TABLE OF CONTENTS

1. GENERAL DESCRIPTION OF THE PROGRAM

1.1. General description of the program

AutoCAD © Structural Detailing - Formwork Drawings has been developed to make it easier to prepare formwork drawings (plans of arrangement of structural positions) of structure components. ***AutoCAD © Structural Detailing - Formwork Drawings*** is the program for defining a structure model, creating drawings (sections, views, projections), creating formwork drawings of single structure elements, preparing tables of material elements and estimating costs of a structure.

AutoCAD © Structural Detailing - Formwork Drawings allows making detailed drawings of a structure (story plans, foundation plans, sections, elevation views, etc.); it is equipped with a complete set of options for drawing and description of structure elements, that are adapted to the needs of an engineer / designer.

The program may be used as:

- a program for exporting elements of a structure model to ***AutoCAD © Structural Detailing - Reinforcement*** and generating required reinforcement
- a tool to use when preparing bids (quick modeling, cost estimation).

AutoCAD © Structural Detailing - Formwork Drawings combined with the program ***AutoCAD © Structural Detailing - Reinforcement*** allows preparing a complete structure project and generating detailed drawings with the possibility to edit prepared drawings.

Basic functions of ***AutoCAD © Structural Detailing - Formwork Drawings*** are:

a) definition of a structure model
b) creating formwork drawings (single structure elements, story plans, elevation views, etc.)
c) export of structure model elements to ***AutoCAD © Structural Detailing - Reinforcement***.

The following objects are singled out in the program:

- Structure model, structure model elements - real structure elements (3D); detailed drawings are created for them
- Position - an object concerned with organization of structure model elements; a (letter - digit) designation of identical elements in a structure
- Document - a set of views of a position in specified projections and views (any number of documents may be generated for each position); a document consists of views. NOTE: A document may be edited only in the edition layout; a document may be printed only after it is inserted to the printout layout.
- View - a single drawing; it is always a document component - if it has been added to a printout, it is a printout element at the same time. NOTE: only a view (drawing) placed in the printout layout may be printed
- Printout - ready-to-use printout composed of views; its equivalent in the AutoCAD © program is a layout together with AutoCAD views placed in it; for each printout there is exactly one layout corresponding to it.

The listed elements are concerned with the stages of work on a project.

1.2. Work stages in the program

AutoCAD © Structural Detailing - Formwork Drawings is a program for creating drawings of buildings (elevation drawings, plans of stories, plans of foundations, sections, etc.). Creation of drawings in the program is an automated process. Drawings are created based on a 3D model of a real building which is defined using the options available in ***AutoCAD © Structural Detailing - Formwork Drawings***. Since ***AutoCAD © Structural Detailing - Formwork Drawings*** works on the AutoCAD © platform, there is also a possibility to use the options available in that program (e.g. edit operations such as copying, moving, etc.).

Creation of a building model and its drawings can be divided into the following stages:

1. ***defining a model / loading a model from other programs***

Structure model includes three-dimensional elements of a real 3D structure; drawings will be prepared for these elements.

Definition of the model consists in determining the locations of structural elements of a building, such as: beams, columns, walls, slabs, foundations, etc.

See – definition of basic structure elements:

Walls
Columns
Beams
Slabs
Spread footings
Continuous footings
Doors
Windows
Lintels
Databases

2. ***assigning positions to structure elements***

Assigning - to elements of a structure model - positions (an object concerned with organization of structure model elements; one or several structure elements are assigned a position for which a common set of drawings may be generated). Positions are available on the two tabs of the ***Object Inspector*** dialog box: *Model* and *Positions*.

See:

Positioning of elements of a structure model
Automatic positioning

3. automatic creation of drawings of a building model

Once positions are assigned to model elements, the program automatically creates drawings of a building based on the configurable templates which define projections, scales, appearance of descriptions, symbols and other parameters concerning graphic representation of an element in a drawing – see:

creating a story plan
creating a foundation plan
creating a 3D view
creating an elevation view
creating a vertical section.

Drawings can be edited using the tools available in ***AutoCAD © Structural Detailing - Formwork Drawings*** or the AutoCAD © standard options. In the case of model modifications, it is possible to automatically update created drawings.

4. export of model elements to other programs / printouts

The following possibilities are offered here:

- saving of formwork drawings of individual structure elements in a DWG file
- export of formwork drawings of structure elements to ***AutoCAD Structural Detailing - Reinforcement***
- automatic generation of reinforcement and drawings of structure element reinforcements in ***AutoCAD Structural Detailing - Reinforcement***.

1.3. Options available in the menu

Below is a list of all options available in ***AutoCAD © Structural Detailing - Formwork Drawings***. The list includes location of options in the text menu, icons that represent particular options, commands that activate options from the command line and short descriptions of options.

Definition

Wall


Opens the ***Wall: definition*** dialog box; the option enables definition of walls in a structure model.
Menu: *Formwork Drawings / Define / Wall* command
Command line: RBCX_DEF_WALL

Column


Opens the ***Wall: definition*** dialog box; the option enables definition of columns in a structure model.
Menu: *Formwork Drawings / Define / Column* command
Command line: RBCX_DEF_COL.

Beam


Opens the ***Beam: definition*** dialog box; the option enables definition of beams in a structure model.
Menu: *Formwork Drawings / Define / Beam* command
Command line: RBCX_DEF_BEAM

Slab


Opens the ***Slab: definition*** dialog box; the option enables definition of floor slabs in a structure model.
Menu: *Formwork Drawings / Define / Slab* command
Command line: RBCX_DEF_SLAB

Stairs


Opens the ***Stairs: definition*** dialog box; the option enables definition of stairs in a structure model.
Menu: *Formwork Drawings / Define / Stairs* command
Command line: RBCX_DEF_STAIRS

Door


Opens the ***Door: definition*** dialog box after indicating a wall; the option is used to define door openings in selected elements of a structure model.
Menu: *Formwork Drawings / Define / Doors* command
Command line: RBCX_DEF_DOOR.

Window


Opens the ***Window: definition*** dialog box after indicating a wall; the option is used to define window openings in selected elements of a structure model.
Menu: *Formwork Drawings / Define / Window* command
Command line: RBCX_DEF_WINDOW.

Lintel


Opens the ***Lintel: definition*** dialog box after selecting a window, door, recess or opening in the wall; the option is used to define lintels above the selected elements of a structure model.
Menu: *Formwork Drawings / Define / Lintel* command
Command line: RBCX_DEF_LINTEL

Opening/recess in wall


Opens the ***Opening: definition*** dialog box after indicating a structure element (wall); the option is used to define an opening/recess in the selected walls of a structure.
Menu: *Formwork Drawings / Define / Opening/recess in wall* command
Command line: RBCX_DEF_HOLE_WALL.

Opening/recess in slab


Opens the ***Opening: definition*** dialog box after indicating a structure element (slab); the option is used to define an opening/recess in the selected slabs of a structure model.
Menu: *Formwork Drawings / Define / Opening/recess in slab* command
Command line: RBCX_DEF_HOLE_SLAB

Spread footing


Opens the ***Spread footing: definition*** dialog box; the option enables definition of spread footings in a structure model.
Menu: *Formwork Drawings / Define / Spread footing* command
Command line: RBCX_DEF_FOOT.

Continuous footing

Opens the ***Continuous footing: definition*** dialog box; the option enables definition of continuous footings in a structure model.
Menu: *Formwork Drawings / Define / Continuous footing* command
Command line: RBCX_DEF_CONT_FOOT.

Ground beam

Opens the ***Ground beam: definition*** dialog box; the option enables definition of ground beams in a structure model.
Menu: *Formwork Drawings / Define / Ground beam* command
Command line: RBCX_DEF_GROUND_BEAM

Raft foundation

Opens the ***Raft foundation: definition*** dialog box; the option enables definition of raft foundations in a structure model.
Menu: *Formwork Drawings / Define / Raft foundation* command
Command line: RBCX_DEF_SLAB_FOUNDATION

Prefabricated element

Opens the ***Prefabricated element: definition*** dialog box; the option enables definition of prefabricated elements in a structure model. Prefabricated elements are volumetric elements of complex geometry (spread footings, stairs, beams, columns, etc.).
Menu: *Formwork Drawings / Define / Prefabricated element* command
Command line: RBCX_DEF_PREF

Workframes

Insert rectangular workframe

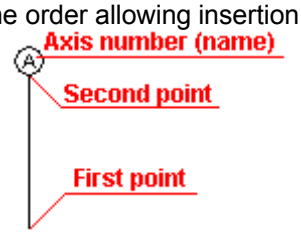
Opens the ***Workframe*** dialog box where parameters of a rectangular workframe can be defined.
Menu: *Formwork Drawings / Workframes / Insert rectangular workframe* command
Command line: RBCX_DEF_WORKFRAME

Insert circular workframe

Opens the ***Workframe*** dialog box where parameters of a circular workframe can be defined.
Menu: *Formwork Drawings / Workframes / Insert circular workframe* command
Command line: RBCX_DEF_WORKFRAME_RADIAL

Insert structural axis

The option is used to insert an axis in a selected place in a drawing. Symbols are drawn according to a default style set in the ***Job Preferences*** dialog box. To insert a structural axis symbol in a drawing, follow the steps below:
1 select the *Formwork Drawings / Insert structural axis* command
2 enter a number (name) of a structural axis
3 indicate the first point of an axis symbol
4 indicate the second point of the axis symbol (see the drawing below)
5 if need be, determine the order allowing insertion of axes in groups.



A number of the axis is proposed according to the settings in the default style; while inserting the axis the user may enter any number (each following one will be inserted according to the recently-specified numbering). There is a possibility to modify the axis number using the relevant option from the context menu.
Menu: *Formwork Drawings / Insert structural axis* command
Command line: RBCX_DEF_SYMBOL_AXIS

Planes

Definition of plane


Opens the ***Definition of planes*** dialog box. The option enables defining intermediate (horizontal) planes or inclined planes on which elements of a structure model can be defined.
Menu: *Formwork Drawings / Planes / Definition of plane* command
Command line: RBCX_PLANE_DEF

Attach to plane


The option allows attaching selected structure elements to an indicated inclined plane. Geometry of structure elements is adjusted to the slope of the inclined plane.
Menu: *Formwork Drawings / Planes / Attach to plane* command
Command line: RBCX_ATTACH_TO_PLANE

Detach from plane


The option allows 'detaching' selected structure elements from an indicated inclined plane (the option has the opposite action to the *Attach to plane* option). The operation results in 'returning' elements to default levels of the reference plane.
Menu: *Formwork Drawings / Planes / Detach from plane* command
Command line: RBCX_DETACH_FROM_PLANE

Group planes


The option allows creating a group of planes; such a group is regarded as a separate object which enables defining elements of a structure model with a complicated plane geometry (e.g. planes of a multi-pitch roof).
To obtain a group of planes, the user should indicate planes that should belong to the group of planes, and next, specify a name of the group of planes. Once created, the group of planes is regarded in the same way as a standard single plane.
Menu: *Formwork Drawings / Planes / Group planes* command
Command line: RBCX_GROUP_PLANES

Ungroup planes


The option allows 'splitting' a group of planes into single separate planes (the option has the opposite action to the *Group planes* option).
Menu: *Formwork Drawings / Planes / Ungroup planes* command
Command line: RBCX_UNGROUP_PLANES

Import/Export

Import structure from CBS Pro

The option allows import of a structure model created in the ***CBS Pro* program.**
Menu: *Formwork Drawings / Import\Export / Import structure from CBS Pro* command
Command line: RBCX_LOAD_ROBIN_MODEL

Export structure to CBS Pro

The option allows opening the ***CBS Pro*** program and saving a structure model in the format of this program.
Menu: *Formwork Drawings / Import\Export / Export structure to CBS Pro* command
Command line: RBCX_SAVE_ROBIN_MODEL

Save model as ACIS solids

The option allows saving a structure model as an ACIS solid (as a drawing in a DWG format).
Menu: *Formwork Drawings / Import\Export / Save model as ACIS solids* command
Command line: RBCX_ACIS_EXPORT

Display

Show whole building


After selecting this option, a whole structure model will be presented on the screen.
Menu: *Formwork Drawings / Display / Show whole building* command
Command line: RBCX_VIEW_BUILDING

Show active story

After selecting this option, only an active (selected) story of a building will be presented on the screen.
Menu: *Formwork Drawings / Display / Show active story* command
Command line: RBCX_VIEW_STOREY

Show only selected elements

After selecting this option, only selected elements of a building will be presented on the screen.
Menu: *Formwork Drawings / Display / Show only selected elements* command
Command line: RBCX_VIEW_SELECT_EL

Show elements

Selecting this option opens the ***Filters – element selection*** dialog box; the option is used to define criteria of selecting elements in a structure model.
Menu: *Formwork Drawings / Display / Show elements* command
Command line: RBCX_VIEW_ELEMENTS

Show planes

Selecting this option opens the ***Manager of plane properties*** dialog box; the option is used for viewing and editing parameters of defined intermediate or inclined planes.
Menu: *Formwork Drawings / Display / Show planes* command
Command line: RBCX_EDIT_PLANES

Create plan of story

After selecting this option the program automatically creates a plan of a selected story in a building.
Menu: *Formwork Drawings / Create plan of story* command
Command line: RBCX_DEF_STOREY_VIEW

Create plan of foundation

After selecting this option the program automatically creates a plan of building foundations.
Menu: *Formwork Drawings / Create plan of foundation* command
Command line: RBCX_DEF_FOUNDATION_VIEW

Create vertical section

After selecting this option the program automatically creates a vertical section of a structure model; to create a vertical section, indicate lines intersecting a structure model and a section 'depth'.
Menu: *Formwork Drawings / Create vertical section* command
Command line: RBCX_DEF_V_SECTION

Create elevation view

After selecting this option the program automatically creates an elevation view of a structure model; to create an elevation view, indicate lines of the elevation view and a direction of the view.
Menu: *Formwork Drawings /Create elevation view* command
Command line: RBCX_DEF_ELEVATION

Created 3D view

After selecting this option the program automatically creates a three-dimensional view of a structure model;
Menu: *Formwork Drawings /Create 3D view* command
Command line: RBCX_DEF_VIEW3D

Element description

The option allows description of individual objects in a structure; descriptions of structure elements are possible in plans (of stories or foundations) and in sections.
Menu: *Formwork Drawings / Element description* command
Command line: RBCX_LABEL

Dimension lines

Group dimension line

The *Group dimension line* option is used to create dimension lines for structure elements presented in created plans or sections of a designed object. The user should indicate elements to be dimensioned and define an

offset of a dimension line from dimensioned elements. The program will automatically recognize selected elements and appropriately group dimension lines.
Menu: *Formwork Drawings / Dimension lines / Group dimension line* command
Command line: RBCX_DIM_GROUP

Simple dimension line

The *Simple dimension line* option is used to create dimension lines for individual structure elements located in generated plans or sections of a designed object; the user should indicate an element(s) to be dimensioned (intersect the element with an auxiliary line) and define an offset of a dimension line from the dimensioned element.
Menu: *Formwork Drawings / Dimension lines / Simple dimension line* command
Command line: RBCX_DIM_SIMPLE

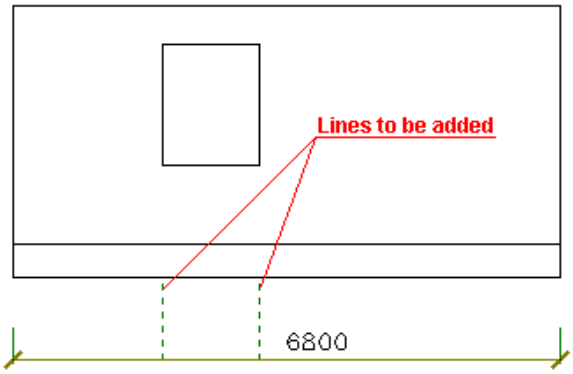
Arc dimension line

The *Arc dimension line* option is used to create dimension lines for arc-shaped structure elements (walls, beams, continuous footings) located in generated plans or sections of a designed object; the user should indicate one arc element to be dimensioned and define an offset of a dimension line from the dimensioned element.
Menu: *Formwork Drawings / Dimension lines / Arc dimension line* command
Command line: RBCX_DIM_ARC

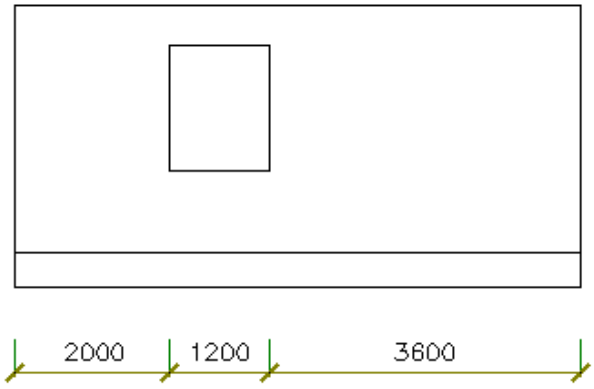
Add division point

The option allows modifying dimension lines created by the program by defining additional points (auxiliary lines) to which a given dimension will refer. Dimensions presented on the dimension line will be recalculated and adjusted to the location of a new point on the dimension line (see the drawing below).

Before adding a line



After adding a line

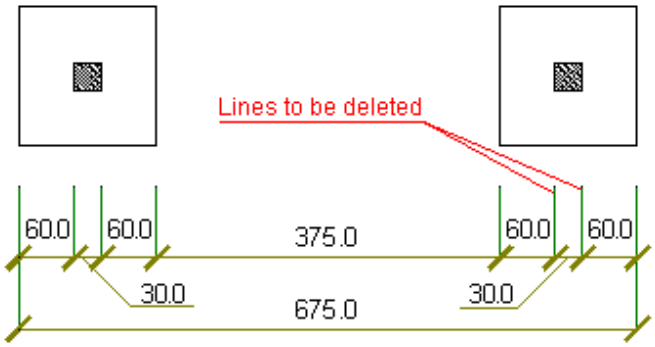


Menu: *Formwork Drawings / Dimension lines / Add division point* command
Command line: RBCX_DIM_UTIL_ADD

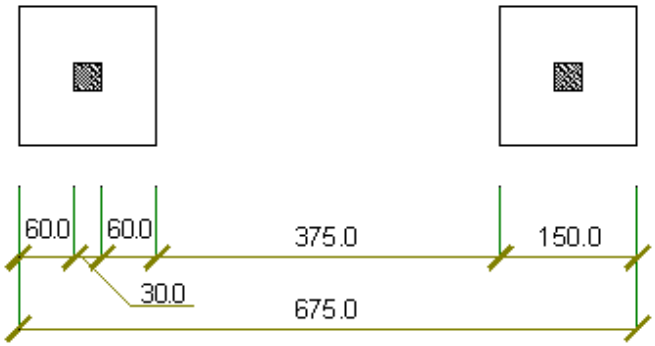
Delete division point

The option allows modifying dimension lines created by the program by deleting indicated points (auxiliary lines) on a dimension line. Once a point (auxiliary line) is deleted from a dimension line, dimensions on the dimension line will be recalculated (see the drawing below).

Before deleting a line



After deleting a line



Menu: *Formwork Drawings / Dimension lines / Delete division point* command
Command line: RBCX_DIM_UTIL_DEL

Insert elevation mark

The option is used to insert an elevation mark in a selected place in a drawing. Symbols are drawn according to a default style set in the ***Styles of symbols*** dialog box. To insert an elevation mark in a drawing, follow the steps below:
1 select the *Formwork Drawings / Insert elevation mark* command
2 indicate a point on a selected level (the program will automatically calculate the height with respect to the base point).
Levels are designated with respect to the base level defined in the ***Building parameters*** dialog box.

Menu: *Formwork Drawings / Insert elevation mark* command
Command line: RBCT_DEF_SYMBOL_COTE

Insert opening symbol in view

The option is used to insert an opening symbol at a selected place in a drawing. Symbols are drawn according to a default style set in the ***Styles of symbols*** dialog box. To insert an opening symbol in a view, follow the steps below:
1 select the *Formwork Drawings / Insert opening symbol in view* command
2 select a point inside the opening in the view.
Menu: *Formwork Drawings / Insert opening symbol in view* command
Command line: RBCT_DEF_SYMBOL_HOLE

Insert opening symbol in cross-section

The option is used to insert an opening symbol in the cross-section. Symbols are drawn according to a default style set in the ***Styles of symbols*** dialog box.
Menu: *Formwork Drawings / Insert opening symbol in cross-section* command
Command line: RBCT_DEF_SYMBOL_HOLECAST

Select labels and dimensions

Opens the ***Filters – element selection*** dialog box; the option is used to define criteria of selection of labels and dimensions of structure objects. Labels (descriptions) of structure elements are available in plans (of stories or foundations) and in sections.
Menu: *Formwork Drawings / Select labels and dimensions* command
Command line: RBCX_LABEL_SELECT

Change size of labels and dimensions

The option allows specifying a scale factor for labels and dimensions of structure objects. After indicating a label or dimension of a structure element, the user should define a factor value by which the size of a label or dimension will be multiplied; the option is available in the edition layout.

Menu: *Formwork Drawings / Change size of labels and dimensions* command
Command line: RBCX_MOD_LABEL_SCALE

Tables

Summary – elements

After running this option, the element summary table is inserted in a drawing. This table may be inserted for:
- whole building (all model elements)
- selected model elements
- filtered model elements.

NOTE: The table may be added only to a completed printout (generated drawing); it cannot be inserted to a structure model or to the edit layout.
Menu: *Formwork Drawings / Tables / Summary – elements* command
Command line: RBCX_LIST_ELEM

Summary – costs

After running this option, the summary table of element costs is inserted in a drawing. This table may be inserted for:
- whole building (all model elements)
- selected model elements
- filtered model elements.

NOTE: The table may be added only to a completed printout (generated drawing); it cannot be inserted to a structure model or to the edit layout.
Menu: *Formwork Drawings / Tables / Summary – costs* command
Command line: RBCX_LIST_PRICE

Detailed – elements

After running this option, the detailed element table (this table includes both summary tables) is inserted in a drawing. This table may be inserted for:
- whole building (all model elements)
- selected model elements
- filtered model elements.

NOTE: The table may be added only to a completed printout (generated drawing); it cannot be inserted to a structure model or to the edit layout.
Menu: *Formwork Drawings / Tables / Detailed – elements* command
Command line: RBCX_LIST_DETA

Detailed – openings

After running this option, a detailed table concerned with openings is inserted in a drawing. This table may be inserted for:
- whole building (all model elements)
- selected model elements
- filtered model elements.

NOTE: The table may be added only to a completed printout (generated

drawing); it cannot be inserted to a structure model or to the edition layout.

Menu: *Formwork Drawings / Tables / Detailed – openings* command
Command line: RBCX_LIST_HOLE

Update table


After the option is switched on, the program updates an indicated table after changes made in geometry / parameters of a structure model.
Menu: *Formwork Drawings / Tables / Update table* command
Command line: RBCX_LIST_ACT

Table Printout / Export / Edit


The option allows printing a table or exporting a table to an *.xls or *.csv format file.
Menu: *Formwork Drawings / Tables / Table Printout / Export / Edit* command
Command line: RBCX_LIST_EXP

Styles

Element description styles

Opens the *Description Styles* dialog box; the option allows defining styles (format) of description for individual elements of a structure model.
Menu: *Formwork Drawings / Styles / Element description styles* command
Command line: RBCX_STYLE_LABEL

Styles – tables

Opens the ***Summary tables – style manager*** dialog box; the option allows defining/modifying summary tables for structure model elements.
Menu: *Formwork Drawings / Styles / Styles – tables* command
Command line: RBCX_STYLE_LIST

Styles – graphic symbols

Opens the ***Styles of symbols*** dialog box; the option allows defining styles (format) of symbols presented in structure drawings (elevation mark, structural axis symbol).
Menu: *Formwork Drawings / Styles / Styles – graphic symbols* command
Command line: RBCT_DEF_SYMBOL_STYLE.

Styles – element graphic display

Opens the ***Structure model – graphic representation of elements*** dialog box; the option allows defining the manner of representing structure model elements on the screen (line thickness and type, colors, etc.).
Menu: *Formwork Drawings / Styles / Styles – element graphic display* command
Command line: RBCX_STYLE_GRAPHIC_PRESENTATION

Styles – drawing templates

Opens the ***Drawing template manager*** dialog box; the option allows determining a method of creating formwork drawings of elements of a building structure.
Menu: *Formwork Drawings / Styles / Styles – drawing templates* command
Command line: RBCX_DRAWING_LAYOUT_MNGR

Databases

Section database

Opens the ***Section list*** dialog box; the options provided in this dialog box allow the user to:
= define a new section shape
= open external section databases (e.g. ***ROBOT*** databases)
= save defined sections in a database (*.mdb file).
Menu: *Formwork Drawings / Databases / Section database* command
Command line: RBCX_TOOL_BASE_SECT

Material database

Opens the ***Material list*** dialog box; the options provided in this dialog box allow the user to:
= define a new material type
= open external material databases (e.g. ***ROBOT*** databases)
= save defined materials in a database (*.mdb file).

Menu: *Formwork Drawings / Databases / Material database* command
Command line: RBCX_TOOL_BASE_MAT

Spread footing database Opens the ***Spread footing list*** dialog box (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a spread footing
= open files with definitions of spread footings
= save defined solid shapes in a database.
Menu: *Formwork Drawings / Databases / Spread footing database* command
Command line: RBCX_TOOL_BASE_SOLID

Stairs database Opens the dialog box ***List of stairs*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of stairs
= open files with definitions of stairs
= save defined stair shapes in a database.
Menu: *Formwork Drawings / Databases / Stairs database* command
Command line: RBCX_TOOL_BASE_SOLID

Prefabricated element database

Beam database Opens the dialog box ***Prefabricated beam list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a prefabricated beam
= open files with definitions of prefabricated beams
= save defined shapes of prefabricated beams in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Beam database* command
Command line: RBCX_TOOL_BASE_PREF_BEAM

Column database Opens the dialog box ***Prefabricated column list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a prefabricated column
= open files with definitions of prefabricated columns
= save defined shapes of prefabricated columns in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Column database* command
Command line: RBCX_TOOL_BASE_PREF_COL

Slab database Opens the dialog box ***Prefabricated slab list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a prefabricated slab
= open files with definitions of prefabricated slabs
= save defined shapes of prefabricated slabs in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Slab database* command
Command line: RBCX_TOOL_BASE_PREF_SLAB

Wall database Opens the dialog box ***Prefabricated wall list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a prefabricated wall
= open files with definitions of prefabricated walls
= save defined shapes of prefabricated walls in a database.

Menu: *Formwork Drawings / Databases / Prefabricated element database / Wall database* command
Command line: RBCX_TOOL_BASE_PREF_WALL

Ground beam database Opens the dialog box ***Prefabricated ground beam list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a (prefabricated) ground beam
= open files with definitions of (prefabricated) ground beams
= save defined shapes of (prefabricated) ground beams in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Ground beam database* command
Command line: RBCX_TOOL_BASE_PREF_GROUND_BEAM

Continuous footing database Opens the dialog box ***Prefabricated continuous footing list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a (prefabricated) continuous footing
= open files with definitions of (prefabricated) continuous footings
= save defined shapes of (prefabricated) continuous footing in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Continuous footing database* command
Command line: RBCX_TOOL_BASE_PREF_CONT_FOOT

Raft foundation database Opens the dialog box ***Prefabricated raft foundation list*** (prefabricated elements, i.e. solids of complex geometry); the options provided in this dialog box allow the user to:
= define a new shape of a (prefabricated) raft foundation
= open files with definitions of (prefabricated) raft foundations
= save defined shapes of (prefabricated) raft foundations in a database.
Menu: *Formwork Drawings / Databases / Prefabricated element database / Raft foundation database* command
Command line: RBCX_TOOL_BASE_PREF_SLAB_FOUNDATION

Window database Opens the ***Window list*** dialog box; the options provided in this dialog box allow the user to:
= define a new window shape
= open external window databases
= save defined window shapes in a database (*.mdb file).
Menu: *Formwork Drawings / Databases / Window database* command
Command line: RBCX_TOOL_BASE_WINDOW

Door database Opens the ***Door list*** dialog box; the options provided in this dialog box allow the user to:
= define a new door shape
= open external door databases
= save defined door shapes in a database (*.mdb file).
Menu: *Formwork Drawings / Databases / Door database* command
Command line: RBCX_TOOL_BASE_DOOR

Remaining openings Opens the ***Opening list*** dialog box; the options provided in this dialog box allow the user to:
= define a new opening shape
= open external opening databases
= save defined opening shapes in a database (*.mdb file).
Menu: *Formwork Drawings / Databases / Remaining openings* command
Command line: RBCX_TOOL_BASE_OPENINGS

Tools

Modify


The option allows modifying parameters of a selected structure element (parameters of a wall, beam, column, etc.). Once the structure element is selected, the dialog box appears on the screen where parameters of the structure element definition (material, geometric parameters, priority, etc.) may be modified.
Menu: *Formwork Drawings / Tools / Modify* command
Command line: RBCX_MOD

Modify graphical parameters


The option allows modification of graphic parameters of a selected structure element. Once the structure element is selected, the dialog box appears on the screen where parameters of the structure element presentation (type of the element contour line, filling, element hatching, etc.) may be modified.
Menu: *Formwork Drawings / Tools / Modify graphical parameters* command
Command line: RBCX_MOD_PROP

Element selection by type


Open the ***Filters – element selection*** dialog box; the option is used to define criteria of selection of elements in a structure model.
Menu: *Formwork Drawings / Tools / Element selection by type* command
Command line: RBCX_TOOL_SELECT_EL

Element selection by section


Opens the ***Filters – element selection*** dialog box; the option is used to define criteria of selection of elements in a structure model.
Menu: *Formwork Drawings / Tools / Element selection by section* command
Command line: RBCX_TOOL_SELECT_EX

Element info


The option allows displaying information concerning an indicated element. Once an element is selected, the dialog box with basic information about the selected element (element type, position, material, length, priority, etc.) appears on the screen.
Menu: *Formwork Drawings / Tools / Element info* command
Command line: RBCX_TOOL_INFO

Change beam to ground beam


The option allows swapping:
- a selected beam for a ground beam
- a selected ground beam for a beam.

After swapping them, all parameters determined for a beam/ground beam are kept (section, position with respect to a story), only the object type changes.
NOTE: If a beam / ground beam has been positioned, then after a swap, positioning of such an element should be repeated.
Menu: *Formwork Drawings / Tools / Change beam to ground beam* command
Command line: RBCX_TOOL_SWAP_BEAM

Copy / Move elements


The options allows copying or moving selected structure elements to a new place indicated by the user. The ***Copy / Move selected elements*** dialog box opens on the screen.
Menu: *Formwork Drawings / Tools / Copy /Move elements* command
Command line: RBCX_TOOL_COPYOBJECTS

Define opening in a beam


The option allows defining an opening in a beam, a ground beam or a continuous footing. To define an opening, the user should:
- indicate a beam, a ground beam or a continuous footing
- in the ***Opening in a beam - Definition*** dialog box – select a type and geometry of an opening in the section of a beam/continuous footing

- indicate the location of an opening along the length of a beam/continuous footing
Menu: *Formwork Drawings / Tools / Define opening in a beam* command
Command line: RBCX_TOOL_HOLE_BEAM

Cut beam/wall to polyline


The option allows cutting a beam, a ground beam, a continuous footing or a wall to the plane determined by a polyline. To do it, the user should:
- indicate a beam, a ground beam, a continuous footing or a wall
- indicate a polyline defining the cutting plane
- indicate a part of a beam/continuous footing/wall to be cut off.
Menu: *Formwork Drawings / Tools / Cut beam/wall to polyline* command
Command line: RBCX_TOOL_CUT_POLYLINE

Refresh structure model


The option allows generating a correct structure model, and in particular, removing possible model incorrectness associated with connections of a structure model element (e.g. intersections of walls with beams / slabs).
NOTE: After performing this operation all documents (story plans, sections of a building, elevation views, 3D views) are out of date. These documents require to be generated again, since changes may have been made in elements of a structure model.
Menu: *Formwork Drawings / Tools / Refresh structure model* command
Command line: RBCX_MODELREFRESH

Job Preferences


Opens the ***Job Preferences*** dialog box; the option enables adopting basic parameters used in ***AutoCAD © Structural Detailing*** (codes, units, materials, etc.)
Menu: *Formwork Drawings / Job Preferences* command
Command line: RBCX_JOB_PREF.

Preferences


Opens the ***Options*** dialog box; the option enables setting parameters of the work environment of ***AutoCAD © Structural Detailing***.
Menu: *Formwork Drawings / Job Preferences* command

Object Inspector – Show / Hide

The option is used to switch on / off presentation of the ***Object Inspector*** dialog box which is in a standard way located in the left-hand part of the screen.
Menu: *Formwork Drawings / Object Inspector – Show / Hide* command
Command line: RBCTOI

2. CONFIGURATION

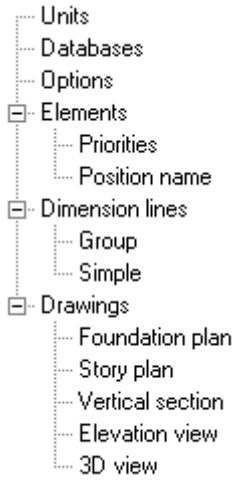
2.1. Job Preferences

The option allows determining basic parameters used in ***AutoCAD © Structural Detailing***. The option is accessible:

- from the menu by selecting the option *Formwork Drawings / Job Preferences*
- by pressing the *Job Preferences* icon
- from the command line: RBCX_JOB_PREF.

The ***Job Preferences*** dialog box can be divided in the two main parts:

- in the left-hand part of the dialog box is a selection tree (see the drawing below) from which the user selects, with the mouse cursor, one of the options of the program job preferences



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

The right part of the dialog box contains the standard buttons (**OK**, **Cancel**, **Help**) and the following ones:
Default - pressing this button saves the values of the job preferences parameters as default ones
Save - pressing this button saves the current status of preference parameters under a specified name
Delete - pressing this button deletes a set of job preferences saved under the currently-selected name.

Once the *General settings* option is chosen in the selection tree located in the left part of the *Structural Detailing* tab (the ***Options*** dialog box of the AutoCAD© program is called up by the menu command of the AutoCAD© program: *Tools / Options* or the command *Formwork Drawings / Preferences* or by pressing the *Preferences* icon), there is a possibility to choose a language version of ***AutoCAD © Structural Detailing***. The following program language versions are available

 USA

 Great Britain

 France

 Poland

 Italy

 Spain

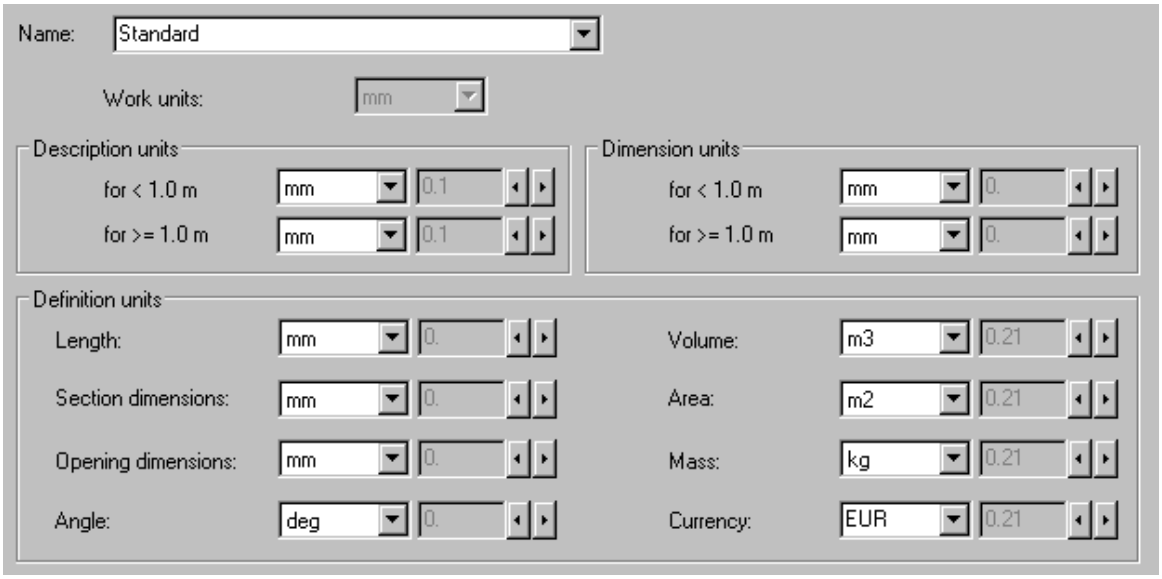
 Romania

 Russia.

After selecting the *General settings* option in the selection tree provided in the left part of the *Formwork Drawings* tab (the ***Options*** dialog box of the AutoCAD© program is called up by the *Tools / Options* command in the AutoCAD© menu or the *Formwork Drawings / Preferences* command or by pressing the *Preferences* icon), it is possible to select a drawing template that will be opened while starting up ***AutoCAD © Structural Detailing - Reinforcement*** (when exporting formwork drawings to ***AutoCAD © Structural Detailing - Reinforcement***).
Moreover, take note that in the ***Options*** dialog box of the AutoCAD © program on the *Structural Detailing* tab the user may select the ***AutoCAD © Structural Detailing*** work template. Templates are located in the CFG folder - they contain settings for a given country, for example the RBCX-001.dwt file is the template for the USA.

2.2. 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 a preference option set.
Below, the user may determine ***AutoCAD © Structural Detailing*** work units. NOTE: Work units may be changed in a project **BEFORE** defining the first arbitrary structure element (wall, foundations, columns, etc.). When the project already includes at least one structure element, the selection list is no longer accessible and work units may not be changed.
The dialog box also holds the options: *Description units* and *Dimension units*. The aim of the option is to determine how to present length / description units less than 1m and greater or equal to 1m; this remark is concerned with values placed in drawings (dimension lines, descriptions, etc.). For example, if centimeters are selected in the *for < 1.0 m* field, then the dimension 0.33 m will be displayed as 33 cm. For these units it is possible to change the manner of presenting the number format. The fields located on the right side enable defining a number of decimal places for each of the quantities.

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

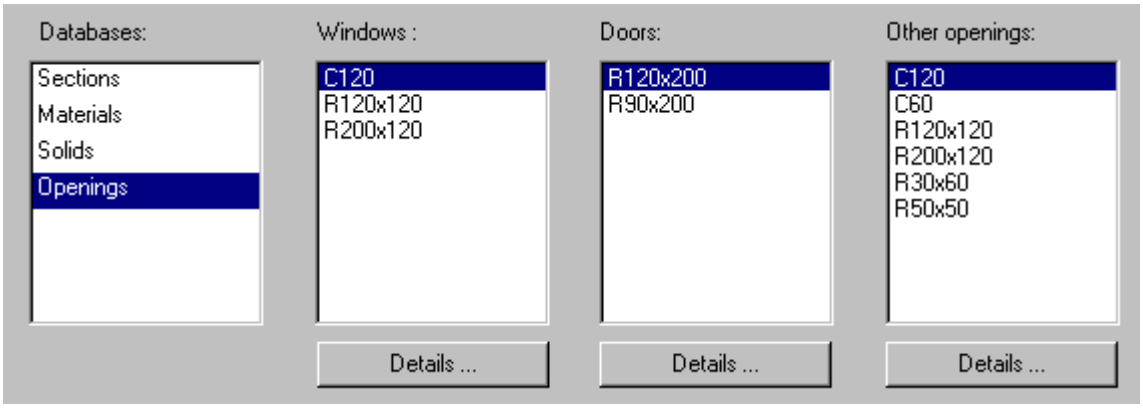
- length unit, including the following components: formwork dimensions, etc.

- dimension unit for a cross-section of structure elements
- dimension unit for openings
- angle unit
- volume unit
- area unit
- mass unit
- currency.

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 number format for the quantities listed. These fields enable defining a number of decimal places for each of the quantities. To change the number of decimal places, click - with the left mouse button - on the arrows ◄ ► (the number of decimal places is increased or decreased, respectively). The unit precision will be reflected in descriptions of elements.

2.3. Databases

After selecting the *Databases* 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 drawing presents a selected opening database).



The databases provided in ***AutoCAD © Structural Detailing - Formwork Drawings*** are used to:
- define new structure elements (cross-sections, materials, doors, windows, openings)
- open existing, external databases (e.g. ***ROBOT*** databases)
- save defined elements in databases (*.mdb files).

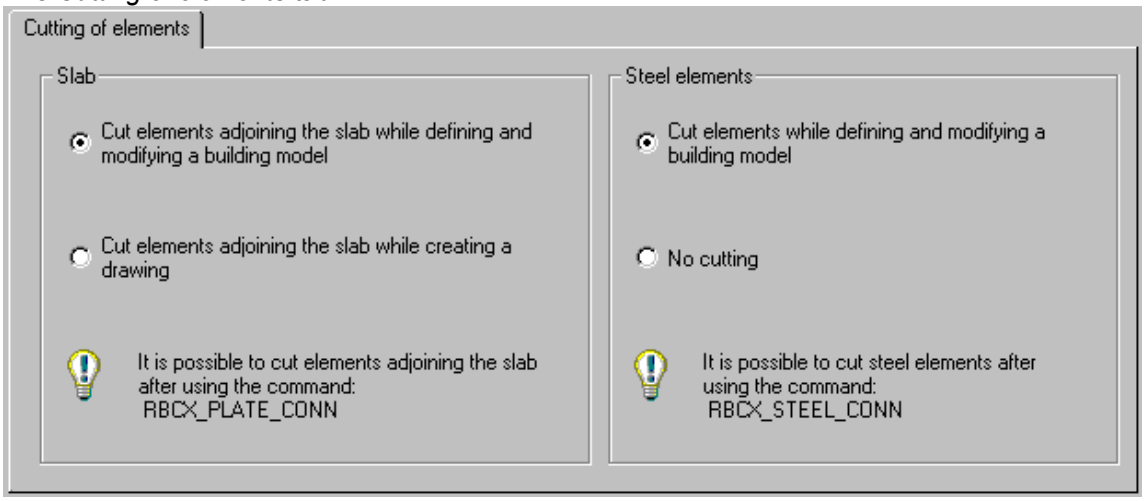
Pressing the **Details** button opens one of the dialog boxes listed below:
- Section list
- Material list
- Opening list (windows, doors or other openings).

When material databases are selected, the **Prices** button is also available; pressing this button opens the ***Prices of formworks and materials*** dialog box.

2.4. Options

After selecting the *Options* 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 tab shown below.

The *Cutting of elements* tab



The above dialog box holds options that enable selecting the method of cutting (cutting to fit) elements of a structure model. The dialog box is split in the following two parts:

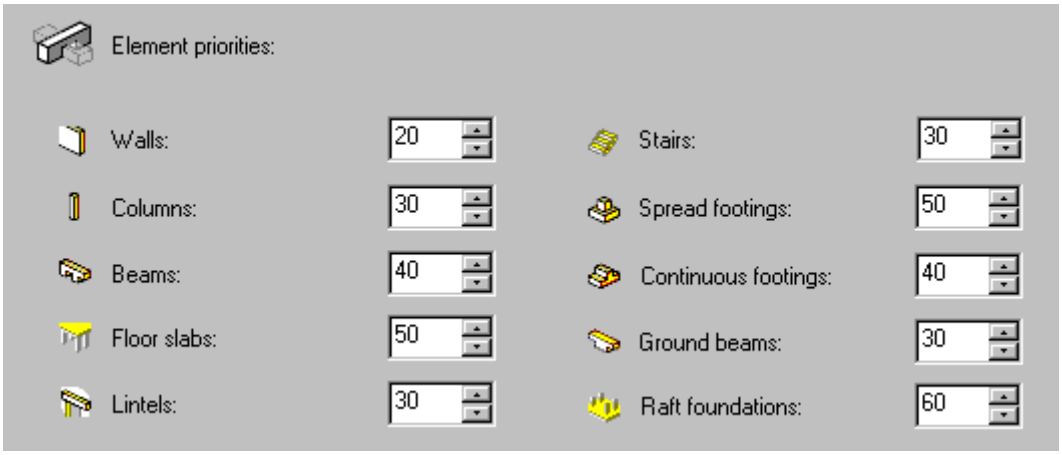
1. part that refers to cutting of RC slabs included in a structure model
   selected option: *Cut elements adjoining the slab while defining and modifying a building model*
   Elements adjoining the slab will be cut on an ongoing basis in the structure model while defining or modifying a model

   selected option: *Cut elements adjoining the slab while creating a drawing*
   Elements adjoining the slab will **not** be cut in the structure model while defining or modifying a model (elements may interpenetrate with the defined slab); cutting of elements will be performed while creating drawings of the structure model (e.g. foundation plans, vertical sections through a building, etc.)

2. part that refers to cutting of steel elements included in the structure model
   selected option: *Cut elements while defining and modifying a building model*
   Steel elements will be cut on an ongoing basis in the structure model while defining or modifying a model
   selected option: *No cutting*
   Steel elements will **not** be cut in the structure model (they will keep their initial length) and in drawings.

2.5. Priorities

After selecting the *Priorities* 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.

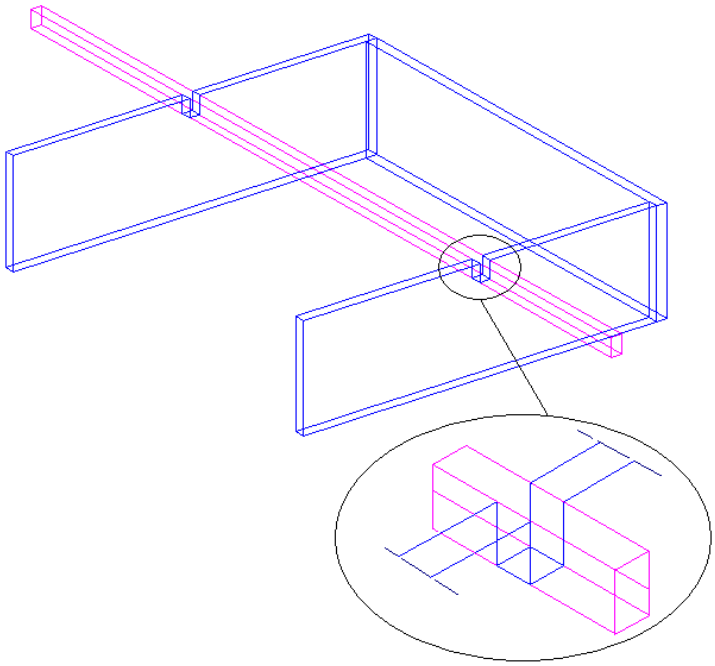


The above dialog box allows defining values of priorities (importance of elements) assigned by default to individual components of a structure. These values may be changed freely while defining structure elements.

A priority is a value which enables managing interpenetration (cutting to fit) of structure elements that overlap with each other. The priority determines importance of an element in relation to other elements in a model.

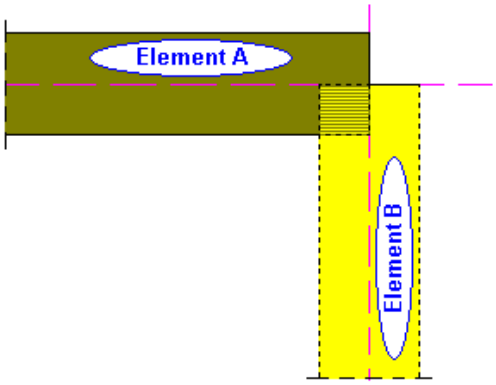
Correct definition of priorities of structure elements allows correct definition of element interpenetration while creating plans and sections; it also has effect on calculation of volume and area of formworks of materials used in a structure.

The basic rule adopted in the program says that elements with a lower value of a priority are cut to fit elements with a higher priority; it means that a higher-priority element does not change, whereas a lower-priority element fits to the element of a higher priority value. As an example, the drawing below illustrates interpenetration of a wall with the priority of 30 and a beam with the priority of 50. The priority of the beam is higher, thus the wall will be cut to the defined beam.

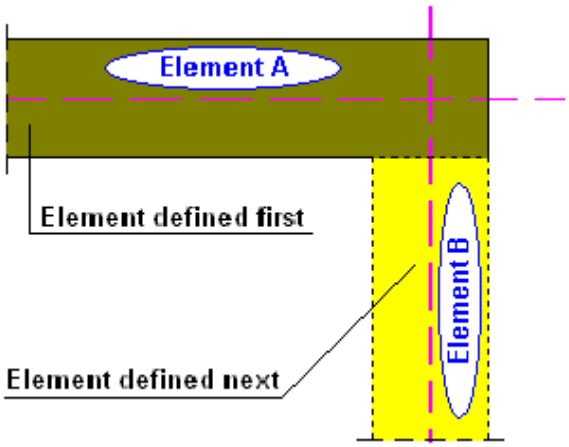


The program adopts default values of priorities for individual structure elements; a hierarchy of structure element importance is defined - from the most important element to the least important one.

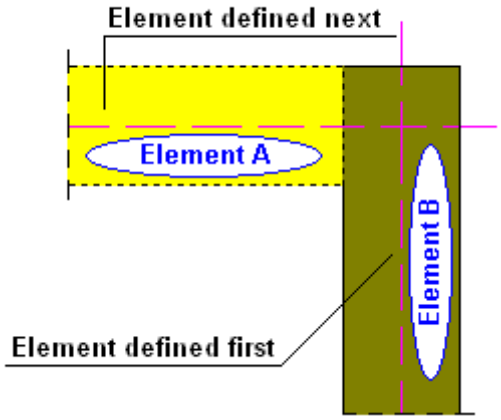
NOTE:
In the case of elements with the same value of priorities, a more important element is the one defined first. This rule is illustrated in the drawings below (for two walls).



Element A defined first

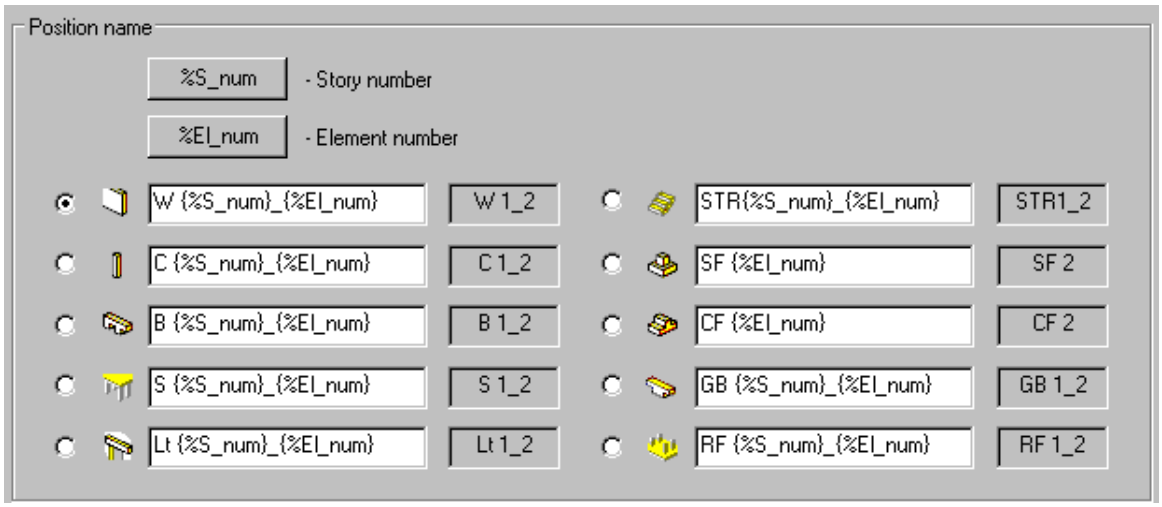


Element B defined first



2.6. Position name

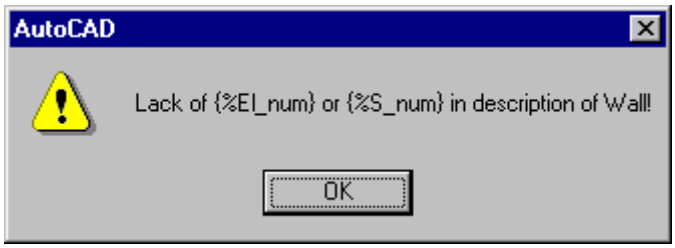
After selecting the *Position name* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



The above dialog box enables defining a syntax of names assigned to positions created in course of automatic positioning of structure elements.

NOTE:
For all structure elements (except spread footing, continuous footing, lintel, ground beam and raft foundation) the syntax of a position name must include a story number and an element number; it means that the variable {%El_num} must be a component of the syntax of each element name.

NOTE:
When closing the dialog box the program verifies if the syntax definition is correct, so that it is not possible to assign the same position to different elements.
Names of all structure elements, except lintels and foundations (spread footings, continuous footings and raft foundations) must include both, the variable of a story number and the variable of an element number. This requirement has been introduced because of reasons concerned with the process of element positioning. If the syntax of a name of an arbitrary structure element (except lintels or foundation elements) does not include both components, an appropriate message appears on the screen (see the drawing below).

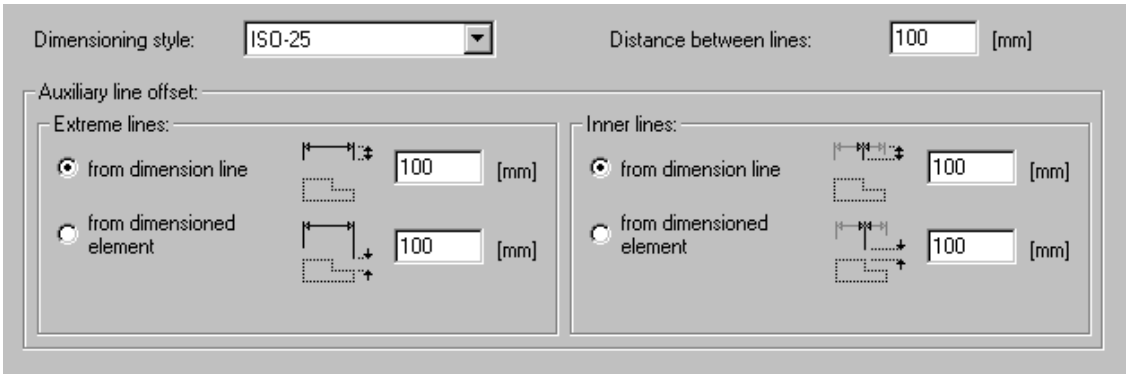


The syntax of a position name may be determined for the following structure elements:
- walls e.g. Wall {%S_num}_{%El_num}; for example, the final description of a wall may take the following form:
  Wall 2_3,
  where:
  2 – denotes the number of a building story
  3 – denotes the successive number of a wall on the building story no. 2
- column: e.g. Column {%S_num}_{%El_num}
- beam / ground beam e.g. Beam {%S_num}_{%El_num}
- lintel: e.g. Lintel {%S_num}_{%El_num}
- floor slab or raft foundation: e.g. Slab {%S_num}_{%El_num}
- stairs: e.g. Stairs {%S_num}_{%El_num}
- spread footing: e.g. Spread footing {%El_num}
- continuous footing e.g. Continuous footing {%El_num}.

After the syntax for an arbitrary structure element is defined, the field on the right side presents a preview of an element name resulting from the defined syntax.

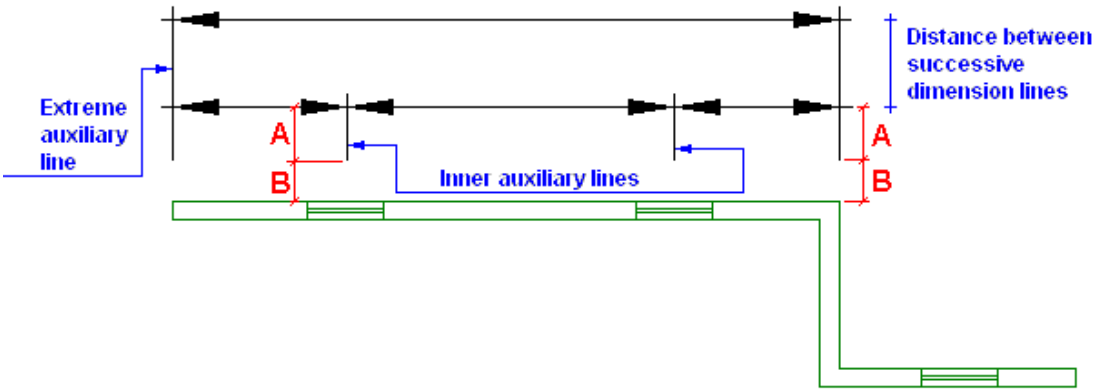
2.7. Dimension lines

After selecting the *Dimension lines* 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 following parameters of dimension lines can be defined in the above dialog box:

- dimensioning style – selected from the list of styles defined with the use of the mechanisms of the AutoCAD © program (NOTE: a dimensioning style must always be selected)
- distance between successive dimension lines (see the drawing below)
- offset of auxiliary lines from a dimension line (the value A in the drawing below) or from a dimensioned element (the value B in the drawing below).



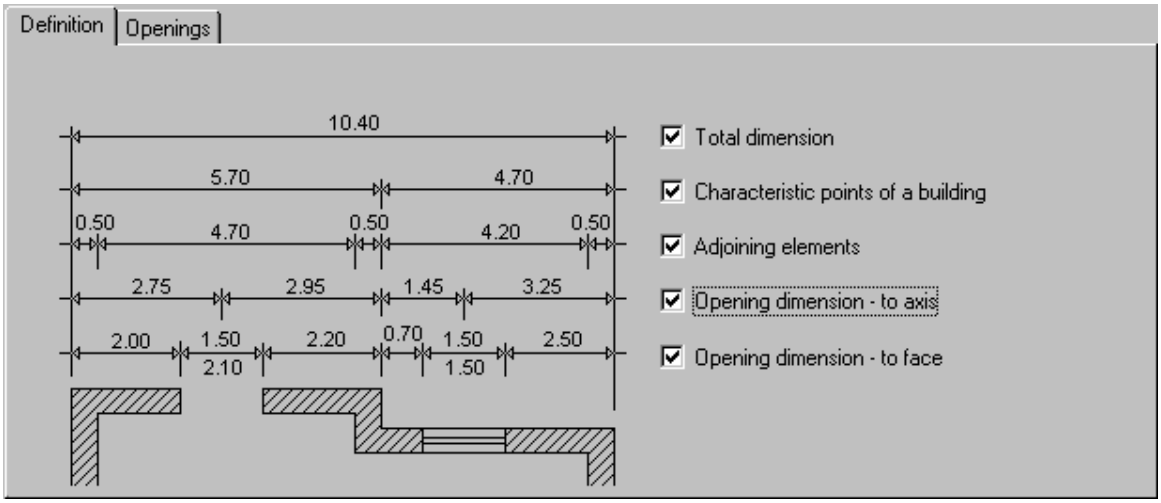
2.8. Group dimension line

After selecting the *Group* 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.

A group dimension line enables creating dimension lines for several structure elements included in a structure plan or view. Modifications like deleting certain elements (e.g. openings) or changing their overall dimensions are instantly reflected in the relevant dimension chains; it means that after modifying a structure dimensions do not need to be defined again.

NOTE:
Created dimension lines may be modified using two options provided in the menu of ***AutoCAD © Structural Detailing - Formwork Drawings****: Add division point to dimension line and Delete division point from dimension line (see description of options in the menu of AutoCAD Structural Detailing - Formwork Drawings).*

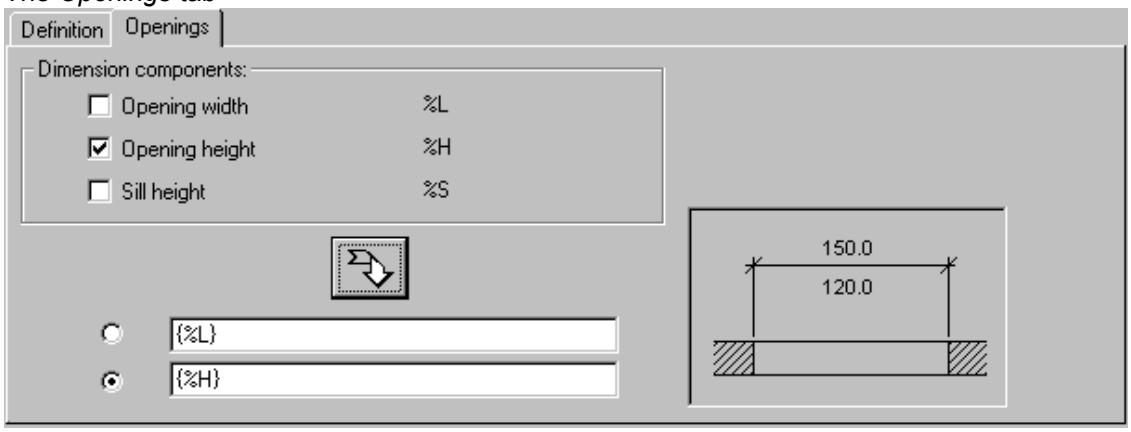
The *Definition* tab



This tab holds options that allow creating dimension lines of objects of a building; the following group dimension lines can be created:

- *Total dimension* - if this option is switched on, then while creating dimension lines of structure objects (group dimension lines), the total dimension of a structure element will be added
- *Characteristic points of a building* - if this option is switched on, then while creating dimension lines of structure objects (group dimension lines), dimensions between characteristic points of objects (e.g. indentation in a wall, beginning or end of a wall, etc.) will be added
- *Adjoining elements* - if this option is switched on, then while creating dimension lines of an object the program also automatically dimensions elements adjoining to the main element (dimensioning refers to selected adjoining elements); this option is mainly concerned with walls - dimensioning concerns the following adjoining elements:
  - columns adjoining to the wall
  - walls adjoining to the wall
  - beams adjoining to the wall
  - columns included in the wall
- *Opening dimension - to axis* - if this option is switched on, then while creating dimension lines of structure objects (group dimension lines), dimensions between characteristic points of an object and opening axes will be added; a description of a dimension line depends on the parameters adopted on the *Openings* tab
- *Opening dimension - to face* - if this option is switched on, then while creating dimension lines of structure objects (group dimension lines) dimensions between characteristic points of an object and faces (edges) of openings will be added; a description of a dimension line depends on the parameters adopted on the *Openings* tab.

The *Openings* tab



This tab holds options that enable the user to determine a method of describing a dimension line for openings (opening dimension - to face*).* The description of an opening dimension line may consists of:

- opening width
- opening height
- sill height (only for window openings).

The description of an opening dimension line may be placed in one line (over the dimension line) or in two lines (over and under the dimension line). To determine the method of describing a dimension line, follow the steps below:

- switch on an appropriate option or several options
- select a location of the description (the upper edit field refers to a description positioned over the dimension line, whereas the lower one – to a description under the dimension line)
- press the arrow  .

2.9. Simple dimension line

After selecting the *Simple* 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.

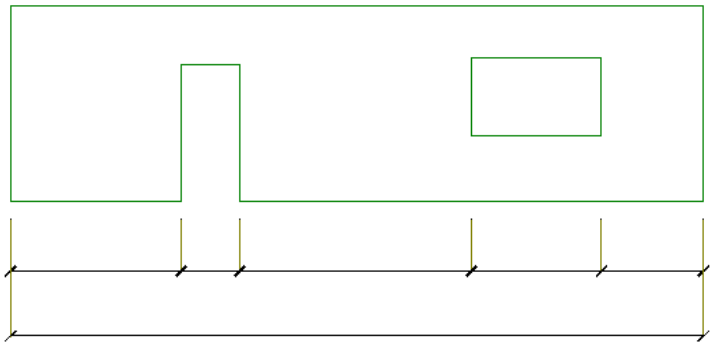
The options in this dialog box enable defining parameters for dimension lines of simple elements presented in drawings; simple elements are basic elements of a structure model (beams, columns, windows, doors, etc.) and drawing elements defined using the AutoCAD© program (lines, arcs, circles, etc.).

If any option is switched on on the *AutoCAD Structural Detailing Elements* or *ACAD Elements* tab (e.g. *Wall* and *Beam*), then, while creating dimension lines, intersecting these elements (i.e. walls and beams in a structure model) with an auxiliary line will result in creating dimension lines for them.

When any option on the *AutoCAD Structural Detailing Elements* or *ACAD Elements* tab is switched off, then such an element will not be considered while creating dimension lines.

The upper part of the dialog box holds the following option:

Add total dimension – if this option is switched on, then a dimension line representing the total dimension of dimensioned elements will be added while creating dimension lines (see the drawing below)

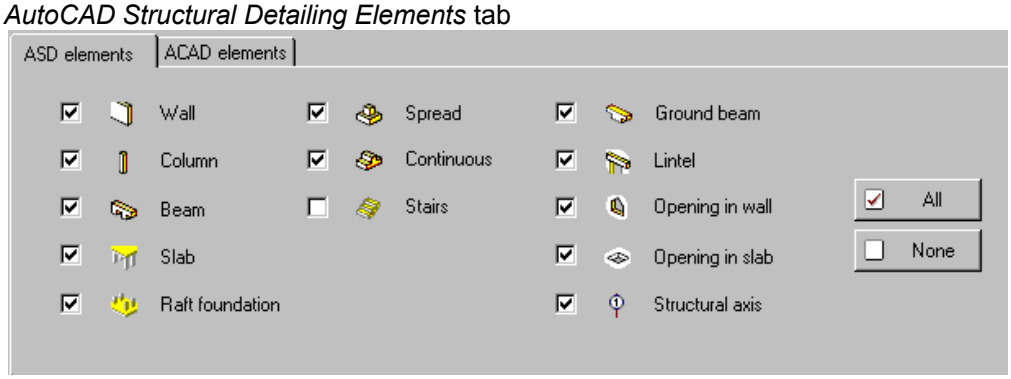


In the upper part of the dialog box there is also the *Show dimensions to axis* option - if this option is switched on, then dimensions on the dimension line refer to axes of dimensioned elements.

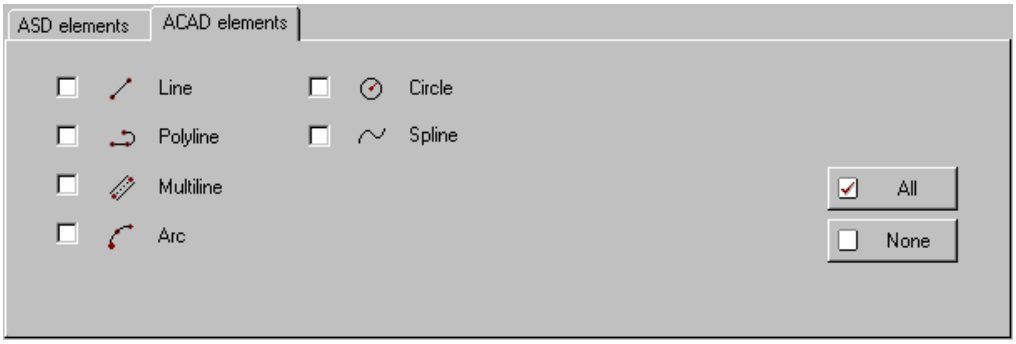
NOTE:
Created dimension lines may be modified using two options provided in the menu of ***AutoCAD Structural Detailing - Formwork Drawings****: Add division point to dimension line and Delete division point from dimension line (see the description of options in the menu of AutoCAD Structural Detailing - Formwork Drawings).*

The two tabs: *AutoCAD Structural Detailing Elements* and *ACAD Elements* are shown in the following drawings.

The *AutoCAD Structural Detailing Elements* tab



The *ACAD Elements* tab



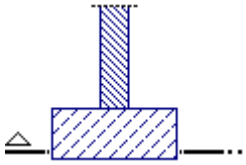
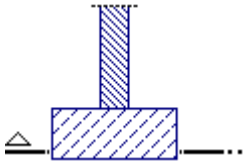
Pressing the **All** button switches on all the options provided on the *AutoCAD Structural Detailing Elements* or *ACAD Elements* tab. Pressing the **None** button switches off all the options provided on the *AutoCAD Structural Detailing Elements* or *ACAD Elements* tab.

2.10. Foundation plan

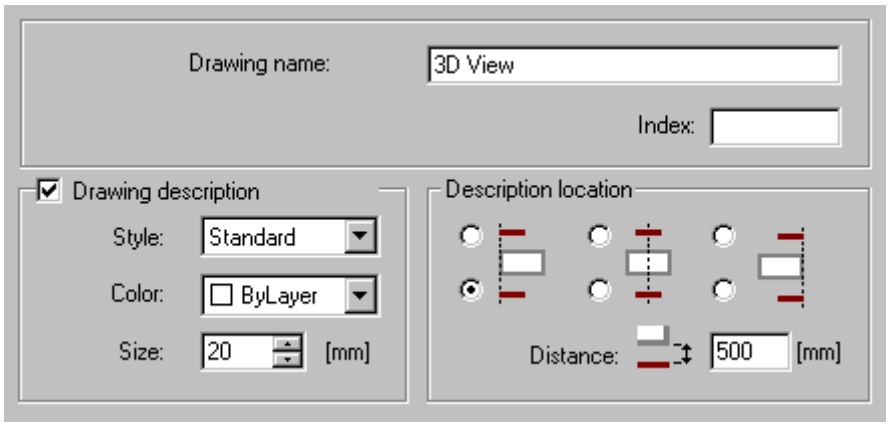
After selecting the *Foundation plan* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes several tabs discussed below.

Definition tab

The options provided on this tab allow defining the location of a foundation section which will be used to create a drawing of the foundation plan:

-  on the foundation reference level ('upward' orientation)
-  above the upper plane of foundations ('downward' orientation) with the possibility to define a distance from the top plane of foundations.

General parameters tab

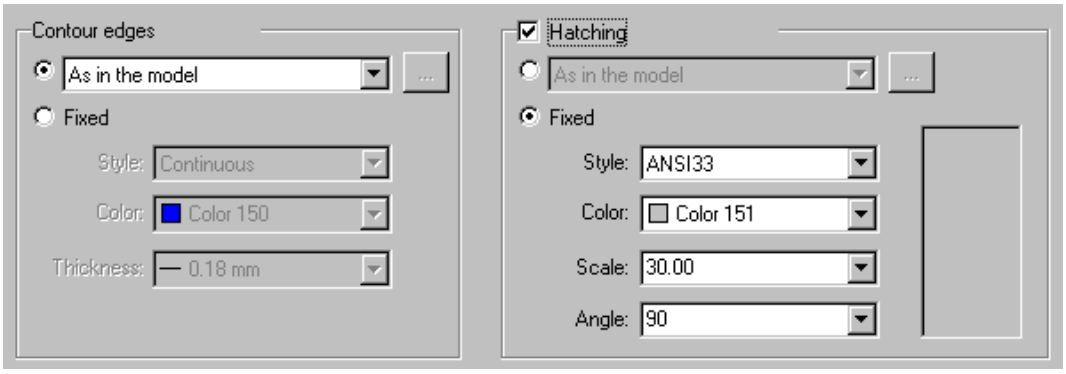


In the upper part of the dialog box a drawing name and index should be specified (e.g. drawing designation A, B, C, etc.).
Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

- *Drawing description* – defining a description of a foundation plan; the following parameters can be set here: color, style and size of a description.
- *Description location* – defining the location of a description in a foundation plan drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

Visible elements tab

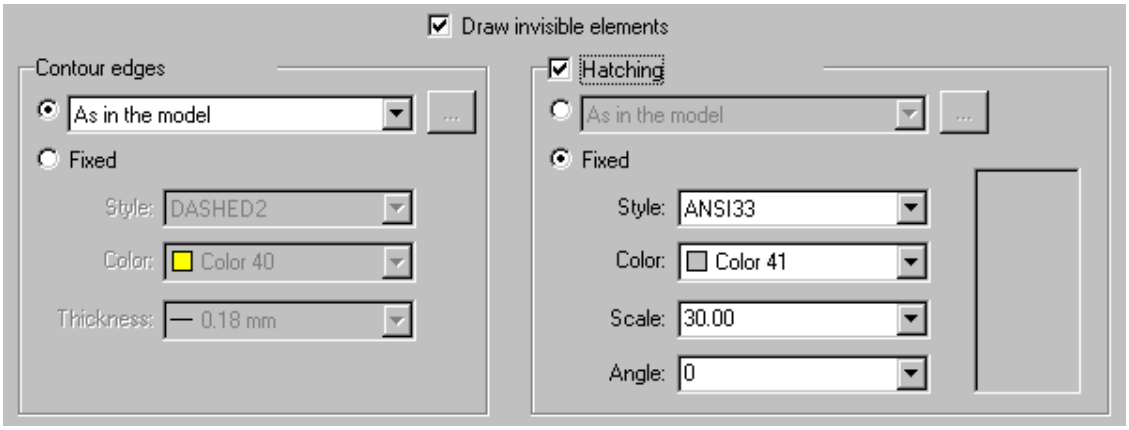


The options provided on the above tab concern parameters of presenting visible structure elements in the foundation plan; for edges of the foundation contour as well as for hatching the following two options are available:

- as in the model / by element type / by material (the last type is available for hatching)
- fixed - determined by the user:
- it is possible to define parameters of edges of the foundation contour (line style, color and thickness) and optionally, parameters of hatching (style, color, scale and angle of hatching).

Cut elements tab
The options provided on the above tab concern parameters of presenting cut structure elements in the foundation plan; this tab includes exactly the same options as the previous tab (*Visible elements*).

Invisible elements tab



The options provided on the above tab concern parameters of presenting invisible structure elements in the foundation plan; this tab includes exactly the same options as the previous tabs (*Visible elements / Cut elements*). Invisible elements are presented in the foundation plan if the *Draw invisible elements* option is switched on.

2.11.Story plan

After selecting the *Story plan* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes several tabs discussed below.

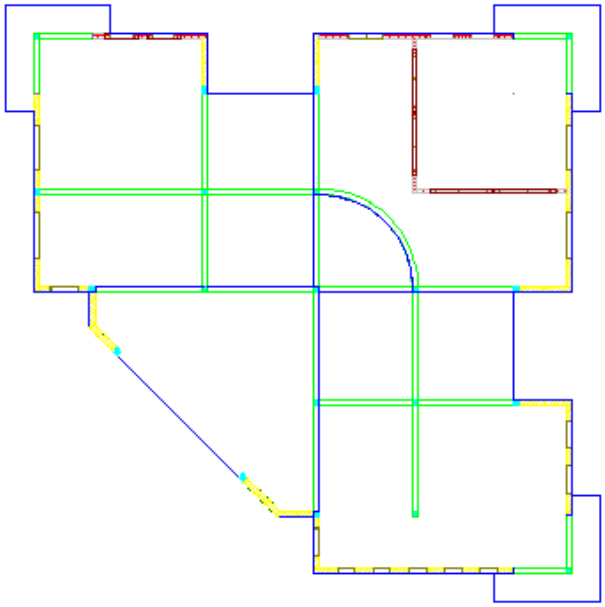
Definition tab
The options provided on this tab allow defining a story plan type:

- story plan according to the model representation - top view
- story plan as a regular horizontal section.

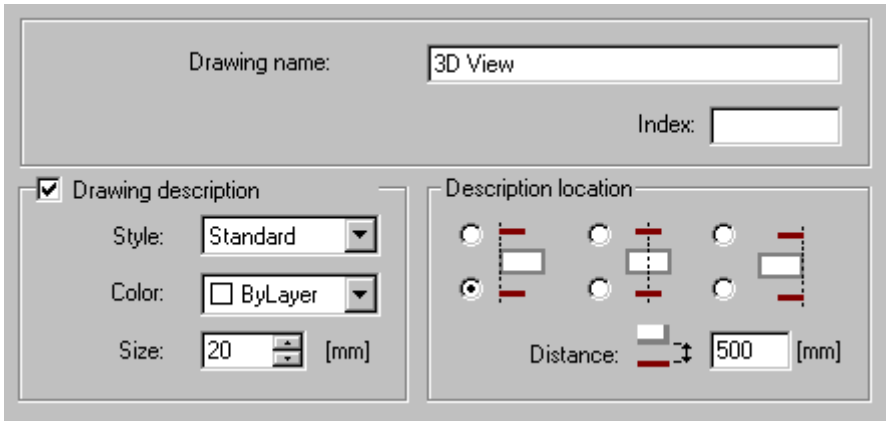
Depending on a selected option, the contents of the available tabs changes.

STORY PLAN - MODEL REPRESENTATION – TOP VIEW
For this plan type the created drawing of the story plan will include all 'visible' elements of a structure model (along with colors, hatching styles, etc. assigned to elements) - see the drawing below.
Only the following two tabs are accessible for this plan type: *Definition* and *General parameters*.

Definition tab
The option group *Draw symbolic elements of a higher story* is available on this tab. If this option is switched on, then the story plan drawing will present structure elements situated on a higher story – the following parameters can be chosen to present contour of these elements: style, color and thickness.



General parameters tab



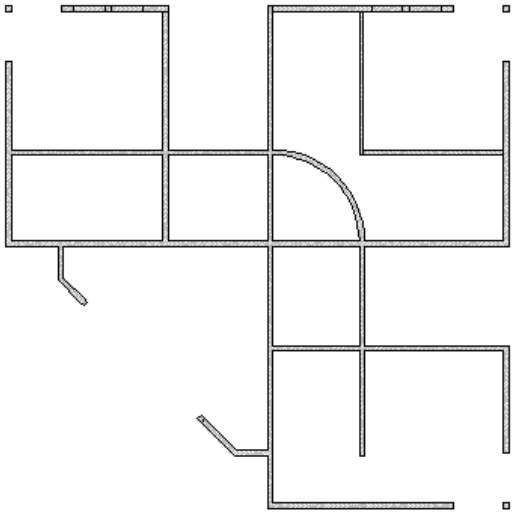
In the upper part of the dialog box a drawing name and index (number) should be specified. Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

- *Drawing description* – defining a description of a story plan; the following parameters can be defined here: color, style and size of a description.
- *Description location* – defining a description location in a story plan drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

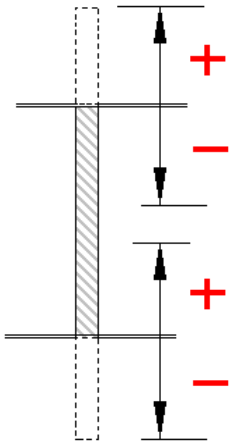
STORY PLAN – REGULAR HORIZONTAL SECTION

The following four tabs are available: *Definition*, *General parameters*, *Contour edges* and *Section edges*.
An example of such a story plan is illustrated in the drawing below.

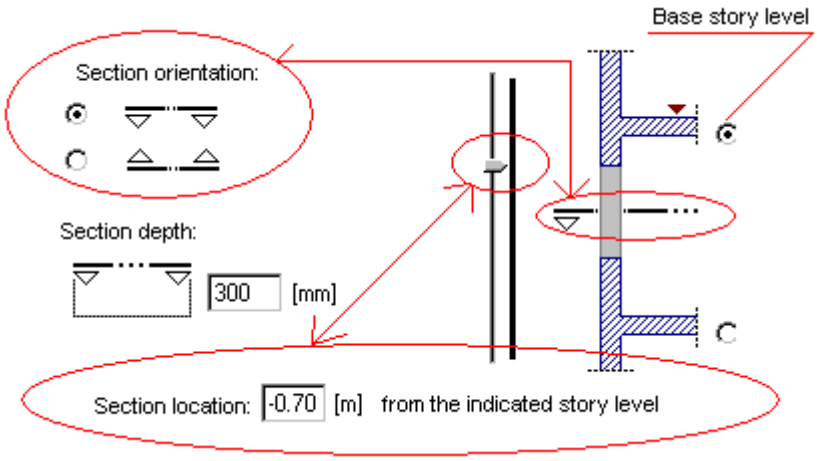


Definition tab
This option holds the options used to define the following parameters of the horizontal plan of a structure model.

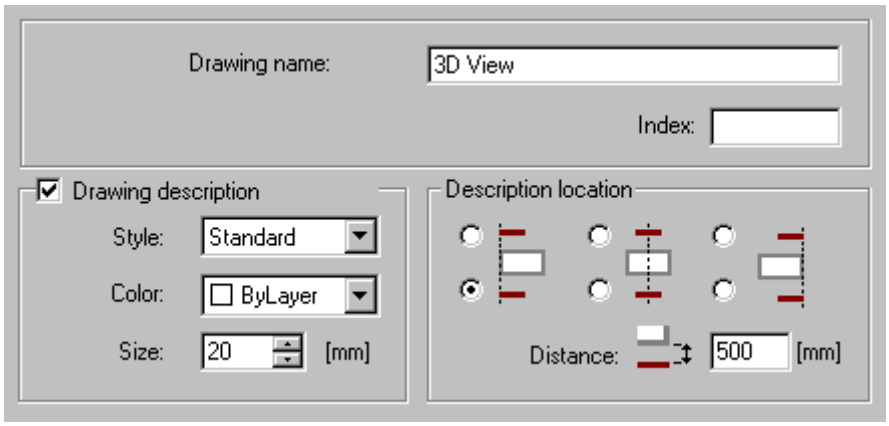
- selection of the base story level
- orientation of the horizontal section ('up' or 'down')
- section depth
- location of a section with respect to a selected story level (it can be defined by means of the slider in the right-hand part of the tab or by entering a number to the *Section location* edit field; the sign convention for values of the section location is presented in the drawing below.



Example parameters of the horizontal section are presented in the drawing below:
- base story level: upper
- section orientation: 'down'
- section depth: 30 cm
- section location: 70 cm from the upper story level.



General parameters tab



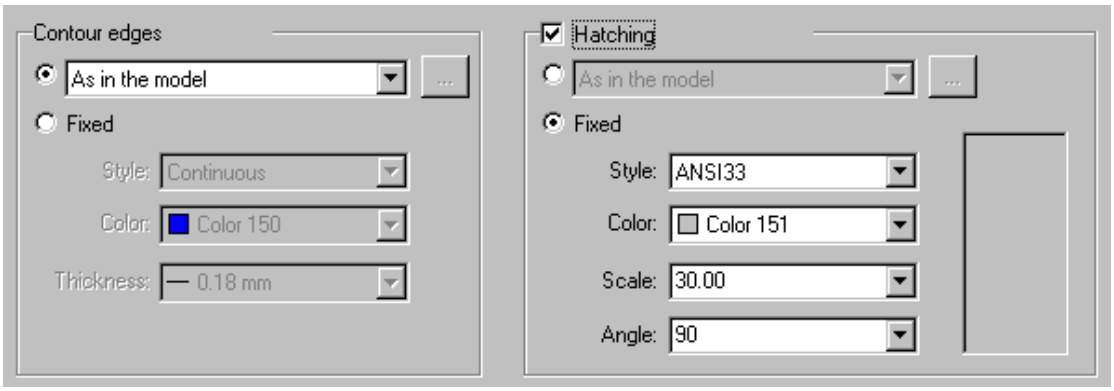
In the upper part of the dialog box a drawing name and index (number) should be specified. Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

- *Drawing description* – defining a description of a story plan; the following parameters can be defined here: style, color and size of a description.
- *Description location* – defining a description location in a story plan drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

For a classic foundation plan, the *General parameters* tab contains an additional option *Present openings/recesses independently of the cutting level – projection*. Once this option is switched on, openings/recesses are included in the story plan irrespective of the selected cutting level; all openings/recesses are projected onto one common level. This option is particularly useful, if in one plan it is necessary to show all openings (usually positioned on different levels) in walls, e.g. ventilation system openings, air conditioning / heating system openings, etc.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

Visible elements tab



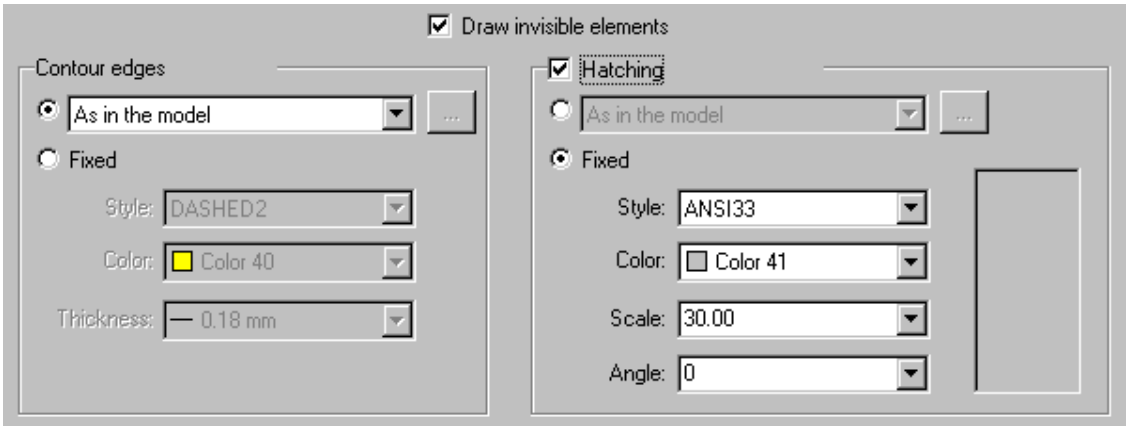
The options provided on the above tab concern parameters of presenting visible structure elements in the story plan; for edges of the contour of structure model elements in the plan as well as for hatching the following two options are available:

- as in the model / by element type / by material (the last type is available for hatching)
- fixed - determined by the user:
  it is possible to define parameters of edges of the element contour in the plan (line style, color and thickness) and optionally, parameters of hatching (style, color, scale and angle of hatching).

Cut elements tab

The options provided on the above tab concern parameters of presenting cut structure elements in the story plan; this tab includes exactly the same options as the previous tab (*Visible elements*).

Invisible elements tab

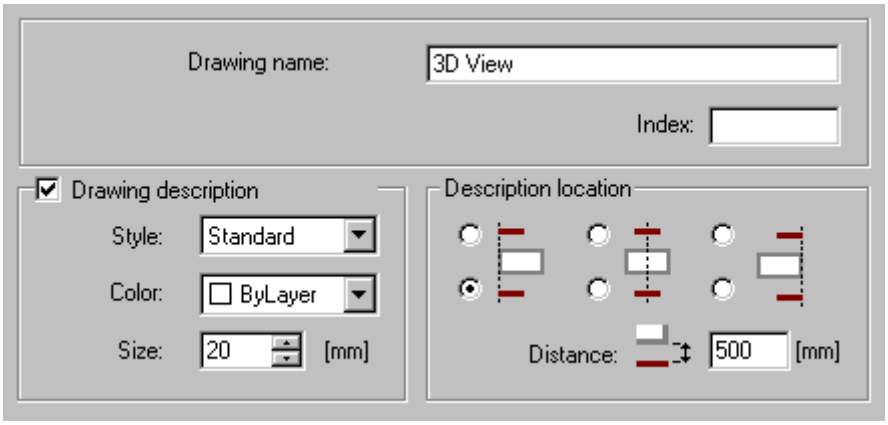


The options provided on the above tab concern parameters of presenting invisible structure elements in the story plan; this tab includes exactly the same options as the previous tabs (*Visible elements / Cut elements*). Invisible elements are presented in the story plan if the *Draw invisible elements* option is switched on.

2.12. Vertical section

After selecting the *Vertical section* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes several tabs discussed below.

General parameters tab

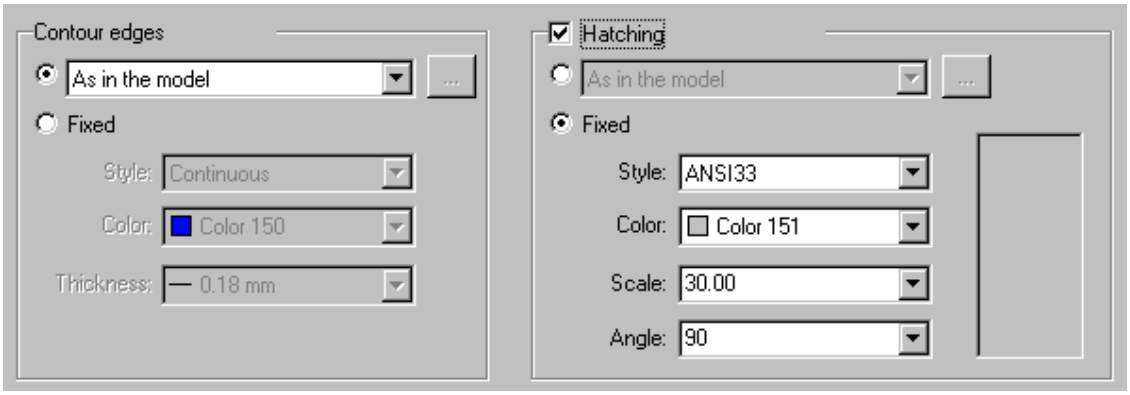


In the upper part of the dialog box a drawing name and index (number) should be specified. Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

- *Drawing description* – defining a description of a vertical section; the following parameters can be defined here: color, style and size of a description.
- *Description location* – defining the location of a description in a vertical section drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

Visible elements tab

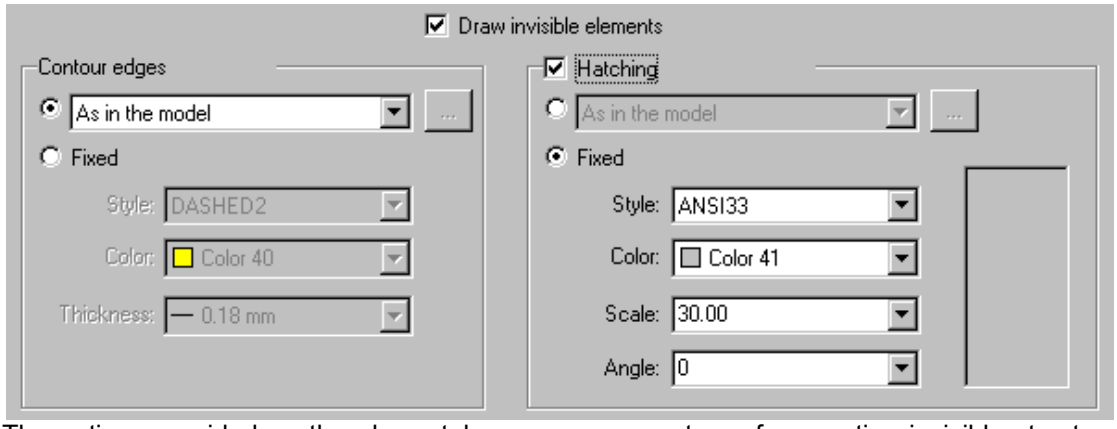


The options provided on the above tab concern parameters of presenting visible structure elements in the vertical section; for edges of the contour of structure model elements as well as for hatching the following two options are available:

- as in the model / by element type / by material (the last type is available for hatching)
- fixed - determined by the user:
  it is possible to define parameters of edges of the contour of structure model elements (line style, color and thickness) and optionally, parameters of hatching (style, color, scale and angle of hatching).

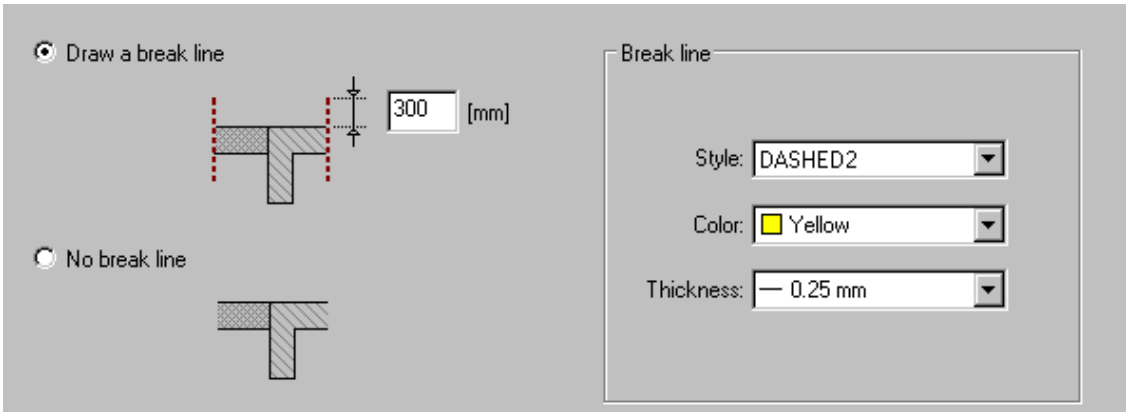
Cut elements tab
The options provided on the above tab concern parameters of presenting cut structure elements in the vertical section; this tab includes exactly the same options as the previous tab (*Visible elements*).

Invisible elements tab



The options provided on the above tab concern parameters of presenting invisible structure elements in the vertical section; this tab includes exactly the same options as the previous tabs (*Visible elements / Cut elements*). Invisible elements are presented in the vertical section if the *Draw invisible elements* option is switched on.

Breaks tab



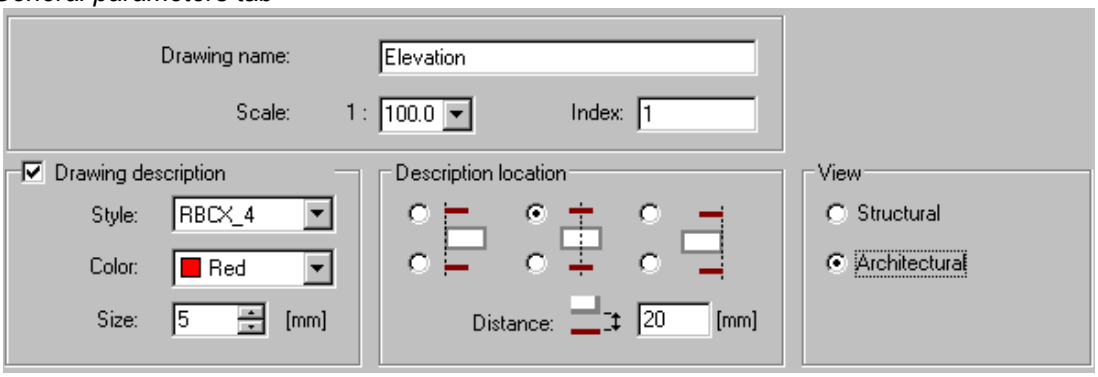
The options provided on the above tab concern parameters of presenting breaks of structure model elements in the vertical section:

- *No break line* - break lines will not be presented (no other options on this tab are available then)
- *Draw a break line* - break lines will be presented; the following options are available on this tab:
  line style, color and thickness of a break line as well as length of the line outside the structure model element.

2.13. Elevation view

After selecting the *Elevation view* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes several tabs discussed below.

General parameters tab



In the upper part of the dialog box a drawing name and index (number) should be specified. Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

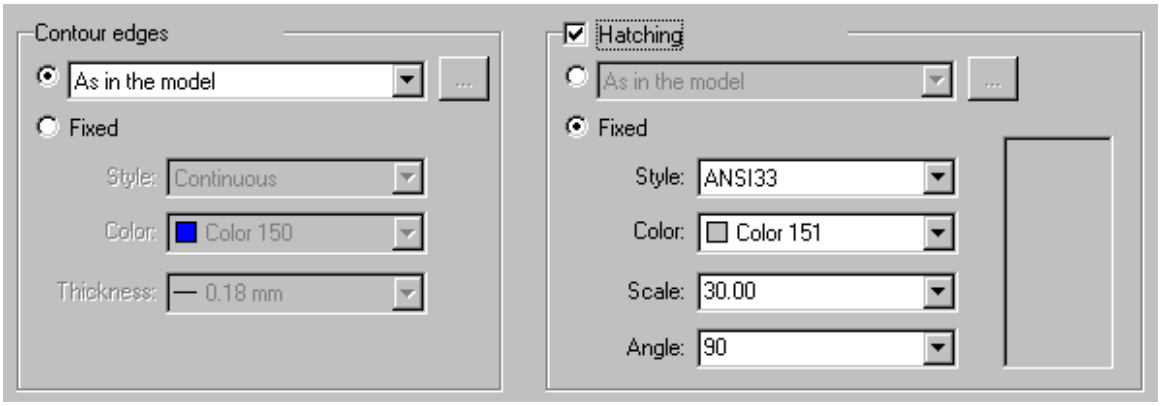
- *Drawing description* – defining a description of an elevation view; the following parameters can be defined here: color, style and size of a description.
- *Description location* – defining the location of a description in an elevation view drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

Moreover, this tab allows determining the manner an elevation view is presented in the drawing:

- structural model
- architectural model.

Visible elements tab

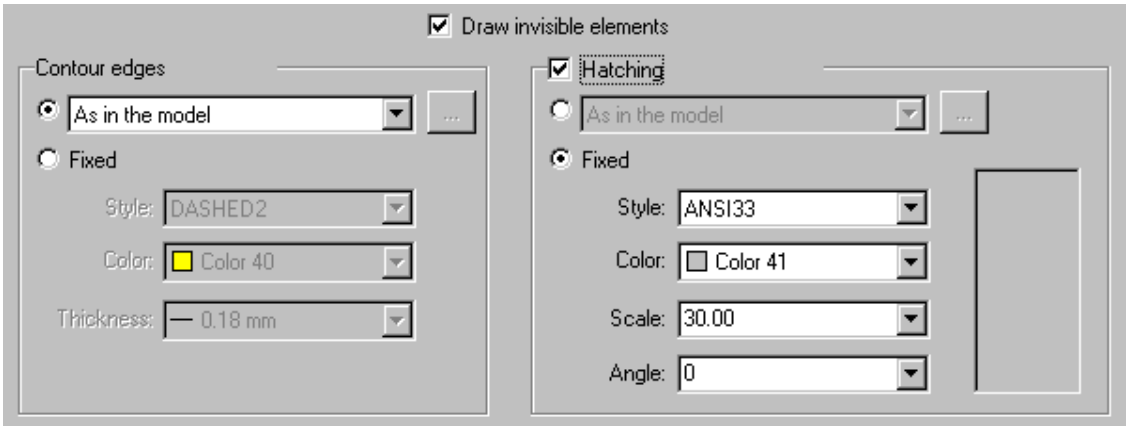


The options provided on the above tab concern parameters of presenting visible structure elements in the elevation view; for edges of the contour of structure model elements as well as for hatching the following two options are available:

- as in the model / by element type / by material (the last type is available for hatching)
- fixed - determined by the user:

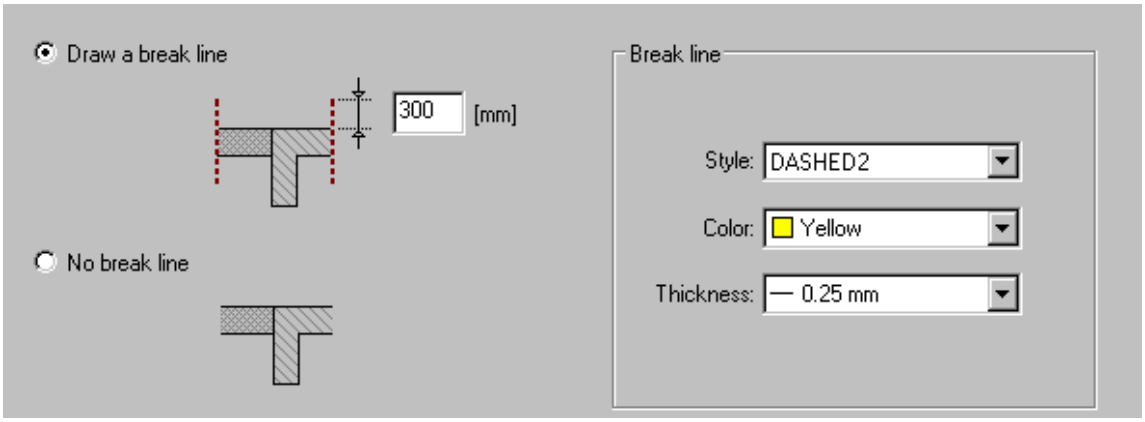
it is possible to define parameters of edges of the contour of structure model elements (line style, color and thickness) and optionally, parameters of hatching (style, color, scale and angle of hatching).

Invisible elements tab



The options provided on the above tab concern parameters of presenting invisible structure elements in the elevation view; this tab includes exactly the same options as the previous tab (*Visible elements*). Invisible elements are presented in the elevation view if the *Draw invisible elements* option is switched on.

Breaks tab



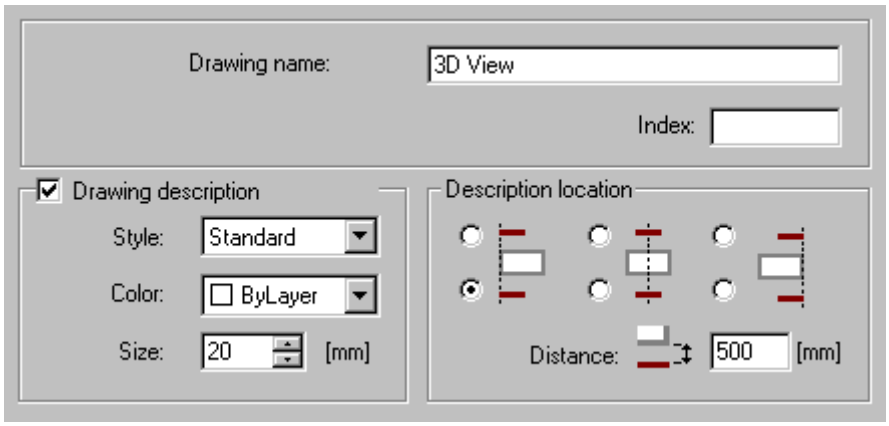
The options provided on the above tab concern parameters of presenting breaks of structure model elements in the elevation view:

- *No break line* - break lines will not be presented (no other options on this tab are available then)
- *Draw a break line* - break lines will be presented; the following options are available on this tab:
  line style, color and thickness of a break line as well as length of the line outside the structure model element.

2.14.3D view

After selecting the *3D view* option from the selection tree located in the left part of the ***Job Preferences*** dialog box, the right part of the dialog box includes several tabs discussed below.

General parameters tab

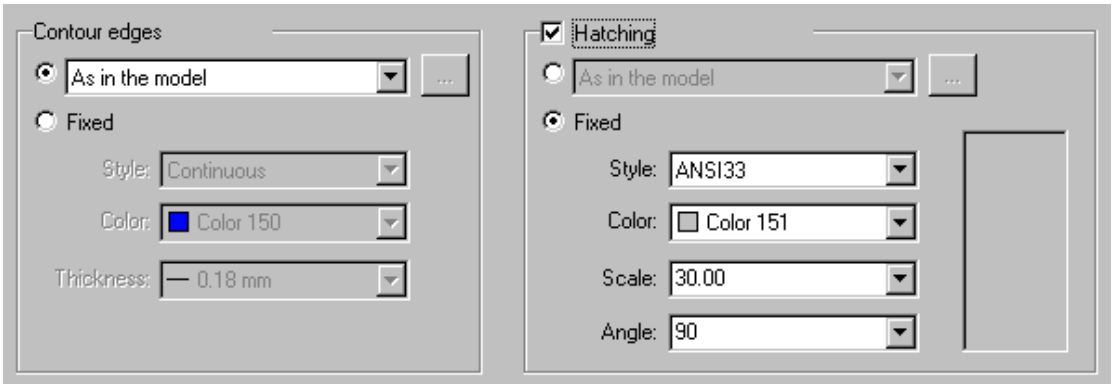


In the upper part of the dialog box a drawing name and index (number) should be specified. Below description parameters can be determined; if the *Drawing description* option is switched on, then the program makes accessible the options in the lower part of the dialog box which allow the following:

- *Drawing description* – defining a description of a 3D view; the following parameters can be defined here: color, style and size of a description.
- *Description location* – defining the location of a description in a 3D view drawing; it is possible to select several description locations (above or under the drawing, on the right side, on the left side, in the middle) and a distance between the description and the view drawing.

NOTE:
If the Drawing description option is switched on, then the defined drawing name will be included in the ***Object Inspector*** *dialog box and in the created drawing (as a drawing title); if the Drawing description option is switched off, then the defined drawing name will be included only in the* ***Object Inspector*** *dialog box.*

Visible elements tab



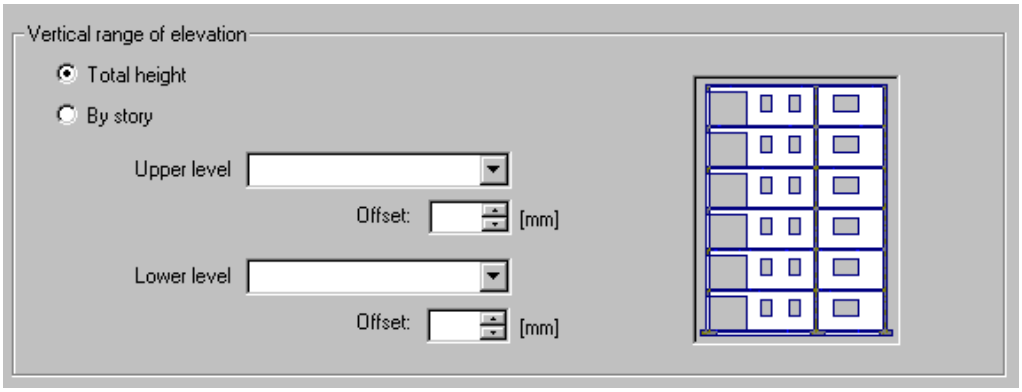
The options provided on the above tab concern parameters of presenting visible structure elements in the 3D view of a structure model; for edges of the contour of structure model elements as well as for hatching the following two options are available:

- as in the model / by element type / by material (the last type is available for hatching)
- fixed - determined by the user:
  it is possible to define parameters of edges of the contour of structure model elements (line style, color and thickness) and optionally, parameters of hatching (style, color, scale and angle of hatching).

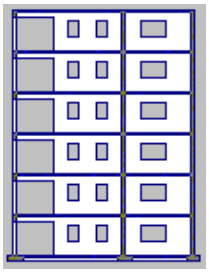
2.15. Modification of drawing parameters

The ***Modification of drawing parameters*** dialog box allow modifying values of parameters for drawings created for a structure model (foundation plan, story plan, vertical section, elevation view and 3D view).
The dialog box opens after selecting a drawing on the *Positions* tab in the ***Object Inspector*** dialog box, pressing the right mouse button and choosing the *Properties* option from the context menu.

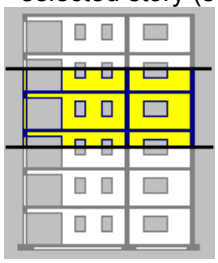
Parameters defined in the ***Modification of drawing parameters*** dialog box are identical to those in the ***Preferences / Drawings*** dialog box for individual drawing types, except the *Definition* tab for the vertical section and the elevation view as well as the *Definition* tab for the foundation plan.
The *Definition* tab for modification of the vertical section and the elevation view



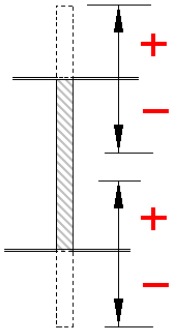
The vertical section through a structure and the structure elevation view can be made for:
- whole structure height



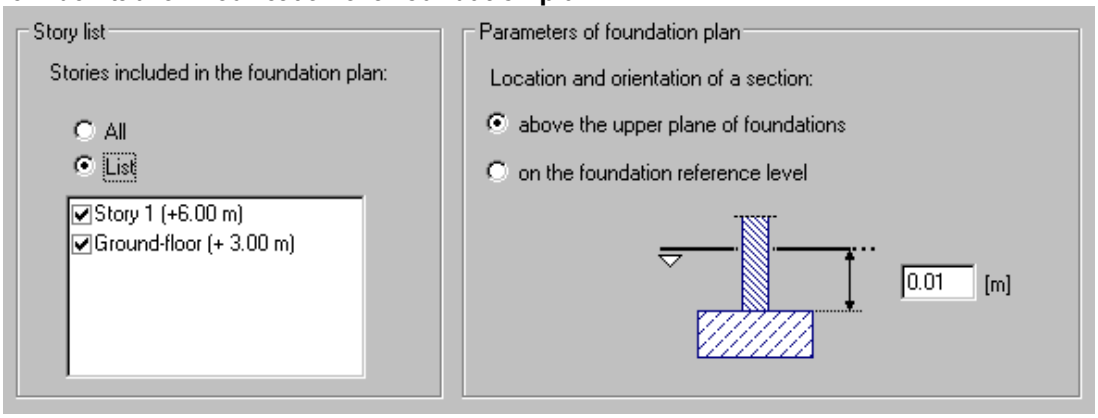
- selected story (selected stories) of a building



for this case the following parameters have to be defined:
= upper level (upper story with an offset, if needed), i.e. the upper limit of the section
= lower level (lower story with an offset, if needed), i.e. the lower limit of the section.
The following sign convention for offset values has been adopted (it refers both to the lower and upper levels of a story):



Definition tab for modification of a **foundation plan**



A foundation plan can be made for:

- all stories in a structure model (this is a default option setting), after selecting the *All* option
- selected stories of a structure model provided on the list of all stories (stories are sorted on the list), after selecting the *List* option; this option is particularly useful when foundations are positioned on several stories – it allows obtaining separate drawings of foundations for individual stories.

NOTE: To obtain changes in a foundation plan, e.g. after selecting stories to be included in the foundation plan, it is necessary to update the foundation plan.

3. OBJECT INSPECTOR

3.1. Description of Object Inspector

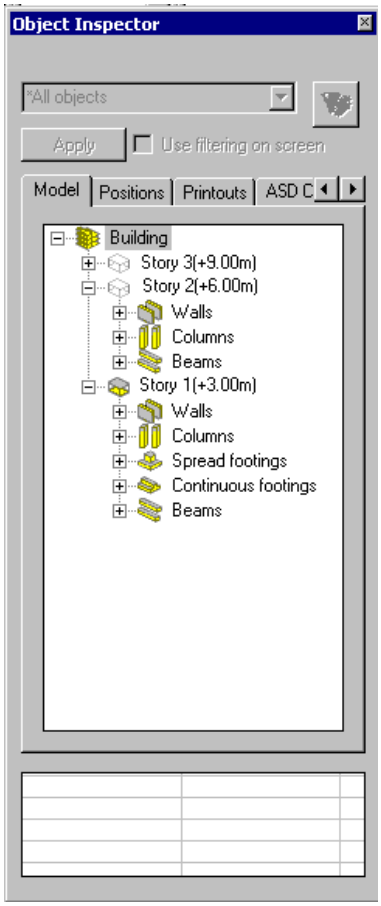
Inspector is a tool for managing elements (objects) included in a project created in ***AutoCAD © Structural Detailing - Formwork Drawings***. By standard, the Inspector dialog box is presented in left part of the program viewer, next to the field for graphical model definition.
The most important tasks carried out by Inspector include:

- presenting contents of a project in the appropriate order (sequence)
- selecting elements that should be acted on by a chosen command
- presenting and modifying properties of project elements (these may be single elements or whole objects)
- activating some of the commands applied to a particular selection (it depends on the inspector mode)
- verifying and modifying positions
- creating and managing drawing documentation of a project.

3.2. Object Inspector

The ***Object Inspector*** dialog box is by standard located to the left of the field of graphical model definition. Width of the dialog box may be freely adjusted to leave as much space as possible for the field of graphical definition of a structure model.
The ***Object Inspector*** dialog box, shown in the drawing below, may be divided into the following parts:

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



The middle part of the dialog box holds the following tabs:
- Model
- Positions
- Printouts
- ASD Center.

This part of the dialog box allows presenting, on the successive tabs, a current list of objects defined in an ***AutoCAD © Structural Detailing - Formwork Drawings*** project. Each of the tabs is intended for presenting elements on different stages of a design process, therefore, each of them comprises different types of objects organized in a slightly different way.

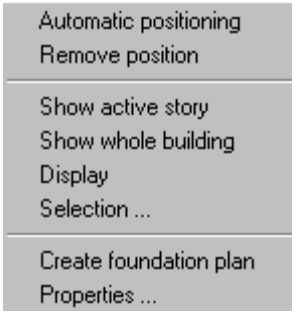
At the bottom of the dialog box is the table with properties of elements selected on the list. The table works as all dialog boxes with properties in the AutoCAD © program. Table features include:
- if one of the elements is selected, then the table presents all properties of the selected element
- if several objects of the same type are selected, then the table presents all the fields with properties relevant to the selected element type, however, only the values that are common for all the elements are presented; the remaining fields are blank (yet they may be changed)
- if several objects of different types are selected, the table presents only the fields with properties that are common for all object types.

3.3. Model tab

This tab presents story by story model elements defined in a structure (beams, walls, columns, spread footings and continuous footings, ground beams, slabs, raft foundations, lintels).
By standard, the whole structure model is called *Building*. This name may be changed in the ***Building Parameters*** dialog box which opens after selecting the *Properties* option in the context menu. Names of successive stories can be defined in this dialog box as well.
Elements found on the list may be selected with the mouse cursor (it is possible to any number of elements). When selected on the list, an element becomes simultaneously highlighted in the graphical editor (the selection operation is interactive, in other words, all the elements chosen in the Inspector dialog box are highlighted in the graphic editor and the other way round: if an element is selected in the graphic editor, then the selection is also presented in the **Inspector** dialog box).

The context menu, that appears in the ***Object inspector*** dialog box after pressing the right mouse button, changes depending on what the user selects: an element group, an element or a building.

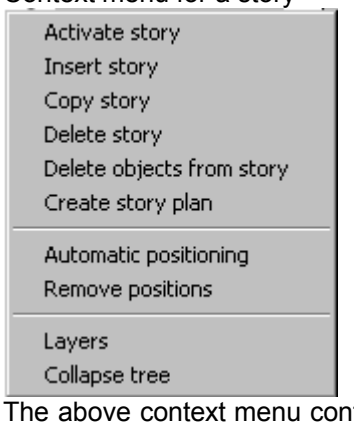
Context menu for a whole building



The above context menu contains the following options that allow performing operations for the whole building:
- *Automatic positioning* – selecting this command allows assigning positions to all elements defined in the whole building; after running this command, the ***Automatic positioning*** dialog box opens on the screen
- *Remove position* – selecting this command removes positions assigned to all elements defined in the whole building;

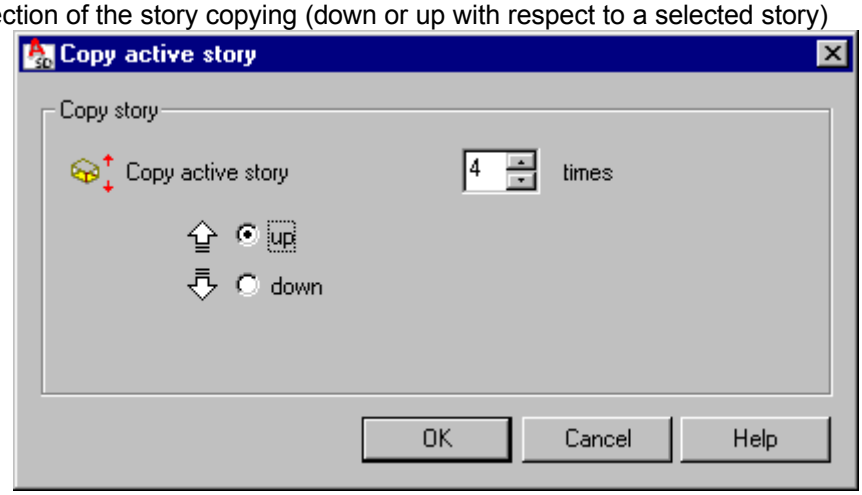
- *Show active story* – after selecting this command, only the active (highlighted in the ***Object Inspector*** dialog box) story is presented on the *Model* layout
- *Show whole building* - after selecting this command, the whole building model is presented on the *Model* layout
- *Display* - selecting this command opens the ***Filters - element selection*** dialog box; this dialog box is used to select elements for presentation on the *Model* layout
- *Selection* - selecting this command opens the ***Filters - element selection*** dialog box; this dialog box is used to choose elements to be selected in a building model
- *Create foundation plan* - selecting this command automatically generates a projection (plan) of a building foundation
- *Properties* - selecting this command opens the ***Building parameters*** dialog box.

Context menu for a story



The above context menu contains the following options that allow performing operations for a selected story:

- *Activate story* - selecting this command activates a selected story in a building
- *Insert story* - selecting this command inserts a story above or below a selected story; the inserted story will have a default story height defined in the ***Building parameters*** dialog box
- *Copy story* - selecting this command copies a chosen story above or below a selected story; the program opens the ***Copy active story*** dialog box, shown in the drawing below, for definition of the copying parameters:
  - number of repetitions
  - direction of the story copying (down or up with respect to a selected story)

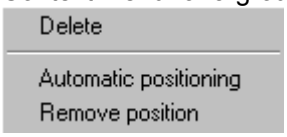


- *Delete story* - selecting this command deletes a selected story from a building model (along with all the objects situated on that story)
- *Delete objects from story* - selecting this command deletes all objects situated on a selected story (the story is not deleted from a building model)
- *Create story plan* - selecting this command automatically generates a projection (plan) of a selected story

- *Automatic positioning* – selecting this command allows assigning positions to all elements defined in a whole building; after running this command, the ***Automatic positioning*** dialog box opens on the screen
- *Remove positions* – selecting this command removes positions assigned to all elements defined in a whole building.
- *Layers* - selecting this command opens the ***Layers*** dialog box.

NOTE:
When copying a story, spread footings, continuous footings, ground beams and raft foundations are not copied to higher stories.

Context menu for a group of elements (e.g. walls) or for an element on a selected story



The above context menu contains the following options that allow performing operations for a selected element (or a group of elements):

- *Delete* - selecting this command deletes a selected element (or a group of elements) from a structure model
- *Automatic positioning* – selecting this command allows assigning a position to a selected element (or a group of elements, e.g. walls); after running this command, the ***Automatic positioning*** dialog box opens on the screen
- *Remove position* - selecting this command removes positions assigned to all elements of a group or a position assigned to a selected element.

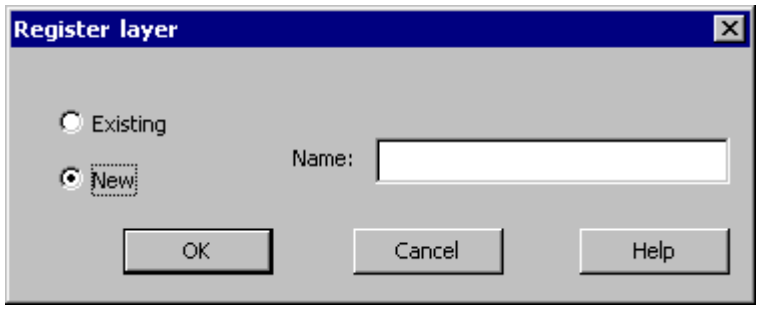
3.4. Layers - Structure model

The dialog box shown below is used to define a layer in a structure model (e.g. a layer of an architectural project of a structure).

Layers are defined separately for each story.
To add a layer to a given story (the story name is presented in the top part of the dialog box), the **Add** button should be pressed. Then the ***Register layer*** dialog box opens.
Pressing the **Delete** button deletes a selected layer from the list of layers available for a given story.

3.5. Register layer - Structure model

The dialog box shown below is used to register a layer in a structure model (e.g. a layer of an architectural project of a structure).

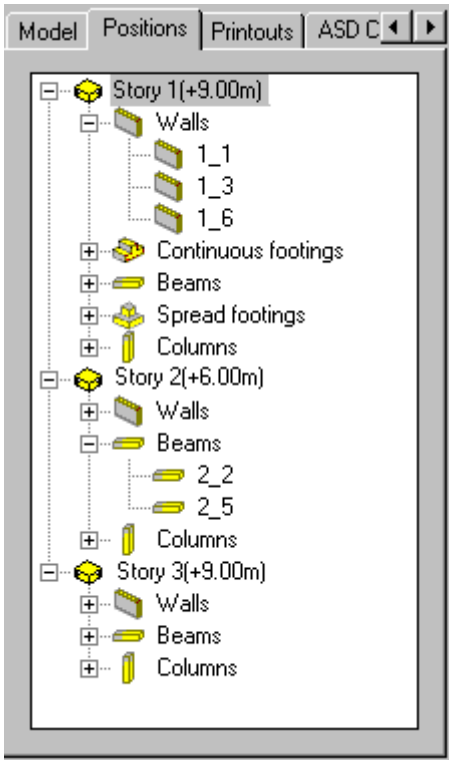


There are two methods of defining / registering a layer:

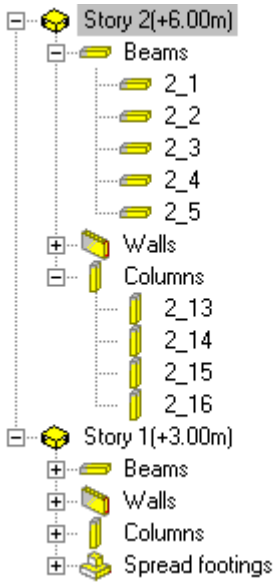
- *Existing* - from the list of defined layers the user should select the name of a layer to be added; NOTE: the layer must be added earlier using the AutoCAD © options
- *New* - after selecting this option, the user should specify a layer name, and next, determine the layer in a structure model.

3.6. Positions tab

This tab presents a list of defined positions; the list is sorted by prefixes and numbers.



This tab presents a list of all positions defined by the user by means of the *Positioning* option; the structure of positions and documents is shown in a form of a tree (see the drawing below).

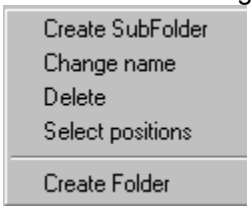


The icon of a position on the list may be presented as (see the description of the context menu for a single position):

- – the icon for documents within an ***AutoCAD © Structural Detailing - Formwork Drawings*** project (it means that a given position has a formwork drawing in the same project as a structure model)
- – the icon means that a given position has a formwork drawing created in ***AutoCAD © Structural Detailing - Reinforcement***. This drawing is saved in a separate file. Double-clicking on this icon automatically runs ***AutoCAD © Structural Detailing - Reinforcement*** and opens the formwork drawing of a given position.

The context menu, that appears in the ***Object inspector*** dialog box after pressing the right mouse button, changes depending on what the user selects: a group of positions (e.g. walls of a selected story) or a single position.

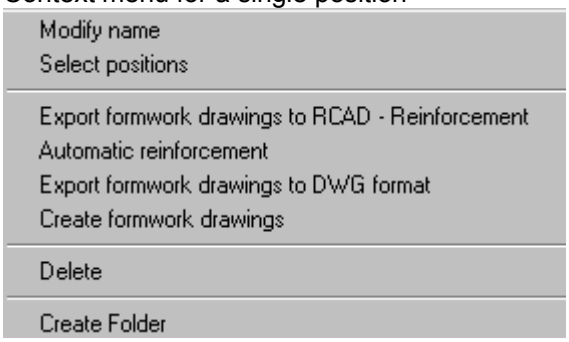
Context menu for a group of positions



The above context menu contains the following options that allow performing operations for a selected a group of positions:

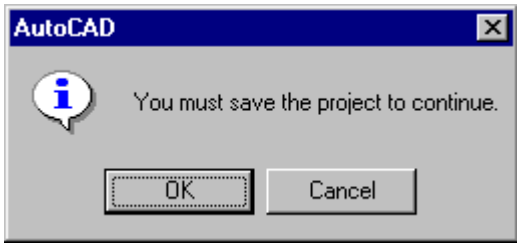
- *Create SubFolder* – selecting this command allows creating an additional subfolder in the position tree
- *Change name* - selecting this command allows changing the name of a selected folder; a new folder name should be typed in the command line
- *Delete* - selecting this command deletes a selected folder (a group of positions)
- *Select positions* - selecting this command allows selection of positions
- *Create Folder* – selecting this command allows creating an additional folder in the position tree.

Context menu for a single position



The above context menu contains the following options that allow performing operations for a selected position:

- *Modify name* - selecting this command allows modifying the name of a selected position; a new position name should be typed in the command line
- *Select positions* - selecting this command allows selection of positions
- *Export formwork drawings to* ***AutoCAD © Structural Detailing*** *- Reinforcement* - selecting this command allows reading formwork drawings by ***AutoCAD © Structural Detailing - Reinforcement***; the program opens the ***Formwork drawing wizard*** dialog box for selection of drawing positions and indicating a file where to save information (see the description of the ***Formwork drawing wizard*** dialog box)
  NOTE: To perform this operation, a project **must** be saved in a file with a specified name; if the project is not saved, the message shown in the drawing below appears on the screen



- *Automatic reinforcement* - selecting this command allows automatic generation of reinforcement for a selected position in ***AutoCAD © Structural Detailing - Reinforcement***; the ***AutoCAD © Structural Detailing - Reinforcement*** program runs then
- *Export formwork drawings to DWG format* - selecting this command allows saving formwork drawings in a *.DWG format file; the program opens the ***Formwork drawing wizard*** dialog box for selection of drawing positions and indicating a file where to save information (see the description of the ***Formwork drawing wizard*** dialog box)
- *Delete* - selecting this command deletes a selected position
- *Create Folder* - selecting this command allows creating an additional folder in the position tree.

NOTE:
The options allowing export of formwork drawings and automatic reinforcement are available in the context menu after positioning of the building elements has been performed.

Additionally, the following options are available for structure plans / views (story plan, foundation plan, elevation view, section through a structure, 3D view):

- *Activate* - selecting this command activates a chosen drawing
- *Update* - selecting this command updates a selected drawing (after changes made in a structure model)
- *Properties* - selecting this command opens the ***Modification of parameters for drawings*** dialog box for a given plan / view of a structure

- *Hide objects* - selecting this command hides some of the objects (section symbols and axes) in structure plans / views
- *Show all hidden objects* - selecting this command restores the display of hidden objects (section symbols and axes) in structure plans / views
- *Layers* - selecting this command opens the ***Layers*** dialog box.

3.7. Printouts tab

This tab enables managing printouts in ***AutoCAD © Structural Detailing - Formwork Drawings***; it presents a list of all printouts defined in an ***AutoCAD © Structural Detailing - Formwork Drawings*** 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. From the logical viewpoint, views are placed in a printout, however, for the user's convenience, the tree also includes the 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 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 a printout in the dialog box is synchronized with the graphic editor - an appropriate drawing is displayed on the screen.

The context menu, which appears after pressing the right mouse button while the cursor is positioned on the *Printouts* tab, contains several options allowing operations on selected printouts:

- *Delete* – selecting this command deletes chosen printouts from a project
- *Change name* - selecting this command allows changing the name of a highlighted printout (the name is entered to the command line)
- *Add printout* - selecting this command adds a blank printout to the project (the name is entered to the command line)
- *Activate* - selecting this command activates a chosen printout
- *Unload printout* - selecting this command makes a chosen printout inactive
- *Save printout* - selecting this command saves a chosen printout to a *.DWG format file.

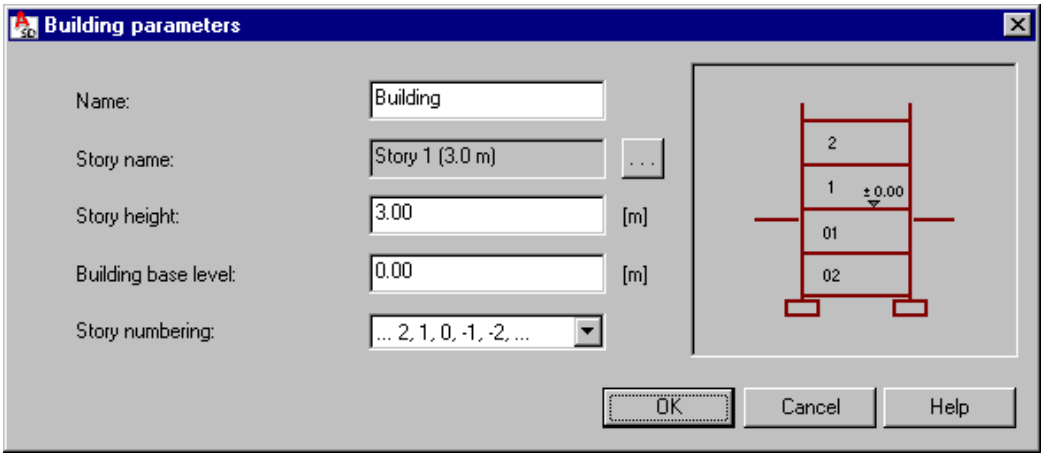
3.8. Structural Detailing Center tab

The options located on this tab are used to copy databases (sections, prefabricated elements, materials, etc.) 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 databases defined in the selected project, which may be used in the current project.
After highlighting a selected database (element), pressing the right mouse button and choosing the *Add* command, the selected database element is available in the current project.

4. BASIC PARAMETERS OF A BUILDING

4.1. Building properties

The ***Building parameters*** dialog box is used to define or modify basic data concerning a designed building. The *Properties* option is available in the context menu on the *Model* tab in the ***Object Inspector*** dialog box.

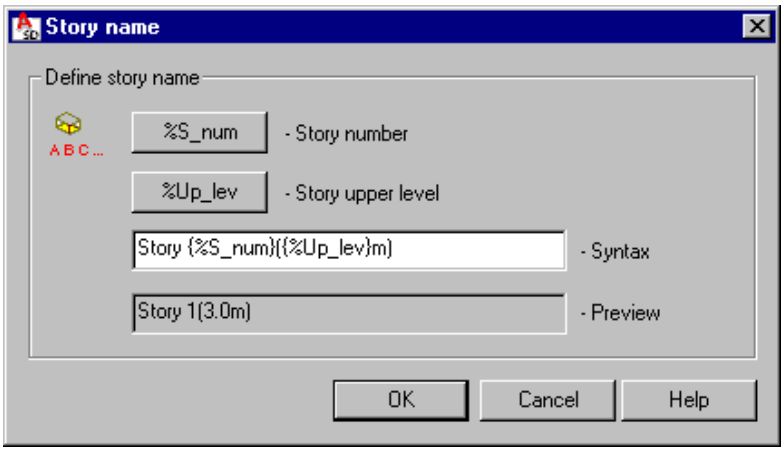


The following parameters of a building can be defined in the above dialog box:

- building name (the name will be presented in the ***Object Inspector*** dialog box and in drawings)
- story name - pressing the button (**...**) opens an additional dialog box ***Story name***
- default story height is used when:
  - defining a new story
  - defining parameters of a story plan
- building base level (reference level) – the level referring to the bottom of the first story; this value has direct effect on the story name (if the variable of the story level is used) in the dialog boxes for definition of structure elements (on the *Vertical definition* tab) and while inserting elevation marks in drawings
- format of the story numbering.

4.2. Story name

The ***Story name*** dialog box allows defining a syntax of names assigned to stories in a building. The dialog box opens on pressing the (**...**) button in the ***Building parameters*** dialog box.



The options provided in this dialog box are used to define a syntax of a story name; apart from the defined general style of names assigned to stories, the above dialog box offers a mechanism for free composition of the syntax and the contents of a story name.
The list of variables included in a story name includes as follows:

%S_num - story number
%Up_lev - story upper level.

By pressing the relevant button a name components are placed in the field including definition of the syntax of a story name. The *Preview* field shows the story name resulting from a defined syntax.
A story name defined using the above syntax is taken into account only when defining a new story or copying an existing one.

5. WORKFRAMES

5.1. Workframe

The option is used to define a workframe which makes definition of a structure model on the plane easier. There are two types of workframes available in the program: rectangular workframe and circular workframe. The option is available from:
RECTANGULAR WORKFRAME

- the menu by selecting the option Formwork Drawings / Workframes / Insert rectangular workframe
- the toolbar by pressing the  icon
- the command line: RBCX_DEF_WORKFRAME.

CIRCULAR WORKFRAME

- the menu by selecting the option Formwork Drawings / Workframes / Insert circular workframe
- the toolbar by pressing the  icon
- the command line: RBCX_DEF_WORKFRAME_RADIAL.

The upper part of the dialog box holds a selection list which enables choosing a workframe created earlier or defining a workframe with a new name (the name should be entered on the selection list). To the right of the selection list there are two buttons:

- **Save** - saves a defined workframe under the name entered in the *Name* selection field
- **Delete** - deletes a selected workframe from the list.

Workframes and axes can be displayed in created plans and views (foundation plan, cross-section, elevation view, etc.) Workframes are not presented in views / plans as one object; each workframe axis is displayed separately. That way of presenting axes enables modification of parameters for each axis separately (color, length, line type, etc.).
Structure views / plans show only axes which:
- intersect with elements of a structure model (plus one axis more on each side)
- are positioned within the defined range of the cross-section.

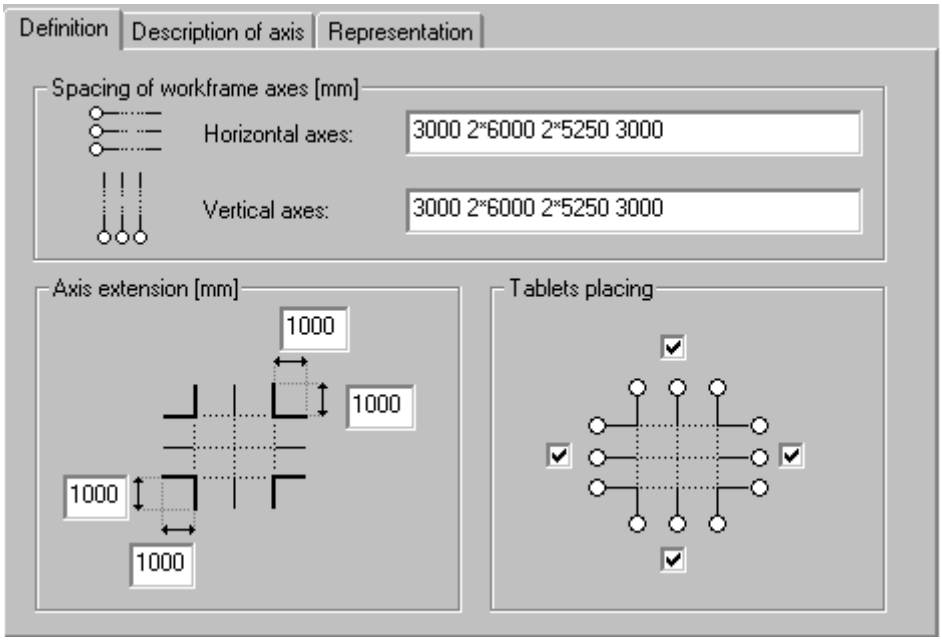
The right part of the dialog box includes two icons:
 *Insert* - creates a workframe based on parameters defined in the dialog box
 *Inherit properties* - adopts parameters of a workframe defined earlier.

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

5.2. Rectangular workframe

The dialog box for definition of a rectangular workframe consists of the three tabs listed below:

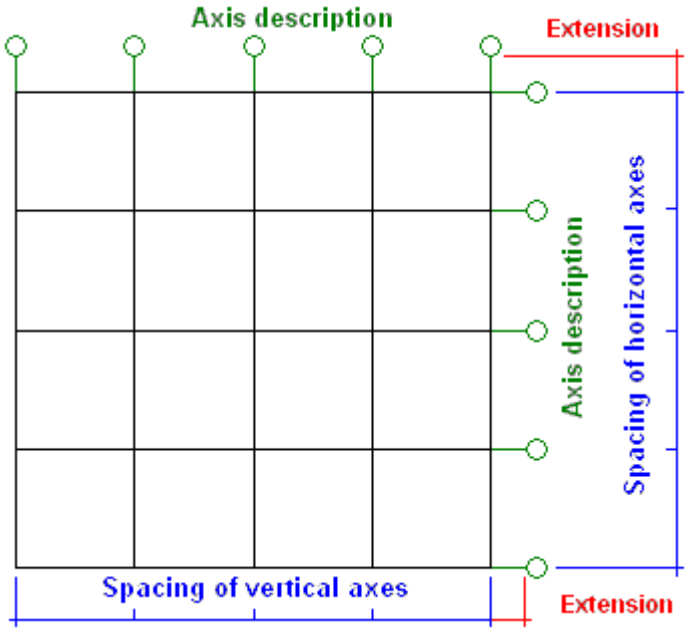
Definition
The *Definition* tab is shown in the following drawing



The dialog box holds the options as follows:

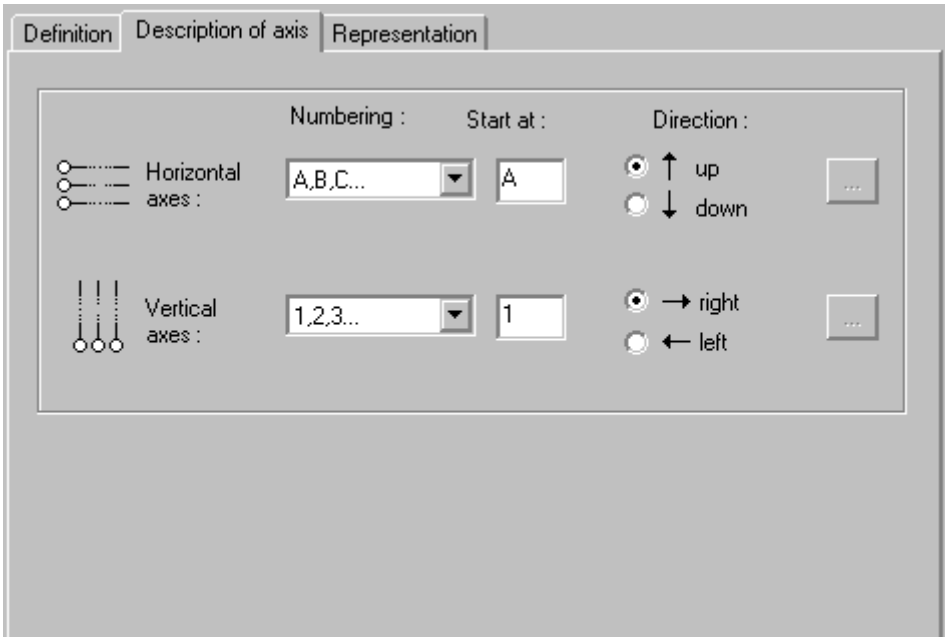
- the *Spacing of workframe axes* field allows defining workframe spacings (vertical and horizontal)
- the *Axis extension* field allows defining values of the axis extension outside the workframe
- the *Tablets placing* field allows selecting locations of axis descriptions.

All these parameters are illustrated in the drawing below (axis extensions and descriptions are defined only at the top and on the right side of the workframe).



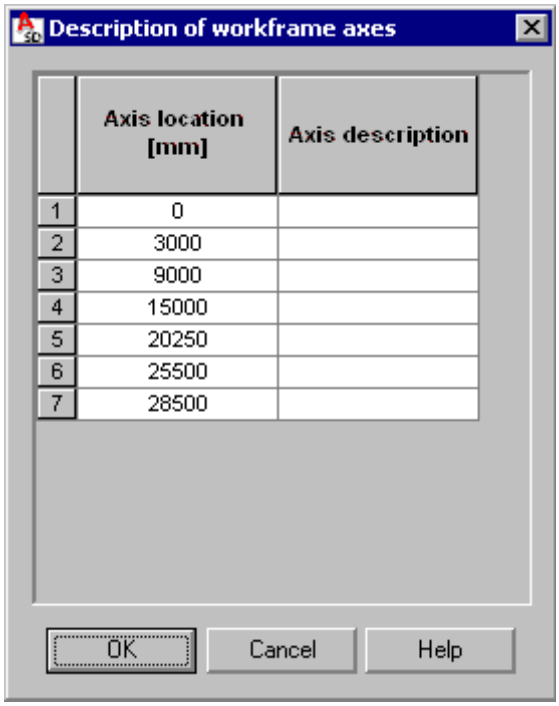
Description of axis

The *Description of axis* tab is shown in the drawing below



The following options are available for horizontal and vertical axes:

- *Numbering* - the field for selecting a method of axis description; the following description types are available:
  1, 2, 3 ...
  A, B, C ...
  I, II, III ...
  a, b, c ...
  user-defined - if the *user-defined* option is selected for any axis, then the (**...**) button is accessible, which when pressed, opens the ***Description of workframe axes*** dialog box where the user may define descriptions of successive workframe axes

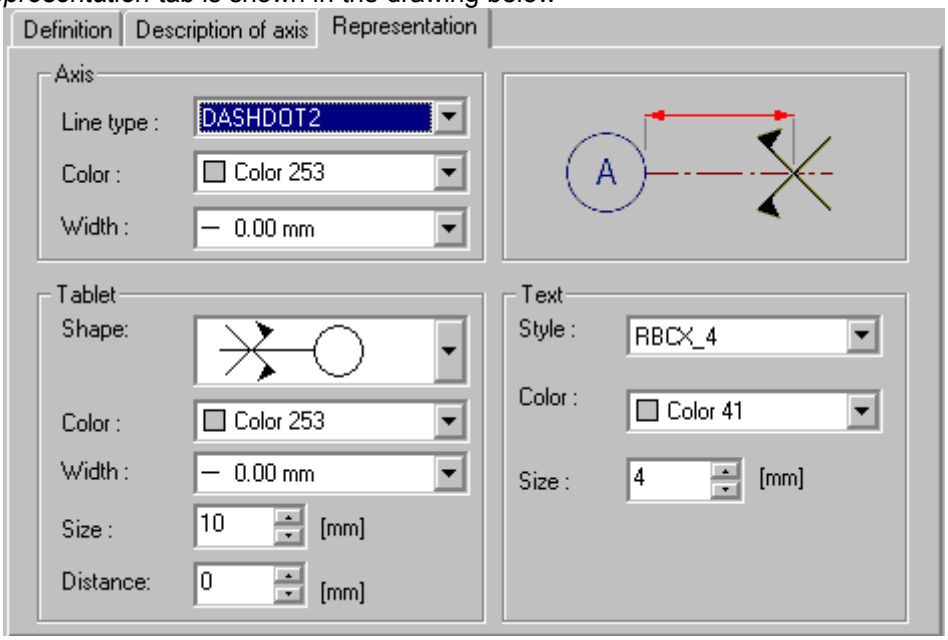


To define descriptions of workframe axes, follow the steps below:
- specify a position and a description for the first axis
- move on to the next table line and specify a position and a description of the second axis

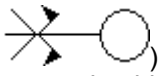
- perform the above operation for defined workframe axes
- press the **OK** button.

- *Initial value* - the *Start at* field is used to determine an initial value of the axis description (depending on a selected axis description type)
- *Description direction* - the options are used to choose a direction of axis description (for horizontal axes: up or down, for vertical axes: right or left).

Representation
The *Representation* tab is shown in the drawing below



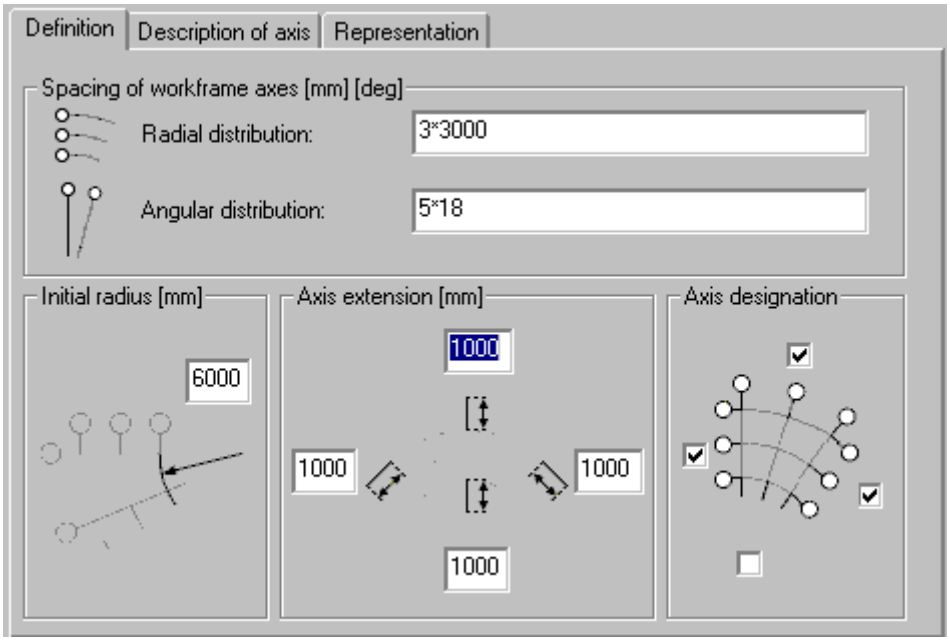
The *Axis* field holds parameters of a line presenting the axis in a drawing: line type, color and thickness.
The *Tablet* field allows determining parameters of a label of the axis description: label shape, size, color and thickness of the label line; the following label shapes are available in the program: circle, ellipse, square, octagon.

If a tablet shape including an axis designation (e.g. ) is selected, then the *Distance* edit field is also available and in this field the user should specify a value of the distance between the two symbols.
The options in the *Text* field refer to the axis description included in a label. The following parameters can be set here: font style, color and size.

5.3. Circular workframe

The dialog box for definition of a circular workframe consists of the three tabs listed below:

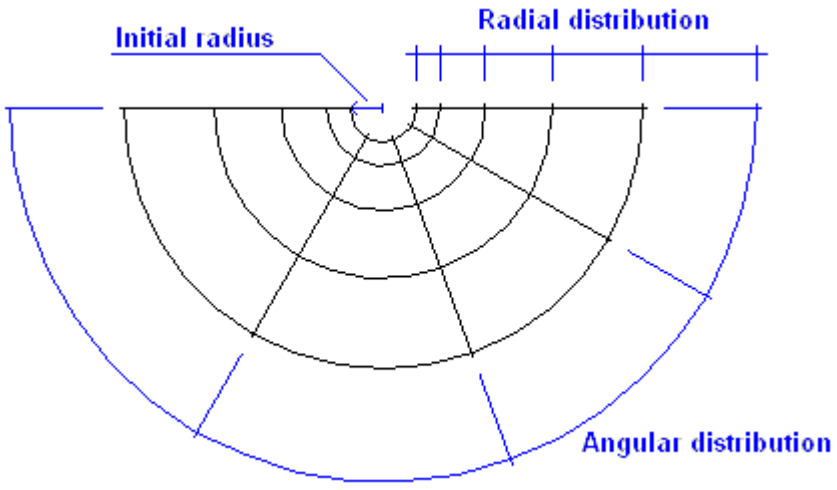
Definition
The *Definition* tab is shown in the following drawing



The dialog box holds the options as follows:

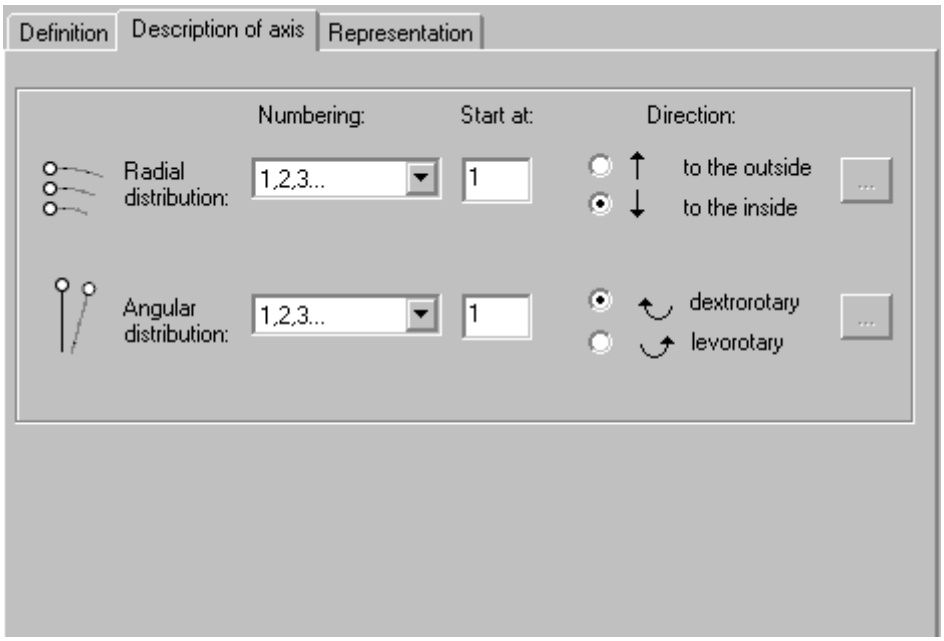
- the *Spacing of workframe axes* field allows defining workframe spacings (radial and angular)
- the *Initial radius* field allows defining a value of the radius with which a radial workframe will begin
- the *Axis extension* field allows defining values of the axis extension outside the workframe
- the *Axis designation* field allows selecting locations of axis descriptions.

All these parameters are illustrated in the drawing below (axis extensions and descriptions are defined only at the top and on the right side of the workframe).



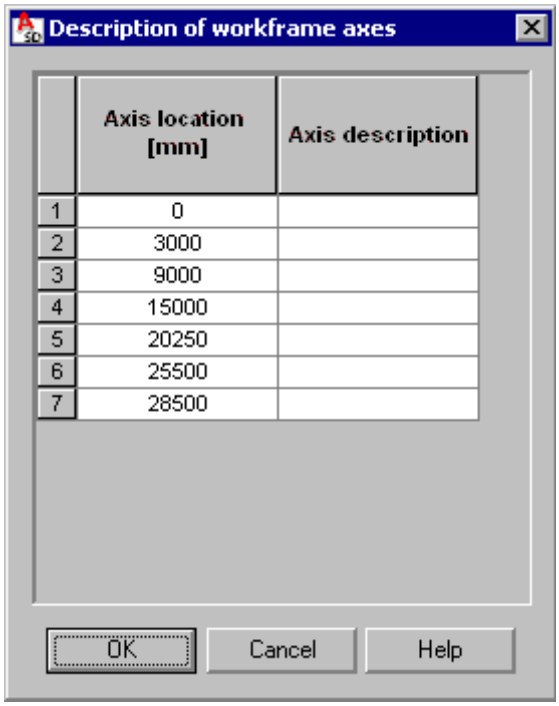
Description of axis

The *Description of axis* tab is shown in the drawing below



The following options are available for horizontal and vertical axes:

- *Numbering* - the field for selection of a method of axis description; the following description types are available:
  1, 2, 3 ...
  A, B, C ...
  I, II, III ...
  a, b, c ...
  user-defined - if the *user-defined* option is selected for any axis, then the (**...**) button is accessible, which when pressed opens the ***Description of workframe axes*** dialog box where the user may define descriptions of successive workframe axes

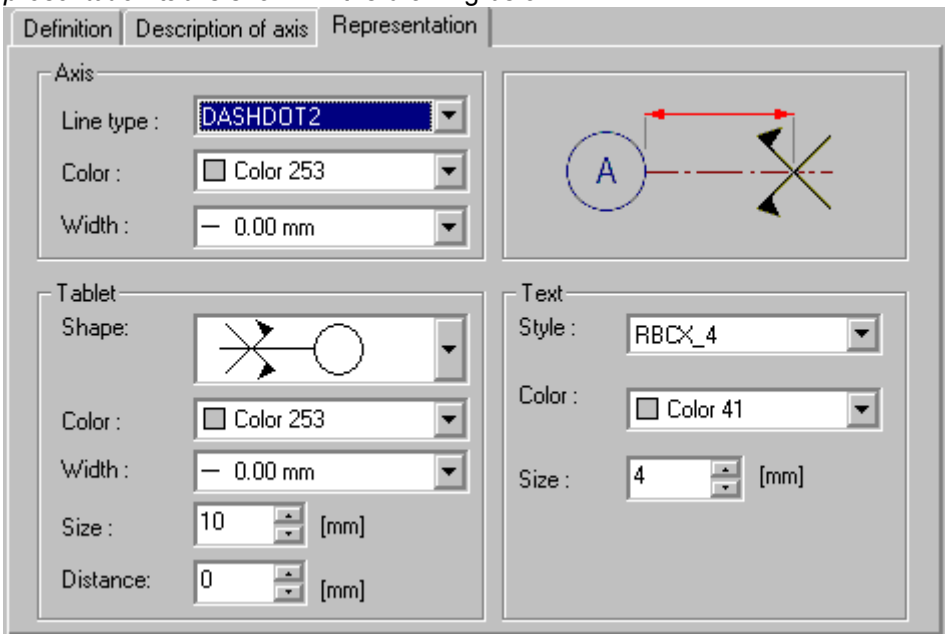


To define descriptions of workframe axes, follow the steps below:
- specify a position and a description for the first axis
- move on to the next table line and specify a position and a description of the second axis

- perform the above operation for defined workframe axes
- press the **OK** button.

- *Initial value* - the *Start at* field is used to determine an initial value of the axis description (depending on a selected axis description type)
- *Description direction* - the options are used to choose a direction of axis description (for the radial distribution: to the outside or to the inside, for the angular distribution: dextrorotary or levorotary).

Representation
The *Representation* tab is shown in the drawing below



The *Axis* field holds parameters of a line presenting the axis in a drawing: line type, color and thickness.
The *Tablet* field allows determining 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.

If a tablet shape including an axis designation (e.g. ) is selected, then the *Distance* edit field is also available and in this field the user should specify a value of the distance between the two symbols.
The options in the *Text* field refer to the axis description included in a label. The following parameters can be set here: font style, color and size.

6. DEFINITION OF STRUCTURE ELEMENTS

6.1. Beams / ground beams

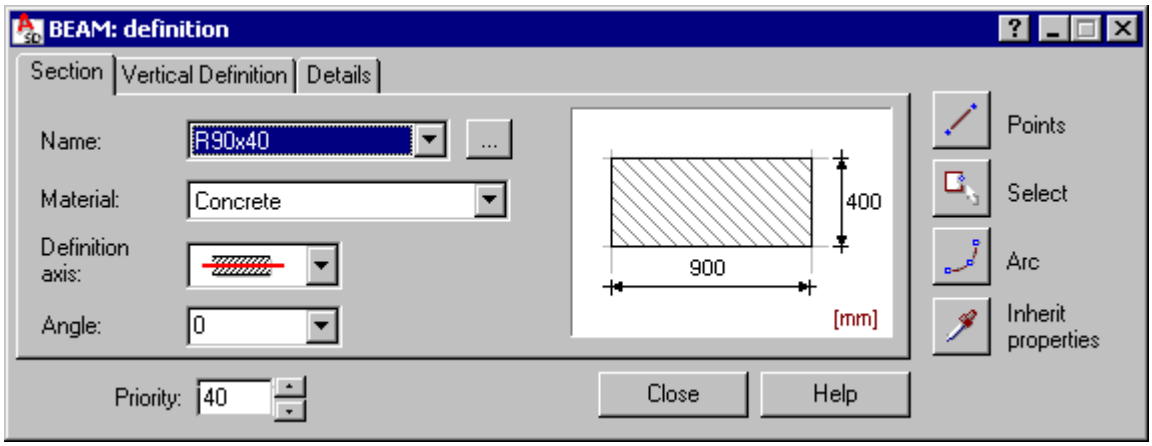
6.1.1. Beam / ground beam - definition

The option enables definition of a beam or a ground beam in a structure model. Beams are inserted in horizontal plans of an active story (in the top view).
The beam may be defined both in the 3D view and the 2D view.
Definition of the beam / ground beam starts with:

- selecting the menu option *Formwork Drawings / Define / Beam* or *Formwork Drawings / Define / Ground beam*
- pressing the icon or
- entering into the command line: RBCX_DEF_BEAM or RBCX_DEF_GROUND_BEAM.

NOTE:
When copying a story, ground beams are not copied to higher stories.

After activating the *Beam definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition
Details.

At the bottom of the dialog box is the Priority field which presents a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for the defined beam.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of the beam. NOTE: only after pressing one of these icons definition of the beam is possible in the graphic viewer of the program.

The beam may be defined as:

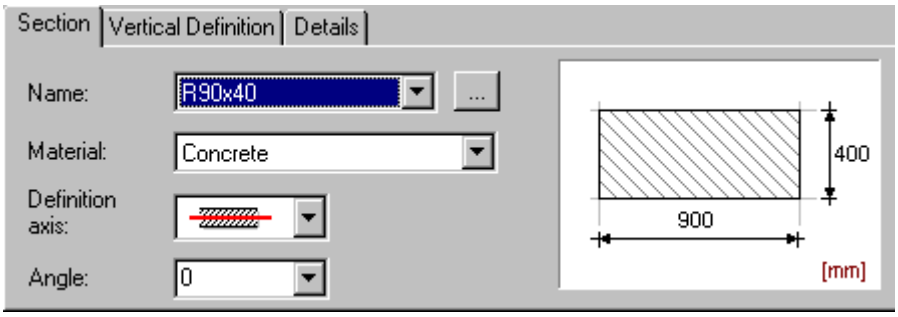
- rectilinear beam: by defining the beginning and end points of the beam in a plan of a building story
- arc beam – curved: by defining three successive points of an arc belonging to the beam in a plan of a building story (see also: Definition of parameters of arc elements – Job Preferences / Options).

Moreover, there is also the  *Select* icon available which, when pressed, enables turning indicated arcs and open polylines to beams.

The icon  located in the bottom right corner is used to inherit parameters from a beam defined earlier.

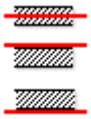
6.1.2. Beam - section

The *Beam: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



This dialog box allows defining parameters of a beam cross-section. The following parameters should be determined for each type of the beam cross-section:

- beam section by selecting the section's name from the list of available sections; pressing the (**...**) button located to the right of the selection list opens the ***Section list*** dialog box which allows adding a new section to the section list
- material the beam is made of
- axis of beam definition; the axis of beam definition may be assumed as:

   center line

  upper line

  lower line

  The methods of definition using the center, upper and lower lines are illustrated schematically in the drawings below.

  Center line is selected:

  Upper line is selected:

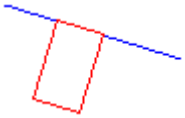
  Lower line is selected:

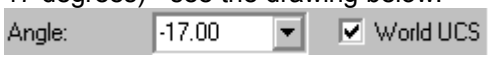
  When defining a beam by means of the outer or inner line, the user may determine an additional parameter (an offset from the definition line) in the edit field next to the icon  ; if the offset equals zero, then the definition line coincides with the upper or lower line.
- angle of rotation of the beam cross-section; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of a rotation angle. NOTE: While defining a beam lying on an inclined plane, there is also the *World UCS* option available (see the drawing below).

The option makes it easier to define the orientation of the beam cross-section with respect to the global coordinate system.
If the *World UCS* option is switched off, then angle values provided on the selection list refer to the orientation of an inclined plane. If the *World UCS* option is switched on, then angle values provided on the selection list refer to directions of axes of the global coordinate system (see the examples below).
NOTE: After switching on the *World UCS* option, an angle is converted to a new reference system.

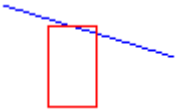
The *World UCS* option switched off
Angle = 0



After switching on the *World UCS* option, for the same position of the cross-section, an angle of plane inclination is calculated on the selection list (in this case, it is the angle = 17 degrees) - see the drawing below.

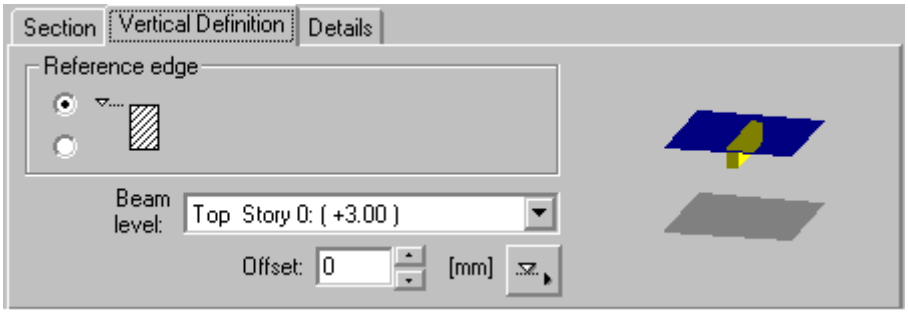


After changing the angle to the one equal to zero and with the *World UCS* option switched on, the position of the beam cross-section changes - see the drawing below.



6.1.3. Beam - vertical definition

The *Beam: definition* dialog box looks as presented in the drawing below after selecting the *Vertical Definition* tab in it.



This dialog box is used to determine a location of a beam with respect to an active story.
The *Beam level* list includes all the levels of an active story of a building defined so far; the location of a beam in a building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2). This list also contains all inclined planes defined for a selected story (see: Definition of a plane).

In addition, there is a possibility to define the reference edge of the beam with respect to the upper or lower level of a story. The following situations are possible:

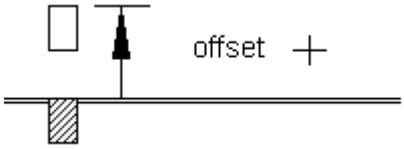
	Reference edge	**Location of the beam with respect to the story level**
1.		

2.  

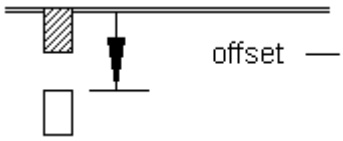
When defining the vertical location of a beam, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story. Pressing the  button allows selecting a horizontal element edge determining an offset value.

The following sign convention for offset values has been adopted (it refers both to the lower and upper levels of a story):

1. positive offset value

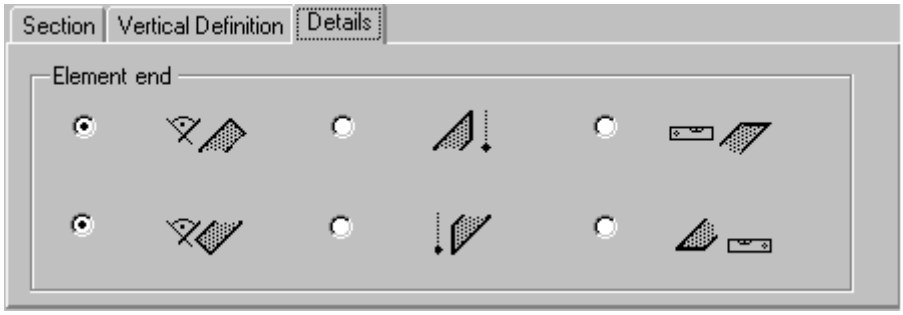


2. negative offset value



6.1.4. Beam - details

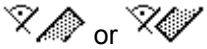
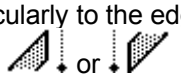
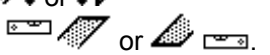
The ***Beam: definition*** dialog box looks as presented in the drawing below after selecting the *Details* tab in it.



NOTE:
Options on this tab are available only then, when an inclined plane has been chosen as a reference level in the dialog box on the Vertical Definition tab.

Options in the above dialog box allow selecting a type of ends of the beam lying on an inclined plane

(at the beginning as well as at the end of a beam):

1. perpendicularly to the edge:  or
2. vertically:  or
3. horizontally:  or .

6.1.5. Beam definition - command line

While defining a beam or a ground beam, the following parameters can be determined in the command line:

Points

Pick the beginning point:
Pick next point [Side / Offset]:

where:
Side - location of the definition axis of a beam (upper, lower, central)
Offset - distance from the lower or upper level of a beam
NOTE: When defining an element on the axis (axis location: central), offset and side are not supported.

Select

Select object:
Finish [Side / Offset]:

where:
Side - location of the definition axis of a beam (upper, lower, central)
Offset - distance from the lower or upper level of a beam
NOTE: Lines, arcs, polyline do not need to be located in the same plane; while creating an element, it is projected on a specified plane(s).

Arc

Pick first point:
Pick second point [Side / Offset]:
Pick third point [Side / Offset]:

where:
Side - location of the definition axis of a beam (upper, lower, central)
Offset - distance from the lower or upper level of a beam

Inherit properties

Select object

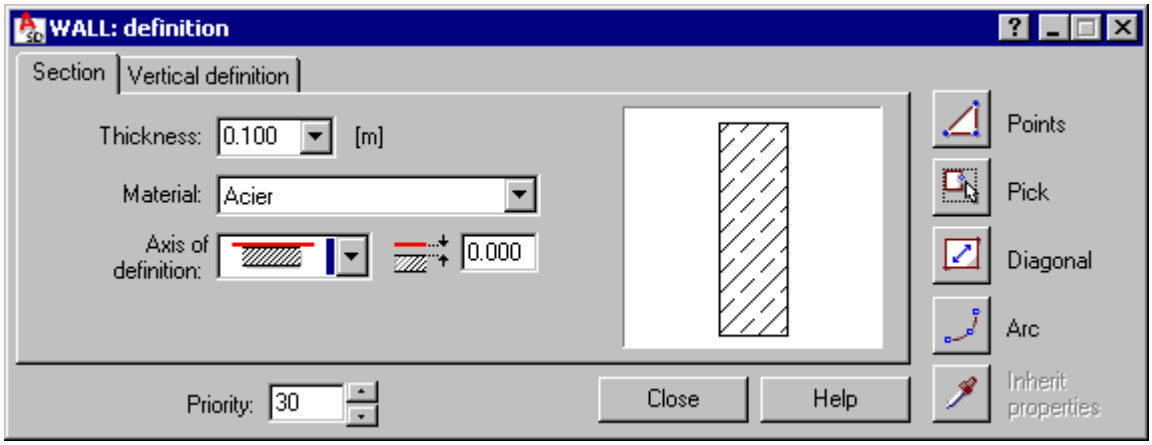
6.2. Walls

6.2.1. Wall - definition

The option enables definition of walls in a structure model. Walls are defined in 3D space, but to make work easier, individual walls are inserted in horizontal plans of a given story (in the top view).
A wall may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu selecting the option *Formwork Drawings / Define / Wall*
- the toolbar by pressing the icon 
- from the command line: RBCX_DEF_WALL.

After activating the *Wall definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition.

At the bottom of the dialog box is the Priority field which presents a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for a defined wall.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of a wall. NOTE: only after pressing one of these icons it is possible to define a wall in the graphic viewer of the program.

A wall may be defined as:

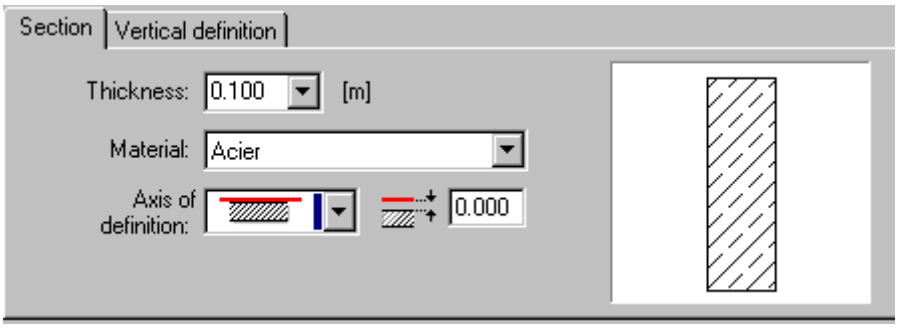
- rectilinear wall by defining the beginning and end points of a wall in a plan of a building story
- arc-shaped wall by defining three successive points of an arc in a plan of a building story (see also: Definition of parameters of arc elements – Job Preferences / Options)
- wall composed of segments – defined by indicating a diagonal; walls will be defined along the perimeter of the rectangle defined by the diagonal.

Moreover, there is also the *Select* icon available which, when pressed, enables turning indicated drawing objects (open polylines, arcs) into walls.

The icon located in the bottom right corner is used to inherit parameters from a wall defined earlier.

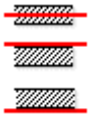
6.2.2. Wall - section

The *Wall: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



The following parameters should be determined for each type of the wall cross-section:

- wall thickness
- material the wall is made of
- axis of wall definition; the axis of wall definition may be assumed as:

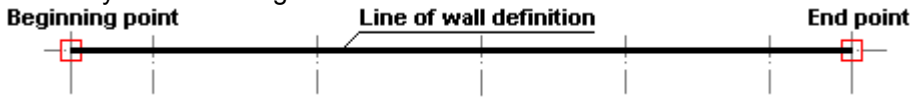


center line

upper line

lower line

The methods of definition using the center, upper and lower lines are illustrated schematically in the drawing below.



Center line is selected:

Upper line is selected:

Lower line is selected:

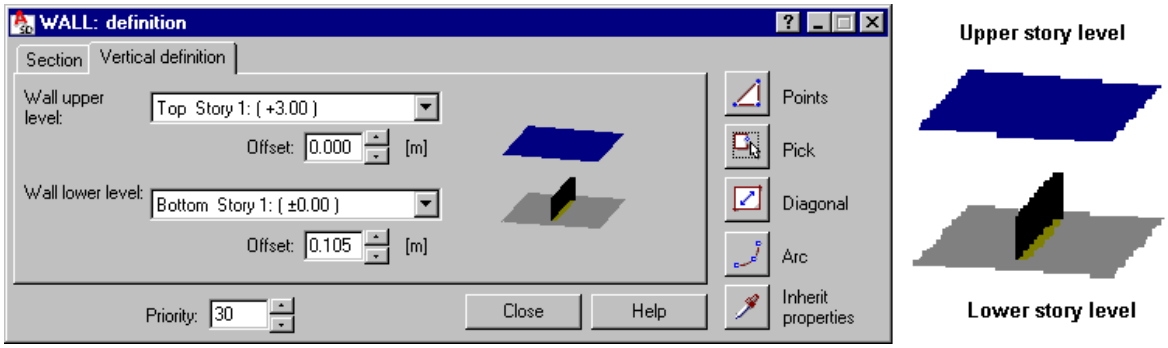
When defining a wall by means of the outer or inner line, the user may determine an additional parameter (an offset from the definition line) in the edit field next to the icon  ; if the offset equals zero, then the definition line coincides with the upper or lower line.

6.2.3. Wall - vertical definition

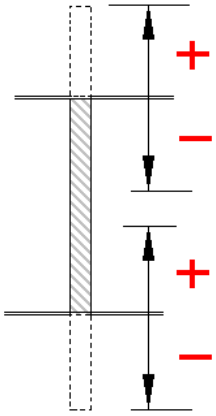
The *Wall: definition* dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.

The options in this dialog box allow selecting levels (upper and lower) of a defined wall.
The lists *Wall upper level* and *Wall lower level* include all the levels of an active story of a building defined so far; the vertical location of a wall in a building is defined by selecting a plane in which the top and bottom faces of the wall should be positioned (e.g. Top - Story 2, Bottom - Story 2).
When defining the vertical location of a wall, the user may define an additional parameter (an offset from the lower or upper level) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane. Pressing the  button allows selecting a horizontal element edge determining an offset value.

NOTE:
It is also possible to define a wall by attaching it only to e.g. the lower level of a story (then the upper and lower levels of the wall assume the value, e.g. Bottom - Story 2). The wall height is defined then by the offset value. Such a situation is presented in the drawing below.



The following sign convention for offset values has been adopted (it refers both to the lower and upper levels of a story):



6.2.4. Wall definition - command line

While defining a wall, the following parameters can be determined in the command line:

Points

Pick the beginning point:
Pick next point [Side / Offset]:

where:
Side - location of the definition axis of a wall (upper, lower, central)
Offset - distance from the lower or upper level of a wall
NOTE: When defining an element on the axis (axis location: central), offset and side are not supported.

Pick

Select object(s):
Finish [Side / Offset]:

where:
Side - location of the definition axis of a wall (upper, lower, central)
Offset - distance from the lower or upper level of a wall
NOTE: Lines, arcs, polyline do not need to be located in the same plane; while creating an element, it is projected on a specified plane(s).

Diagonal

First corner:
Second corner [Side / Offset]:

where:
Side - location of the definition axis of a wall (upper, lower, central)
Offset - distance from the lower or upper level of a wall

Arc

Pick first point:
Pick second point [Side / Offset]:
Pick third point [Side / Offset]:

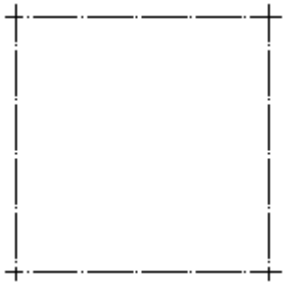
where:
Side - location of the definition axis of a wall (upper, lower, central)
Offset - distance from the lower or upper level of a wall

Inherit properties

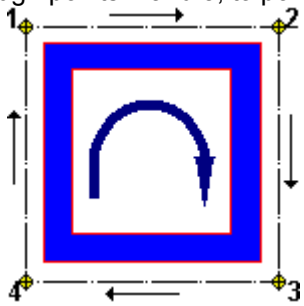
Select objects

6.2.5. Definition rule - offset different from zero

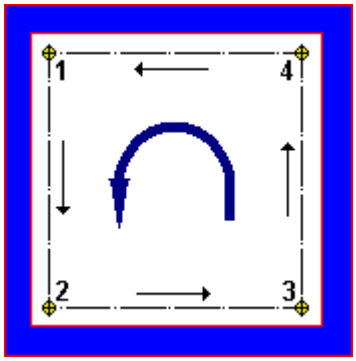
While defining a structure element, an offset value may differ from zero (the definition axis is positioned as: outer line or inner line). To illustrate the rule of definition of a structure element with an offset, such an element will be defined on the contour shown in the drawing below.



Walls, continuous footings, beams:
If the offset value is greater than zero (e.g. 30), and successive elements are defined clockwise, then the user obtains elements 'inside' the contour shown above – the drawing below (definition from point 1, through points 2 and 3, to point 4).

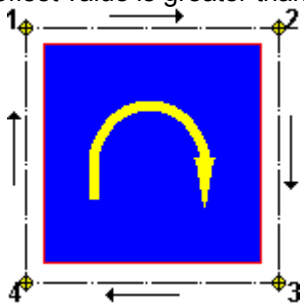


If the offset value is greater than zero (e.g. 30), and successive elements are defined counterclockwise, then the user obtains elements 'outside' the contour shown above – the drawing below (definition from point 1, through points 2 and 3, to point 4).

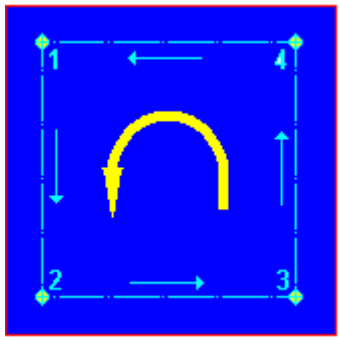


Slabs:

The offset value is greater than zero



The offset value is less than zero



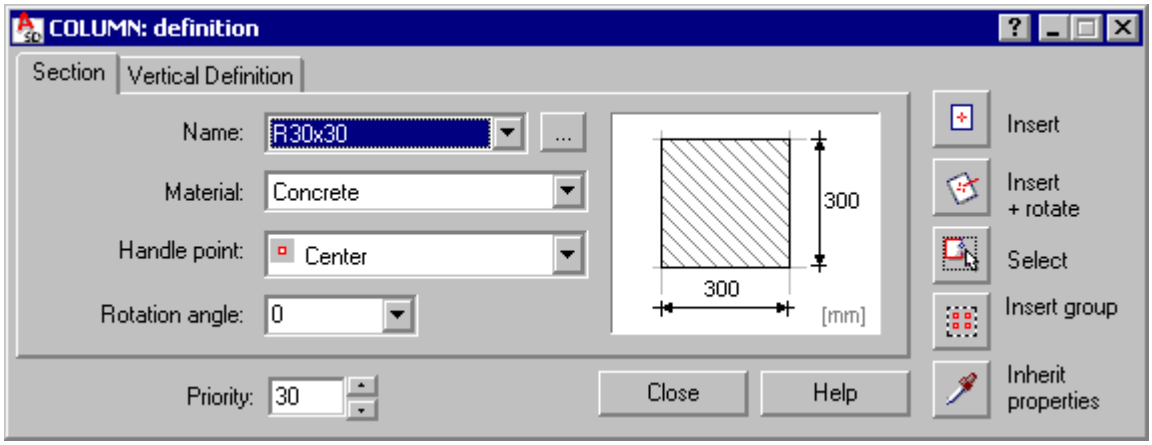
6.3. Columns

6.3.1. Column - definition

The option enables definition of columns in a structure model. Columns are defined in 3D space, but to make work easier, individual columns are inserted in horizontal plans of a given story (in the top view).
A column may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu selecting the option *Formwork Drawings / Define / Column*
- the toolbar by pressing the icon 
- from the command line: RBCX_DEF_COL.

After activating the *Column definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition.

At the bottom of the dialog box is the Priority field presenting a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for the defined column.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of the column. NOTE: only after pressing one of these icons it is possible to define the column in the graphic viewer of the program.

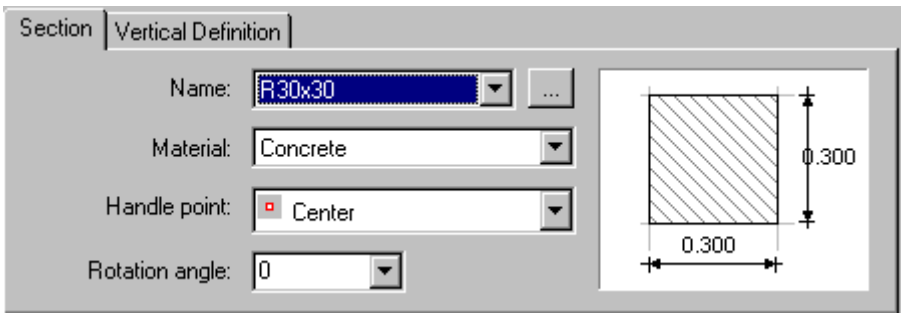
The column may be defined:

- by indicating a point in a plan of a building story with a value of the angle of section rotation chosen from the selection list
- by indicating a point in a plan of a building story and defining graphically the rotation angle of the column cross-section (the angle is defined by indicating a point in the direction of which the column will be 'facing')
- by indicating several points of a workframe; to do it, the user should select two points of a (rectangle-shaped) region - a column with the specified parameters will be defined at all workframe points positioned inside that region.

Moreover, there is also the *Select* icon available which, when pressed, enables turning indicated drawing objects (rectangle, circle, closed polyline) to a section and a column.
The icon located in the bottom right corner is used to inherit parameters from a column defined earlier.

6.3.2. Column - section

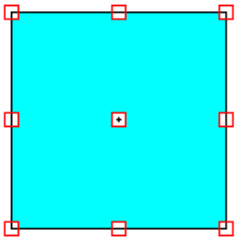
The *Column: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



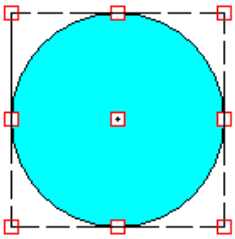
This dialog box allows defining parameters of a column cross-section. The following parameters should be determined for each type of the column cross-section:

- type of the column section by selecting the section's name from the list of available sections; pressing the (**...**) button located to the right of the selection list opens the ***Section list*** dialog box which allows adding a new section to the section list
- material the column is made of
- handle points of the column while defining it on the screen; handle points for individual types of the column-cross sections are presented in the drawings below

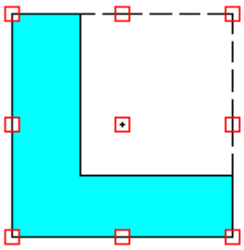
rectangular section



round section



L-shaped section



- rotation angle of the column cross-section; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of the rotation angle.

6.3.3. Column - vertical definition

The *Column: definition* dialog box looks as presented in the drawing below after selecting the *Vertical Definition* tab in it.

The options in this dialog box allow selecting levels (upper and lower) of a defined column.

The lists *Column upper level* and *Column lower level* include all levels of an active story of a building defined so far; the vertical location of the column in a building is defined by selecting a plane in which the top and bottom points of the column should be positioned (e.g. Top - Story 2, Bottom - Story 2).
When defining the vertical location of the column, the user may define an additional parameter (an offset from the lower or upper level) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane. Pressing the  button allows selecting a horizontal element edge determining an offset value.

NOTE:
It is also possible to define the column by attaching it only to e.g. the lower level of a story (then the upper and lower levels of the column assume the value, e.g. Bottom - Story 2). The column height is defined then by the offset value.

6.3.4. Column definition - command line

While defining a column, the following parameters can be determined in the command line:

Insert
Pick insertion point:

Insert + rotate
Pick insertion point:
Rotate column [Orientation]:

Pick
Select object(s):

NOTE: In the current program version, this command allows selecting only objects such as a circle and a rectangle.

Inherit properties
Select objects

6.4. Floor slabs / raft foundations

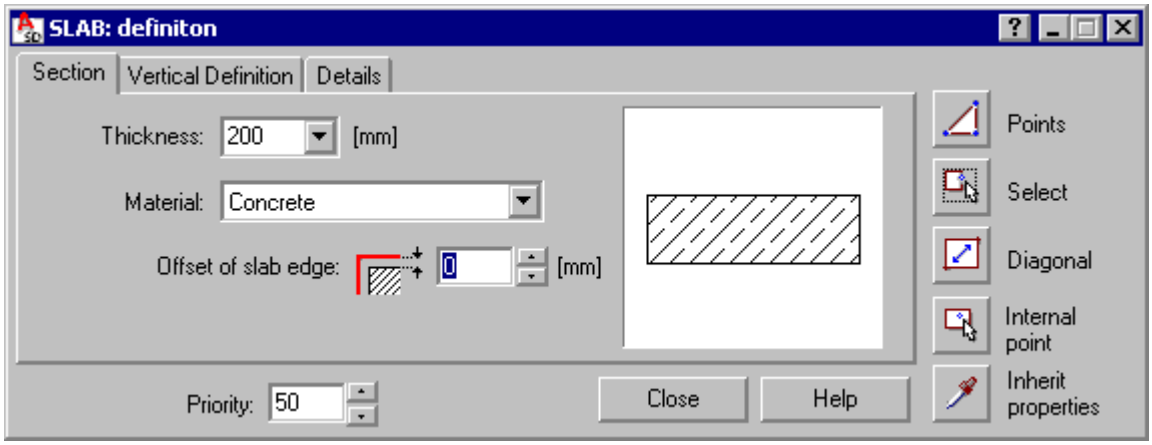
6.4.1. Floor slab / raft foundation - definition

The option enables definition of floor slabs and raft foundations in a structure model. Slabs and raft foundations are inserted in horizontal plans of an active story (in the top view).
A slab / raft foundation may be defined both in the 3D view and the 2D view.
Definition of the floor slab / raft foundation starts after:

- selecting the menu option *Formwork Drawings / Define / Slab* or *Formwork Drawings / Define / Raft foundation*
- pressing the icon  or 
- entering into the command line: RBCX_DEF_SLAB or RBCX_DEF_SLAB_FOUNDATION.

NOTE:
When copying a story, floor slabs are not copied to higher stories.

After activating the *Slab definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition
Details.

At the bottom of the dialog box is the Priority field which presents a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for a defined slab / raft foundation.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of the slab / raft foundation. NOTE: only after pressing one of these icons it is possible to define the slab / raft foundation in the graphic viewer of the program.

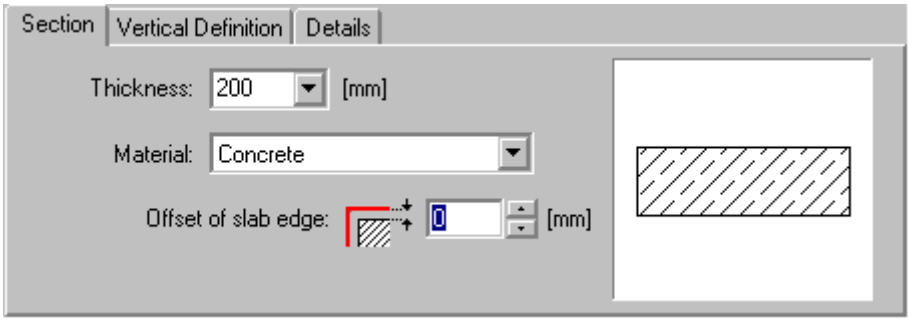
The slab / raft foundation may be defined:

- by giving successive points defining geometry of the slab / raft foundation
- by indicating a closed polyline that determines a contour of the slab / raft foundation
- by indicating a diagonal; the slab / raft foundation will be defined on a contour of the rectangle defined by the diagonal
- by indicating an internal point of a closed contour.

The icon located in the bottom right corner is used to inherit parameters from a slab defined earlier.

6.4.2. Slab - section

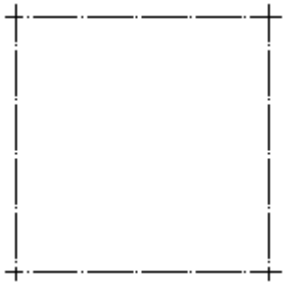
The *Slab: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



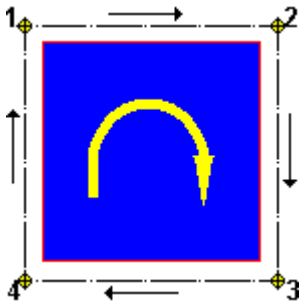
The following parameters should be determined for each type of the slab cross-section:

- slab thickness
- material the slab is made of

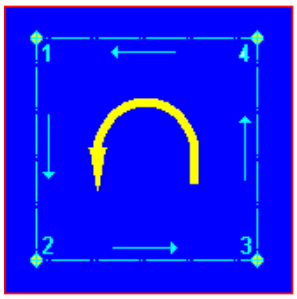
- offset of the slab edge; as an example of definition of the slab with the offset, the drawing below illustrates definition of the slab on the presented contour



1. slab offset is greater than zero and characteristic points of the slab are defined clockwise

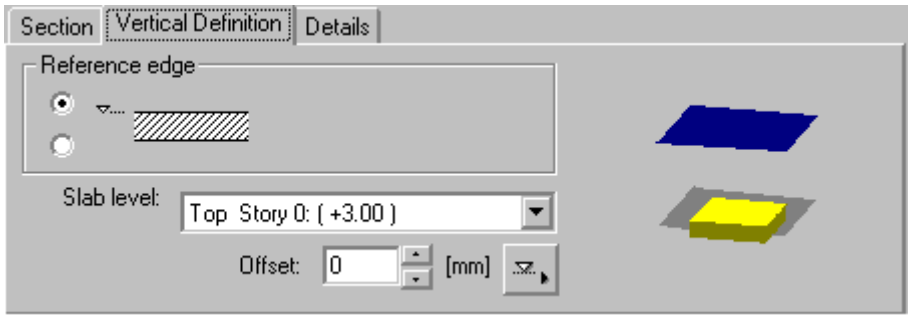


2. slab offset is greater than zero and characteristic points of the slab are defined counterclockwise



6.4.3. Slab - vertical definition

The *Slab: definition* dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



This dialog box is used to determine a location of a slab with respect to an active story.
The *Slab level* list includes all levels of an active story of a building defined so far; the location of a slab in the building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2). This list also contains all inclined planes defined for a selected story (see: Definition of a plane).
In addition, there is a possibility to define the reference edge of a slab with respect to the upper or lower level of a story. The following situations are possible:

Reference edge

1. 

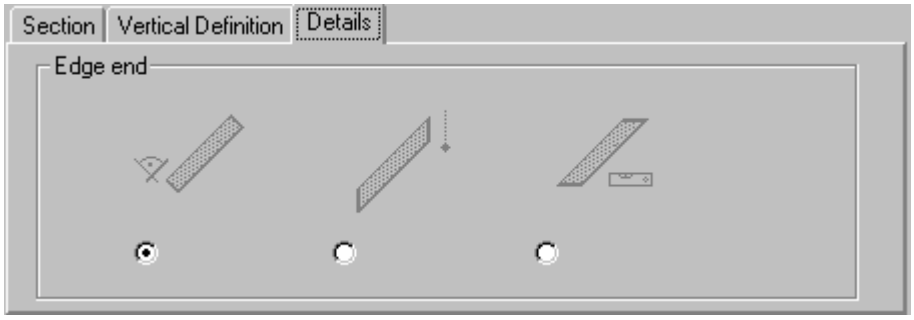
2. 

Location of the slab with respect to the story level

When defining the vertical location of a slab, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story. Pressing the  button allows selecting a horizontal element edge determining an offset value.

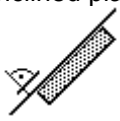
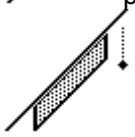
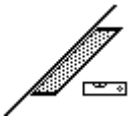
6.4.4. Slab - details

The *Slab: definition* dialog box looks as presented in the drawing below after selecting the *Details* tab in it.



NOTE:
Options on this tab are available only then, when an inclined plane has been chosen as a reference level in the dialog box on the Vertical Definition tab.

Options in the above dialog box allow selecting a type of the edge end of a slab lying on an inclined plane:

1.  perpendicularly to the edge
2.  vertically
3.  horizontally.

6.4.5. Slab definition - command line

While defining a slab or a raft foundation, the following parameters can be determined in the command line:

Points

Pick the beginning point:
Pick next point [sElection / Arc]:
Selection
Select edge

Arc
Pick second arc point
Pick third arc point

Select
Select object(s):
Finish [Side / Offset]:

where:
Side – location of the reference edge (upper, lower)
Offset - distance from the lower or upper level of a slab
NOTE: Lines, arcs, polyline do not need to be located in the same plane; while creating an element, it is projected on a specified plane(s).

Diagonal
First corner:
Second corner [Side / Offset]:

where:
Side – location of the reference edge (upper, lower)
Offset - distance from the lower or upper level of a slab

Internal point
Select contour internal point

Inherit properties
Select objects

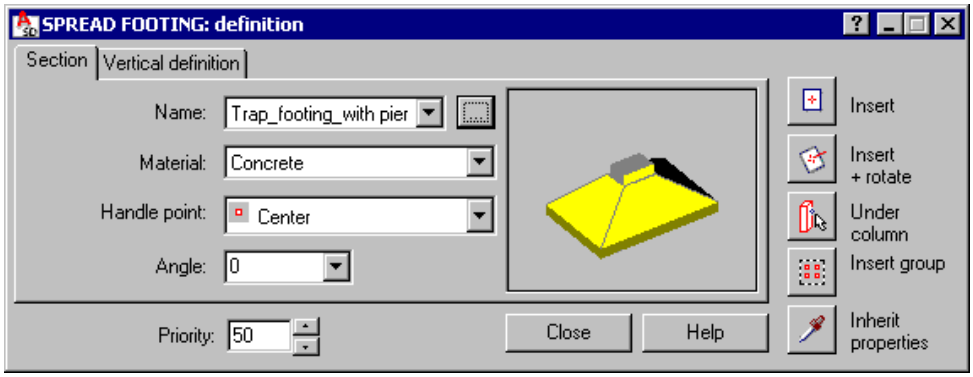
6.5. Spread footings

6.5.1. Spread footing - definition

The option enables definition of spread footings in a structure model. Spread footings are defined in 3D space, but to make work easier, individual spread footings are inserted in horizontal plans of a given story (in the top view). In the program, spread footings are prefabricated elements (volumetric elements of complex geometry).
A spread footing may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu selecting the option Formwork Drawings / Define / Spread footing
- the toolbar by pressing the icon
- from the command line: RBCX_DEF_FOOT.

After activating the *Spread footing definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition.

At the bottom of the dialog box is the Priority field which presents a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for a defined spread footing.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of the spread footing. NOTE: only after pressing one of these icons it is possible to define the spread footing in the graphic viewer of the program.

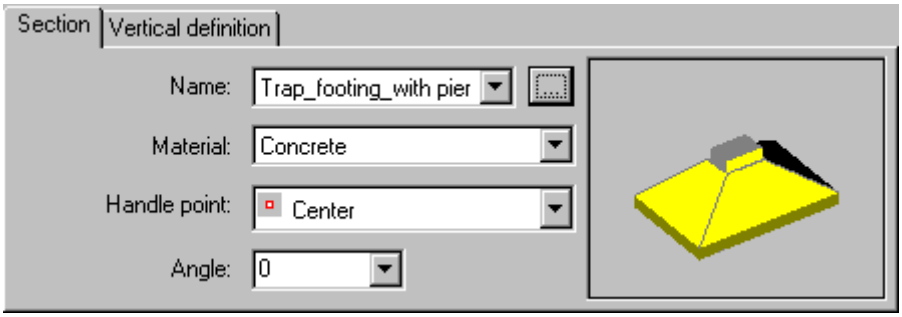
The spread footing may be defined in the following ways:

- by indicating a point in a plan of a building story with a value of the angle of section rotation chosen from the selection list
- by indicating a point in a plan of a building story and defining graphically the rotation of the spread footing with respect to the insertion point (the angle is defined by indicating a point in the direction of which the spread footing will be 'facing').
- by defining the spread footing under the column
- by indicating several points of a workframe; to do it, the user should select two points of a (rectangle-shaped) region - a spread footing with the specified parameters will be defined at all workframe points positioned inside that region.

The icon located in the bottom right corner is used to inherit parameters from a spread footing defined earlier.

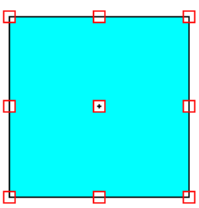
6.5.2. Spread footing - section

The *Spread footing: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.

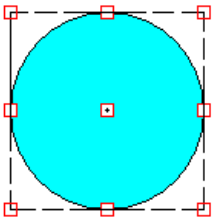


This dialog box allows defining shape parameters of a solid (spread footing). The following parameters should be determined for each type of the spread footing cross-section:

- type of the spread footing section by selecting a solid name from the list of available solids; pressing the (**...**) button located to the right of the selection list opens the dialog box which allows adding a new spread footing type to the list of available spread footings
- material the spread footing is made of
- handle points of the spread footing while defining it on the screen; handle points for individual types of the spread footing cross sections are presented in the drawings below
  rectangular section



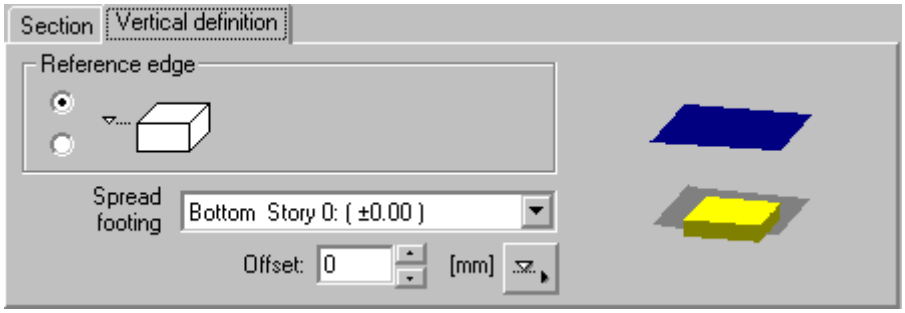
round section



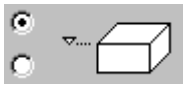
- angle of rotation of the spread footing cross-section; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of the rotation angle.

6.5.3. Spread footing - vertical definition

The *Spread footing: definition* dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



This dialog box is used to determine a location of a spread footing with respect to an active story.
The *Spread footing* list includes all the levels of an active story of a building defined so far; the location of a spread footing in a building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2).
In addition, there is a possibility to define the reference edge of a spread footing with respect to the upper or lower level of a story. The following situations are possible:

Reference edge	**Location of a spread footing with respect to a story level**
1. 	
2. 	

When defining the vertical location of a spread footing, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story.
Pressing the  button allows selecting a horizontal element edge determining an offset value.

6.5.4. Spread footing definition - command line

While defining a spread footing, the following parameters can be determined in the command line:

Insert

Pick insertion point:

Insert + rotate
Pick insertion point:
Rotate spread footing [Orientation]:

Under column
Select column(s):

NOTE: The command works both for a single column and for several selected columns.
NOTE: Orientation of spread footings should be compatible with orientation of columns.

Inherit properties
Select objects

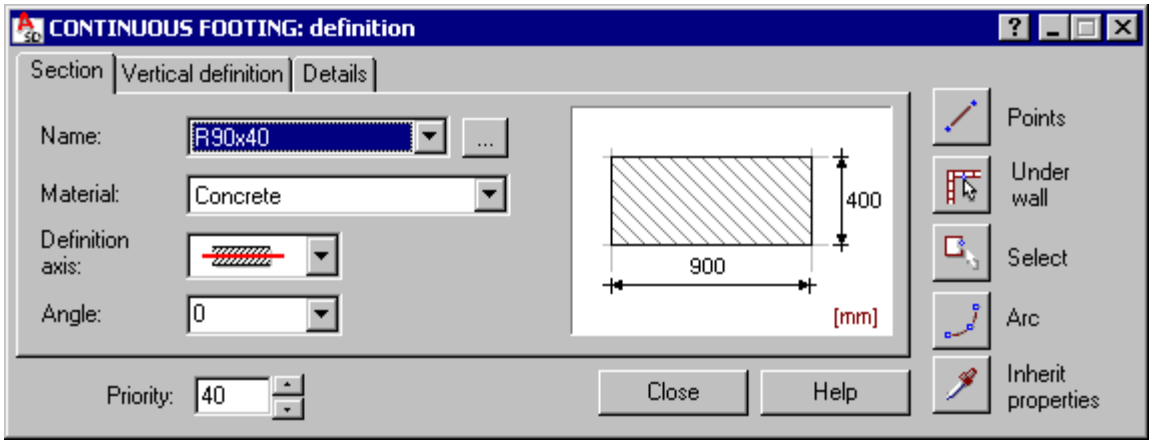
6.6. Continuous footings

6.6.1. Continuous footing - definition

The option enables definition of continuous footings in a structure model. Continuous footings are inserted in horizontal plans of an active story (in the top view).
A continuous footing may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu selecting the option Formwork Drawings / Define / Continuous footing
- the toolbar by pressing the icon
- from the command line: RBCX_DEF_CONT_FOOT.

After activating the *Continuous footing definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition
Details.

At the bottom of the dialog box is the Priority field which presents a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for the defined continuous footing.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of the continuous footing. NOTE: only after pressing one of these icons it is possible to define a continuous footing in the graphic viewer of the program.

The continuous footing may be defined as:

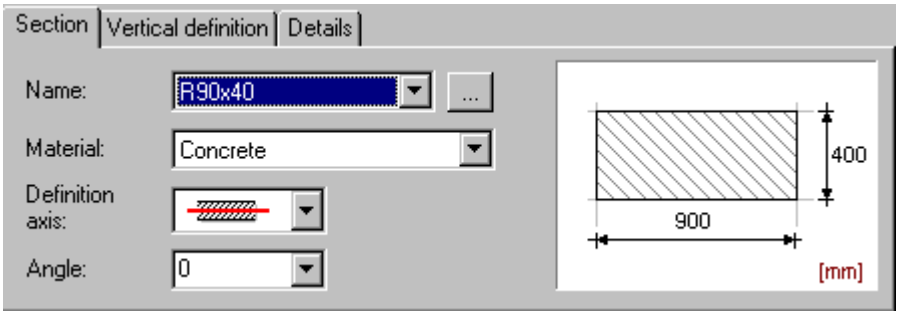
- rectilinear continuous footing by defining the beginning and end points of the continuous footing in a plan of a building story
- curved continuous footing by defining three successive points of an arc belonging to the continuous footing in a plan of a building story
- continuous footing under an already-defined wall.

Moreover, there is also the *Select* icon available which, when pressed, enables turning indicated arcs and open polylines to continuous footings.

The icon located in the bottom right corner is used to inherit parameters from a continuous footing defined earlier.

6.6.2. Continuous footing - section

The *Continuous footing: definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



This dialog box allows defining parameters of a cross-section of a continuous footing. The following parameters should be determined for each type of the continuous footing cross-section:

- continuous footing section by selecting the section's name from the list of available sections; pressing the (**...**) button located to the right of the selection list opens the ***Section list*** dialog box which allows adding a new section to the section list
- material the continuous footing is made of
- axis of definition of the continuous footing (beam); the axis of continuous footing definition may be assumed as:

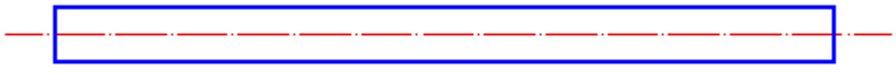
  center line

  upper line

  lower line

  The methods of definition using the center, upper and lower lines are illustrated schematically in the drawings below.

  Center line is selected:

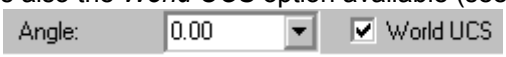


  Upper line is selected:

  Lower line is selected:

When defining the continuous footing by means of the outer or inner line, the user may determine an additional parameter (an offset from the definition line) in the edit field next to the icon  ; if the offset equals zero, then the definition line coincides with the upper or lower line.

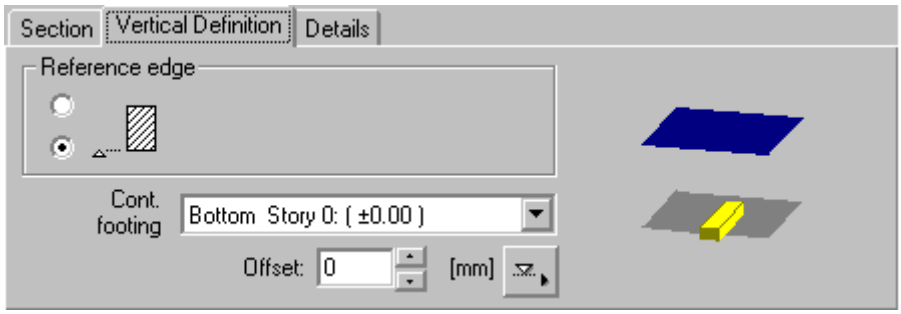
- angle of rotation of the continuous footing cross-section; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of a rotation angle. NOTE: While defining a continuous footing lying on an inclined plane, there is also the *World UCS* option available (see the drawing below).



The option makes it easier to define the orientation of a continuous footing cross-section with respect to the global coordinate system. Examples of application of this option are given in the topic dedicated to the beam definition.

6.6.3. Continuous footing - vertical definition

The *Continuous footing: definition* dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



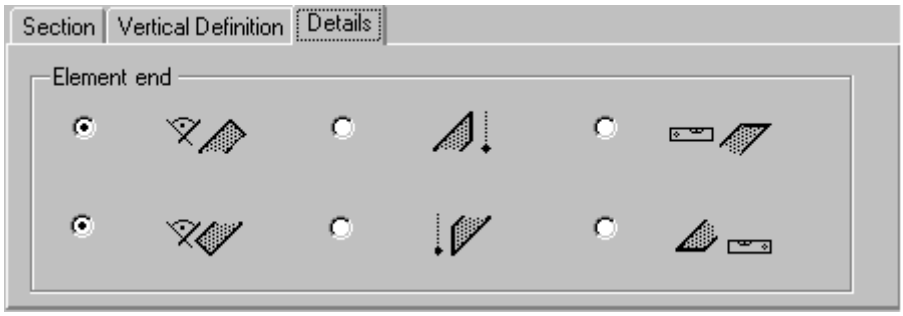
This dialog box is used to determine a location of a continuous footing with respect to an active story.
The *Cont. footing* list includes all levels of an active story of a building defined so far; the location of the continuous footing in a building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2). This list also contains all inclined planes defined for a selected story (see: Definition of a plane).
In addition, there is a possibility to define the reference edge of the continuous footing with respect to the upper or lower level of a story. The following situations are possible:

Reference edge	**Location of the continuous footing with respect to the story level**
1. 	
2. 	

When defining the vertical location of the continuous footing, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story.
Pressing the  button allows selecting a horizontal element edge determining an offset value.

6.6.4. Continuous footing - details

The ***Continuous footing: definition*** dialog box looks as presented in the drawing below after selecting the *Details* tab in it.



NOTE:
Options on this tab are available only then, when an inclined plane has been chosen as a reference level in the dialog box on the Vertical Definition tab.

Options in the above dialog box allow selecting a type of the edge end of the continuous footing positioned on an inclined plane (at the beginning as well as at the end of the continuous footing):

1. perpendicularly to the edge: or
2. vertically: or
3. horizontally: or .

6.6.5. Continuous footing definition - command line

While defining a continuous footing, the following parameters can be determined in the command line:

Points

Pick the beginning point:
Pick next point [Side / Offset]:

where:
Side - location of the definition axis of a continuous footing (upper, lower, central)
Offset - distance from the lower or upper level of a continuous footingNOTE: When defining an element on the axis (axis location: central), offset and side are not supported.

Under wall

Select wall(s):

NOTE: The command works both for a single wall and several selected walls.

Select

Select object(s):
Finish [Side / Offset]:

where:
Side - location of the definition axis of a continuous footing (upper, lower, central)
Offset - distance from the lower or upper level of a continuous footing
NOTE: Lines, arcs, polyline do not need to be located in the same plane; while creating an element, it is projected on a specified plane(s).

Arc

Pick first point:
Pick second point [Side / Offset]:
Pick third point [Side / Offset]:

where:
Side - location of the definition axis of a continuous footing (upper, lower, central)
Offset - distance from the lower or upper level of a continuous footing

Inherit properties

Select object

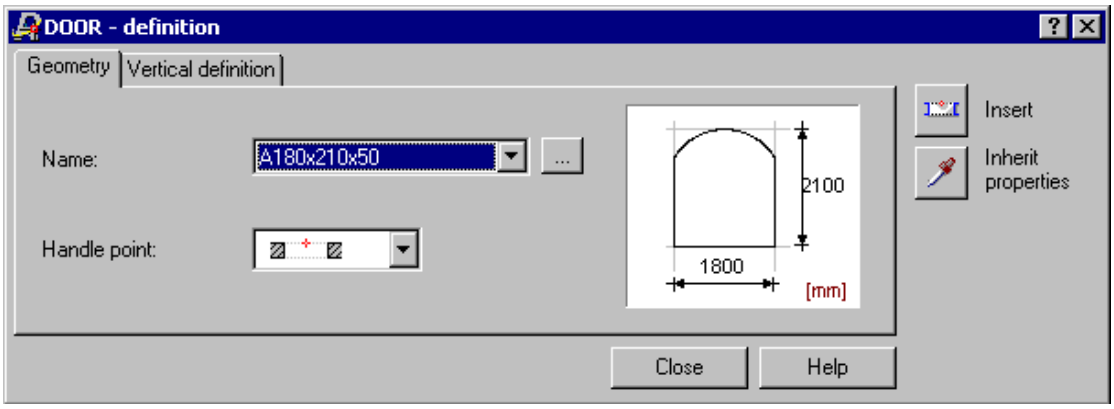
6.7. Doors

6.7.1. Door - definition

The option enables definition of doors in walls of a structure model. Doors are defined in 3D space, but to make work easier, individual door openings are inserted in horizontal plans of a given story (in the top view).
A door may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu by selecting the option Formwork Drawings / Define / Doors
- the toolbar by pressing the icon
- from the command line: RBCX_DEF_DOOR.

After activating the *Door definition* option and indicating a wall in a structure model, the dialog box shown in the drawing below appears on the screen.



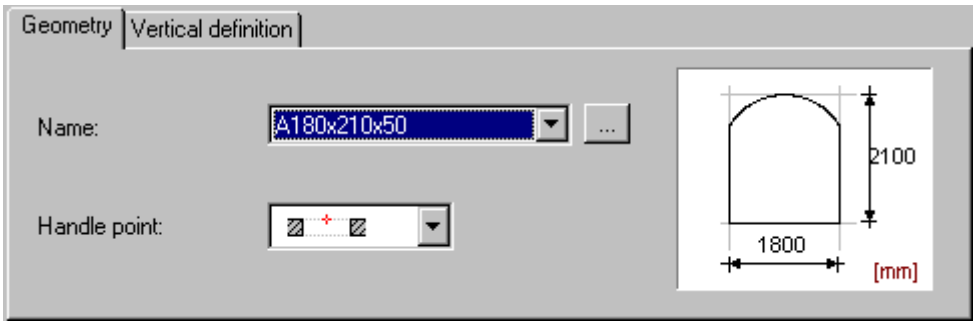
The dialog box is composed of the following tabs:
Geometry
Vertical definition.

The door may be defined after pressing the *Insert* icon located in the right-hand part of the dialog box. After switching to the graphic viewer the location of the door in a selected wall is displayed on an ongoing basis.

The icon located in the bottom right corner is used to inherit parameters from a door defined earlier.

6.7.2. Door - geometry

The *Door: definition* dialog box looks as presented in the drawing below after selecting the *Geometry* tab in it.

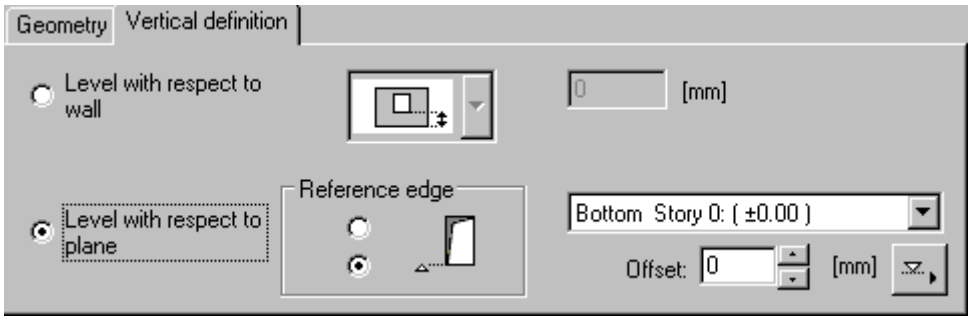


While defining geometry of a door, the following parameters should be determined:

- type of shape of a door opening by selecting the section's name from the list of available shapes; pressing the (**...**) button located to the right of the selection list opens the ***Door list*** dialog box which allows adding a new shape to the list of shapes
- door handle point (it is a point within an opening in relation to which the door opening in a wall be inserted); the door handle point may be positioned:
  – on the left
  – in the middle
  – on the right.

6.7.3. Door - vertical definition

The ***Door - definition*** dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



There are two methods of defining a vertical location of a door in the program:

- by defining the level with respect to the wall
  level (location) of a door opening along the wall height (the Z coordinate); two possibilities are available (a default value is 0.0 from the lower wall level):
  - location of an opening with respect to the lower wall level measured to the lower part of the door
  - location of an opening with respect to the lower wall level measured to the upper part of the door

- by defining the level with respect to the plane
  level (location) of a door opening is determined with respect to a plane (any plane defined earlier in a structure model) selected from the drop-down list of planes; additionally, the following parameters can be determined:
  - reference edge of a door with respect to the upper or lower level
  - value of the offset from the lower or upper story level in the *Offset* edit field.
  This method of defining a door allows the user to determine a door location with respect to the plane, e.g. of the floor (and not of the story level).

6.7.4. Door definition - command line

While defining a door, the following parameters can be determined in the command line:

Insert

Select objects:
Define opening position [Reference point / Attachment]:

Reference point
Indicate new reference point or [Specify distance]

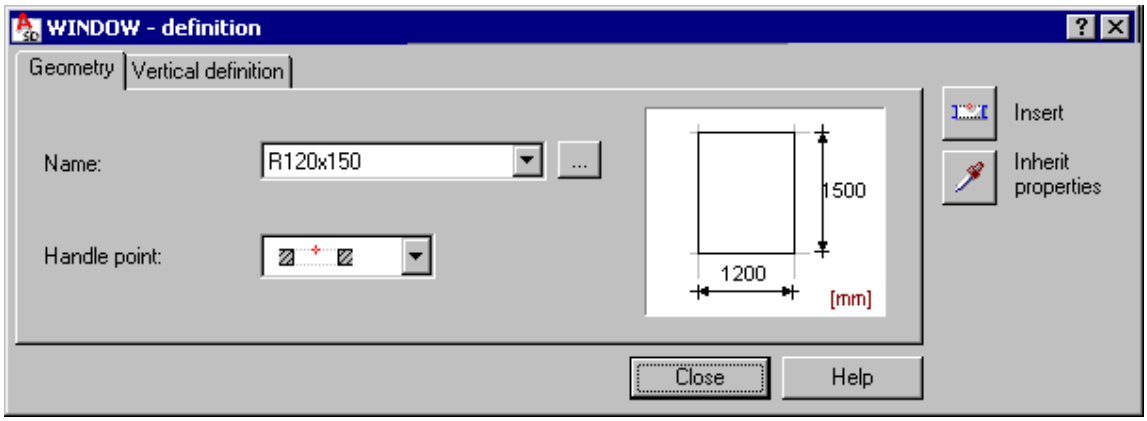
6.8. Windows

6.8.1. Window - definition

The option enables definition of windows in walls of a structure model. Windows are defined in 3D space, but to make work easier, individual window openings are inserted in horizontal plans of a given story (in the top view).
Windows may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu by selecting the option *Formwork Drawings / Define / Window*
- the toolbar by pressing the icon
- from the command line: RBCX_DEF_WINDOW.

After activating the *Window definition* option and indicating a wall in a structure model, the dialog box shown in the drawing below appears on the screen.



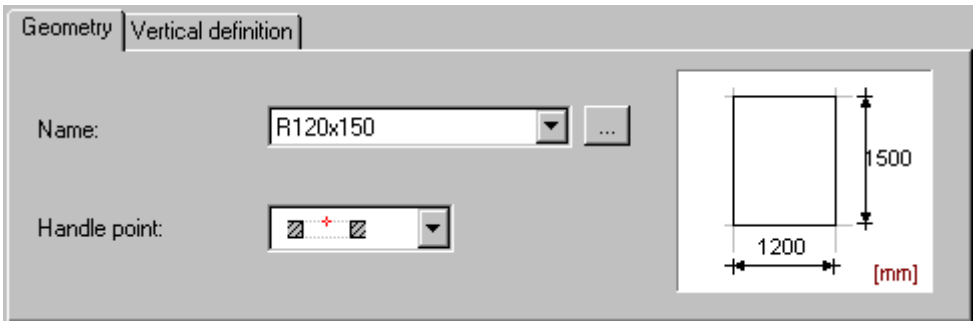
The dialog box is composed of the following two tabs:
Geometry
Vertical definition.

A window may be defined after pressing the *Insert* icon located in the right-hand part of the dialog box. After switching to the graphic viewer the position of the window in a selected wall is displayed on an ongoing basis.

The icon located in the bottom right corner is used to inherit parameters from a window defined earlier.

6.8.2. Window - geometry

The *Window: definition* dialog box looks as presented in the drawing below after selecting the *Geometry* tab in it.

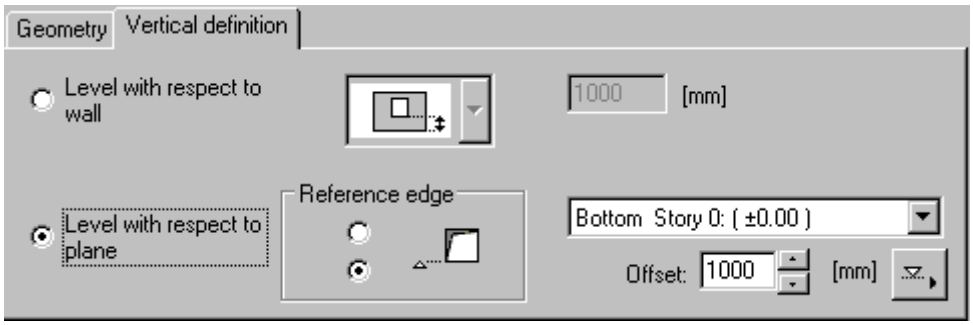


While defining geometry of a window, the following parameters should be determined:

- type of a window opening shape by selecting the shape name from the list of available shapes; pressing the (**...**) button located to the right of the selection list opens the ***Window list*** dialog box which allows adding a new shape to the list of shapes
- window handle point (it is a point within an opening in relation to which the window opening in a wall will be inserted); the window handle point may be positioned:
  - – on the left
  - – in the middle
  - – on the right.

6.8.3. Window - vertical definition

The ***Window - definition*** dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



There are two methods of defining a vertical location of a window in the program:

- by defining the level with respect to the wall
  level (location) of a window opening along the wall height (the Z coordinate); two possibilities are available (a default value is 100.0 cm from the floor level):

  – location of an opening with respect to the lower wall level measured to the lower part of the window

  - location of an opening with respect to the lower wall level measured to the upper part of the window

- by defining the level with respect to the plane
  level (location) of a window opening is determined with respect to a plane (any plane defined earlier in a structure model) selected from the drop-down list of planes; additionally, the following parameters can be determined:
  - reference edge of a window with respect to the upper or lower level
  - value of the offset from the lower or upper story level in the *Offset* edit field.
  This method of defining a window allows the user to determine a window location with respect to the plane, e.g. of the floor (and not of the story level).

6.8.4. Window definition - command line

While defining a window, the following parameters can be determined in the command line:

Insert

Select objects:
Define opening location [Reference point / Attachment]:

Reference point
Indicate new reference point or [Specify distance]

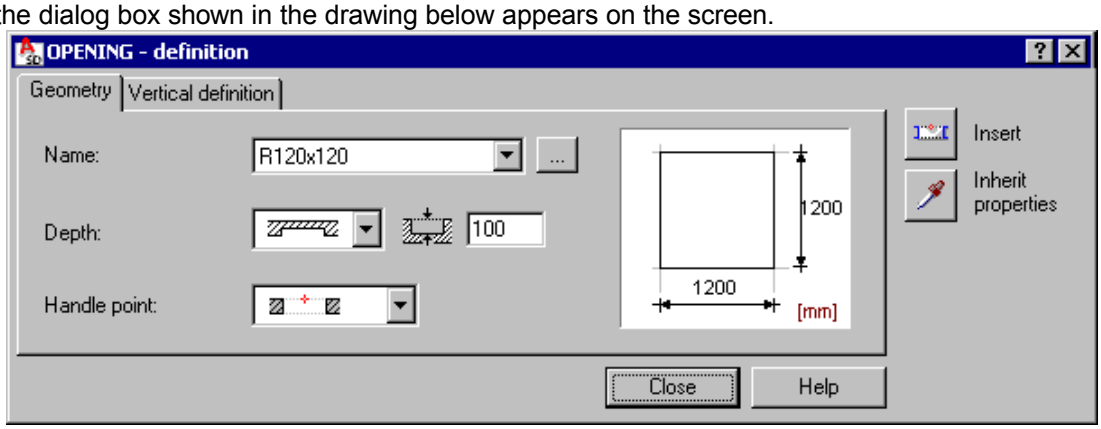
6.9. Openings/recesses - walls

6.9.1. Opening / recess - definition

The option enables definition of openings or recesses in walls of a structure model. Openings or recesses are defined in 3D space, but to make work easier, individual openings are inserted in horizontal plans of a given story (in the top view).
Note should be taken here that if an opening is defined in the wall which touches another element of a structure model, then it is possible to define the opening in such a way so that it is also positioned in the structure element touching the wall.
Openings may be defined both in the 3D view and the 2D view.
The option is accessible from:

- the menu by selecting the option Formwork Drawings / Define / Opening/recess in wall
- the toolbar by pressing the icon
- from the command line: RBCX_DEF_HOLE_WALL.

After activating the *Opening/recess definition* option and indicating a wall in a structure model, the dialog box shown in the drawing below appears on the screen.

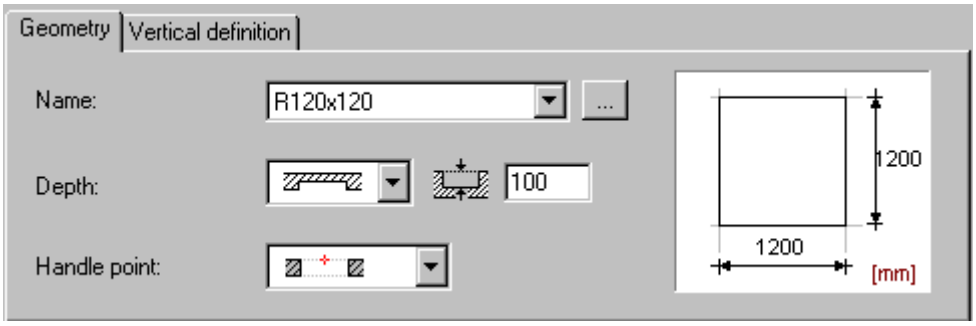


The dialog box consists of the following tabs:
Geometry
Vertical definition.

An opening/recess may be defined after pressing the *Insert* icon located in the right-hand part of the dialog box. After switching to the graphic viewer the location of the opening in a selected wall is displayed on an ongoing basis.

6.9.2. Opening / recess - geometry

The *Opening / recess: definition* dialog box looks as presented in the drawing below after selecting the *Geometry* tab in it.



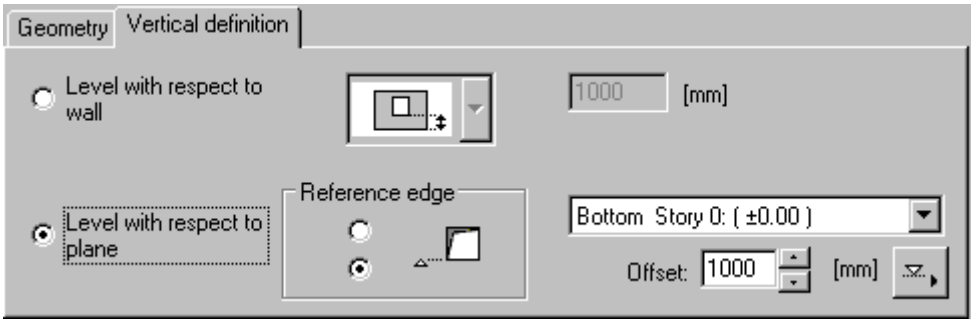
While defining geometry of an opening, the following parameters should be determined:

- type of an opening shape by selecting the shape name from the list of available shapes; pressing the (**...**) button located to the right of the selection list opens the ***Opening list*** dialog box which allows adding a new shape to the list of shapes
- depth if a recess is selected; the selection list includes the following possibilities:
  – opening in a wall
  or – recess in a wall
  when a recess is selected, an edit field for defining the recess depth is available.
  The method of recess definition
  Once geometry of a recess is defined (it does not matter which recess type has been selected), indicate the wall edge in which the recess will be cut out - see the drawing below.

- opening handle point (it is a point within an opening in relation to which the opening in a wall be inserted); the opening handle point may be positioned:
  – on the left
  – in the middle
  – on the right.

6.9.3. Opening/recess - vertical definition

The ***Opening / recess - definition*** dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



There are two methods of defining a vertical location of a opening/recess in the program:

- by defining the level with respect to the wall
  opening level (location); two possibilities are available (a default value is 100.0 cm from the lower wall level):
  - location of an opening with respect to the lower wall level measured to the lower part of an opening

 - location of an opening with respect to the lower wall level measured to the upper part of an opening

- by defining the level with respect to the plane
  level (location) of an opening is determined with respect to a plane (any plane defined earlier in a structure model) selected from the drop-down list of planes; additionally, the following parameters can be determined:
  - reference edge of a window with respect to the upper or lower level
  - value of the offset from the lower or upper story level in the *Offset* edit field.
  This method of defining an opening/recess allows the user to determine an opening location with respect to the plane, e.g. of the floor (and not of the story level).

6.9.4. Opening definition - command line

While defining openings, the following parameters can be determined in the command line:

OPENING / RECESS IN WALL

Insert

Select wall:
Define opening location [Reference point / Attachment]:

Reference point
Indicate new reference point or [Specify distance]

OPENING / RECESS IN SLAB

Insert

Select slab:
Indicate location of the opening or [Reference point]:
Reference point
Indicate new reference point or [Specify distance]

Insert + rotate

Select slab:
Indicate location of the opening or [Reference point]:

Reference point
Indicate reference point / Define opening location

Define opening rotation [Orientation]:

Cut

Select slab:
Pick the opening contour or define the opening contour as [Diagonal / pOints]:

Diagonal
First corner
Second corner

Points
Pick first point [Arc]:
Pick next point [Arc]:

Arc
Pick second point of the arc [Line]

Pick third point of the arc

Inherit properties
Select objects

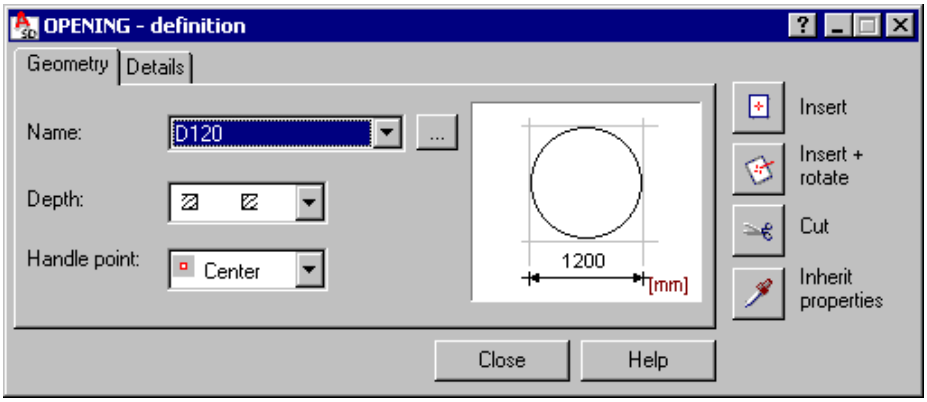
6.10. Openings/recesses - slabs

6.10.1. Opening / recess - definition (slab)

The option enables definition of openings or recesses in slabs of a structure model. Openings may be defined both in the 3D view and the 2D view.
The option is available from:

- the menu selecting the option Formwork Drawings / Define / Opening/recess in slab
- the toolbar by pressing the icon
- the command line: RBCX_DEF_HOLE_SLAB.

After activating the *Definition* of *opening/recess in slab* option and indicating a slab in a structure model, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Geometry
Details.

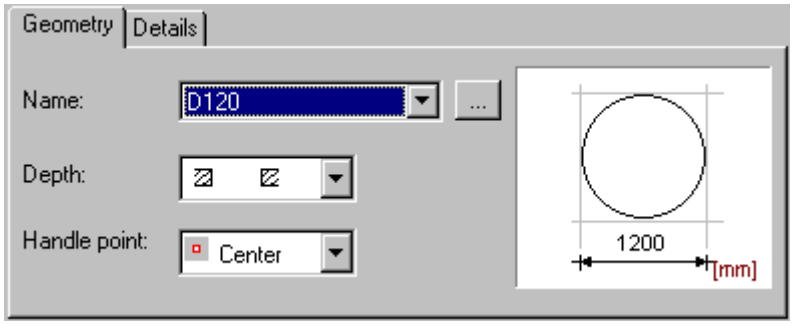
An opening/recess in a slab may be defined after pressing one of the icons:

- indicating a point that determines the location of an opening in a slab
- indicating a point that determines the location of an opening in a slab and a rotation angle of an opening
- defining an arbitrary shape of an opening in a slab.

The icon located in the bottom right corner is used to inherit parameters from an opening defined earlier.

6.10.2. Opening/recess - geometry (slab)

The ***Opening / recess (slab)*** dialog box looks as presented in the drawing below after selecting the *Geometry* tab in it.



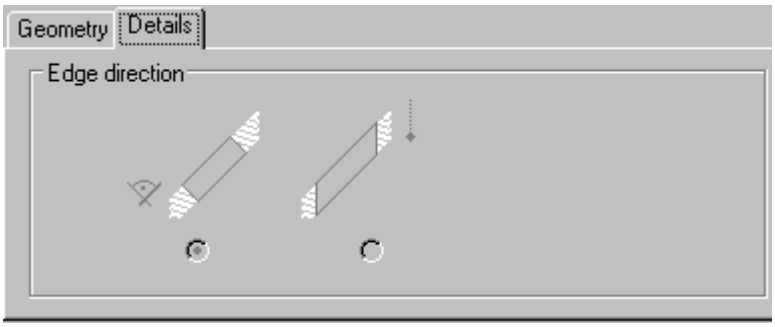
While defining geometry of an opening, the following parameters should be determined:

- type of an opening shape by selecting the shape name from the list of available shapes; pressing the (**...**) button located to the right of the selection list opens the ***Opening list*** dialog box which allows adding a new shape to the list of shapes
- depth if a recess is selected; the selection list includes the following possibilities:
  - opening in a slab
  - or - recess in a slab

  when a recess is selected, an edit field for defining the recess depth is available
- opening handle point (this is a point within an opening in relation to which the opening in a slab will be inserted); the opening handle point may be assumed as:
  - upper left
  - upper center
  - upper right
  - center left
  - center
  - center right
  - lower left
  - lower center
  - lower right.

6.10.3. Opening/recess - details (slab)

The ***Opening / recess(slab)*** dialog box looks as presented in the drawing below after selecting the *Details* tab in it.



NOTE:
The options provided on this tab are available only for inclined slabs.

Options in the above dialog box allow selecting a method of defining an opening (direction of the opening edge) with respect to the slab geometry:
- perpendicularly to the slab plane
- vertically (according to the Z axis of the global coordinate system).

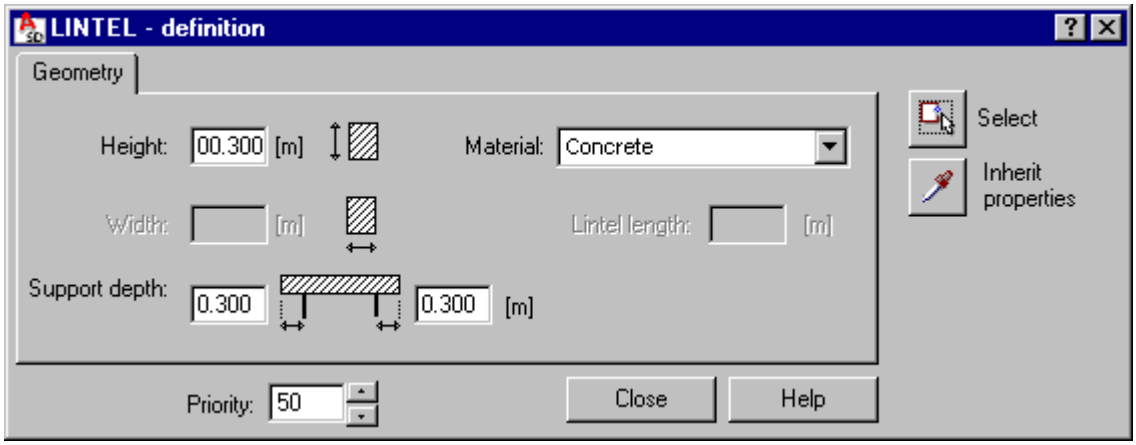
6.11.Lintels

6.11.1. Lintel - definition

The option is used to define lintels. Lintels may be defined above door or window openings as well as niches and recesses in the wall, both in the 3D view and in 2D views.
The option is accessible from:

- the menu by selecting the option Formwork Drawings / Define / Lintel
- the toolbar by pressing the icon
- the command line: RBCX_DEF_LINTEL.

After activating the *Lintel definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box includes the *Geometry* tab where the following lintel parameters may be defined:
- lintel height (height of the lintel cross-section)
- material, the lintel is made of
- lintel support depth on both sides of the door or window, recess, niche.

The lintel length is determined automatically based on the opening geometry and a value of the support depth.

Once lintel parameters are determined, the user may begin definition of the lintel in elements of a structure model. To define the lintel, the user should:

- press the *Select* icon provided in the right-hand part of the dialog box
- in a structure model, indicate objects (door or window openings, recesses, niches), above which lintels should be defined.

The icon located in the bottom right corner is used to inherit parameters from a lintel defined earlier.
At the bottom of the dialog box is the *Priority* field presenting a default value of a priority which allows managing interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for a defined lintel.

Command line:

Select
Indicate opening in the wall

6.12. Prefabricated elements

6.12.1. Prefabricated element / stairs - definition

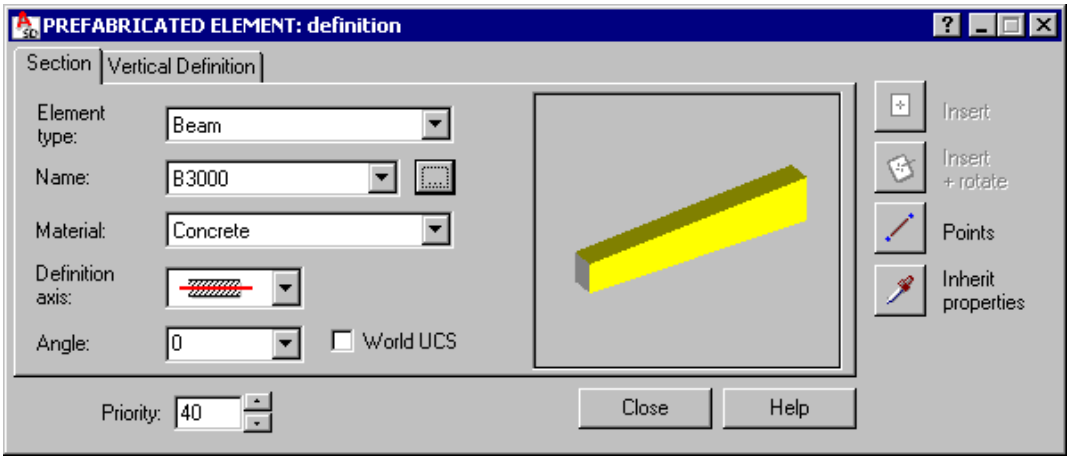
The dialog box which is used to define the following objects in a structure model:
- stairs
- prefabricated elements.

6.12.2. Prefabricated element - definition

The option enables definition of a prefabricated element in a structure model. Prefabricated elements are volumetric elements of complex geometry (spread footings, stairs, beams, columns, etc.).
Definition of a prefabricated element may start after:

- selecting the menu option Formwork Drawings / Define / Prefabricated element
- pressing the icon
- entering into the command line: RBCX_DEF_PREF.

After activating the *Prefabricated element definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical definition.

At the bottom of the dialog box is the Priority field presenting a default value of a priority which allows steering interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for a defined prefabricated element.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of a prefabricated element. NOTE: only after pressing one of these icons it is possible to define a prefabricated element in the graphic viewer of the program.

Definition of a prefabricated element depends on a selected type of a structure element (beam, column, slab, etc.) Not all the icons provided in the right part of the dialog box are available for individual types of structure elements; a prefabricated element may be defined:

for walls, beams, ground beams and continuous footings:

- by defining the beginning and end points of a rectilinear prefabricated element in a plan of a building story

for columns, slabs and raft foundations:

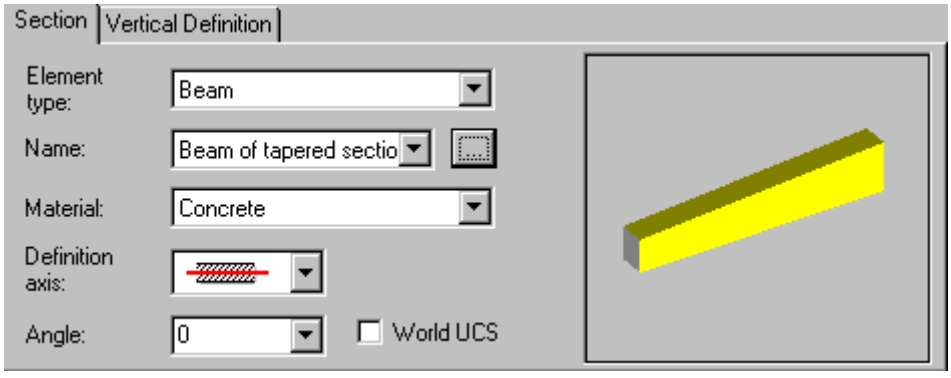
- by indicating a point in a plan of a building story with a value of the rotation angle of the column section or a slab rotation chosen from the selection list
- by indicating a point in a plan of a building story and defining graphically the rotation angle of the column cross-section or a slab rotation (the angle is defined by indicating a point in the direction of which the column section or a slab will be 'facing').

NOTE:
A prefabricated wall and a prefabricated column are 'attached' to the top or bottom story (these are not elements 'stretched' between the stories); in consequence, when the height of a story changes, the height of a prefabricated column or a prefabricated wall WILL NOT BE modified automatically (the prefabricated column or wall will not be adjusted to the change of the story height).

The icon located in the bottom right corner is used to inherit parameters from a prefabricated element defined earlier.

6.12.3. Prefabricated element - section

The ***Prefabricated element: definition*** dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



This dialog box allows defining parameters of the cross-section of a prefabricated element. The following parameters should be determined for each type of the cross-section:

- prefabricated element type; the following elements are available:
  beam
  column
  wall

continuous footing
slab / raft foundation
ground beam

- type of a prefabricated element section, by selecting a name of a solid from the list of available solids; pressing the (**...**) button located to the right of the selection list opens the dialog box which allows adding a new prefabricated element to the list
- material a prefabricated element is made of.

Options in the lower part of the tab depend on a selected type of a prefabricated element:

- for walls, beams, ground beams and continuous footings the following options are available:
  definition axis of a prefabricated element; the definition axis of a prefabricated element may be assumed as:
   center line
   upper line
   lower line
  When defining a prefabricated element by means of the outer or inner line, the user may determine an additional parameter (an offset from the definition line) in the edit field next to the icon  ; if the offset equals zero, then the definition line coincides with the upper or lower line.

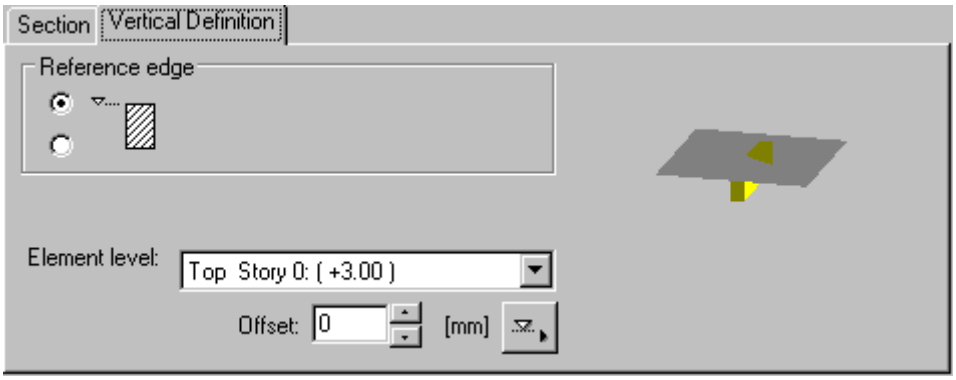
  rotation angle of the cross-section of a prefabricated element; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of the rotation angle.
  NOTE: While defining a prefabricated element lying on an inclined plane, there is also the *World UCS* option available. The option makes it easier to define the orientation of the cross-section of a prefabricated element with respect to the global coordinate system. If the *World UCS* option is switched off, then angle values provided on the selection list refer to the orientation of an inclined plane. If the *World UCS* option is switched on, then angle values provided on the selection list refer to directions of axes of the global coordinate system.

- for columns, slabs and raft foundations the following options are available:
  handle points of a prefabricated element during definition on the screen; the following handle points of prefabricated elements are available:
  upper left
  upper center
  upper right
  center left
  center
  center right
  lower left
  lower center
  lower right.

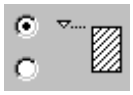
  rotation angle of the cross-section of a prefabricated element; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of the rotation angle.

6.12.4. Prefabricated element - vertical definition

The ***Prefabricated element: definition*** dialog box looks as presented in the drawing below after selecting the *Vertical definition* tab in it.



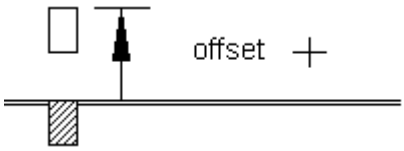
This dialog box is used to determine a location of a prefabricated element with respect to the active story.
The *Element level* list includes all the levels of the active story of a building defined so far; the location of a prefabricated element in a building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2). This list also contains all inclined planes defined for a selected story (see: Definition of a plane).
In addition, it is possible to define the reference edge of a prefabricated element with respect to the upper or lower level of a story. The following situations are possible:

Reference edge	**Location of a prefabricated element with respect to the story level (the example of a beam)**
1. 	
2. 	

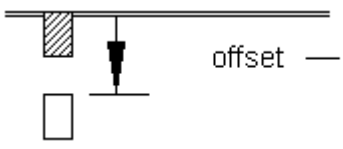
When defining the vertical location of a prefabricated element, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story. Pressing the  button allows selecting a horizontal element edge determining an offset value.

The following sign convention for offset values is adopted (it refers both to the lower and upper levels of a story):

1. positive offset value



2. negative offset value



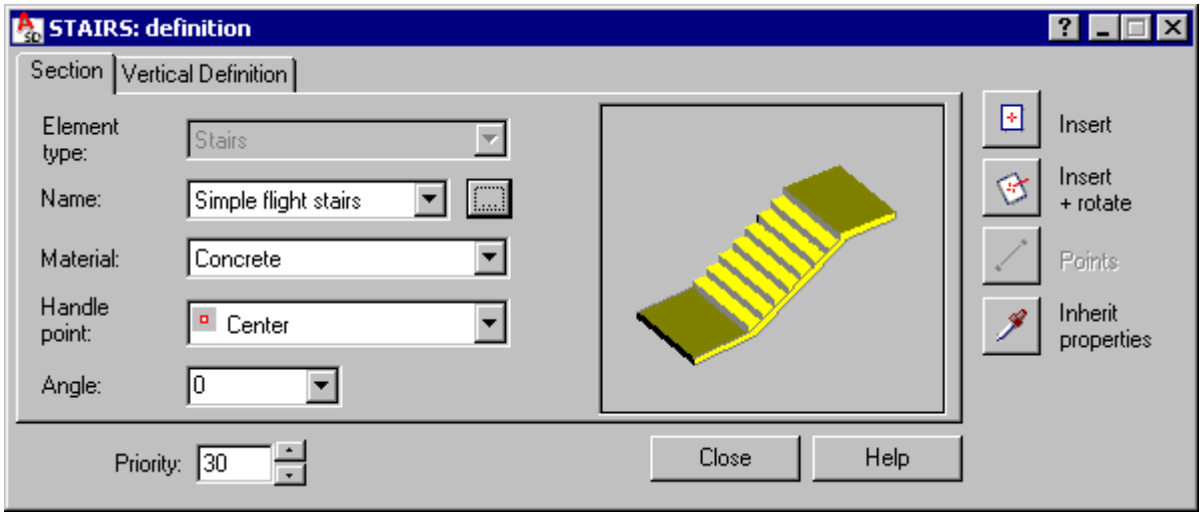
6.13. Stairs

6.13.1. Stairs - definition

The option enables definition of stairs in a structure model. Stairs are prefabricated elements (volumetric elements of complex geometry).
Definition of stairs may start after:

- selecting the menu option *Formwork Drawings / Define / Stairs*
- pressing the icon 

entering into the command line: RBCX_DEF_STAIRS.
After activating the *Stairs - definition* option, the dialog box shown in the drawing below appears on the screen.



The dialog box consists of the following tabs:
Section
Vertical Definition.

At the bottom of the dialog box is the Priority field presenting a default value of a priority which allows steering the interpenetration (cutting to fit) of structure elements that overlap with each other. This value may be modified for defined stairs.

The right part of the dialog box holds several icons that are used to select a mode of graphic definition of stairs. NOTE: only after pressing one of these icons it is possible to define stairs in the graphic viewer of the program.

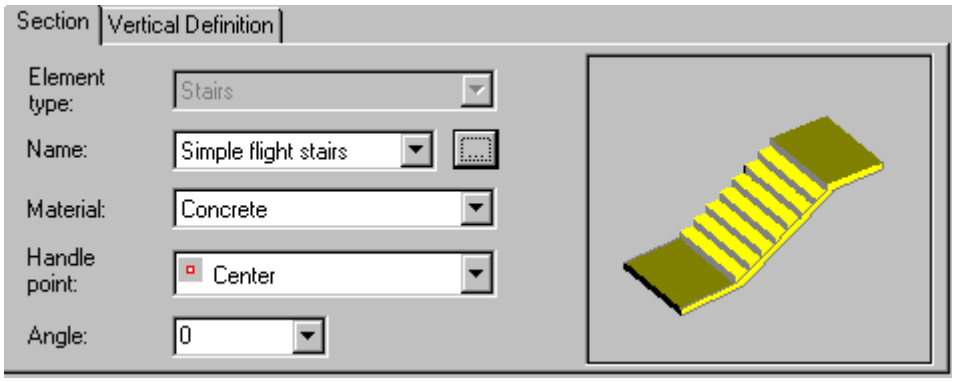
Stairs may be defined by:

-  indicating a point in the plan of a building story with a value of the angle of section rotation chosen from the selection list
-  indicating a point in the plan of a building story and defining graphically the rotation of stairs with respect to the insertion point (the angle is defined by indicating a point in the direction of which the stairs will be 'turned').

The icon  located in the bottom right corner is used to inherit parameters from stairs defined earlier.

6.13.2. Stairs - section

The *Stairs - definition* dialog box looks as presented in the drawing below after selecting the *Section* tab in it.



This dialog box allows defining parameters of a solid shape (stairs). The following parameters should be determined for the cross-section of stairs:

- type of the section of stairs (stairs' name) by selecting the solid's name from the list of available solids; pressing the (**...**) button located to the right of the selection list opens the dialog box which allows adding a new prefabricated element to the list
- material the stairs are made of
- handle points of stairs during the definition on the screen:
  - upper left
  - upper right
  - upper center
  - center left
  - center right
  - center
  - lower left
  - lower right.
  - lower center
- rotation angle of the cross-section of stairs; the following typical values of the rotation angle are available: 0, 45, 90, 135 and 180 degrees; this field also allows entering an arbitrary value of the rotation angle.

6.13.3. Stairs - vertical definition

The *Stairs - definition* dialog box looks as presented in the drawing below after selecting the *Vertical Definition* tab in it.

This dialog box is used to determine a location of stairs with respect to an active story.
The *Element level* list includes all the levels of an active story of a building defined so far; the location of stairs in a building is defined by selecting a plane (e.g. Top - Story 2, Bottom - Story 2).

In addition, there is a possibility to define the reference edge of stairs with respect to the upper or lower level of a story. The following situations are possible:

Reference edge	**Location of stairs with respect to the level of a story**
1. 	
2. 	

When defining the vertical location of stairs, the user may define an additional parameter (an offset from the lower or upper level of a story) in the *Offset* edit field. If the offset equals zero, then the definition line coincides with the upper or lower plane of a story. Pressing the  button allows selecting a horizontal element edge determining an offset value.

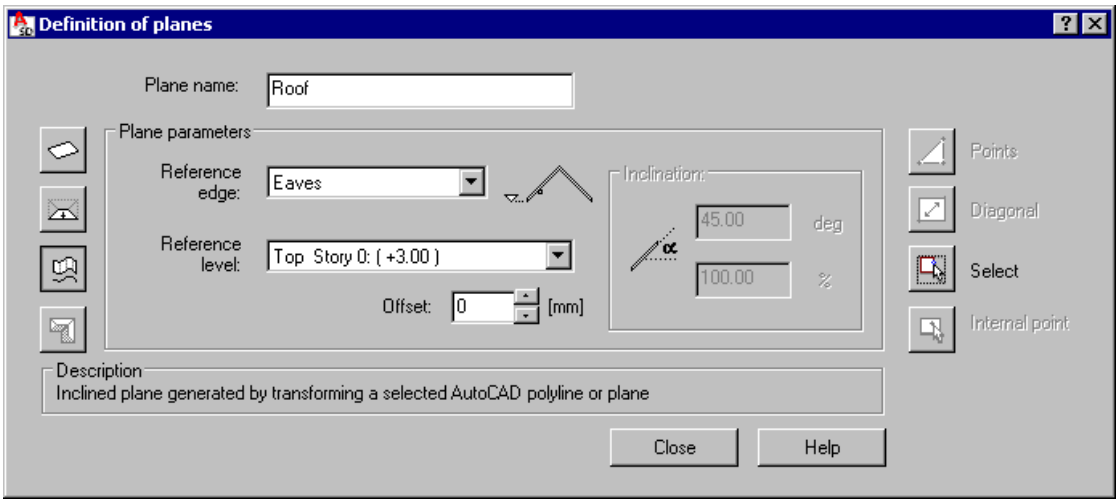
6.14.Planes

6.14.1. Definition of planes

The option enables defining intermediate (horizontal) planes or inclined planes on which elements of a structure model can be defined.
Definition of planes may start after:

- selecting the menu option *Formwork Drawings / Planes / Definition of plane*
- pressing the icon 
- entering into the command line: RBCX_PLANE_DEF.

After activating the *Definition of planes* option, the dialog box shown in the drawing below appears on the screen.



The current program version allows defining plane types as follows:

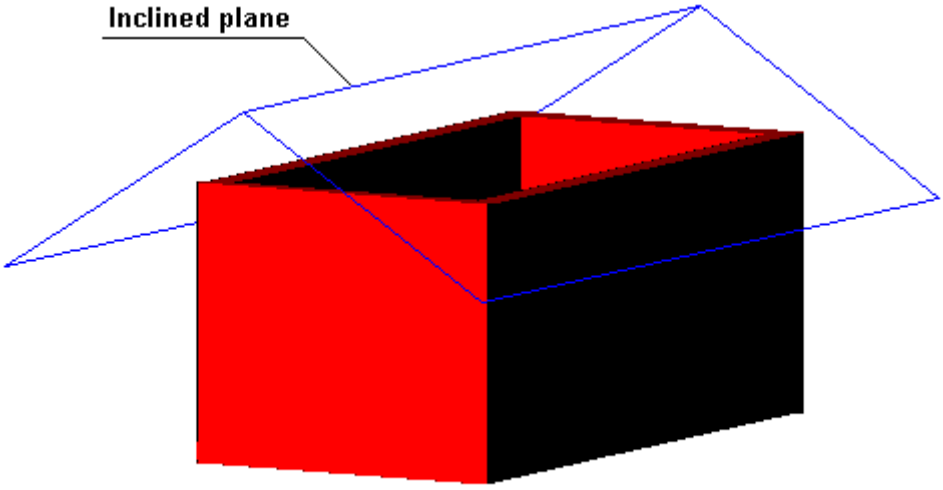
1.  intermediate plane (it is always a horizontal plane)
2.  inclined plane of a defined shape and inclination
3.  inclined plane defined according to the geometry of a selected polyline.

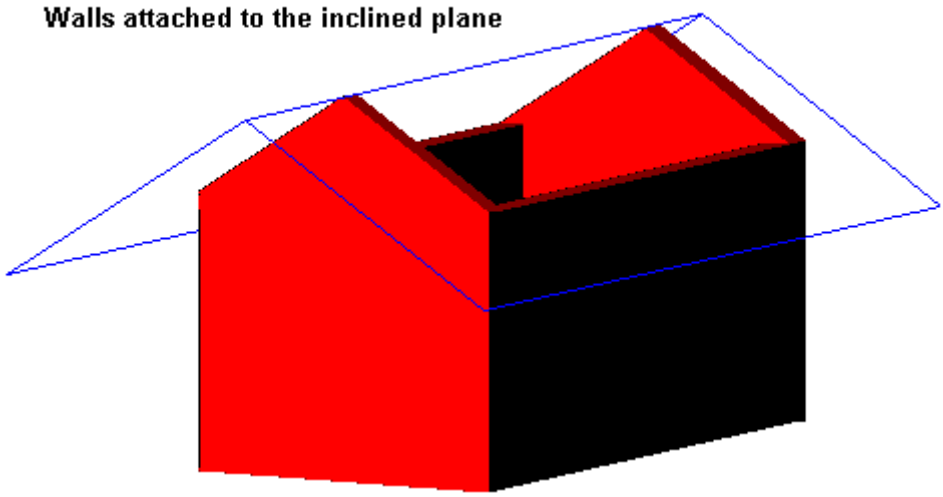
Defined planes are available on the selection lists of levels provided in the dialog boxes for definition of structure model elements (beams, slabs, walls, etc.). NOTE: defined planes are shown on these lists only for stories on which planes have been defined.

To define inclined elements (beams, slabs, etc.) on a chosen story, the user should select – in the dialog box for definition of a structure element – a defined inclined plane as a reference level (on the *Vertical Definition* tab). A type of the inclined element end (for beam elements as well as for slabs) can be defined on the additional *Details* tab in the dialog box for definition of structure elements. Options on this tab are available only then, when an inclined plane has been chosen as a reference level in the dialog box for definition of structure elements (beams, slabs).
Planes can be grouped. The *Group planes* option provided in the menu (see the description of the option in the Options available in the menu topic) allows creating a group of planes which makes it easier to define elements of a structure model (e.g. elements belonging to a multi-pitch roof).
Structure elements can be 'attached' to an inclined plane:

- by selecting a name of the inclined plane on the *Vertical Definition* tab in the dialog box for modification of a single element
- after selecting the *Attach to plane* option included in the menu for element selection (see the description of the option given in the Options available in the menu topic).

An example of attaching walls to the inclined plane is shown in the drawing below.





The icon  located in the bottom right corner is used to inherit parameters from a plane defined earlier.

6.14.2. Definition of planes - intermediate horizontal plane

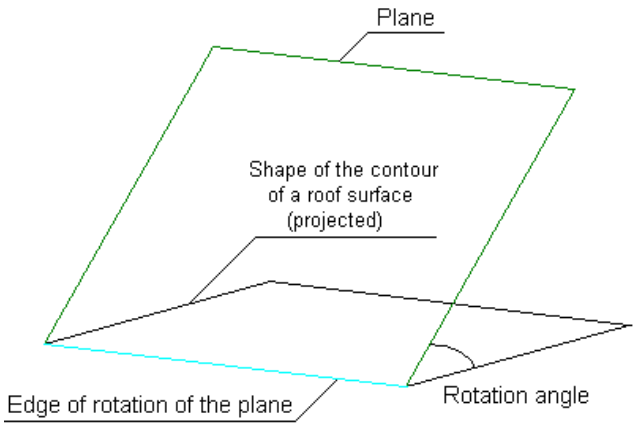
An intermediate plane is always a horizontal plane parallel to the top/bottom reference level of a story. It is defined for a selected (active) story. It allows defining structure elements positioned between the bottom and top level of a story; thus, the intermediate plane is an alternative to offsets defined while creating structure elements (walls, columns, beams, etc.).
To define the intermediate plane, the user should:

- specify a plane name in the *Plane name* edit field
- press the  icon which enables defining a horizontal intermediate plane
- determine parameters of the plane location:
  - reference level (bottom or top of a story)
  - offset of the plane with respect to a selected reference level
- press the  icon located in the right-hand part of the dialog box.

A defined plane is added to the list of planes provided in the ***Manager of plane properties*** dialog box.
Defined planes are available on the selection lists of levels for a given story included in the dialog boxes for definition of structure model elements (beams, slabs, walls, etc.).

6.14.3. Definition of planes - inclined plane of a defined shape and inclination

An inclined plane of a determined shape and inclination is a plane generated through transforming an indicated projection of the roof surface contour. It is defined for a selected (active) story. It allows definition of successive roof surfaces on which elements of the roof will be defined.
To define this plane type, the user should (see the drawing below):
- indicate a shape of the (projected) plane contour
- determine the rotation axis (the edge of rotation of the plane)
- specify a rotation angle.

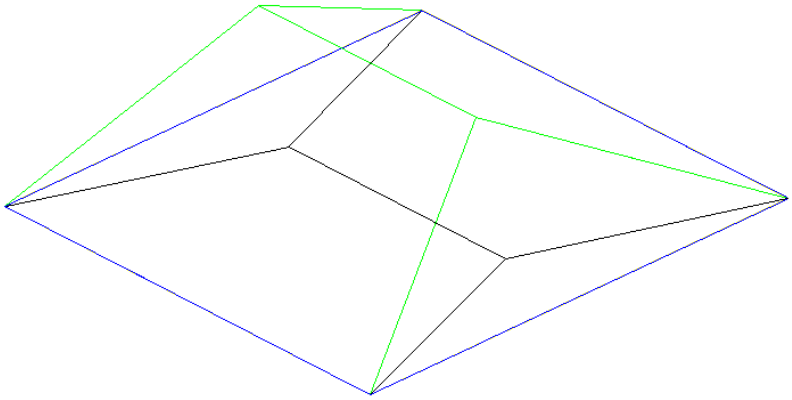


The program offers the following methods of defining a shape of the contour of an inclined plane:
-  by indicating successive points belonging to the contour
-  by indicating the diagonal of the contour (shape of a rectangle)
-  by indicating the contour (closed contour)
-  by indicating an internal point of the contour (closed contour).

This type of plane allows defining multi-pitch roofs. Each roof surface needs to be defined separately (if the contour is symmetrical, only half of the roof surface may be defined).

NOTE:
The contour of an inclined plane (projection) may be modified only on the plane (in 2D projection); as regards modification of the rotation angle (inclination of the plane), it is possible only in the 3D view – there is a possibility to shift points of the plane.

An example of a multi-pitch roof created using this type of plane is presented in the drawing below.



6.14.4. Definition of planes - inclined plane

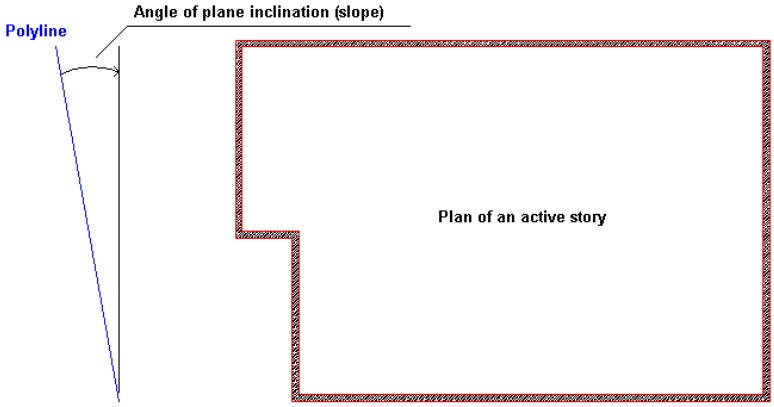
An inclined plane is a plane generated as a result of transforming a polyline indicated by the user (the polyline has to be defined before starting definition of the inclined plane). It is defined for a selected (active) story. It enables defining structure elements inclined at a chosen angle to the horizontal plane of stories such as roof elements.

NOTE:
The inclined plane may be deleted only then, if no structure elements are attached to it; if structure elements have been attached to the inclined plane, then it is impossible to delete such a plane.

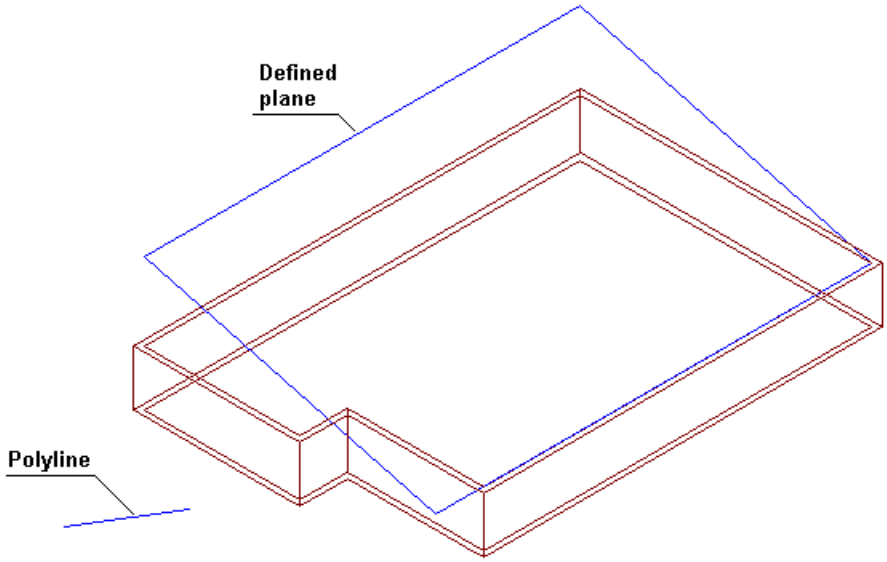
The program allows for two methods of defining the inclined plane:

1. Method - definition on a story plan
To define the inclined plane, the user should:

- indicate a polyline defined on a story plan (see the drawing below); NOTE: an inclination angle of the polyline determines the slope of the defined inclined plane

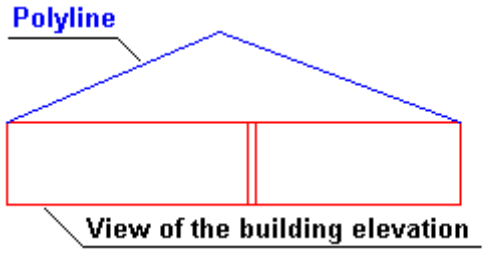


- specify a plane name in the *Plane name* edit field
- press the icon which enables defining the inclined plane
- determine parameters of the plane location:
- - reference edge
- - reference level (bottom or top of a story) - see the drawings below
- - offset of the plane with respect to a selected reference level, if needed
- press the icon located in the right-hand part of the dialog box
- indicate the polyline
- determine the direction of polyline projection (in the direction of the X axis or Y axis)
- indicate two points defining an area of the inclined plane.

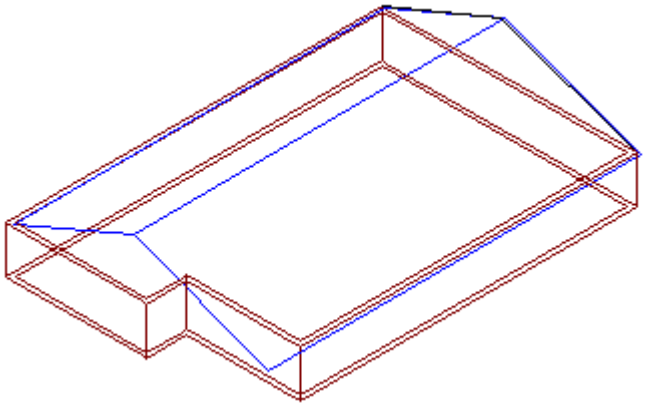


2. Method - definition in the side view of a structure

- indicate a polyline defined in the side view of structure (see the drawing below); NOTE: geometry of the polyline determines the geometry of the defined inclined plane

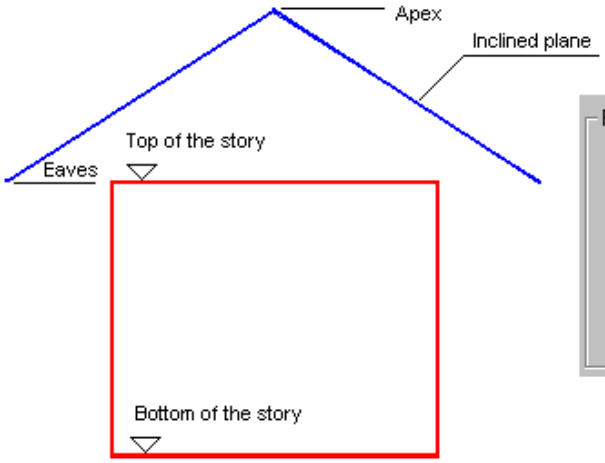


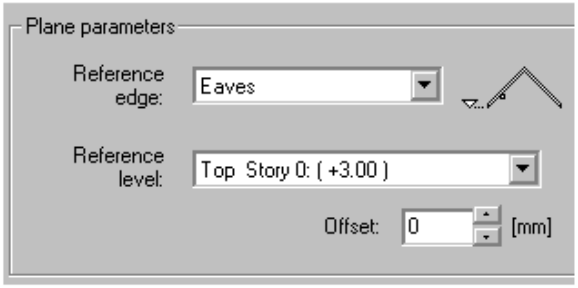
- specify a plane name in the *Plane name* edit field
- press the icon which enables defining the inclined plane
- determine parameters of the plane location:
  - reference edge
  - reference level (bottom or top of a story) - see the drawings above for the 1st method
  - offset of the plane with respect to a selected reference level, if needed
- press the icon located in the right-hand part of the dialog box; NOTE: the structure view presented on the screen is changed automatically to the active story plan
- indicate two points defining an area of the inclined plane.



6.14.5. Definition of planes - parameters of plane location

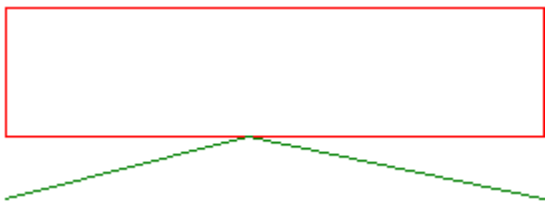
For an inclined plane it is possible to determine the following plane locations:





selected plane parameters:
reference edge: *apex*
reference level: *top story*

selected plane parameters:
reference edge: apex
reference level: *bottom story*



selected plane parameters:
reference edge: *eaves*
reference level: *top story*

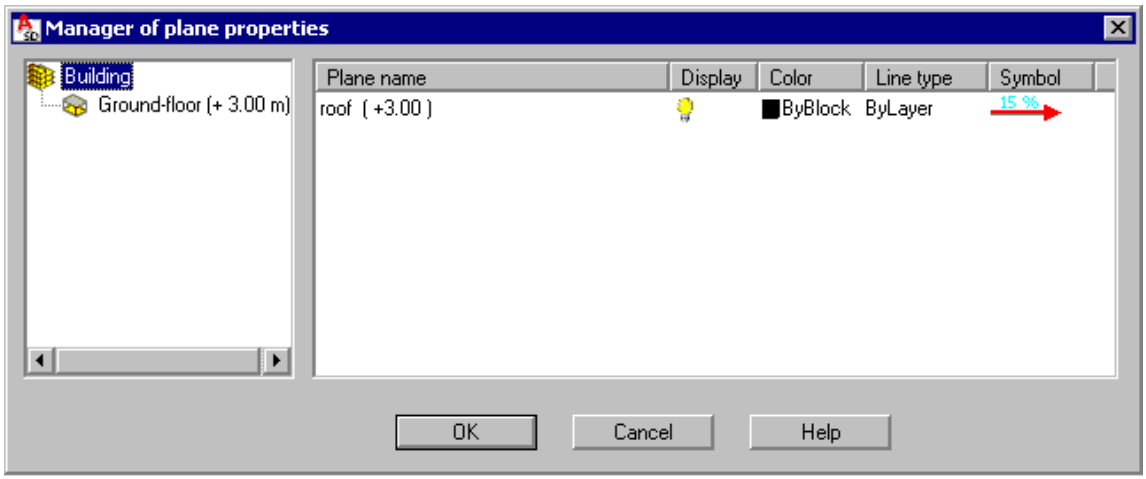
selected plane parameters:
reference edge: *eaves*
reference level: *bottom story*

6.14.6. Manager of plane properties

The option allows viewing or editing parameters of (intermediate or inclined) planes defined in a structure model. The option is accessible after:

- selecting the menu option Formwork Drawings / Display / Show planes
- pressing the icon
- entering into the command line: RBCX_EDIT_PLANES.

After activating the *Manager of plane properties* option, the dialog box shown in the drawing below appears on the screen.



The dialog box provides the following information concerning defined planes:

- plane name (with additional information regarding the level of plane definition)
- plane display (a plane may be displayed or not on the screen) - pressing the light bulb symbol in the *Display* column changes the plane display
- color of plane presentation on the screen
- type of the line for plane presentation
- symbol of the plane inclination (none, arrow, arrow with a value of the plane inclination).

In the dialog box, in the table presenting parameters of planes there is a context menu (after pressing the right mouse button) including the option as follows:

- *Edit* - selecting this option opens the dialog box for modification of parameters of a selected plane
- *Delete* - selecting this option deletes a selected plane.

7. DATABASES

7.1. Available databases

7.1.1. Databases

The option is used to define different structure elements (cross sections, materials, doors, windows, openings), open existing, external databases (e.g. ***ROBOT*** databases) and save defined elements in databases (*.mdb files).
The following elements may be defined in ***AutoCAD © Structural Detailing - Formwork Drawings***:
Section database
Material database
Door database
Window database
Opening database.

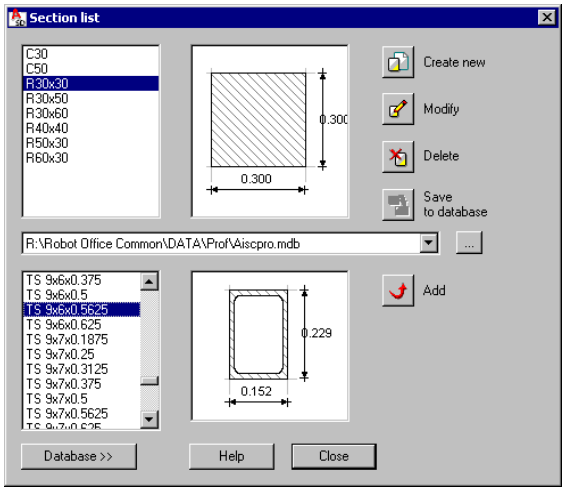
Moreover, the following databases of prefabricated elements (i.e. solids of complex geometry) are available in the program:
Spread footing database.
Prefabricated beam database
Prefabricated column database
Stairs database
Slab database
Wall database
Continuous footing database
Raft foundation database
Ground beam database.

7.1.2. List of cross-sections

The dialog box, shown in the drawing below, allows the user to:

- define a new section shape using basic mechanisms of the AutoCAD © program
- open external section databases (e.g. ***ROBOT*** databases) and use sections included there in ***AutoCAD © Structural Detailing - Formwork Drawings***
- save defined sections in the database (an *.mdb file).

NOTE:
The dialog box presented in the drawing below is expanded (larger than the dialog box opened in a standard way); it means that there are the options in the lower part of the dialog box which are used for adding sections from a selected database. To collapse or expand the dialog box, press the ***Database*** *button.*



The top left part of the dialog box holds a list of defined cross-sections of structure elements. These sections will be presented on the selection lists of element cross-sections. In the middle part of the dialog box is a schematic drawing representing a section type (rectangular, circular section, etc.) chosen from the list of defined sections along with characteristic section dimensions.

In the right-hand side of the dialog box are the icons that are used to:


open the ***Section definition*** dialog box where a new cross-section may be defined; the section is added to the list of available sections in the current project


open the ***Section modification*** dialog box where parameters of a selected section (names, dimensions) can be modified


delete a selected section from the list of available sections in the project

save to an external file (database).

The dialog box expands after pressing the **Database** button at the bottom of the dialog box. This part of the dialog box includes:

- field for selection of a cross-section database (an *.mdb format file), from which a section may be added to the list of sections available in ***AutoCAD © Structural Detailing - Formwork Drawings***
- list of cross-sections defined in a database
- schematic drawing representing a section type (rectangular, circular section, etc.) and its dimensions.

To add a section from a database to the list of sections available in ***AutoCAD © Structural Detailing - Formwork Drawings***, press the icon  . The section will be added to the list in the top left part of the dialog box.

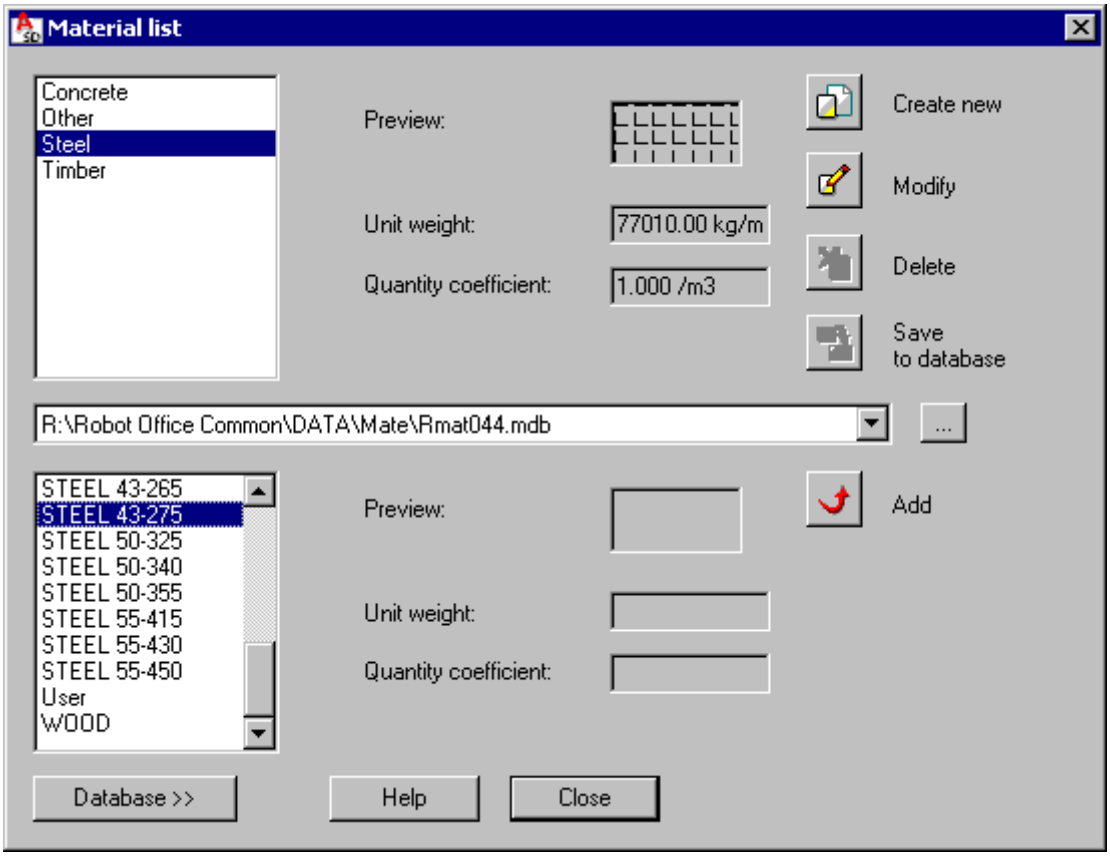

NOTE:
*All section databases used in the **ROBOT** program can be imported to **AutoCAD © Structural Detailing - Formwork Drawings**.*

7.1.3. Material list

The dialog box, shown in the drawing below, allows the user to:

- define a new material
- open external material databases (e.g. ***ROBOT*** databases) and use materials included there in ***AutoCAD © Structural Detailing - Formwork Drawings***
- save defined materials in the database (an *.mdb file).


NOTE:
*The dialog box presented in the drawing below is expanded (larger than the dialog box opened in a standard way); it means that there are the options in the lower part of the dialog box which are used for adding materials from a selected database. To collapse or expand the dialog box, press the **Database** button.*



The top left part of the dialog box includes a list of defined materials. These materials will be presented on the material selection lists in ***AutoCAD © Structural Detailing - Formwork Drawings***. In the middle part of the dialog box are basic parameters of a selected material (presentation of a material, unit weight).

In the right-hand side of the dialog box are the icons that are used to:


open the ***Material definition*** dialog box where a new material may be defined; the defined material is added to the list of available materials in the project


open the ***Material modification*** dialog box where parameters of a selected material (name, parameters) can be modified


delete a selected material from the list of available materials in the project

save to an external file (database).

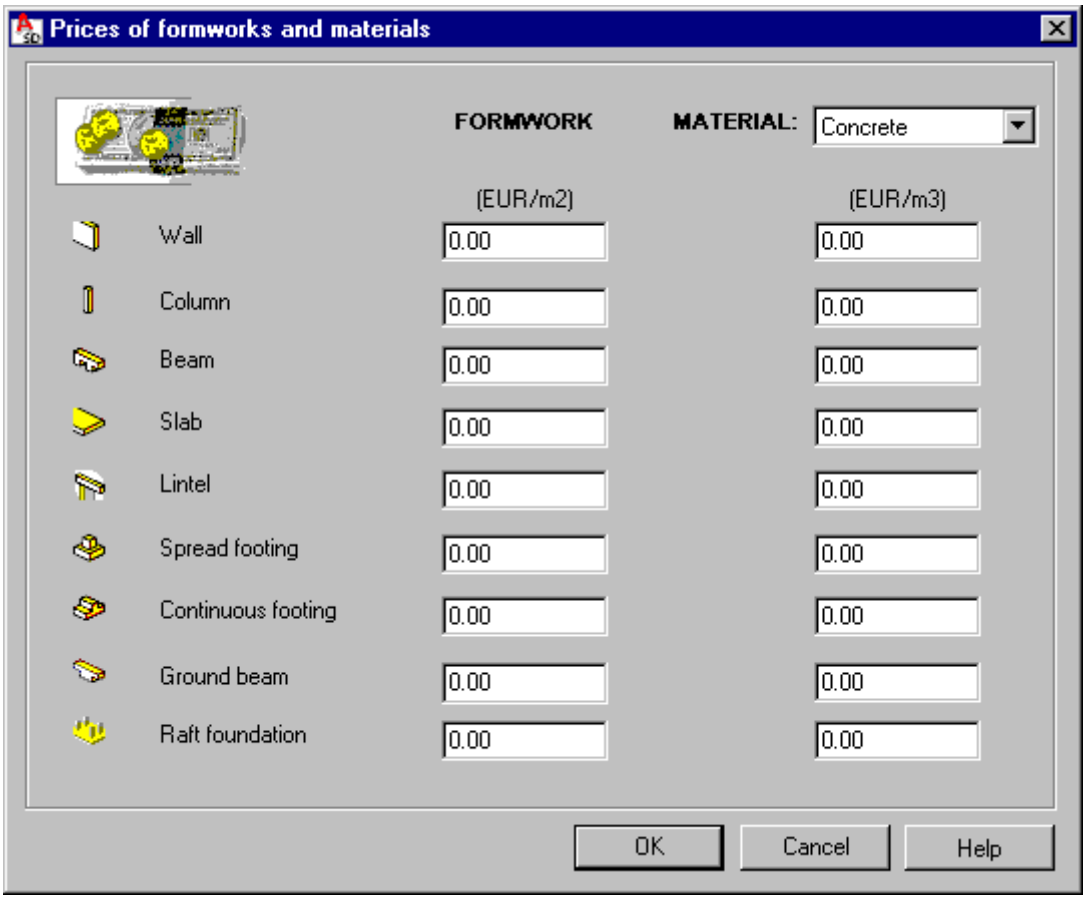
On pressing the **Database** button at the bottom of the dialog box, the dialog box expands. This part of the dialog box includes:

- field for selection of a material database (an *.mdb format file), from which a material may be added to the list of materials available in ***AutoCAD © Structural Detailing - Formwork Drawings***
- list of materials defined in a database
- basic parameters of a chosen material (graphic presentation of a material, unit weight).

To add a material from the database to the list of materials available in ***AutoCAD © Structural Detailing - Formwork Drawings*** the user should use the icon  . The material will be added to the list in the top left part of the dialog box.

7.1.4. Prices of formworks and materials

The dialog box below opens on pressing the **Prices** button located in the ***Job Preferences*** dialog box (the Databases tab) for material databases. The options provided in this dialog box allow selecting prices of formworks of RC structure elements and prices of materials (concrete, steel, timber, etc.). These prices will be used in tables generated by the program.



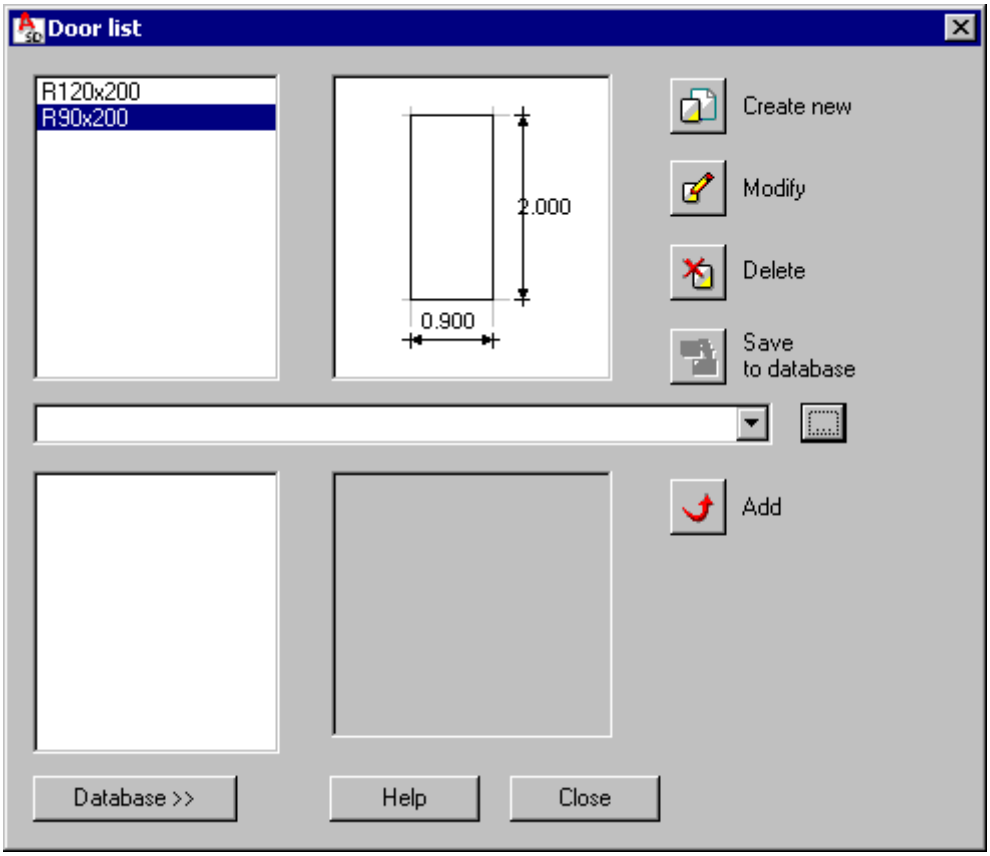
The above dialog box enables determining the prices of:

- formworks of individual component elements of RC structures (in EUR per square meter of a formwork)
- materials (e.g. concrete, steel, etc.) for individual component elements of a structure (in EUR per cubic meter).

7.1.5. Opening list (doors, windows, remaining openings)

The dialog box, shown in the drawing below, allows defining dimensions of new openings in building elements and saving them in the database (an *.mdb file).
It is also possible to open external databases of openings and use opening definitions included there in ***AutoCAD © Structural Detailing - Formwork Drawings***.

NOTE:
The dialog box presented in the drawing below is expanded (larger than the dialog box opened in a standard way); it means that there are the options in the lower part of the dialog box which are used for adding openings from a selected database. To collapse or expand the dialog box, press the ***Database*** *button.*



The top left part of the dialog box includes a list of defined openings. These openings will be presented on the opening selection lists. In the middle part of the dialog box is a schematic drawing representing an opening type (rectangular, arc-shaped section, etc.) chosen from the list of defined openings along with characteristic opening dimensions.

In the right-hand side of the dialog box are the icons that are used to:


open the ***Opening definition*** dialog box where a new opening may be defined; the defined opening is added to the list of available openings in the project


open the ***Opening modification*** dialog box where parameters of a selected opening (name, dimensions) can be modified


delete a selected opening from the list of available openings in the project

save to an external file (database).

On pressing the **Database** button at the bottom of the dialog box, the dialog box expands. This part of the dialog box includes:

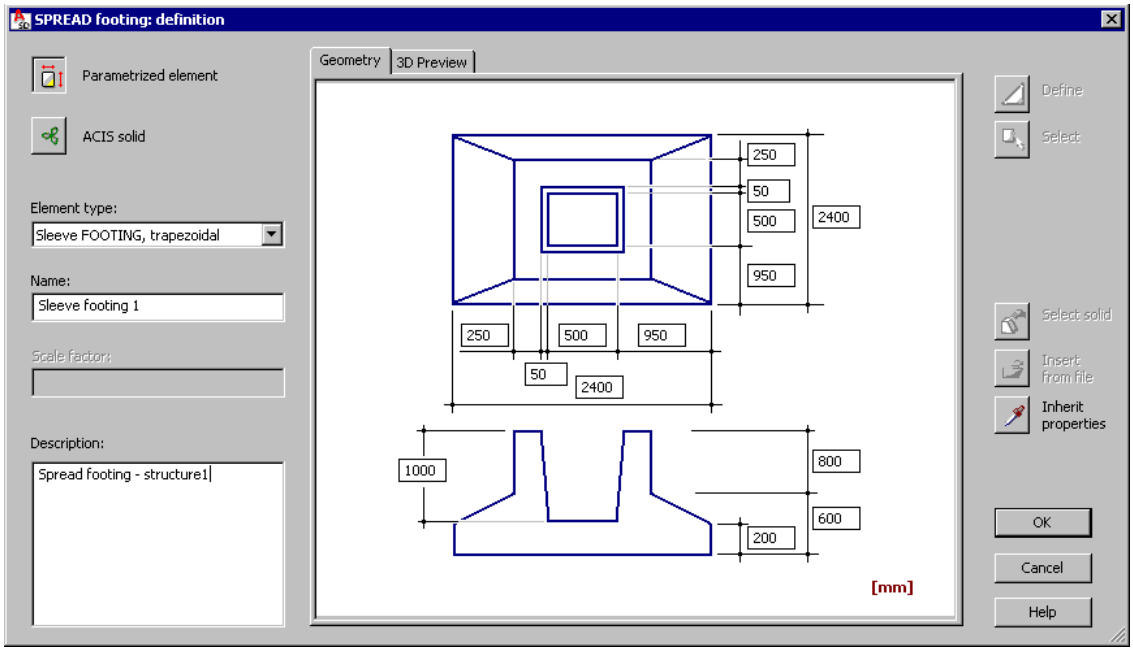
- field for selection of a opening database (an *.mdb format file), from which an opening may be added to the list of openings available in ***AutoCAD © Structural Detailing - Formwork Drawings***
- list of openings defined in a database
- schematic drawing representing an opening type (rectangular, circular section, etc.) and its dimensions.

To add an opening from the database to the list of openings available in ***AutoCAD © Structural Detailing - Formwork Drawings***, press the icon  . The opening will be added to the list in the top left part of the dialog box.

7.2. Databases of volumetric elements

7.2.1. Definition of prefabricated elements

The dialog box, shown in the drawing below, allows defining a new prefabricated element (a volumetric element of complex geometry, e.g. spread footings, stairs, etc.).



There are two possibilities of definition:

- - *Parametrized element*: by specifying dimensions that define the geometry of a selected type of prefabricated element; a defined element is saved to the database and is available on the lists for a selected type of prefabricated element
  NOTE: for spread footings of arbitrary shape there are two icons available in the right-hand part of the dialog box:
  Define - defines a shape of the cross-section of a spread footing
  Select - defines a shape of the cross-section of a spread footing by selecting a polyline (created with the use of the AutoCAD © options) that forms a closed contour

- - *ACIS solid*: enables definition of a prefabricated element by defining or indicating an ACIS solid (the SOLID 3D element of AutoCAD © or a group of objects of the FACE type which defines a coherent, closed solid); NOTE: in the latter case, a group of objects of the FACE type must be a block; after selecting this option, the right-hand part of the dialog box holds the two icons:
  - allows indicating an existing ACIS solid defined by means of the available AutoCAD© options
  - allows inserting a solid from an indicated DWG file (NOTE: a DWG file may include a definition of only one solid).
  A defined element is saved to the database and is available on the lists for a selected type of prefabricated element.

The central part of the dialog box - the *Geometry* tab shows a schematic drawing which presents a prefabricated element type along with characteristic dimensions of the section. The *3D Preview* tab includes a 3D drawing of a defined element.

The left-hand part of the dialog box holds the following options:

- the *Element type* list: the contents of this list depends on the type of prefabricated element for which the dialog box has been opened; the following element types are available:
  Spread footings
  Prefabricated beams
  Prefabricated columns

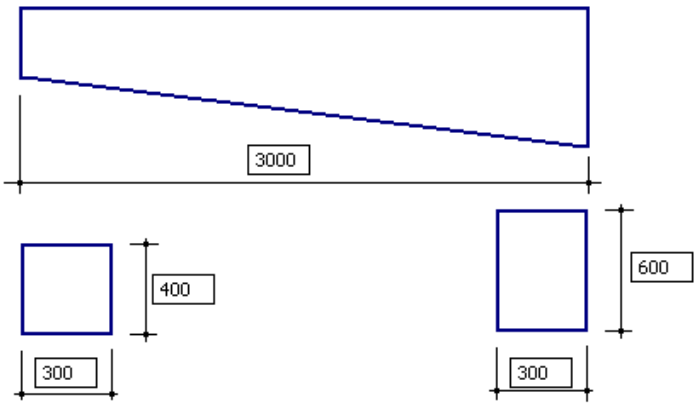
Stairs
Slabs
Walls
Continuous footings
Raft foundations
Ground beams
NOTE: Some types of prefabricated elements may be defined **only** through definition or indication of an ACIS solid (they may not be defined through a parametrized element)

- the *Name* edit filed - a field where the user may specify a name of a defined element
- the *Scale factor* edit field - a field where the user may specify a value of the scale factor according to the AutoCAD © rules
- the *Description* edit field - a field where the user may give an additional description of a defined element; this description will be provided in the INFO window (the Element info option).

7.2.2. Prefabricated elements - beams

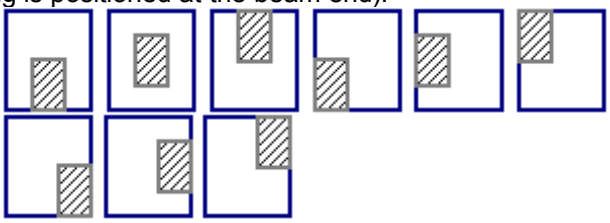
AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined types of prefabricated beams (prefabricated elements):

1. beam with a rectangular cross-section (tapered section)

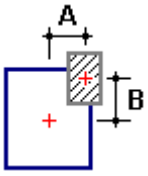


Apart from the definition of beam dimensions, for this type of beam it is also possible to define the following positions of the cross-section at the beginning and end of a beam (i.e. how the cross-section defined at the beam beginning is positioned at the beam end):

- several characteristic positions of cross-sections at the beginning / end of a beam



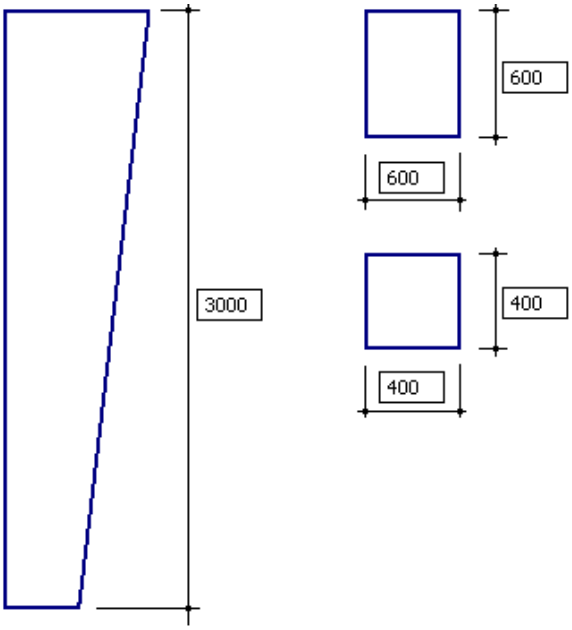
- an arbitrary position of the cross-section determined by specifying values of the parameters A and B.



7.2.3. Prefabricated elements - columns

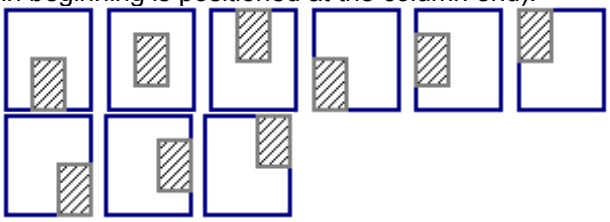
AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined types of prefabricated columns (prefabricated elements):

1. column with a rectangular cross-section (tapered section)

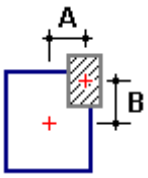


Apart from the definition of column dimensions, for this type of column it is also possible to define the following positions of the cross-section at the beginning and end of a column (i.e. how the cross-section defined at the column beginning is positioned at the column end):

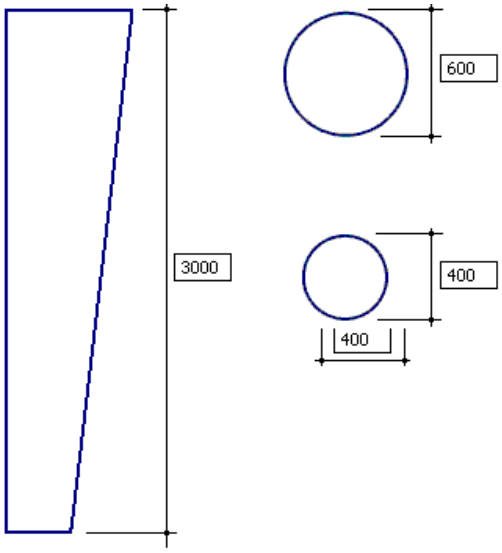
- several characteristic positions of cross-sections at the beginning / end of a column



- an arbitrary position of the cross-section determined by specifying values of the parameters A and B.

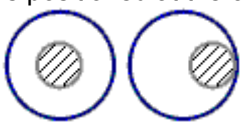


2. column with a round cross-section (tapered section)

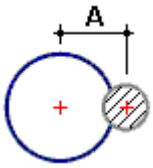


Apart from the definition of column dimensions, for this type of column it is also possible to define the following positions of the cross-section at the beginning and end of a column (i.e. how the cross-section defined at the column beginning is positioned at the column end):

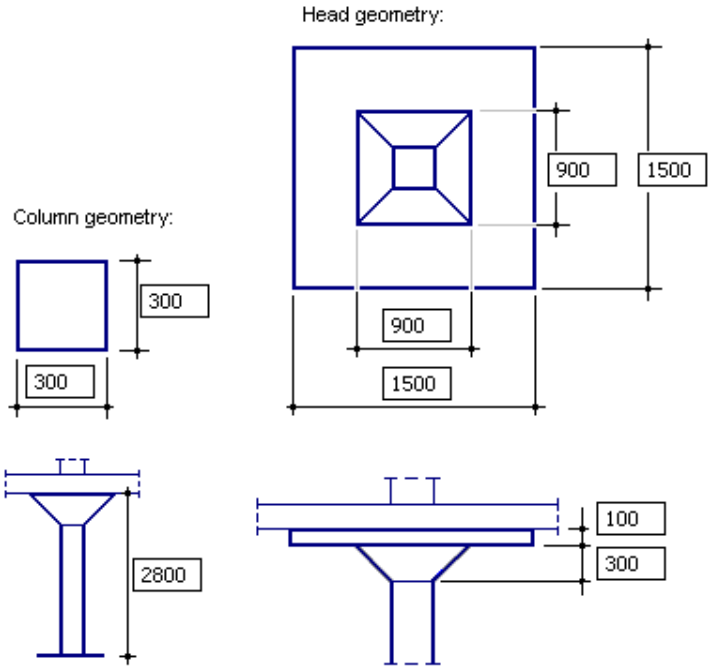
- several characteristic positions of cross-sections at the beginning / end of a column



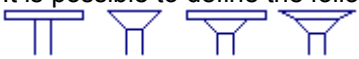
- an arbitrary position of the cross-section determined by specifying a value of the parameter A.



3. column with a rectangular or round cross-section, with a head



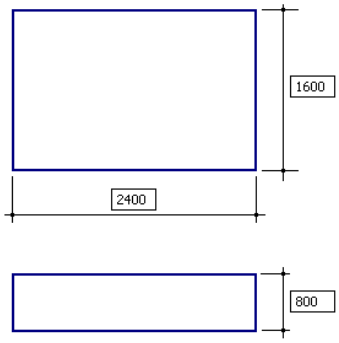
It is possible to define the following column head types for this type of column:



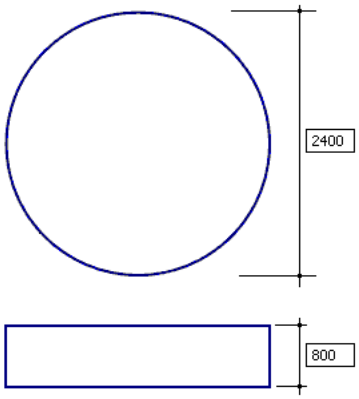
7.2.4. Prefabricated elements - spread footings

AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined types of spread footings (prefabricated elements):

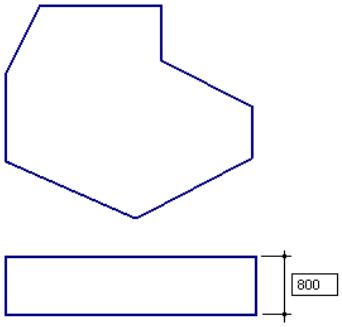
1. rectangular footing



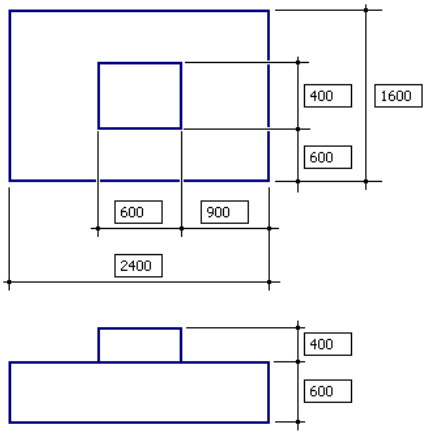
2. circular footing



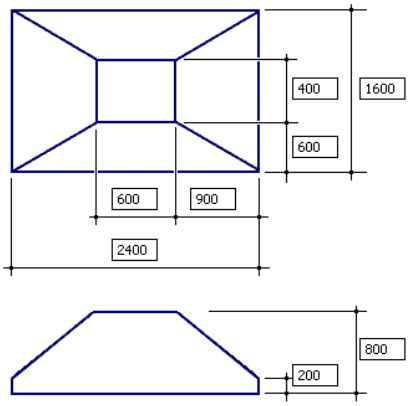
3. footing of arbitrary shape



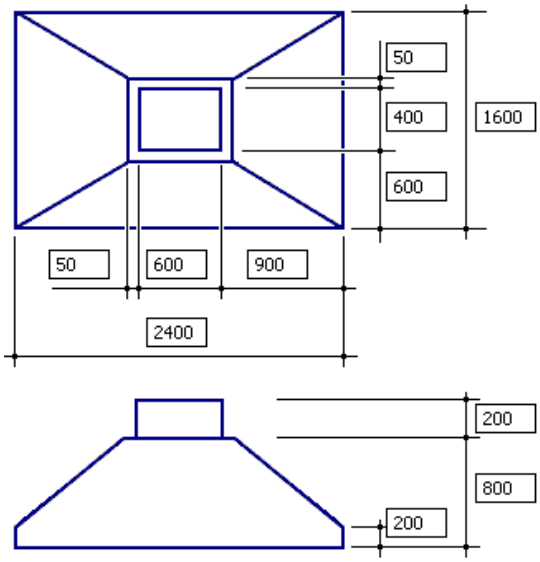
4. rectangular footing with a pier



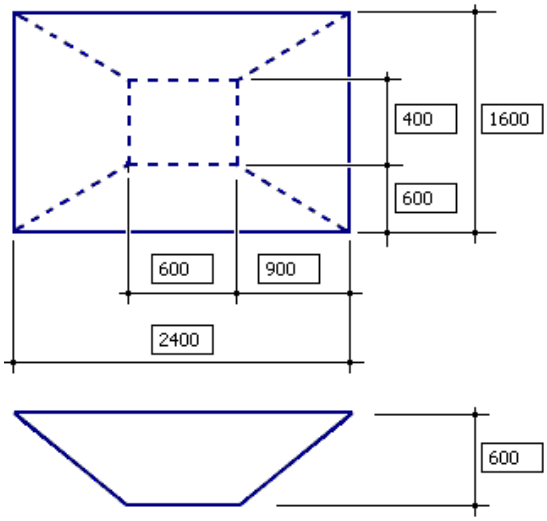
5. trapezoidal footing



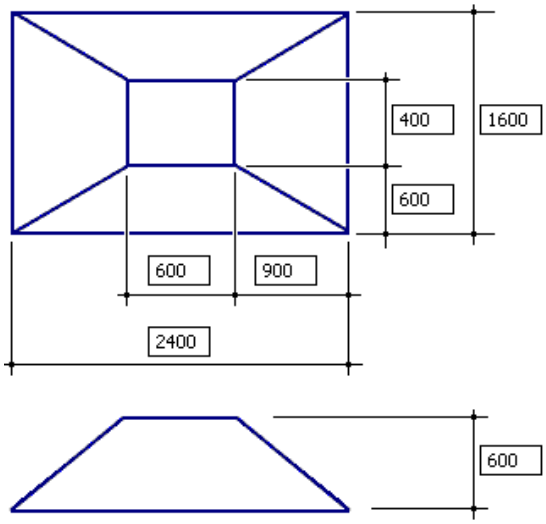
6. trapezoidal footing with a pier



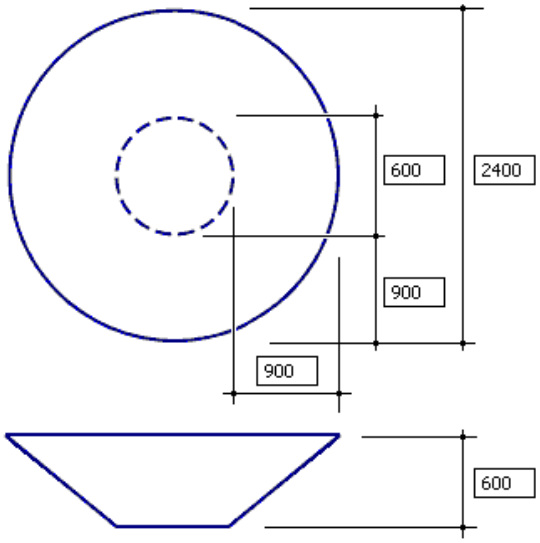
7. trapezoidal head type1



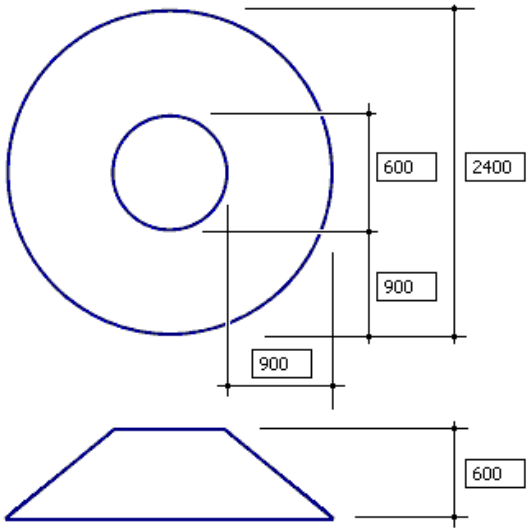
8. trapezoidal head type2



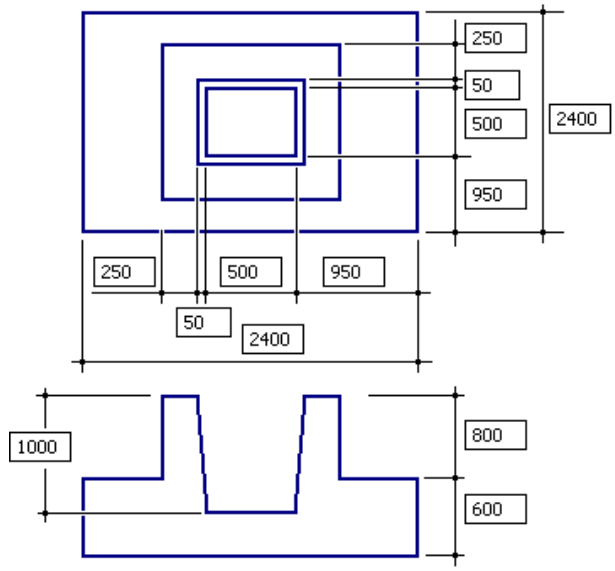
9. conical head type1



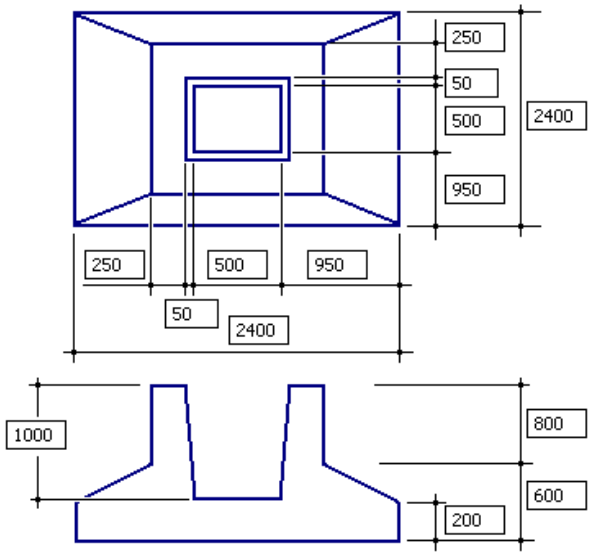
10. conical head type2



11. sleeve footing (rectangular)



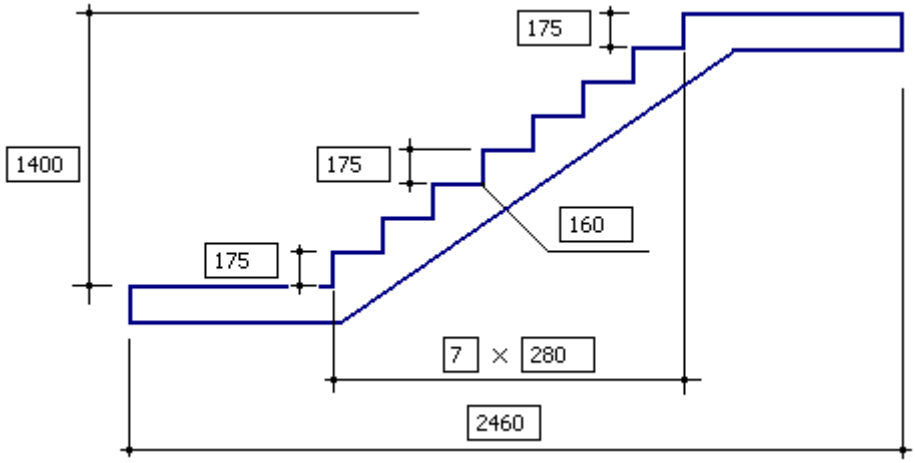
12. sleeve footing (trapezoidal)



7.2.5. Prefabricated elements - stairs

AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined types of stairs (prefabricated elements):

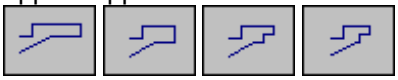
1. simple flight stairs



It is possible to define the following support types for this type of stairs:

lower support:

upper support:



7.3. Definition / modification of database elements

7.3.1. Definition / modification of sections and openings

The option is used to define or modify different structure elements (cross-sections, doors, windows, openings) provided on the selection lists.

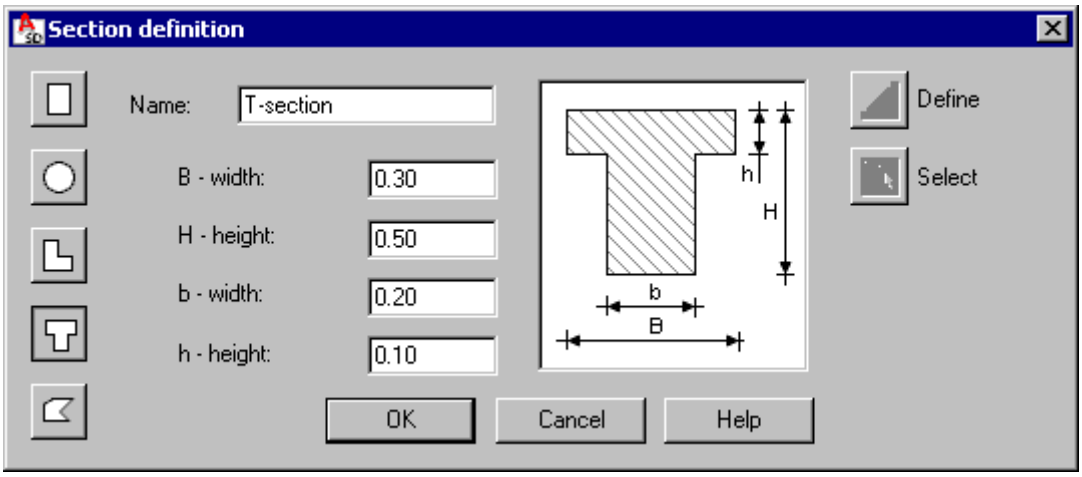
The following elements may be defined/modified in ***AutoCAD © Structural Detailing - Formwork Drawings***:

- cross-sections

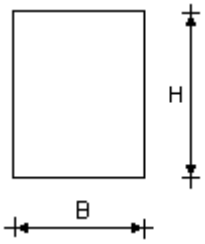
- doors
- windows
- remaining openings.

7.3.2. Definition / modification of a cross-section

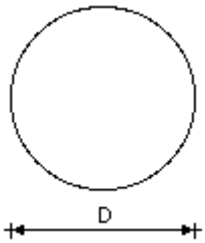
The dialog box presented in the drawing below allows defining / modifying the following predefined types of cross-sections:



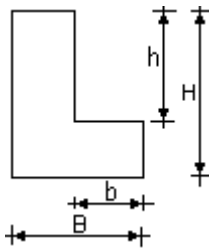
Rectangular opening - parameters: H - height
B - width



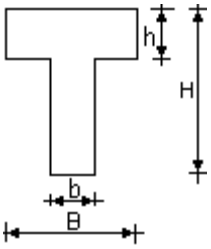
Round opening - parameters: D - diameter



L-shaped opening - parameters: h, H - heights
b, B - widths



T-shaped opening - parameters: h, H - heights
b, B - widths



Section of an arbitrary shape; two possibilities of definition:

 *Define* – a shape of a cross-section is defined by indicating on the screen successive characteristic points of a section

 *Select* - a shape of a cross-section is defined by selecting a polyline (created with the use of the AutoCAD © options) that forms a closed contour.

To add a new section to the list of sections available in ***AutoCAD © Structural Detailing - Formwork Drawings***, follow the steps below:

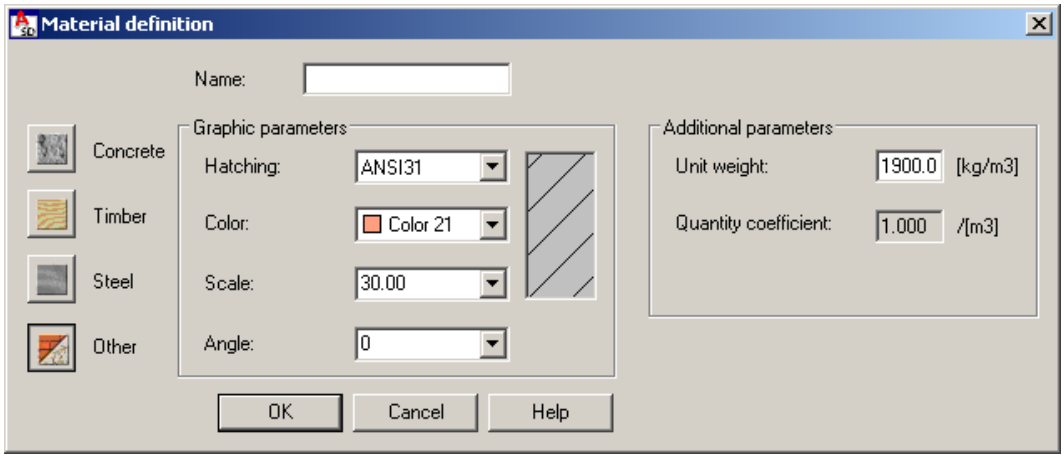
- specify a section name in the *Name* field
- select a cross-section type (rectangular, round section, etc.)
- determine dimensions of a cross-section appropriate for a selected cross-section type
- press the **OK** button.

When defining an arbitrarily-shaped section, specify a name of a cross-section and choose one of the methods of section contour definition (definition of a contour on the screen or selection of an existing contour on the screen).

7.3.3. Definition / modification of a material

AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined material groups:

-  concrete
-  timber
-  steel
-  other materials.



The following parameters can be defined for each material:
- graphic parameters (hatching, color and scale of a cross-section, angle of hatching)
- additional parameters (unit weight, quantity coefficient).

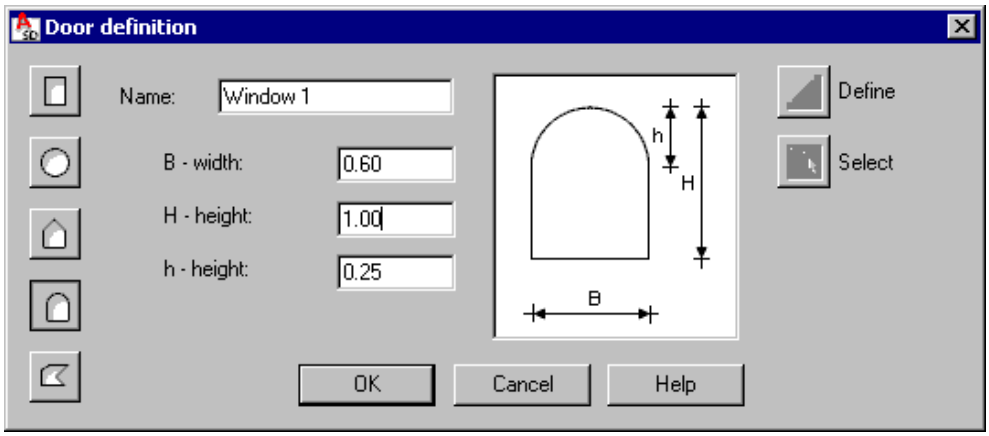
To add a new material to the list of materials available in ***AutoCAD © Structural Detailing - Formwork Drawings***, follow the steps below:

- select a type of material (concrete, timber, steel, etc.)
- specify a material name in the *Name* field

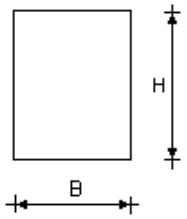
- determine material parameters
- press the **OK** button.

7.3.4. Definition / modification of openings

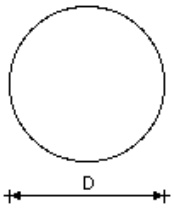
AutoCAD © Structural Detailing - Formwork Drawings allows definition / modification of the following predefined types of openings (windows, doors, etc.):



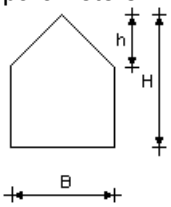
Rectangular opening - parameters: H - height
B - width



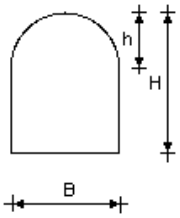
Round opening - parameters: D - diameter



Pentagon-shaped opening - parameters: h, H - heights
B - width



Arc-shaped opening - parameters: h, H - heights
B - width



Section of an arbitrary shape; two possibilities of definition:

 *Define* – an opening shape is defined by indicating on the screen successive characteristic points of an opening

 *Select* – an opening shape is defined by selecting a polyline (created with the use of the AutoCAD © options) that forms a closed contour.

To add a new opening to the list of openings available in ***AutoCAD © Structural Detailing - Formwork Drawings***, follow the steps below:

- specify an opening name in the *Name* field
- select an opening shape
- determine characteristic dimensions of an opening appropriate for a selected opening shape
- press the **OK** button.

When defining an arbitrarily-shaped section, specify a name of an opening and choose one of the methods of opening contour definition (definition of a contour on the screen or selection of an existing contour on the screen).

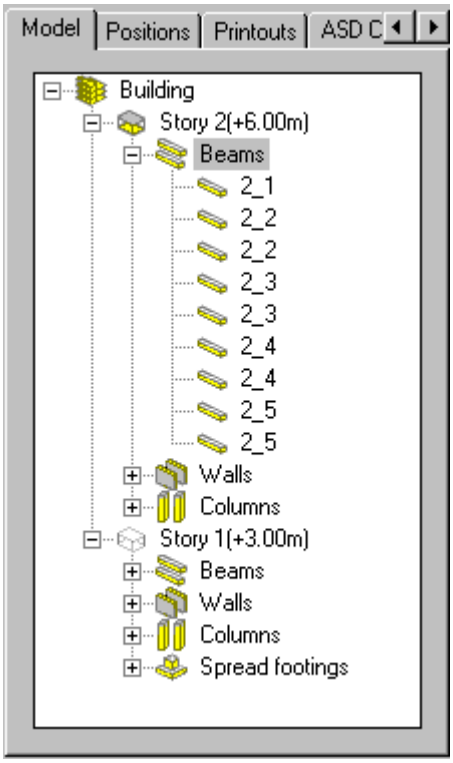
8. ELEMENT POSITIONING

8.1. Positioning of structure elements

AutoCAD © Structural Detailing - Formwork Drawings allows automatic positioning of objects defined in a structure model.
Positioning is performed for an object selected in the tree on the *Model* tab in the ***Object Inspector*** dialog box. Positioning may refer both to a structure part (e.g. story) and to single elements of the structure.
Positioning is aimed at giving names to elements (assigning an identifier - a position), detecting all identical and differing elements in individual groups; it also gives a number of the same elements in individual positions.

To run positioning of a structure model, follow the steps below:

- select a whole structure or a structure part in the structure model tree (a story or an element type – the element will be highlighted)
- click the right mouse button
- select the *Automatic positioning* command from the context menu.

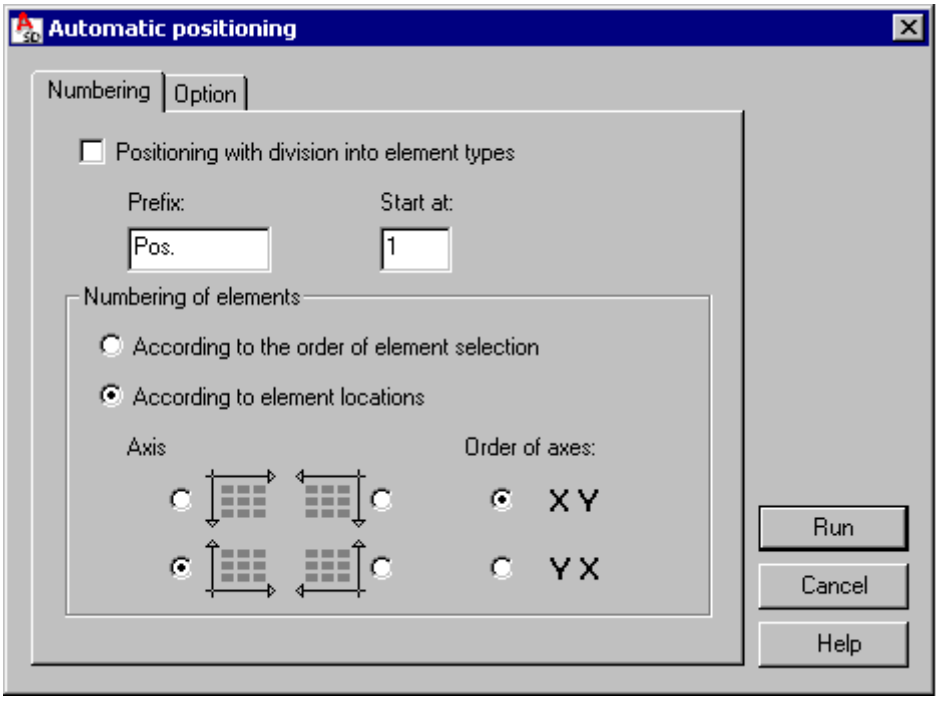


8.2. Automatic positioning

The option enables assigning positions automatically to defined structure elements. It is available from the context menu in the ***Object Inspector*** dialog box.
The operation of automatic assigning of a position is possible for a whole structure model or for its selected part (a story, an element type). Once positioning is run, the ***Automatic positioning*** dialog box appears on the screen; it consists of two tabs:
- *Numbering*
- *Options*.

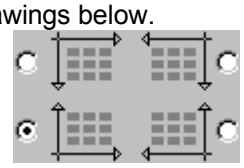
Pressing the **Run** button activates the operation of assigning positions automatically. Positioning is performed automatically for all selected structure elements.

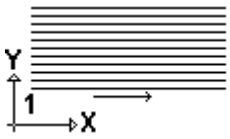
The *Numbering* tab

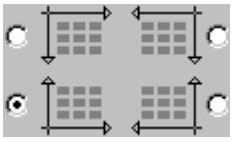


This tab holds the following options:

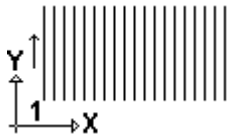
- *Positioning with division into element types*
  If this option is switched on, the ***Prefix*** field is inaccessible; names of positions will be assigned according to the syntax adopted for individual structure elements in the ***Job Preferences*** dialog box (the Position name option); there is a possibility to determine the start number of positions.
  If this option is switched off, the program makes accessible the ***Prefix*** field for defining a prefix for positions generated for all structure elements (regardless of an element type); all element types (beams, columns, walls, etc.) will be assigned names beginning with the character string specified in the *Prefix* field; there is a possibility to determine the start number of positions
- *Numbering of elements*
  If the *According to the order of element selection* option is switched on, then after pressing the **Run** button the user should indicate structure elements on the screen; the order in which they are indicated will be the criterion of assigning numbers to positions.
  If the *According to element locations* option is switched on, the *Axis* and *Order of axes* options are accessible; these options enable defining a direction of the element numbering on a story; examples of methods of the element numbering are presented in the drawings below.

1. *Axis*:  *Order of axes*: **X Y**
   Numbering of positions will run as follows



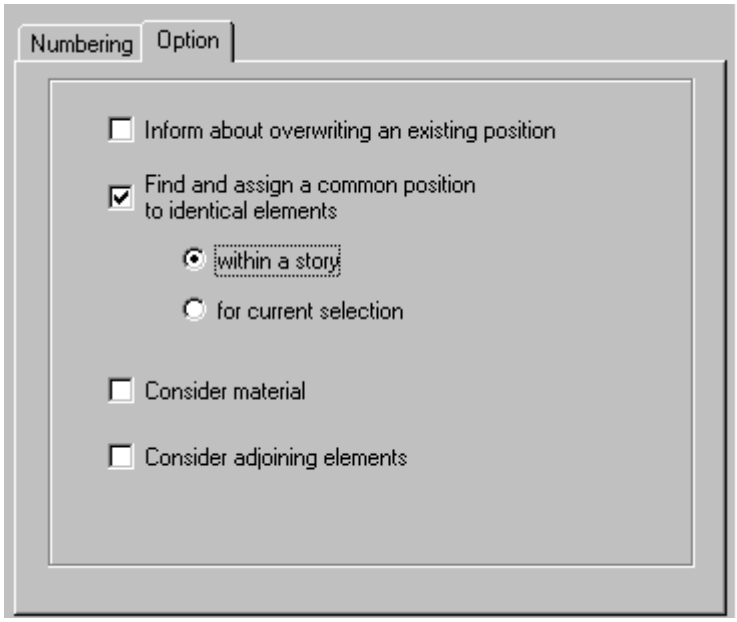
2. *Axis*:  *Order of axes*: **Y X**

Numbering of positions will run as follows



The criterion that determines the method of numbering of structure elements is the location of the element center (the midpoint in the length of a beam, column).

The *Options* tab



This tab holds the following options:

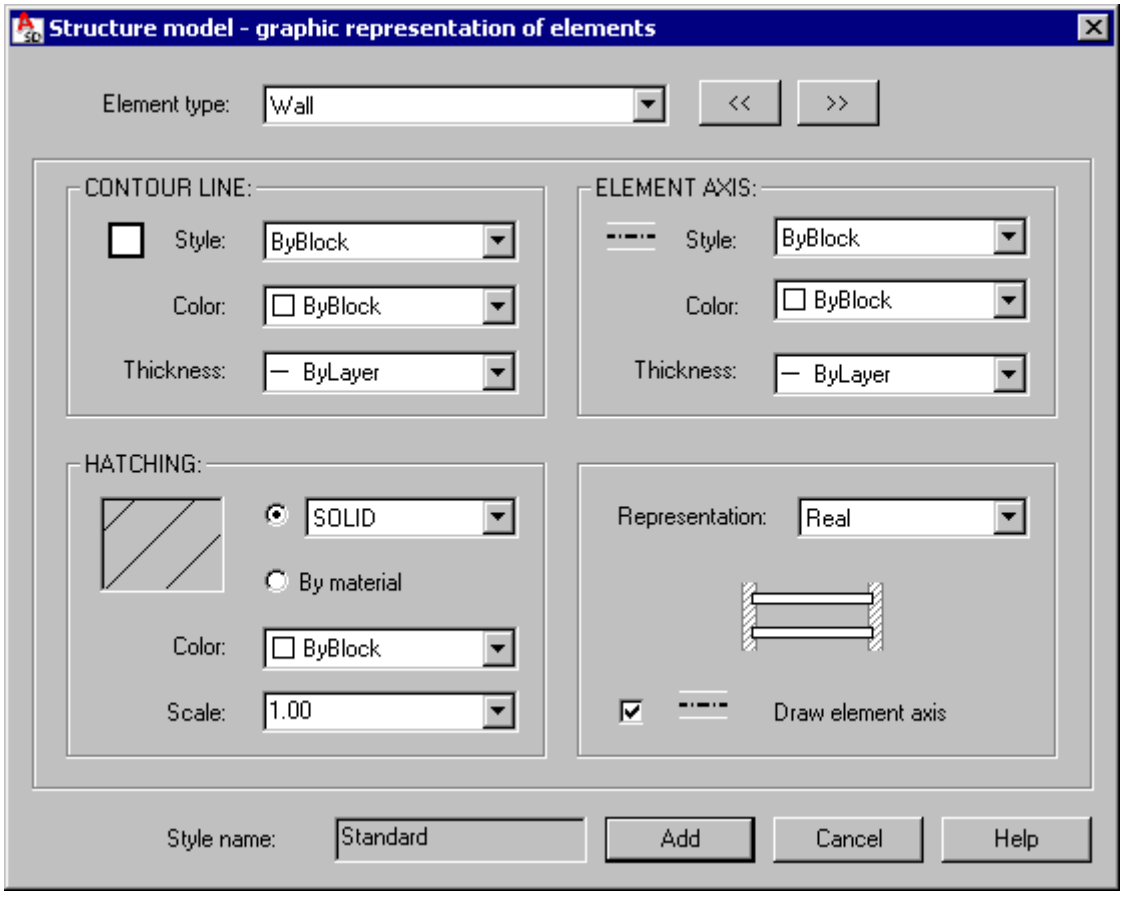
- *Inform about overwriting an existing position*
  If this option is switched on, while changing a name of a position that has been previously assigned a position, the program will display a question asking whether to overwrite the existing position
- *Find and assign a common position to identical elements*
  If this option is switched on, then geometrically-identical elements will be assigned the same position:
  - within a story
  - for current selection
- *Consider material*
  If this option is switched on, elements geometrically identical but made of different materials (e.g. an RC or brick column, a beam made of a different-class concrete), will be assigned different positions
  If this option is switched off, elements geometrically identical but made of different materials (e.g. an RC or brick column, a beam made of a different-class concrete), will be assigned the same positions
- *Consider adjoining elements*
  If this option is switched on, then geometrically-identical elements adjoined by elements with different geometrical dimensions (e.g. an RC column adjoining from top with a different section or no column from top for one of the columns), will be assigned different positions
  If this option is switched off, adjoining elements have no effect on definition of positions.

9. STYLES OF ELEMENT GRAPHIC DISPLAY

9.1. Graphic representation of structure elements

Graphic representation of structure elements determines how elements of a structure model should be presented on the screen (line thickness and type, colors, etc.). It may also be one of the methods of presenting plans of building stories (see also: Preferences / Drawings).
A method of presenting structure elements in drawings can be selected in the ***Graphic representation of elements*** dialog box shown in the drawing below. The dialog box opens from:

- the menu by selecting the option *Formwork Drawings / Styles / Styles – element graphic display*
- the command line RBCX_STYLE_GRAPHIC_PRESENTATION.



Contents of this dialog box depends on a type of structure element chosen from the selection list at the top of the dialog box.

For all types of structure elements the following fields are provided in the left-hand part of the dialog box:

- *Contour line*
  the following parameters are selected for the element contour line: line style, color and thickness
- *Hatching*
  the following parameters are selected for hatching:
  = line style or hatching based on the material
  = line color
  = line scale
  = angle of hatching.

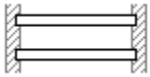
The right-hand part of the dialog box may include additional options for the following structure elements:

1. Wall, beam, continuous footing, ground beam

- E*lement axis*
  the option is available only if the *Draw element axis* option is switched on in the lower part of the dialog box; the following parameters may be selected for the element axis: line style, color and thickness
- *Element representation*

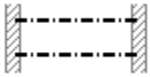
There are two possibilities of presenting a structure element in a drawing:

= real presentation



an element is represented in a drawing by a whole element contour (with the possibility of adding an element axis)

= symbolic presentation



an element is represented in a drawing only by element axes

2. Lintel, opening in the beam, opening/recess in the wall, opening/recess in the slab

- *Element axis*
  the option is available only if the *Draw element axis* option is switched on in the lower part of the dialog box; the following parameters can be selected for an element axis: line style, color and thickness.
- *Element representation*

There are two possibilities of presenting a lintel in a drawing:

= real presentation

= symbolic presentation - an element is represented in a drawing only by element axes

NOTE:

After selecting symbolic presentation of an opening/recess in wall another button - ***Details*** *is available in the dialog box.* When pressed, it opens the ***Opening symbol in top view*** dialog box where parameters of presenting an opening/recess in the drawing can be selected.

10. CREATING OF STRUCTURE VIEWS

10.1. Views / plans created for a structure model

Once definition and edition of a structure model is completed, it is possible to create automatically plans (projections) and views for individual parts of a building. The following operations can be performed on the created views / plans:

- inserting of dimension lines (see methods of defining dimension lines: for single elements and group dimensions)
- inserting of elevation marks (see description of available elevation mark styles)
- description of structure elements (see description of available element description styles).

After the structure model is defined, the user may launch an option that allows assigning a description to individual structure objects. NOTE: Descriptions of structure elements can be placed in plans (of stories or foundations) and in sections.
The following plans / views of a building can be created in the current version of ***AutoCAD © Structural Detailing - Formwork Drawings***:
- plan of a building story
- plan of a building foundation
- vertical section of a building
- elevation view of a building
- 3D view of a building.

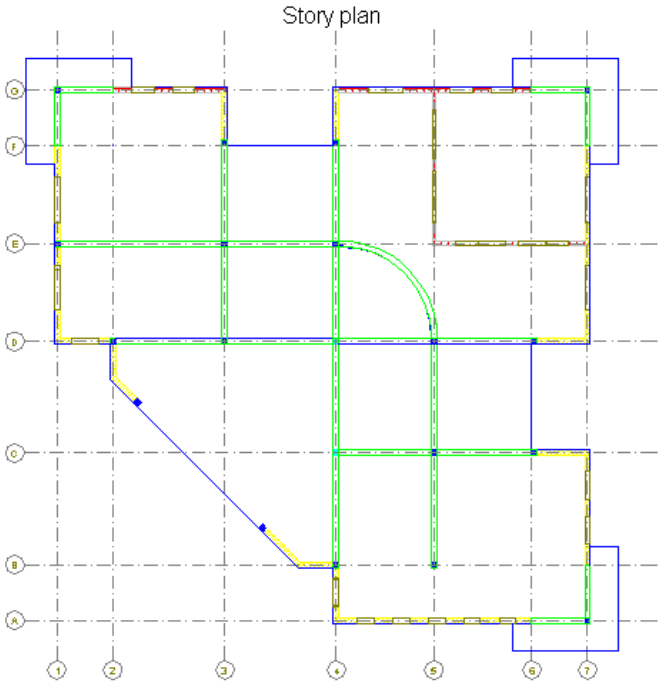
The way elements of a structure model are presented in plans (projections) or views depends on the settings chosen in the ***Job Preferences*** dialog box (the *Drawings* tabs).

10.2. Creating of a plan (projection) of a building story

Once edition of a structure model is completed, it is possible to create automatically a projection of a selected story. To do it, follow the steps below:

- select a story in the tree located on the *Model* tab in the ***Object Inspector*** dialog box (the story will be highlighted)
- press the right mouse button and from the context menu choose the *Create story plan* option or press the  icon.

A new edition layout is generated at the bottom of the screen. The generated projection (plan) of a story is placed in the tree on the *Positions* tab in the ***Object Inspector*** dialog box (a name of the plan may be defined as e.g. *Story plan - Story-1*).
After indicating the name of a story plan, pressing the right mouse button and choosing the *Activate* option from the context menu, a drawing of the required story plan is created in the graphic edition field.

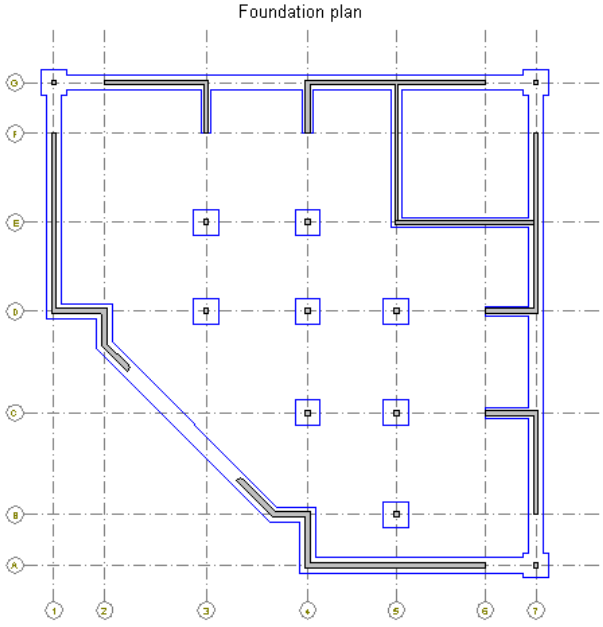


10.3. Creating of a plan (projection) of foundations

Once edition of a structure model is completed, it is possible to create automatically a projection of structure foundations. To do it, follow the steps below:

- select a story in the tree located on the *Model* tab in the ***Object Inspector*** dialog box (the story will be highlighted)
- press the right mouse button and from the context menu choose the *Create plan of foundations* option or press the icon.

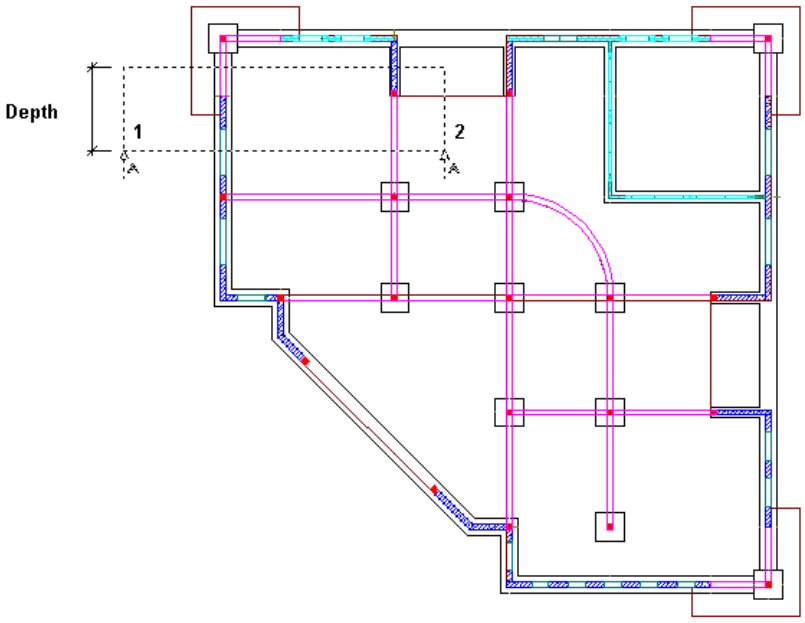
A new edition layout is generated at the bottom of the screen. The generated projection (plan) of foundations is placed in the tree on the *Positions* tab in the ***Object Inspector*** dialog box (a name of the plan may be defined as e.g. *Foundation plan*).
After indicating the name of a foundation plan, pressing the right mouse button and choosing the *Activate* option from the context menu, a drawing of the foundation plan is created in the graphic edition field.



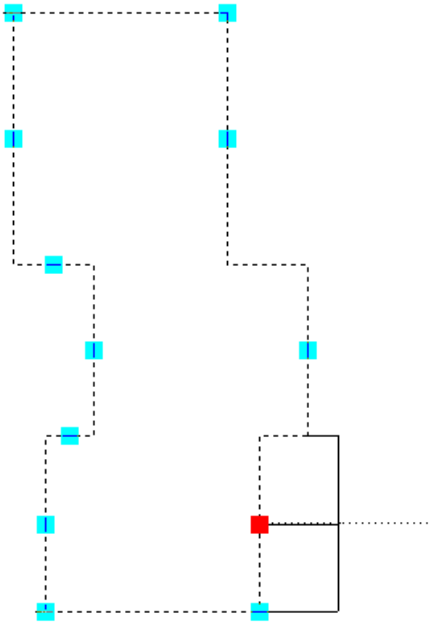
10.4. Creating of a vertical section of a building

Once edition of a structure model is completed, it is possible to create a vertical section of the structure model at the place indicated by the user. To do it, select the option from the menu *Formwork Drawings / Create vertical section* or press the icon.

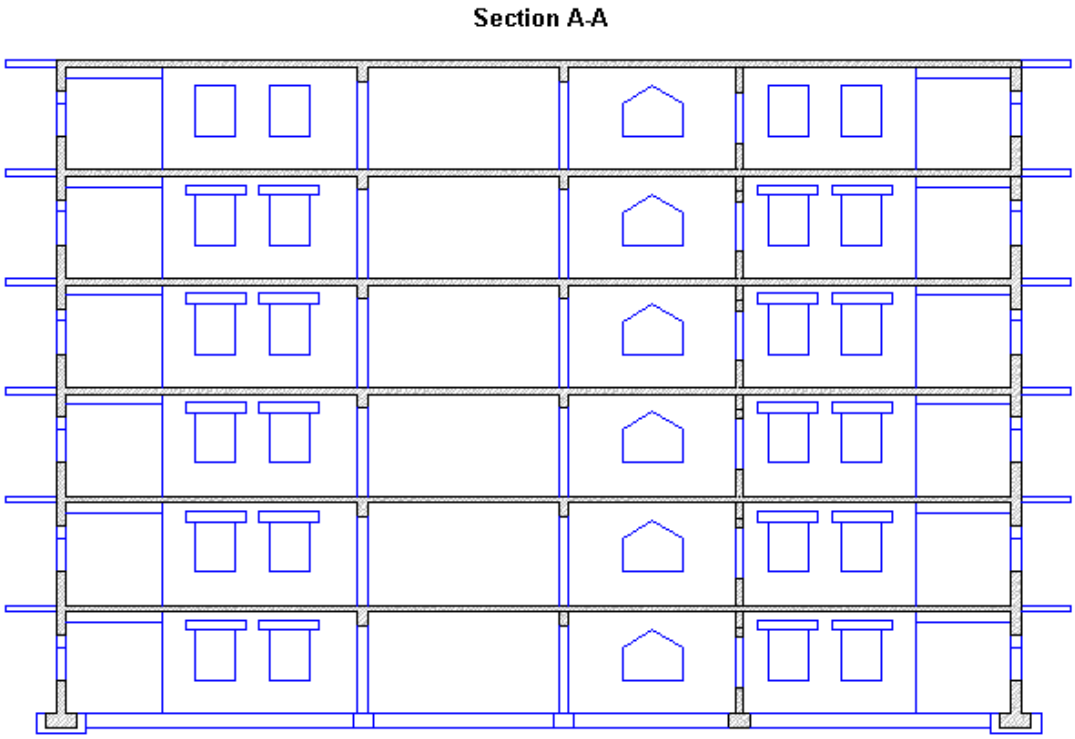
On the screen the user should indicate lines intersecting through the structure model and the depth (points defining the intersecting line and the 'depth' of the view are illustrated in the drawing below); once they are indicated, a vertical section is created in the edition layout.



NOTE:
It is possible to define independently a depth for each segment of a broken line which determines a 'cut' of a structure model. It can be done by changing a position of the line determining a depth of the view for a selected segment of a broken line (see the drawing below).

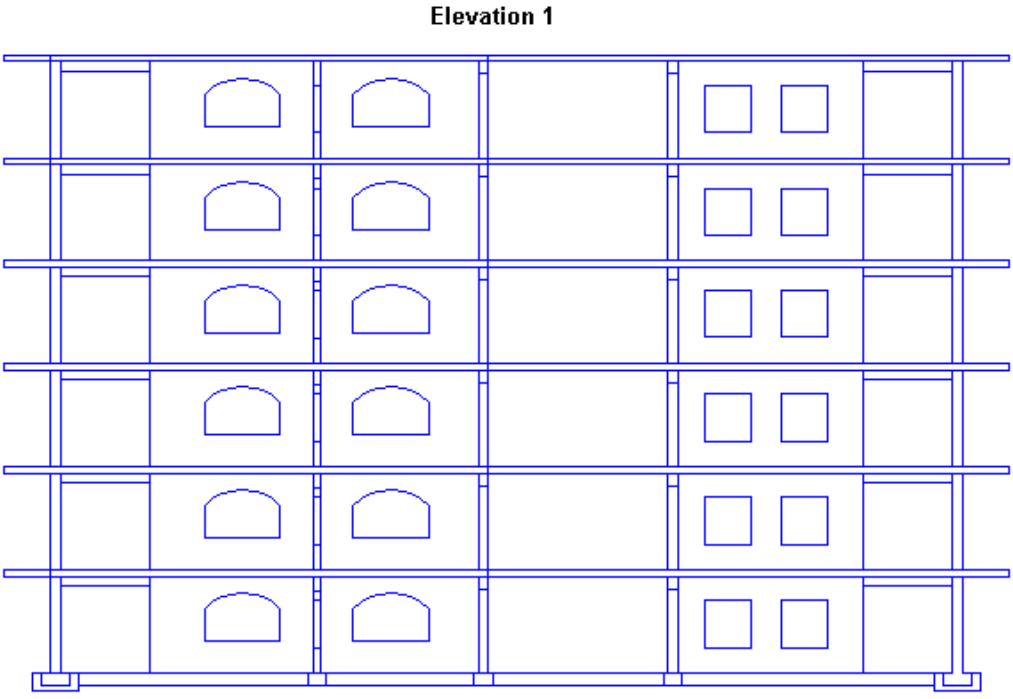


The generated vertical section, the example of which is shown in the drawing below, is placed in the tree on the *Positions* tab in the ***Object Inspector*** dialog box.



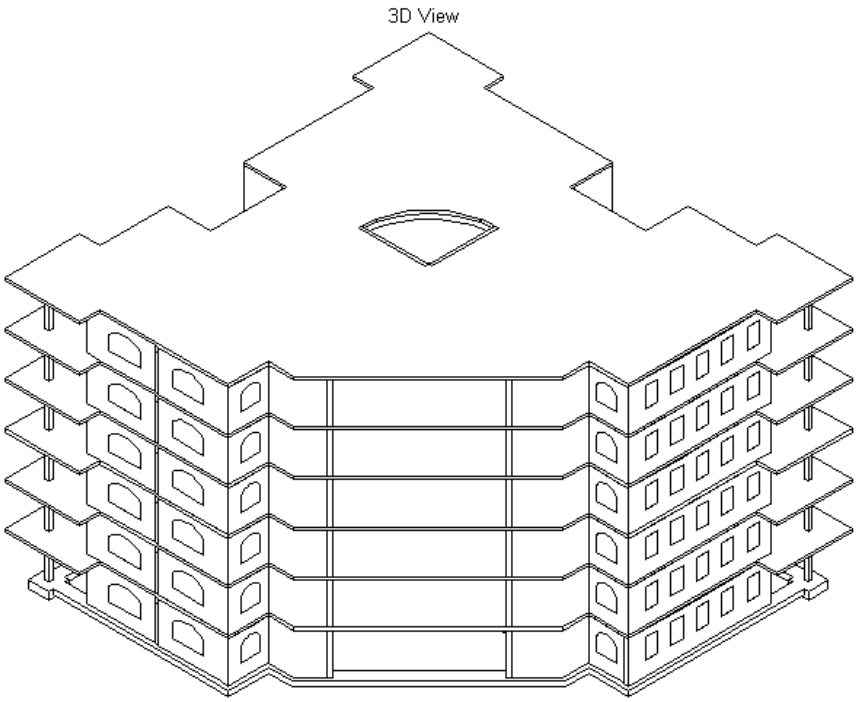
10.5. Creating of an elevation view of a building

Once edition of a structure model is completed, it is possible to create an elevation view of the structure model at the place indicated by the user. To do it, select the option from the menu *Formwork Drawings / Create elevation view* or press the icon.
On the screen the user should indicate a line of the elevation view (beginning and end points); once they are specified, the elevation view is created in the edition layout.
The generated elevation view is placed in the tree on the *Positions* tab in the ***Object Inspector*** dialog box.



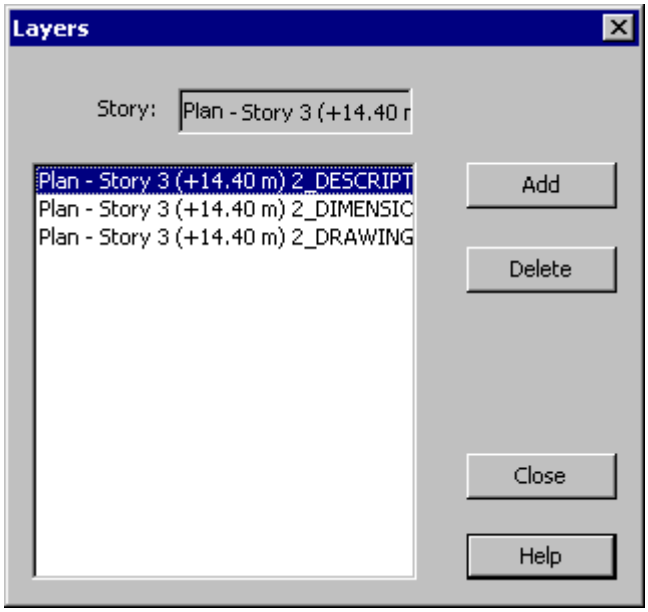
10.6. Creating of a 3D view of a building

Once edition of a structure model is completed, it is possible to create automatically a three-dimensional view of the structure model. To do it, select the option from the menu *Formwork Drawings / Create 3D view* or press the icon. On the edition layout the program creates the 3D view of the structure model made from the current structure view on the *Model* tab.
The generated 3D view is placed in the tree on the *Positions* tab in the ***Object Inspector*** dialog box.



10.7. Layers - Drawing positions

The dialog box shown below is used to define a layer in drawings (e.g. a layer of an architectural project of a structure).

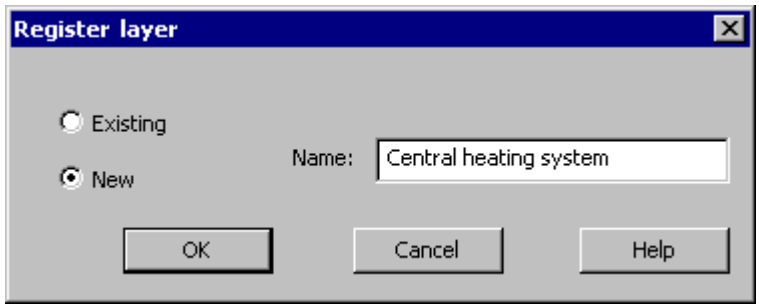


In the dialog box concerning layers in a drawing there are three standard layers:
- layer concerning a drawing
- layer concerning dimensions of drawing elements
- layer concerning descriptions of drawing elements and elevation marks.

The user may add any layer to those layers listed above. To add a layer to a given story (the story name is presented in the top part of the dialog box), the **Add** button should be pressed. Then the ***Register layer*** dialog box opens.
Pressing the **Delete** button deletes a selected layer from the list of layers available for a given story.

10.8. Register layer - Drawing positions

The dialog box shown below is used to register a layer in drawings (e.g. a layer of an architectural project of a structure).



There are two methods of defining / registering a layer:
- *Existing* - from the list of defined layers the user should select the name of a layer to be added; NOTE: the layer must be added earlier using the AutoCAD © options
- *New* - after selecting this option, the user should specify a layer name.

11. DESCRIPTION OF STRUCTURE ELEMENTS

11.1. Element descriptions

Once definition and edition of a structure model is completed, it is possible to describe elements of a structure model. NOTE: Descriptions of structure elements can be inserted in plans (of stories or foundations) and in sections.

To add a description of a structure element select the *Element description* option (from the menu – the option *Formwork Drawings / Element description* or the Abc icon), and next, indicate objects to be described. It can be performed either by clicking on a chosen element or by selecting a group of elements with a window. The latter method enables describing different types of elements (the program on its own recognizes different elements – columns, beams, spread footings, etc. - and describes them automatically).

After indicating elements, the program provides them with descriptions; the user should choose a location of a description at a selected place in a drawing.

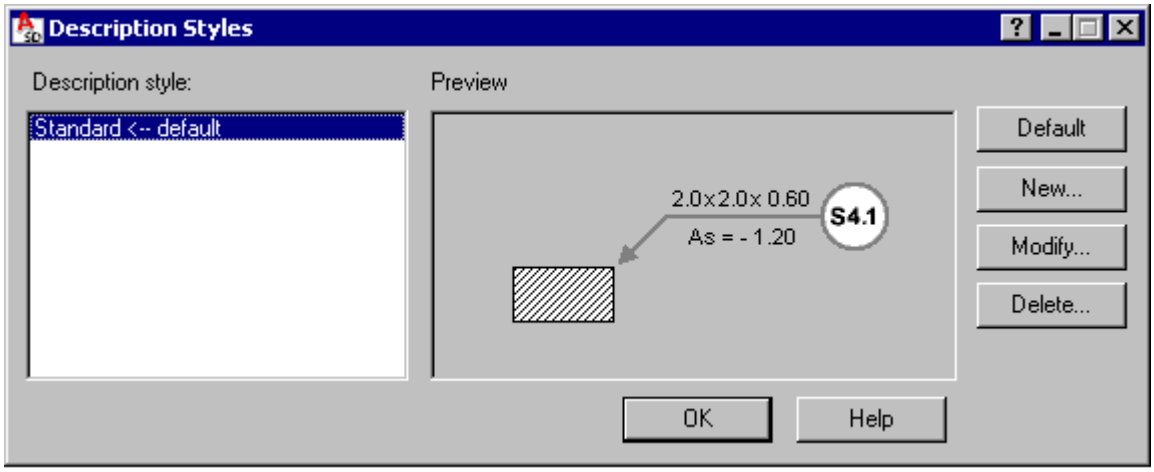
11.2. Description styles

11.2.1. Styles of element description

After the structure model is defined, the user may launch an option that allows assigning a description to individual structure objects. NOTE: Descriptions of structure elements can be inserted in plans (of stories or foundations) and in sections.

Styles of element description may be defined using the options provided in the ***Description Styles*** dialog box presented in the drawing below. The dialog box is opened:

- from the menu by selecting the option Formwork Drawings / Styles / Element description styles
- from the command line: RBCX_STYLE_LABEL.



The *Description style* field includes styles defined for a selected description of structure elements.

The *Preview* field presents parameters set graphically for an indicated style of the structure element description.

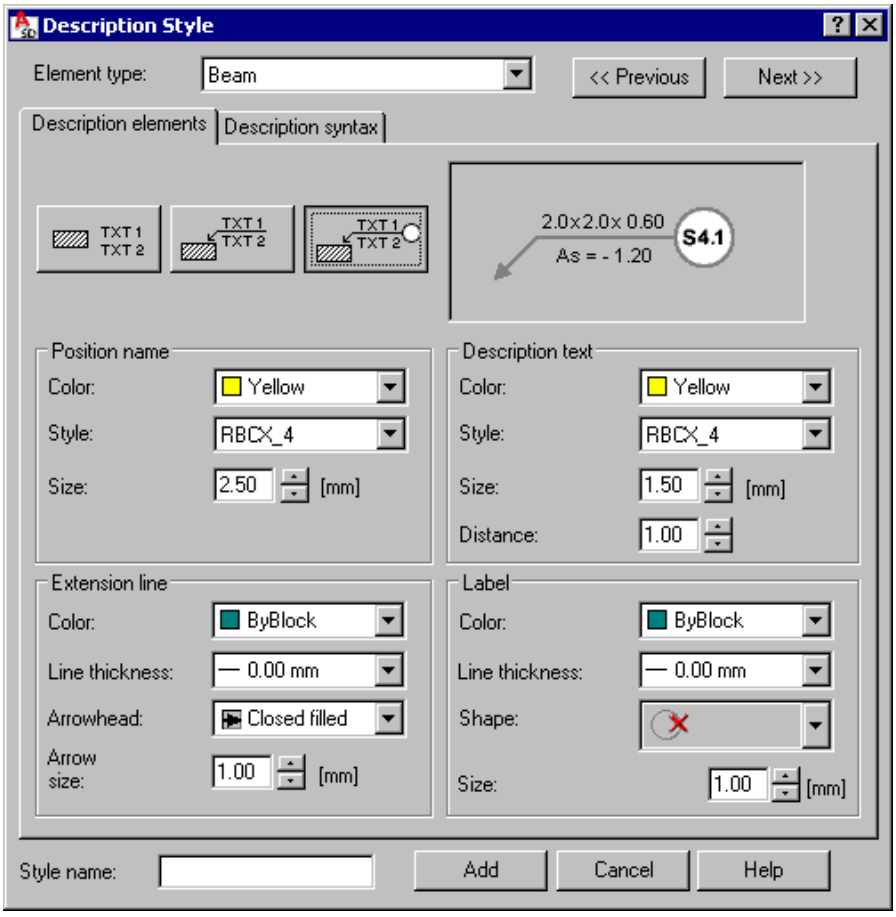
The right part of the dialog box contains the following buttons (apart from the standard ones **OK**, **Cancel** and **Help**):

- **Default** – pressing this button selects a default (standard) style of the structure element description
- **New** - pressing this button opens the ***Description Style*** dialog box where a new style of the structure element description may be defined

- **Modify** - pressing this button opens the ***Description Style*** dialog box where a selected type of the structure element description may be modified
- **Delete** - pressing this button deletes a highlighted style of the structure element description from the list of styles available in the *Description style* field.

11.2.2. Description style - definition

The dialog box is used to define a style of the structure element description.
The ***Description style*** dialog box opens on pressing the **New** or **Modify** button in the ***Description Styles*** dialog box. A style name may be specified in the *Style name* field located at the bottom of the dialog box (while modifying a style, the *Style name* field is not accessible; it shows a name of a selected style of the element description).



A defined or modified style includes settings of a description of all available structure elements (beams, columns, walls, slabs, etc.).
The upper part of the dialog box contains a list of available structure elements; the following elements are available in the current program version: beam, column, wall, slab, raft foundation, spread footing, continuous footing, ground beam, stairs, lintel, door, window, opening/recess in the wall, opening/recess in the slab, opening in the beam. An element on the list can be changed by:

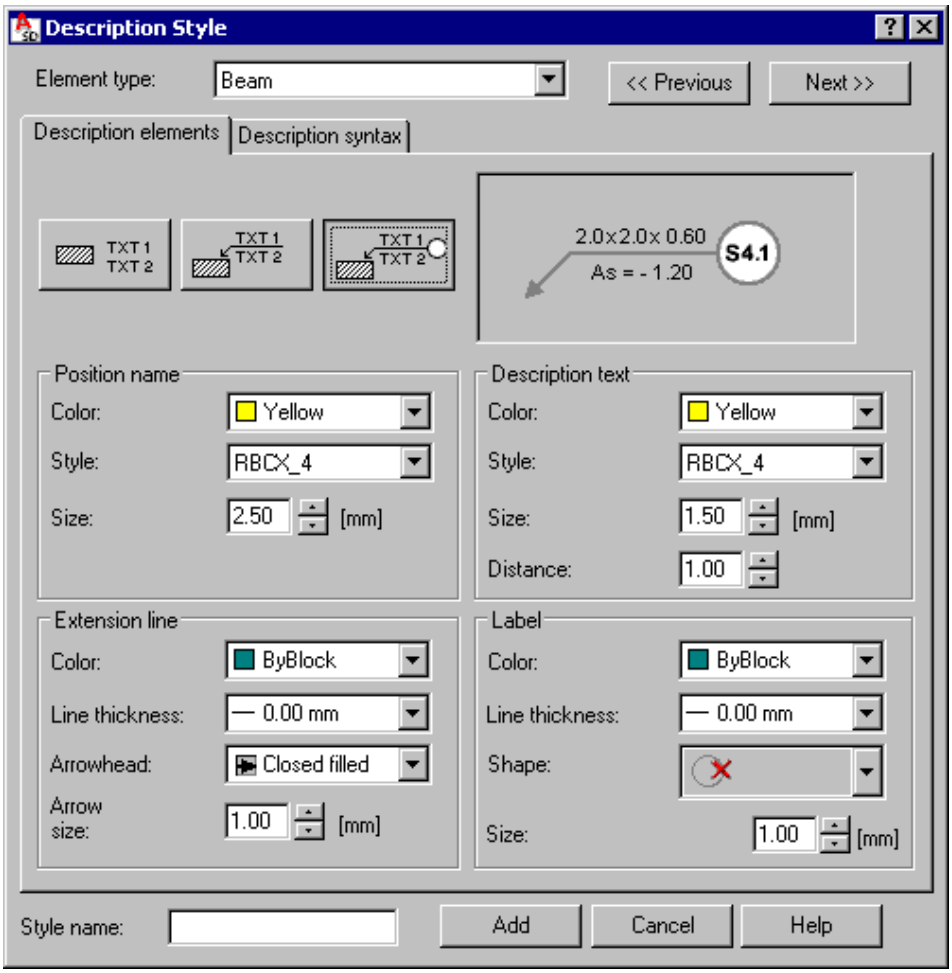
- selecting an element from the list
- pressing the button **Next >>** (**<< Previous**) which results in selecting the next (or previous) element on the list.

The ***Description Style*** dialog box consists of the following two tabs:
Description elements
Description syntax.

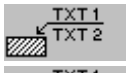
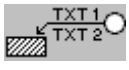
To add a new element description style, press the **Add** button after defining description parameters.

11.2.3. Description elements

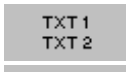
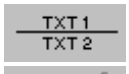
The Description Style dialog box looks as presented in the drawing below after selecting the *Description elements* tab in it.



The first tab is used to define main settings (colors, line thickness, size and style of fonts, etc.) of the element description.
There are three methods of element description:
- for all elements except slabs and raft foundations

1.  Element description as multiline text
2.  Element description as text on an extension line
3.  Element description as text on an extension line + label.

- for slabs and raft foundations:

1.  Element description as multiline text
2.  Element description as multiline text on the diagonal of a slab
3.  Description in a label positioned on the diagonal of a slab

The options located in the *Position name* field concern parameters of a position name from Object Inspector. The following parameters of an element name can be set here: font style, color and size of a description.

The options provided in the *Description text* field refer to structure element description. The following parameters can be set here: font style, color and size of description as well as distance to the auxiliary line

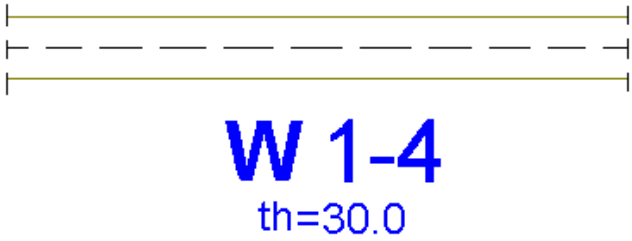
The options in the *Extension line* field allow defining the following parameters of a line connecting a structure element description with a structure element:
- for beams, columns, walls, spread footings and continuous footings:
  color, thickness and arrowhead of the line (apart from that, the arrow size should be determined)
- for slabs:
  color and thickness of the line.

The options in this field are available if 2nd or 3rd method of the element description has been chosen.

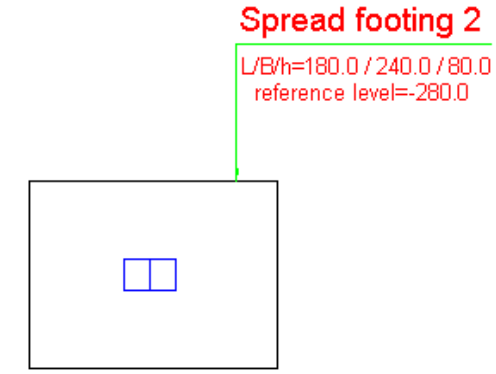
The options in the *Label* field allow defining the following parameters of the label positioned on the end of the line connecting a structure element description with a structure element: color, line thickness, label shape and size of a description in the label.
The options in this field are available if 3rd method of the element description has been chosen.

Example of descriptions are illustrated in the drawings below. Take note that a position name has other text parameters (larger font) than the description text.

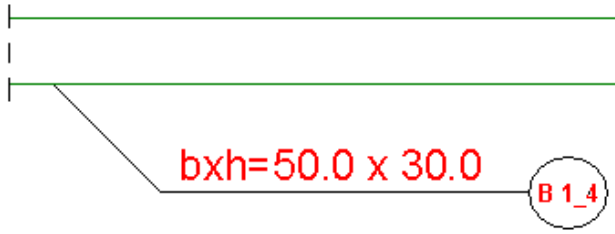
Description of a wall with the use of 1st description method



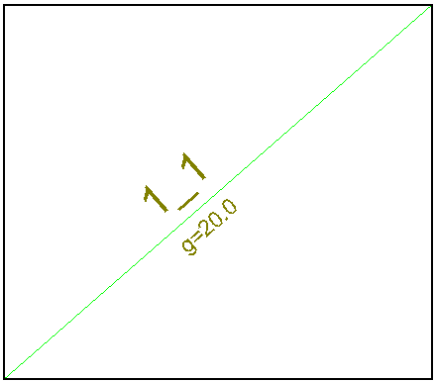
Description of a spread footing with the use of 2nd description method



Description of a beam with the use of 3rd description method

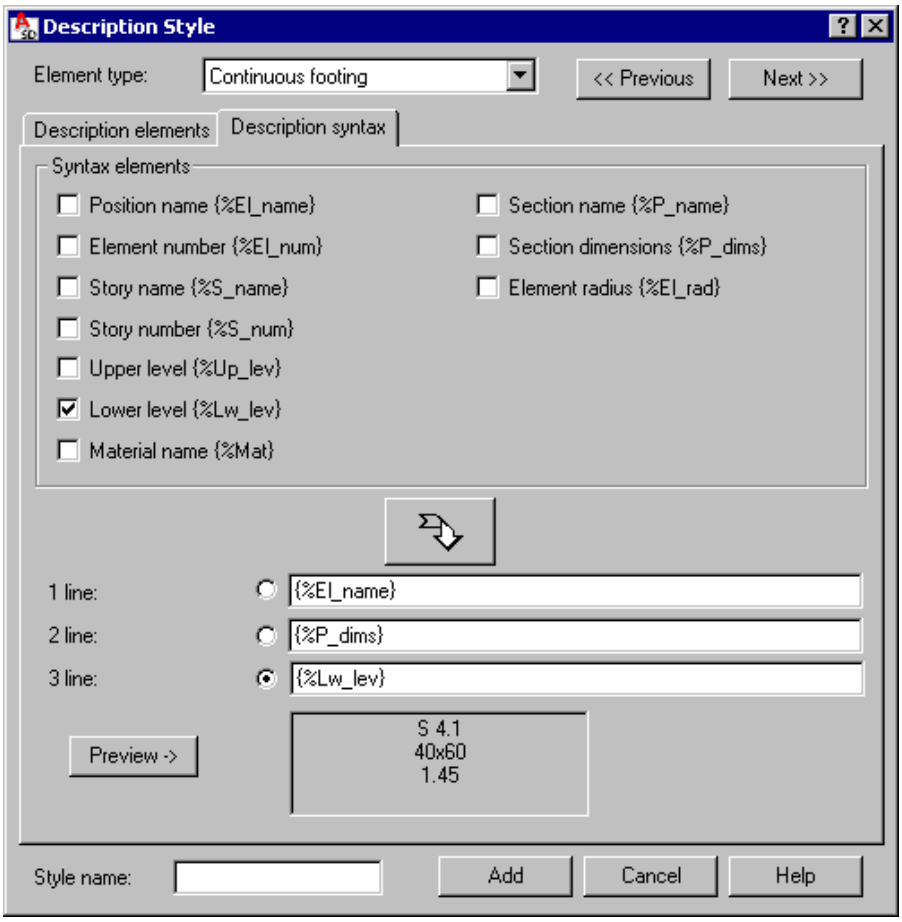


Description of a slab with the use of 2nd description method



11.2.4. Description syntax

The Description Style dialog box looks as presented in the drawing below after selecting the *Description syntax* tab in it.



The options provided on this tab are used to define a syntax of an element description; along with a defined general style of the element description the above dialog box offers a mechanism for freely composing of a syntax and contents of the element description.
By switching on appropriate description elements included in the *Syntax elements* field they can be moved to the fields*: 1 line, 2 line, 3 line* (pressing the arrow in the middle of the dialog box), where it is possible to arbitrarily arrange variables with user-defined texts, symbols, etc. An example description: S 4.1 bxh = 40x60.
A description may consists of three parts, therefore, three edit fields are available. After pressing the arrow, selected description elements are moved to an active edit field.
The *Preview* field shows the element description resulting from a defined syntax. This description is based on numerical values saved as fixed ones and responds to changes in preferences (change of unit, precision). NOTE: a unit is not displayed in a preview.

If it is necessary to define more than three description lines, then in the last line text may be broken by using the symbol \P. This is a standard symbol used in the AutoCAD © program (see also AutoCAD program mechanisms).
The list of variables included in the element description is as follows:

%El_name	- position name or opening name
%El_num	- element number
%S_name	- story name
%S_num	- story number
%Up_lev	- upper level
%Lw_lev	- lower level
%Mat	- material name
%P_name	- section name
%P_dims	- section dimensions
%El_rad	- element radius
%El_h	- element height (refers to a column and a wall)
%El_th	- element thickness (refers to a wall and a slab)
%P_dims	- dimensions of a spread footing or dimensions of an opening
%S	- height of the sill (refers to openings).

 *NOTE:*
Variables that may be applied in a syntax, must be put in parentheses {}; the user may add any texts between the successive variables put in these parentheses.

11.2.5. User description - mechanisms of the AutoCAD program

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 the formatting, the 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.

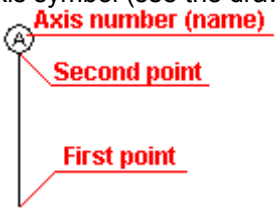
12. GRAPHIC SYMBOLS

12.1.How to insert graphic symbols

The following graphic symbols can be defined in the program:

- axis symbol
- elevation mark
- section symbol
- opening symbol.

To insert the axis symbol in a drawing, follow the steps below:
1 select the command *Formwork Drawings / Insert structural axis* or press the icon
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).



A number of the axis is proposed according to the settings in the default style; while inserting the axis the user may enter any number (every following number will be inserted according to the recently-specified numbering). The axis number can be modified using the relevant option from the context menu.

To insert the elevation mark in a drawing, follow the steps below:
1 select the command *Formwork Drawings / Insert elevation mark* or press the icon
2 indicate a point on a selected level (the program will calculate automatically the height with respect to the base level).
Levels are designated with respect to the base level determined in the ***Building parameters*** dialog box.

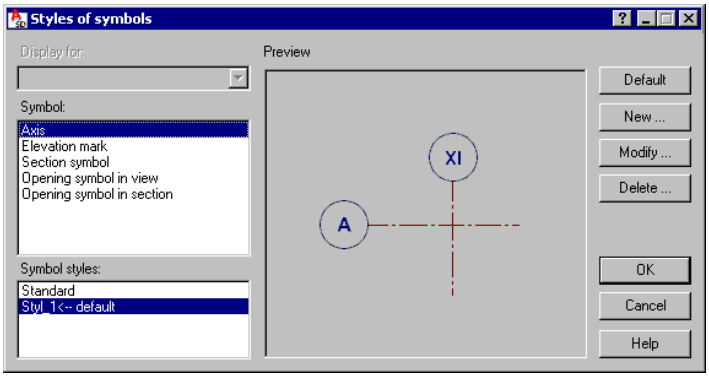
12.2.Styles of graphic symbols

12.2.1. Styles of symbols

The option allows defining / modifying symbols of axes, levels and sections used in elements of an RC structure. The option is accessible from:

- the menu by selecting the option Formwork Drawings / Styles / Styles – graphic symbols
- from 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 structure elements (presented in the *Symbol* field in the above dialog box):

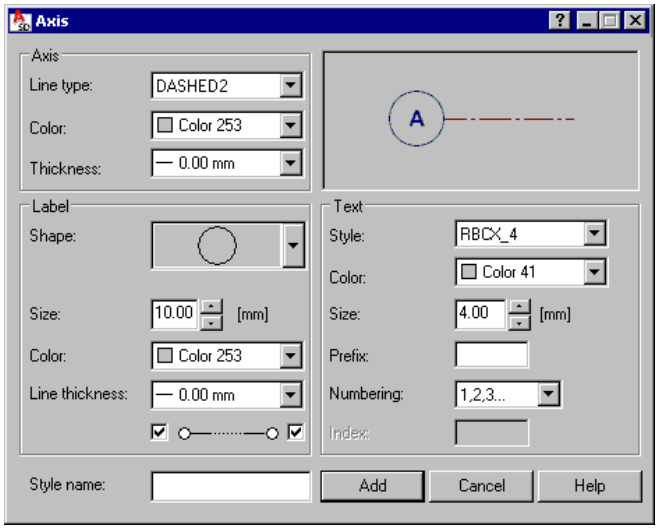
- axis symbol
- elevation mark
- section symbol
- opening symbol in view
- opening symbol in cross-section.

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 part of the dialog box contains the following buttons (apart from the standard ones **OK**, **Cancel** and **Help**):

- **Default** - pressing this button restores a default (standard) symbol of an axis, level or section
- **New** - pressing this button opens one of the dialog boxes: Axis, Section symbol, Opening symbol in view, Opening symbol in cross-section or Elevation Mark 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, Section symbol, Opening symbol in view, Opening symbol in cross-section or Elevation Mark 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.

12.2.2. 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 **New** or **Modify** button in the Styles of symbols dialog box if an axis style is chosen in the *Symbol* field.

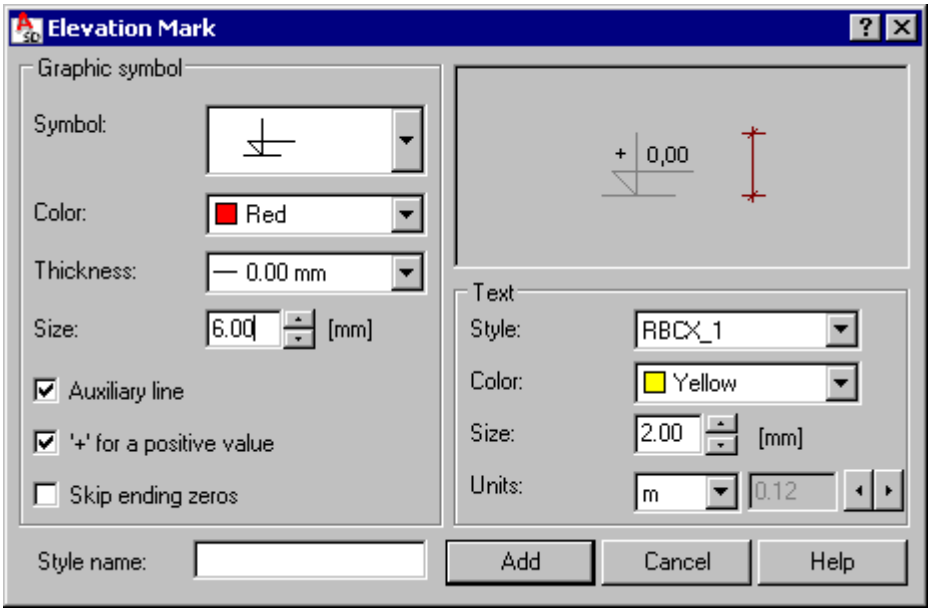


The Axis field holds parameters of the line presenting an axis in a drawing: line type, color and thickness.
The *Label* field allows determining parameters of a label of an axis description: label shape, size, color and thickness of the label line; the following label shapes are available in the program: circle, ellipse, square, octagon. At the bottom of this field there are options that allow switching on / off the display of a label at both ends of the axis.
The options in the *Text* field refer to the axis description provided in the label. The following parameters can be set here: font style, color and size; additionally, it is possible to choose a prefix of the axis 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.

12.2.3. 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 **New** or **Modify** button in the Styles of symbols dialog box if the elevation marks style is chosen in the *Symbol* field.



The *Graphic symbol* field holds parameters of a symbol representing a level in the drawing: symbol type, symbol color, symbol size and thickness of the line in the symbol. Below are the following options:

- *Auxiliary line* – if this option is switched on, then apart from the elevation mark, the drawing will include an auxiliary line connecting the elevation mark with the level of an RC structure element; when the option is switched off, the elevation mark will be shown without any additional lines
- *'+' for a positive value* - if this option is switched on, then the '+' symbol will appear additionally when the value of a number presented in the elevation mark is greater than zero; if this option is switched off, a positive value will be presented in the elevation mark without any additional symbol
- *Skip ending zeros* - if this option is switched on, then if there are zeros (one or more) in decimal places at the end of a number, these zeros will be ignored (e.g. the number 12,23 will be displayed instead of the number 12.2300).

The options in the *Text* field refer to the level description provided in the elevation mark. The following parameters can be set here: font style, color and size; additionally, it is possible to select a unit used to present the level value and the number precision (a number of decimal places).

To add a new elevation mark, specify a name of the elevation mark description style (in the *Style name* field) and press the **Add** button.

12.2.4. Section symbol

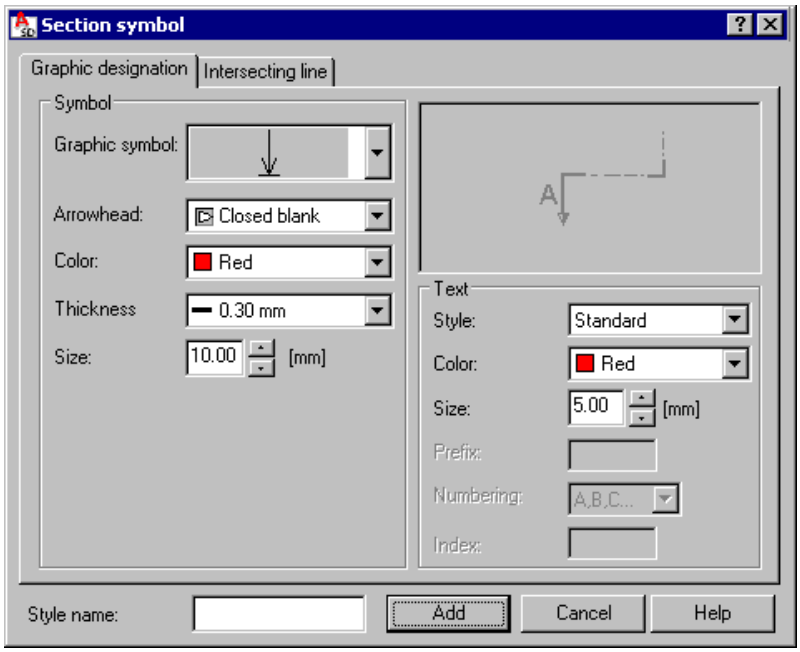
The dialog box is used to define a new style or to modify an existing style of a section through the RC structure element. It opens on pressing the **New** or **Modify** button in the Styles of symbols dialog box if a section symbol style is chosen in the *Symbol* field.

The dialog box consists of the two tabs:
Graphic designation
Intersecting line.

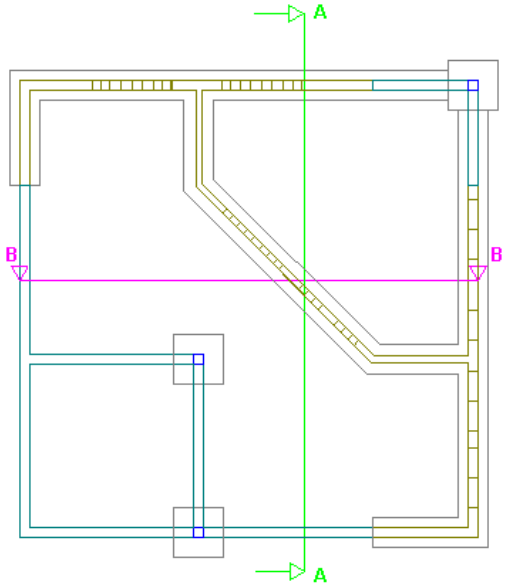
To add a new section symbol, specify a name of the section symbol style (in the *Style name* field) and press the **Add** button.

12.2.5. Graphic designation

The Section symbol dialog box looks as presented in the drawing below after selecting the *Graphic designation* tab in it.



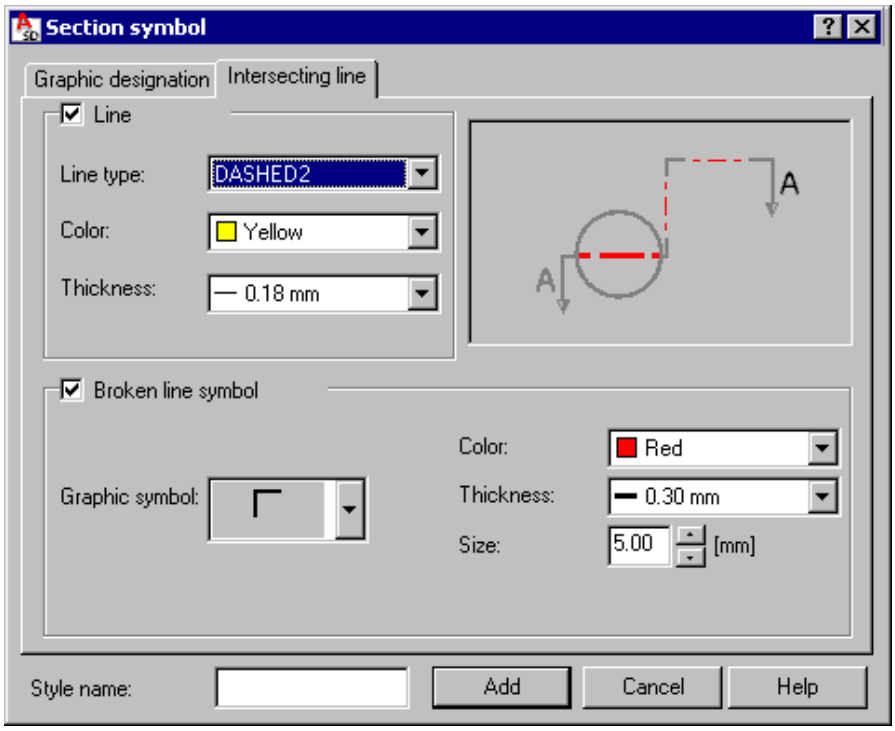
The *Symbol* field holds parameters of a symbol representing the section of a structure element in a drawing: graphic symbol type, arrowhead of the section symbol, symbol color, size and thickness of a line in the symbol; besides, it is possible to select the numbering of section symbols: with letters: A, B, C, with numerals: 1, 2, 3, with numerals: I, II, III or by determining any other designations (after selecting the *Define* option). An example of a section symbol is shown in the drawing below.



The options provided in the *Text* field refer to a section description. The following parameters can be set here: font style, color and size.

12.2.6. Intersecting line

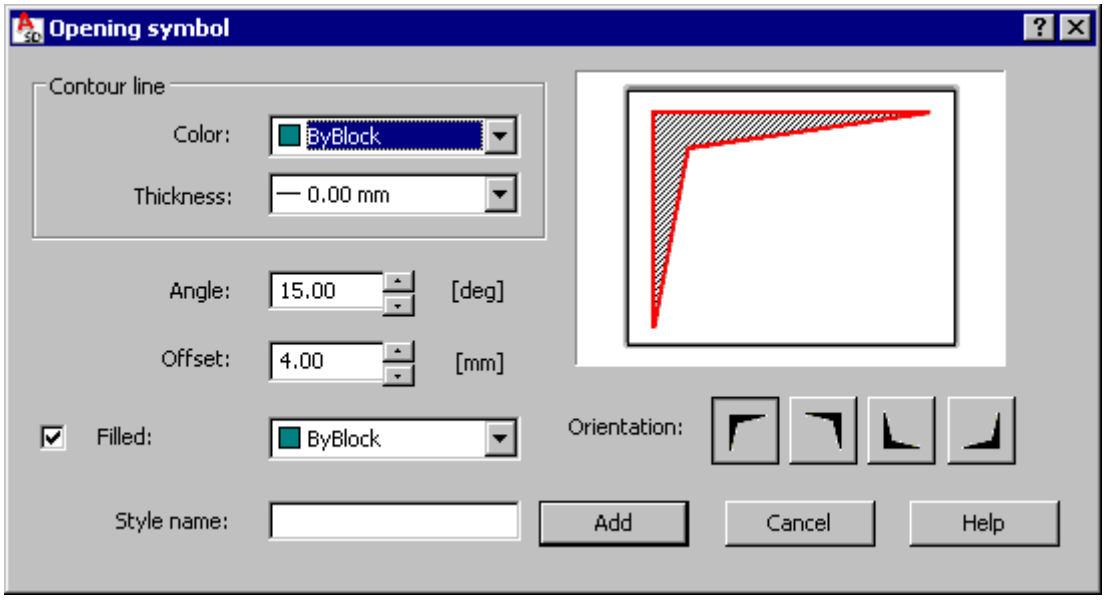
The Section symbol dialog box looks as presented in the drawing below after selecting the *Intersecting line* tab in it.



Once the *Line* option is switched on, the options for definition of parameters of an intersecting line (cross-section of a structure model) become available. The following parameters can be set here: line type, color and thickness.
In the lower part of the dialog box there is the *Broken line symbol* option; if it is switched on, parameters of the symbol which denotes 'breaking' the intersecting line (when defining an intersecting line by means of a polyline) can be specified. The following parameters of a broken line symbol can be defined:
- type of a broken line symbol
- color, thickness and size of a broken line symbol.

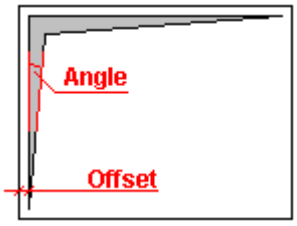
12.2.7. Opening symbol in view

The dialog box is used to define a new style or to modify an existing style of an opening symbol in view. It opens on pressing the **New** or **Modify** button in the Styles of symbols dialog box if a style of an opening symbol in view is chosen in the *Symbol* field.



The *Contour line* field holds parameters of a line presenting the contour of an opening symbol in a drawing: color and thickness of the line.
Below the following parameters can be determined:

- value of the *angle* for a defined symbol - see the drawing below
- value of the *offset* for a defined symbol - see the drawing below

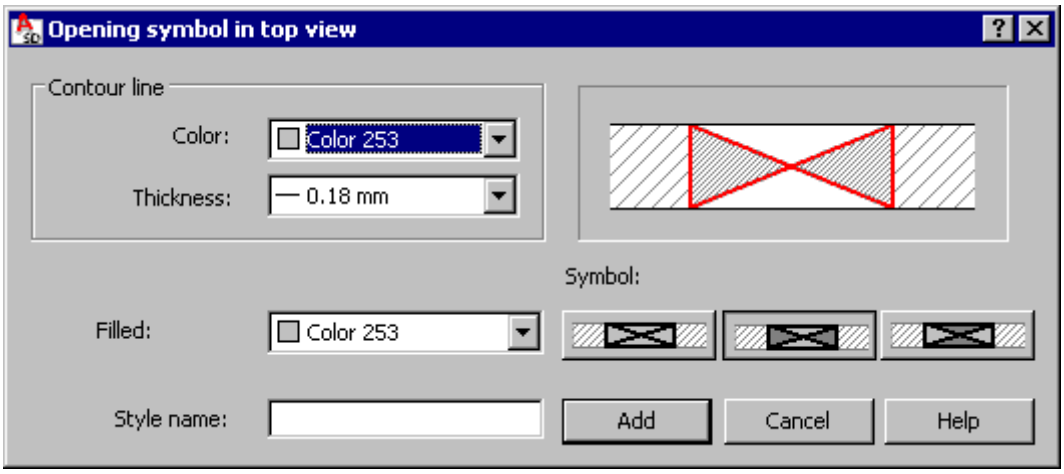


- *filled;* filling of an opening symbol can be defined if the option is switched on (the √ symbol appears to the left of the option name); a filling type is chosen from the list
- *orientation;* four positions of an opening symbol are possible:  ,  ,  and  .

To add a new opening symbol, specify a name of the opening symbol style (in the *Style name* field) and press the **Add** button.

12.2.8. Opening symbol in cross-section

The dialog box is used to define a new style or to modify an existing style of description of openings in a top view. The dialog box opens on pressing the **New** or **Modify** button in the Styles of symbols dialog box if a style of an opening symbol in cross-section is chosen in the *Symbol* field.
The dialog box below also opens after selecting symbolic presentation of an opening/recess in the wall and pressing the **Details** button in the ***Graphic representation of element*** dialog box.



Three types of presentation of an opening in wall are available in the program:

- 
- 
-  .

For each of the listed presentation types the following parameters can be defined:

- parameters of a contour line (color and thickness)
- filling of an opening symbol.

To add a new description of an opening in top view, the user should specify a name of the style of an opening symbol in view (in the *Style name* field) and press the **Add** button.

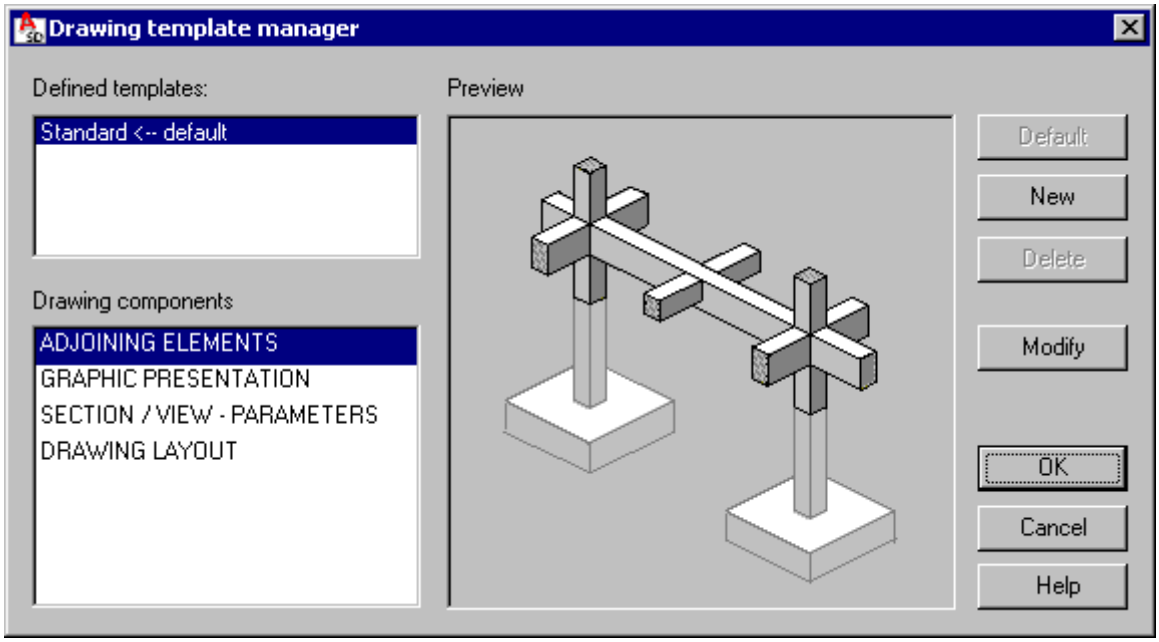
13. STYLES OF DRAWING TEMPLATES

13.1. Drawing template manager

Drawing template manager enables the user to determine a method of creating formwork drawings of elements of a building structure. It includes defining parameters of appearance of individual structure elements (colors, scale, plans, etc.); in this way drawing templates for each type of element are created.

NOTE:
All parameters defined in this dialog box refer to the link between the programs ***AutoCAD © Structural Detailing - Reinforcement*** *and* ***AutoCAD © Structural Detailing - Formwork Drawings****.*
Parameters of drawings generated in ***AutoCAD © Structural Detailing - Formwork Drawings*** *can be set in the* ***Job Preferences*** *dialog box.*

The ***Drawing template manager*** dialog box can be opened from:
- from the menu by selecting the option Formwork Drawings / Styles / Styles - drawing templates
- from the command line: RBCX_STYLE_DRAWINGS.



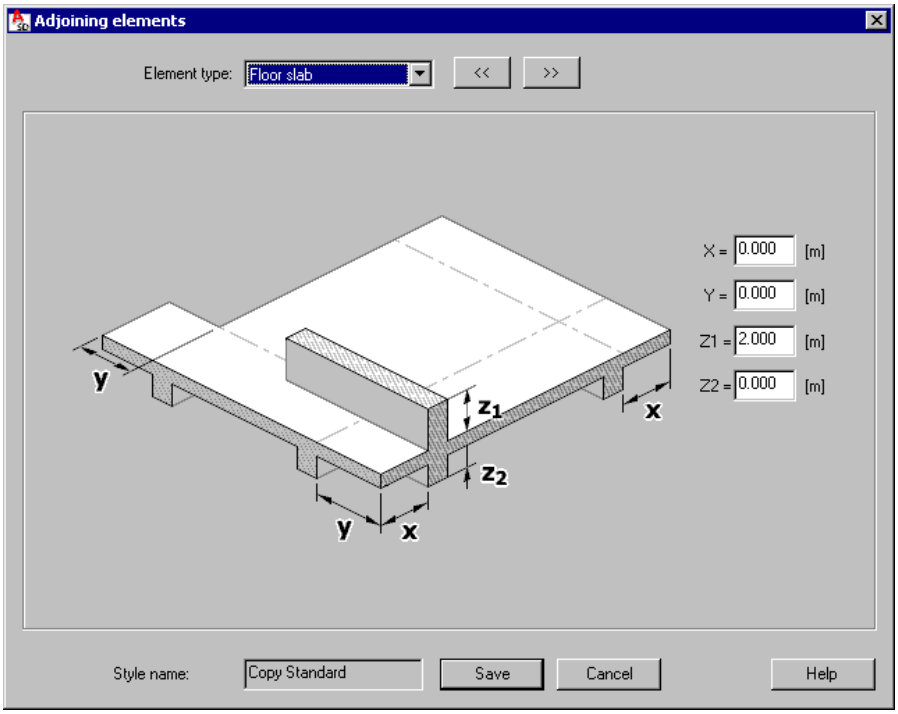
The following drawing components are available in the dialog box:
Adjoining elements
Graphic presentation
Section / View - parameters
Drawing layout

The right part of the dialog box contains the following buttons (apart from the standard ones: **OK**, **Cancel** and **Help**):
- **Default** – pressing this button makes a selected template a default drawing template for a chosen drawing component
- **New** - pressing this button opens a dialog box where a new template of drawing components may be defined
- **Modify** - pressing this button opens a dialog box where a selected template of a drawing component may be modified

- **Delete** - pressing this button deletes a highlighted template from the list of templates in the *Defined templates* field.

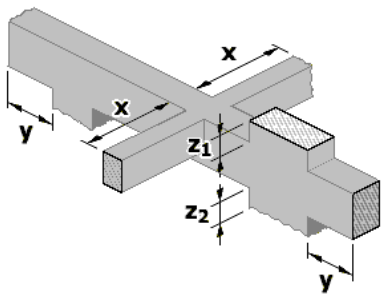
13.2. Adjoining elements

The dialog box is used to define the surroundings of a selected element type. These surroundings determine the length of adjoining elements in a drawing.
The ***Adjoining elements*** dialog box opens after choosing ADJOINING ELEMENTS in the *Drawing components* field and pressing the button **New** or **Modify** in the ***Drawing template manager*** dialog box. A style name may be specified in the *Style name* field located at the bottom of the dialog box (when modifying a style, the *Style name* field is not accessible; the name of a selected style of element description is displayed there).



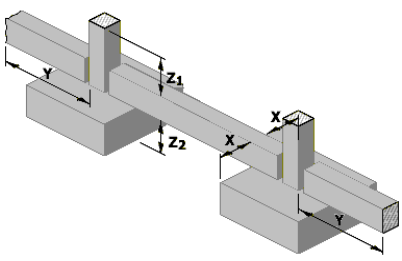
In the dialog box above, dimensions that form the surroundings of a selected element type are specified.
For individual element types the user may choose the following parameters of adjoining elements that can be placed in drawings of selected types of elements:

Beam



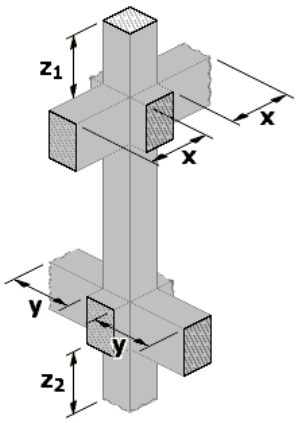
Parameters to be defined:
X Y
Z1, Z2

Ground beam



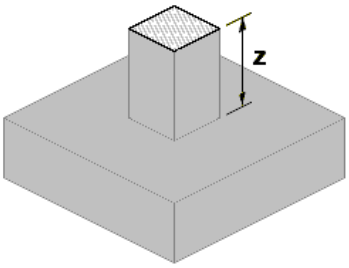
Parameters to be defined:
X, Y
Z1, Z2

Column



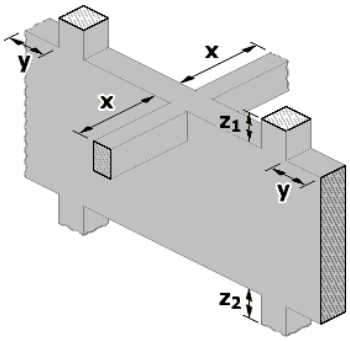
Parameters to be defined:
X Y
Z1, Z2

Spread footing



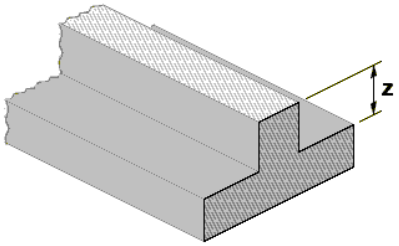
Parameters to be defined:
Z

Wall



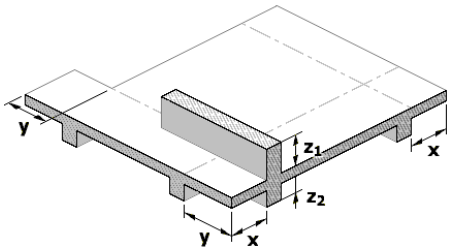
Parameters to be defined:
X Y
Z1, Z2

Continuous footing



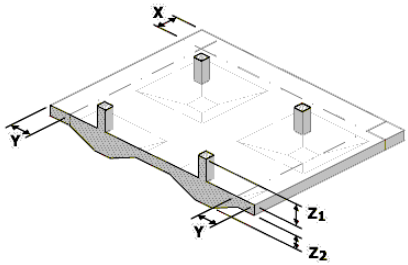
Parameters to be defined:
Z

Floor slab



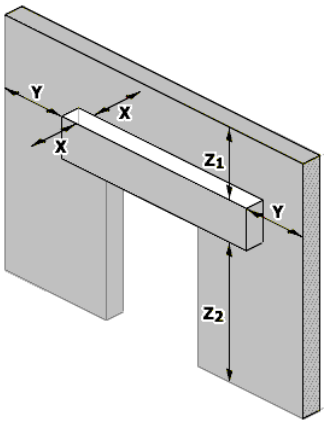
Parameters to be defined:
X Y
Z1, Z2

Raft foundation



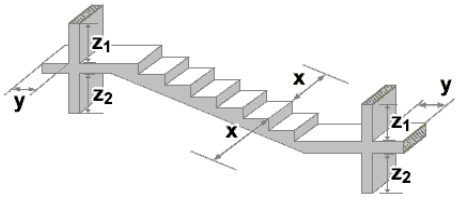
Parameters to be defined:
X, Y
Z1, Z2

Lintel



Parameters to be defined:
X, Y
Z1, Z2

Stairs



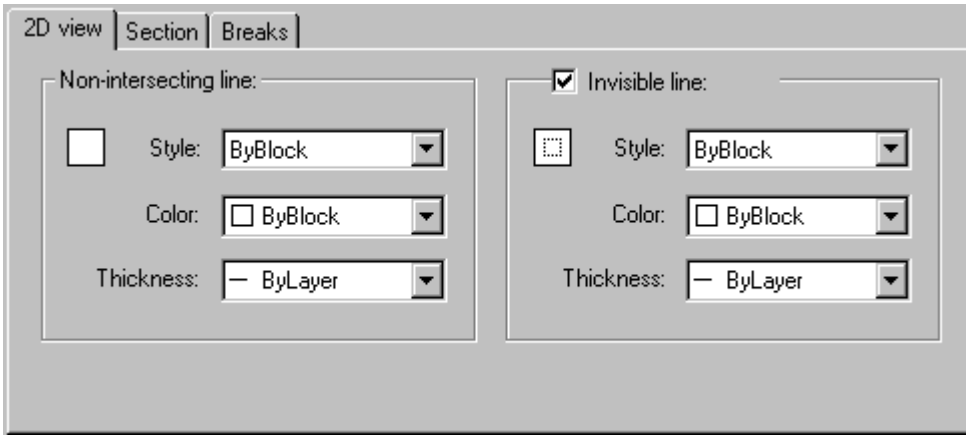
Parameters to be defined:
X, Y
Z1, Z2

13.3. Graphic presentation

The dialog box is used to determine parameters of graphic presentation of views and sections through elements of a structure model in formwork drawings.
The ***Graphic presentation*** dialog box opens after choosing GRAPHIC PRESENTATION in the *Drawing components* field and pressing the button **New** or **Modify** in the ***Drawing template manager*** dialog box. A style name may be specified in the *Style name* field located at the bottom of the dialog box (while modifying a style, the *Style name* field is not accessible; it shows a name of a selected style of element description).

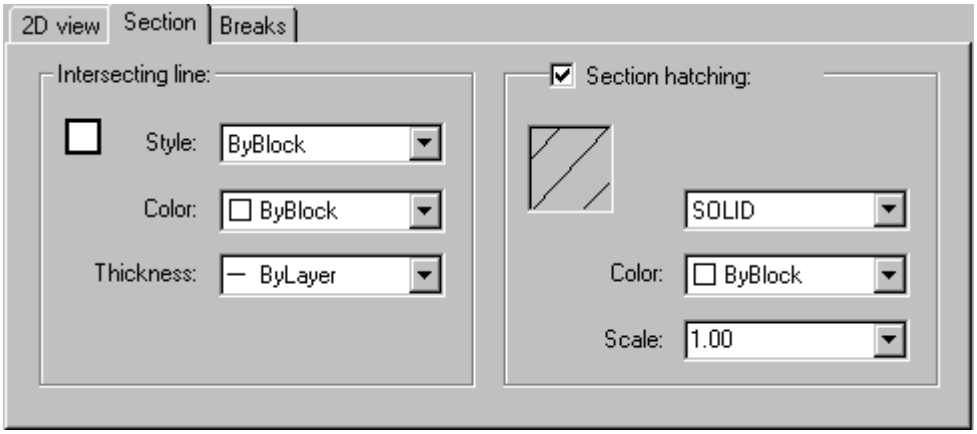
The dialog box consists of the three tabs: *2D view*, *Section* and *Breaks*.

The *2D view* tab



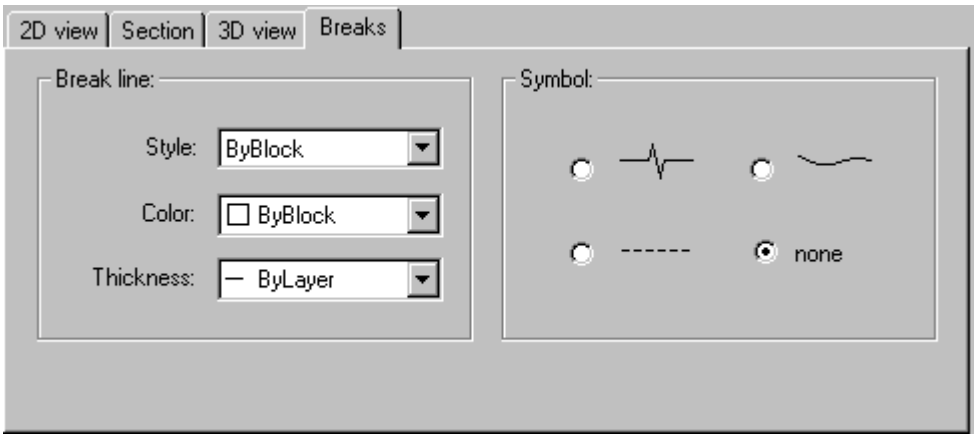
The options provided in the above tab concern parameters of presenting structure elements in 2D views (projections); the following parameters of a non-intersecting line and invisible line can be set (optionally): line style, color and thickness.

The *Section* tab



The options provided in the above tab concern parameters of presenting sections of elements of a structure model; the following parameters can be set here:
- for the intersecting line: line type, color and thickness.
- for section hatching (optionally): style, color and scale of hatching.

The *Breaks* tab



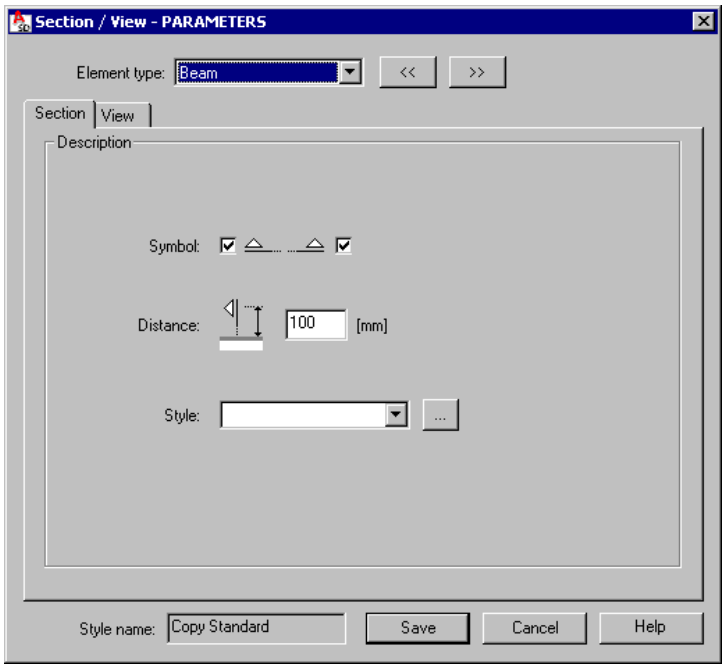
The options provided in the above tab concern parameters of presenting breaks of elements of a structure model in views; the following parameters can be set here:
- contour line: line style, color and thickness.

13.4. Section / view - parameters

The dialog box is used to determine parameters of section symbols and descriptions of views presented in formwork drawings of selected types of structure elements.
The ***Section / view - parameters*** dialog box opens after choosing SECTION / VIEW - PARAMETERS in the *Drawing components* field and pressing the button **New** or **Modify** in the ***Drawing template manager*** dialog box. A style name may be specified in the *Style name* field located at the bottom of the dialog box (while modifying a style, the *Style name* field is not accessible; it shows a name of a selected style of element description).

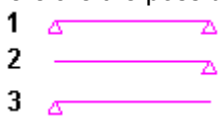
In the upper part of the dialog box is a selection list for selection of a structure element type (beam, column, spread footing, etc.) for which parameters of sections and views will be defined.
The dialog box consists of the two tabs: *Section* and *View*.

The *Section* tab



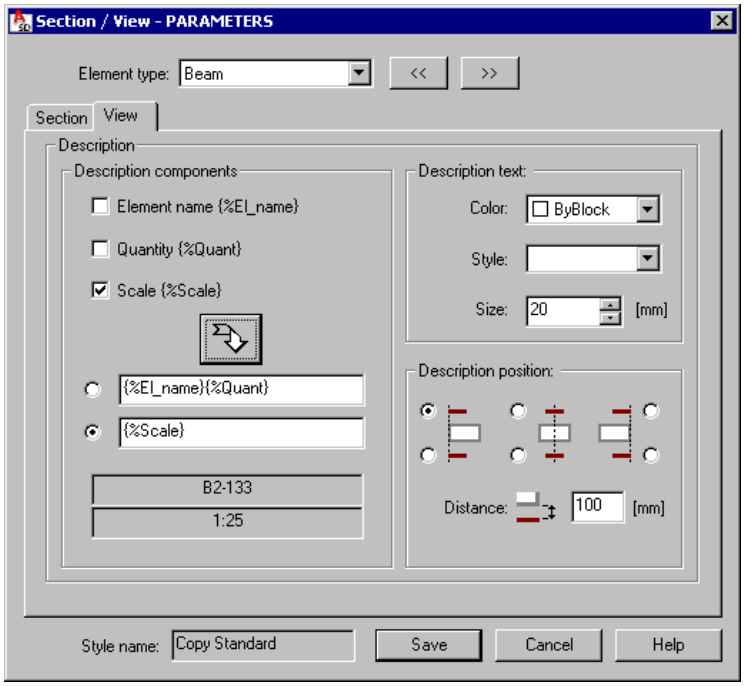
The above dialog box offers the following options referring to a cutting line:

- *symbol* – it indicates the way a section symbol will be presented in formwork drawings; there are the possibilities as follows:



- *distance* – an edit field to specify a distance between the section symbol and the element contour
- *style* – the list includes all the defined styles of section symbols; pressing the (**...**) button opens the ***Styles of symbols*** dialog box which enables selecting another existing style of the section symbol or modifying an existing style of that symbol

The *View* tab

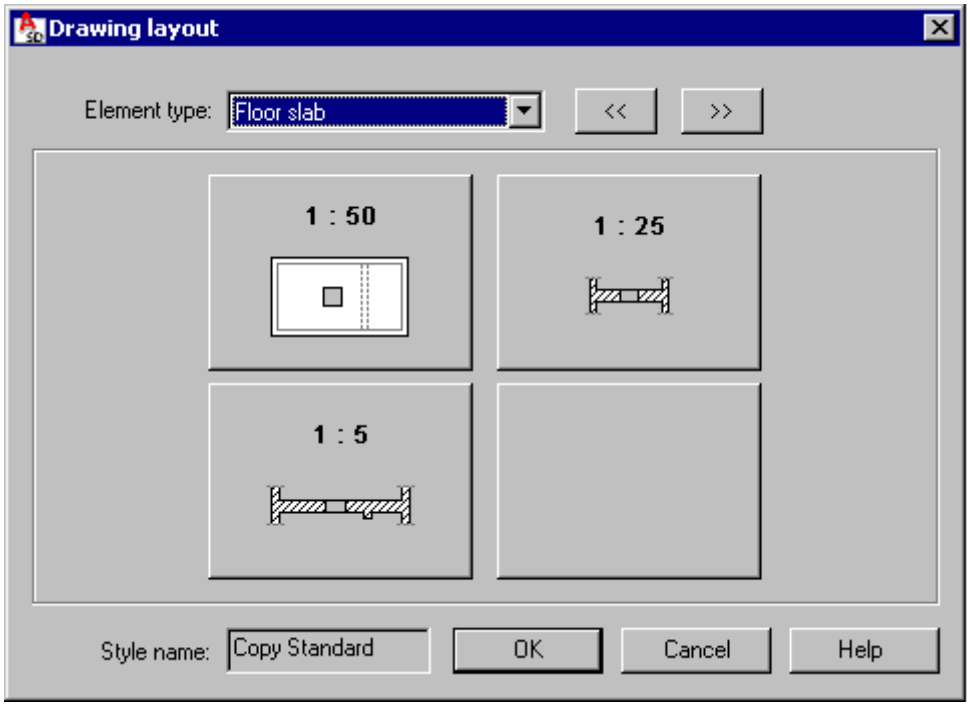


The above dialog box offers the following options used to parametrize the description of a view of an element drawing:

- *description components* – definition of a syntax of a section description; the description may consist of two parts, therefore, two edit fields are available (after pressing the arrow, selected description components are moved to an active edit field – below are inaccessible fields which show a preview of the view description resulting from a defined syntax); a list of variables that may appear in a section description includes:
  %El_name - element name (position name)
  %Quant - quantity (number of positions)
  %Scale - scale of the drawing of an element view
- *description text* - definition of a section description; the following parameters can be set here: color, style and size of a description.
- *description position* - definition of the description position in a drawing of an element view; there is a possibility to select several description positions and a distance between the description and the view drawing.

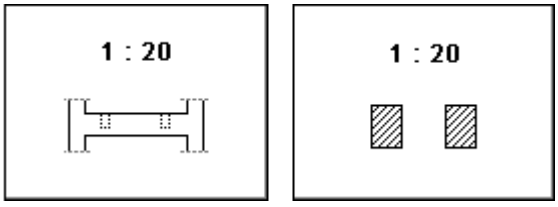
13.5. Drawing layout

The dialog box is used to determine a drawing layout (which drawings and in what scale will be generated) for selected types of structure elements.
The ***Drawing layout*** dialog box opens after choosing DRAWING LAYOUT in the *Drawing components* field and pressing the button **New** or **Modify** in the ***Drawing template manager*** dialog box. A style name may be specified in the *Style name* field located at the bottom of the dialog box (while modifying a style, the *Style name* field is not accessible; it shows a name of a selected style of element description).

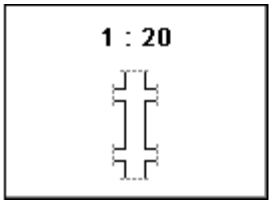


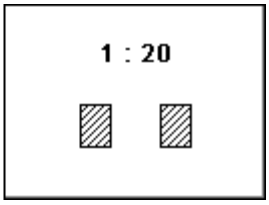
By default, for individual types of structure elements the program creates drawings in the presented layout and defined scale:

Beam

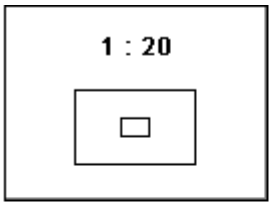


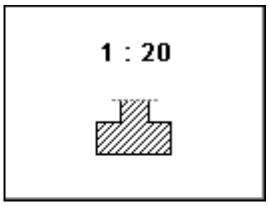
Column

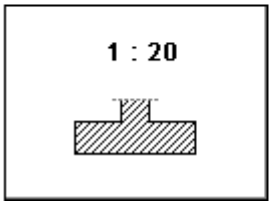




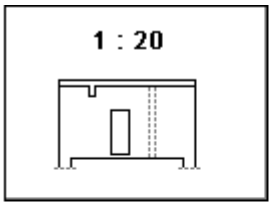
Spread footing

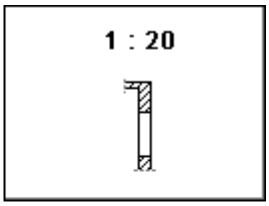


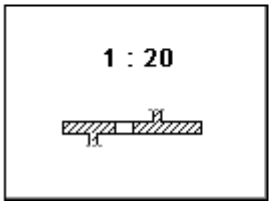




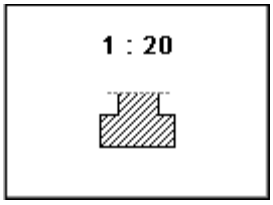
Wall



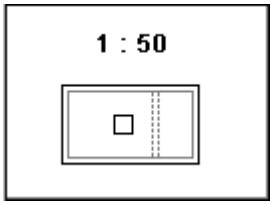




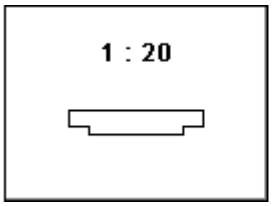
Continuous footing

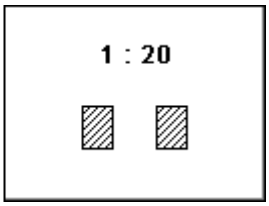


Floor slab

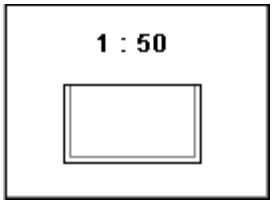


Ground beam

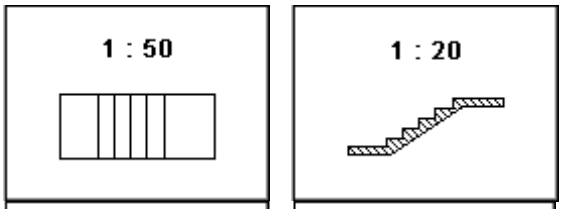




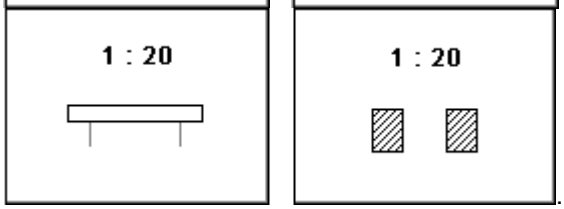
Raft foundation



Stairs



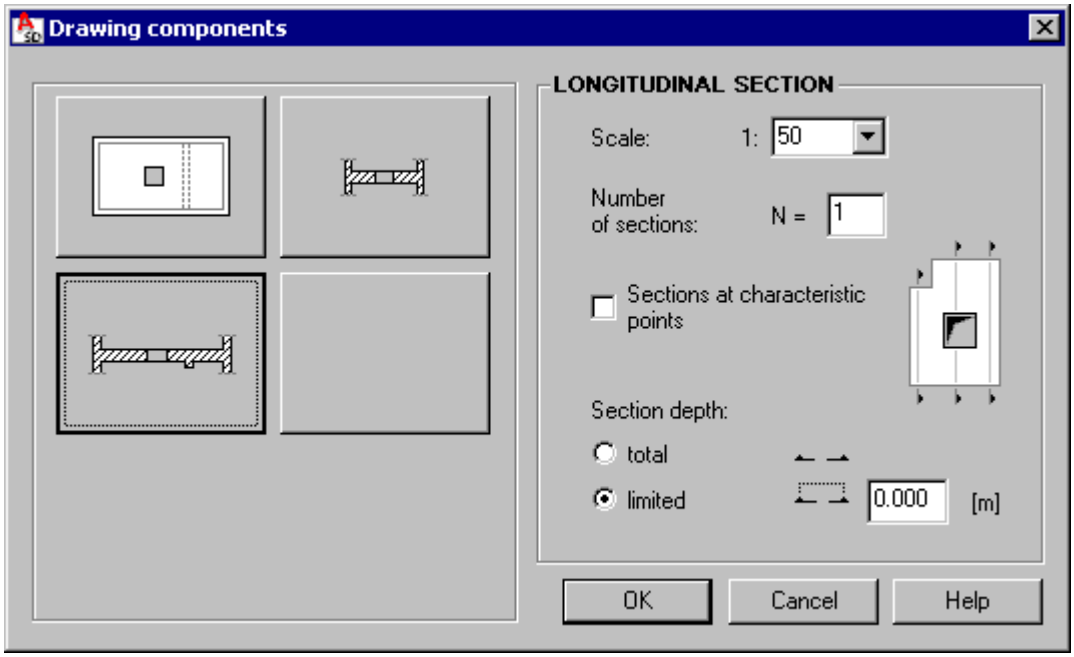
Lintel



If for a selected drawing layout and a selected structure element type there is the  icon in the above dialog box, it indicates the field which does not contain any drawing.
Pressing the icon that symbolizes a generated drawing opens another dialog box - ***Drawing components***.

13.6.Drawing components

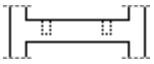
The dialog box is used to determine a drawing layout (which drawings and in what scale will be generated) for a selected type of structure elements.
The ***Drawing components*** dialog box opens on pressing the icon that denotes a generated drawing in the ***Drawing layout*** dialog box.



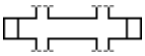
The contents of this dialog box depends on the type of a structure element for which it has been opened. Below are presented options for individual types o elements.

Beam

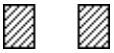
Generation of the following drawings is possible:

 **Front view (elevation)**

Parameters:
View scale

 Top view

Parameters:
View scale

 Cross-section view

Parameters:
View scale
Number of sections in a beam
Section depth:
- total (section through the whole beam)
- limited to a value specified

Moreover, the *Sections at characteristic points* option may be switched on, which results in creating drawings in sections through the beam which are associated with additional points (openings, adjoining beams, etc.).

Column

Generation of the following drawings is possible:

 **Front view (elevation)**

Parameters:
View scale

 Side view

Parameters:
View scale

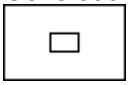
 Cross-section view (several sections set horizontally / vertically one next to another)

Parameters:
View scale
Number of sections in a column
Section depth:
- total (section through the whole column)
- limited to a specified value

Moreover, the options *Lower column section and Upper column section* may be switched on, which results in creating drawings of a section through the column positioned above or below a given column. Sections through a column may be presented in drawings in a row (one next to another set horizontally) or in a column (one next to another set vertically).

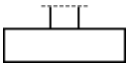
Spread footing

Generation of the following drawings is possible:

 **Top view**

Parameters:

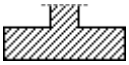
View scale

 Front view

Parameters:
View scale

 Side view

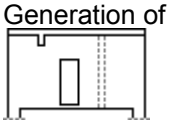
Parameters:
View scale

 or  Cross / longitudinal section

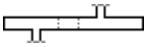
Parameters:
View scale
Section depth:
- total (section through the whole spread footing)
- limited to a specified value

Wall

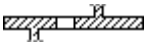
Generation of the following drawings is possible:

 **Front view (elevation)**

Parameters:
View scale

 Top view

Parameters:
View scale

 or  Cross / longitudinal section

Parameters:
View scale
Number of sections in a wall
Section depth:
- total (section through the whole wall)
- limited to a specified value

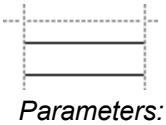
Moreover, the *Sections at characteristic points* option may be switched on, which results in creating drawings in sections through the wall that are associated with additional points (openings, adjoining beams, etc.).

Continuous footing

Generation of the following drawings is possible:

 Horizontal / vertical section

Parameters:
View scale
Section depth:
- total (section through the whole continuous footing)
- limited to a specified value

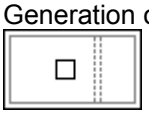
 **Front view (elevation)**

Parameters:

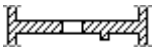
View scale

Floor slab

Generation of the following drawings is possible:

 Top view

Parameters:
View scale

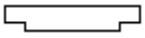
 or  Longitudinal / cross section

Parameters:
View scale
Number of sections in a slab
Section depth:
- total (section through the whole slab)
- limited to a specified value

Moreover, the *Sections at characteristic points* option may be switched on, which results in creating drawings in sections through the slab that are associated with additional points (openings, points on a broken-line-shaped edge of a slab, etc.).

Ground beam

Generation of the following drawings is possible:

 **Front view**

Parameters:
View scale

 Top view

Parameters:
View scale

 Cross-section view

Parameters:
View scale
Number of sections in a ground beam
Section depth:
- total (section through the whole ground beam)
- limited to a specified value

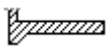
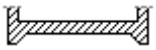
Moreover, the *Sections at characteristic points* option may be switched on, which results in creating drawings in sections through the ground beam that are associated with additional points (change of the beam section, etc.).

Raft foundation

Generation of the following drawings is possible:

 Top view

Parameters:
View scale

 or  Longitudinal / cross section

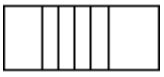
Parameters:
View scale
Number of sections in a raft foundation
Section depth:
- total (section through the whole raft foundation)
- limited to a specified value

Moreover, the *Sections at characteristic points* option may be switched on, which results in creating drawings in sections

through the raft foundation that are associated with additional points (points on a broken-line-shaped edge of a slab, etc.).

Stairs

Generation of the following drawings is possible:

 or  Top view / Side view

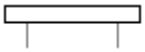
Parameters:
View scale

 Cross-section

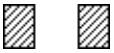
Parameters:
View scale
Number of sections of stairs
Section depth:
- total (section through the whole stairs)
- limited to a specified value

Lintel

Generation of the following drawings is possible:

 **Front view (elevation)**

Parameters:
View scale

 Cross-section view

Parameters:
View scale
Number of sections in a lintel
Section depth:
- total (section through the whole lintel)
- limited to a specified value

If for a selected drawing layout and a selected structure element type there is the  icon in the above dialog box, it indicates the field which does not contain any drawing.

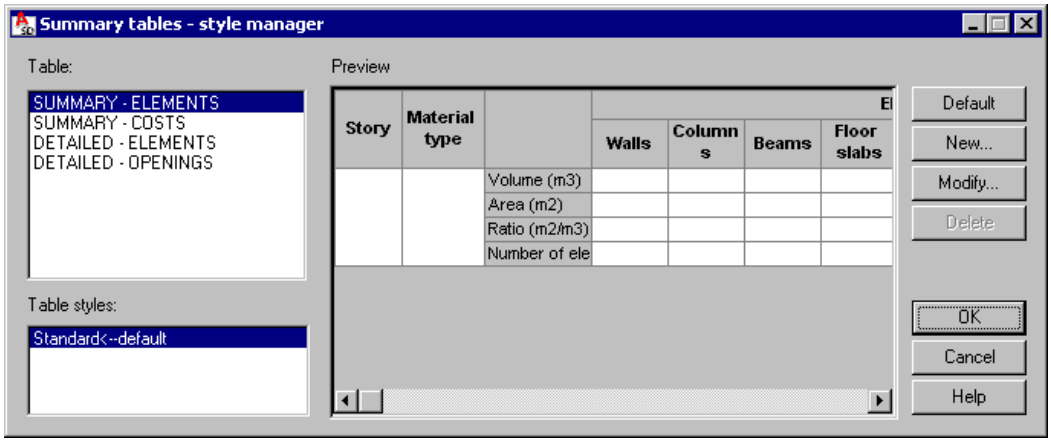
14. SUMMARY TABLES

14.1.Summary tables (style manager)

The option allows defining/modifying summary tables. The option is accessible from:

- from the menu by selecting the option Formwork Drawings / Styles / Styles - tables
- from the command line: RBCX_STYLE_LIST.

After activating the option, the ***Summary tables - style manager*** dialog box, shown in the drawing below, appears on the screen.



The program offers the following types of tables (presented in the *Table* field in the above dialog box):

- Summary - elements
- Summary - costs
- Detailed - elements
- Detailed - openings.

Individual table types present the information as follows:

- *Summary - elements* - this is a table with a list of elements in a structure model
- *Summary - costs* - this is a table including costs of elements of a structure model
- *Detailed - elements* - a summary presentation of elements of a structure model (including both summary tables).
- *Detailed - openings* - a summary presentation of openings defined in elements of a structure model.

For each table type a standard table style (presented in the *Table style* field) has been defined in the program. After highlighting a table type and a table style, a view of the chosen table style is presented in the middle part of the dialog box (in the *Preview* field).
The right part of the dialog box contains the following buttons (apart from the standard ones: **OK**, **Cancel** and **Help**):

- **Default** - pressing this button sets an indicated style as a default table layout (description style)
- **New** - pressing this button opens the Definition of a new table style dialog box where a new style of a selected table type may be defined (based on a style that already exists)
- **Modify** - pressing this button opens the ***Modification of table style*** dialog box where a selected table type and table style can be modified
- **Delete** - pressing this button deletes a highlighted table style from the list of styles available in the *Table styles* field.

14.2. Definition/modification of a summary table style

14.2.1. Definition/modification of a table style

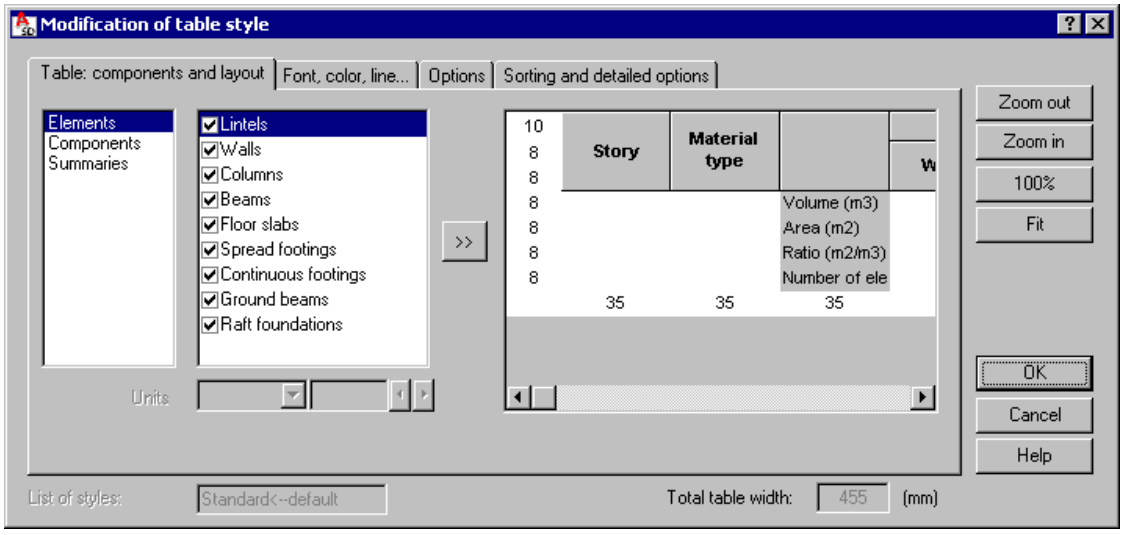
The dialog box is used to define a new style or to modify an existing style of a table type.

The ***Definition of a new table style*** dialog box opens after pressing the **New** button in the Summary tables (style manager) dialog box (the ***Modification of table style*** dialog box opens after pressing the **Modify** button).

The dialog box consists of the four tabs:
Table: components and layout
Font, color, line
Options
Sorting and detailed options.

14.2.2. Table: components and layout

The Definition/modification of a table style dialog box looks as presented in the drawing below after selecting the *Table: components and layout* tab in it.



NOTE:
The options located on this tab depend on a table type selected in the ***Summary tables - style manager*** *dialog box. The drawing above shows the options available after selecting the detailed table.*

At the bottom of the dialog box is the *List of styles* edit field; there a name of a defined table style should be entered (when modifying a table style, the *List of styles* field is inaccessible).
To define/modify a table style, follow the steps below:

- select a set of table components (e.g. in the dialog box shown above these will be: *Element*, *Prices*, *Position*, *Story*, *Material*)
- in the next field switch on the components to be included in the table (the option is switched on when the √ symbol appears)
- press the **>>** button.

In the right part of the dialog box the defined table layout is presented.

NOTE:
Location of columns and column names may be freely modified for every table style.

The above dialog box also holds the following options:

- *Total table width* – an inaccessible edit field which displays a width of a defined table determined by the program
- buttons: **Zoom in, Zoom out, 100%,** which allow zooming in / zooming out a table presented
- **Fit** button, pressing which enables adjusting width of table columns to length of texts in column headers.

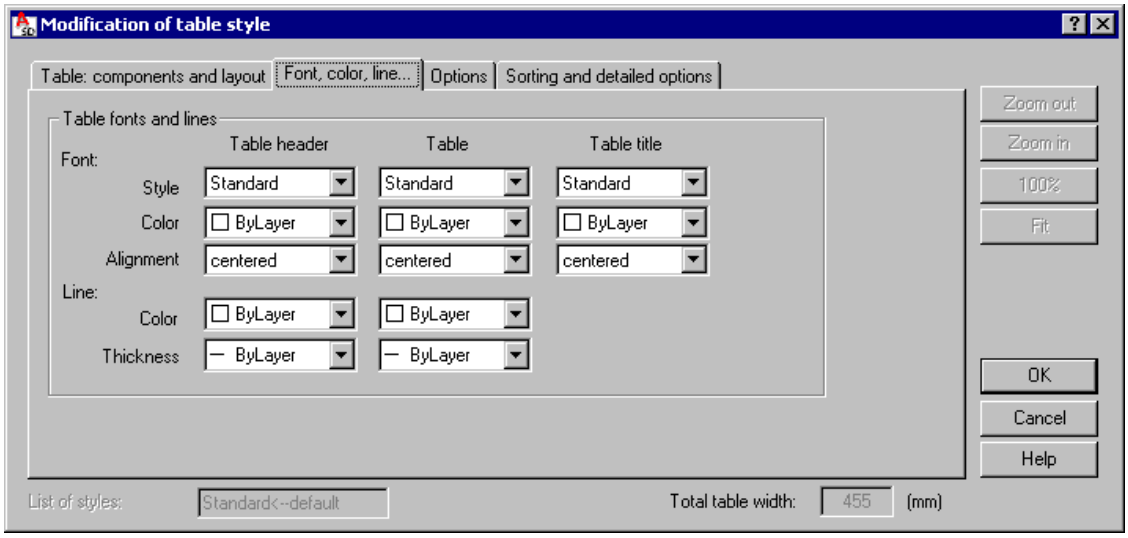
The order of individual table columns may be freely arranged (it refers only to the detailed table). To do it, select a whole column, and next while keeping the left mouse button pressed, move the column to a chosen location. Apart from that, it is possible to increase height of table cells and in table headers, to define additional user descriptions or to change names of existing columns.

The tables enable the user:

- inserting additional blank lines at the beginning and end of a table
- inserting blank columns in tables
- exact definition of table dimensions (width of columns and height of lines) – dimensions of column widths and line heights are presented under the table and on the left side of the table, respectively.

14.2.3. Font, color, line

The Definition/modification of a table style dialog box looks as presented in the drawing below after selecting the *Font, color, line* tab in it.

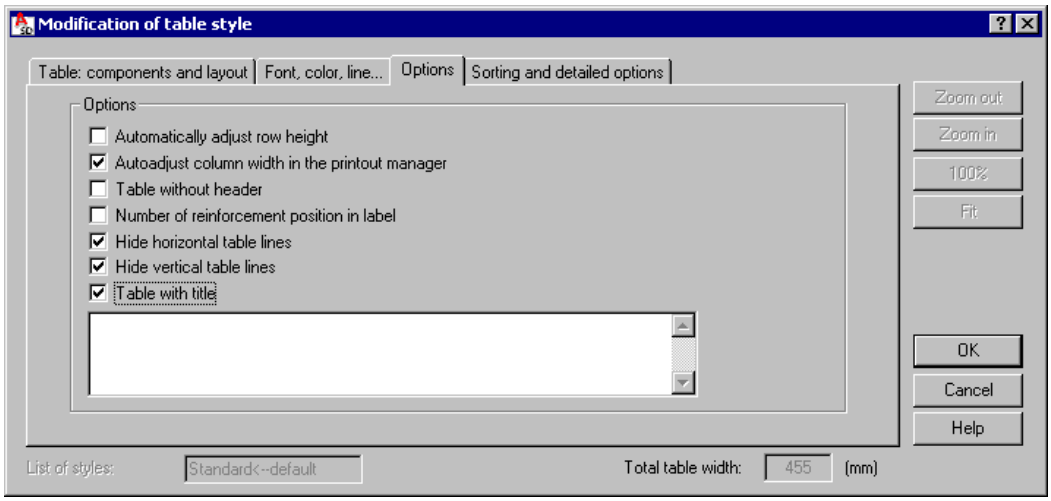


The options available in the above dialog box enable the user to:

- determine a font used in a table (in the table header and all table cells): style, color and alignment of a table header and texts in table cells
- select table lines: thickness and color
- define a font used in a table title (if the *Table with title* option is activated on the *Options* tab): style, color and alignment of the table title.

14.2.4. Options

The Definition/modification of a table style dialog box looks as presented in the drawing below after selecting the *Options* tab in it.

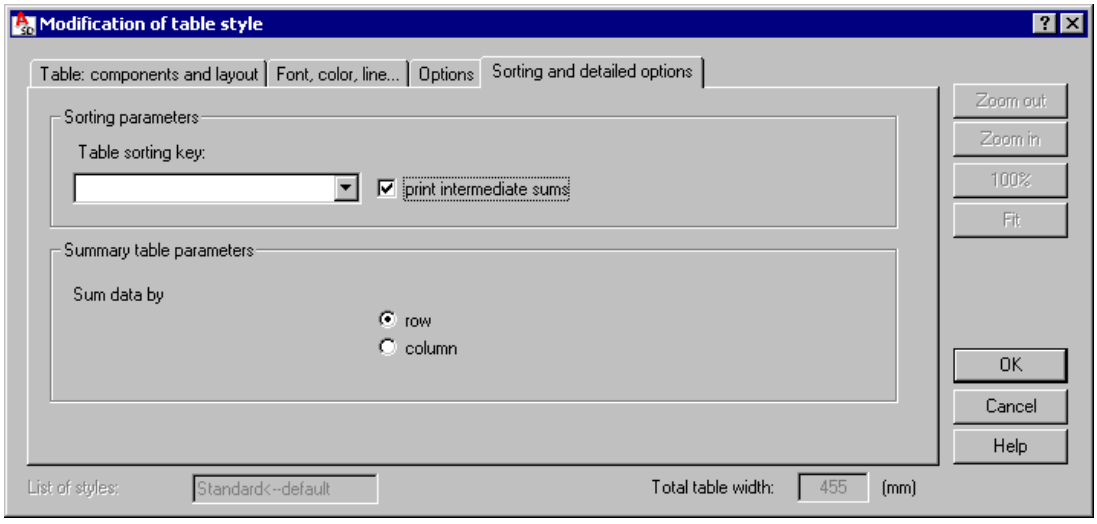


The following options are available in the *Options* field:

- *automatically adjust row height* - if this option is switched on (the √ symbol appears), then height of table rows will be automatically adjusted to the size of symbols of element shapes presented in the table
- *table without header* - if this option is on (the √ symbol appears), then there will be no header in a defined table
- number of reinforcement position in label - if this option is switched on (the √ symbol appears), then a position number in a table will be presented in a round label
- *hide horizontal table lines* - if this option is switched on (the √ symbol appears), then horizontal lines will not be displayed in a table
- *hide vertical table lines* - if this option is switched on (the √ symbol appears), then vertical lines will not be displayed in a table
- *table with title* - if this option is switched on (the √ 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.

14.2.5. Sorting and detailed options

The Definition/modification of a table style dialog box looks as presented in the drawing below after selecting the *Sorting and detailed options* tab in it.



When the detailed table is selected, the Key for table sorting option is accessible on the above tab (in the Sorting parameters field). It enables sorting the element table by: position-name, element type, section or weight (by standard, the table is sorted by positions).
The option *Sum data by* rows or columns is available only for SUMMARY tables.

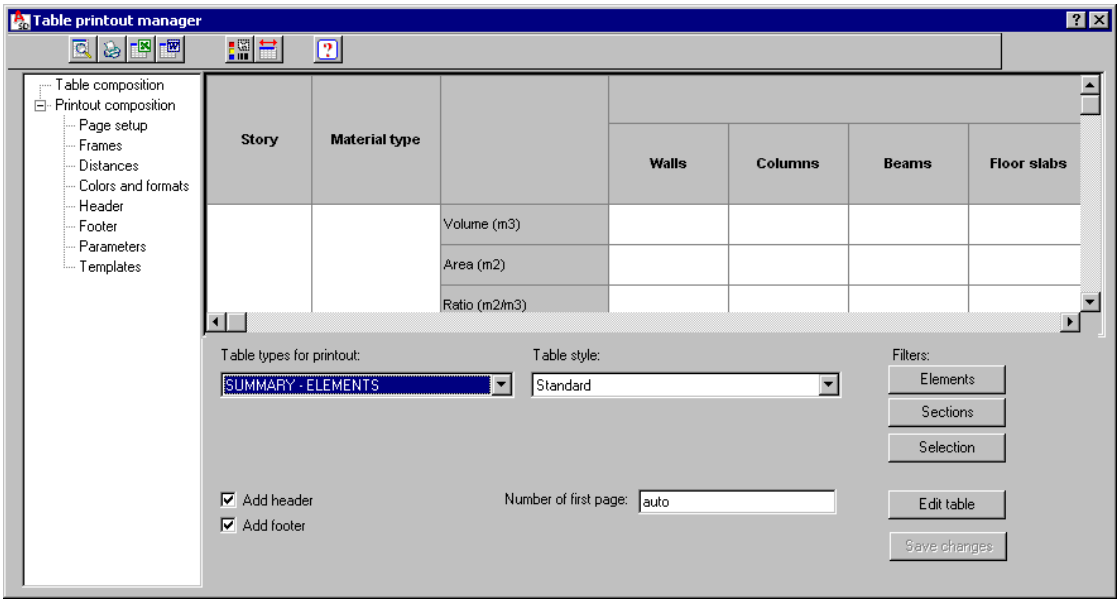
15. PRINTOUT - TABLES

15.1. Table printout manager

The option allows defining/modifying the appearance of a printout of summary tables. The option is accessible from:

- the menu by selecting the option *Formwork Drawings / Tables / Table Printout/Export/Edit*
- the toolbar by pressing the icon
- the command line RBCX_LIST_EXP.

After activating the option, the ***Table printout manager*** dialog box, shown in the drawing below, appears on the screen.



The ***Table printout manager*** dialog box can be divided in the two main parts:

- in the left-hand part of the dialog box is a selection tree (see the drawing below) from which the user selects with the mouse cursor one of the options of the printout manager

- the part of the dialog box to the right of the selection tree includes parameters relevant for the option selected by the user from the selection tree; the dialog box is updated after selecting an option. The upper part of the dialog box shows the layout of a selected table type.

The upper part of the dialog box holds the options as follows:


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 MS Excel © format. A table may be saved to:
- *.CSV (Comma Separated Values) format files - 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 in which a table may be saved in an MS Word file of 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 adjusts the width of table columns to the length of names of table columns


Help - pressing this icon opens Help.

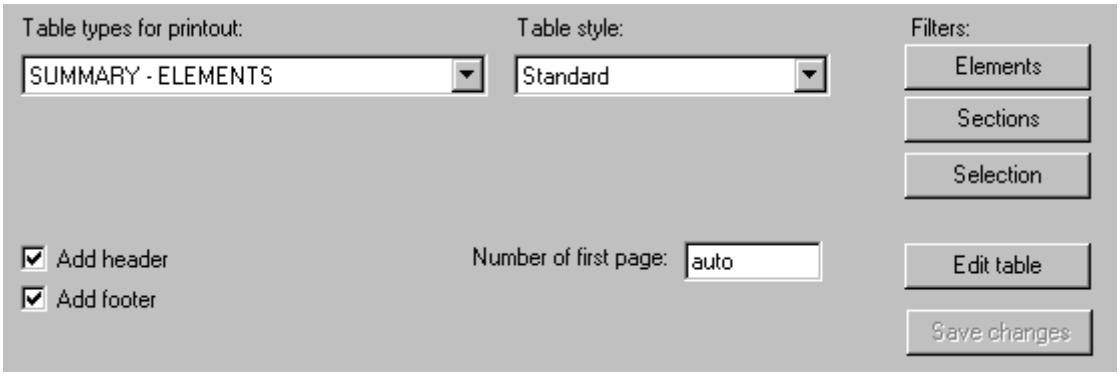
It should be mentioned here that, although a table shape (cell height, column width) depends directly on a defined table style, the user may freely determine a cell height or a column width.

Moreover, a table contains a context menu with the following options:

- table cells - options: group (merges several table cells into one cell) and ungroup
- text orientation – vertical, horizontal
- inserting or removing a column from the table
- inserting or removing a row from the table.

15.2. Table composition

After selecting the *Table composition* 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 following table types are available in the program:

- Summary - elements
- Summary - costs
- Detailed - elements
- Detailed - openings.

For each table type the user may choose a table style defined previously for the selected table type.

Pressing the **Elements** button in the right-hand part of the dialog box (the *Filters* option) opens the ***Filters - element selection (by type)*** dialog box for defining criteria of element selection (by element type).

Pressing the **Sections** button in the right-hand part of the dialog box (the *Filters* option) opens the ***Filters - element selection (by section)*** dialog box for defining criteria of element selection (by element section).

Pressing the **Selection** button in the right-hand part of the dialog box (the *Filters* option) allows the user to switch to the graphical viewer in the element selection mode.

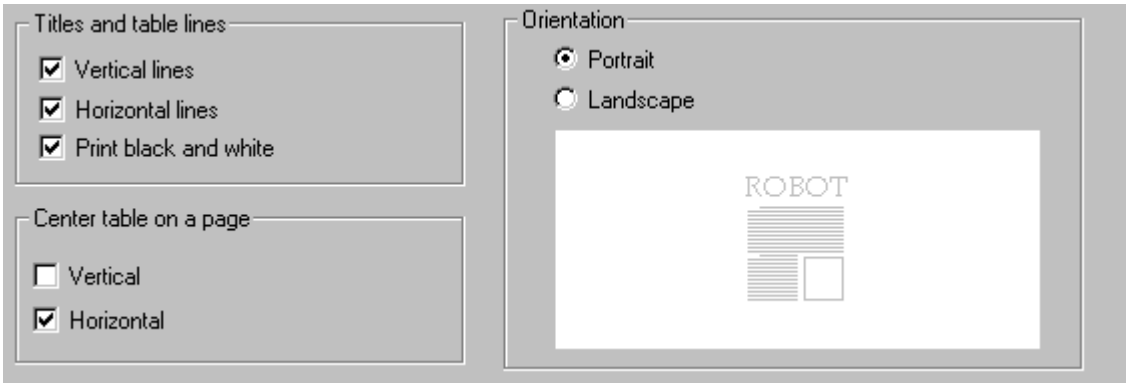
Pressing the **Edit table** button allows the user to indicate graphically a table to be edited. Changes made in a table may be saved after pressing the **Save changes** button.

The upper part of the dialog box holds the options as follows:

- Number of first page - the field in which the user may specify a number of the first printed page
- *Add header* - if this option is switched on, then a defined header will appear on a printout
- *Add footer* - if this option is switched on, then a defined footer will appear on a printout.

15.3. Page setup

After selecting the *Page setup* 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.

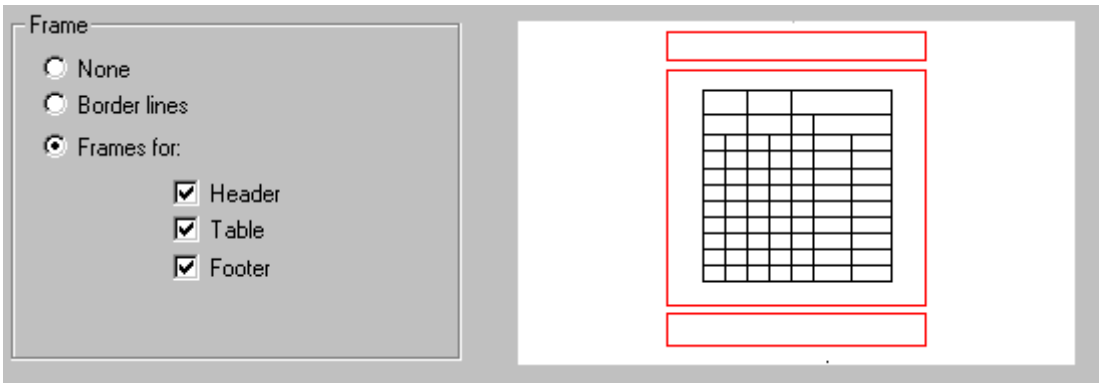


The dialog box above allows determining the manner of table presentation:

- in the *Titles and table lines* field:
  - if the *Vertical lines* option is switched off, then vertical lines in a table are not displayed
  - if the *Horizontal lines* option is switched off, then horizontal lines in a table are not displayed
  - if the *Print black and white* option is switched off, then a table is printed using the defined colors
- the options in the *Center table on a page* field are used to determine how a table should 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).

15.4. Frames

After selecting the *Frames* 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.



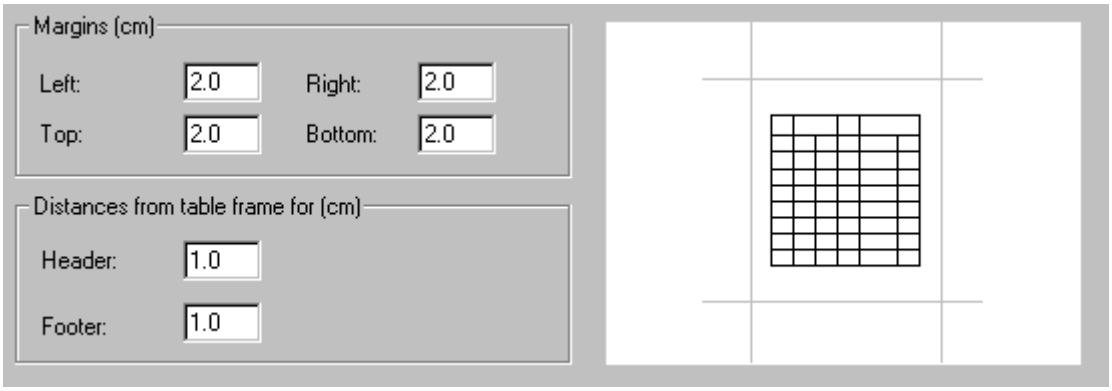
The dialog box above allows determining the manner of table presentation on a page:

- no separation between a header/footer and a table
- with a border line that separates the 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 mentioned elements).

15.5. Distances

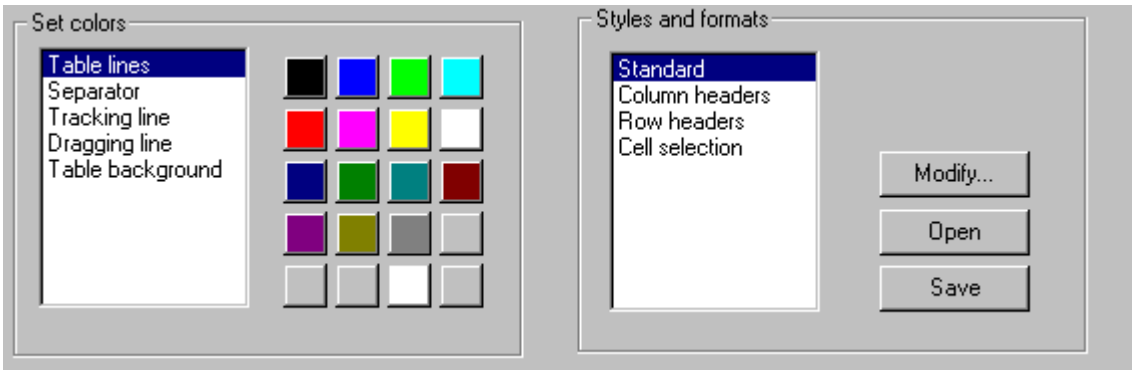
After selecting the *Distances* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



The above dialog box allows determining (similarly, as in every text editor) page margins: left, right, top and bottom. Moreover, it is possible to define distances between the table frame and the header or footer.
A size of the header and footer is calculated automatically in the program; the parameters mentioned depend on the size of a font used, size of a drawing with the company logo and a number of lines required in a header or footer.

15.6. Colors and formats

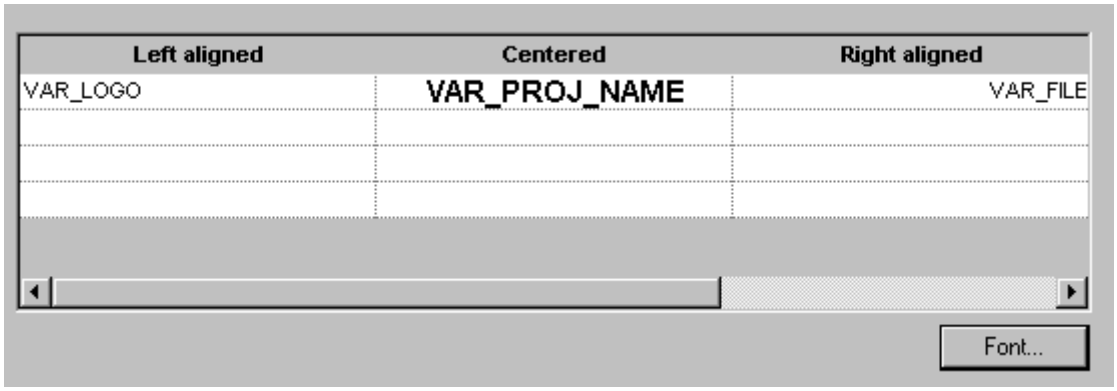
After selecting the *Colors and formats* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



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 in a table. Pressing the **Modify** button opens the dialog box where a format (font, font color, alignment) for the listed table elements may be chosen.

15.7.Header

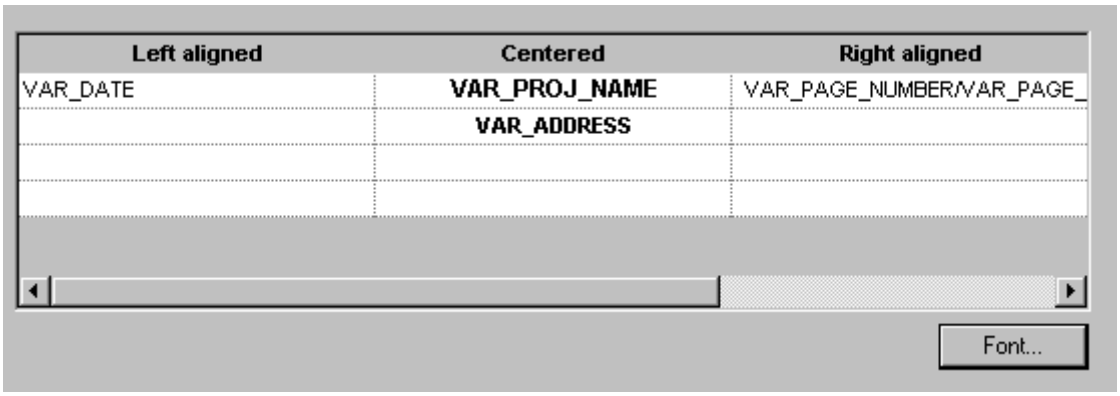
After selecting the *Header* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



The above dialog box shows a layout of the printout header.
To change a header layout, press the table field presenting the header layout; then the list of available variables unfolds and from this list an appropriate variable may be selected. When the cursor is positioned in a table field 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.

15.8.Footer

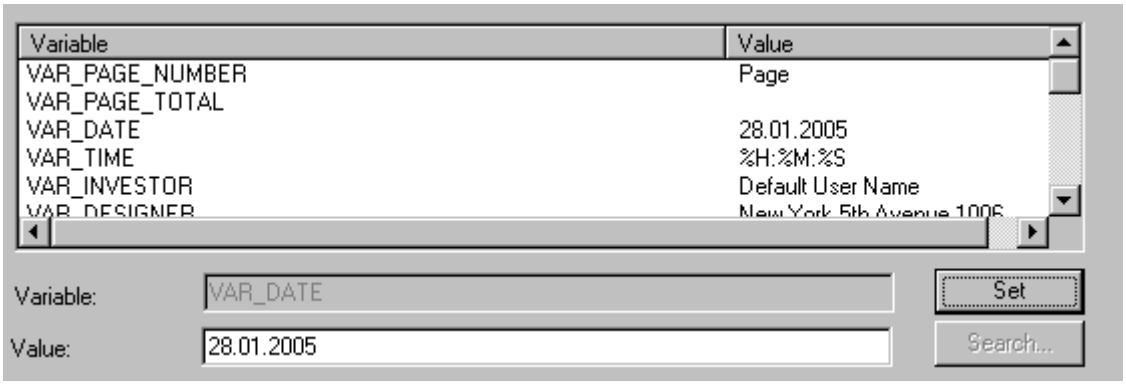
After selecting the *Footer* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.



The above dialog box shows a layout of the printout footer.
To change a footer layout, press the table field presenting the footer layout; then the list of available variables unfolds and from this list an appropriate variable may be selected. When the cursor is positioned in a table field 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.

15.9.Parameters

After selecting the *Parameters* 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 above dialog box contains all the variables defined in the system and their names.

NOTE:
*Setting of a variable and next, its modification **must** be confirmed by pressing the **Set** button.*

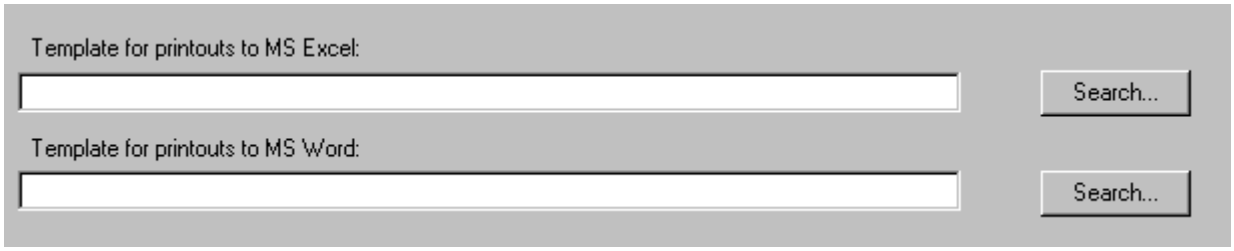
Variables are used for formatting a printout header and footer:
VAR_PAGE_NUMBER - variable that allows assigning the current printout page. The total number of pages will be preceded by the text ascribed to the variable if the VAR_PAGE_TOTAL variable is used (e.g. if the "Page **VAR_PAGE_NUMBER** " value 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. The text assigned to it may be preceded with a number of the current page if the VAR_PAGE_NUMBER variable is used
VAR_DATE, **VAR_TIME -** these variables may be assigned any text and a combination of the key words presented below (it enables printing the current date/time on a printout); allowable formats include:
%A - full name of a week day (Monday)
%a - abbreviated name of a week day (Mon)
%B - full name of a month (January)
%b - abbreviated name of a month (Jan)
%c - standard representation of a date and time
%d - day of a month (01-31)
%H - time (24-hour clock) (00-23)
%I - time (12-hour clock) (00-12)
%j - day of a 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 - week of a year (first day - Sunday) (00-53)
%W - week of a year (first day - Monday) (00-53)
%w - day of a week (0-6, Sunday is denoted by 0)
%X - standard representation of time
%x - standard representation of a date
%Y - year and century
%y - year without the century (00-99)
%Z - name of a time zone
%% - 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 are used only for convenient classification while formatting a

printout. ***AutoCAD © Structural Detailing - Formwork Drawings*** program enables using the following variables associated with a printout:
VAR_INV_NAME – investor's name
VAR_INV_ADDRESS - investor's address
VAR_INV_PHONE – investor's phone
VAR_INV_FAX - investor's fax
VAR_INV_EMAIL – investor's e-mail address
VAR_OFF_NAME – name of a design office
VAR_OFF_NAME – name of a design office
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 a *.bmp file.
These variables may be also used 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.

15.10. 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.

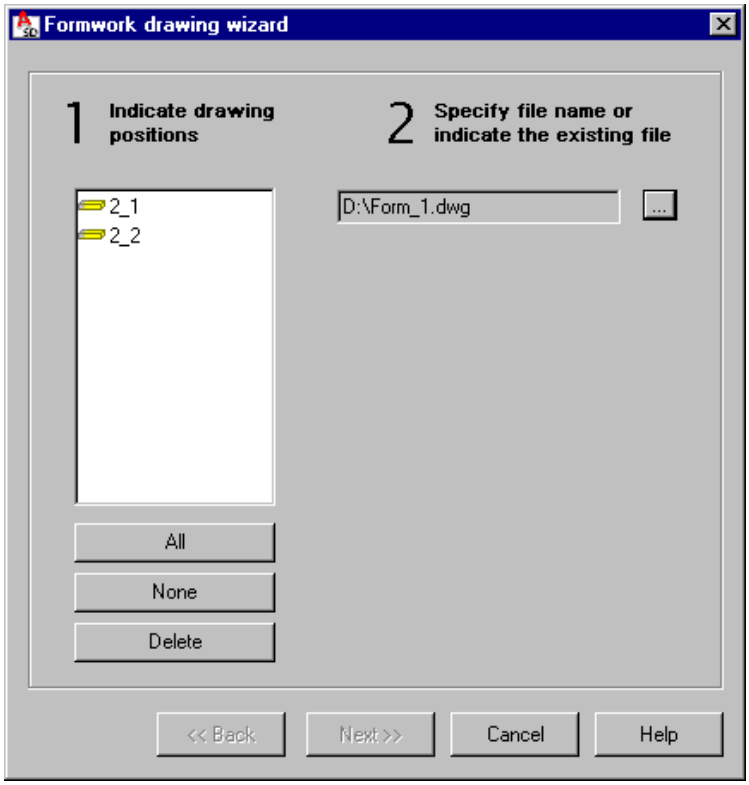


The dialog box above allows selecting templates for printing tables to the following two programs:
- MS Word ©: *.dot format files
- MS Excel ©: *.xlt format files.
In the edit fields the user may specify file names with a full access path; after pressing the **Search** button, it is possible to indicate the template file on the computer hard disk.

16. DRAWINGS

16.1. Formwork drawing wizard

The option is used to save formwork drawings or to transfer them to ***AutoCAD © Structural Detailing - Reinforcement*** program. It also allows automatic generation of reinforcement for selected structure elements. The contents of the ***Formwork drawing wizard*** dialog box shown in the drawing below depend on a context menu option selected on the Positions tab in the ***Object Inspector*** dialog box.

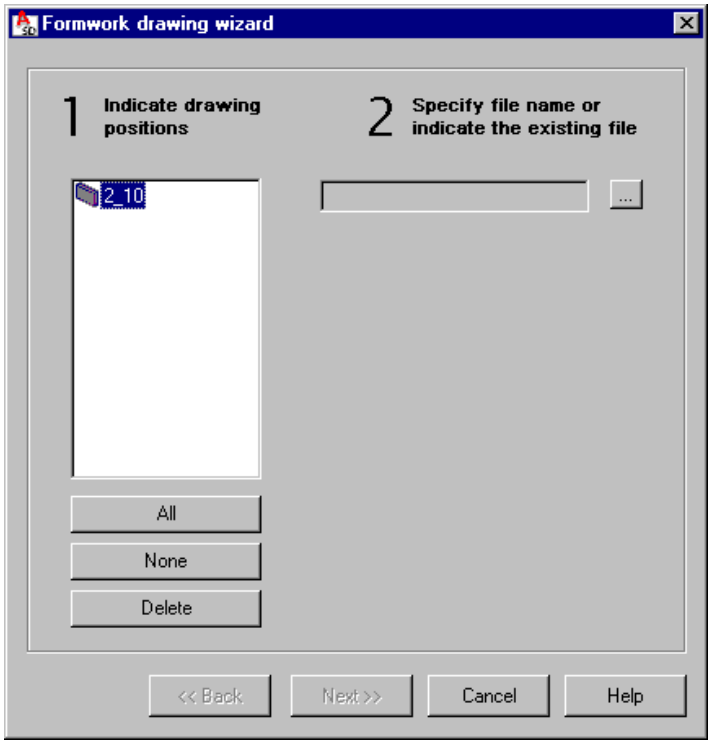


The dialog box opens during:
- export of formwork drawings to AutoCAD © Structural Detailing - Reinforcement
- export of formwork drawings to a DWG format file
- automatic reinforcement generation with the use of the AutoCAD © Structural Detailing - Reinforcement macros.

16.2. Export of formwork drawings to DWG

The option is used to save a formwork drawing of a selected position (element) to a DWG format file. To open the dialog box below, do as follows:

- run positioning of structure elements (see: Automatic positioning)
- select a position on the *Positions* tab in the ***Object Inspector*** dialog box
- press the right mouse button and in the context menu choose the option *Export formwork drawings to DWG.*



In the dialog box above the following operations should be performed:
1. indicate a drawing position (for a selected element type) in the left part of the dialog box
2. indicate the file where the formwork drawing will be saved
3. press the **Next >>** button.

After the formwork drawing is saved, it may be opened and edited in the AutoCAD © or ***AutoCAD © Structural Detailing - Reinforcement*** programs.

The following objects can be saved to a DWG format file, too:
- foundation plan
- story plans
- vertical sections through a building
- elevation views
- 3D views.

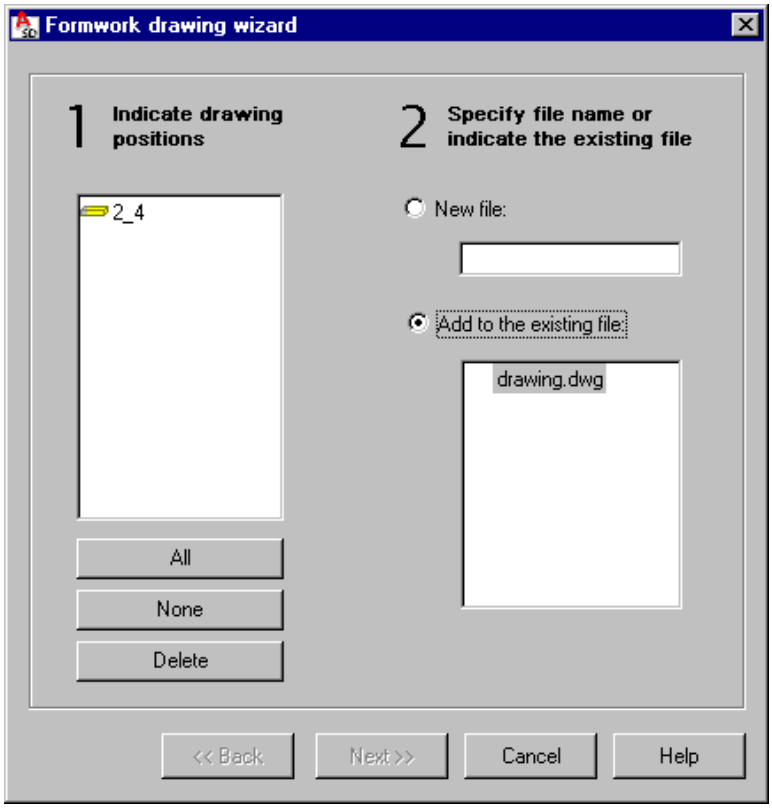
To do it, perform the following operations:
1. indicate a view / plan
2. press the right mouse button and select the *Save to file* command from the context menu
3. specify the file name.

16.3. Export of formwork drawings to AutoCAD © Structural Detailing - Reinforcement

The option is used to transfer formwork drawings of selected positions (elements) of a structure model to ***AutoCAD © Structural Detailing - Reinforcement***.

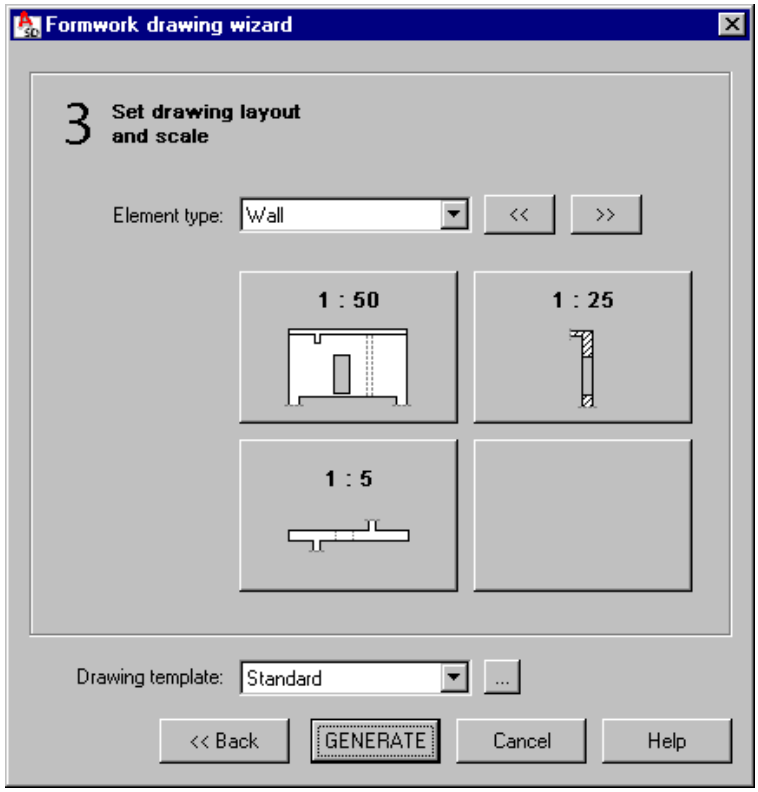
To open the dialog box below, do as follows:

- run positioning of structure elements (see: Automatic positioning)
- select positions (of the same or different types) on the *Positions* tab in the ***Object Inspector*** dialog box
- press the right mouse button and in the context menu, choose the option *Export formwork drawings to AutoCAD © Structural Detailing - Reinforcement.*



NOTE:
In the case of integrated work of the programs ***AutoCAD © Structural Detailing - Formwork Drawings*** *and* ***AutoCAD © Structural Detailing - Reinforcement*** *(when generating formwork drawings in* ***AutoCAD © Structural Detailing - Reinforcement****),* ***AutoCAD © Structural Detailing - Reinforcement*** *keeps the units used in* ***AutoCAD © Structural Detailing - Formwork Drawings****; it means that in the* ***AutoCAD © Structural Detailing - Reinforcement*** *program it is not possible to change the units used (the selection list for changing the work units in* ***AutoCAD © Structural Detailing - Reinforcement*** *is not available).*

In the dialog box above the following operations should be performed:
1. indicate drawing positions in the left part of the dialog box
   drawing positions may be of the same type (the same element type, e. g beams) or of different types, e.g. beams, columns, spread footings, etc.
2. indicate the file where a formwork drawing will be generated:
   - by entering its name in the *New file* edit field
   - by pointing the file saved previously on the disk (in the *Add to the existing file* field)
3. press the **Next >>** button; the following dialog box will appear on the screen:



4. determine the drawing layout and scale for all the selected types of structure elements
   the drawing layout and scales are assumed according to the settings in the ***Drawing template manager*** dialog box
   the *Element type* list contains only these types of elements whose representatives have been selected
5. press the **Generate** button.

AutoCAD © Structural Detailing - Reinforcement integrated with ***AutoCAD © Structural Detailing - Formwork Drawings*** runs. A *.dwg format file is a ***AutoCAD © Structural Detailing - Reinforcement*** file (this is the file integrated with the ***AutoCAD © Structural Detailing - Formwork Drawings*** project).

NOTE:
*Return to the initial **AutoCAD © Structural Detailing - Formwork Drawings** file is possible after selecting the name of this file from the menu (the list of opened files is provided in the Window submenu).*

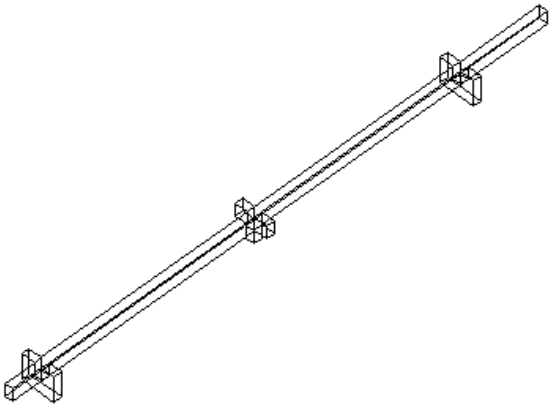
After launching ***AutoCAD © Structural Detailing - Reinforcement*** combined with ***AutoCAD © Structural Detailing - Formwork Drawings*** (when exporting formwork drawings), there is an additional toolbar – ***Formwork Drawings*** available in ***AutoCAD © Structural Detailing - Reinforcement***, with options allowing management of objects read from ***AutoCAD © Structural Detailing - Formwork Drawings***.

The above toolbar (as well as the *RBCRElx* menu in the ***AutoCAD © Structural Detailing - Reinforcement*** context menu) holds the options as follows:

- *Modify graphical parameters* – once this option is selected the ***Modification of display parameters*** dialog box opens on the screen; it enables modifying parameters of graphical display of selected objects (see also: Graphic representation of objects)
- *Create section* - once this option is selected, a section through a chosen object can be defined; to create the section, follow the steps below:

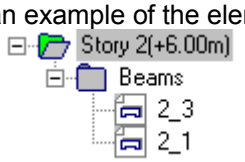
- press the icon on the toolbar shown above
- indicate a view (drawing)
- determine successive points defining the section
- define the section depth
- indicate the section location in a drawing

- *Create 3D view* (the option available only in the context menu) - once this option is selected, a three-dimensional drawing (representation) of a selected object can be created; the 3D view of an element includes all the elements adjoining the chosen element (according to the settings defined in the ***Adjoining elements***) dialog box; to create the 3D view, follow the steps below:
  - select the *Create 3D view* option provided in the RBCRELX context menu
  - indicate the contour of an element for which the 3D view should be generated
  - indicate the location of the 3D view in a drawing; an example 3D view of a beam is shown below.

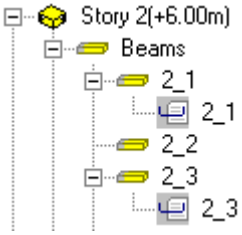


Elements loaded from ***AutoCAD © Structural Detailing - Formwork Drawings*** to the ***AutoCAD © Structural Detailing - Reinforcement*** program are placed in the ***Object Inspector*** dialog boxes:

- on the *Model* tab in ***AutoCAD © Structural Detailing - Reinforcement***, considering the element structure (hierarchy) – see the description of the ***Element manager*** dialog box; an example of the element structure is presented in the drawing below



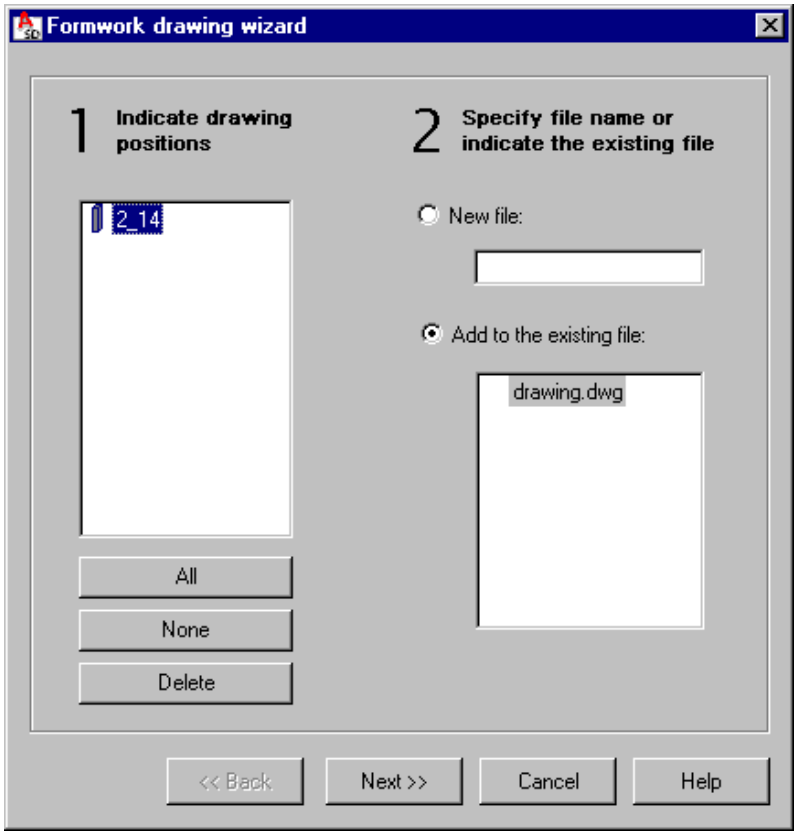
- on the *Positions* tab in ***AutoCAD © Structural Detailing - Formwork Drawings***; positions are marked with the icon which means that a given position has a formwork drawing created in ***AutoCAD © Structural Detailing - Reinforcement***; double-clicking on this icon automatically runs ***AutoCAD © Structural Detailing - Reinforcement*** and opens the formwork drawing of the given position; an example of the position structure is presented in the drawing below.



16.4. Automatic reinforcement

The option is used for automatic generation of reinforcement for a selected position (element) of a structure model.
To open the dialog box below, do as follows:

- run positioning of structure elements (see: Automatic positioning)
- select a position on the *Positions* tab in the ***Object Inspector*** dialog box
- press the right mouse button and in the context menu select the *Automatic reinforcement* option.



In the dialog box above the following operations should be performed:
1. indicate a drawing position (for a selected element type) in the left part of the dialog box
2. indicate the file where a formwork drawing together with reinforcement will be generated:
   - by entering its name in the *New file* edit field
   - by pointing the file saved previously on the disk (in the *Add to the existing file* field).
3. press the **Next >>** button.

Then a ***AutoCAD © Structural Detailing - Reinforcement*** macro appropriate for a chosen structure element type runs. This macro allows defining parameters of necessary reinforcement of a selected element type.
After defining reinforcement parameters, the formwork drawing of the structure element along with the generated reinforcement is opened in ***AutoCAD © Structural Detailing - Reinforcement***.

NOTE:
Reinforcement is generated for a single structure element (one selected position); it is not possible to generate reinforcement for several elements of the same or different types.

RC macros of ***AutoCAD © Structural Detailing - Reinforcement*** for generating reinforcement are composed of two basic parts:
- part for defining geometry of an RC structure element
- part for defining reinforcement parameters.

All options provided in the ***AutoCAD © Structural Detailing - Reinforcement*** macros for definition of reinforcement parameters are available for each type of the RC structure element; however, not all options located on the *Geometry* tabs of the ***AutoCAD © Structural***

Detailing - Reinforcement macros are available after calling up the *Automatic reinforcement* option in ***AutoCAD © Structural Detailing - Formwork Drawings*** (see the table below).

The list of limitations for options provided on the *Geometry* tab of the ***AutoCAD © Structural Detailing - Reinforcement*** macro run from the ***AutoCAD © Structural Detailing - Formwork Drawings*** program (NOTE: if the conditions described below are not satisfied, the *Automatic reinforcement* option is not available in the context menu).

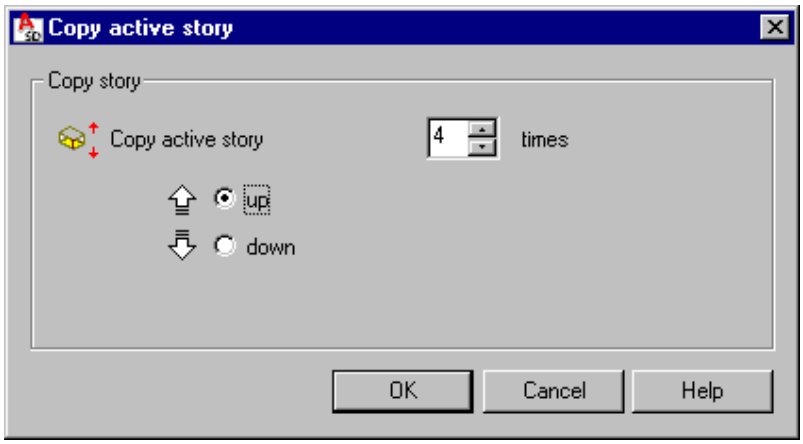
To generate reinforcement for a selected element, the following conditions have to be fulfilled:

Spread footing:	- spread footing shape: only spread footings with a rectangular cross-section - a column must adjoin to the spread footing - column shape: a column with a rectangular or round cross-section
Continuous footing:	- continuous footing shape: only schemes nos. 1 and 3 - cross-section of a continuous footing: only rectangular (to be placed under 1 or 2 walls) - a wall must adjoin to the continuous footing - continuous footing geometry: straight (not arc-shaped) continuous footing - wall geometry: straight (not arc-shaped) wall
Beam:	- beam cross-section: only rectangular - beam geometry: n spans
Column:	- all 4 types of columns are available
Ground beam:	- shape: only scheme no. 1 (beam without cuts) - beam cross-section: only rectangular - beam geometry: straight (not arc-shaped) beam
Lintel:	- shape: single-span beam with a rectangular cross-section - supports: brick supports on both ends of a beam.

17. TOOLS

17.1.Copy story

The option enables copying a selected structure story. It is available in the context menu of the ***Object Inspector*** dialog box (on the *Model* tab).

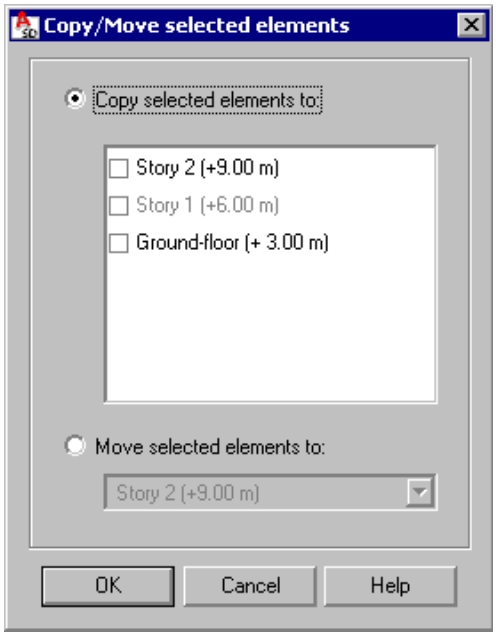


The options provided in the above dialog box allow copying a selected story above or under a chosen story; the following copying parameters can be defined in the dialog box:
- number of repetitions
- direction of copying a story (down or up with respect to a selected story).

17.2.Copy / move selected elements

The option allows copying or moving selected structure elements to a new place indicated by the user. The option is accessible after:
- selecting the command *Formwork Drawings / Tools / Copy /Move elements* from the menu
- pressing the icon .

Once elements to be copied or moved are selected, the dialog box shown in the drawing below appears on the screen.

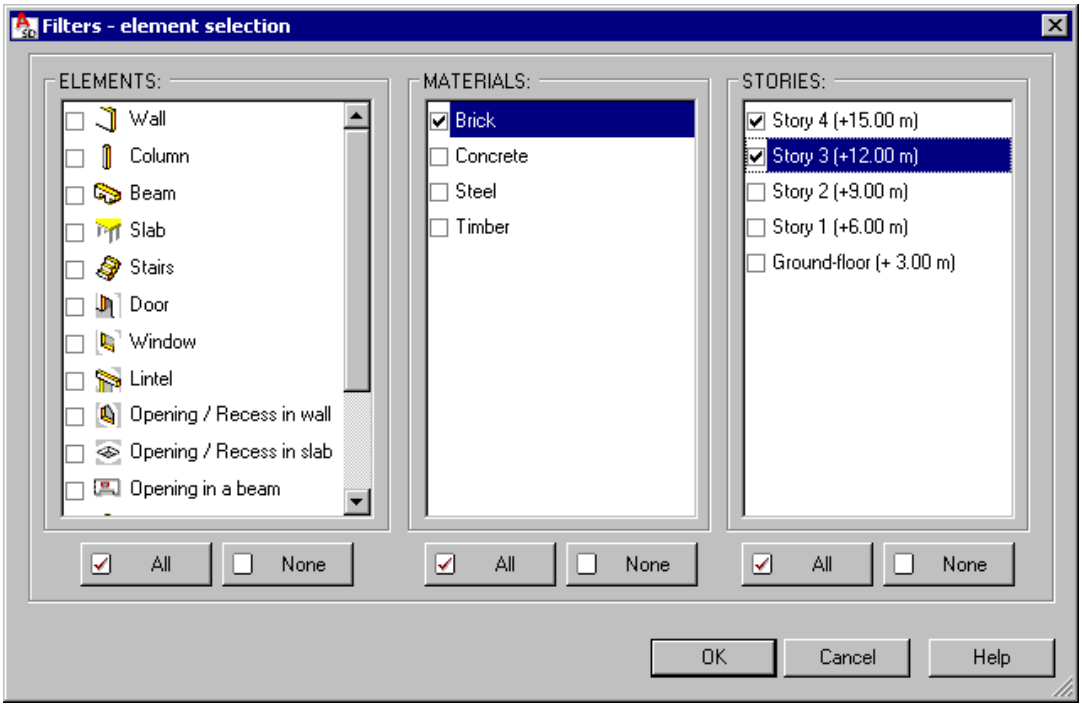


The following options can be chosen in the dialog box above:

- *Copy selected elements to* – from the list of defined stories the user should select a story or stories (√ symbol appears next to them) to which the selected elements should be copied
- *Move selected elements to* - from the list of defined stories the user should select a story to which the selected elements should be moved.

17.3.Filters - element selection by type

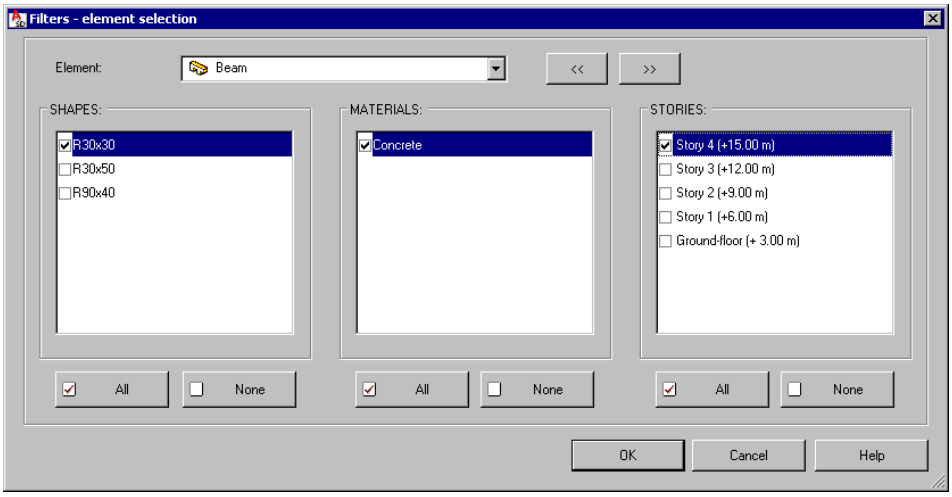
The option is used to define criteria of element selection (by element type). The option is accessible after:
- selecting the menu command: *Formwork Drawings / Tools / Element selection by type*
- pressing the icon .
The dialog box shown in the drawing below appears on the screen then.



Clicking with the mouse cursor in the selection field next to a given object (the √ symbol appears) in the field *ELEMENTS, MATERIALS, STORIES* activates a chosen selection criterion. For example, to activate all RC beams belonging to the stories 4 and 5 defined in a structure, switch on the following options: *Beams* in the *ELEMENTS* field, *Story 4* and *Story 5* in the *STORIES* field and *Concrete* in the *MATERIALS* field.
Pressing the ***All*** button switches on all objects available in the appropriate fields (*ELEMENTS*, *MATERIALS, STORIES*) of the above dialog box. Pressing the ***None*** button switches off all objects available in the appropriate fields (*ELEMENTS*, *MATERIALS, STORIES*) of the above dialog box.

17.4.Filters - element selection by section

The option is used to define criteria of selecting elements (by element sections). The option is available by:
- selecting the menu command: *Formwork Drawings / Tools / Element selection by section*
- pressing the icon .
The dialog box shown in the drawing below appears on the screen then.



In the upper part of the dialog box is the selection list for choosing a structure element type (beam, column, spread footing, etc.) which will the subject of selection. An element on the list can be changed by:

- selecting an element from the list
- pressing the button **>>** (**<<**) - results in selecting the previous (or next) element on the list.

Clicking with the mouse cursor in the selection field next to a given object (the √ symbol appears) in the fields *SHAPES, MATERIALS, STORIES* activates a chosen selection criterion. For example, to select all RC beams with the section R30x50, belonging to the stories 4 and 5 defined in a structure, the following options should be switched on: *R30x50* in the *SHAPES* field, *Story 4* and *Story 5* in the *STORIES* field and *Concrete* in the *MATERIALS* field.

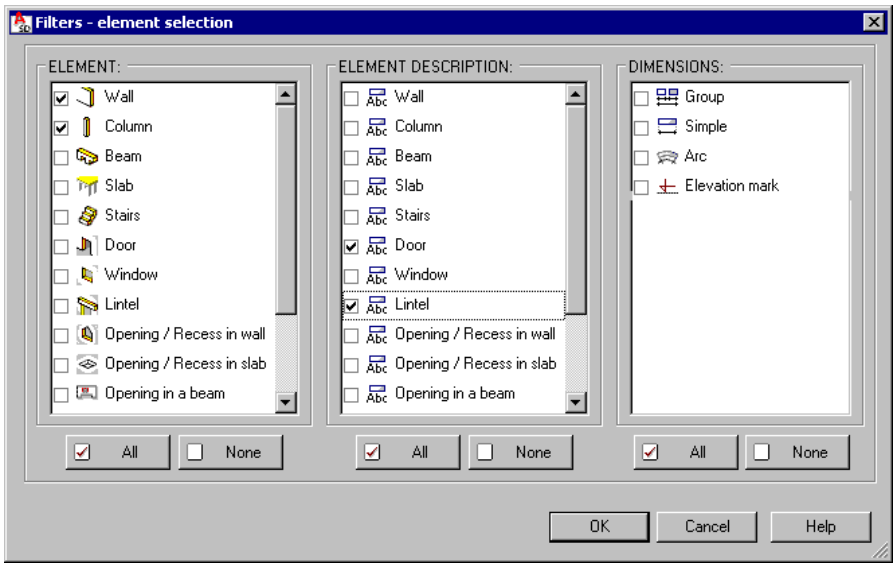
Pressing the ***All*** button switches on all objects available in the appropriate fields (*SHAPES*, *MATERIALS, STORIES*) of the above dialog box. Pressing the ***None*** button switches off all objects available in the appropriate fields (*SHAPES*, *MATERIALS, STORIES*) of the above dialog box.

17.5. Filters - element selection

The option is used to define criteria of selecting descriptions and dimensions of elements. The option is available by:

- selecting the menu command: *Formwork Drawings / Select labels and dimensions*
- pressing the icon  .

The dialog box shown in the drawing below appears on the screen then.



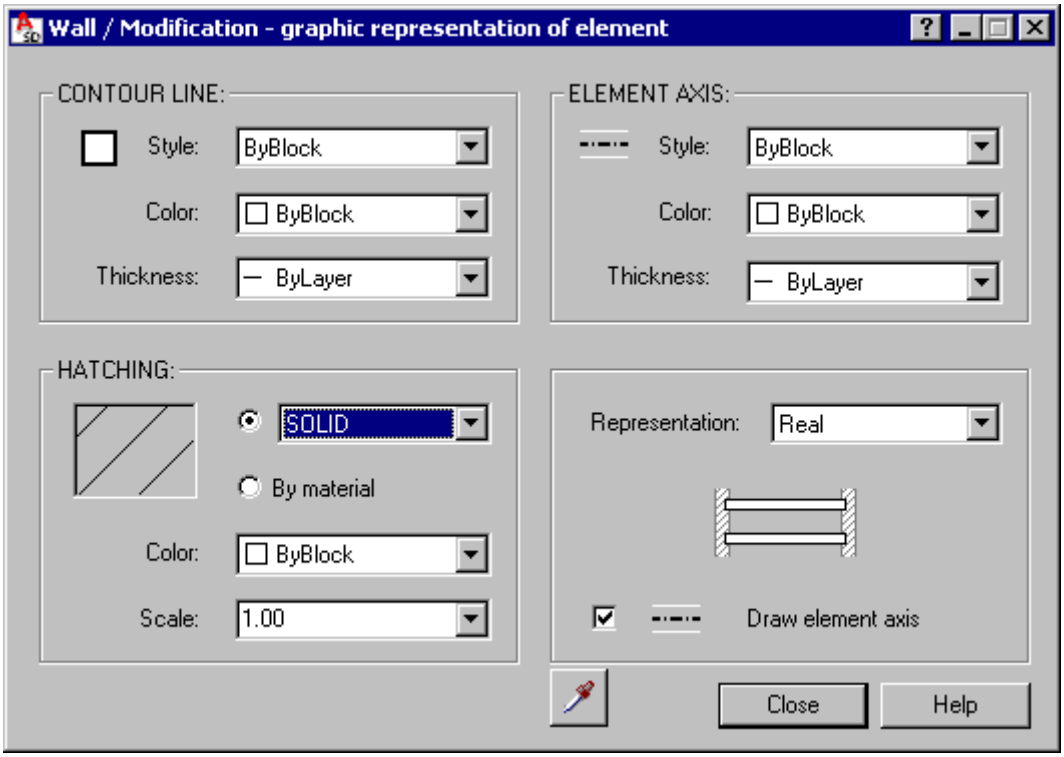
Clicking with the mouse cursor in the selection field next to a given element (the √ symbol appears) in the fields *ELEMENT, ELEMENT DESCRIPTION, DIMENSIONS* activates a chosen selection criterion.
Pressing the ***All*** button switches on all objects available in the appropriate fields (*ELEMENT*, *ELEMENT DESCRIPTION, DIMENSIONS*) of the above dialog box. Pressing the ***None*** button switches off all objects available in the appropriate fields (*ELEMENT*, *ELEMENT DESCRIPTION, DIMENSIONS*) of the above dialog box.

17.6. Modification

17.6.1. Wall - modification of element graphic representation

There is a possibility to change graphic representation of structure elements defined in a building model. The options provided in the dialog box in the drawing below are used to modify graphic representation of walls. The dialog box opens after choosing walls in a structure model and:

- selecting the menu option Formwork Drawings / Tools / Modify graphical parameters
- pressing the icon 
- entering in the command line: RBCX_MOD_PROP.



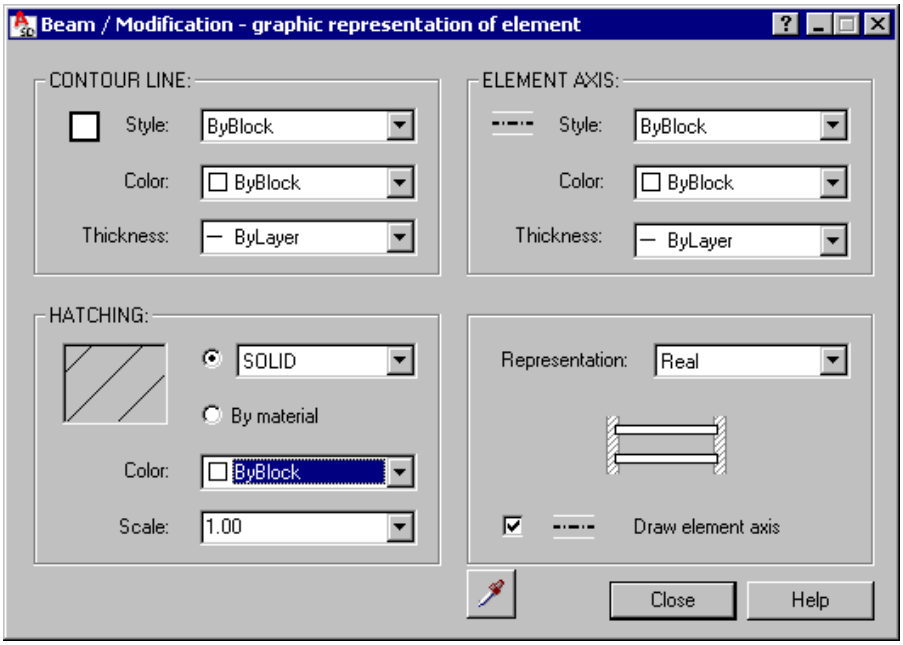
The options concerned with graphic representation of walls, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.2. Beam / ground beam - modification of element graphic representation

There is a possibility to change graphic representation of structure elements defined in a building model. The options provided in the dialog box in the drawing below are used to modify graphic representation of beams. The dialog box opens after choosing beams in a structure model and:

- selecting the menu option Formwork Drawings / Tools / Modify graphical parameters
- pressing the icon 

- entering in the command line: RBCX_MOD_PROP.

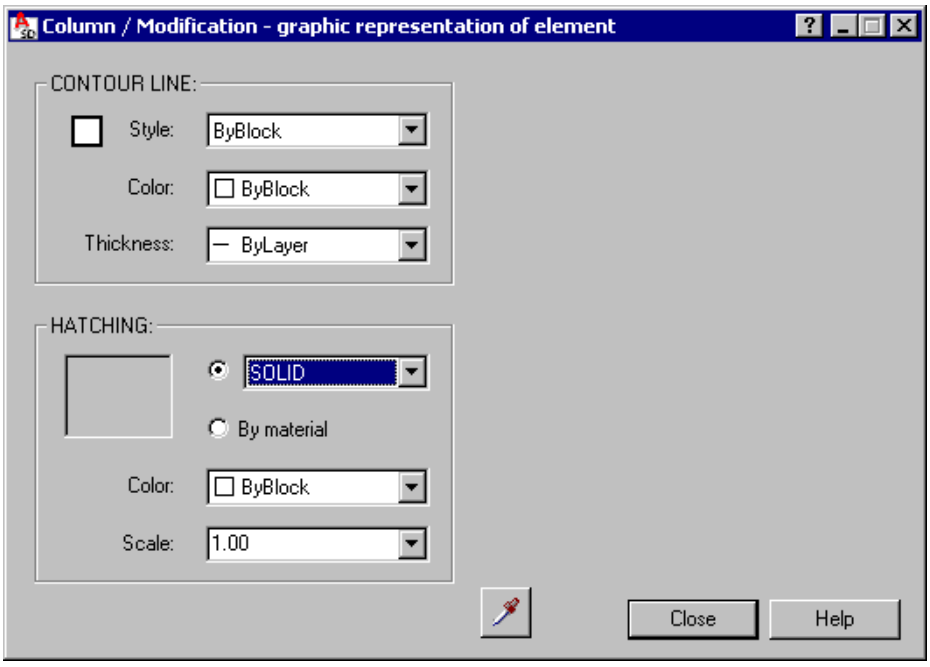


The options concerned with graphic representation of beams, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.3. Column - modification of element graphic representation

There is a possibility to change graphic representation of structure elements defined in a building model. The options provided in the dialog box in the drawing below are used to modify graphic representation of columns. The dialog box opens after choosing columns in a structure model and:

- selecting the menu option *Formwork Drawings / Tools / Modify graphical parameters*
- pressing the icon 
- entering in the command line: RBCX_MOD_PROP.

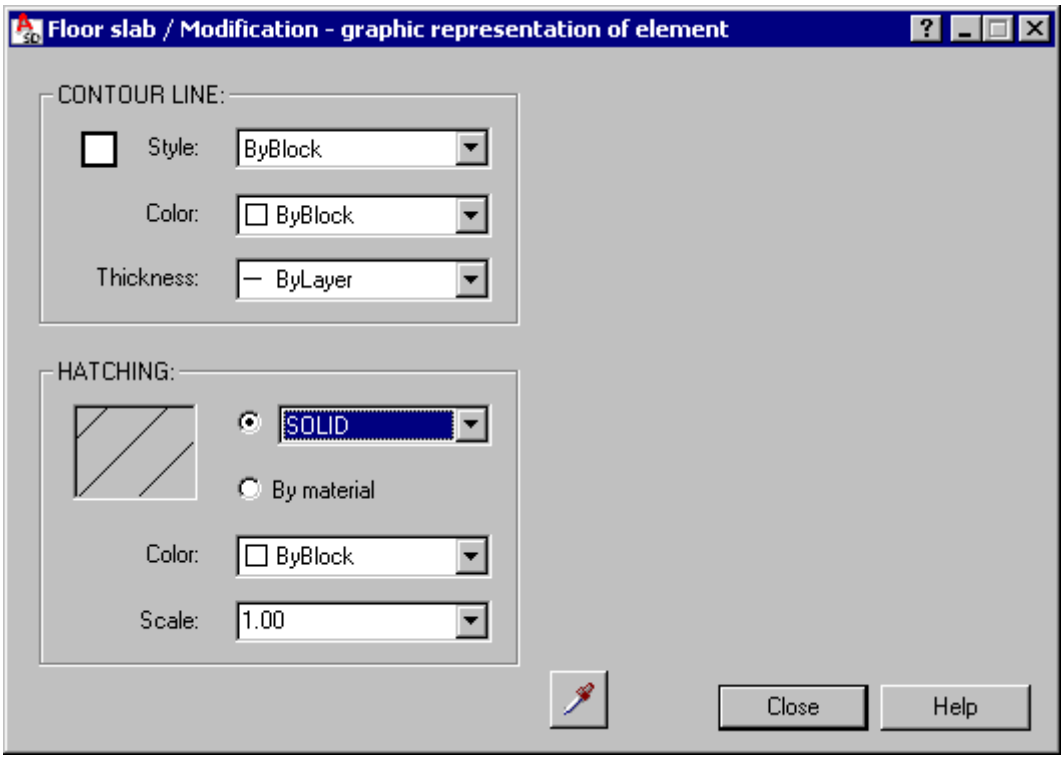


The options concerned with graphic representation of columns, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.4. Slab/raft foundation - modification of element graphic representation

Graphic representation of structure elements defined in a building model can be changed. The options provided in the dialog box in the drawing below are used to modify graphic representation of slabs or raft foundations. The dialog box opens after choosing slabs in a structure model and:

- selecting the menu option *Formwork Drawings / Tools / Modify graphical parameters*
- pressing the icon
- entering in the command line: RBCX_MOD_PROP.

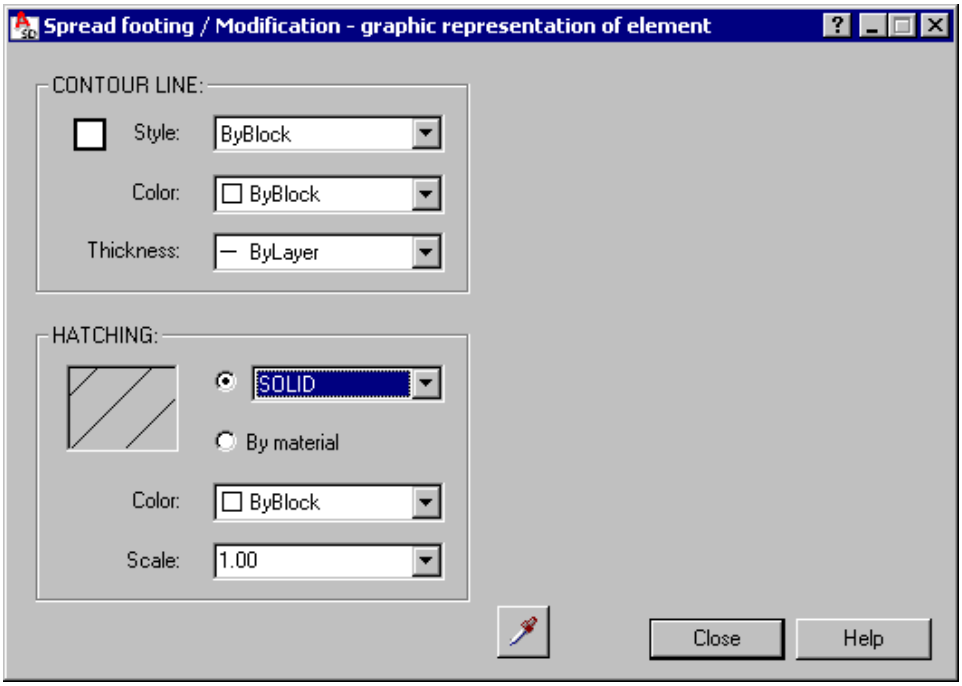


The options concerned with graphic representation of slabs or raft foundations, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.5. Spread footing - modification of element graphic representation

Graphic representation of structure elements defined in a building model may be changed. The options provided in the dialog box in the drawing below are used to modify graphic representation of spread footings. The dialog box opens after choosing spread footings in a structure model and:

- selecting the menu option *Formwork Drawings / Tools / Modify graphical parameters*
- pressing the icon
- entering in the command line: RBCX_MOD_PROP.

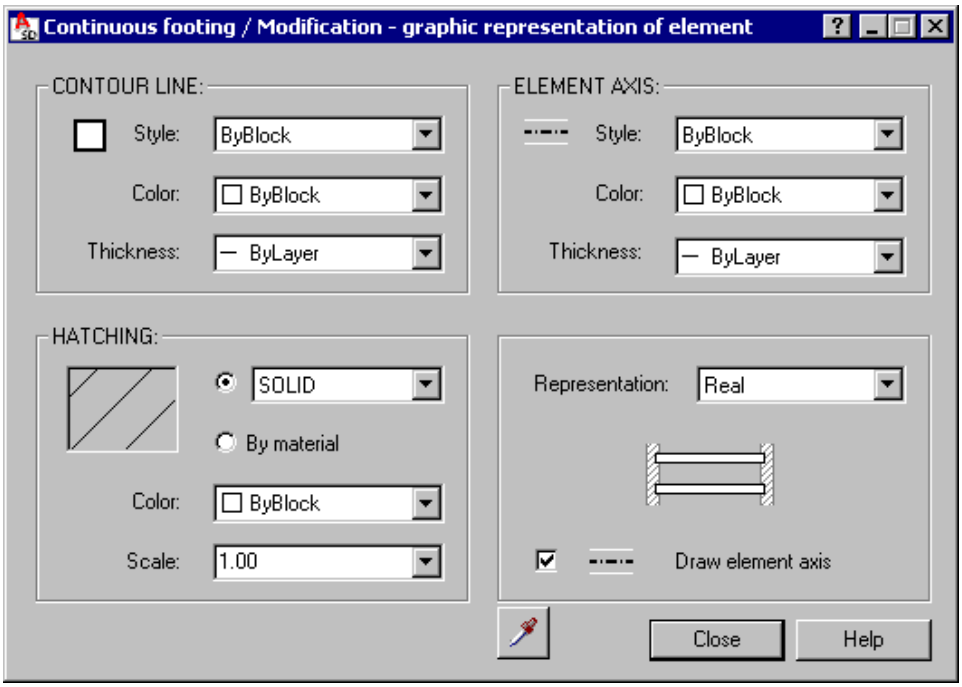


The options concerned with graphic representation of spread footings, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.6. Continuous footing - modification of element graphic representation

There is a possibility to change graphic representation of structure elements defined in a building model. The options provided in the dialog box in the drawing below are used to modify graphic representation of continuous footings. The dialog box opens after choosing continuous footings in a structure model and:

- selecting the menu option Formwork Drawings / Tools / Modify graphical parameters
- pressing the icon
- entering in the command line: RBCX_MOD_PROP.

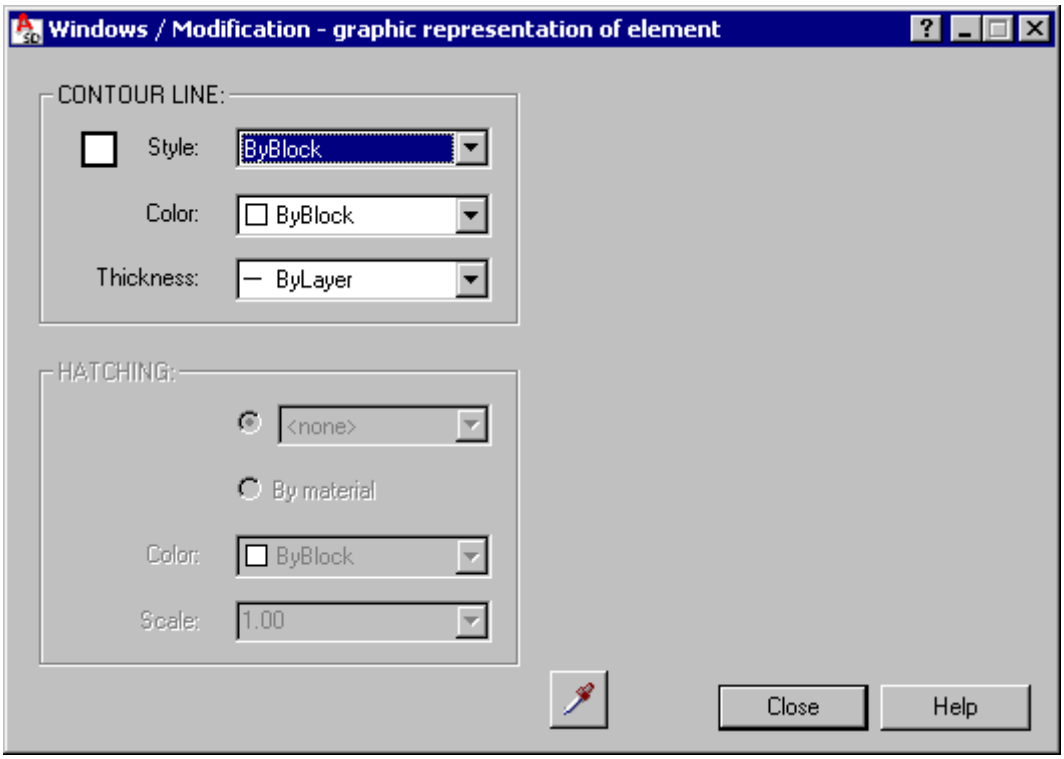


The options concerned with graphic representation of continuous footings, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

17.6.7. Openings (windows, doors) - modification of element graphic representation

There is a possibility to change graphic representation of structure elements defined in a building model. The options provided in the dialog box in the drawing below are used to modify graphic representation of openings (doors, windows). The dialog box opens after choosing openings in a structure model and:

- selecting the menu option *Formwork Drawings / Tools / Modify graphical parameters*
- pressing the icon 
- entering in the command line: RBCX_MOD_PROP.



The options concerned with graphic representation of openings, located in the above dialog box, have been presented in the topic on the method of graphic representation of structure model elements.

18. EXAMPLE - PROCEDURE WHILE WORKING IN THE PROGRAM

18.1.Short example - use of AutoCAD Structural Detailing - Formwork Drawings

Below is presented the procedure applied while working in ***AutoCAD © Structural Detailing - Formwork Drawings***:

1. run ***AutoCAD © Structural Detailing - Formwork Drawings***
2. set basic parameters used in the program:
   - job preferences (units, priorities, databases, parameters of plans and views)
   - building properties, story name
3. define structure elements (on the 1st story)
   - walls, columns, foundations, beams, slabs, etc.
   - additional elements, such as: windows, doors, lintels, etc.
4. copy stories (see the description of the Model tab in the ***Object Inspector*** dialog box)
5. run element positioning (see the description of positioning of building elements)
6. create: plans (projections), views and sections
7. drawings
   - generation of formwork drawings (in co-work with ***AutoCAD © Structural Detailing - Reinforcement***)
   - generation of reinforcement for RC elements (in co-work with ***AutoCAD © Structural Detailing - Reinforcement***).