assigning a current printout page. A text assigned to the variable will be preceding the total number of pages, if the VAR_PAGE_TOTAL variable is used (e.g. if the "Page **VAR_PAGE_NUMBER**" value is ascribed to the variable, then on the printout each page will be printed in the following form: Page 1, Page 2, etc.)
VAR_PAGE_TOTAL - value of this variable indicates the total number of printout pages. A text assigned to it may be preceded by a current page number, if the VAR_PAGE_NUMBER variable is used.
VAR_DATE, **VAR_TIME -** these variables may be assigned any text and combination of the key words presented below (it enables printing the current date/time on a printout); allowable formats include:
%A - full week day name (Monday)
%a - abbreviated week day name (Mon)
%B - full month name (January)

%b - abbreviated month name (Jan)
%c - standard presentation of date and time
%d - month day (01-31)
%H - time (24-hour clock) (00-23)
%I - time (12-hour clock) (01-12)
%j - successive day of the year (001-366)
%M - minute (00-59)
%m - month (01-12)
%p - local equivalent of the English abbreviations AM / PM
%S - second (00-59)
%U - successive week of the year (first day - Sunday) (00-53)
%W - successive week of the year (first day - Monday) (00-53)
%w - day of the week (0-6, Sunday is denoted by 0)
%X - standard time representation
%x - standard date representation
%Y - year and century
%y - year without the century specified (00-99)
%Z - time zone name
%% - percent mark.
Standard date representation is the following string of variables: %a %b %d %Y
Standard time representation is the following string of variables: %H:%M:%S
Standard date and time representation is the following string of variables: %a %b %d %H:%M:%S %Y.

The remaining variables listed below, do not contain other values than texts ascribed to them by the user. Their names serve only for the purpose of convenient classification while formatting.

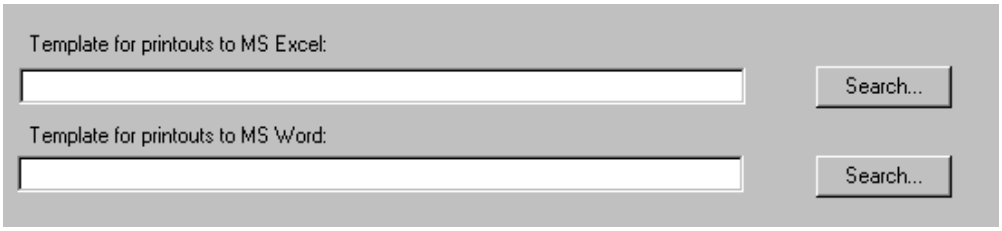
The following variables linked with the printout may be used in ***AutoCAD Structural Detailing - Reinforcement***:
VAR_INV_NAME – investor name
VAR_INV_ADDRESS – investor address
VAR_INV_PHONE – investor phone
VAR_INV_FAX – investor fax
VAR_INV_EMAIL – investor e-mail address
VAR_OFF_NAME – design office name
VAR_OFF_ADDRESS - design office address
VAR_OFF_PHONE - design office phone
VAR_OFF_FAX - design office fax
VAR_OFF_EMAIL - design office e-mail address.
VAR_SCALE – drawing scale
VAR_DRAW_NAME – drawing name
VAR_FILE – name of a DWG file including a drawing
VAR_DESIGNER - designer
VAR_VERIF - verification
VAR_PROJ_NAME, **VAR_PROJ_NUM**
VAR_REV_NAME, **VAR_REV_NUM**
VAR_LOGO – access path to an *.bmp file.
These variables may be also inserted when creating user's own printout layouts. When inserting such a layout the program will automatically fill out variables with values set in the table printout manager.

Moreover, the following variables linked with the printout may be used in ***AutoCAD Structural Detailing - Steel***:
VAR_POS - position name
VAR_SCALE - drawing scale
VAR_PART - part name

VAR_MATERIAL - part material
VAR_LENGTH - part length
VAR_WEIGHT - part weight
VAR_PAINTAREA - painting area
VAR_QUANTITY - number of items of a given position in the project.

23.10. Templates

After selecting the *Templates* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.

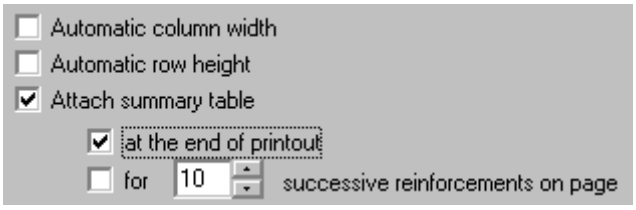


The dialog box above allows selecting templates for printing tables to the following two programs:
- MS Word ©: *.dot format files
- MS Excel ©: *.xlt format files.

In the edit fields the user may specify file names with a full access path; after pressing the **Search** button, it is possible to indicate the template file on the computer hard disk.

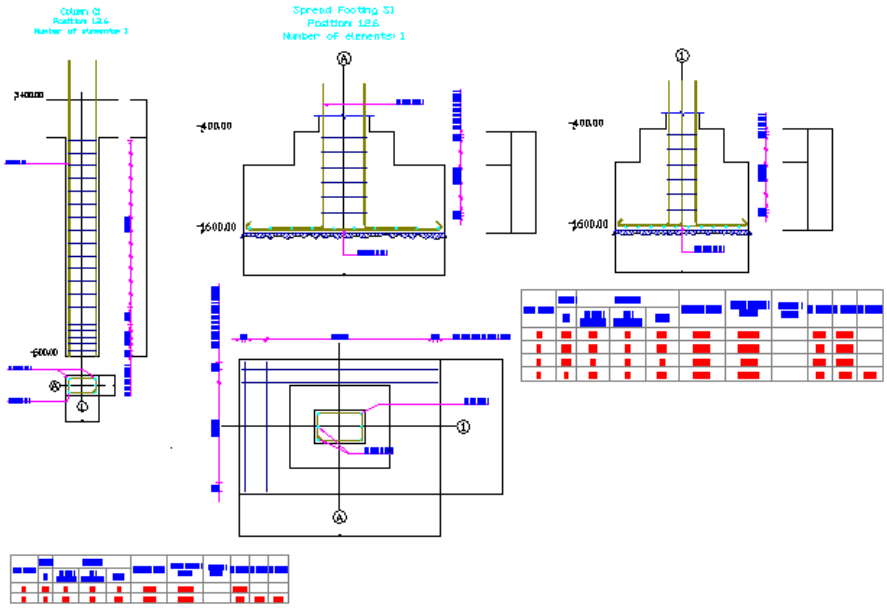
In ***AutoCAD Structural Detailing - Reinforcement***, at the bottom of the dialog box there are additional options available for printouts to MS Word©. They allow inserting an additional summary table including information about the reinforcement (the *Attach summary table* option is switched on); it is possible to set the table either at the end of the entire printout, or for several successive reinforcements on a page.
Switching on the following option:

- *Automatic column width* results in widths of table columns being automatically adjusted to fit the length of names of table columns
- *Automatic row height* results in heights of table rows being automatically adjusted to contents of table rows.



23.11. Example of definition of views and generation of the final drawing

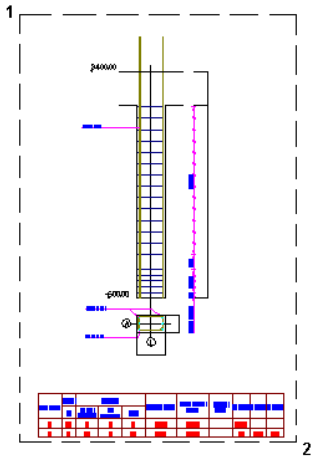
The example illustrates generation of drawings of a spread footing and an RC column generated with the use of the macros available in ***AutoCAD Structural Detailing - Reinforcement*** (the examples of the spread footing and the RC column defined on the *Model* tab are presented in the drawing below).



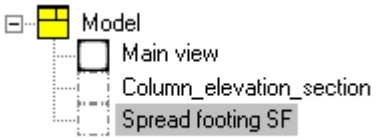
The first stage of drawing generation is to create views in the model space. To create views for projections of the spread footing and the column, do as follows:

- press the *Create view* icon
- indicate the first and second vertex determining e.g.: a view of the column with a reinforcement table (see the drawing below)

NOTE:
A column view and a table view can be created separately (the table and the column in separate views).



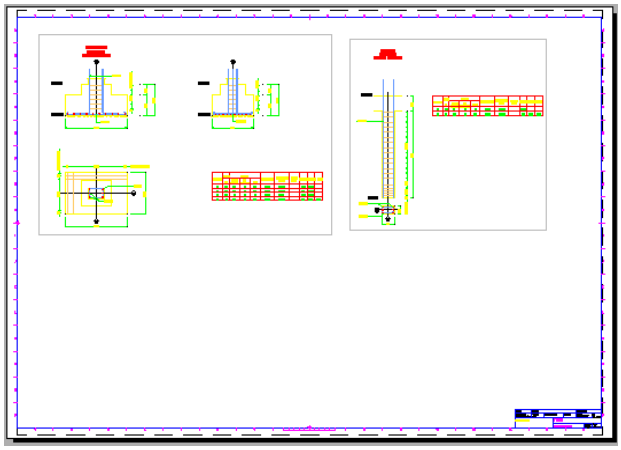
- in the command line enter a name of the view, e.g. *Column_elevation_section* and press the ENTER key
- assume the default scale value 1:20 and press the ENTER key; the name of the defined view appears on the *Positions* tab in the ***Object Inspector*** dialog box
- act similarly when creating a view of the spread footing projections (the view name – Spread footing SF1, default scale)



The second stage of drawing generation is to move the created views to the printout layout. To place the views in the printout layout, do as follows:

- go on to the *Printouts* tab in the ***Object Inspector*** dialog box; add a new printout tab, e.g. template A1 ASD
- in the ***Object Inspector*** dialog box on the *Printouts* tab activate *A1 ASD* (indicate the name A1 ASD and after pressing the right mouse button choose the *Activate* option from the context menu)
- move to the *Positions* tab in the ***Object Inspector*** dialog box
- after indicating the column view (the view *Column_elevation_section*) and pressing the right mouse button choose the *Add to current printout* option from the context menu
- indicate location of the view in the printout layout
- act similarly while placing the spread footing projections on the printout.

The generated printout (the final drawing) has been presented in the drawing below.



23.12. List of commands available in the printout module

In the module that enables printouts the following commands are available:

RBCT_ACTDOC
Available for a selected document located in the option tree; the option's task is to activate - in the edition layout - a selected document (for its edition); there may be many documents created, while only one of them is active, i.e. may be edited - in the edition layout.

RBCT_ADDTOPRINT
Available for a selected view located in the position tree; the layout to which the view is to be added, must be active; the option is intended for composing a final printout; the command adds a selected view to the current layout (printout); NOTE: a view may be contained only on one printout.

RBCT_ADDALLTOPRINT
Available for a selected document provided in the position tree; the layout to which the view is to be added, must be active; the option is intended for composing a final printout; the command adds all the views belonging to a given document (not added to other printout, yet) to the current layout.

RBCT_FITVIEWS
Available for an active document - the edition layout must be active, as well; its task is to provide greater work convenience to the user; the command results in adjusting dimensions of views included in the edition layout to the current size of the AutoCAD ® program window.

RBCT_DELPRINTOUT
Available for a selected printout in the printout tree; the aim of the option is to delete a printout; the command also causes deletion of the layout corresponding to it.

RBCT_REMFROMRINT
Available for a selected view provided in the printout tree; the aim of the option is to remove a view from a printout; the command removes the view only from a printout - it remains in a document and may be reused later on (e.g. by adding it to other printout).

RBCT_DELALLFROMPRINT
Available for a selected document located in the printout tree; it is used to remove views belonging to a selected document from the printout; the command removes the views only from the printout - they remain in a document and may be reused later on (e.g. by adding them to other printout).

RBCT_DELDOC
Available for a selected document located in the position tree; the option's task is to delete a document from the list of documents created for a given position; the command deletes all the drawings belonging to the document.

RBCT_EDITVIEW
Available for a selected view provided in the printout tree; it is used to switch to the edition mode for a selected view; the command enables the user to switch in a simple manner from edition of a printout to edition of a view (drawing) included in it; the command activates the edition layout and a document whose component is the selected view.

RBCT_EDITDOC
Available for a selected document provided in the printout tree; it is used to switch to the edition mode of a selected document; the command enables the user to switch in a simple manner from edition of a printout to edition of views (drawings) included in it; the command activates the edition layout and the selected document.

RBCT_MEDIT_ON
Available for an active document containing one view; it allows editing a document in the model layout.

RBCT_MEDIT_OFF
The command restores the standard functionality of the model layout.

RBCT_REG_LAYER
Available for the active document and view; after adding a layer, the user may change its name by means of the options available in the AutoCAD ® program; the program manages layers within the available views and documents; a layer must be added using the ***AutoCAD Structural Detailing*** options (the problem concerns only the layers used in the edition layout).

RBCT_DELETEPOS
Available for a position selected in the position tree; the option's task is to delete a position; NOTE: only positions defined by the user may be deleted - positions created on the basis of model elements cannot be deleted.

RBCT_RENAMEPOS
Available for a position selected in the position tree; the command enables changing the name of a position.

RBCT_RENAMEVIEW
Available for a view selected in the position tree; the command enables changing the name of a view.

RBCT_RENAMEDOC
Available for a document selected in the position tree; the command enables changing the name of a document.

RBCT_RENAMEPRINTOUT
Available for a printout selected in the printout tree; the command enables changing the name of a printout; a printout name is always identical to the name of a layout (printout) corresponding to it.

RBCT_ACTVIEW
Available for a view selected in the position tree; the command makes the view of the AutoCAD ® program which corresponds to the selected view, become active; NOTE: a view must be a component of the active document.

RBCT_ADDTEMPLATE
The command results in adding a new template; in the dialog box which opens once this option is selected, the user may specify the name of a new template and choose one of the registered template types.

RBCT_DELTEMPLATE
Available for a template selected in the template tree; the command deletes a selected template.

RBCT_RENAMETEMPLATE
Available for a template selected in the template tree; the command allows changing a name of a selected template.

RBCT_ACTTEMPLATE
Available for a template selected in the template tree; the command activates a selected template in the template layout (it enables its edition).

RBCT_ADDVIEWPORT
Accessible for an active template in the template layout; the command adds a view to the active template; in the dialog box which opens once this option is selected, the user may choose view name, scale and type; name and scale of the view may be changed at any time, as regards the type, it cannot be modified.

RBCT_DELVIEWPORT
Accessible for a view selected in the template tree; the command enables deleting a view.

RBCT_APPENDDOC
Available for a position selected in the position tree; the command allows adding a document to the selected position; in the dialog box which opens once this option is selected, the user may specify the name of a document and select a template based on which the document is to be created.

RBCT_CNGSCALE
Accessible for the active view located in the edition layout; the command allows changing the view scale; the scale is expressed as the natural number ***n***, which denotes the scale 1 : ***n***.

RBCT_SETVIEWRANGE
Available for the active view located in the edition layout; the command allows setting the view area visible on the printout; the user selects with a rectangle a part of the view which is to be visible on the printout; it should be remembered that edit operations in the edition layout do not result in modification of a defined print area.

RBCT_REFRESHDOC
Accessible for a document selected in the position tree; the command enables refreshing a selected document; if the element of a structure model for which the position has been defined, changes, then the documents created for this position are refreshed; a document that needs to be refreshed is marked with a red diagonal.

RBCT_REFRESHALLDOC
Available for a position selected in the position tree; the command enables refreshing all the documents belonging to a selected position; if the element of a structure model for which the position has been defined, changes, then the documents created for this position are refreshed; a document that needs to be refreshed is marked with a red diagonal.

RBCT_PRINTVFRAMEON
Available for all defined printout views.

RBCT_PRINTVFRAMEOFF
Available for all defined printout views; areas marked with rectangles (they determine views of the AutoCAD ® program) will not be printed.

RBCT_ADDDETAILVIEW
Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view; once the command is selected, the user should select with a rectangle a part of the active view that is to become a new view; a view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout; a new view is added to the position tree; the new view contains copies of the elements included in the selected rectangle; thus, the created view may be edited independently of the initial view; the command is useful in the situation when it is necessary to have another drawing of the same part to edit it independently or two drawings of different scale.

RBCT_ADDVIEW
Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view; once the command is selected, the user should select with a rectangle a part of the active view that is to become a new view; a view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout; a new view is added to the position tree; the new view contains the same elements as those included in the selected rectangle; if the created view is edited, it also makes changes in the initial view; the command is useful in the situation when several independent views is to be created out of one large view (drawing) which is not added to the printout; all the views created are assigned the same scale.

RBCT_DELVIEW
Available for a view selected in the position tree; the command enables deleting a view that has been added by means of the following commands: RBCT_ADDDETAILVIEW or RBCT_ADDVIEW; NOTE: the original view contained in a template, from which the document has originated, cannot be deleted.

RBCT_SHOWIEW
Accessible for a view selected in the position tree; the command allows edition in the active view; if additional views have been added to the document during edition, then not all the views are visible in the edition template.

RBCT_REGMODELLAYERS
Available for the whole project; the command enables work optimization; once this command is selected, in the dialog box that is displayed on the screen, the user may declare the layers intended for work in the model; while working in the edition layout, layout (printout) or template layout, the layers are frozen - it brings about optimization of the **REGENALL** command operation.

24. TOOLS

24.1. Drawing scale

For all the objects drawn by means of the options available in ***AutoCAD Structural Detailing*** units chosen in the Job Preferences dialog box (the *Codes / Units* tab) are applied. Units are selected for reinforcement length, reinforcement cover, reinforcement diameter, etc.
While working in ***AutoCAD Structural Detailing*** the user may adopt different work units (mm, cm, m). When starting work in ***AutoCAD Structural Detailing - Reinforcement*** the default model scale 1 : 20 is set (*Object Inspector / Positions / Main view*); it means that all descriptions of bars and bar distributions as well as dimensions will be inserted in the scale 1:20 (NOTE: bars are always drawn in the scale 1:1). The default scale may be changed in the lower part of the ***Object Inspector*** dialog box.

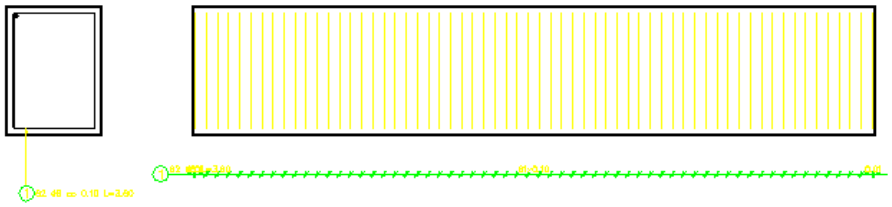
It is possible to work in the program in several scales at the same time using active views. To do this, create a few views and define an appropriate scale for them, e.g. : 1:20, 1:25 or 1:50.

Too draw objects in the scale, e.g. 1:50, activate the view with this scale (it will be highlighted in red) and into this view insert reinforcing bars (the scale of their description will be 1:50). It should always be remembered that on the *Model* tab the scale of an active view is adopted.

The scale of a drawing presented in a selected view may be changed by modifying the scale of this view. Scale modification concerns description of reinforcement distribution, dimension lines, texts and line scale (ltscale) of reinforcement description; once this option is executed, a drawing presented on the *Model* tab shows changed size of letters, labels and dimension lines. When views created in different scale are inserted into a printout layout, the size of fonts and dimension lines are identical.

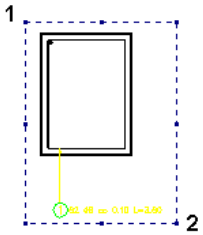
24.2. View scale definition

Drawings for the document shown in the figure below are to be prepared (cross section of an RC beam with a stirrup and longitudinal beam section with stirrup distribution along the beam length).



To create views for the figure above the user should:

- press the *Create view* icon
- indicate the first and second apex defining a view of the beam cross section (see the figure below)



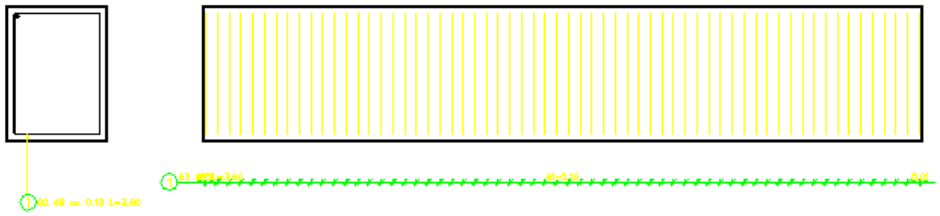
- specify name of the view e.g. *Cross section view* and press the ENTER key
- define scale of the view or assume the default scale value 1:20 and press the ENTER key.

The program has generated a view with the specified name in the active document for the current position.

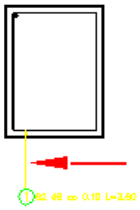
Similarly, follow the above steps to create the other view – view of the beam longitudinal section.

24.3. Different scale of drawings in a view and preparation of printout

Drawings with different scale applied are to be prepared for the document shown in the figure below (cross section of an RC beam with a stirrup and view of a beam with stirrup distribution along the beam length).



Create views in the same manner as described in the topic View scale definition.
To change scale of one of the views (the scale of the view named *Cross section view* will be changed), the user should highlight the cross section view (from the context menu, with the right mouse button select the *Show view* command or double-click on Cross section view) in the ***Object Inspector*** dialog box. Next, in the lower part of the ***Object Inspector*** dialog box define appropriate scale for a given view. All the descriptions included in the cross section view will be modified.



The created views will now be added to the printout. To do that, follow the steps below:

- move to the printout layout tab (e.g. layout1 which is by standard provided in the **AutoCAD ®** program)
- choose printout size (e.g. A2 paper size)
- select the previously created view (e.g. *Cross section view)* in the ***Object Inspector*** dialog box
- press the right mouse button and select the *Add to current Printout* option in the context menu
- indicate the view position on the printout.

Similarly, follow the above steps to obtain the other view (*Longitudinal view*); once this operation is performed, the printout will consist of a beam cross section and a beam longitudinal section with transversal reinforcement presented.

24.4. Find reinforcement

The option enables the user to find a reinforcement element with a given position number in a drawing. The option is available from:

- the menu by choosing the option: *Reinforcement / Tools / Find reinforcement*
- the toolbar by pressing the icon
- the command line: RBCR_TOOL_FINDR.

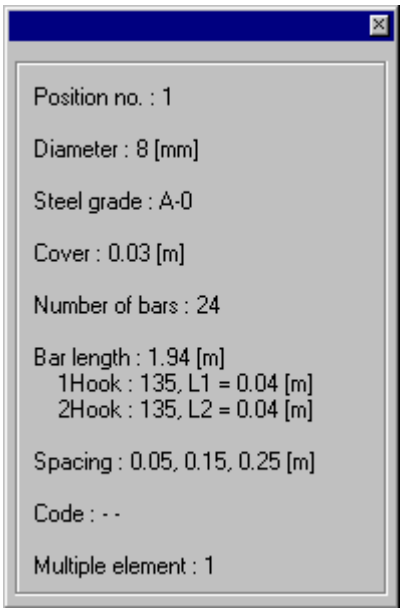
In the command line the user should enter a number of reinforcement position which is looked for. It is also possible to find the next (previous) reinforcement position of the same type (bar, distribution, etc.).

24.5. Reinforcement - information

The option allows displaying basic information about a reinforcing bar or distribution. The option is accessible from:

- the menu by choosing the option *Reinforcement / Tools / Reinforcement - Information*
- the toolbar by pressing the icon
- the command line: RBCR_TOOL_INFO.

If this option is selected, the mouse cursor changes and assumes the shape of a 'cross'. When the cursor is located close to a reinforcement position (reinforcing bar, reinforcement distribution) presented in a drawing, a small dialog box is displayed on the screen in which information concerning the indicated reinforcement is presented. If any piece of information about reinforcement is inaccessible (e.g. position no. or spacing), then in place of a value the '---' symbol is displayed. An example of reinforcement information is shown in the drawing below.



24.6.Renumbering of reinforcement position

The option enables changing numbering of reinforcement. The option is accessible from:

- the menu by selecting the option *Reinforcement* / *Tools* / *Renumbering of reinforcement position*
- the toolbar by pressing the  icon
- the command line: RBCR_TOOL_RENUM.

To carry out renumbering, it is necessary to choose element or group of elements whose numbers will be changed. Renumbering is carried out in an active document.
After generating a final drawing (printout), a change of the numbering in a model results in an automatic change of the numbering in a printout preserving consistency of the numbering; the consistency of numbering in a printout implies checking if a given layout does not include different reinforcement elements with the same position number.

In the default work mode selection of the option causes a single change of a position number of the indicated reinforcement. Additionally, there are the following modes of renumbering available in the program:

- Global renumbering
- Consolidation of renumbering
- Renumbering of identical bars.

Command line:

Select/enter reinforcement no. or [Shift / Consolidation / Identical reinforcement]:
The user should indicate reinforcement description (reinforcement with position) graphically or enter a position number

New reinforcement no.:
The user should enter a new position number; afterwards, the program checks, if the number entered already exists. If yes, then it is verified if both numbers are assigned to identical reinforcement. If not, then a warning is displayed and the user should enter a new reinforcement number again. After completing the renumbering, tables are updated automatically.

Shift

Shift of numbering from position <1> : 5
By default, a first reinforcement position (the one with the lowest number) is proposed by the program. If a different position number (higher number) is entered, then operation of renumbering will be performed for all the positions whose numbers are higher than the specified position number (and for the position with the number entered).

NOTE: The specified position number (in this case 5) must be included in a drawing. If such a position number does not exist, then a warning is displayed and a new position number should be entered.

Position [5] will be changed to: 13
The user should enter a new position number; after completing the renumbering, tables are updated automatically.

Example of shift operation
There are positions nos.: 1 2 3 4 5 6 13 14 15
Shift of numbering from position <1> : 5
Position [5] will be changed to: 13
Result of the renumbering: 1 2 3 4 13 14 21 22 23
Shift of numbering has preserved numbering discontinuities.

Consolidation

Renumbering from position <1>: 5

By default, a first reinforcement position (the one with the lowest number) is proposed by the program. If a different position number (higher number) is entered, then operation of renumbering will be performed for all the positions whose numbers are higher than the specified position number (and for the position with the number entered).

NOTE: The specified position number (in this case 5) must be included in a drawing. If such a position number does not exist, then a warning is displayed and a new position number should be entered

After completing the renumbering, tables are updated automatically.

Example of consolidation operation
There are positions nos.: 1 4 5 6 13 14 15
Renumbering from position <1>: 5
Result of the renumbering: 1 4 5 6 7 8 9
Discontinuities in the numbering starting from the selected position number on, have been removed.
Identical reinforcement

Renumbering from position <1>: 5

By default, a first reinforcement position (the one with the lowest number) is proposed by the program. If a different position number (higher number) is entered, then operation of renumbering (finding identical reinforcement) will be performed for all the positions whose numbers are higher than the specified position number (and for the position with the number entered).

NOTE: The specified position number (in this case 5) must be included in a drawing. If such a position number does not exist, then a warning is displayed and a new position number should be entered.

Once the search is completed, the program displays a report presenting changes made in position numbering. Changes in position numbering are always made in such a manner that a higher number is changed to a lower one.

Example of the operation of finding identical reinforcement
Position: 12,15,65 - changed to [5]
Position: 99 – changed to [13]