User Guide

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1. PROGRAM DESCRIPTION

1.1. General description of the program

AutoCAD Structural Detailing is used to prepare detailed drawings of designed steel structures. After completing the stage of architectonic structure design and generating a structure calculation model (this stage comprises calculations and verification of structure elements), the stage of preparing project technical documentation with required final drawings takes place.

AutoCAD Structural Detailing - ***Steel*** is a program which allows generation of complete detailing documentation of steel structures (drawings + tables of materials). The process of creating drawings in ***AutoCAD Structural Detailing*** - ***Steel*** is automatic. Drawings are generated based on a 3D structure model using the options available in ***AutoCAD Structural Detailing*** - ***Steel*** with the possibility to apply options available in the AutoCAD ® program. In case a structure model is modified, it is possible to update automatically the created documentation.

AutoCAD Structural Detailing - ***Steel*** is a program enabling generation of detailing drawings; it is divided into a few parts which are responsible for:

1. link with the calculation program (***Autodesk Robot Structural Analysis***): data exchange, structure recalculation, if needed
2. edition of a drawing or part of a drawing (projections, cuts, etc.) - including additional drawing elements, correction of existing structure elements, adding structure element dimensions
3. generation of final drawings
4. printout management.

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

- generation of a structure model and structure calculations
- structure design
- generation of final drawings allowing edition of prepared drawings.

Basic tasks that are performed in ***AutoCAD Structural Detailing*** - ***Steel*** after reading in a structure model from the calculation program, include:

- possibility to define connections of structure elements
- posibility of elements' machining (fitting to a line or object, chamfering)
- possibility to single out assemblies and groups in a structure model – according to the actual procedure of structure assembly
- possibility to define user descriptions of parts
- possibility to obtain a drawing of a whole structure as well as to prepare the overall bill of materials
- generation of detailed drawings.

The following objects have been distinguished in the program:

- Structure model, structure model elements - real (3D) structure elements; for them final drawings are created
- Position - an object concerned with organization of structure model elements; a (letter – digit) designation of identical parts or assemblies in a structure
- Document - a set of views of a position in determined 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
- (Document) template - a predefined set of views composing a logical whole (e.g. 3 projections of a given object); each document is created based on a template determining which drawings (projections, sections) should be included in the document

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

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

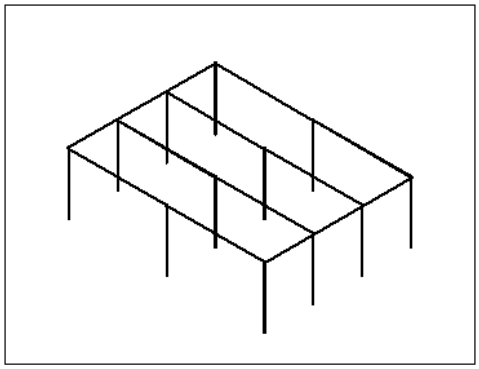
1.2. Stages of work on structure project drawings

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

1. Defining or loading a structure model

A structure model may be defined in ***AutoCAD Structural Detailing - Steel***; it is also possible to prepare it in another program and open in ***AutoCAD Structural Detailing - Steel*** (see: AutoCAD Structural Detailing - Robot link)

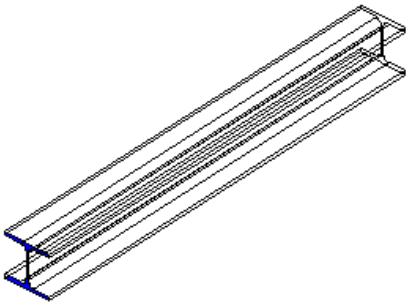
A structure model - three-dimensional elements of a real 3D structure; for these elements drawings will be prepared.



All operations are performed in the model layout. The model layout is an object of the AutoCAD ® program which is used to work with a structure model.

2. Assigning positions to structure elements

A position - an object concerned with organization of structure model elements; one or several structure elements is/are assigned a position, for which a common set of drawings may be generated.

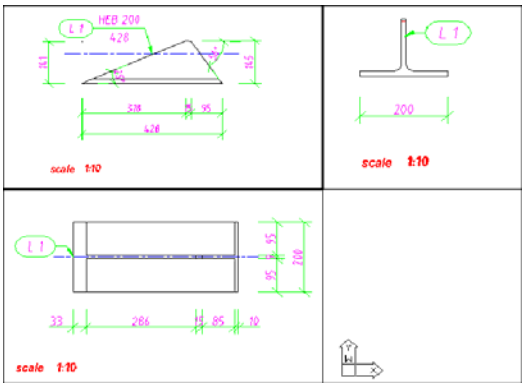


It is available on two tabs of the ***Object inspector*** dialog box: *Model* and *Parts Edition*; positions defined are presented on the *Positions* tab.

3. Defining documents

A document - a set of drawings for a position (for each position any number of documents may be generated); a document consists of views (a document cannot be printed).

Documents are generated on the basis of templates, in other words, predefined sets of views composing a logical whole (e.g. 3 projections of a given object); a template determines which drawings (projections, sections) should be included in a document.

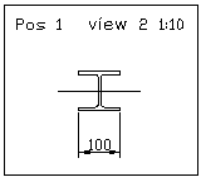


All operations are performed in edition layouts.

The edition layout is an object of the AutoCAD ® program. Only one document may be active at a time and only this document may be edited in the edition layout. Contents of the edition layout should not be printed.

4. Defining (editing) views/projections

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

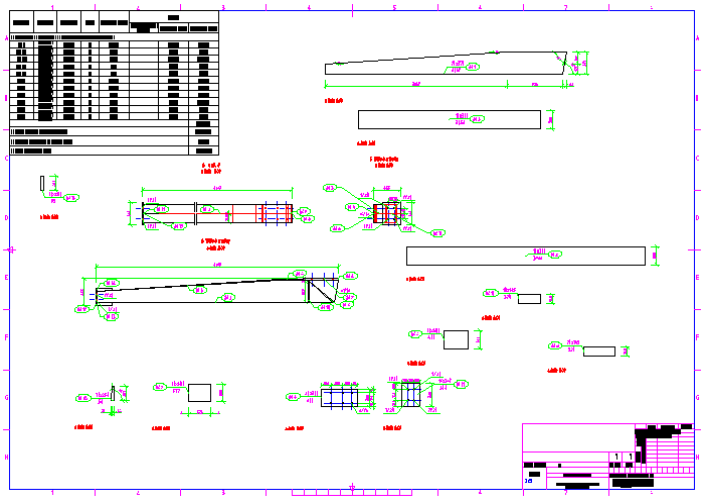


Projections of assemblies are prepared on the basis of the coordinate system of the main part (by default it is the longest part of an assembly). To modify projections, the user may either rotate the view or, before drawings are generated, change the main part of the assembly (using the command *Change main part of group* provided in the context menu of the ***Object Inspector***, on the *Model* tab).

Edition is performed in the edition layout.

5. Generating a printout

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



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

1.3. Options available in the menu

All the options available in ***AutoCAD Structural Detailing*** are presented below (menu of the steel part). The list includes: option position in the context menu, icon symbolizing the option, command activating the option from the command line as well as short description of the option.

Create workframe


Opens the ***Workframe*** dialog box; the option allows application of a workframe which makes definition of a structure model in 3D space easier.
Menu: *Steel / Workframes / Create workframe* command
Command line: RBCS_WF

Add node


The option allows creating an additional node in the existing workframe.
Menu: *Steel / Workframes / Add node* command
Command line: RBCS_ADDNODE

Delete node


The option allows deleting a node in the existing workframe.
Menu: *Steel / Workframes / Delete node* command
Command line: RBCS_WF_DELNODE

Add line


The option allows creating an additional line in the existing workframe.
Menu: *Steel / Workframes / Add line* command
Command line: RBCS_WF_ADDLINE

Delete line


The option allows deleting a line in the existing workframe.
Menu: *Steel / Workframes / Delete line* command
Command line: RBCS_WF_DELLINE

Profiles


Opens the ***Profile*** dialog box; the option enables defining a bar of a selected section.
Menu: *Steel / Profile* command
Command line: RBCS_PROFILE

Plates

Opens the ***Plate*** dialog box; the option enables plate definition.
Menu: *Steel / Plates* command
Command line: RBCS_PLATE

Bent profiles

Opens the ***Bent profile*** dialog box; the option enables defining a curvilinear bar of selected profile.
Menu: *Steel / Bent profiles* command
Command line: RBCS_BENT

User parts

Opens the ***User parts*** dialog box; the option enables defining (selecting) user objects.
Menu: *Steel / User parts* command
Command line: RBCS_USERPART

Grates

Opens the ***Grates*** dialog box; the option enables defining a grate of specified dimensions and weight.
Menu: *Steel / Grates* command
Command line: RBCS_GRATE

Compound profiles

Opens the ***Compound profiles*** dialog box; the option enables defining a compound section bar.
Menu: *Steel / Compound profiles* command
Command line: RBCT_MMACRO

Castellated beam

Opens the ***Special profiles - castellated beam*** dialog box; the option enables defining a castellated beam.
Menu: *Steel / Castellated beam* command
Command line: RBCT_MMACRO

Connections
The submenu and toolbar to automatically create different types of connections between selected elements of steel structures.

Plate girders
The submenu and toolbar to automatically create plate girders; using this software version you can define a column, beam and multisegment beam.

Cut by line

The option enables cutting a part (profile, plate) so that it fits the plane determined by a line.
Menu: *Steel / Machining / Cut by line* command
Command line: RBCS_CUTBYLINE

Fit to line

The option enables trimming a given profile so that it fits the plane determined by a line, perpendicular to UCS.
Menu: *Steel / Machining / Fit to line* command
Command line: RBCS_FITTOLINE

Fit to polyline

The option enables trimming a given part so that it fits the broken plane determined by a polyline, perpendicular to UCS.
Menu: *Steel / Machining / Fit to polyline* command
Command line: RBCS_FITTOPOLY

Fit to object

Opens the ***Fit to object*** dialog box. The option enables trimming a given part so that it fits another steel part.
Menu: *Steel / Machining / Fit to object* command
Command line: RBCS_FITOBJ

Cut by bisector


Opens the ***Cut by bisector*** dialog box. The option enables trimming two intersecing profiles so that they fit each other using a bisector of the angle between their axes.
Menu: *Steel / Machining / Cut by bisector* command
Command line: RBCS_CUTBISEC

Lengthen


The option allows the user to lengthen an existing profile along its axis.
Menu: *Steel / Machining / Lengthen* command
Command line: RBCS_LENGTHEN

Shorten


The option allows the user to shorten an existing profile along its axis.
Menu: *Steel / Machining / Shorten* command
Command line: RBCS_SHORTEN

Bent plate


The option enables bending an indicated plate.
Menu: *Steel / Machining / Bent plate* command
Command line: RBCS_BENTPLATE

Remove bending from plate


The option enables removing an existing bend from the indicated plate.
Menu: *Steel / Machining / Remove bending from plate* command
Command line: RBCS_UNBENTPLATE

Split profile


The option enables division of an existing profile into several independent parts.
Menu: *Steel / Machining / Split profile* command
Command line: RBCS_SPLIT

Merge profiles


The option enables connecting independent profiles into one profile.
Menu: *Steel / Machining / Merge profiles* command
Command line: RBCS_MERGE

Split plate


The option enables division of an existing plate into several independent parts. The user should indicate the plate and define a cutting line (the cutting line can be defined as a line, polyline or arc). Moreover, a spacing between parts of the split plate can be defined (the spacing is divided symmetrically on both sides of the cutting line).
NOTE: If the cutting line intersects chamfers defined in the plate, then these chamfers are removed from the plate. If the cutting line does not intersect chamfers, then they remain in the split parts of the plate.
Menu: *Steel / Machining / Split plate* command
Command line: RBCS_SPLIT

Merge plates


The option enables connecting two independent plates into one plate. To connect two plates into one, the following conditions have to be satisfied:
- plates have to lie in one plane
- plates have to touch each other.

The result of merging plates is a plate (type: user) made of the material of which the plate indicated first is made. All machinings of merged plates are kept.
Menu: *Steel / Machining / Merge plates* command
Command line: RBCS_MERGE

Add plate vertex


The option allows adding a vertex (or vertexes) in a polyline which defines an indicated plate. Adding a new plate vertex changes the type of the plate (to the user-defined plate).
Menu: *Steel / Machining / Add plate vertex* command
Command line: RBCS_ADDVERTEX

Remove plate vertex


The option allows removing a vertex (or vertexes) belonging to a polyline which defines an indicated plate. NOTE: removing a plate vertex is possible only then, when after removing the vertex, the plate contour will be a closed polyline (it means that, for example, it is impossible to remove a vertex from a triangle-shaped plate).
Removing a new plate vertex changes the type of the plate (to the user-defined plate).
Menu: *Steel / Machining / Remove plate vertex* command
Command line: RBCS_REMOVEVERTEX

Chamfer plate corner


The option enables the operation of chamfering a plate corner. There are a few types of chamfering available.
Menu: *Steel / Machining / Chamfer plate corner* command
Command line: RBCS_CHAMFCORN

Insert arc


The option enables inserting an arc (bent profile) between two indicated profiles.
After indicating two profiles, the following question appears in the command line: *'Do you want to join profiles?'* Depending on the answer to this quiestion, the resultant profile may be:
- NO: a composition of three profiles (two initial profiles and a bent profile)
- YES: one profile (three component profiles will be joined into one profile).

Menu: *Steel / Machining / Insert arc* command
Command line: RBCS_BENDCORNER

Drill/Bolt


Opens the ***Drilling*** dialog box; the option enables definition of openings and bolts.
Menu: *Steel / Machining / Drill\Bolt* command
Command line: RBCS_DRILL

Weld


Opens the ***Welds*** dialog box; the option enables definition of welded connections.
Menu: *Steel / Machining / Weld* command
Command line: RBCS_WELD

Multi-weld


Opens the ***Welds*** dialog box; the option allows defining welded connections for several elements simultaneously. Once several objects are selected (profiles, plates), the program performs an analysis of the contact area for the indicated elements. After selecting a weld type in the ***Welds*** dialog box, welded connections of the indicated elements are generated.
Menu: *Steel / Machining / Multi-weld* command
Command line: RBCS_MWELD

Copy weld


The option enables copying selected welds.
Menu: *Steel / Machining / Copy weld* command
Command line: RBCS_COPYWELD

Copy drills / bolts

The option enables copying selected drills or bolts.
Menu: *Steel / Machining / Copy drills/bolts* command
Command line: RBCS_COPYDRILL

Adopt drills

The option enables adopting all properties of a selected drill.
Menu: *Steel / Machining / Adopt drills* command
Command line: RBCS_TAKEDRILL

Insert bolts

The option enables inserting a bolt into a selected drill.
Menu: *Steel / Machining / Insert bolts* command
Command line: RBCS_BOLTHOLES

Explode group of holes/bolts

The option enables exploding a group of holes/bolts into basic objects of the AutoCAD ® program.
Menu: *Steel / Machining / Explode group of holes/bolts* command
Command line: RBCS_EXPLDRILL

Remove bolts

The option enables removing bolts.
Menu: *Steel / Machining / Remove bolts* command
Command line: RBCS_REMBOLTS

Copy connection

The option enables copying a selected connection.
Menu: *Steel / Machining / Copy connection* command
Command line: RBCS_CONNECTION

Update connections

The option enables updating selected connections after modifications in structure geometry; connections are adjusted to new structure dimensions; the connection update mode depends on the option settings in the Project preferences dialog box.
Menu: *Steel / Machining / Update connections* command
Command line: RBCS_CONNUPDATE

Tools

Explode

The option enables exploding some complex elements (groups, connections) into single elements.
Menu: *Steel / Tools / Explode* command
Command line: RBCS_EXPLODE

Modify properties

The option allows modifying parameters of selected objects; the program opens the dialog box where the user may change parameter values for a selected object.
Menu: *Steel / Tools / Modify properties* command
Command line: RBCS_PROPERTIES

Family Manager

Opens the ***Family Manager*** dialog; use this option to manage families of steel elements.
Menu: *Steel / Tools / Family Manager* command
Command line: RBCS_FAMILIES

Group

The option allows the user to create groups of selected objects.
Menu: *Steel / Tools / Group*
Command line: RBCS_GROUP

Group assemblies


The option allows the user to group assemblies based on the defined connections.
Menu: *Steel* / *Tools* / *Group assemblies* command
Command line: RBCS_ASSEMBLY

User sections


Use this option to define a profile and save it to a profile database or locally, in a project.
Menu: *Steel* / *Tools* / *User sections* command
Command line: RBCS_USRPROFILE

Inquire

Object info


Opens the ACAD text window; the option enables obtaining information about the indicated object.
Menu: *Steel* / *Tools* / *Inquire* / *Object info* command
Command line: RBCS_INQ_ELEMENT

Group


Opens the ACAD text window; the option enables obtaining information about the group to which the indicated object belongs.
Menu: *Steel* / *Tools* / *Inquire* / *Group* command
Command line: RBCS_INQ_GROUP

Assembly


Opens the ACAD text window; the option enables obtaining information about the assembly to which the indicated object belongs.
Menu: *Steel* / *Tools* / *Inquire* / *Assembly* command
Command line: RBCS_INQ_ASSEMBLY

Center of gravity


Opens the ACAD text window; the option enables calculating coordinates of the gravity center for a structure or its selected part.
Menu: *Steel* / *Tools* / *Inquire* / *Center of gravity* command
Command line: RBCS_INQ_COG

Model size


Opens the ACAD text window; the option enables calculation of dimensions and weight of a structure or its selected part.
Menu: *Steel* / *Tools* / *Inquire* / *Model size* command
Command line: RBCS_INQ_MSIZE

Collision detection


The option enables finding a collision of structure components (profiles, plates, bolts, etc.).
The command line shows a number of detected collisions between structure elements. It is possible to save collision tests; they are saved in the 'RBCS_Collision_Detection' layer and represented by AutoCAD ® 3D Solid objects.
Menu: *Steel* / *Tools* / *Inquire* / *Collision detection* command
Command line: RBCS_INQ_COLISION

Filters

Show current selection


The option allows filtering objects in a drawing (only selected objects are displayed). The filter is also accessible from the ***Object Inspector*** dialog box.
Menu: *Steel* / *Tools* / *Filters* / *Show current selection* command
Command line: RBCT_SHOWCURSEL

Hide current selection

The option allows filtering objects in a drawing (only unselected objects are displayed). The filter is also accessible from the ***Object Inspector*** dialog box.
Menu: *Steel / Tools / Filters / Hide current selection* command
Command line: RBCT_HIDECURSEL

Show all

The option allows filtering objects in a drawing (all objects are displayed). The filter is also accessible from the ***Object Inspector*** dialog box.
Menu: *Steel / Tools / Filters / Show all* command
Command line: RBCT_SHOWALLOBJ

Workspace

Show / Hide Inspector

Use this option show or hide the *Object Inspector* dialog which is by default displayed in the left part of the screen.
Menu: *Steel / Tools / Workspace / Show / Hide Inspector* command
Command line: RBCTOI

Object snap settings

Opens the ***Snap Mode*** dialog; use this option to set active snap points for steel profiles.
Menu: *Steel / Tools / Workspace / Object snap settings* command
Command line: RBCS_PSNAP

Lock X direction

Use this option to select or deselect blocking the local X axis of the bar profile; if this option is selected, you can perform the operation of extending a bar only along the bar axis; if this option is deselected, a profile can be extended in any direction you selected.
Menu: *Steel / Tools / Workspace / Lock X direction* command
Command line: RBCS_LOCKX

2D copying / moving
Options for moving or copying selected elements in a specified direction (along the axis or in the plane).

Select assembly

Use this option to select a part of an assembly; it selects a whole assembly that this part belongs to.
Menu: *Steel / Tools / Workspace / Select assembly* command
Command line: RBCS_SELASSEMBLY

Dynamic UCS

Use this option to fit the view plane and active UCS to the UCS of a selected object.

Menu: *Steel / Tools / Workspace / Dynamic UCS* command
Command line: RBCS_DYNUCS

Elevation mark - model

Use this option to insert an elevation mark in a selected plane and in a selected location in the drawing
Menu: *Steel / Tools / Workspace / Elevation marl - model* command
Command line: RBCS_MELEV

Clipping plane on

Use this option to limit the model view with 2 planes
Menu: *Steel / Tools / Workspace / Clipping plane on* command
Command line: RBCS_CLIPON

Clipping plane off

Use this option to deselect limiting the model view with 2 planes.
Menu: *Steel* / *Tools* / *Workspace* / *Clipping plane off* command
Command line: RBCS_CLIPOFF

Positions

Assign position

Opens the ***Manual positioning*** dialog box; the option enables the user to ascribe the position to a selected part.
Menu: *Steel* / *Positions* / *Assign position* command
Command line: RBCS_MANUALPOS

Run automatic positioning

Opens the ***Automatic positioning*** dialog box; the option enables the user to ascribe positions to structure elements automatically.
Menu: *Steel* / *Positions* / *Run automatic positioning* command
Command line: RBCS_AUTOPOS

Remove position

The option allows the user to remove the ascribed position from a selected part.
Menu: *Steel* / *Positions* / *Remove position* command
Command line: RBCS_REMPOS

Verify positions

Opens the ***Position verification*** dialog box; the option enables the user to run verification of selected positions.
Menu: *Steel* / *Positions* / *Verify positions* command
Command line: RBCS_VERIFYPOS

Parametric structures
Submenu and toolbar with options enabling definition of typical elements of steel structures.

Multi-bay frame

The option allows definition of a parametric structure - a multi-bay frame.
Menu: *Steel* / *Parametric structures* / *Multi-bay frame* command

Roof truss

The option allows definition of a parametric structure – a roof truss.
Menu: *Steel* / *Parametric structures* / *Roof truss* command

Bracing

The option allows definition of a parametric structure – a bracing.
Menu: *Steel* / *Parametric structures* / *Bracing* command

Brace

The option allows definition of a parametric structure – a brace.
Menu: *Steel* / *Parametric structures* / *Brace* command

Stairs

The option allows definition of a parametric structure – stairs.
Menu: *Steel* / *Parametric structures* / *Stairs* command

Stairs spiral

The option allows definition of a parametric structure – spiral stairs.
Menu: *Steel* / *Parametric structures* / *Stairs spiral* command

Staircase

The option allows definition of a parametric structure – a staircase.
Menu: *Steel* / *Parametric structures* / *Staircase* command

Railing

The option allows definition of a parametric structure – the railing.
Menu: *Steel* / *Parametric structures* / *Railing* command

Ladder

The option allows definition of a parametric structure – the ladder.
Menu: *Steel* / *Parametric structures* / *Ladder* command

Cage

The option allows definition of a parametric structure – the cage.
Menu: *Steel / Parametric structures / Cage* command

Purlin spreading

The option allows defining a distribution of purlins.
Menu: *Steel / Parametric structures / Purlin spreading* command

Grate spreading

The option allows defining a distribution of grates.
Menu: *Steel / Parametric structures / Grate spreading* command

Drawings edition

Change style

Opens the ***Design styles*** dialog box; the option permits changing a design style for a selected drawing.
Menu: *Steel / Drawings Edition / Change style* command
Command line: RBCS_CNGSTYLE

Adjust style

Opens the ***Design styles*** dialog box; the option permits changing a design style for a selected drawing.
Menu: *Steel / Drawings Edition / Adjust style* command
Command line: RBCS_ADJSTYLE

Lock dimensions

Use this option to freeze generated drawings with existing dimensions or descriptions. This is the option to protect existing drawings (changed manually by the user) from losing the changes while the drawing is updated. Automatically generated drawings are quite often changed by adding, deleting or moving dimensions or descriptions. To avoid losing changes while the drawing is updated, select this option and lock dimensions in the drawing.

After executing this command, the command line displays a message about dimensions in the drawing (locked dimensions and dimensions that are not locked); in addition, an icon displays in the ***Object Inspector*** pane.

Menu: *Steel / Drawings Edition / Lock dimensions* command
Command line: RBCS_DIMLOCK

Edit view in full screen

Selection of this option enables edition of a created drawing on the full monitor screen.
Menu: *Steel / Drawings Edition / Edit view in full screen* command
Command line: RBCT_FULLSCREEN_ON

Full screen off

Selection of this option switches off edition of a created drawing on the full monitor screen.
Menu: *Steel / Drawings Edition / Full screen off* command
Command line: RBCT_ FULLSCREEN_OFF

Insert weld symbol

The option enables the user to insert a weld symbol in a drawing.
Menu: *Steel / Drawings Edition / Insert weld symbol* command
Command line: RBCS_ADDWELD

Insert elevation mark

The option enables the user to insert an elevation mark in a drawing.
Menu: *Steel / Drawings Edition / Insert elevation mark* command
Command line: RBCS_ADDELEV

Insert part label

The option allows the user to insert description of parts in a drawing.
Menu: *Steel / Drawings Edition / Insert part label* command
Command line: RBCS_PARTLBL

Insert assembly label

The option allows the user to insert an assembly description in a drawing.
Menu: *Steel / Drawings Edition / Insert assembly label* command
Command line: RBCS_ASLBL

Add view (detail)

The option enables the user to create a new view (view of a section detail) in a drawing.
Menu: *Steel / Drawings Edition / Add view (detail)* command
Command line: RBCS_ADDDETAILVIEW

Add cut

The option enables the user to create a new view (view of a section) in a drawing. To create a cut:
1. specify points defining the cut
2. specify a range
3. specify a cut name
4. select a drawing template.

Menu: *Steel / Drawings Edition / Add view (detail)* command
Command line: RBCS_ADDCUT

Automatic cuts

Use this option to automatically create new views (cuts) in the drawing.
Menu: *Steel / Drawings Edition / Automatic cuts* command
Command line: RBCS_AUTOCUT

Move cut

Use this option to change a cut in the drawing. You can change a position or range of the cut, and a symbol description.
Menu: *Steel / Drawings Edition / Move cut* command
Command line: RBCS_MOVCUT

Change view scale

The option enables the user to change scale of a view (projection) in a drawing.
Menu: *Steel / Edit drawings / Change view scale* command
Command line: RBCS_CNGSCALE

Rotate view

The option allows the user to rotate a view (projection) in a drawing.
Menu: *Steel / Drawings Edition / Rotate view* command
Command line: RBCS_ROTATEVIEWPORT

Shorten drawing

The option allows the user to shorten a selected element in a drawing; the option is useful in the case of long elements (in particular, for these element segments which do not contain any snap points); the element shortening is presented as a gap in the element subjected to shortening.
Menu: *Steel / Drawings Edition / Shorten drawing* command
Command line: RBCS_SHORTDRAW

Delete all shortenings

The option allows the user to delete all the defined element shortenings in a drawing.
Menu: *Steel / Drawings Edition / Delete all shortenings* command
Command line: RBCS_DELSHORTDRAW

Dimensioning General / Detailed

Use this option to refresh the view (including the dimensioning of elements) based on the settings of styles that are associated with details. You can select the following dimensioning style: standard, detailed or schematic.

Menu: *Steel / Drawings Edition / Dimensioning Genaral / Detailed* command
Command line: RBCS_CNGDET

Explode drawing

The option enables the user to explode drawings into basic objects of the AutoCAD ® program.
Menu: *Steel / Drawings Edition / Explode drawing* command
Command line: RBCS_EXPLODE

Drawing template designer

The option enables the user to create and modify drawing templates.
Menu: *Steel / Drawings Edition / Drawing template designer* command
Command line: RBCS_ADDTEMPLATE

Automatic printouts

Use this option to create automatic printouts (one part in one drawing). NOTE: This option is not available for schemes of elements disposition.
Menu: *Steel / Automatic printouts* command
Command line: RBCS_AUTOPRINT

Engineering drawings

Engineering drawings

Use this option to define a group of structure elements to make a scheme drawing of a structure model. Drawings presenting structure elements in a simplified way are created for a defined scheme.
Menu: *Steel / Engineering drawings / Engineering drawings command*
Command line: RBCS_SCHEME

Create mark

Use this option to create a mark, that is an object consisting of selected structure elements. It is usually an element (composed of parts such as plates, profiles, user parts or connectors) which has a different function in structure calculations, such as column, beam, frame or bracing.

Marks that have the same function in the structure and that have the same geometry can be labeled with the same name (referred to as a mark). The degree of similarity of elements (and of names of marks) is user-defined.

The basic functionality of a mark is that you can draw and describe it as a single element in the drawing.

To define a mark:

1. select elements that should belong to the mark
2. specify a mark name
3. define values of forces.

Menu: *Steel / Engineering drawings / Create mark command*
Command line: RBCS_MARK

Create mark from assembly

Use this option to create marks form all assemblies in the selection. If single elements are also selected, they are ignored while creating a mark.

To create a mark:

1. select assemblies
2. specify a mark name
3. define values of forces.

Menu: *Steel / Engineering drawings / Create mark from assembly* command
Command line: RBCS_ASMARK

Create mark from single part


Use this option to create marks with the same names, that are entirely included in the selection of elements. Marks created using this option consist of a single element set.

To create a mark:

1. select parts
2. specify a mark name
3. define values of forces.

Menu: *Steel / Engineering drawings / Create mark from single part* command
Command line:RBCS_SPMARK

Tube development


Use this option to create a drawing of a developed tube profile.
Menu: *Steel / Tube development* command

Styles

Dimensioning style

Opens the ***Dimensioning styles*** dialog box used for determining the styles of dimensioning for elements provided in a drawing.
Menu: *Steel / Styles / Dimensioning styles* command
Command line: RBCS_DIM

Description styles

Opens the ***Styles of descriptions*** dialog box which is used to determine styles of descriptions for elements provided in a drawing.
Menu: *Steel / Styles / Description styles* command
Command line: RBCS_DESCRIPTION

Tables

Bill of materials


The option allows the user to prepare tables of all structure elements.
Menu: *Steel / Tables / Bill of materials* command
Command line: RBCS_BILL

List of profiles


The option allows the user to prepare tables of profiles.
Menu: *Steel / Tables / List of profiles* command
Command line: RBCS_PROFILELIST

List of profiles (by section types)


Use this option to create tables of profiles by profile cross-sections.
Menu: *Steel / Tables / List of profiles by section types* command
Command line: RBCS_PROFILELISTSUM

List of plates


The option allows the user to prepare tables of plates.
Menu: *Steel / Tables / List of plates* command
Command line: RBCS_PLATELIST

List of user parts


The option allows the user to prepare tables of user parts.
Menu: *Steel / Tables / List of user parts* command
Command line: _RBCS_USEROBJLIST

List of assemblies


The option allows the user to prepare assembly tables.
Menu: *Steel / Tables / List of assemblies* command
Command line: RBCS_ALIST

Mounting list

The option allows the user to prepare mounting tables.
Menu: *Steel / Tables / Mounting list* command
Command line: RBCS_MONTLIST

Bolt list

The option allows the user to prepare connector tables.
Menu: *Steel / Tables / Bolt list* command
Command line: RBCS_BLIST

Update table

The option allows the user to update a table presented in a drawing.
Menu: *Steel / Tables / Update table* command
Command line: RBCS_ACTTABLE

Tables manager

Opens the ***Table printout manager*** dialog box; the option enables the user to define/modify appearance of the printout of tables used to prepare steel summary tables.
Menu: *Steel / Tables / Tables manager* command
Command line: RBCS_LISTPRINT

Styles

Opens the ***Steel summary tables - style manager*** dialog box; the option enables the user to define/modify tables used to prepare steel summary tables.
Menu: *Steel / Tables / Styles* command
Command line: RBCS_LISTTEMPL

Static analysis - *Autodesk Robot Structural Analysis*

Option allows export of a structure model to the calculation program ***Autodesk Robot Structural Analysis***.
Menu: *Steel / Static analysis - Robot* command
Command line: RBCS_R2R

DSTV data export

The option enables reading / saving a DSTV format file.
Menu: *Steel / DSTV data export* command
Command line: RBCS_DSTV

Save model as ACIS solids

The option enables saving a structure model to a DWG format file (ACIS format). You can select the full (complete description of solid details) or simplified save option. If the simplified save option is selected, profiles are saved without fillets and openings (except anchor openings), bolts, washers, nuts and anchors are ignored.
Menu: *Steel / Save model as ACIS solids* command
Command line: RBCS_EXPORTMODEL

Insert / Send model to Revit
This option allows the link between ***AutoCAD Structural Detailing - Steel*** and ***Autodesk Revit***.
Menu: *Steel / insert \ Send model to Revit*

Preferences

Opens the ***Options*** dialog box; the option enables the user to set parameters of the work environment for ***AutoCAD Structural Detailing***.
Menu: *Steel / Preferences* command
Command line: RBCS_PREF

Project preferences

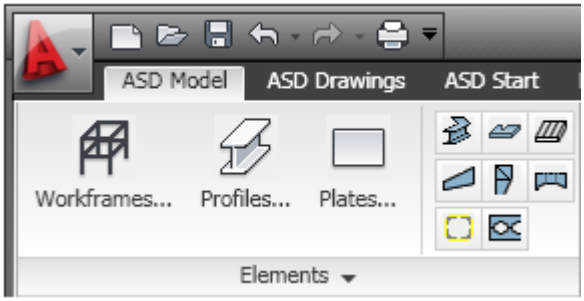
Opens the ***Project preferences*** dialog box; the option enables the user to adopt basic parameters of the current project in ***AutoCAD Structural Detailing***.
Menu: *Steel / Project preferences* command
Command line: RBCS_PRJPREF

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

Divide project 
The option enables dividing a project into several parts (distributed work on a document); it allows decreasing the size of a file with project data.
Menu: *Steel / Divide project* command
Command line: RBCT_DISTRIBUTE

1.4. Ribbon

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



The ribbon consists of several panels, grouped on tabs that are named by task or subject. The ribbon panels include many commands that have been on toolbars and in dialogs so far, such as icons, drop-down lists, sliders, text fields and other elements characteristic of a given tab. Using the ribbon, you do not have to display many toolbars; thus the application displays fewer functions and increases the allowable workspace placing the whole interface on a small area that can be anytime shown or hidden.

The ribbon displays automatically when a drawing is created or opened using the *2D Drafting* or *3D Modeling* workspace. You can display the ribbon manually using either of the following methods:

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

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

- click 

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

NOTE:
You can display the ribbon horizontally, vertically or as a floating palette.

Using the editor you can also switch between workspaces (such as the classic workspace without the ribbon). To do it, select the *Customize* tab > *Workspaces* and select *Set current* from the context menu.

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

1.5. Divide project (distributed work)

The option enables the user to divide a file with structure model into smaller files. The option is available from:

- menu by pressing the *Steel / Divide project* option
- toolbar by pressing the  icon
- command line: RBCT_DISTRIBUTE
- when running automatic drawings - by switching on the *Start distributed work* option.

The option creates automatically a subfolder, where in separate *.dwg files information about positions is saved. Thanks to this operation the main *.dwg file is smaller (e.g. loading such a file takes up much less time than loading one file with information about all structure elements).

 *NOTE:*
*If the user has already divided a *.dwg file (has started distributed work on the project), there is no possibility to return to one large *.dwg file containing information about all structure elements.*

To start distributed work, the user needs:

- a model with already defined positions
- to select the *Steel / Divide project* option or press the  icon
- to select positions to be saved to external files
- to press the **Enter** key.

Once a project is divided, successive positions / drawings added to a project are saved in separate files (the *Divide project* option works for new positions / drawings); the main project file will not be too large, since most information will be saved in separate *.dwg files.

 *NOTE:*
*If the user would like to copy a created project in which distributed work has begun, then apart from the main *.dwg file, he/she has to copy the subfolder with separate *.dwg files containing descriptions of individual positions / drawings.*

2. CONFIGURATION

2.1. Project preferences

2.1.1. Project preferences

The option allows the user to adopt basic parameters applied in ***AutoCAD Structural Detailing*** (these parameters are saved in a DWG file). The option is available from:

- the menu by selecting the *Steel* / *Project preferences* option
- the toolbar by pressing the  icon
- the command line: RBCS_PRJPREF.

Two main parts may be distinguished in the ***Project preferences*** dialog box:

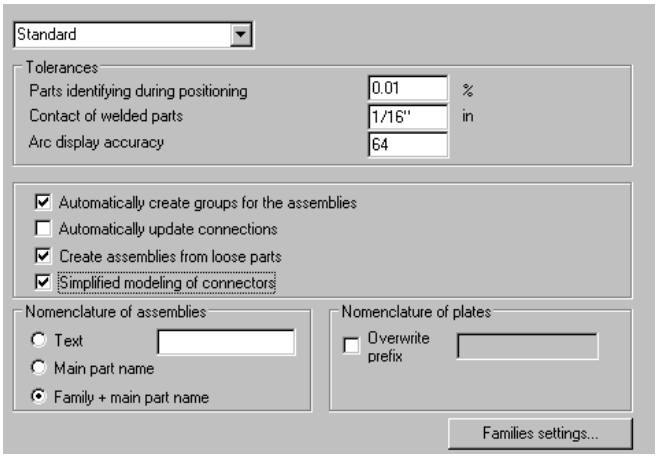
- the left part of the dialog box contains the selection tree (see the drawing below) from which the user, by means of the mouse, selects one of the project preference options

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

The right part of the dialog box includes standard buttons (**OK**, **Cancel**, **Help**) as well as the following buttons:
Default - if this button is pressed, values of the project preference parameters are saved as default values
Save - if this button is pressed, the current project preference parameters are saved under the name provided in the ***General*** dialog box
Delete - if this button is pressed, a set of project preferences saved under the name currently presented in the ***General*** dialog box, is deleted.

2.1.2. General

Once the *General* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are provided in the right part of the dialog box.

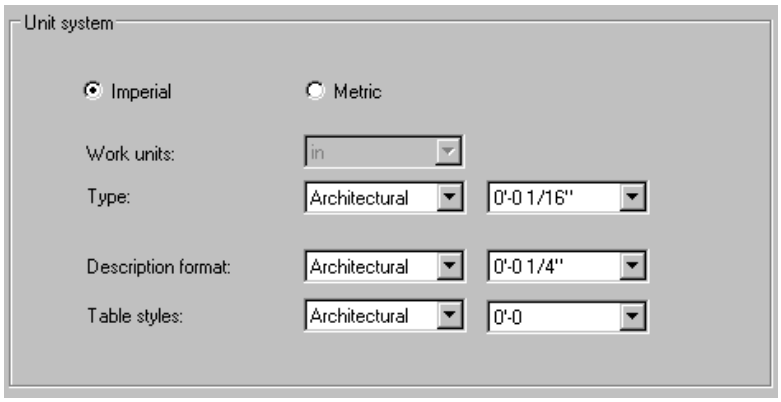


The following parameters may be determined in this dialog box:

- the *Tolerances* field enables defining the following quantities:
  Parts identifying during positioning - an admissible deviation in dimensions of two parts which are to be treated as identical during positioning
  Contact of welded parts - an admissible distance between edges of the parts during detection of a contact line (used while welding the parts)
  Arc display accuracy – a number of straight segments (with respect to a round angle - 360 degrees) into which a created arc is dividied
- *Automatically create groups for the assemblies* - if this option is turned on, then the elements joined by means of workshop connectors (bolts or welds) are grouped automatically while these connectors are being inserted
- *Automatically update connections* – if this option is turned on, then after modification of structure geometry the existing connections are adjusted to new structure dimensions; turning on the option is recommended for small structures (and fast computers); for a large structure model the operation of updating the structure may be very time consuming
- *Create assemblies from loose parts* - if this option is turned on, then all parts which do not belong to any assembly will be ascribed a new position type: assembly (during automatic generation of assemblies); if this option is turned off, then the position type of parts not belonging to any assembly will not be changed
- *Simplified modeling of connectors* – if selected, all connectors are displayed in a simplified way; it decreases computer memory use, particularly for large structures.
- *Nomenclature of assemblies* – enables selection of names of assemblies created automatically in the program:
  main part name – an assembly name will consist of a name of main element (e.g. HP14x102)
  family + main part name - an assembly name will consist of a family name and a main element name (e.g. Beam_HP14x102)
  text - an assembly name will consist of any text defined by the user
- *Nomenclature of plates*
  Overwrite prefix - if selected, a name defined in the field overwrites the default prefix for plates (the name created automatically when defining a plate)
- **Families settings** - pressing this button opens the ***Family manager*** dialog box.

2.1.3. Units

After selecting the *Units* option from the selection tree in the ***Project Preferences*** dialog, the following options display in the right part of this dialog.



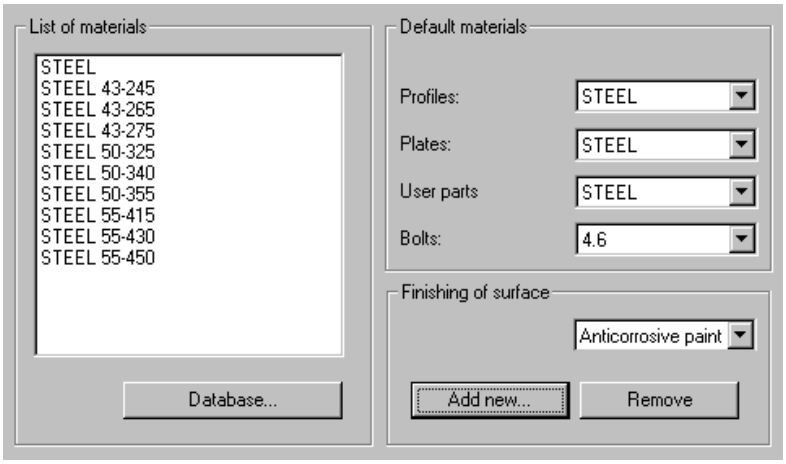
Use this tab to define ***AutoCAD Structural Detailing*** work units. You can select one of the following unit systems:

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

The unit system settings can be selected for the description format and table styles.
You can also change the displayed number format. Use the selection lists to select a number of decimal places for each quantity. To change a number of decimal places, select an appropriate symbol from the selection list.

2.1.4. Materials

Once the *Materials* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are provided in the right part of the dialog box.



The following parameters may be determined in this dialog box:

- the *List of materials* field contains materials available in the current project; the same material list is included on the lists accessible in the *Default materials* field (profiles, plates, user parts);
  Pressing the **Database** button opens the Material database dialog box where the list of materials may be defined; this dialog box allows the user to add materials from the selected database.
- in the *Default materials* field the selection lists of profiles, plates, user parts and bolts are provided; they allow selection of a material, which will be adopted by default in the dialog boxes for definition of profiles, plates, user parts or bolts.

NOTE:
*A default bolt grade chosen in the above dialog box is taken into account in the **AutoCAD Structural Detailing** macros (connections, typical structures) and while using the options in the case of which bolts are inserted automatically; if bolts are inserted 'manually' by the user, the most-recently-selected bolt grade is remembered in the dialog box.*

In the lower part of the dialog box is the *Finishing of surface* field; it allows defining and/or selecting types of finishing of steel element surfaces (hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat). A type of surface finishing is an additional attribute of structure objects (profiles, plates, assemblies).
The selection list contains all defined types of surface finishing (also the *None* type indicating a surface without the finishing – this type cannot be removed fom the list). This part of the dialog box includes, as well, the following two buttons:

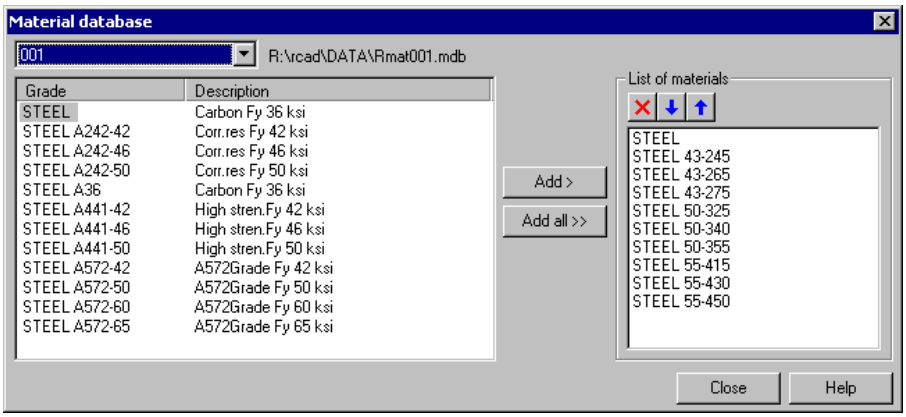
- **Add new** – opens the ***Add new*** dialog box for defining a name of a new type of surface finishing
- **Remove** – removes a surface finishing type chosen on the selection list.

NOTE:
A type of surface finishing is saved in the current project; it means that a list of types is saved in a DWG file.

NOTE:
An assembly adopts a surface finishing type from the type selected for the main part of that assembly.

2.1.5. Material database

The ***Material database*** dialog box shown in the drawing below opens after pressing the ***Database*** button located in the ***Project preferences*** dialog box (the Materials tab).



In the above dialog box the user may add materials from databases accessible in the program to the list of available materials. To add a material to the list of available materials, the user should follow the steps below:

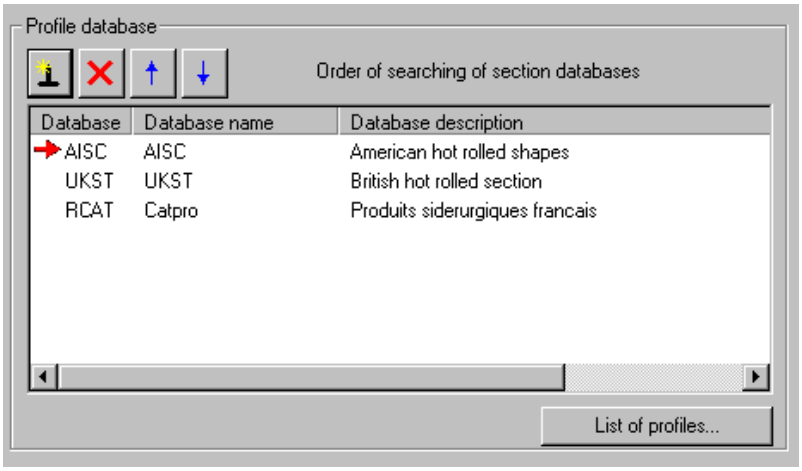
- choose a material database from the selection list provided in the top part of the dialog box, e.g. 001 (American material database)
- select a material type in the left panel
- press the **Add >** button.

Pressing the **Add all >>** button adds all the materials included in the left panel.

Pressing the  icon deletes a selected (highlighted) material from the list of available materials.

2.1.6. Profiles

Once the *Profiles* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are provided in the right part of the dialog box.



In this dialog box, the *Profile database* field presents a list of profile folders (databases) available in the current project. Pressing the **List of profiles** button opens the dialog box in which profiles applied in a given project are displayed. The following icons are provided in the top part of the dialog box:

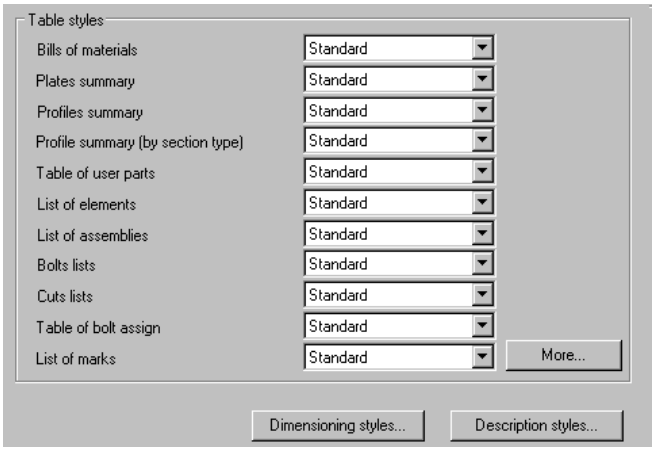
 – pressing this icon opens the dialog box for selection of a database to be added to the list of available profile databases

 – pressing this icon deletes a selected profile database

 – pressing these icons moves a database one position up or down on the list; the order is significant while searching through available profile databases.

2.1.7. Styles

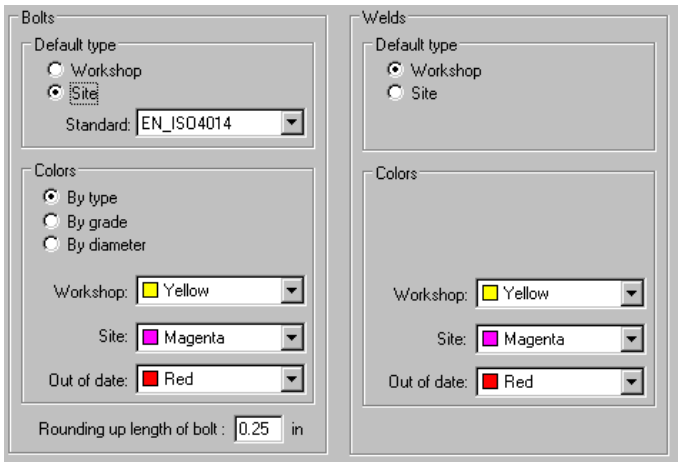
Once the *Styles* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are provided in the right part of the dialog box.



This dialog box enables definition of table styles for the table types available in the program (Bills of materials, Profile summary, Plate summary, etc.); the selection lists include all the styles defined in a project.
Pressing the **More…** button opens the Steel summary tables – style manager dialog box. In the bottom part of the dialog box the following two buttons are provided: **Dimensioning styles** and **Description styles**, which when pressed open the dialog boxes Dimensioning styles and Styles of descriptions, respectively.

2.1.8. Connectors

Once the *Connectors* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are displayed in the right part of the dialog box.



The following parameters of bolts and welds may be determined in this dialog box:

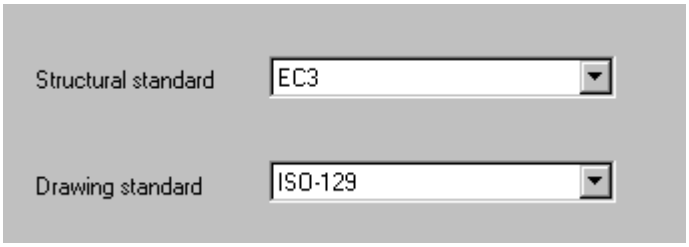
- in the *Default weld type* field the user may determine a default weld type (workshop weld or site weld) if weld is created automatically (e.g. connections)
- in the *Default bolt type* field the user may determine a default bolt type (workshop bolt or site bolt) if bolt is created automatically (e.g. connections); it is also possible to select a standard (database) of bolts used in the program
- in the *Colors* field the user may determine the color of the following types of connectors presented in drawings: site connectors, wokshop connectors or out-of-date connectors (these are no longer current connectors after changes have been made in a structure model e.g. after translating a bar a site weld is no longer needed in a given place).

Moreover, for bolts it is possible to:

- select a method of assigning colors to bolts: by bolt type, by bolt grade or by bolt diameter
- determine a rounding up length of bolts (in the program the bolt length is automatically calculated for connections of elements; a default value of the rounding up length is 5 mm; the user can modify the value of the rounding up length). NOTE: a value of this parameter is saved together with the project in a DWG file.

2.1.9. Standards

Once the *Standards* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the drawing below are displayed in the right part of the dialog box.

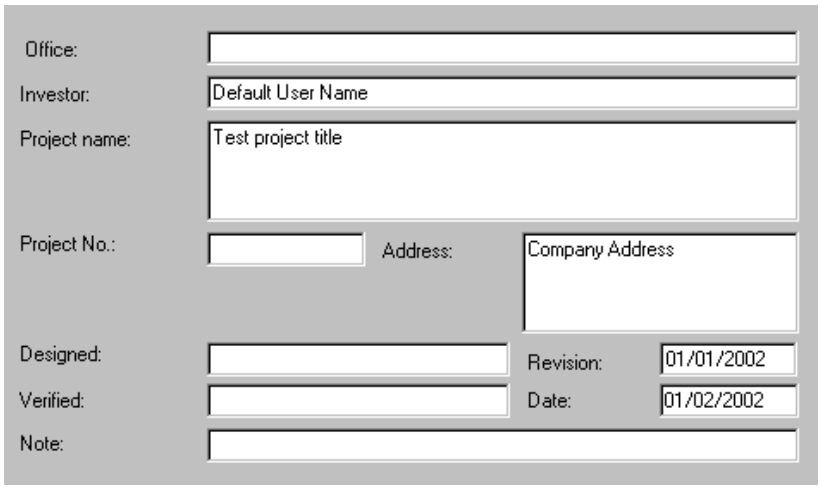


The above dialog box enables selection of standards used in ***AutoCAD Structural Detailing***: a standard of the steel structure design and a drawing standard (drawings of steel structure elements). A selected drawing standard imposes appropriate designations, hatching symbols, etc. valid in a country from which a given standard comes from. The following standards are available in the current program version:

- structural standard:
  - Eurocode 3
  - Polish code PN-90/B-03200
  - French code CM66
  - Russian code SNiP II-23-81
- drawing standard (corresponding to a structural standard):
  - ISO 129
  - Russian code GOST 21.101-97.

2.1.10. Project Info

Once the *Project Info* option is chosen from the selection tree in the left part of the Project preferences dialog box, the options shown in the figure below are provided in the right part of the dialog box.



The above dialog box contains several edit fields which allow the user to enter general information about the project. These texts correspond to the variables used in cells of tables included in drawings as well as in printout headers. The following variables are applied in the edit fields provided in the dialog box above:
Office - **VAR_OFFICE**
Investor - **VAR_INVESTOR**
Project name - **VAR_PROJ_NAME**
Project No. - **VAR_PROJ_NUM**
Address - **VAR_ADDRESS**
Designed - **VAR_DESIGNER**
Verified - **VAR_VERIF**
Revision - **VAR_REV**
Date - **VAR_DATE**.
Apart from that, the variables below are accessible:
Scale - **VAR_SCALE**
Page No. - **VAR_PAGENO**.

2.2. Preferences

2.2.1. Preferences

The option allows the user to adopt basic parameters used in ***AutoCAD Structural Detailing***. The option is available from:

- the menu by selecting the *Steel / Preferences* option
- the toolbar by pressing the  icon.

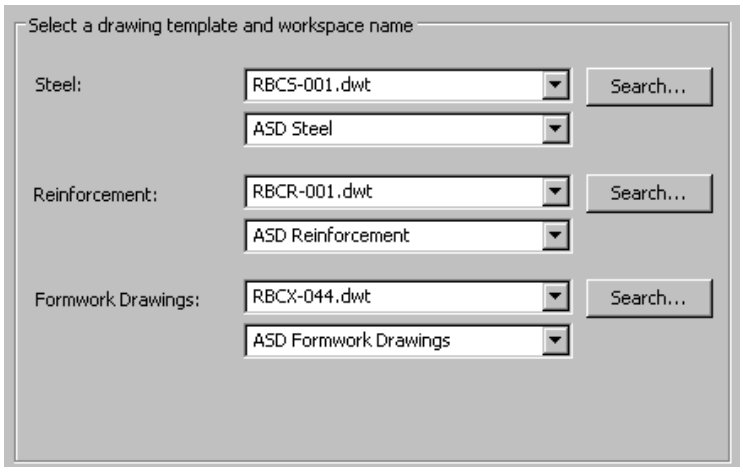
Two main parts may be distinguished on the *Steel* tab in the ***Options*** dialog box of the AutoCAD ® program:

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

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

2.2.2. General settings

After selecting the *General settings* option from the selection tree on the *Structural Detailing* tab (the AutoCAD ® ***Options*** dialog), in the right part of this dialog you can select a drawing template and a workspace name (such as ASD, ASD Classic) for ***AutoCAD Structural Detailing*** modules.



In ***AutoCAD Structural Detailing*** there are the following countries available for which the working language may be selected:

 Great Britain

 USA

 Italy

 Russia.

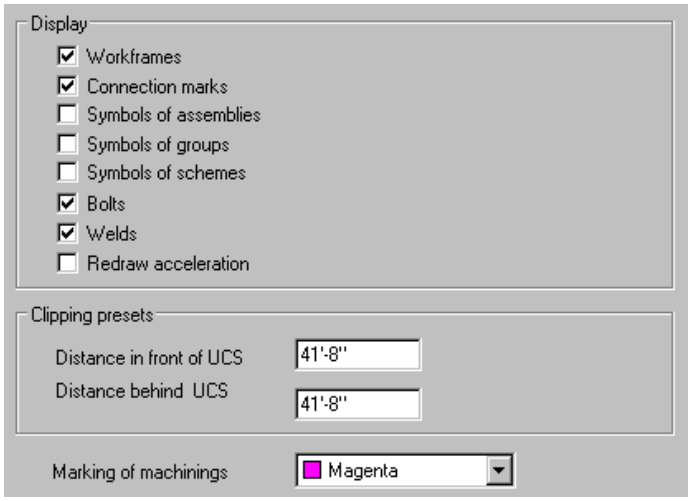
 Poland

 France

 Spain

2.2.3. Display settings

Once the *Display settings* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.



In this dialog box the user may switch on /off display of the following elements in the graphical editor:

- workframes
- matching and connection marks
- symbols of assemblies
- symbols of groups
- symbols of schemes
- bolts
- welds.

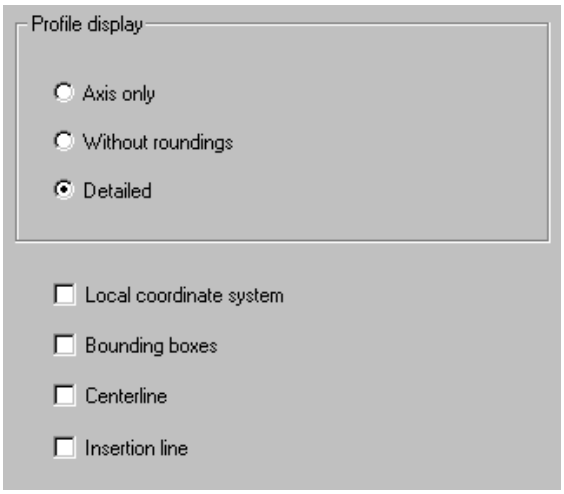
Below is the *Redraw acceleration* option; if this option is switched on, then while presenting a view of a whole structure on the screen, a part of structure elements will not be shown (e.g. connection details, connection symbols). It results in acceleration of model display on the screen. It is recommended to activate this option in the case of large structures.

In the middle part of the dialog box the user may determine values of distances between clipping planes and UCS.

In the lower part of the dialog box is the field for selecting a color of machinings made in elements of a structure model.

2.2.4. Profiles

Once the *Profiles* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.

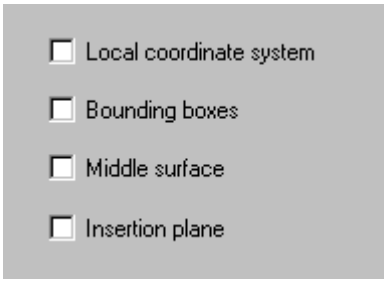


In this dialog box the user may determine detailed display parameters for bar profiles:

- the *Profile display* dialog box allows selection of the following options:
  axis only - if this option is selected, then a profile (bar) will be displayed as a line connecting the beginning point and end point of the profile
  without roundings - if this option is selected, then a simplified view of a profile (without roundings) will be displayed
  detailed - if this option is selected, then a profile (bar) will be presented with all the details
- *Local coordinate system* - if this option is selected, then the local coordinate system will be presented for all the profiles defined in the project
- *Bounding boxes* - if this option is selected, then bounding boxes will be presented for all the profiles defined in the project
- *Centerline* - if this option is selected, then the bar axis will be presented for all the profiles defined in the project (regardless of that which option has been chosen in the *Profile display* field)
- *Insertion line* - if this option is selected, then the line of profile insertion will be presented for all the profiles defined in the project; this option may be applied to present profile offsets.

2.2.5. Plates

Once the *Plates* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.

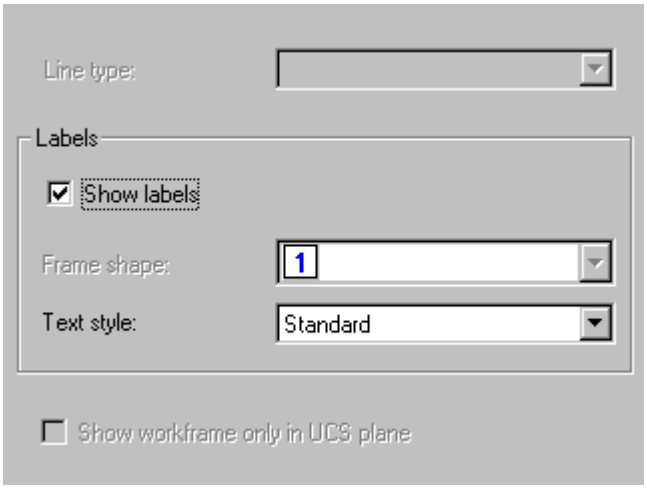


In this dialog box the user may determine detailed display parameters for the defined plates:

- *Local coordinate system* - if this option is selected, then the local coordinate system will be presented for all the plates defined in the project
- *Bounding boxes* - if this option is selected, then plate bounding boxes will be presented for all the plates defined in the project
- *Middle surface* - if this option is selected, then the middle surface will be presented for all the plates defined in the project
- *Insertion plane* - if this option is selected, then the insertion plane will be presented for all the plates defined in the project; this option may be applied in offset presentation.

2.2.6. Workframes

Once the *Workframes* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.

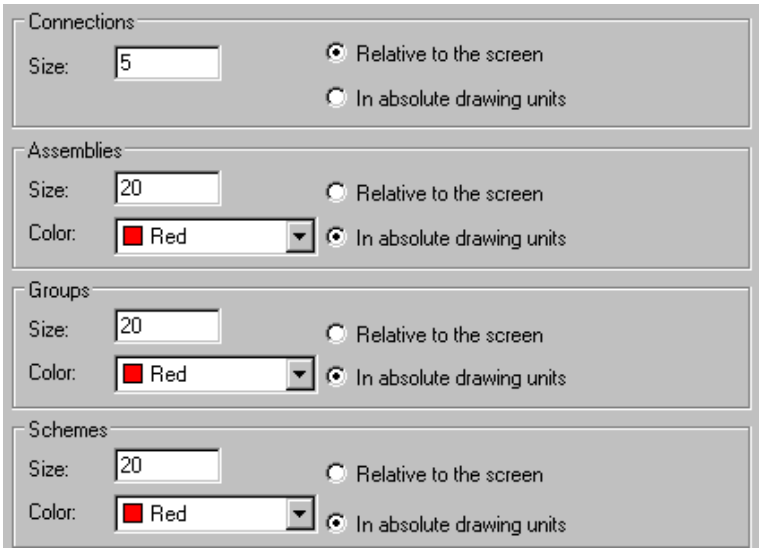


In this dialog box the user may determine detailed display parameters for the defined workframes:

- the *Line type* list contains all the line types available in the project of the AutoCAD ® program; from this list the user may select a line type applied to lines of the workframes presented in a structure model
- the *Labels* field allows selection of the following options:
  Show labels - the option allows switching on/off the description of workframe lines
  Frame shape - the option allows selecting the shape of descriptions of workframe lines; there are three possibilities available: round, rectangular or without frame
  Text style - it contains all the text styles available in the project of the AutoCAD ® program; from this list the user may select a text style applied to descriptions of lines belonging to the workframes presented in a structure model
- *Show workframe only in UCS plane* - if this option is switched on, then only these workframe lines which are positioned in the active local coordinate system are presented in the model.

2.2.7. Connections

Once the *Connections* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.

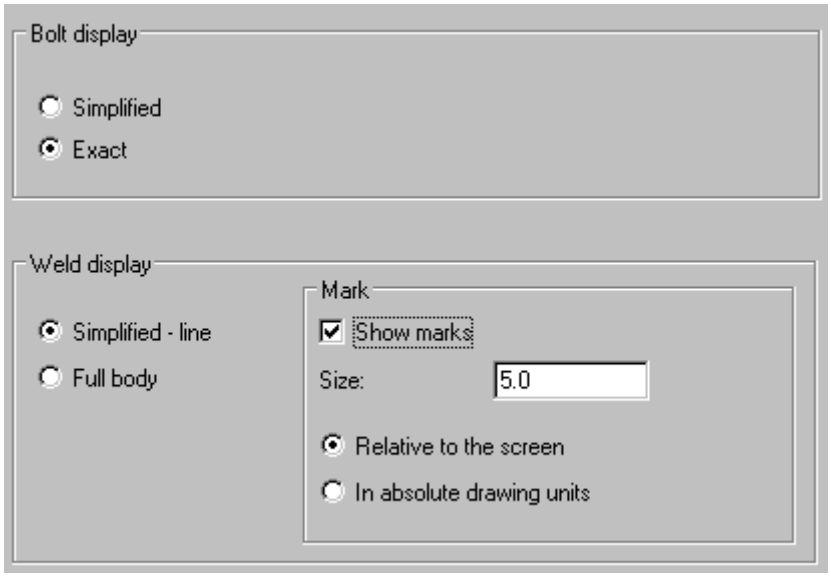


In this dialog box the user may determine detailed display parameters for labels of defined connections (spheres), assemblies and groups:

- the *Connections* field enables selection of the following options:
  Size - the edit field in which the user may specify the size of a connection symbol; a size value may be defined as follows:
  Relative to the screen - if this option is selected, then the size is expressed as percents
  In absolute drawing units - if this option is selected, then the size is expressed in units used in the AutoCAD ® program
- the *Assemblies* field enables selection of the following options:
  Color - a list for selecting a color of the assembly symbol
  Size - an edit field for defining a size of the assembly symbol; a size value may be specified in the following ways:
  Relative to the screen - if this option is selected, then the size is given in percent
  In absolute drawing units - if this option is selected, then the size is given in units used in AutoCAD ®.
- the *Groups* field enables selection of the following options:
  Color - a list for selecting a color of the group symbol
  Size - an edit field for defining a size of the group symbol; a size value may be specified in the following ways:
  Relative to the screen - if this option is selected, then the size is given in percent
  In absolute drawing units - if this option is selected, then the size is given in units used in AutoCAD ®.
- the *Schemes* field enables selection of the following options:
  Color - a list for selecting a color of the scheme symbol
  Size - an edit field for defining a size of the scheme symbol; a size value may be specified in the following ways:
  Relative to the screen - if this option is selected, then the size is given in percent
  In absolute drawing units - if this option is selected, then the size is given in units used in AutoCAD ®.

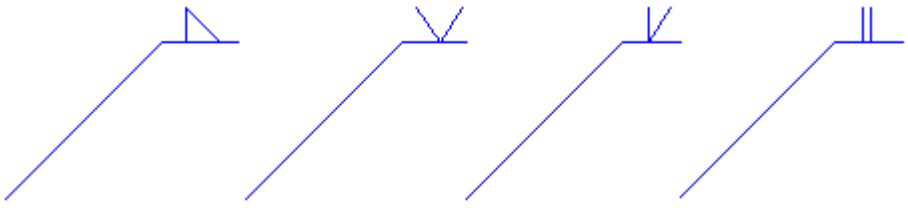
2.2.8. Bolts and welds

Once the *Bolts and welds* option is chosen from the selection tree in the left part of the Steel tab (the ***Options*** dialog box of the AutoCAD ® program), the options shown in the figure below are provided in the right part of the dialog box.



In this dialog box the user may determine detailed display parameters for the defined bolts and welds:

- the *Bolt display* field allows selection of the following options:
  Simplified - if this option is switched on, then bolts will be presented as a line fragment (segment)
  Exact - if this option is switched on, then bolts will be presented with all shape details
- the *Weld display* field allows selection of the following options:
  Simplified - line - if this option is turned on, then welds will be presented as a line fragment (segment)
  Full body - if this option is turned on, then welds will be presented with shape details displayed.
- in the *Mark* field the following options can be chosen:
  Show marks - the option is used to switch on/off display of a weld mark on the screen; a mark shape also determines a weld type (see the drawing below which presents marks of several weld types)
  An extension line on which a weld mark is positioned is always perpendicular to the weld



  Size - an edit field for defining a size of a weld mark; a size value may be specified in the following ways:
  Relative to the screen - if this option is selected, then the size is given in percent
  In absolute drawing units - if this option is selected, then the size is given in units used in AutoCAD ®.
  Indicating a weld mark results in selecting a weld in a drawing.

3. OBJECT INSPECTOR

3.1. Description of Object Inspector

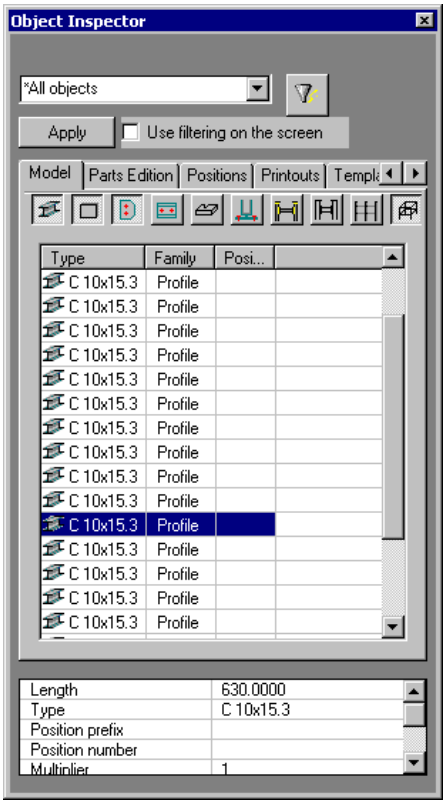
Inspector is a tool that allows managing elements (objects) included in a project created in ***AutoCAD Structural Detailing***. By standard, the Inspector dialog box is presented in left part of the program window, next to the field of graphical model definition.
The most important tasks carried out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

- presenting the project contents in the appropriate order (sequence)
- selecting elements that should be subjected to action of a selected command
- presenting and modifying project element properties (these may be single elements or whole objects)
- activating some of the commands applied to a particular selection (it depends on the inspector mode)
- defining, verifying and modifying groups
- verifying and modifying positions
- filtering model elements (objects)
- creating and managing drawing documentation of a project.

3.2. Object Inspector

The ***Object inspector*** dialog box is located to the left of the area of graphical model definition. Width of the dialog box may be freely adjusted so that as much space as possible is left for the area of graphical model definition.
The ***Object Inspector*** dialog box shown in the drawing below, can be divided into three parts:

- options that enable filtering model elements
- six tabs containing lists (set) of project elements depending on the stage of the design process (modeling / positions / printouts)
- table presenting properties of selected objects.



Options contained in the top part of the dialog box allow limiting the number of elements presented on the tabs describing model elements by means of filters. Filters also enable the user to decrease the number of model elements shown in the graphical editor viewer.
The selection list located in the top part of the dialog box contains the list of filters defined by the user in the current project. A filter may be defined in the Filter definition dialog box that can be opened once the  button is pressed.

The list always contains three default filters (they cannot be deleted):

- *all objects* - cancels any filter and displays all objects defined in the current project on the list
- *current selection* - an element list is limited to the elements that are currently selected in a model presented in the field of graphical model definition
- *hide current selection* – hides elements currently selected in a model presented in the field of graphical model definition (obviously, if there are no objects selected, then none of them can be hidden); the way this filter works is similar to that of the *Current selection* filter, however, the result the *Hide current selection* filter causes is an inversion of the result the *Current selection* filter brings.

Filtering is carried out after pressing the **Apply** button. All objects defined once filtering has been performed, will be displayed on the list (even if they do not satisfy a filtering condition) till the next filtering operation is made.
Once the ***Filter definition*** dialog box is closed, the list of filters defined is available in the ***Object Inspector*** dialog box.
If the *Use filtering on screen* option is switched on, the model graphical editor will present only these elements that are shown on the list in the Inspector dialog box.

The middle part of the dialog box includes the following tabs:

- *Model*
- *Parts edition*
- *Positions*

- *Printouts*
- *Templates*
- *Structural Detailing Center.*

This part of the dialog box allows presenting on the successive tabs, the current list of objects defined in the project in ***AutoCAD Structural Detailing - Steel***. 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 manner.

The bottom part of the dialog box contains the table with properties of elements selected on the list. The table operates in the identical manner as all the properties' dialog boxes in the AutoCAD ® program. Table characteristics include as follows:

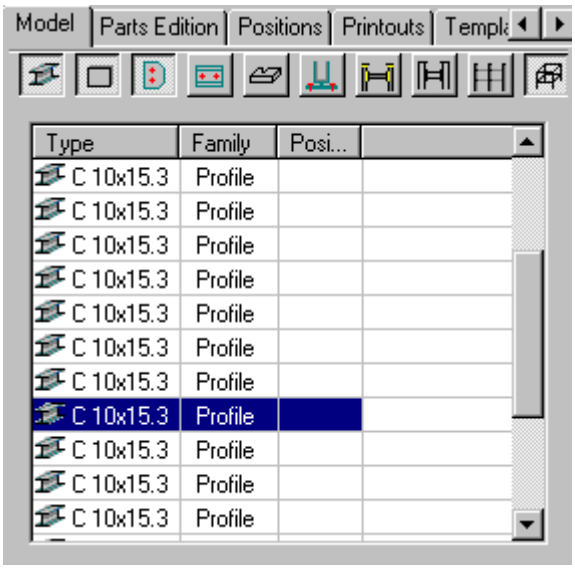
- if one of the elements is selected, then the table presents all properties of a selected element
- if several objects of the same type are selected, then the table presents all the fields with properties relevant to the selected element type, however, only the values that are common for all the elements are presented; the remaining fields are empty (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.

The first table column shows a name of a property (information), while the second one displays its current value. A table cell that can be changed may act like an option (edit field, selection list, button, etc.).

The table with object properties has been designed in such a manner so that it could present these object properties that are important from the engineering viewpoint. It usually shows only basic information (properties). Not all the pieces of information displayed in the table can be changed.

3.3. Model

This tab of the ***Object Inspector*** dialog box manages elements of a project model.



The following model objects defined in a structure may be presented on this tab:

- Profiles
- Plates
- Subparts

- User parts
- Connections
- Groups
- Workframes.

The enumerated object types may be presented on a list or may be excluded from presentation of objects; the following icons located in the top part of the dialog box serve this purpose:

 - Profiles

 - Plates

 - Subparts (plates)

 - Subparts (profiles)

 - User parts

 - Connections

 - Groups

 - Workframes.

 *NOTE:*
The listed types of objects are filtered while being presented/hidden; the currently selected filter is applied then.

Elements found on the list may be selected by indicating with the mouse cursor (the user may select 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 vice versa: if an element is selected in the graphic editor, then the selection is also presented in the Inspector dialog box).
An object list may be sorted; this can be done by applying one of the following two methods:

- by double-pressing the table column header (the list will be sorted then in ascending or descending order)
- by pressing the right mouse button and selecting an appropriate option in the context menu.

 *NOTE:*
Sorting is always carried out separately for each element type.

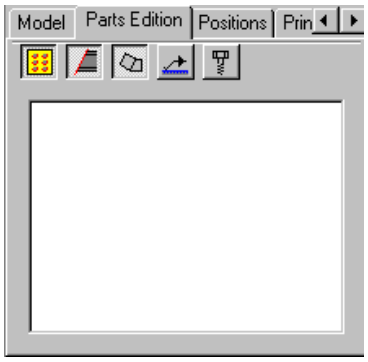
The context menu appearing once the right mouse button is pressed while the cursor is located on the Model tab, includes several options that allow performing operations on selected objects:

- *Select all* - choosing this command results in selecting all elements presented on the list
- *Deselect all* - selecting this command cancels selection of all elements on the list
- *Remove* - selecting this command results in removing selected elements from a project
- *Zoom selection* - selecting this command zooms a structure view in such a way so that the currently-selected elements are zoomed in
- *Modify properties* - selecting this command results in opening the properties dialog box (the dialog box contents depend on a selected object type); this command is available only when selected elements are of the same type
- *Group* - selecting this command results in creating a group that consists of selected elements; this command is available when several elements are selected
- *Add to group* - selecting this command enables adding a selected element to an existing group

- *Remove from group* - selecting this command enables removing an indicated element from a selected group
- *Change main part of group* - selecting this command enables changing the main part (main element) in a group; it is necessary to indicate an element that should be the main part of the group
- *Ungroup* - selecting this command results in splitting the group into single elements (objects); this command is available only when a group (or several groups) has been selected on the list
- *Assign position* - selecting this command allows assigning a position to selected elements; when this command is activated, the ***Manual Positioning*** dialog box opens
- *Remove position* - selecting this command results in removing positions from selected elements
- *Auto positioning* - selecting this command enables assigning positions to selected elements; after activating this command, the ***Automatic positioning*** dialog box opens on the screen.

3.4. Parts edition

This table presents edit operations performed on an element (e.g. cuttings, drills) as well as the contents of assemblies and groups. An element (or several elements) currently selected is presented in the form of a tree.



The list of filter selection and the button allowing filter definition are inaccessible on this tab. The *Current selection* filter is adopted by default.
Once the **Apply** button is pressed, selected objects are shown on the list. If a group is selected, then the group structure is presented on the screen. If a single element is selected, then edit operations carried out on this element are presented.
The manner of presentation can be changed by applying the following context menu options: *Show hierarchy of: Groups or Connections*.
The following object types can be presented in the tree:

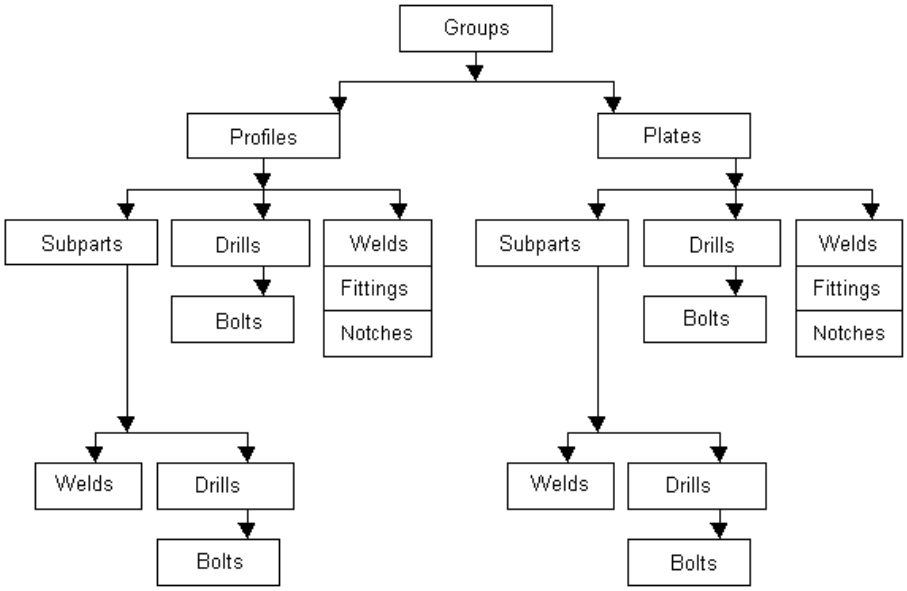
- Groups
- Connections
- Profiles
- Plates
- Subparts
- Drills
- Connectors (bolts, welds)
- Cuttings (fits)
- Notches.

The enumerated object types may be presented on a list or excluded from the presented objects; the following icons located in the top part of the dialog box serve this purpose:

 - drills
 - cuttings (fits)
 - bent profiles
 ,  - connectors (welds, bolts).
All the listed icons are switched on by default.

Elements found on the list may be selected by indicating with the mouse cursor (the user may select 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 vice versa: if an element is selected in the graphic editor, then the selection is also presented in the Inspector dialog box).

The hierarchy applied to present individual elements is illustrated in the figure below. There is a separate hierarchy for each connection.

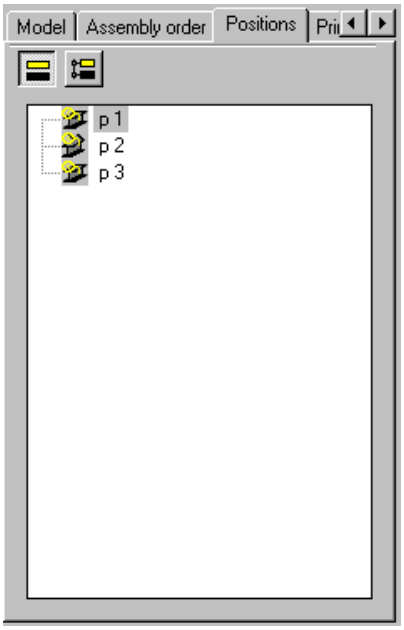


The context menu appearing once the right mouse button is pressed while the cursor is located on the *Parts Edition* tab, includes several options that allow performing operations on selected objects:

- *Remove* - selecting this command results in removing selected elements from a project
- *Zoom selection* - selecting this command zooms a structure view in such a way so that the currently-selected elements are zoomed in
- *Modify properties* - selecting this command results in opening the properties dialog box (the dialog box contents depend on a selected object type); this command is available only when one element is selected on the list
- *Assign position* - selecting this command allows assigning positions to selected elements; activating this command opens the ***Manual Positioning*** dialog box
- *Remove position* - selecting this command results in removing positions from selected elements.

3.5. Positions

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



The position types shown below may be presented on the list or may be excluded from presentation; the following icons, located in the upper part of the dialog box, are used for this purpose:

 - Profiles

 - Plates

 - Assemblies

 - Groups

 – User parts

 - Schemes.

Below are three selection options (*Parts*, *Families*, *Groups*) that make it possible to activate a mode of presentation (sorting) of positions.

The *Parts* option selected

In this mode elements are shown in one level, grouped by position types in the order as follows (the first order criterion for individual parts is a prefix, the next one is a number):

1. Profiles in the alphabetical order of position designations
2. Plates in the alphabetical order of position designations
3. Assemblies in the alphabetical order of position designations
4. Groups in the alphabetical order of position designations
5. User parts in the alphabetical order of position designations
6. Schemes in the alphabetical order

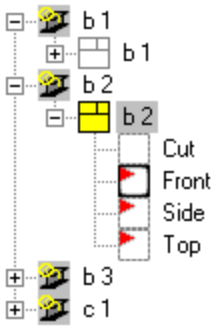
The *Families* option selected

In this mode elements are presented in the form of a tree with the following levels:

1. Position type; the order of the items: assemblies, profiles, plates, user parts
2. Family (in the alphabetical order)
3. Within a family positions are presented in the alphabetical order of designations

The *Groups* option selected
This mode offers multi-level presentation of elements; it is reflected in this mode which parts are assigned to which assemblies and groups. Positions are presented in the form of a tree with the following levels:
1. Position types; the order of the items: groups, assemblies, profiles, plates, user parts, schemes
For the types Groups and Assemblies successive sub-levels are created:
for groups: elements included in a given group
for assemblies: single parts included in a given assembly.

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



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

- in yellow - it means that this document is active on the edition layout
- as a red slash - it means that the element of a structure model for which the document has been prepared, has changed
- in gray - it means that the document has been exported to a separate dwg file.

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

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

- *Delete* - selecting this command results in removing selected positions from a project
- *Select positions* - choosing this command enables selecting all positions
- *Attach document* - choosing this command enables selection of a printout template; the template is saved on the *Templates* tab
- *Update documents* - selecting this command allows update of documents available for a position
- *Delete documents* - selecting this command enables deleting documents available for a position
- *Automatic drawings* - selecting this command enables automatic generation of drawings for a selected position; after running this command, the ***Automatic drawing generation*** dialog box opens on the screen.

3.6. Printouts

This tab enables management of printouts in ***AutoCAD Structural Detailing***; it presents the list of all printouts defined in the ***AutoCAD Structural Detailing*** project.
Printouts are presented together with a set of views. The printout list contains all the printouts, even those which do not include any views. The structure of user-defined printouts and views is shown in a form of a tree. Due to logical reasons, views are placed in a printout, however, for the user's convenience, the tree also includes an intermediate level, so that it is obvious to which document given views belong. If the printout layout is active, then the icon of a printout corresponding to the active printout layout is presented in red color.
Printouts provided on the list may be selected by indicating them with the mouse cursor (take note that only elements of one printout may be selected at a time - it is impossible to select elements of two different printouts). Selection of the printout in the dialog box is synchronized with the graphic editor - an appropriate drawing is displayed on the screen.
The top part of the dialog box contains two icons that enable presentation of the existing positions in the tree in the following modes:
-  model mode
-  drawing mode.

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

- *Delete* - selecting this command results in removing selected printouts from a project
- *Change name* - choosing this command enables changing the name of a highlighted printout (the name is entered to the command line)
- *Add printout* - selecting this command adds an empty printout to the project (the name is entered to the command line)
- *Activate* - selecting this command activates a selected printout
- *Unload printout* – after selecting this command, a selected printout is no longer active
- *Save printout* - selecting this command saves a selected printout in a *.DWG format file
- *Update* - selecting this command updates a selected printout after making modifications in positions
- *Update all printouts* - selecting this command updates all printouts after making modifications in positions.

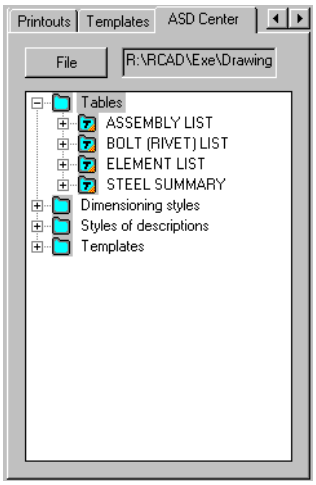
3.7. Templates

This tab enables management of printout templates in ***AutoCAD Structural Detailing***; it presents the list of all printout templates defined in the ***AutoCAD Structural Detailing*** project; the template structure is presented in a form of a tree. The icon of an active template (in the template layout) is presented in yellow.
The context menu, that appears after pressing the right mouse button at the moment when the cursor is located on the *Templates* tab, contains several options which allow performing operations on selected templates:

- *Delete* - selecting this command results in removing selected templates from a project
- *Change name* - choosing this command enables changing the name of a highlighted template (the name is entered to the command line)
- *Activate* - choosing this command causes a highlighted template to become active
- *Add view* - selecting this command adds a view to an existing template
- *Change style* - choosing this command enables changing a style in a selected template
- *Add document template* - selecting this command adds the template to a selected view.

3.8. Structural Detailing Center

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



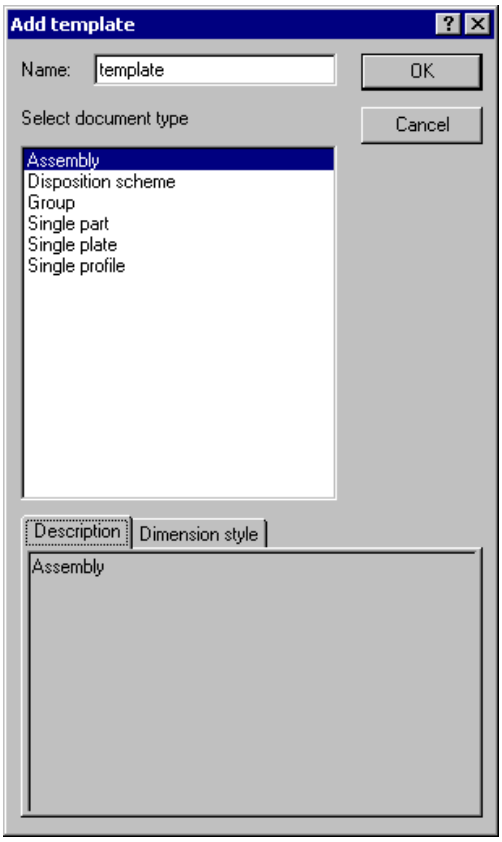
After pressing the **File** button and selecting a file with an earlier-saved project, the Inspector dialog box shows all the styles defined in the selected project, which may be used in the current project.
After highlighting a selected style, pressing the right mouse button and choosing the *Copy*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

3.9. Add template

The dialog box is used to define a printout template for drawings of positions defined in the program. The option is available from:

- *Steel / Drawing template designer* - the text menu
- the  icon - the toolbar
- the ***Object inspector*** dialog box after pressing the right mouse button and selecting the *Add document template* option on the *Templates* tab.

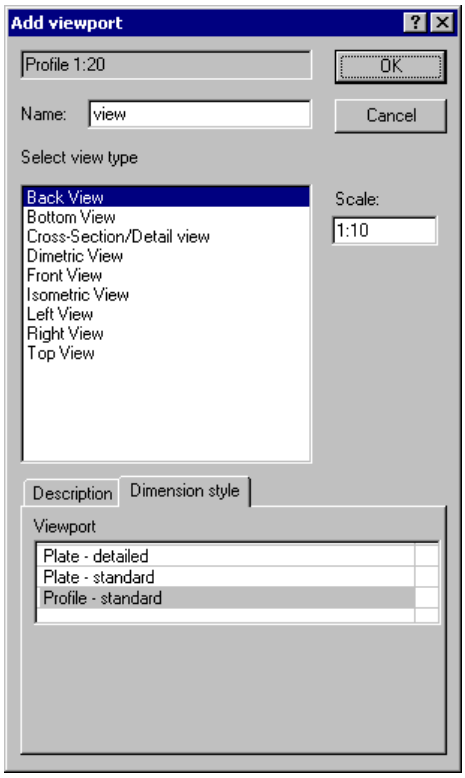
The dialog box presented below appears on screen.



The dialog box above allows defining a template for created drawings. In the steel part the user should select one of the allowable document types concerned with the position type: assembly, disposition scheme, group, single part, single plate or single profile.
After providing the name of a template and pressing the **OK** button, the template is added to the list of available templates located on the *Templates* tab.
In the bottom part of the dialog box the following two tabs are included: *Description* (this tab presents description of a selected position type) and *Dimension style*. The latter tab shows available dimension styles for a selected position type.

3.10. Add view

Once the template name and type are determined, the user may define views included in a given template. The ***Add viewport*** dialog box is available after pressing the right mouse button and choosing the *Add view* option on the *Templates* tab in the ***Object inspector*** dialog box.
The dialog box presented below appears on screen.



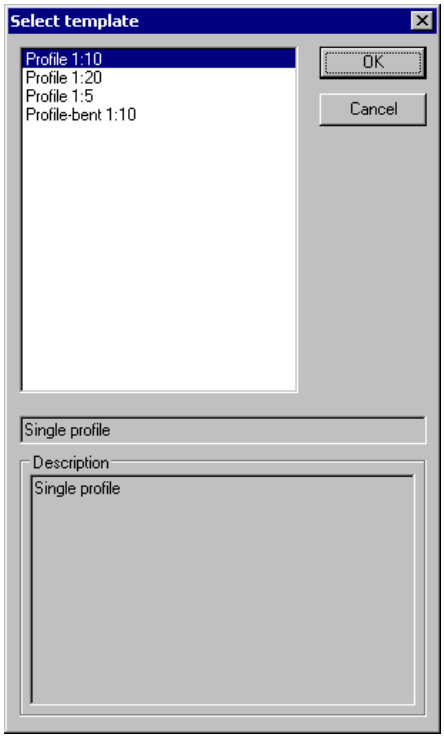
The following parameters may be determined in the above dialog box:

- name of a created view (in the top part of the dialog box a name of the currently-applied template is specified)
- the user may select view (projection) of a structure element
- the user may specify the scale for a selected view.

After pressing the **OK** button, within the template layout the placement of a view in a template (not yet in a final drawing) should be indicated.
The steps described above should be repeated in case of successive views placed in a template.

3.11.Select template

Once a position is defined, the user may select a drawing template for it. The ***Select template*** dialog box is available after pressing the right mouse button and choosing the *Attach document* option on the *Positions* tab in the ***Object inspector*** dialog box.
The dialog box presented below appears on screen.



In the above dialog box, from the list of available (defined) templates the user should choose a template to be applied for generating a drawing. Once the **OK** button is pressed, the template is assigned to a selected position.
After selecting the *Add to current Printout* option (context menu on the *Positions* tab), a created document will be placed in a final drawing (on the Layout tab).

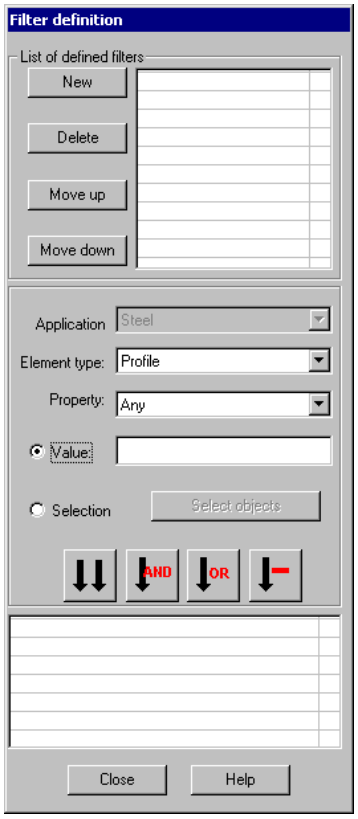
3.12. Templates/views - steps to be followed while creating a printout

In order to make a final drawing in the program, the user should follow the steps listed below (in the case of steel structure elements):

1. define a template:
   - ⇒ specify the name and select the type of structure element (single part…) - the ***Add template*** dialog box
   - ⇒ determine views (projections) that are to be included in a template
2. define positions for structure elements - the ***Positioning*** (manual or automatic) dialog box
3. assign template to a selected position - the ***Select template*** dialog box
4. place documents in a final drawing (printout).

3.13. Definition of filters

Filters may be defined in the ***Filter definition*** dialog box presented in the drawing below; the dialog box is opened after pressing the  icon placed in the Object inspector dialog box.



Filter definition may be based on types of model elements and their properties. A filter defined is identified by means of a name presented on the filter selection list located in the Object Inspector dialog box.

The ***Filter definition*** dialog box allows defining a new filter or modifying a filter that already exists (except for the two default filters). User-defined filters may be deleted from the list after pressing the Delete button.

The top part of the dialog box contains the list of all the filters defined by the user in a project. A selected (highlighted) filter is an active filter (i.e. a filter selected when the dialog box was being opened).

The bottom part of the dialog box includes the field which presents a defined expression determining the filter selected. The steps taken while defining and modifying a filter are similar; therefore, below a description of a new filter definition is presented.

To start definition of a new filter, the user should press the **New** button provided in the top part of the dialog box. Then the ***New filter name*** dialog box is displayed in the screen; the user should enter a name of a filter being defined into the edit field located in this dialog box. If the *Inherit definition* option is on, then a new filter will inherit a definition from the recently-selected filter (which means that the expression of the recently-selected filter will be entered on the list located in the bottom part of the ***Filter definition*** dialog box); if the *Inherit definition* option is off, then a new filter will be created without any definition of expression (the list located in the bottom part of the ***Filter definition*** dialog box will be empty). After closing the ***New filter name*** dialog box, a filter defined is presented on the list contained in the ***Filter definition*** dialog box.

After providing a filter name, the user may determine the expression defining a filter.

From the *Type* drop-down list the user may select a type of element that is to be included in the filter defined. The contents of the *Properties* list (see also List of available elements, properties and values) changes depending on the element type selected.
The user may select a filtering criterion from the *Properties* list; the criteria available depend on the selected element type.
Depending on the selected type of property, the user may determine the list of selected property values in the *Value* edit field. This field allows entering values that appear in the expression defining a filter.
If the *Properties* and *Value* field are not filled in (they are empty), it means that all the elements of the selected type (all values) will be included in the filter defined.
For some of the properties (e.g. length), the user may define a range of values in the *Value* field; the following syntax can be applied then:

- single values separated with a semicolon e.g. 1;2;3;10; - only the elements listed will be included in a filter
- range of values defined as follows: 1-100;
- values greater than and less than: -100; 10-; (respectively).

Apart from definition of values, it is also possible to define graphically the elements that are to be included in a filter. It can be done once the *Selection* option is chosen. Then the **Select objects** button becomes accessible; pressing this button results in closing the dialog box, whereas the cursor is in the selection mode. Once structure elements are selected and the right mouse button (or the ENTER key) is pressed, the dialog box appears again with the elements selected entered into the field for defining an expression that determines a filter.
For an expression or selection to be included in a filter, it is necessary to use one of the buttons presented below:

-  - pressing this button results in replacing all the expressions defined hitherto by the expression currently determined
- 

- pressing this button results in adding the currently-determined expression to expressions defined in a filter earlier (the AND operator is added in front of the currently-determined expression)
- 

- pressing this button results in adding the currently-determined expression to expressions defined in a filter earlier (the OR operator is added in front of the currently-determined expression)
-  - pressing this button results in adding the currently-determined expression to expressions defined in a filter earlier (the AND NOT operator is added in front of the currently-determined expression).

All expressions added to a defined/modified filter are presented in the bottom part of the dialog box with appropriate operators. Each expression is entered into a separate line; lines are separated with logical operators which refer to expressions below.
The user cannot modify the contents of a single expression in a filter; however, it is possible to delete lines from definitions of expressions in a filter. That can be carried out by means of two options located in the context menu that is activated after pressing the right mouse button while the mouse cursor is placed within the bottom part of the ***Filter definition*** dialog box.

- *Delete line* - once this command is selected, the program deletes a highlighted line together with the operator located before the line being deleted
- *Delete all* - once this command is selected, the program deletes a whole filter definition.

Each of the filters defined can be modified at any moment while working in a project. The list of filters defined can be seen in the top part of the ***Filter definition*** dialog box. A filter defined in the project can be deleted from the list of available filters by highlighting a line with a filter name and pressing the **Delete** button. The order of filters defined in a project can be changed on the filter list by applying the buttons: **Move up** or **Move down**. If they are pressed, a highlighted filter is moved one position up or down on the list of defined filters.

3.14. How to define a filter

What should be done to define a filter that allows selection of all columns made of RSJ 102 x 102 x 23.07 and RSJ 76 x 76 x 12.65 sections?

- press the **New** button; once the ***New filter name*** dialog box appears, enter the name of a defined filter
- close the ***New filter name*** dialog box (the *Inherit definition* option is switched off); a new filter is displayed in the ***Filter definition*** dialog box
- select *Section* as *Type, Section type* as *Properties*; on the *Values* list select lines with sections RSJ 76 x 76 x 12.65 and RSJ 102 x 102 x 23.07 (the lines are highlighted)
- add an expression by pressing the ↓↓ button; there appears the following expression: Sections (type) = RSJ 76 x 76 x 12.65; RSJ 102 x 102 x 23.07
- select *Section* as *Type; Family* as *Properties*; on the Values list select the *Column* family
- add an expression by pressing the ↓AND button; there appears the following expression: Sections (type) = RSJ 76x76x12.65; RSJ 102 x 102 x 23.07 and Sections (family) = Column.

The operation above completes the filter definition.

3.15. Objects and their properties

The Object Inspector may include:

Model elements (physical elements)

- Profiles
- Plates (single plates)
- User parts
- Subparts (plates, profiles, ribs, shear plates, etc.)
- Welds
- Bolts
- Connections
- Groups
- Assemblies – a set of parts (profiles or plates) grouped automatically in the program based on the connectors with the 'workshop' attribute existing between them; in practice, these are elements sent as a whole from a workshop to a construction site
- Workframes.

Model elements (nonphysical elements)

- Drills
- Openings
- Cuttings/fits
- Notches.

Drawing elements

- Positions
- Documents (sets of views)
- Views
- Printouts.

Below, the table of available elements, properties and values is presented.

Objects	**Properties**	**Values**
	All properties for objects	All the values appropriate for object properties
Profiles		
	Profile type	All the profiles available in the current project
	Family	All the families intended for the "profiles" type available in the current project
	Material	All the materials set in preferences
	Length	Any real positive value
	Position prefix	All the position prefixes used in the current project
	Position number	Any integral positive number
	Subpart	Yes / No depending on that whether a selected profile is a subpart
Plates		
	Plate thickness	All the thicknesses used hitherto in a project or any real positive value
	Family	All the families available in the current project intended for the "plate" type
	Material	All the materials set in preferences
	Type	Rectangular, circular, user-defined
	Position prefix	All the position prefixes used in the current project
	Position number	Any integral positive number
	Subpart	Yes / No depending on that whether a selected plate is a subpart
Subparts		
		Properties as for a profile or plate depending on the subpart type
Connections		
	Connection type	All the available connection types: (e.g. endplate)
Groups	Name	Any text
	Position prefix	All the position prefixes used in the current project
	Position number	Any integral positive number
	Subpart	Yes / No depending on that whether a selected profile is a subpart
Workframes	Workframe name	All the workframe names available in the current project
	Workframe type	box, wedge, prism
Positions	Prefix	All the position prefixes used in a project
	Number	Position numbers defined in the *edit field*
	Level of positioning	Single part, assembly, group
Printouts	Number	Numbers defined in the *edit field*
	Name	All the drawing names available in the current project
	Format	All the formats used in the current project
	Positions	All the positions defined in the current project

4. TYPICAL STRUCTURES

4.1. Typical structures

The option allows definition of typical (parametric) structures. The option is available from:

- menu by selecting one of the options in the submenu *Steel / Parametric structures*
- pressing one of the icons in the *Parametric structures* toolbar.

The following typical structures are available in the current version of ***AutoCAD Structural Detailing***:

- Multi-bay frame
- Roof truss
- Stairs
- Stairs spiral
- Staircase
- Bracing
- Ladder
- Railing
- Cage
- Brace
- Grate spreading
- Purlin spreading
- Compound profile
- Castellated beam
- Column (plate girder)
- Beam (plate girder)
- Multisegment beam (plate girder).

Parametric structures are available from the menu (*Steel / Parametric structures)* and the *Parametric structures* toolbar.

* Multi-bay frame - 
* Roof truss - 
* Stairs - 
* Stairs spiral - 
* Staircase - 
* Bracing - 
* Ladder - 
* Railing - 
* Cage - 
* Brace - 
* Compound profile - 
* Grate spreading - 
* Purlin spreading -  .

Plate girders are avialble from the menu (*Steel / Plate girders*) and from the *Plate girder* toolbar.

* Column (plate girder) 
* Beam (plate girder) 
* Multisegment beam (plate girder) 
* Castellated beam  .

After selecting a structure an additional dialog box is displayed on the screen in which a user may determine parameters of a selected steel structure. The shape of this dialog box depends on a selected structure type or structure element.

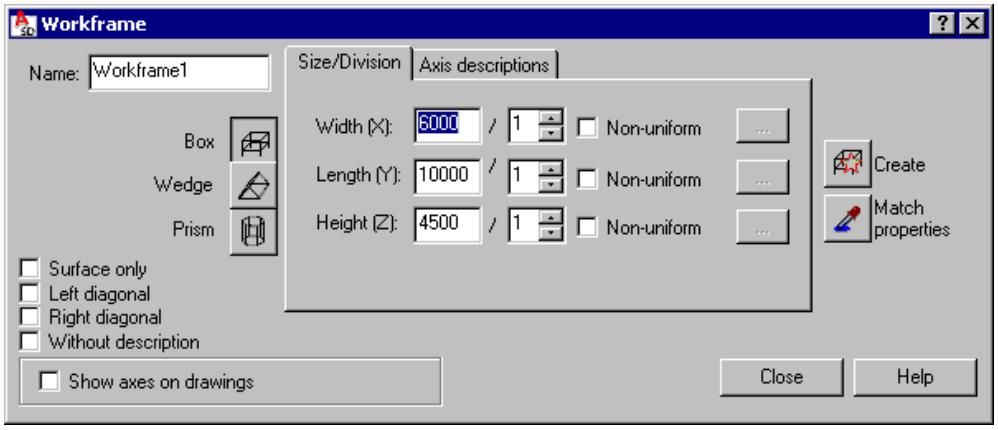
5. WORKFRAME

5.1. Workframe

The option allows determining a workframe that facilitates definition of a structure model in 3D space. The option is available from:

- the menu by selecting the *Steel / Workframes / Create workframe* option
- from the toolbar by pressing the icon
- the command line: RBCS_WF.

Once the *Workframes* option is selected, the dialog box shown in the drawing below, is displayed on the screen.



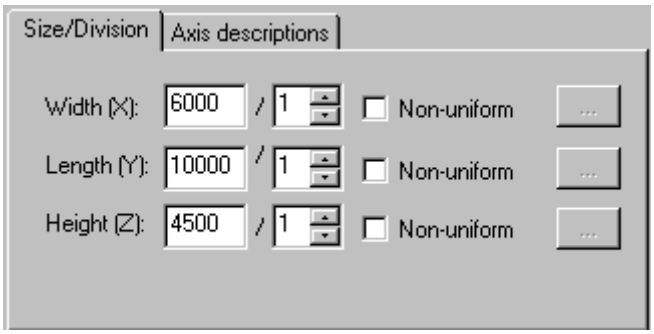
The ***Workframe*** dialog box can be divided into three parts:
in the left part of the dialog box the user may define basic information concerning a workframe:
workframe name
workframe type:
box
wedge
prism

- the bottom part of the dialog box contains the following options:
  Surface only - if this option is switched on, only external lines of a workframe are created; if this option is switched off, then all workframe lines are created
  Left diagonal - if this option is switched on, additional lines are also created, these are the diagonals of a quadrangle area inclined to the left
  Right diagonal - if this option is switched on, additional lines are also created, these are the diagonals of a quadrangle area inclined to the right
  Without description - if this option is switched on, descriptions of axes are not provided; if this option is switched off, then workframe axes will be described
  Show axes in drawings - if this option is switched on, then axes of a defined workframe are presented in drawings; if this option is switched off, then workframe axes play only auxiliary role in modeling and are not shown in any drawing
- the middle part of the dialog box includes two tabs: *Size/Division* and *Axis descriptions*; their contents depend on a selected type of a workframe
- the right part of the dialog box contains two icons:
  - creates a workframe based on parameters determined in the dialog box
  - is used to adopt (inherit) parameters from a workframe that is already defined.

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

5.2. Box

Once box is selected as a workframe type, the Workframe dialog box contains two tabs: *Size/Division* and *Axis descriptions*.
The *Size/Division* tab is shown in the figure below.

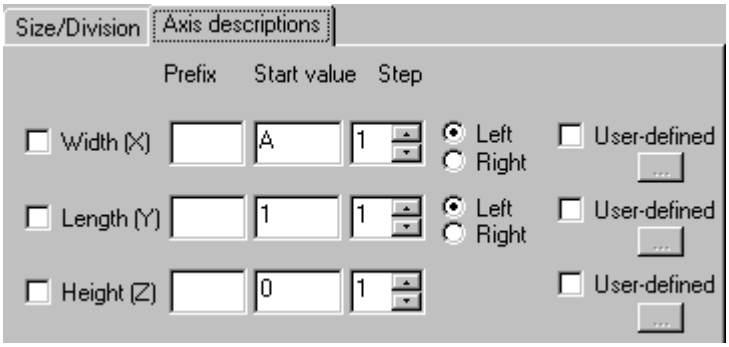


The following options are provided in this dialog box:

- *Length* (Y) - fields where length of a workframe and number of divisions along the workframe length are determined
- *Width* (X) - fields where width of a workframe and number of divisions along the workframe width are determined
- *Height* (Z) - fields where height of a workframe and number of divisions along the workframe height are determined.

If the *Non-uniform* option next to the options listed, is switched off, then a workframe will be generated automatically based on the parameter values defined in the edit fields discussed. If the *Non-uniform* option is switched on, then the edit fields become inaccessible; instead, the (**...**) button becomes available which - when pressed, opens the Lines distribution dialog box where position of successive workframe axes may be determined.

The *Axis descriptions* tab is shown in the figure below.



The following options are provided in this dialog box:

- *Length* (Y) - fields where an axis description along the workframe length is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated
- *Width* (X) - fields where an axis description along the workframe width is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated
- *Height* (Z) - fields where an axis description along the workframe height is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated.

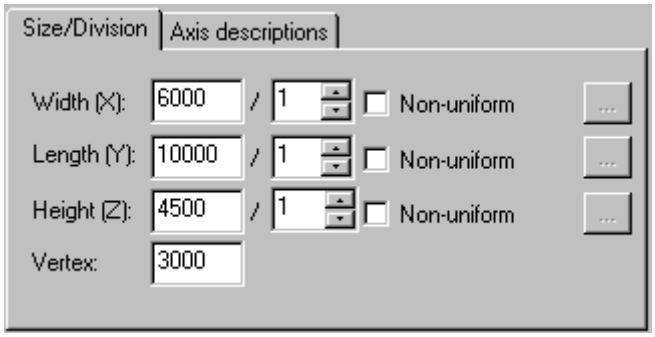
For each of the options listed three fields are accessible:

- *Prefix* - field where a prefix of workframe axes can be determined for the automatic description of workframe axes
- *Start value* - field where a start value for workframe axes can be determined for the automatic description of workframe axes
- *Step* - field where a value of increment for successive workframe axes can be determined for the automatic description of workframe axes.

For the width (X) and the length (Y) it is possible to choose the location of a description: on the right or on the left side of a workframe line.
If the *User-defined* option next to the options listed, is switched off, then a workframe description will be generated automatically based on the parameter values defined in the edit fields discussed. If the *User-defined* option is switched on for any of the options, then the edit fields for that option become inaccessible; instead, the (**...**) button becomes available which - when pressed, opens the Lines description dialog box where the description of successive workframe axes may be determined.

5.3. Wedge

Once wedge is selected as a workframe type, the Workframe dialog box contains two tabs: *Size/Division* and *Axis descriptions*.
The *Size/Division* tab is shown in the figure below.

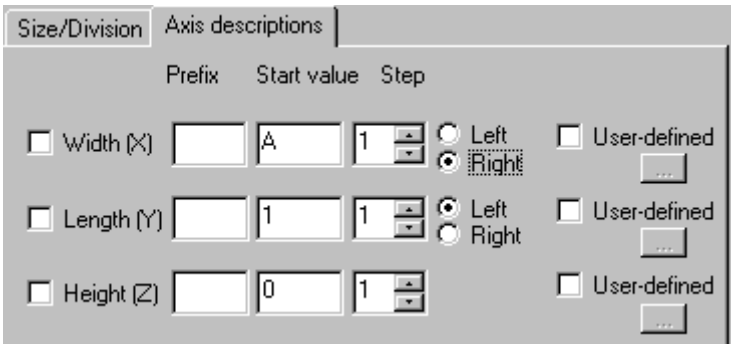


The following options are provided in this dialog box:

- *Length* (Y) - fields where length of a workframe and number of divisions along the workframe length are determined
- *Width* (X) - fields where width of a workframe and number of divisions along the workframe width are determined
- *Height* (Z) - fields where height of a workframe and number of divisions along the workframe height are determined.
- *Vertex* – a field which determines a coordinate of the vertex position for a wedge-shaped workframe.

If the *Non-uniform* option next to the options listed, is switched off, then a workframe will be generated automatically based on the parameter values defined in the edit fields discussed. If the *Non-uniform* option is switched on, then the edit fields become inaccessible; instead, the (**...**) button becomes available which - when pressed, opens the Lines distribution dialog box where position of successive workframe axes may be determined.

The *Axis descriptions* tab is shown in the figure below.



The following options are provided in this dialog box:

- *Length* (Y) - fields where an axis description along the workframe length is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated
- *Width* (X) - fields where an axis description along the workframe width is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated
- *Height* (Z) - fields where an axis description along the workframe height is determined; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated.

For each of the options listed three fields are accessible:

- *Prefix* - field where a prefix of workframe axes can be determined for the automatic description of workframe axes
- *Start value* - field where a start value for workframe axes can be determined for the automatic description of workframe axes
- *Step* - field where a value of increment for successive workframe axes can be determined for the automatic description of workframe axes.

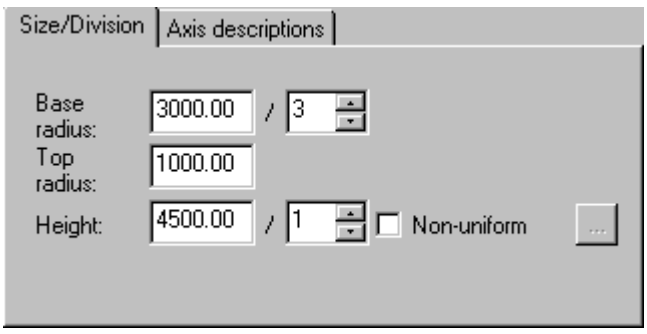
For the width (X) and the length (Y) it is possible to choose the location of a description: on the right or on the left side of a workframe line.

If the *User-defined* option next to the options listed, is switched off, then a workframe description will be generated automatically based on the parameter values defined in the edit fields discussed. If the *User-defined* option is switched on for any of the options, then the edit fields for that option become inaccessible; instead, the (**...**) button becomes available which - when pressed, opens the Lines description dialog box where the description of successive workframe axes can be determined.

5.4. Prism

Once prism is selected as a workframe type, the Workframe dialog box contains two tabs: *Size/Division* and *Axis descriptions*.

The *Size/Division* tab is shown in the figure below.

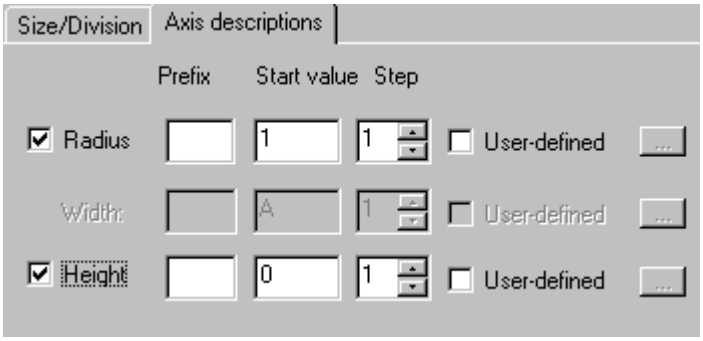


The following options are provided in this dialog box:

- *Base radius* - fields that determine a radius of the circle circumscribed on the prism base and a number of radial divisions
- *Top radius* - field that determines a radius of the circle circumscribed on the prism top
- *Height* - fields that determine height of a workframe and number of divisions along the workframe height.

If the *Non-uniform* option next to the *Height* option is switched off, then a workframe will be generated automatically based on the parameter values defined in the edit fields discussed. If the *Non-uniform* option is switched on, then the edit fields become inaccessible; instead, the (**...**) button becomes available which when pressed, opens the Lines distribution dialog box where position of successive workframe axes can be determined.

The *Axis descriptions* tab is shown in the figure below.



The following options are provided in this dialog box:

- *Radius* - fields that determine an axis description along the workframe radius; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated
- *Width* - the option is not available
- *Height* - fields that determine an axis description along the workframe height; if the option is switched on (the √ symbol appears), the description will be generated, and if the option is switched off, the axis description will not be generated.

For each of the options listed three fields are accessible:

- *Prefix* - field where a prefix of workframe axes can be determined for the automatic description of workframe axes
- *Start value* - field where a start value for workframe axes can be determined for the automatic description of workframe axes
- *Step* - field where a value of increment for successive workframe axes can be determined for the automatic description of workframe axes.

If the *User-defined* option next to the options listed, is switched off, then a workframe description will be generated automatically based on the parameter values defined in the edit fields discussed. If the *User-defined* option is switched on for any of the options, then the edit fields for that option become inaccessible; instead, the (**...**) button becomes available which when pressed, opens the Lines description dialog box where the description of successive workframe axes can be determined.

5.5. Lines distribution

If the *Non-uniform* option is switched on for any of the options on the *Size/Division* tab of the Workframe dialog box, then the edit fields become inaccessible; pressing the (**...**) button opens the ***Lines distribution*** dialog box which provides the table that allows the user to determine position of the successive workframe axes.
To define the position of workframe axes, the user should:

- define spacing between successive workframe axes or define the coordinate of the workframe axis (calculated from the workframe beginning)
- perform the above operation for the successive workframe axes
- press the **OK** button.

5.6. Lines description

If the *User-defined* option is switched on for any of the options on the *Axis descriptions* tab of the Workframe dialog box, then the edit fields become inaccessible; pressing the (**...**) button opens the ***Lines description*** dialog box which provides the table that allows the user to determine descriptions of the successive workframe axes.
To define descriptions of workframe axes, the user should:

- define description of the first axis
- move to the next table line and define description of the second axis
- for some descriptions (e.g. width - X and length - Y) select location of the axis description; on the right or on the left side of the workframe line
- perform the above operation for the workframe axes defined
- press the **OK** button.

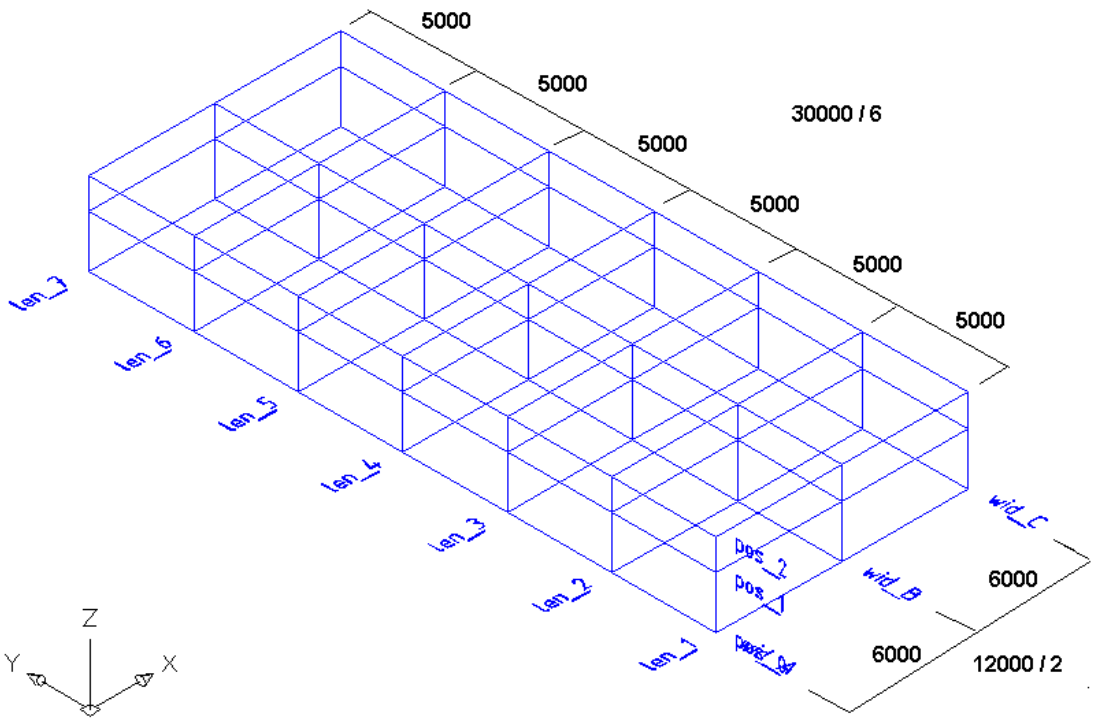
5.7. Example of definition of a workframe

To start definition of a workframe, do as follows:

- select the menu option *Steel / Workframes / Create workframe* or press the icon 
- in the ***Workframe*** dialog box determine the following parameters:
  in the *Name* edit field enter: Workframe - 3D Workshop
  select the workframe type: *Box*
  switch off the following options: *Surface only*, *Left diagonal*, *Right diagonal*, *Without description*
  switch on the option *Show axes in drawings*
- on the *Size / Division* tab enter the following workframe dimensions with respect to the system axes:
  Width (X): 12000 / 2
  Length (Y): 30000 / 6
  Height (Z): switch on the *Non-uniform* option and press the (**...**) button located next to this option; in the ***Line distribution (length)*** dialog box enter the values as follows:
  in the *Coordinate* column: 5000, 7500, 9000; values of axis spacings will be defined automatically; press the **OK** button in the ***Line distribution (length)*** dialog box
- move on to the *Axis descriptions* tab, switch on descriptions for all the dimensions (the √ symbols appear) and choose the following descriptions for individual workframe axes:
  Width (X): Prefix: wid_, Start value: A, Step: 1, left
  Length (Y): Prefix: len_, Start value: 1, Step: 1, left
  Height (Z): Prefix: pos_, Start value: 0, Step: 1
- press the *Create*  icon located in the right-hand part of the ***Workframe*** dialog box

- specify the workframe origin and determine the direction of the workframe X axis: direction of the X axis of the global coordinate system (NOTE: the direction of the X axis of the workframe does not need to be compatible with the X axis of the global coordinate system).

The generated workframe is illustrated in the drawing below.



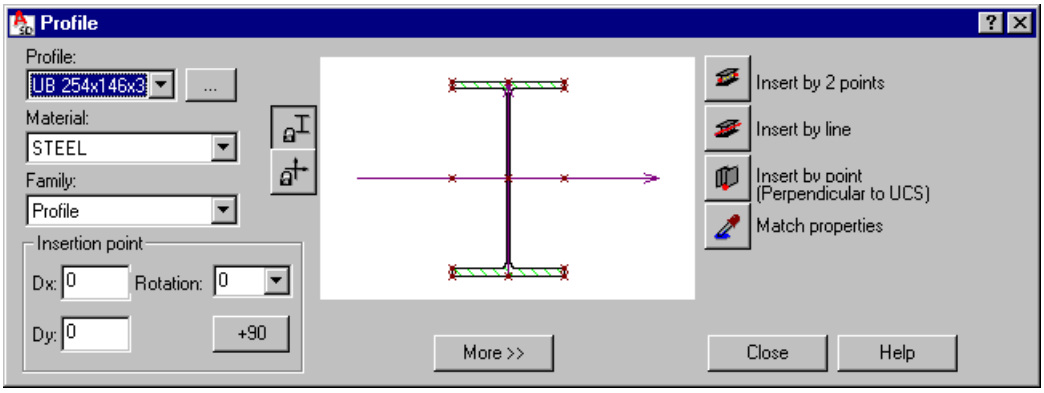
6. PROFILES

6.1. Profiles

The options allows defining a member with a selected profile. The option is available from:

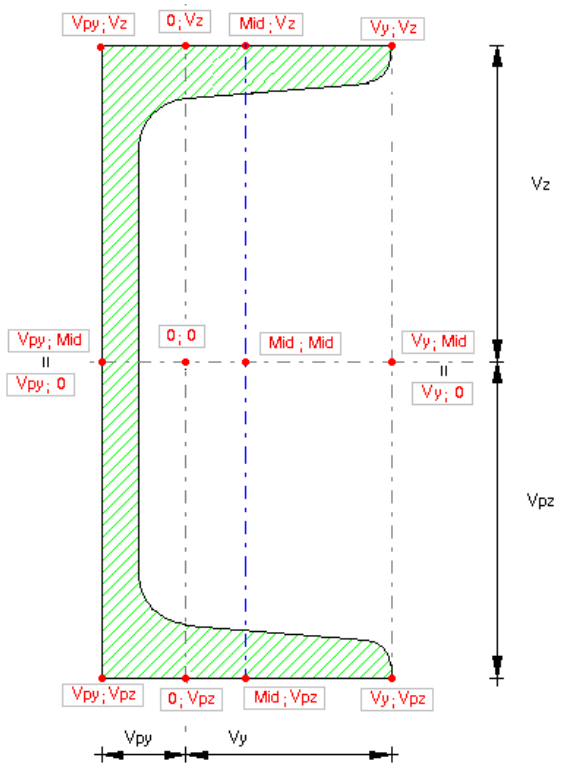
- the menu by selecting the *Steel / Profiles* option
- the toolbar by pressing the icon
- the command line: RBCS_PROFILE.

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



The ***Profile*** dialog box may be divided into three parts:

- in the left part of the dialog box a user may define basic information concerning a profile of a defined member:
  on the *Profile* list a user may select a profile for a defined member (on the list the member profile defined recently is proposed); pressing the (**...**) button opens the Profile List dialog box in which profiles from the databases available in the program may be added to the list of available profiles.
  on the *Material* list the user may select type of a material assigned to a defined member
  on the *Family* list the user may select a family (group) of members to which a defined member will be assigned
  in the *Insertion point* field a user may determine coordinates of the member shift (offset), i.e. Dx, Dy as well as rotation of a member profile about the member longitudinal axis (a value of rotation may be selected from the values proposed on the list or may be entered into the *Rotation* field); the offset denotes shift of a profile center with respect to the insertion line
- the middle part of the dialog box comprises the graphical field presenting a selected profile of a defined member positioned in relation to the insertion line (under the graphical field characteristic distances of the profile fibers from the profile axis are presented); the graphical field enables graphical definition of offsets: it can be done by shifting a profile with respect to the axis by means of the mouse or by indicating characteristic points of a profile.
  Options provided in the dialog box also enable definition of relative offsets (with respect to values related to the cross section and characteristic points of a profile); location of section characteristic points (their number depends on the section type and shape) may be determined by specifying certain values connected directly with the cross section (these do not have to be absolute values) – characteristic points include:
  - (0;0) center of gravity of a section
  - points presented in the drawing below whose coordinates are determined by the values Vy, Vpy, Vz, Vpz, Mid (they define a relative location of a point in the cross section)



Profile characteristic points are marked in red in the dialog box (in the drawing of a selected profile); after clicking on a characteristic point of a profile, the profile is translated and the edit fields *Dx* and *Dy* in the *Insertion point* field are filled out automatically.
Under the schematic drawing of a profile is the **More >>** (or **Less <<**) button, which when pressed opens (closes) an additional field where:
1. a surface finishing type for a profile (e.g. hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat) can be chosen; a default type of surface finishing can be selected in the ***Project preferences*** dialog box on the *Materials* tab; this dialog box also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types
2. profile properties (area, moments of inertia, etc.) are presented – see the description of the Profile list dialog box; after positioning the mouse cursor in this field and pressing the right mouse button, the context menu appears on the screen – the following options are available there:
- *Select All* – selects the contents of the edit field
- *Copy* – copies the contents of the edit field to the Clipboard
- *Paste* – pastes the Clipboard contents to the edit field

The following two icons are also available in this part of the dialog box:

 - once it is pressed, position of axes in the graphical field is blocked and user may shift a profile

 - once it is pressed, position of a profile in the graphical field is blocked and user may shift profile axes

- the right part of the dialog box contains the following icons that allow defining a member by means of:

 - 2 points

 - line

 - 1 point (a member will be defined as perpendicular to UCS)

and the  icon - used to adopt (inherit) parameters from a member profile that is already defined.

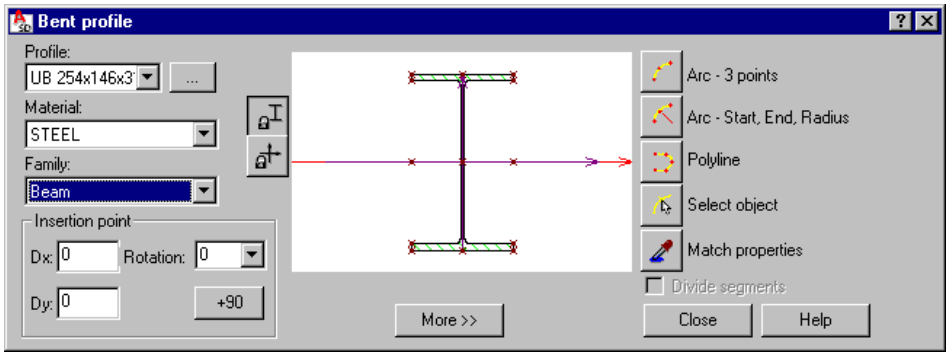
The dialog box opens showing parameters of the member profile defined recently.

6.2. Bent profiles

The option enables defining a bent bar of selected profile. The option is available from:

- menu by selecting the option *Steel / Bent profiles*
- toolbar by pressing the  icon
- command line: RBCS_BENT.

After selecting the *Bent profiles* option, the dialog box shown in the drawing below appears on the screen.



Three parts may be distinguished in the ***Bent profile*** dialog box:

- in the left part of the dialog box basic information concerning the profile of a defined bar may be determined:
  from the *Profile* list the user may choose a profile of a defined bar (the last-defined bar profile is proposed on the list); pressing the (**...**) button opens the Profile List dialog box, where profiles from the databases accessible in the program may be added to the list of available profiles
  from the *Material* list a type of material assigned to a defined bar may be selected
  from the *Family* list the user may select a bar family (group), to which a defined bar will be assigned
  the *Insertion point* field allows the user to determine coordinates of the bar offset: Dx, Dy and rotation of the bar profile about the bar longitudinal axis (value of rotation may be chosen from the values proposed on the list or entered in the *Rotation* field); the offset denotes translation of the profile center with respect to the insertion line
- the middle part of the dialog box contains the graphical field presenting a selected profile of a defined bar positioned in relation to the insertion line (under the graphical field characteristic distances of the profile fibers from the profile axis are presented); the graphical field enables graphical definition of offsets: it can be done by translating a profile with respect to the axis by means of the mouse or by indicating characteristic points of a profile.
  Under the schematic drawing of a profile is the **More >>** (or **Less <<**) button, which when pressed opens (closes) an additional field where:
  1. a surface finishing type for a profile (e.g. hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat) can be chosen; a default type of surface finishing can be selected in the ***Project preferences*** dialog box on the *Materials* tab; this dialog box also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types

2. profile properties (area, moments of inertia, etc.) are presented – see the description of the Profile list dialog box; after positioning the mouse cursor in this field and pressing the right mouse button, the context menu appears on the screen – the following options are available there:
- *Select All* – selects the contents of the edit field
- *Copy* – copies the contents of the edit field to the Clipboard
- *Paste* – pastes the Clipboard contents to the edit field

There are also two icons available in this part of the dialog box:

 - when pressed, position of axes in the graphical field is blocked and the user may translate a profile

 - when pressed, position of a profile in the graphical field is blocked and user may translate profile axes

- the right part of the dialog box contains the following icons that allow definition of a bar by:

 - 3 arc points

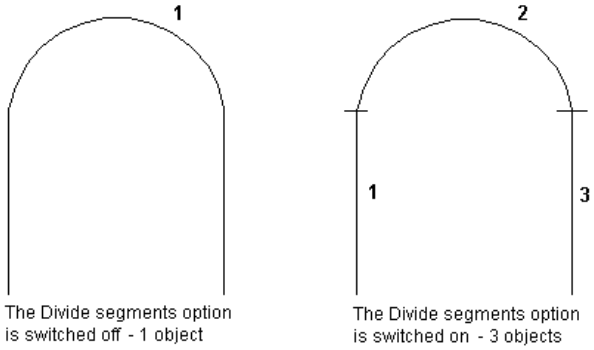
 - arc: start, end and radius of an arc

 - polyline

 - selection of object

and the  icon - used to adopt (inherit) parameters from a bar profile defined earlier.

In the bottom part of the dialog box the *Divide segments* option is provided; if the option is switched off, then a bent profile is defined as one object (despite that that it may consist of several 'parts', e.g. a polyline composed of arc, segment and arc); if this option is switched on, the a bent profile is divided into separate objects when the type of profile parts changes (a polyline composed of arc, segment and arc is divided into three separate parts: 2 arcs and a segment). The drawing below illustrates schematically how this option works.



The dialog box opens showing parameters of the bar profile defined recently.

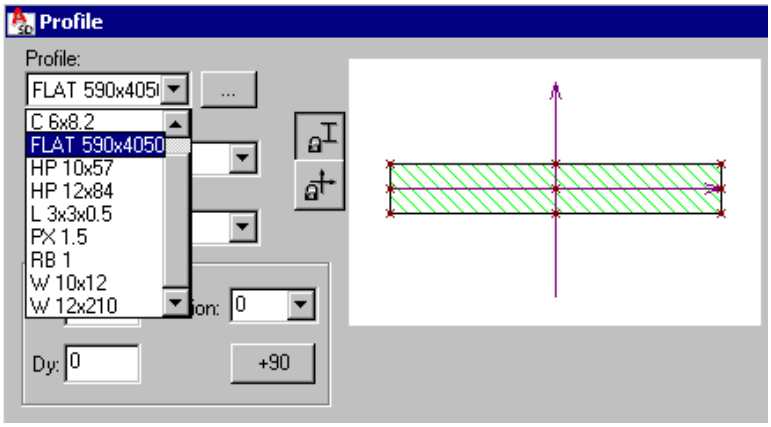
6.3. Example of definition of a user-defined section

The example below presents definition of a section of a steel flat bar that may be used in definition of a structure model. To define a user-defined section (bar section), do as follows:

- applying the AutoCAD ® program options:
  - draw a rectangle (the option *Draw / Rectangle*) with the dimensions 590x4050 (see the drawing below)
  - select a region (the option *Draw / Region*) and indicate the rectangle drawn

- applying ***AutoCAD Structural Detailing - Steel*** options:
  - run the option *Steel / User-defined sections* or press the icon 
  - indicate the defined region (rectangle)
  - select saving of the section in the current project (the *Locally* option in the command line)
  - specify a section name, e.g.: FLAT 590x4050 (a name may consist of 4 letters and designations describing profile dimensions)
  - define a profile shape code.

NOTE:
Section codes are adopted according to designations used in ***Robot****; code descriptions are provided in the* ***Robot*** *help.*
Thus-defined section of a profile will be added to the list of available profiles (e.g. in the ***Profiles*** dialog box, a part of which is shown in the drawing below). The section will be available only in the current project.



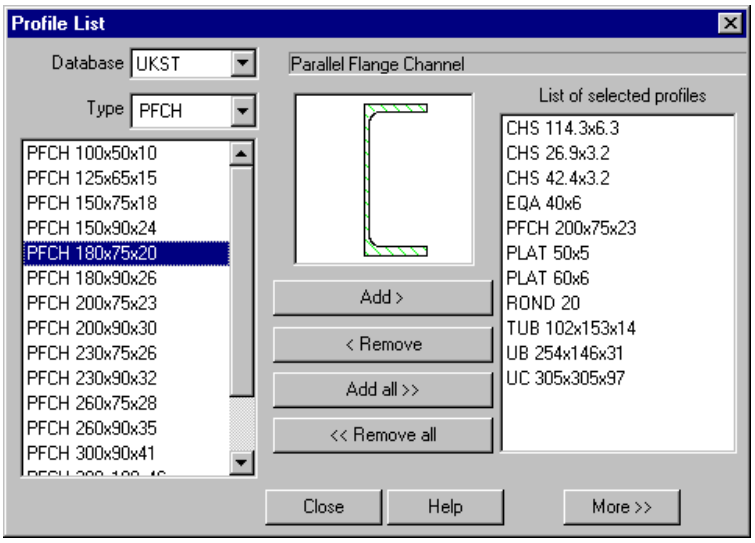
If the *Database* option is selected, then to save a profile to a database, the user should follow the steps listed below in the ***Saving section to database*** dialog box:

- specify a name: maximally four letters (Attention: using digits is not allowed)
- define dimensions of the profile (real numbers defining information about the profile):

= if one dimension is given, specify Dimension 1
= if two dimensions are given, specify Dimension 1 and Dimension 3.

6.4. Profile list

The ***Profile list*** dialog box shown in the figure below opens after pressing the **(…)** button provided in the Profile dialog box.



The above dialog box enables adding profiles from databases accessible in the program to the list of profiles available in ***AutoCAD Structural Detailing - Steel***. To add a profile to the list of available profiles, the user should:

- select a profile folder in the *Database* field, e.g. AISC (American Hot Rolled Shapes)
- select a profile type
- select a profile in the left panel
- press the **Add >** button.

Pressing the **Add all >>** button results in adding all the profiles included in the left panel.
Pressing the **< Remove** button deletes a selected (highlighted) profile from the list of available profiles. Pressing the **<< Remove all** button results in deleting all the profiles provided on the list of available profiles (in the right panel) except for the profiles that have already been used in the project.
The *Remove* option is unavailable for profiles that have already been applied in a project.
The bottom right corner of the dialog box contains the **More >>** (or **Less <<**) button; if pressed, it opens an additional part of the dialog box with information on basic section properties (dimensions of the cross section, area, moments of inertia, section modulus for calculation of torsion stresses, elastic section moduli (bending), mass). This additional part of the dialog box is closed on pressing the **Less >>** button.

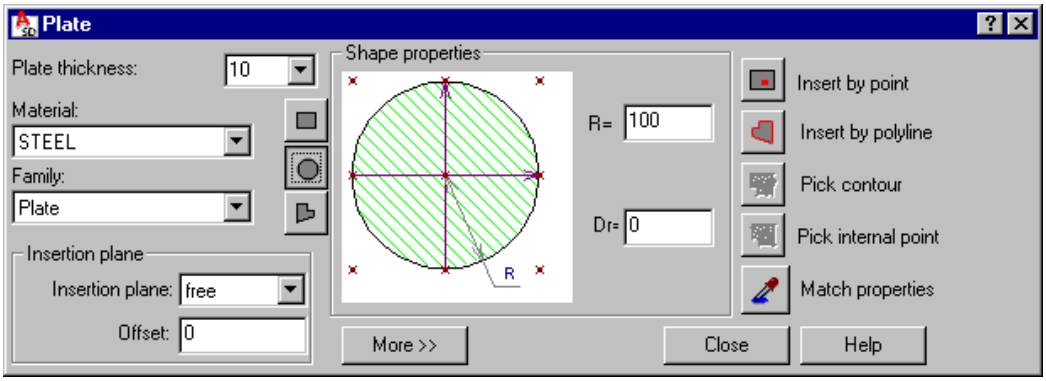
7. PLATES

7.1. Plates

The option allows defining a plate. The option is available from:

- the menu by selecting the *Steel / Plates* option
- from the toolbar by pressing the icon
- the command line: RBCS_PLATE.

Once the *Plates* option is selected, the dialog box shown in the drawing below, is displayed on the screen.



The ***Plate*** dialog box may be divided into three parts:

- in the left part of the dialog box a user may define basic information concerning a defined plate:
  on the *Thickness* list a user may select a value of plate thickness from the values proposed on the list or enter this value into the *Thickness* field
  on the *Material* list the user may select the type of material assigned to a defined plate
  on the *Family* list the user may select a family (group) of plates to which a defined plate will be assigned
  the following three icons are also available:
  - once this icon is pressed, the defined plate will be a rectangular plate
  - once this icon is pressed, the defined plate will be a circular plate
  - once this icon is pressed, a plate shape will be defined by a user (it will be a closed polyline)

  the *Insertion plane* field enables a user to determine the plane of plate insertion as upper, lower, middle and free; a user may also determine here a value of the offset denoting the distance to the insertion plane related to UCS
  the middle part of the dialog box comprises a graphical field presenting a defined plate positioned in relation to the insertion point; in the graphical field a user may define offsets graphically: it is possible by shifting the plate with respect to the point by means of the mouse or indicating characteristic points of the plate
  the edit fields included in the middle part of the dialog box are used to determine plate dimensions; depending on the plate type - circular, rectangular, arbitrarily shaped - the following parameters are available:
  circular plate - value of R radius and value of Dr offset that determines the distance of the insertion point from the center of a circular plate
  rectangular plate - dimensions of W and L sides and dimensions of Dx and Dy offsets that determine distances between the insertion point and center of a rectangular plate
  Under the schematic drawing of a plate there is the **More >>** (or **Less <<**) button, which when pressed opens (closes) an additional field where a surface finishing type for a plate

(e.g. hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat) can be chosen; a default type of surface finishing can be selected in the ***Project preferences*** dialog box on the *Materials* tab; this dialog box also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types

- the right part of the dialog box contains the following icons that allow plate definition by means of:

   - point

   - polyline

   - contour (a contour should be indicated) – the option is available for plates defined by the user

   – internal point (an internal point of the contour should be indicated) - the option is available for plates defined by the user

  and the  icon - used to adopt (inherit) parameters from a plate that is already defined.

The dialog box opens showing the plate parameters defined recently.

7.2. Plate definition

In order to define a plate, the user should first follow the steps listed below:

- select the plate shape type (rectangular, circular or user-defined)
- define plate dimensions (number of dimensions depends on a selected shape of a plate)
- define plate thickness
- determine a material that a plate will be made of
- determine a family to which a plate will belong
- if needed, select a surface finishing type for a plate (after pressing the **More >>** button)
- determine a plane of plate insertion and if need be, a value of the offset denoting the distance to the insertion plane related to UCS.

Once these actions are performed, a plate should be defined by pressing one of the icons:

 a plate will be defined by indicating the insertion point and direction

 a plate will be defined by defining a polyline, rectangle or circle depending on the selected type of plate

 a plate will be defined by indicating an existing contour (polyline) - the option is available for plates defined by the user

 a plate will be defined by indicating an internal point of the existing contour (polyline) - the option is available for plates defined by the user.

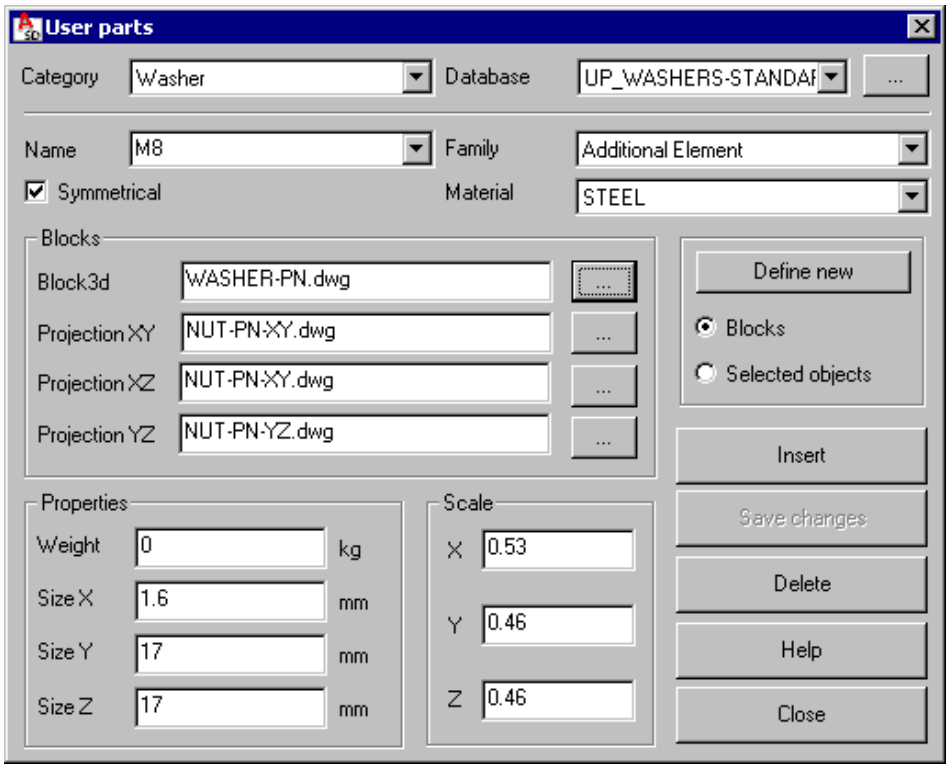
8. USER PARTS

8.1. User parts

The option enables definition of user parts (non-standard structure elements, e.g. prefabricated elements, user-defined elements, etc.). User parts have properties of a standard element, e.g. weight, position symbol, etc.; these parts are presented in documentation. The option is accessible from:

- the menu by selecting the option *Steel / User parts*
- the toolbar by pressing the icon
- the command line: RBCS_USERPART.

After calling up the *User parts* option, the dialog box shown below appears on the screen.



Objects that are user parts are defined in the model layout; they are presented on printouts as well. A form how they are presented on printouts is determined by the user through blocks linked with user parts. Blocks may consist of any objects of the AutoCAD ® program.

User parts are saved to databases which, in turn, are grouped into categories (one category may include any number of databases). Databases are separate *.mdb format files; each database has a description displayed in the dialog box. There is a possibility to add user's own categories and add databases in selected categories.

NOTE:
A database may belong only to one category.

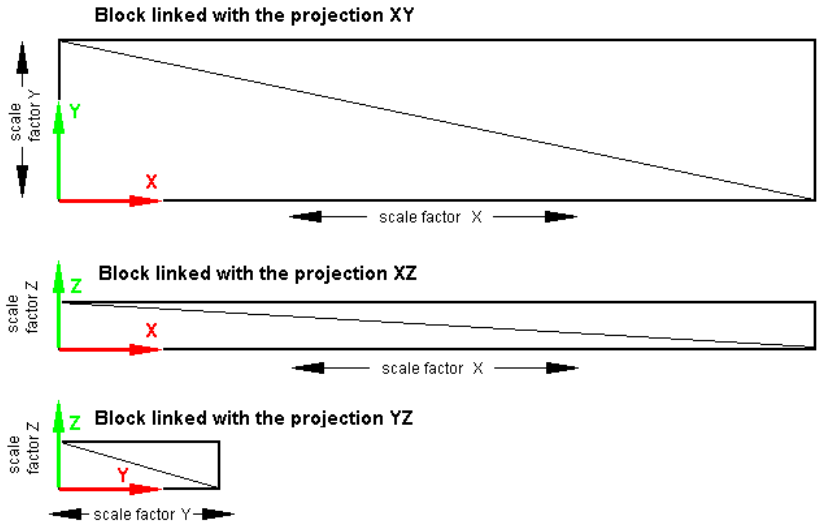
User parts that belong to one or several databases may be linked with the same blocks. A block must exist and be saved on the hard disk. User parts included in databases are linked with blocks (*.dwg format files), which determine the way they are presented (symbol) in a model and on printouts.

User parts belong to families; that where they belong determines their properties such as color, layer, etc. User parts may be assigned positions, however, note should be taken that while positioning the program does not check geometry of user parts. The criteria for identification of user elements are: their name, category as well as properties saved in the database.
A user part weight defined in the dialog box above is taken into account while calculating the weight of a selected structure part and the center of gravity of a structure.

As mentioned above, user parts are defined in the model layout and presented in this layout in the form of blocks linked with them. For a chosen part, the block may be scaled independently in the directions of the local coordinate system of an object. Scale factors refer to the directions determined by the local coordinate system of a 3D block. The same scale factors are applied both to a three-dimensional block in the model layout and to 2D blocks presenting user parts in drawings.

Once positioned, user parts may be drawn as single parts or be presented in drawings of assemblies or groups. Drawings of user parts are generated automatically, considering the location and orientation (the local coordinate system) of a part defined in the model layout. In drawings user parts are shown as blocks linked with them.

Below is presented a drawing that illustrates which scale factors are used for views and directions.



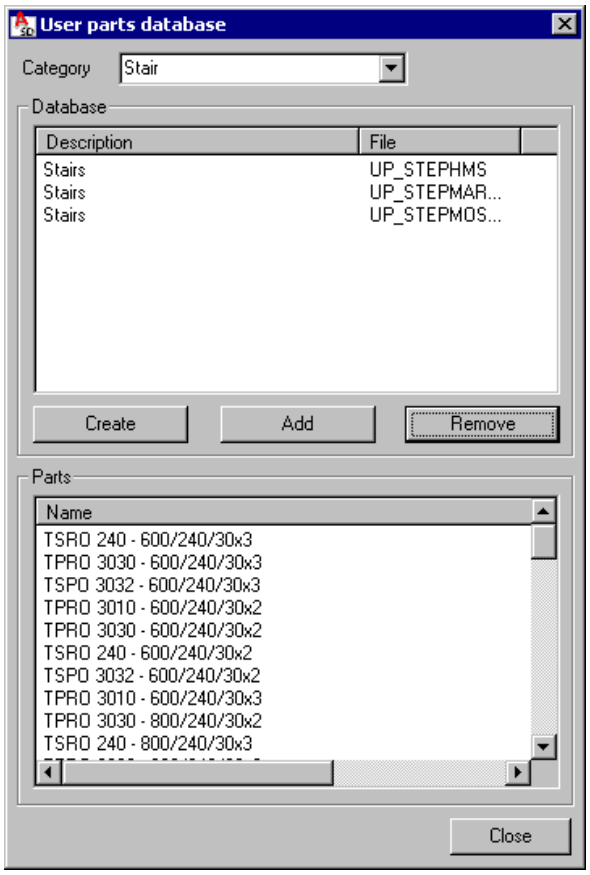
User parts are presented in drawings only in othogonal projections.
User parts shown in drawings of assemblies or groups may be presented (depending on the user selection) using the line type and the line color chosen in a dimensioning style.
User parts cannot be dimensioned; descriptions for them are generated in the same way as for single parts (with the possibility to use variables).

User parts defined in the model layout are modifiable; take note, however, that not all parameters of user parts can be changed.
It is possible to execute the basic edit commands of the AutoCAD ® program on user parts, like copying, translation, etc.
User parts may be components of assemblies, groups or schemes.

NOTE:
There is no possibility to perform the machining operations using the ***AutoCAD Structural Detailing*** *options on user parts.*

8.2. User part database

The ***User parts database*** dialog box illustrated in the drawing below makes it possible to perform operations in databases and categories. It also enables viewing the contents of individual databases.
The dialog box opens on pressing the (**…**) button located in the ***User parts*** dialog box.



The basic operations that may be performed in the dialog box above include:

Adding a new category
It allows creating a new category.

After pressing the ***Add new*** button, information about a new category may be given: the category name in the *Category* field as well as the category description and file.

Creating a new database
It allows creating a new database (file) in the default folder; the database will belong to a category chosen from the selection list.

In the dialog box, a name and a description of the database should be specified.

8.3. Definition of a new user part

Below is presented the procedure adopted while defining a new user part.

Definition of new user part
Pressing the (**...**) button in the top part of the ***User parts*** dialog box adds a new user part to the current database belonging to a selected category.

A part may be added in two ways (by choosing one of the options mentioned):
- by blocks
- by selection of parts.

Blocks
Once this option is selected, adding a new object consists in linking parts with blocks saved in external *.dwg files.
After pressing the **Define new** button, define a name of a user part in the *Name* field. Values in the remaining edit fields for the newly-defined user part are inherited from the previous user part.
In the central part of the dialog box specify access paths to external *.dwg files. The insertion point and the local coordinate system of an object are defined in DWG files.
In the lower part of the dialog box determine parameters of the user part (a weight, a scale and dimensions for individual directions).

Selected objects
Once this option is selected, adding a new object consists in indicating on the screen objects that will make up blocks linked to the user part.
After pressing the **Define new** button, the ***User parts*** dialog box closes; specify a name of the user part in the command line. Next, the command line shows the following commands that enable:
- selection of objects that will make up the blocks linked with the 3D projection
- selection of objects that will make up the blocks linked with the projections: XY, XZ and YZ.

For all objects it is necessary to determine the base point and a point positioned on the positive side of the axis. Blocks for projections are saved in *.dwg files.
Once selection of objects for the mentioned projections is complete, the ***User parts*** dialog box appears on the screen. Edit fields in the upper and central parts of the dialog box are filled out with names given while defining the projections.
In the lower part of the dialog box determine parameters of the user part (a weight, a scale and dimensions for individual directions); by default values of the weight and dimensions equal zero, whereas scale factor values equal 1.0.

Insertion of a user part
Pressing the **Insert** button inserts the user part chosen in the dialog box into the model layout at the point indicated by the user.

Saving of modifications
Pressing the **Insert** button accepts changes made in the existing databases (the button is inactive until changes are made).

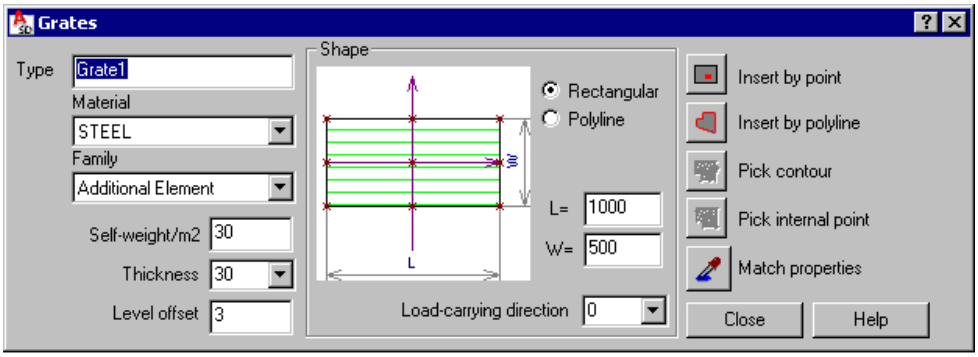
9. GRATES

9.1. Grates

The option enables definition of grates. The option is available from:

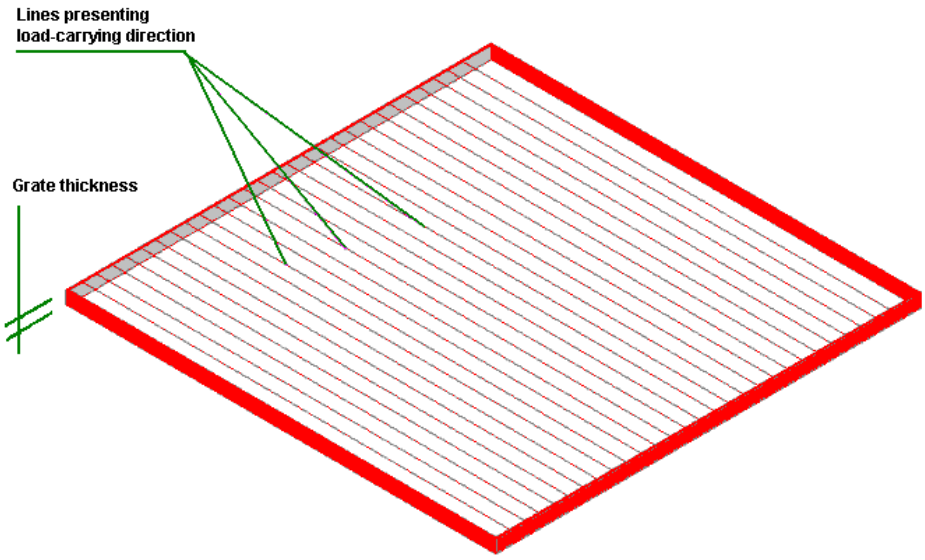
- the menu by selecting the option *Steel / Grates*
- the toolbar by pressing the icon
- the command line: RBCS_GRATE.

After calling up the *Grates* option, the dialog box as shown below, appears on the screen.



The ***Grates*** dialog box may be divided into three parts:

- the left part of the dialog box is used to determine basic information about a defined grate:
  in the *Type* field a name of the defined grate may be specified
  the *Material* list allows selecting a material type assigned to the defined grate
  from the *Family* list it is possible to choose a family (group) of grates to which the defined grate will be ascribed (e.g. Additional Element)
  moreover, values of the following parameters have to be defined:
  - self-weight – based on this value the program calculates a grate weight
  - thickness – a grate thickness value may be selected out of the values on the list or typed in the *Thickness* field
  - level offset – a value of the distance between the grate insertion point and the bottom left corner of the grate; an offset equals zero, if the the bottom left corner of the grate coincides with the grate insertion point
- in the central part of the dialog box is a graphic field presenting the defined grate
  two types of the grate shape are available:
  - rectangular grate
  - grate with any shape defined by means of a polyline.
  edit fields located in the central part of the dialog box are used to determine dimensions of the grate (it refers to a rectangle-shaped grate): side dimensions W and L
  Apart from that, there is a possibility to define a load-carrying direction for the grate (an angle in the XY plane); the direction is presented in the model layout and drawings using the symbol ; the longer axis of the symbol is an axis parallel to the load-carrying direction of the grate; the symbol of the load-carrying direction is presented (in the same color and the same style as the grate edges) only in a top view and a 3D view of the grate in the plane parallel to the grate plane; the grate with its basic elements is illustrated below



- the right part of the dialog box holds the following icons that make possible definition of the grate by means of:

  - point (this method is not available for a shape defined by polyline)
  - polyline
  - contour (a contour should be indicated) – this option is accessible for grates defined by the user (this method is not available for the rectangular shape of the grate)
  - internal point (an internal point of a contour should be indicated) - this option is accessible for grates defined by the user (this method is not available for the rectangular shape of the grate)

  and the icon – is used to adopt parameters of the earlier-defined grate.

The dialog box opens showing the most-recently defined parameters of the grate.

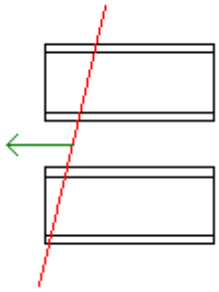
10. MACHINING

10.1. Fit to line

The option enables the user to perform the operation of fitting a profile to a defined line. The option is available from:

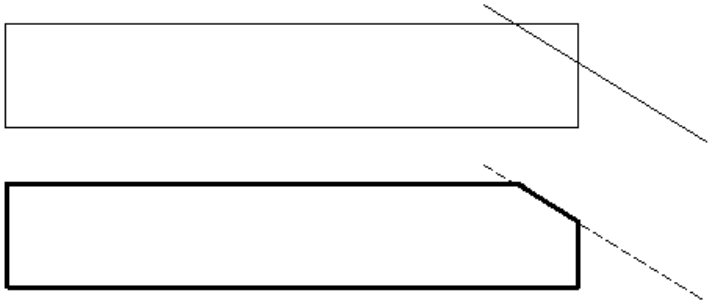
- the menu by selecting the *Steel / Machining / Fit to line* option
- the toolbar by pressing  icon
- from the command line: RBCS_FITTOLINE.

The operation of fitting a profile may be performed either by means of a line defined using the AutoCAD ® program tools or by defining the position of two points determining a line. The user may, as well, fit several profiles positioned on the cutting line (see the drawing below).

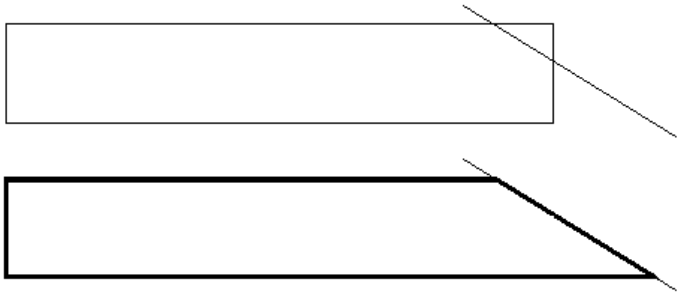


The *Cut by line* option operates in the similar manner. Moreover, the direction is indicated (a point on one side of the cutting line); it determines the part of an object which remains after the cutting operation is performed. The difference in the manner the options *Fit to line* and *Cut by line* operate is presented below - the drawings show an object before performing the fitting and cutting operations (together with a defined line) and an object generated after performing the fitting and cutting operations.

Cut by line



Fit to line

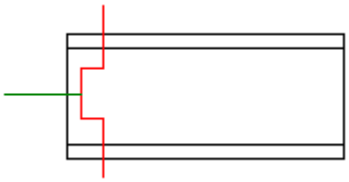


10.2. Fit to polyline

The option enables the user to perform the operation of fitting an object to a defined broken line (polyline). The option is available from:

- the menu by selecting the *Steel / Machining / Fit to polyline* option
- the toolbar by pressing icon
- from the command line: RBCS_FITTOPOLY.

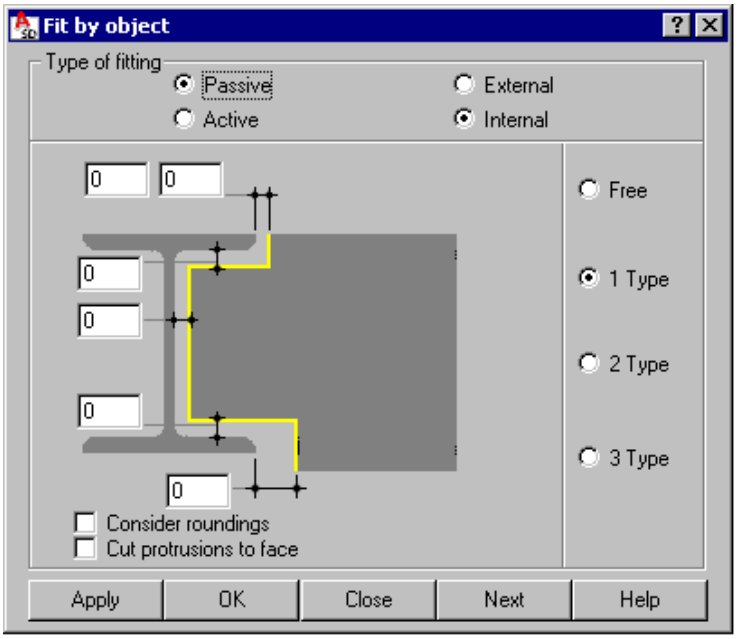
The plane of fitting (cutting off) consists of a few segments, as shown in the drawing below.



10.3. Fit to object

The option enables the user to perform the operation of fitting two structural elements to each other. The option is available from:

- the menu by selecting the *Steel / Machining / Fit to object* option
- the toolbar by pressing icon
- from the command line: RBCS_FITOBJ.



Once the *Fit to object* option is selected, the user should indicate two structure elements that are to be fitted to each other: object that is being adjusted and object to which the element selected earlier should be adjusted. After selecting the objects, the dialog box shown in the drawing below is displayed on the screen.
In the top part of the dialog box – in the *Type of fitting* field the user may select as follows:
mode of object fitting:

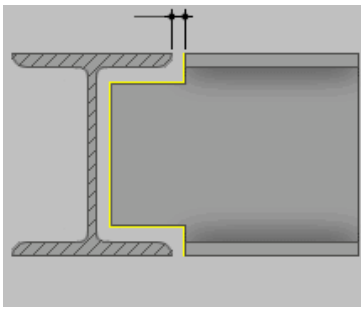
- *Active* - if this option is selected, then fitting of objects remembers the object to which the selected part is being fitted; fitting of objects is updated each time after modification of the fitted object and the object to which the object is being fitted (after modifying a position of any of the objects, the parts will be automatically fitted applying all the parameters defined in the dialog box); if the object to which the selected part is being fitted, is deleted, then fitting of objects is deleted, as well
- *Passive* – if this option is selected, then fitting of objects is not updated each time after modification of the fitted object and the object to which the object is being fitted.

NOTE:
Once fitting of objects is completed, the mode of object fitting (passive, active) may be changed in the ***Properties*** *dialog box.*

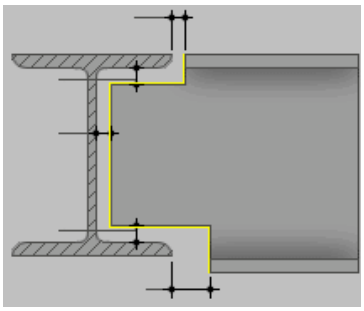
type of object fitting:

- *External* - mode aimed at fitting one of the chosen objects to the external contour of another object
- *Internal* - mode aimed at fitting one of the chosen objects to the internal contour of another object.

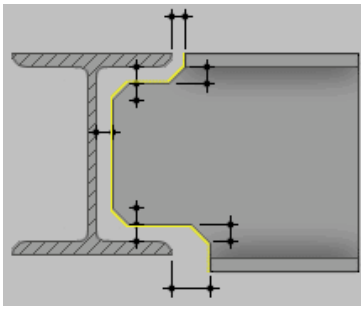
If the internal mode is selected, there are four options available:



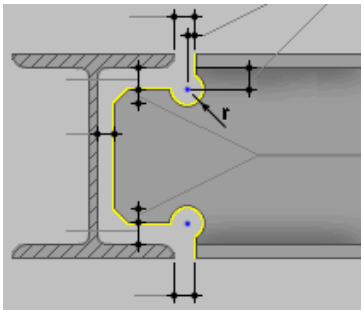
Free
mode applied to perform quick fitting operation; in this case determining parameter values is not required.
If due to technological reasons more exact parameter determination is required, then the user may apply one of the three methods described below.



1 Type
A distance value may vary for flanges and web of the element which undergoes fitting.



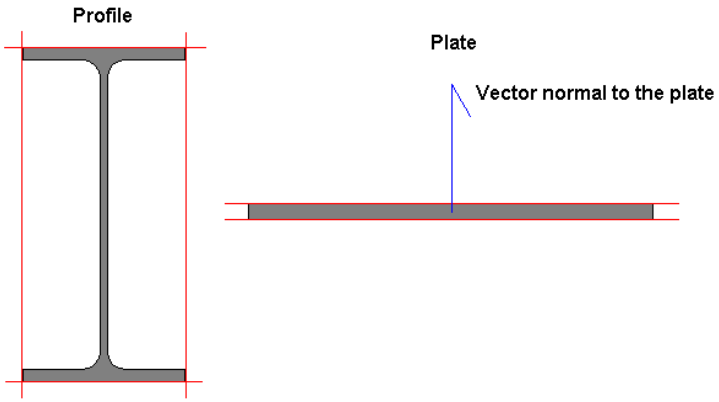
2 Type
For this type of fitting the user may switch on/off chamfering. Note: An angle applied during the chamfering operation performed on some of the elements always equals 45 degrees.



3 Type
This type of fitting enables adjusting objects with additionally-defined openings. Opening parameters include: position of the opening center, translations of the opening center position and radius value. It is also possible to perform the chamfer operation on some elements of the parts which undergo fitting.

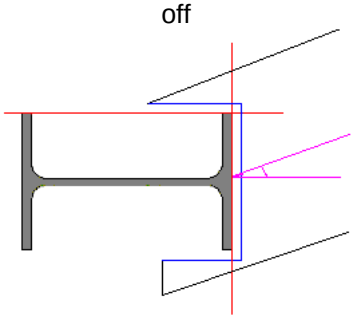
In the bottom part of the dialog box there are the following options:

- *Consider roundings* - if this option is switched off, then roundings of rolled profiles will not be considered while generating a cutting plane
- *Cut protrusions to face*; if the option is switched on, additional planes that form a contour of a part (profile, plate) are accounted for in a cut shape; these planes are shown in red in the drawings below; out of these planes the program selects this one, in the case of which the value of the inclination angle between the plane and the object cut is the closest to 90 degrees.

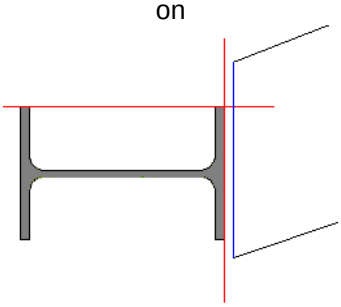


The operation of the *Cut protrusions to face* option is illustrated in the drawings below.

The *Cut protrusions to face* option is switched off



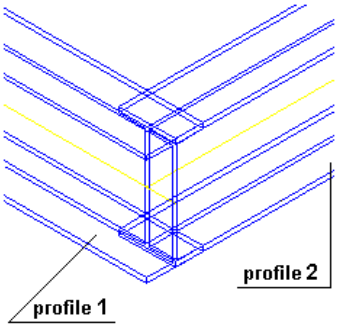
The *Cut protrusions to face* option is switched on



Pressing the **Apply** button changes parameters of the selected objects. If the **OK** button is pressed, it results in performing the defined operation of object fitting based on the adopted parameters and closes the dialog box.
When pressed, the **Close** button causes the defined operation to be conducted without saving the changes made recently.
Pressing the **Next** button starts fitting of successive objects.

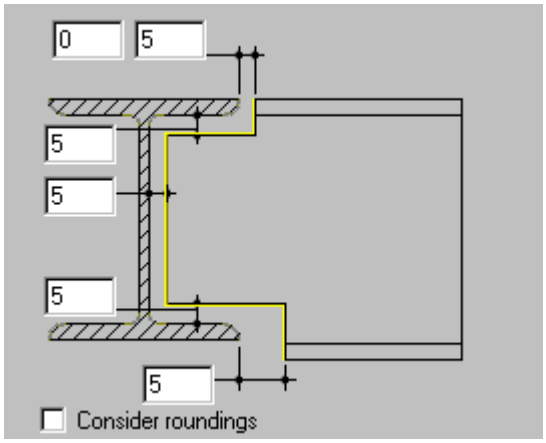
10.4. Example of the fit to object operation

To start fitting to each other the two I-sections illustrated in the drawing below, do as follows:

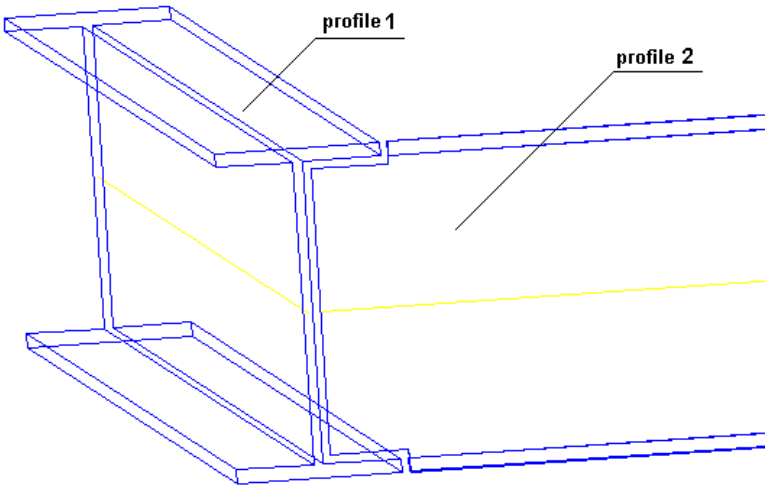


- select the menu option *Steel / Machining / Fit to object* or press the icon 
- indicate the profile to which the other profile should be fitted (in the drawing this is the profile no. 1) and the profile which should be fitted to the profile no. 1 (in the drawing this is the profile no. 2)

- in the ***Fit by object*** dialog box determine the parameters as follows:
  Type of fitting: *Passive*, *Internal*
  Type: *1 Type*
  switch off the *Consider roundings* option
  adopt the fitting dimensions as presented in the drawing below



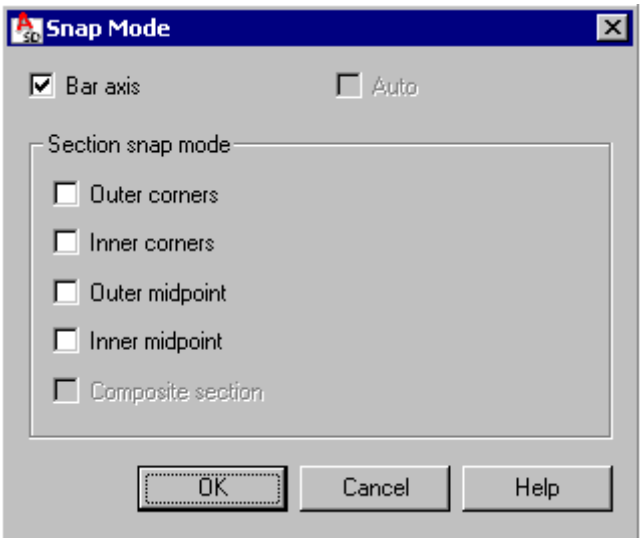
- press the **Apply** button in the ***Fit by object*** dialog box; the I-sections fitted are presented in the following drawing.



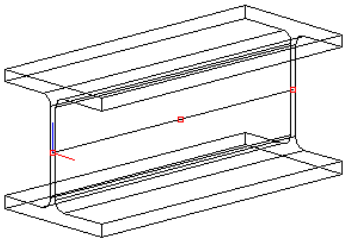
10.5. Object snap settings

While making a drawing the user may position the cursor at snap points of objects (profiles or plates). The options which enable placing the cursor at selected points may be activated in the ***Snap Mode*** dialog box shown in the figure below. The option is available from:

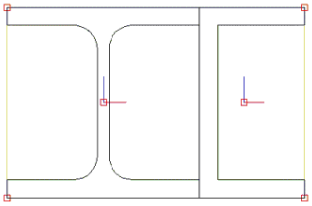
- the menu by selecting the *Steel / Workspace / Object snap settings* option
- the toolbar by pressing the icon
- the command line: RBCS_PSNAP.



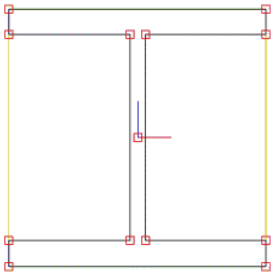
The above dialog box enables the user to determine the following snap points at which the cursor may be positioned:



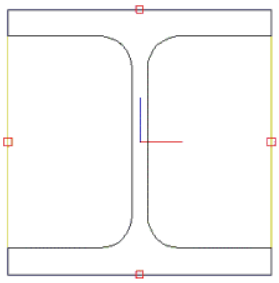
axis - snap point:
plate - plate center
profile - three snap points on the bar axis: beginning, middle and end



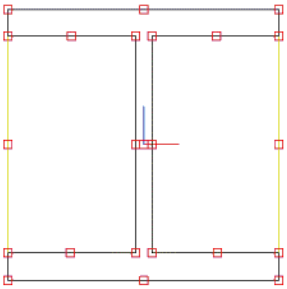
outer corners - snap points are the corners of the outer contour circumscribed on a cross section



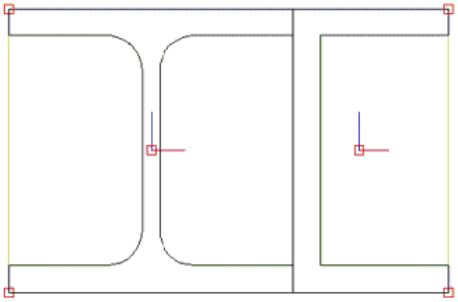
inner corners - snap points are the inner corners of a section as shown in the drawing



outer midpoints - snap points are the midpoints of outer contour sides of a section.



inner midpoints - snap points are the midpoints of section inner sides.



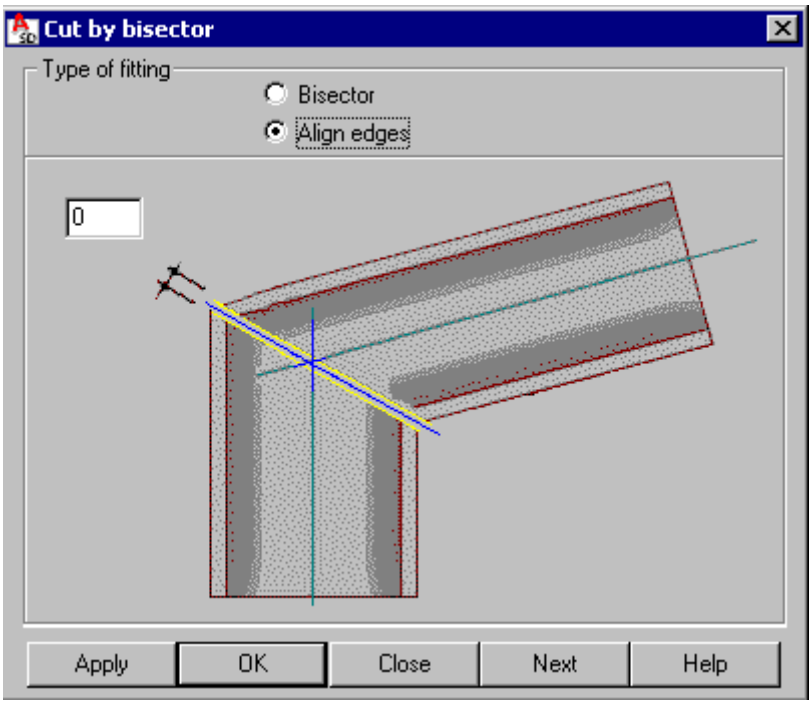
composite sections - the option is intended for determining snap points of composite sections; in this mode, snap points are specified for each part of a composite section.

In addition, the *Auto* option is available; if it is switched on, then snap points are positioned on the axis.

10.6.Cut by bisector

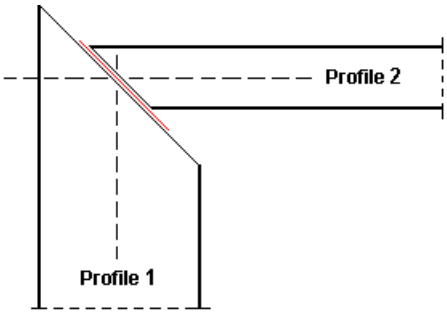
The option enables the operation of fitting two profiles to each other using a bisector. The option is accessible from:

- the menu by selecting the option *Steel / Machining / Cut by bisector*
- the toolbar by pressing the icon 
- the command line: RBCS_CUTBISEC.

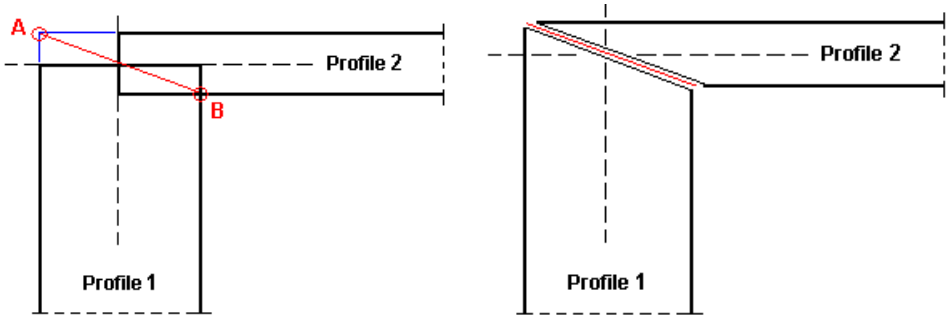


The cutting operation may be performed:

- as cutting with a bisector; the result of this option is a cut as shown in the drawing below.



- as cutting with alignment of profile edges (the cutting line goes between the points A and B); the result of this option is a cut as shown in the drawing below.



For the both types of cutting it is possible to define a distance between the cut profiles.

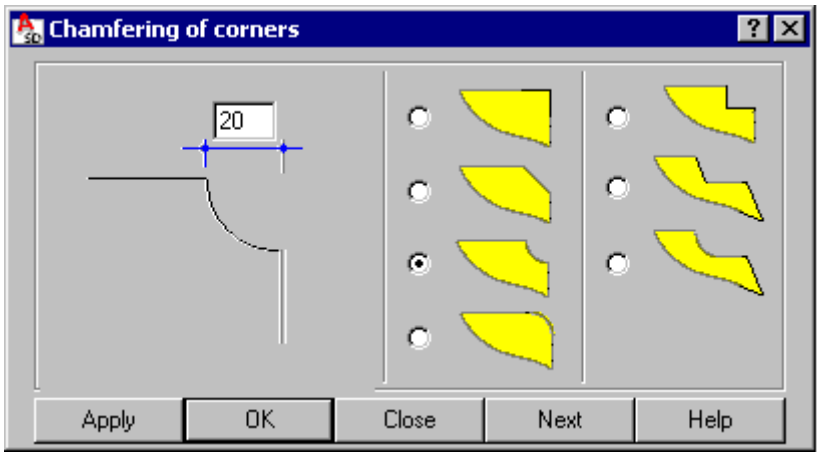
On pressing the **Apply** button parameters in selected objects are modified. Pressing the **OK** button performs the defined operation of cutting the profiles based on the adopted parameters and closes the dialog box.
After pressing the **Close** button, the defined operation is performed without saving the changes made recently. Cutting of successive profiles starts on pressing the **Next** button.

10.7.Chamfering of corners - plates

The option enables the user to perform the operation of chamfering plate corners. The option is available from:

- the menu by selecting the option: *Steel / Machining / Chamfer plate corner*
- the toolbar by pressing the icon
- the command line: RBCS_CHAMFCORN.

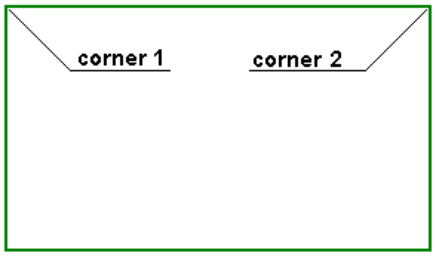
After activating the option, indicating a plate and the plate corner, the dialog box shown in the drawing below is displayed on the screen.



Once the chamfer type is chosen, chamfer dimensions should be defined (the number of available dimensions depends on the selected chamfer type). The chamfer is an active object, which means that it may be edited later on (change of chamfer type, modification of parameter values); deletion of a plate chamfer is also possible.
Pressing the **Apply** button modifies chamfer parameters. Pressing the **OK** button runs the defined chamfer operation based on the parameters adopted and closes the dialog box.
After pressing the **Close** button the program carries out the defined operation without saving the changes made recently. Pressing the **Next** button starts chamfering of the next plate corner.

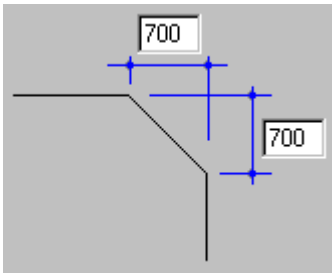
10.8.Example of chamfering plate corners

To start chamfering corners of the plate illustrated in the drawing below, follow the steps below:



- select the menu option *Steel / Machining /Chamfer plate corner* or press the icon
- indicate the plate whose corners will be chamfered
- indicated corners of the selected plate which should be chamfered (in the drawing above these are corner 1 and corner 2)
- in the ***Chamfering of corners*** dialog box determine the parameters as follows:

  Chamfering type:
  adopt the chamfer dimensions as shown in the drawing below

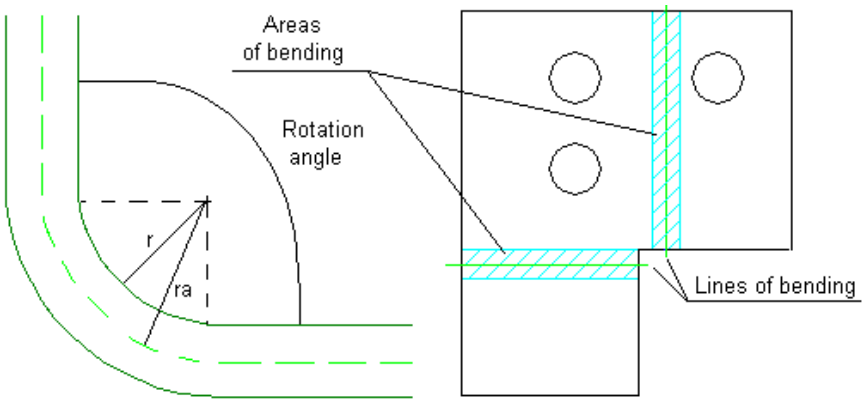


- press the **Apply** button in the ***Chamfering of corners*** dialog box; the plate after performing the operation of chamfering of corners is illustrated in the drawing below.

10.9.Bending of plates

A bent plate is an element made by bending a standard plate defined in a model. Bending of plates is a reversible operation (it is possible to remove a bend made in a selected plate).
After performing the operation of bending, a plate becomes a bent plate in the program.

A bend in a plate may be defined by giving the information as follows (see the drawing below):
- bending line
- angle of bending
- reference surface; it determines whether a radius is defined on the internal surface of the plate (the r radius) or on the middle surface of the plate (the ra radius)
- side of the plate to be bent.



NOTE:
Areas of bending may not overlap.

While defining a bend in a plate, the following commands will appear in the command line:
Choose method [by2Points/byLine]:
Specify start point:
Specify end point:
or
Select line:
Location of bending line [Center/Edge]
Specify internal radius or [Axial]:
Specify angle:
Pick side to be wrapped:

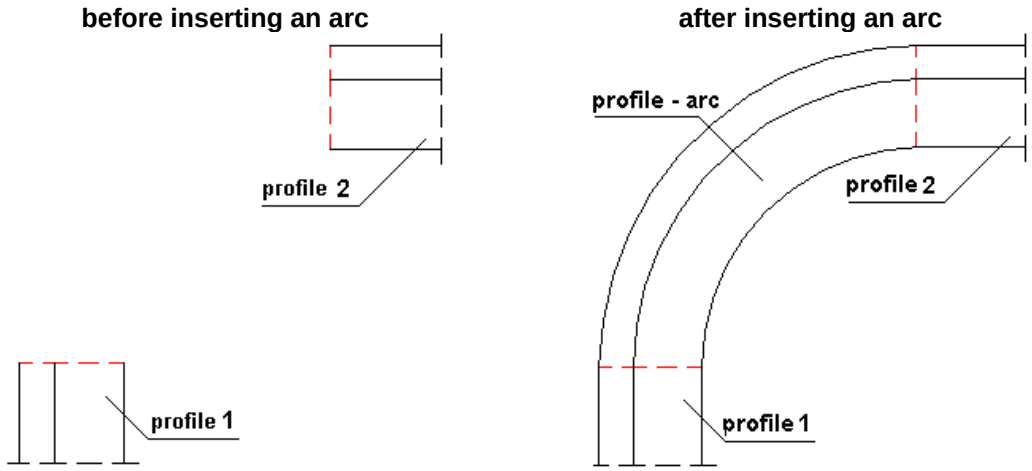
10.10. Insert arc - profiles

The option enables connection - by means of an arc element – of two profiles not touching each other, lying in the same plane. The option is accessible from:

- menu by selecting the option: *Steel / Machining / Insert arc*
- toolbar by pressing the icon
- command line: RBCS_BENDCORNER.

To insert an arc between two profiles (see the drawing below), the user should:

- select the *Steel / Machining / Insert arc* option or press the icon
- indicate two profiles to be connected by an arc
- specify a radius of the inserted arc
- press the **Enter** key.



10.11. Copy connection

The option enables copying all types of connections defined in ***AutoCAD Structural Detailing***. Copying comprises all the elements of a connection. To define a connection, the user should follow the steps below:

1. select the command: Steel / *Machining / Copy connection* or press the icon
2. indicate the connection to be copied
3. indicate the profiles to which the selected connection will be ascribed (if all profiles in the structure model are indicated, then the program will find all the profiles which can be ascribed the selected connection)
4. press the **Enter** key.

Take note that a connection will be copied if the following conditions concerned with the profiles in the copied connection and the profiles where to the connection will be copied, are satisfied:

- 'target' profiles are identical to those in the indicated connection
- mutual orientation of profiles is identical to that in the indicated connection.

NOTE:
If a connection involves machining operations that have been defined 'manually" by the user, they are deleted from the elements of this connection when regenerating this connection.

10.12. Collision detection

The option enables finding structure elements that overlap. To detect a collision of elements in the structure model, the user should follow the steps below:

1. select the command: *Steel / Collision detection* or press the icon
2. indicate elements of the structure model to be verified (if all elements in the structure model are indicated, then the program will find all the collisions of structure component elements)
3. press the **Enter** key.

Once verification of a structure model is completed, the program displays in the command line the number of collisions found between structure elements (profiles, plates, bolts, etc.). Collision sites are marked in a structure model.
The command line shows a number of collisions found between structure elements. It is possible to save collision tests; they are saved in the 'RBCS_Collision_Detection' layer and represented by AutoCAD ® 3D Solid objects.

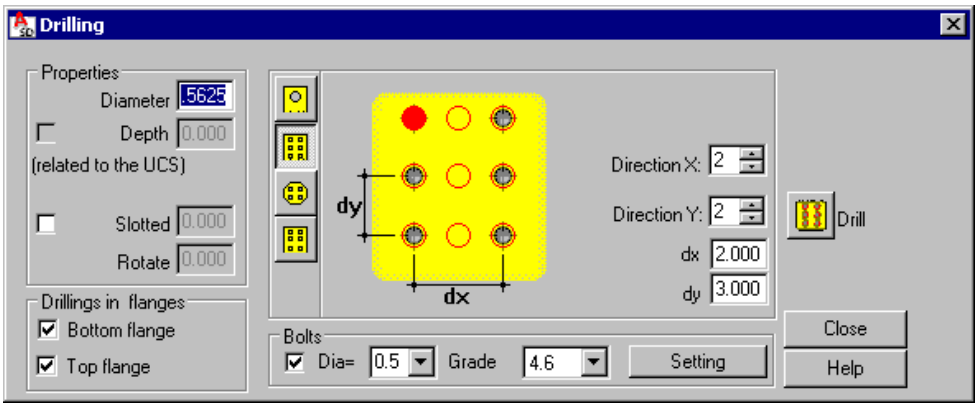
11. DRILLS/BOLTS/WELDS

11.1.Drills

The option allows defining bolt openings. The option is available from:

- the menu by selecting the *Steel / Machining / Drill\Bolt* option
- the toolbar by pressing the icon
- the command line: RBCS_DRILL.

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



The ***Drilling*** dialog box may be divided into three parts:

- in the left part of the dialog box a user may define basic information concerning bolt openings:
  after switching on the *Depth* option, an edit field becomes accessible in which a user may determine the drilling depth; if the option is switched off, then an opening is drilled through a whole element
  once the *Slotted* option is turned on, the defined opening will be slot-shaped; then two edit fields become accessible in which a user may determine: length of the opening rectilinear part and opening rotation angle, respectively; if the *Slotted* option is turned off, the opening will be circular
  after switching on the *Bottom flange* or *Top flange* option, drillings will be positioned in the bottom/top flange of a member section
- in the middle part of the dialog box a user may select one of the possible modes of defining bolt openings / drillings (the selected mode is presented in the drawing):
  - single bolt opening
  - rectangular arrangement of openings / drillings with a regular spacing of openings in both directions - when this opening type is selected, the following parameters need to be determined:
  - number of rows and columns of openings / drillings (direction X and direction Y)
  - distances between rows and columns of openings / drillings (dx and dy)
  - polar arrangement of openings / drillings (openings will be defined on an arc of a circle)
  - when this opening type is selected, the following parameters need to be determined:
  - number of openings / drillings
  - radius of a circle on which the centers of openings / drillings will be positioned
  - angle between the successive openings/bolts; if the *Auto fill* option is turned on, then openings will be arranged along the entire circumference length and the angle will be determined automatically
  - rectangular arrangement of openings / drillings with an irregular spacing of openings - when this opening type is selected, the following parameters need to be determined:

- number of rows and columns of openings / drillings (direction X and direction Y)
- distances between successive openings in rows and columns of openings / drillings (dx and dy), e.g.: 50 75 50 100

below the *Bolts* option is located, if it is switched off, then bolts will not be placed in the openings (all the edit fields are inaccessible); if the option is switched on, then bolts will be placed in the openings and the fields for defining bolt parameters: diameter and class, are accessible; moreover, the **Setting** button is available which when pressed, opens the Bolt Settings dialog box

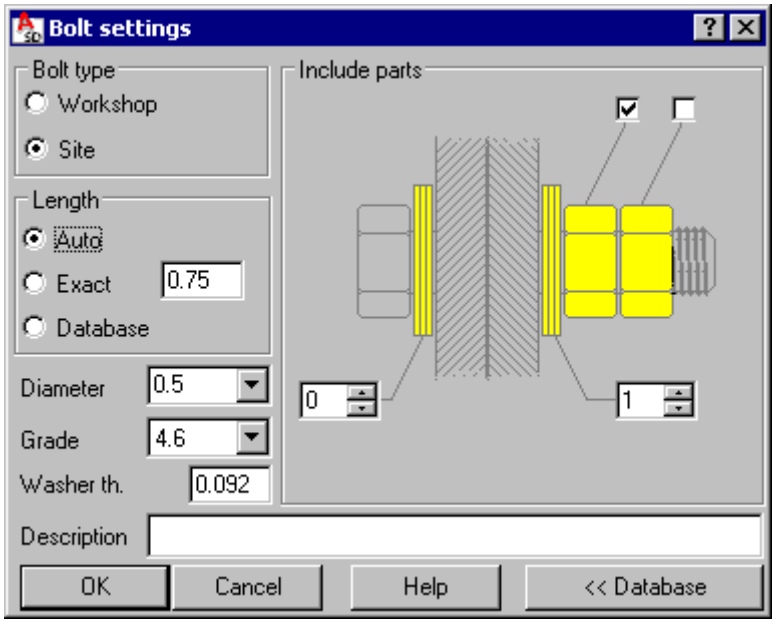
- the right part of the dialog box contains the following icon allowing definition of openings / drillings:

 - indicates objects in which the openings are to be made and drills bolt openings based on parameters defined in the dialog box.

The dialog box opens showing the opening/bolt parameters defined recently.

11.2. Bolt settings

The ***Bolt settings*** dialog box presented in the drawing below opens after pressing the **Setting** button in the Drilling dialog box.



The left part of the dialog box contains basic information concerning applied bolts:

- bolt diameter
- bolt grade
- bolt type (site, workshop)
- washer thickness; in ***AutoCAD Structural Detailing***, bolts are generated with washers (NOTE: the washer thickness is taken into account while calculating the bolt length)
- bolt length; the length may be:
  - *Auto* option: determined automatically
  - *Exact* option: specified directly by the user
  - *Database* option: selected or calculated based on data in a chosen bolt database (see: Principles of calculation of the bolt length)
- additional description of a bolt.

There is the *Include parts* field provided in the right part of the dialog box. If an option is turned on in this field, it results in adding an appropriate part, such as:

- one or two nuts
- washers: head washers or nut washers.

When pressed, the **Database** button opens the bottom panel of the ***Bolt settings*** dialog box. It comprises a table with data regarding bolts, nuts and washers available in the selected bolt folder.

11.3.Principles of calculation of the bolt length

If in the ***Bolt settings*** dialog box the *Database* option has been selected, then a bolt of the appropriate length is automatically selected from an available bolt database.

The following data is needed to determine a bolt length:

1. sum of thicknesses of all the elements connected with each other
2. parameters of bolts:
   - bolt diameter
   - bolt type connected with a selected bolt database
3. parameters of washers:
   - number of washers
   - washer type connected with a selected washer database
4. parameters of nuts:
   - number of nuts
   - nut type connected with a selected nut database

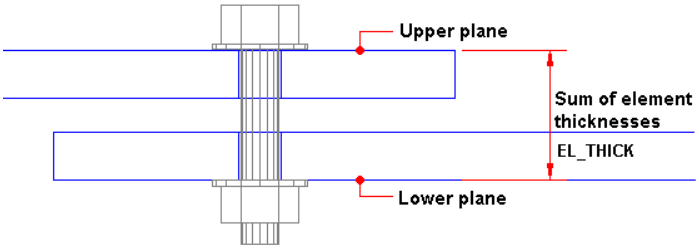
If the above information is known, i.e.

- there is a database including a table of prestressed shank lengths
- databases of nuts and washers are chosen
- there is information about a recommended number of nuts and washers,

then a bolt length is selected directly from the database including a table of prestressed shank lengths.

If not all the conditions mentioned above are fulfilled (e.g. there is no database including information about prestressed shank lengths), a bolt length is calculated in the program according to the description below:

1. calculations of thicknesses of connected elements EL_THICK



This thickness is a distance between the upper plane of the first connected element and the lower plane of the last connected element. The distance is measured at the point where the bolt axis intersects the connected elements.

2. calculations of the minimum length of the bolt shank B_SHANK_LENGTH

Once thickesses of all washers and nuts are added to thicknesses of all connected elements, the value obtained is referred to as the minimum length of the bolt shank.

B_SHANK_LENGTH = EL_THICK + nw * thw + nn* thn

where:

nw - number of washers

thw - washer thickness (from a database)

nn - number of nuts

thn - nut thickness (from a database).

3. searching of a bolt of the required length of the shank and the thread in a selected database; the two conditions presented below have to be fulfilled:

BSld – B_SHANK_LENGTH ³ 0.5 mm

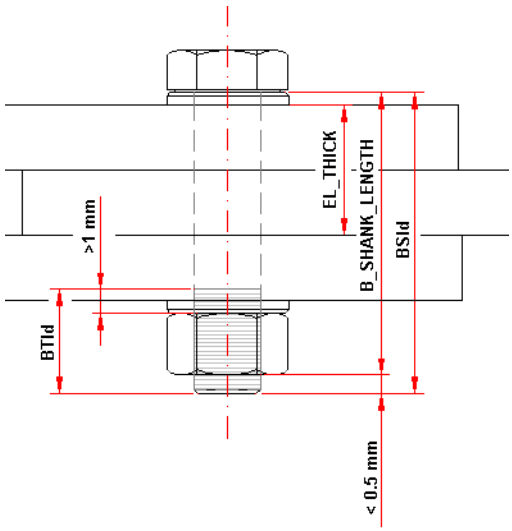
(BSld – nw * thw – EL_THICK) - BTld ≤ - 1.0 mm

where:

BSld - length of the bolt shank (from a database)

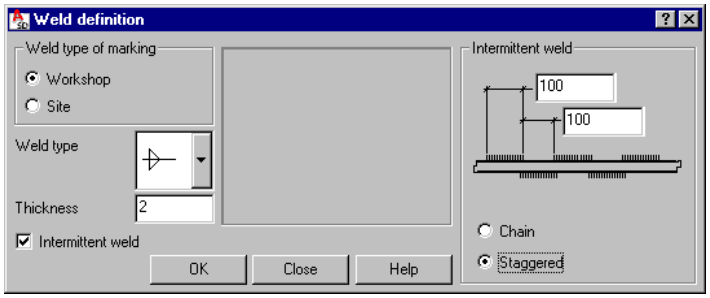
BTld - length of the bolt thread (from a database)

The bolt list presented in the lower part of the ***Bolt settings*** dialog box includes all bolts of a given diameter from a database. The bolt found is highlighted on this list.



11.4.Weld definition

The ***Weld definition*** dialog box shown in the below-presented drawing is used to define weld parameters.



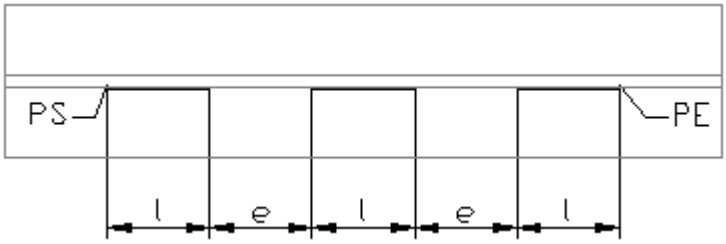
Once the weld type of marking (workshop or site) is selected, the user may determine parameters of the defined weld:

- weld type
- weld thickness (calculated automatically)
- dimension(s) for edge preparation are shown in a schematic drawing - the number of parameters depends on a weld type.

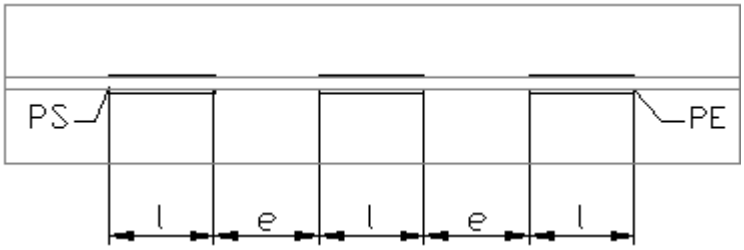
Switching on the *Intermittent weld* option allows definition of an intermittent fillet weld (composed of many segments of single wleds). The drawings below include designations as follows:
PE - beginning of a weld
PS - end of a weld
l - length of weld segments
e - spacing between weld segments.
All segments of a weld are of the same length (l); spacings between segments of welds are also of the same length.

The following types of intermittent welds are available in the program:

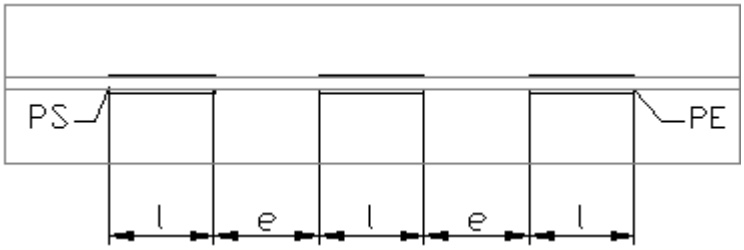
- intermittent weld – one-sided



- intermittent weld – double-sided - chain



- intermittent weld – double-sided - staggered.



12. POSITIONING

12.1. Position - definition and description

The design process in ***AutoCAD Structural Detailing*** may be divided into two primary stages: model definition and preparation of drawing documentation. The intermediate stage constitutes positioning, in other words, assigning - to structure elements - the "position" designations, which are used to identify elements in the documentation. Assigning a position is necessary in order to prepare a drawing of a given element.
An additional objective of positioning is to limit a number of drawings in such a manner that identical structure parts are presented in one drawing. Three types of objects that may be positioned, have been distinguished in ***AutoCAD Structural Detailing***:

- Single parts
- Assemblies (only automatic positioning)
- Groups.

The position is presented in the form of an element (element group) label. The position syntax consists of two separate parts:
Prefix + number, where

- *Prefix* is a short, editable text, e.g. *pos*, that may be linked to the element's family (single part, groups, etc.); this part of a name may be disregarded
- *Number* is saved in the numeric or alphanumeric format (it may be ascribed automatically or manually); As a result of adopting a position number, the classification is set properly on the list.

Positioning levels
Positions may be assigned independently on two positioning levels:

- for single parts
- for assemblies.

The process of positioning is carried out separately for each of the levels listed.

The program allows two methods of assigning a position:

- manual positioning (manual definition or modification of positions for selected elements – single parts or groups)
- automatic positioning (automatic definition of a position for the entire model or its selected part).

Each of the positioning levels listed is connected with the type of drawings (the template used for automatic drawing generation): single part drawing, workshop drawing and site drawing. Beneath there are the elements presented that may be chosen for individual positioning levels:

Single part	Single parts: profiles, plates, connectors
Assembly	Single parts, groups
Group	Groups

There is also the *Remove position* option provided in the program. It is available from:

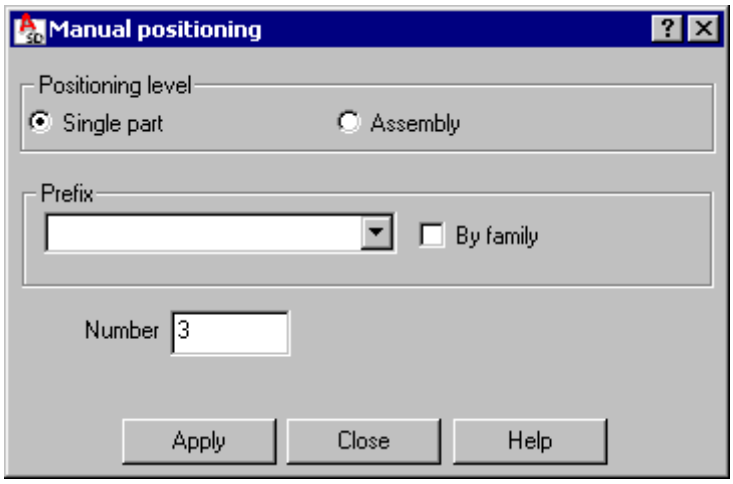
- the menu by selecting the *Steel* / *Positions* / *Remove position* option
- the toolbar by pressing the  icon
- the command line: RBCS_REMPOS.

To remove the positions assigned to elements, the user should select the elements and activate the *Remove position* option. Positions will be removed from all the selected elements.

12.2. Manual positioning

The option enables assigning the position manually. It is available from:

- the menu by selecting *Steel / Positions / Assign position manually* option
- the toolbar by pressing the icon
- the command line: RBCS_MANUALPOS.



In order to perform the operation of assigning the position manually, the user should:

- select an element or several identical elements
- in the above dialog box - determine the following: positioning level, position label (prefix, number and comment - see the position description)
- press the **Apply** button.

NOTE:
While assigning a position manually only basic criteria of element recognition are considered; user-defined criteria are available only in the case of automatic positioning.

A position may also be assigned (modified) any time while working in ***AutoCAD Structural Detailing*** by means of the *Object properties* option (the ***Inspector*** dialog box).

12.3. Automatic positioning

The option enables assigning positions automatically. It is available from:

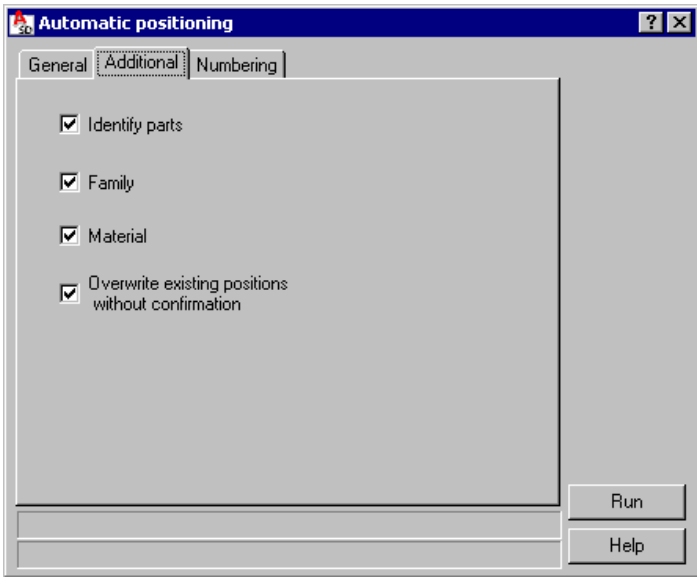
- the menu by selecting the *Steel / Positions / Run automatic positioning* option
- the toolbar by pressing the icon
- the command line: RBCS_AUTOPOS.

The operation of assigning the position automatically may be performed for a whole structure model or for its selected part. Once structure elements are selected, the ***Automatic positioning*** dialog box is displayed on the screen; the dialog box consists of three tabs:
General
Additional
Numbering.

Pressing the **Run** button activates the operation of assigning the position automatically. Positioning is performed automatically for all selected structure elements. All the positions assigned during one positioning operation are given identical prefix and comment.

12.4. Additional

The dialog box assumes the form shown in the drawing below after selecting the *Additional* tab in the Automatic positioning dialog box.



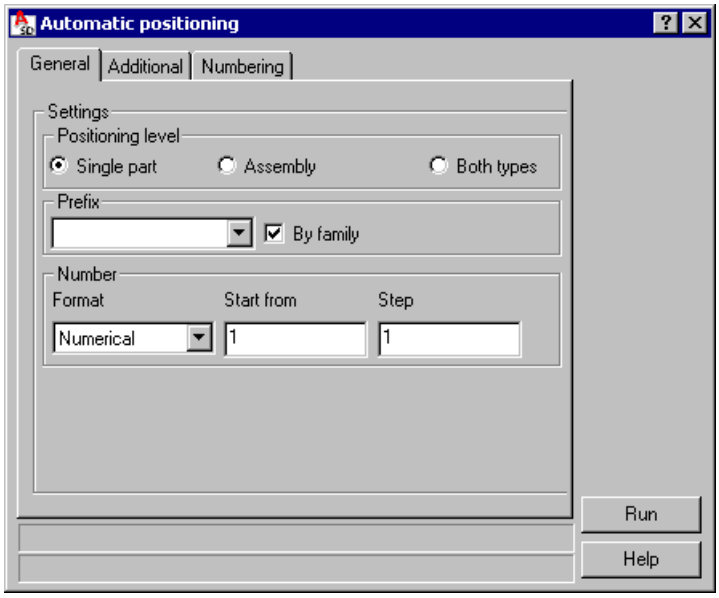
The above dialog box contains the following options:

- *Identify parts*
  if this option is switched on (this is the default option setting), the identity of a structure part is checked; if two identical parts are found, then the same position is assigned to them
  if this option is switched off, each structure part is assigned a successive position
- *Family*
  if this option is switched on, then families are considered while checking the identity of a structure part; it means that if structure elements belong to different families, then these structure elements will be ascribed different positions; if the *Identify parts* option is switched off, the *Family* option is inaccessible
- *Material*
  if this option is switched on, then materials are considered while checking the identity of a structure part; it means that if structure elements are assigned different materials, then these structure elements will be ascribed different positions
- *Overwrite existing positions without confirmation*
  if this option is switched on, then the structure elements that have been selected and that have already been assigned positions, are automatically ascribed new positions (previous positions are overwritten)
  if this option is switched off and during the automatic positioning an element with the position already assigned is found, then in the command line the message is displayed asking if the position for the element is to be overwritten: there are three possible answers:
  No - the position will not be changed
  Yes - the position will be changed to a new one
  Yes for all - the position will be changed for all the successive elements that have already been assigned positions.

If the *Identify parts* option is turned on and several identical parts have been found during positioning, then a number ascribed to the element is the number of the first identical part found during positioning. All the parts identical to the element that has already been assigned a position, are ascribed the same position as the first element found.

12.5.General

The dialog box assumes the form shown in the drawing below, once the *General* tab is selected in the Automatic positioning dialog box.

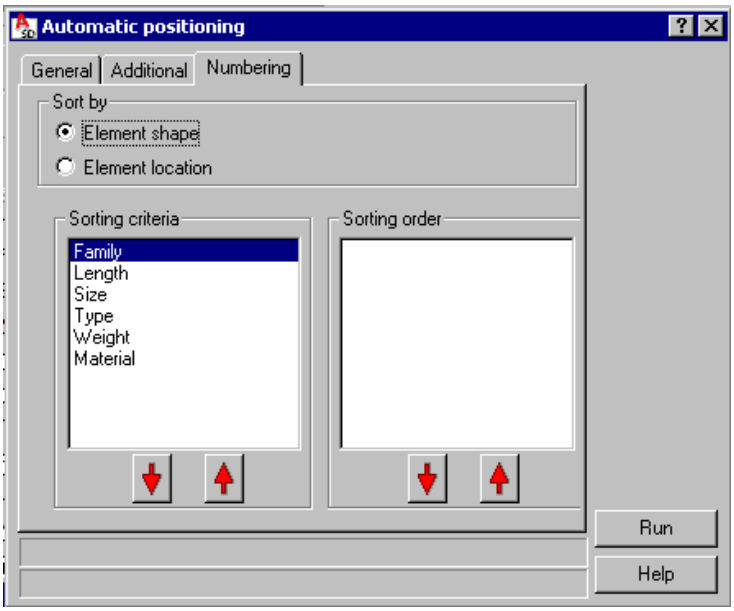


The following options are provided in the *Settings* field:

- *Positioning level* - if a given option is indicated, then the selected positioning level is assigned to all the elements chosen for the positioning operation;
  once the *Single part* option is selected, then groups are not considered in the positioning process
  once the *Assembly* option is selected, positions are assigned both to single elements and to groups
  once the *Group* option is selected, single parts are not considered in the positioning process
- *Prefix* - the field (list) allowing the user to specify a character string (it is allowable to disregard a prefix) that will be ascribed as a prefix to all positions; a prefix may be taken from a family
- *Number* - determines the format of numbering the positions ascribed to selected structure elements; the following options are available in this field:
  Format - selection of the format for a position number (numbers or letters)
  Start - depending on a selected format, the field allows defining (natural) numbers or alphabet letters; in this field the beginning value for the position numbering may be determined - successive positions will be numbered depending on a value provided in the *Step* field
  if the alphanumeric format has been chosen, then the consecutive alphabet letters are ascribed to consecutive positions (the alphabet contains 26 letters); if the 26th letter (Z letter) has already been applied, position numbering will assume the following form: AA, AB, AC, etc.
  Step - the field for defining a value of the position numbering increment.

12.6.Numbering

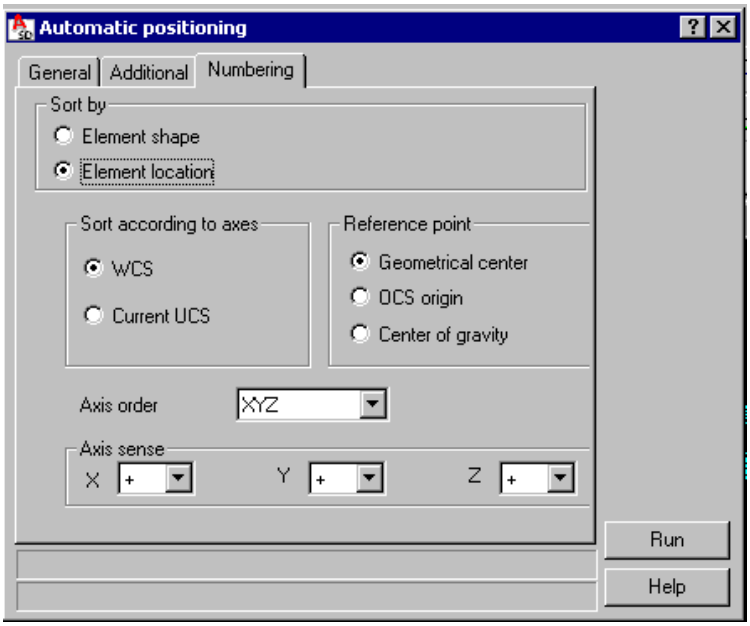
Once the *Numbering* tab is selected in the Automatic positioning dialog box, the contents of the dialog box depends on the option selected in the *Sort by* field.
If the *Sort by element shape* option is chosen, then the dialog box assumes the form presented in the drawing below.



In this mode position numbers are sorted with the element geometry taken into account. The numbering order is determined based on selected properties of the element shape:
The dialog box above includes two panels:

- left panel containing available sorting criteria (6 element properties, that may be considered while defining the numbering order, are presented on the list); highlighting a line in the left panel and pressing the ↑↓ buttons results in moving the selected property up or down the list; in this manner the user may obtain the required sorting order
- right panel presents the (ascending) order of quantities defined in the program for a selected sorting criterion (e.g. all profile types); highlighting a line in the right panel and pressing the ↑↓ buttons results in moving the selected property up or down the list; in this manner the user may obtain the required sorting order.

If the *Sort by element location* option is chosen, then the dialog box assumes the form presented in the drawing below.



In this mode position numbers are sorted considering the element location in a structure model. Successive numbers are ascribed based on the position of the element reference point in the coordinate system. The above dialog box allows determining the following parameters:

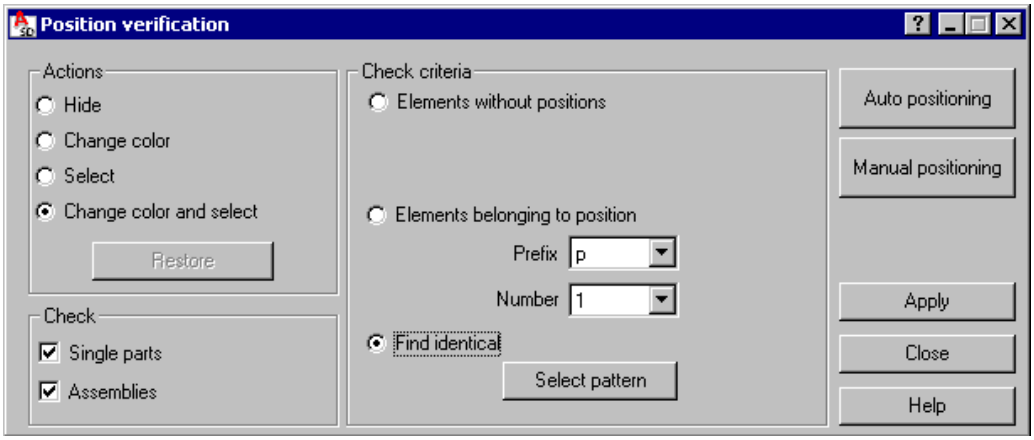
- in the *Sort according to axes* field - the user chooses whether the location of positioned elements is to be verified with respect to the global coordinate system (WCS) or with respect to the current user local coordinate system (UCS)
- in the *Reference point* field - the user determines the point which identifies the point position in space
- from the *Axis order* list the user may select the method of searching the parts in the selected coordinate system
- in the *Axis sense* field - the user determines if the numbering increases according to the axis sense (+) or if the numbering increasing in the direction opposite to the axis sense (-) is assumed.

12.7.Position verification

The option is intended for controlling designations of positions ascribed to elements in the process of manual or automatic positioning.
Positioning is verified through searching for positions that do not fulfill (or do fulfill) selected conditions of positioning. The option is available:

- from the menu by selecting the option: *Steel / Positions / Verify positions*
- from the toolbar pressing the  icon
- from the command line: RBCS_VERIFYPOS.

Verification of positions is performed for the entire structure model. After activating the option, the ***Position verification*** dialog box is displayed on the screen.



The *Actions* field includes several options which allow performing operations on elements found during position verification (hide, change color / select). The effect of an action is remembered until the **Restore** button is pressed or until the dialog box is closed and regeneration is carried out.
In the *Check* field the user may choose elements (single parts or assemblies) to be taken into account in structure verification.
In the *Check criteria* field there are options provided which enable indicating the elements to be searched in the process of verification:

- *Elements without positions* – if this option is selected, then the program will be searching for all the elements to whom no position has been assigned
- *Elements belonging to position* - if this option is selected, then the program will be searching for the elements that contain the specified prefix and position number
- *Find identical* - if this option is selected, then the program will be searching for the elements in conformity with the pattern object selected by the user (group or single part); the pattern object may be indicated on the screen after pressing the **Select pattern** button.

Pressing the **Apply** button runs verification (the options in the *Check criteria* field); after verification is carried out on all objects that satisfy the selected criterion, an action chosen in the *Actions* field may be performed. In addition, the command line provides information about the number of elements found that fulfill the defined criterion, e.g.:
Position verification
Identical elements: 10 found.

In the top right corner of the dialog box there are two buttons which enable the user to run positioning of elements found and selected during verification, if the *Select* option has been chosen in the *Actions* field:

- **Manual positioning** – pressing this button enables performing manual positioning for elements found in the verification process
- **Auto positioning** - pressing this button enables performing automatic positioning for elements found in the verification process.

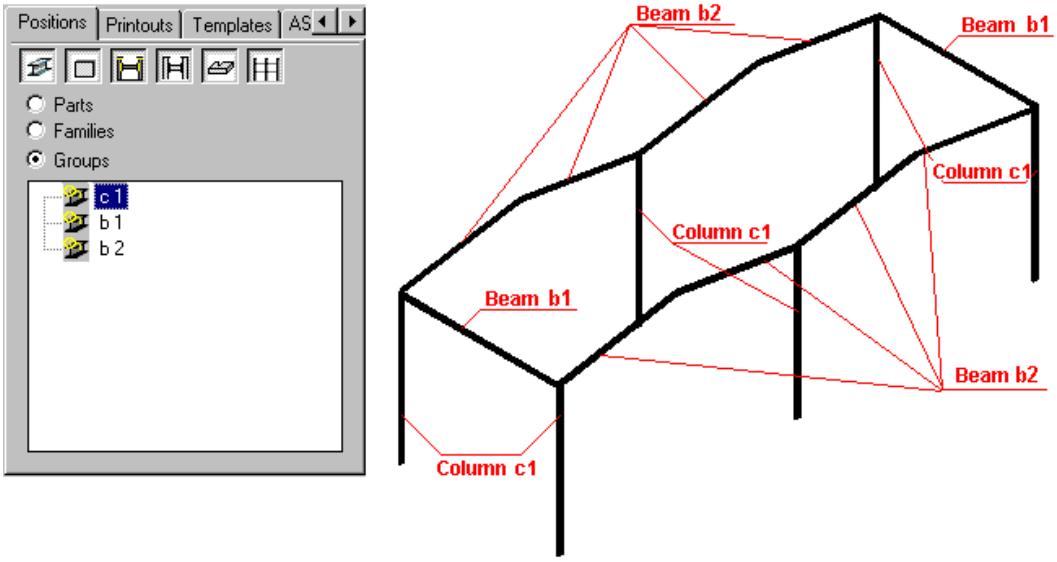
12.8.Example of positioning and automatic printout

Positioning will be illustrated on the example of a simple 3D frame generated with the use of the Multi-span frame macro available in ***AutoCAD Structural Detailing - Steel***. To define a multi-span frame, follow the steps below:

- select the menu command: *Steel* / *Parametric structures* / *Multi-span frame* or press the icon 
- determine the frame insertion point
- in the ***Multispan Hall*** dialog box, on the *Geometry* tab specify the following parameters:
  number of spans = 2
  number of frames = 2
  frame spacing = 8000 mm
  switch on the options: *Rafters* and *External spandrel beams*
- adopt any user-defined dimensions of the frame in the lower part of the dialog box
- move on to the *Profiles* tab and choose the following parameters for component elements of the frame:
  columns - profiles: *C 10x15.3*, family: *Column*
  rafters: profiles: *C 10x15.3*, family: *Beam*
  spandrel beams: profiles: *C 10x15.3*, family: *Beam*
- press the **OK** button at the bottom of the dialog box; the program will generate a model of the 3D frame.

The profiles generated in the created 3D frame have been added to the ***Object inspector*** dialog box on the *Model* tab; to assign positions to the elements of the created frame, do as follows:

- in the ***Object inspector*** dialog box on the *Model* tab select all the profiles (the profiles will be highlighted)
- while indicating - with the mouse cursor - the profiles located in the ***Object inspector*** dialog box, press the right mouse button; select the *Auto positioning* option from the context menu
- in the ***Automatic positioning*** dialog box choose the parameters as follows:
  on the *General* tab
  Positioning level: *single part*
  Prefix: the *By family* option switched on
  Number: numerical format, start from 1, step = 1
  on the *Additional* tab
  all the options switched on
  on the *Numbering* tab
  Sort by: *element shape*
  Sorting criteria: *family*
  Sorting order: column, beam
- press the **Run** button; all the elements of the 3D frame will be assigned positions according to the parameters determined in the ***Automatic positioning*** dialog box; since some frame elements are identical, on the *Positions* tab in the ***Object inspector*** dialog box three positions will be created: columns c1, beams b1 and beams b2; the drawing below illustrates the created positions and location of the structure elements that make up individual positions



- move on to the *Positions* tab in the ***Object inspector*** dialog box
- in the ***Object inspector*** dialog box, on the *Positions* tab select all the created positions (the positions will be highlighted)
- while indicating - with the mouse cursor - the positions located in the ***Object inspector*** dialog box, press the right mouse button; select the *Automatic Drawings* option from the context menu
- in the ***Automatic drawing generation*** dialog box choose the parameters as follows:
  on the *Templates* tab
  Column: *Profile 1:10*
  Beam: *Profile 1:10*
  on the *Formats and scales* tab
  Part type: *single profiles*
  Switch on the *Automatic scale* option
  Format: A4 ASD
  Arrange views: accept default values of the parameters
  on the *Options* tab
  Names of printouts: *Names consistent with position name*
  Start printout numbers: 1
  All the remaining options switched off
  on the *Bill of materials* tab
  the *Add table* option switched on
  description: Standard
  position of the bill of materials: top left corner of the printout
- press the **Generate** button; for all the positions (b1, b2 and c1) detailed drawings will be generated; the program will create additional layouts (b1, b2 and c1) which will present drawings of selected positions; in the ***Object Inspector*** dialog box, on the *Positions* tab documents (projections of positions: views and sections) for each position will be added.

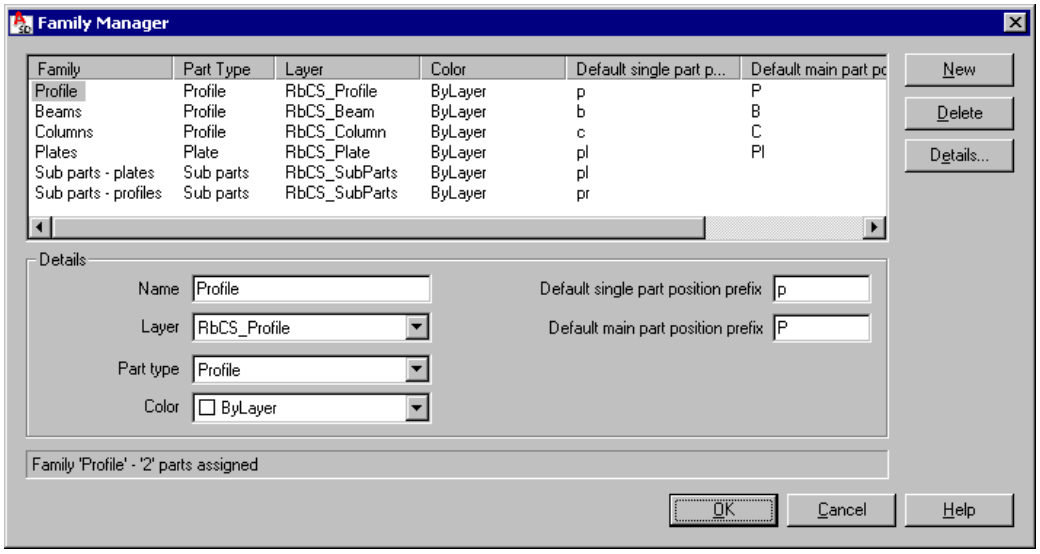
13. FAMILY MANAGER

13.1.Family manager

The option enables adding (removing) families to a project as well as modifying family properties. The option is available from:

- the menu by selecting the *Steel / Family manager* option
- the toolbar by pressing the icon
- the command line: RBCS_FAMILY.

Once the *Family manager* option is selected, the dialog box presented in the drawing below is displayed on the screen.



Basic rules that apply while defining/modifying families in ***AutoCAD Structural Detailing*** are presented below:

- the family is an attribute that may be assigned to a profile, bent profile, plate or user part defined in a structure model; thus, it is a tool that facilitates management of structure model parts
- the main goal of defining families is to collect structure elements into certain sets and put them in order so that they are recognizable during filtering, positioning, etc.
- any number of families may be defined in a project; a set of families may be saved in a project template (in a DWT file)
- the family may be assigned to each of structure parts (profile, bent profile, plate or user part) during definition of this structure element (dialog boxes contain the lists that enable selection of a family) or may be ascribed after defining the structure elements listed as a property of that structure element.

NOTE:
The family may be assigned only to a single part (for groups or assemblies the family is understood as a family assigned to the main part).

The above dialog box allows definition of the following family parameters:

- *Name* - the field that allows providing the name identifying a family; a family name cannot repeat in a project

- *Layer* - default layer on which family elements will be positioned during definition; however, it should be emphasized here, that the connection of the family and layer is not permanent: after definition, an element may be transferred to another layer without losing the family attribute
- *Part Type* - determines the type of structure element for which the group will be defined; the family may be defined for *profiles* (with *bent profiles* included), *plates or subparts* ; it should be stressed here that a family defined for profiles cannot be assigned to plates (and vice versa); subparts created automatically during operations such as a connection definition, are ascribed to a default family (it depends on the option selected in Preferences)
- *Color* - color that will be assigned to parts belonging to the family being defined
- *Default single part position prefix* - field that allows entering a prefix (text) that may be used during automatic positioning of single parts of the family being defined
- *Default main part position prefix* - field that allows entering a prefix (text) that may be used during automatic positioning of main parts of the family being defined.

Defined families are available in:

- the appropriate selection lists provided in the following dialog boxes: Profile, Bent profile and Plate
- in profile or plate properties (families may be modified in the ***Properties*** or ***Inspector*** dialog boxes).

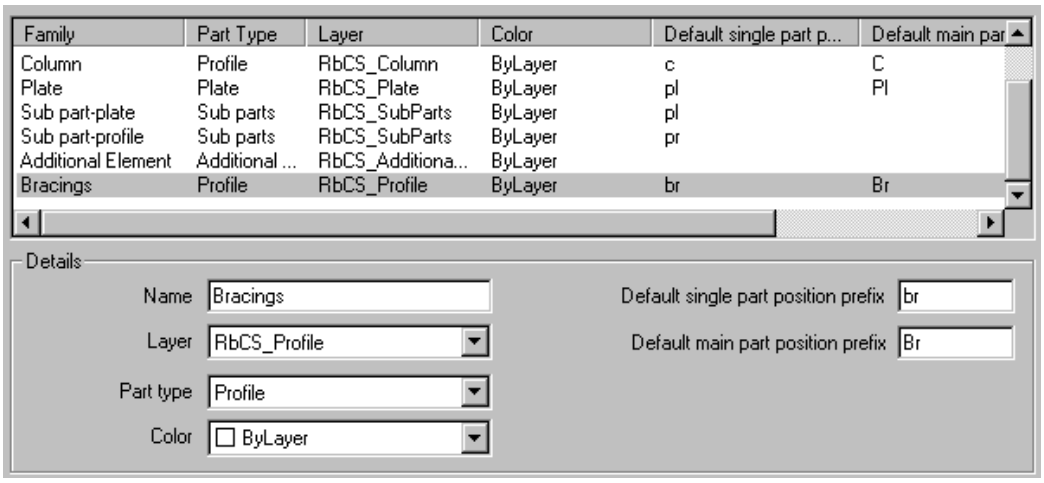
The right part of the dialog box contains the following buttons:

- **New** - pressing this button enables adding a new family to the list of available families; an added family is assigned a default name (the user may change the name of the created family); all the properties are adopted (inherited) from the family recently highlighted on the list of available families (if none of the families has been highlighted, then the properties are adopted (inherited) from the first family on the list)
- **Delete** - pressing this button removes a family selected (highlighted) on the list of available families; it should be added here that all structure elements which were ascribed the removed family, remain in a structure model without the family assigned
- **Details** - pressing this button results in displaying (or hiding) the bottom part of the dialog box provided under the list of available families; the options located in this part of the dialog box enable defining parameters for a selected family or selected families; when several families are selected, the properties are identical for all the selected families.

13.2. Example of definition of a profile family

To define a new profile family, follow the steps below:

- select the menu option *Steel / Family manager* or press the icon 
- in the ***Family Manager*** dialog box press the **New** button; the program will create a new family with the properties of the most-recently-selected (highlighted) family
- in the lower part of the ***Family Manager*** dialog box determine the parameters as follows:
  - name: Bracings
  - layer: RbCS_Profile
  - part type: Profile
  - color: ByLayer
  - default single part position prefix: br
  - default main part position prefix: Br
- the new profile family has been added to the list (see the drawing below)
- press the **OK** button in the ***Family Manager*** dialog box.



The dialog box above also enables edition of parameters of a selected family; after indicating a family on the list of families in the upper part of the dialog box, parameters of the family presented in the lower part of the dialog box should be modified.

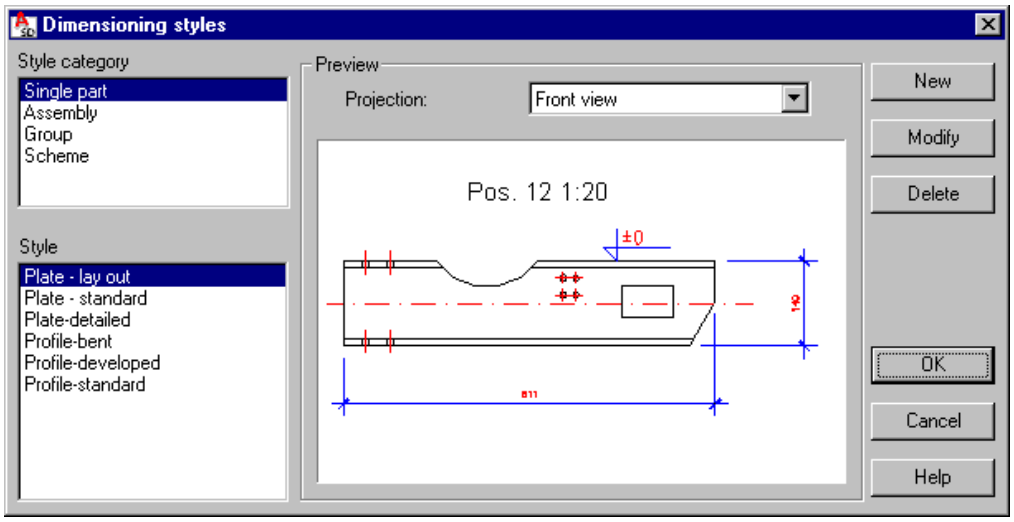
14. DIMENSIONING STYLES

14.1.Dimensioning styles

The option enables defining styles of element dimensioning. The option is available from:

- the menu by selecting the *Steel / Styles / Dimensioning styles* option
- the command line: RBCS_DIM.

Once the *Dimensioning style* option is activated, the dialog box shown in the drawing below appears on the screen.



Dimensioning consists in positioning the elements listed below in a 2D projection of a position:

- geometrical dimensions: linear dimensions of individual elements or elements joined together to create an element chain, arc dimensions, angle dimensions, diameters and radial dimensions
- symbols and names: weld symbols, bolt (opening) symbols, elevation marks, designations of assemblies and groups (profile types, positions), etc.
- comments (additional text): variables describing created views and presented objects (name, scale, etc.), user-defined texts.

Dimensioning is performed automatically based on the user-defined settings; it is also possible to modify created dimension lines and their descriptions. Dimensioning may be conducted within the edition contour. All dimensional elements (dimension lines, text style, ends of dimension lines, etc.) are set in the dimension styles available in the AutoCAD ® program.
The user may define any number of dimensioning styles. It should be remembered, however, that each of the defined styles belongs to one of the categories concerning the type of a drawing position (single part, assembly, group, scheme – a group of elements constituting a structure model).

The dimensioning style defines the following:

- dimension style of the AutoCAD ® program to be applied
- elements and the element projections in which the individual elements should be included
- manner of arranging elements in the drawing.

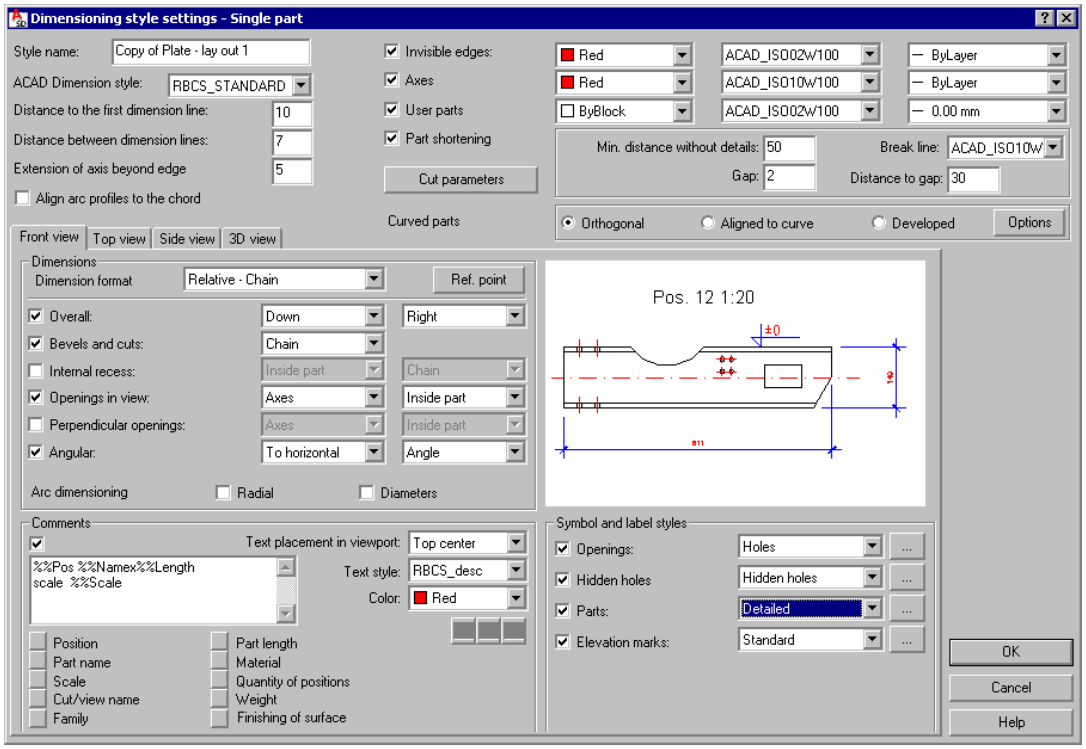
A set of defined styles is saved in a DWG format file or in a template (a DWT file).

In the above dialog box, when one of the types of drawing positions is selected in the *Style category* field, it enables presenting the styles defined for this position type; after highlighting one of the categories provided in the *Style* field, all defined styles for the selected category are displayed. A preview of a selected style is shown in the *Preview* field. For each style there are different settings defined for each of available element views (top view, front view, side view, 3D view).
One of the styles defined for each drawing position type is chosen to be a default style; it means that this style will be suggested as a default one while defining a drawing template. The default type may be changed by highlighting a style within a given category and pressing the **Default** button.
The dialog box above contains also the following buttons (apart from standard ones):

- **New** - pressing this button opens the Dimensioning style settings dialog box where the user may define a new style for a selected category; name of a new style and all its settings are inherited from the style currently selected
- **Modify** - pressing this button opens the Dimensioning style settings dialog box where the user may change settings of a selected style
- **Delete** - pressing this button results in deleting the style currently selected from the list of available styles for a given category.

14.2.Definition/modification of a dimensioning style

The dialog box is used to define a new style or modify an existing dimensioning style.
The ***Dimensioning style settings*** dialog box opens after pressing the **New** or **Modify** button in the Dimensioning styles dialog box.



The contents of the dialog box shown above depends on the category selected in the ***Dimensioning styles*** dialog box; most options included in this dialog box are identical, however, there are considerable differences in the part concerned with geometrical dimensions. The dialog box differs considerably if a dimensioning style of disposition scheme has been chosen.

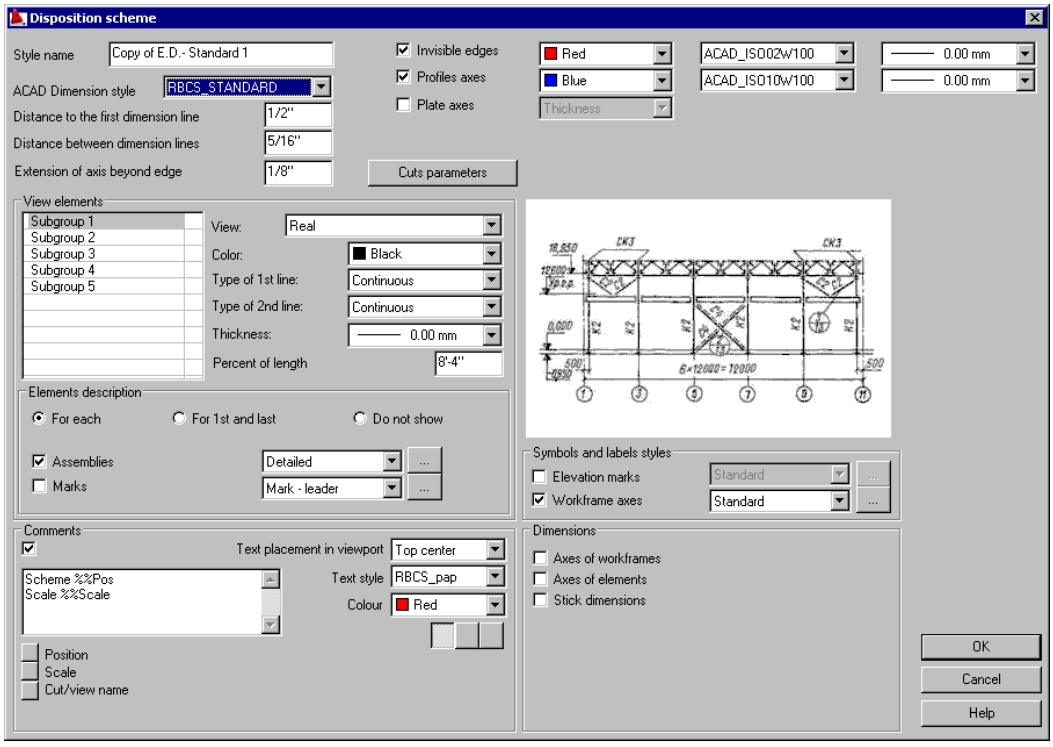
The above dialog box may be divided into the following parts (from the top of the dialog box):

- general part - parameters defined in this part are used in case of all generated views
- part concerned with parameters of dimensioned curved parts
- part concerned with definition of geometrical elements which are to undergo dimensioning - this part consists of four tabs (*Front view, Top view, Side view, 3D view)*; if an option in this part of the dialog box is switched on, then an element selected in the drawing will undergo dimensioning; in the selection fields the user may determine dimensioning parameters
- field which presents a preview of the style currently defined
- part that allows determining additional texts (comments) generated automatically for views
- part that allows defining description styles and symbols - it enables indicating a symbol that is to be created automatically in the selected style; pressing one of the (**...**) buttons opens the dialog box used for defining description styles which determine the manner of presenting a description/symbol in a drawing
- the upper part of the dialog box contains an edit field that allows specifying a style name, whereas the lower part holds the standard buttons **OK**, **Cancel**, **Help**.

14.3. Definition/modification of a dimensioning style - disposition scheme

The dialog box is used to define a new style or to modify an existing dimensioning style for the disposition scheme (a group of elements constituting a structure model).

The ***Dimensioning style settings (Disposition scheme)*** dialog box opens after pressing the **New** or **Modify** button in the Dimensioning styles dialog box (the scheme has to be selected as a style category).



The following parameters can be determined in the top part of the dialog box:

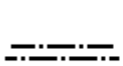
- *ACAD Dimension style* - enables selection of the AutoCAD ® program dimensioning style which will be applied in the dimensioning style currently selected; the list contains all the dimensioning styles defined in a DWG format file

- *Distance to the first dimension line* – an edit field where the user may specify the distance between the edge of a dimensioned part and the first dimension line generated automatically
- *Distance between dimension lines* – an edit field where the user may specify the distance between successive dimension lines
- *Extension of axis beyond edge* – an edit field where the user may specify the length of extension of lines outside the element edges on both sides of the element
- *Invisible edges* - if this option is switched off, then invisible edges of an element are hidden in a final drawing; if this option is switched on, then invisible edges are presented in a drawing - their parameters are set on the selection lists located to the right of the option: line color, line type, line thickness
- *Profile axes* - if this option is switched off, then profile axes are hidden in a final drawing; if this option is switched on, then profile axes are presented in a drawing - their parameters are set on the selection lists located to the right of the option: line color, line type, line thickness; profile axes are presented in all projections - in final drawings axes are extended by 5 mm outside the edge of a dimensioned object

Click the **Cuts parameters** button in the upper part of the dialog to open the Cut parameters dialog where you can specify parameters of hatching the cut and parameters for designating the cut in disposition schemes.

Options in the central part allow choosing parameters of an element view for a disposition scheme (color, type of the first and/or second line, line thickness); the following types of views are available:

- *real* – a real view of each element
- *rectangular box* – a view of each element is represented as a rectangular contour
- *line* - a view of each element is represented as a single line
- *double line* - a view of each element is represented as a double line




- *column line* - a view of each element is represented as a column line


- *truss scheme* – an element view is represented as a truss scheme

- *rectangular box - group* - for all elements of a group (also for elements in subgroups) it is represented as a rectangular contour circumscribed on elements of the group
- *line - group* - represented as the largest central (single) line of the rectangular contour circumscribed on elements of the group
- *double line - group* - represented as the largest central (double) line of the rectangular contour circumscribed on elements of the group
- *column line - group* - represented as the largest central line (column line) of the rectangular contour circumscribed on elements of the group.

Designations can be presented (for assemblies and marks):
- for each element of a scheme
- only for the first and last elements.
It is also possible not do display designations at all.
In addition, styles can be selected for assemblies and marks.

Moreover, a value of shortening (in percent) of an element length can be specified.

The drawings may also include:
- *Stick dimensions* - distances of elements from axes in a scheme (if an element lies on the axis, then a zero dimension is not shown)
- *Level marks* – level symbols, separately for each subgroup of the scheme
- *Axial dimensions* – dimensions from axis to axis of elements of a scheme
- *Level dimensions* – additional dimension lines of elements (on the side of a scheme, near the level marks); a dimension chain from the lowest to the highest level in the scheme is created.

If the *Comment* option is switched on, then automatically, to the views of a dimensioned object a title (name) is added, which may be a text (comprising one or several lines) containing variables or any character string defined by the user. In the bottom part of this field accessible variables are provided - if they are pressed, then a variable is added to the defined comment to the edit field where the cursor is positioned - a whole text of the comment is displayed in the edit field.
A comment may include the following variables:
%%Pos - if this option is turned on, then description of a position is added to a text
%%Scale - if this option is turned on, then a scale is added to a text, e.g. 1:20
%%Vname - if this option is turned on, then an automatic name of a view or section is added to a text

The user may set a comment in a drawing: the following can be determined: text style, text aligning (align right, align left, center) as well as position of text in a view.

In this part of the dialog box the user may select styles for the following descriptions and symbols (designations) presented in final drawings:
- Elevation marks
- Workframe axes.

14.4. Orientation of element views and dimensions in a drawing

To enable definition of position projections, the following terms have been adopted:
- *Front view* - XZ plane of the element local coordinate system
- *Top view* - XY plane of the element local coordinate system
- *Side view* - YZ plane of the element local coordinate system
- *3D view* – isometric view.

The element local coordinate system for a selected position type is defined in the following manner:
- single parts
  Profiles:
  X axis - along the profile axis
  Y axis - section horizontal axes - identically as in the case of profile saving in the database (for most profiles it is the axis of a larger moment of inertia)

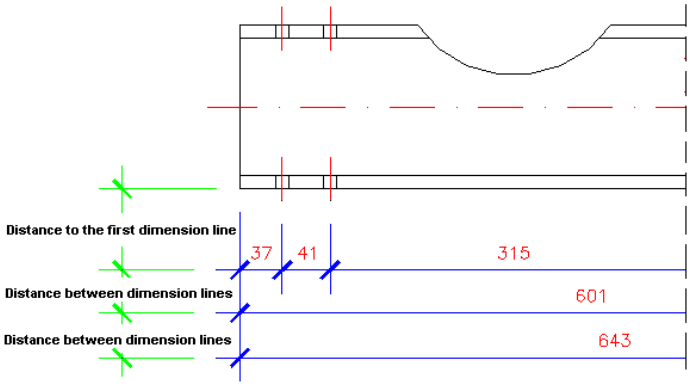
Z axis - section vertical axes - identically as the case of profile saving in the database (for most of the profiles it is the axis of a smaller moment of inertia)

Plates:
rectangular
X axis - along the plate length
Y axis - along the plate width
Z axis - perpendicularly to the plate plane
circular
X axis - along the radius (diameter) determined during plate definition
Y axis - perpendicularly to X axis (in the plate plane)
Z axis - perpendicularly to the plate plane
user-defined
X axis - along the longest segment belonging to the plate contour
Y axis - perpendicularly to X axis (in the plate plane)
Z axis - perpendicularly to the plate plane

14.5.General part

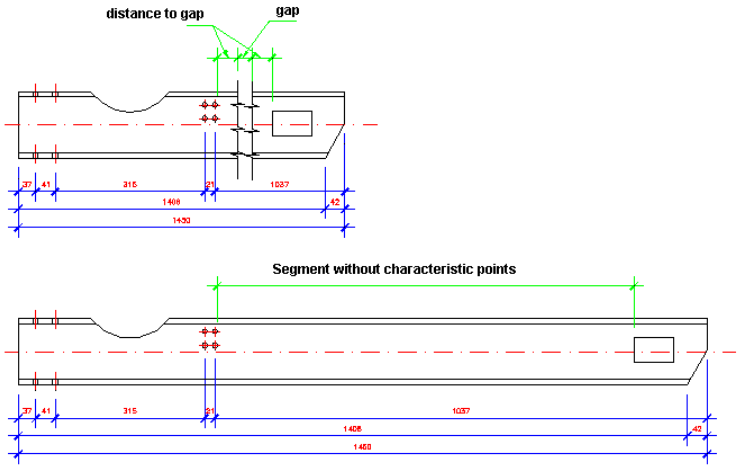
In this part of the dialog box the user may determine the following parameters:

- *ACAD dimension style* - enables selecting a dimensioning style of the AutoCAD ® program that will be applied in the dimensioning style currently selected; the list contains all the dimensioning styles defined in a DWG format file
- *Distance to the first dimension line* – an edit field where the user may specify the distance between an edge of a dimensioned part and first, generated automatically, dimension line
- *Distance between dimension lines* – an edit field where the user may specify the distance between successive dimension lines (see the drawing below)
- *Extension of axis beyond edge* - an edit field where the user may specify the length of extension of lines outside the element edges on both sides of the element



- *Part shortening* - if this option is switched on, then long elements (these element segments which do not include any characteristic points) will be shortened in a final drawing; the element shortening is presented as a gap in a shortened element.
  The operation of element shortening will be performed if the element segment not including any characteristic points, is longer than a value of the user-defined minimal distance without details; below the drawing illustrates the same element - shortened (top part of the drawing) and not subjected to the operation of shortening (bottom part of the drawing)
  Moreover, the user may define a type of the break line when performing the operation of element shortening; the list contains all the available line types of the AutoCAD ® program saved in a DWG format file - thickness and color of the break line are identical as those of an element contour.

When performing the operation of element shortening, the following parameters may also be specified (see the drawing below): gap value as well as value of the distance to gap (distance between a characteristic point and a gap).



- *Invisible edges* - if this option is switched off, then in a final drawing invisible edges of an element are hidden; if this option is switched on, then invisible edges are presented in a drawing - their parameters are set on the selection lists located to the right of the option: line color, line type, line thickness
- *Axes* - if this option is switched off, then profile axes are hidden in a final drawing; if this option is switched on, then profile axes are presented in a drawing - their parameters are set on the selection lists located to the right of the option: line color, line type, line thickness; profile axes are presented in all projections - in final drawings axes are extended by 5 mm outside an edge of a dimensioned object
- *User parts* - if this option is switched off, then user parts are hidden in a final drawing; if this option is switched on, then user parts are presented in a drawing - their parameters are set on the selection lists located to the right of the option: line color, line type, line thickness
- **Cut parameters** – pressing this button opens another dialog box, in which the user may determine parameters of section hatching and parameters of designation of a part section.

For styles of assemblies or groups, take note that *Distance to the first dimension line* indicates the distance measured to the edge determined by a rectangle describing an assembly projection (rectangle sides are parallel to the axes of the element local coordinate system).

When defining a modification of an assembly style there is also *the Draw parts in actual location* option available. If this option is switched off, then projections of an assembly are generated in the coordinate system determined by the main part of the assembly; if this option is switched on, then projections of the assembly are generated in WCS (the Global coordinate system).

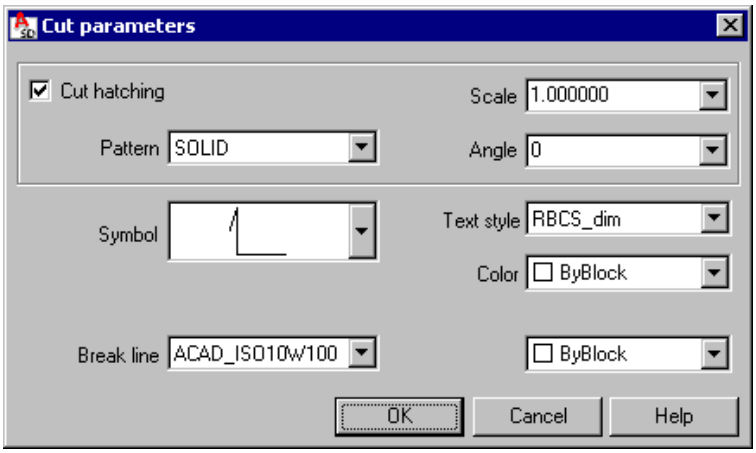
14.6. Geometrical dimensions

This part of the dialog box used to define or modify a dimensioning style depends on the selected category:

- single part
- assembly
- group.

14.7. Cut parameters

The dialog box shown in the drawing below opens on pressing the **Cut parameters** button in the ***Definition / modification of dimensioning style*** dialog box.



If the *Cut hatching* option is switched on, surfaces seen as cuts through element walls are hatched. The following hatching parameters can be defined:

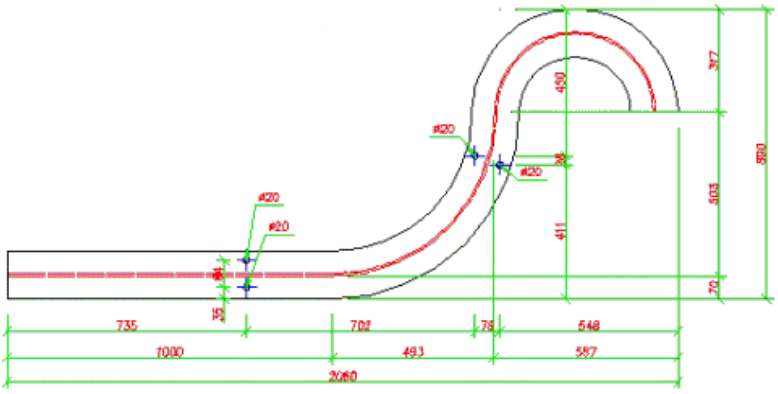
- *Pattern* – a list of available patterns of the AutoCAD ® program
- *Angle* – a list containing values of the inclination angle of the hatching pattern
- *Scale* – a list of scale values that cause elongating or shortening of the hatching pattern.

In the lower part of the dialog box it is possible to determine parameters of a cut through elements: cut symbol, style and color of a text, style and color of a break line.

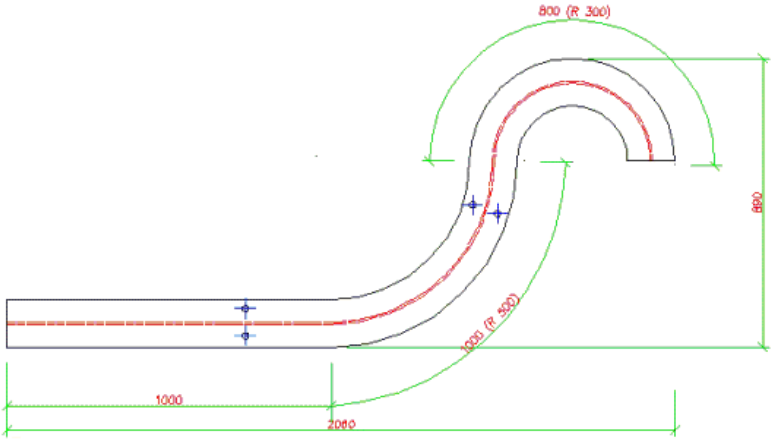
14.8. Dimensioning of curved parts

Curved elements may be presented on the screen and dimensioned in the following way:
Single parts:

- as standard objects using the *Orthogonal* option; objects are presented so that their actual shape is shown and dimensions are generated in the orthogonal system (compare example dimensions generated with the use of this option in the drawing below)



- as curved objects using the *Aligned to curve* option; elements are presented so that their actual shape is shown and dimensions are generated in the system adjusted to the profile curvature (compare example dimensions generated with the use of this option in the drawing below); total dimensions are presented in the orthogonal system (coordinate system of the curved part) – the remaining dimensions are generated in the longitudinal / transversal system; longitudinal dimensions are always parallel to the profile axis (dimension values are calculated along the length of arc axis, moreover, a radius value is given in parentheses); transverse dimensions are segments perpendicular to the profile axis and passing through the dimension point.



- using the *Developed* option; object dimensions are presented in the developed form in the orthogonal system; pressing the **Options** button opens the additional dialog box: Developed parts.

Assemblies

- as standard elements using the *Orthogonal* option; objects are presented so that their actual shape is shown and dimensions are generated in the orthogonal system
- as curved elements using the *Aligned to curve* option; elements are presented so that their actual shape is shown and dimensions are generated in the system adjusted to the profile curvature; total dimensions are presented in the orthogonal system (coordinate system of the curved part) – the remaining dimensions are generated in the longitudinal / transversal system; longitudinal dimensions are always parallel to the main part axis (dimension values are calculated along the length of arc axis, moreover, a radius value is given in parentheses); total transverse dimensions are determined by cutting planes (external transverse dimensions are parallel to total transverse dimensions); internal transverse

dimensions are segments perpendicular to the profile axis and passing through the dimension point.

Single parts

For the options: *Orthogonal* and *Aligned to curve* projections of the actual element shape are obtained; elements are projected onto planes of the coordinate system of a curved part. The coordinate system of a curved part is a coordinate system used only in drawings; it is defined in the following manner:

For profiles:

If a profile includes linear segments, then the longest segment is selected and the coordinate system of a curved part coincides with the coordinate system of this segment (as if it were a single simple profile).

If a profile does not contain linear segments, then the coordinate system of a curved part may be defined in two ways:

A. X axis of the coordinate system of a curved part is positioned along the chord of the longest (among all the parts) arc segment
B. X axis of the coordinate system of a curved part is a tangent to the chord of the longest (among all the parts) arc segment at the beginning point.

The method of defining the coordinate system of a curved part is selected in the ***Dimensioning Style Settings*** dialog box:

- if the *Align arc profiles to the chord* option is switched off, the coordinate system is defined applying method A
- if the *Align arc profiles to the chord* option is switched on, the coordinate system is defined applying method B.

In both cases Y axis of the coordinate system of a curved part is positioned in the plane of part bending, irrespective of profile rotation.

For plates:

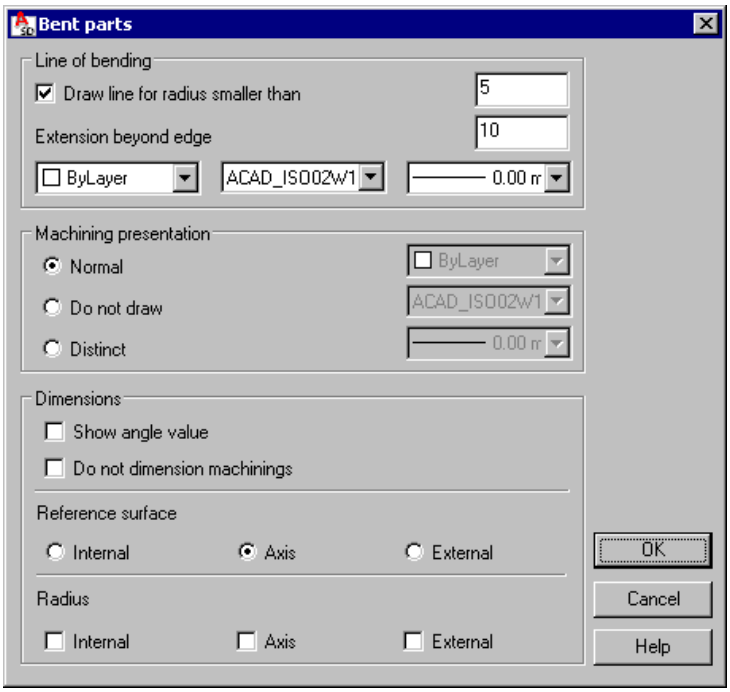
The coordinate system of a curved part is identical with the part UCS.

Assemblies

Assemblies are considered curved if the main part of an assembly is curved. For curved assemblies projections of the actual element shape are obtained; elements are projected onto planes of the coordinate system of a curved part (main part of an assembly). The coordinate system of a curved part is defined as for a single part.

14.9. Developed parts - options

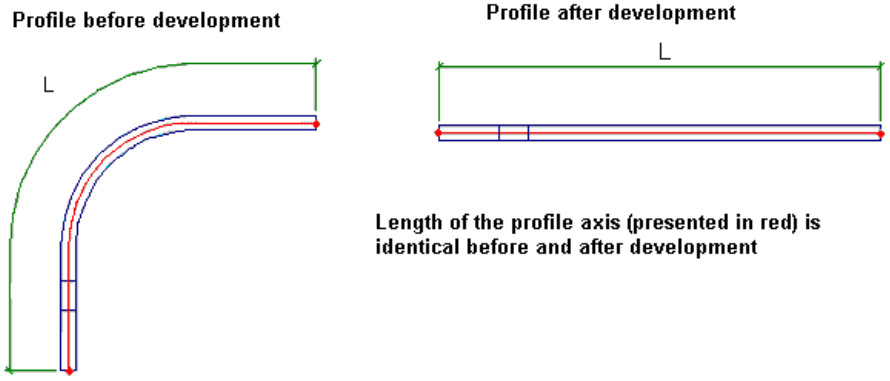
The dialog box shown in the drawing below opens after pressing the **Options** button in the Dimensioning style settings dialog box (the option is available for curved parts presented in a developed form).



The above dialog box includes the following fields:

- *Line of bending* – the options located in this field are used to set parameters of bending edges
- *Machining presentation* – options provided in this field are used to determine how to present machining in a drawing; selection lists in the right part of the field are accessible only when the *Distinct* option is selected.

The idea of element development consists in transforming the curve representing the element axis into a straight line in such a way so that the axis length and distances between points on the axis remain the same (see the drawing below).



In the top part of the dialog box parameters of the line of bending may be defined; this line is positioned in the middle of each arc-shaped segment. Thickness, color and type of this line is determined in the bottom part of the *Line of bending* field; three selection lists are used for that.
The line of bending is presented in a drawing if the *Draw line for radius smaller than* option is switched on. A radius value may also be specified – the line is usually drawn for small radiuses; thus the user defines the limit (minimum) radius value for which the line of bending will be drawn.

In the *Extension beyond edge* edit field the user may determine a value of distance, i.e. extension of the line beyond the element edges on both sides of the element.

The *Machining presentation* field enables selection of one of the following types of presentation (NOTE: the options below are not available in version 5.0):

- *Do not draw* – if this option is selected, then only profile shape will be presented in a drawing (machinings will not be shown in a drawing)
- *Normal* - if this option is selected, all openings, cuttings, etc. will be presented in a drawing consistently with the style adopted for standard parts
- *Distinct* - if this option is selected, then machinings will be presented in a drawing; a shape of developed machinings will be defined identically as for normal presentation type; in addition, from the three selection lists available to this type of presentation the user may choose the following machining parameters: thickness, color and line type.

Curved parts are dimensioned identically as rectilinear parts. There is a possibility to switch off dimensioning of machinings which may be shown in a drawing of a developed part. If the *Do not dimension machinings* option is switched on, then all the dimension points defining machinings are disregarded and only total dimensions and dimenions to the line of bending are presented.

The options provided in the lower part of the dialog box refer to drawings of developed bent plates. NOTE: these drawings present the original shape and dimensions of plates (without plastic deformations).

The options in this part of the dialog box allow insertion of the following items in a drawing: bending lines, angle of bending, radius (external or to axis), dimensions and machinings of a plate.

14.10. Geometrical dimensions (single part)

Dimension lines are positioned outside an element in the following order starting from the edge of a dimensioned element:

1. elements located inside the dimensioned part
2. drilled openings
3. chamfers and cuts
4. overall dimensions.

This part of the dialog box consists of three identical tabs on which the following parameters may be determined:

- *dimension format* – for all the dimensions generated automatically in a view the following dimension settings are available:
  relative - chain – the program generates a complete dimension chain with standard (relative) dimensions
  absolute - chain – the program generates a dimension chain which begins in the reference point; dimensions are positioned on extension lines - a dimension value indicates the coordinate of a dimensioned point with respect to the reference point
  relative + absolute - chain - the program generates an accumulative dimension chain and on the dimension line provides real (relative) dimensions of segments between the dimensioned points
  to base – each dimension is positioned on a separate dimension line attached to the reference point

Pressing the **Ref. point** button opens the Reference point dialog box, where the user may determine an additional dimension point during generation of some dimensions (e.g. absolute dimensions).

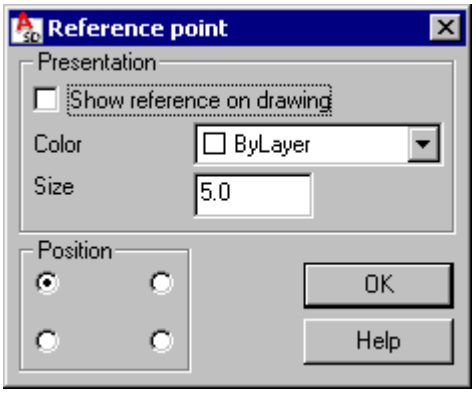
- *overall* - these are the linear dimensions presenting an overall size of a dimensioned part; these dimensions are generated if the option is turned on; they are always positioned along the horizontal or vertical axis (parallel to the plane of the element local coordinate system); horizontal dimensions may be placed above or under the dimensioned element (it is also possible to exclude dimensions from presentation), as regards vertical dimensions, they are positioned to the left or to the right of the dimensioned element (it is also possible to exclude dimensions from presentation); selection of the *optimal* option results in positioning the dimension line on this side where the longest edge of the dimensioned element is placed (so that extension lines on the edges do not need to be lengthened)
- *bevels and cuts* - these dimensions concern all the discontinuances on element edges (corners, cuts, chamfers, etc.); they do not pertain to drilled openings; there are three options available:
  chain - all dimensions are presented on the common dimension line; lines are always positioned along the vertical and horizontal axes; dimension lines are presented on this element side where a discontinuance is found - dimension lines are drawn to reach the most distant corners (edges) of an element - identically as for overall dimensions
  from edge - each dimension is presented on a separate dimension line; successive dimensions show distances from the common point to successive characteristic points of an element; dimension lines are presented on this element side where a discontinuance is found
  aligned - each segment of a dimensioned element's edge is ascribed its own dimension line which is parallel to the dimensioned segment; in the case of arc segments, the chord is subjected to the dimensioning
- *internal recess* - these are the linear dimensions of recesses included inside the element contour; there are the following options available:
  inside part - dimension lines are drawn inside the contour of a dimensioned element; all dimensions are presented on one dimension line (dimension chain) - vertical or horizontal; the dimension line is extended to the closest edge of a dimensioned element
  outside part - dimension lines are drawn outside the dimensioned element (the same rules are applicable as in the case of bevels and cuts)
- *openings in view* - these are the linear dimensions that indicate positions of drilled openings; dimension points may be positioned at opening centers or at external points of the contour of a circle representing an opening; the options: *chain* and *from edge* are also available (as for bevels and cuts)
- *perpendicular openings* - these are the linear dimensions that indicate positions of drilled openings located in the plane perpendicular to the projection plane; the same rules apply here as for openings in view; the options: *chain* and *from edge* are available, as well (as for bevels and cuts)
- *angular* - these are the dimensions presented for the edges of a dimensioned element that are not parallel to the axes of the element local coordinate system; the following parameters are specified:
  line with respect to which the dimension is to be determined (with respect to horizontal or vertical line)
  value of the angular dimension expressed as: angle value or edge inclination.

Moreover, this part of the dialog box contains options that enable arc dimensioning (these dimensions do not concern drilled openings and circles):

- *diameters* - for all circles (not arcs), a circle diameter is presented; a dimension line coincides with the circle diameter positioned at an 45-degree angle to the horizontal axis
- *radial* - for all arcs and circles, the radius is presented; circles are dimensioned in this manner if the *Diameters* option is switched off; a dimension line coincides with the radius set at an 30-degree angle starting from the beginning point of the arc.

14.11. Reference point

The reference point is an auxiliary point positioned at one of the four points of the main part; it may be shown in a drawing as .
The reference point is used as an additional dimensioned point in generation of some dimensions (e.g. absolute dimensions). It is positioned on one of the ends of the main part's edge parallel to its axis (by default the Bottom, Left point is chosen).
Position, size and color of the reference point can be configured in the dialog box shown in the drawing below, opened by pressing the **Ref. point** button in the New dimensioning style dialog box.



14.12. Geometrical dimensions (assembly)

This part of the dialog box consists of three identical tabs which allow the user to determine the following parameters:

- *dimension format* – as for single parts
- *overall* - the option operates identically as in the case of single parts
- *part spacing* - the aim of these dimensions is to present mutual position of the assembly component parts; the dimensions are shown on external dimension lines (vertical or horizontal); they are generated parallel to the overall dimension lines; there are the following options available:
  contour
  dimensions along the main axis (horizontal dimensions) – beginning and end of a subpart is projected onto the direction parallel to the axis of the element's main part
  dimensions transverse with respect to the axis (vertical dimensions) - beginning and end of a subpart is projected onto the direction parallel to the specified direction of dimensioning; the reference point should also be projected on this line; if any of thus determined dimension points is positioned on one of the edges of the main part, parallel to the axis of the main part, this point is not dimensioned; a given part is projected on the dimension chain positioned on this side where the edge closer to the dimensioned part is located
  axes
  dimensions along the main axis (horizontal dimensions) – the point of intersection of the element axis and the main part's edge, parallel to the axis and the closest to the geometrical center of the dimensioned subpart
  dimensions transverse with respect to the axis (vertical dimensions) – significant only for elements whose axis is parallel to the axis of the main element - the axis is projected onto the direction perpendicular to it; a dimension is placed on the vertical dimension line; if the axis of a dimensioned subpart coincides with the axis of the main part, then the dimension is not generated

point
dimensions along the main axis (horizontal dimensions) – one of the edge points of a subpart is projected onto the direction parallel to the main part's axis (a point positioned closer to the main part's edge is selected); if both edge points of a subpart are located on the main part's edge, then the point closer to the reference point is dimensioned
dimensions transverse with respect to the axis (vertical dimensions) – the first (counting from the reference point) point met on the subpart's edge is projected onto the direction parallel to the specified direction of dimensioning; the reference point should be projected onto this line, as well; if any of thus determined dimension points is positioned on one of the edges of the main part, parallel to the main part's axis, this point is not dimensioned; a given part is projected on the dimension chain positioned on this side where the edge closer to the dimensioned part is located
optimal
dimensions along the main axis (horizontal dimensions) – selection depends on a subpart type:
plates perpendicular to the main part's axis, set orthogonally – the *axes* option
plates perpendicular to the main part's axis, set slanting - the *point* option
plates parallel to the main part's axis – the *contour* option
profiles perpendicular to the main part's axis – the *point* option
profiles parallel to the main part's axis - the *contour* option
dimensions transverse with respect to the axis (vertical dimensions) - selection depends on a subpart type:
plates perpendicular to the main part's axis, set orthogonally - the *point* option
plates perpendicular to the main part's axis, set slanting - the *point* option
plates parallel to the main part's axis - the *contour* option
profiles perpendicular to the main part's axis - the *contour* option
profiles parallel to the main part's axis - the *contour* option

Additionally, the following options are accessible (they parametrize the position of a dimension line):
With overall – dimension lines are always placed on the same side as overall dimension lines; if position of overall dimensions is not defined, then lines are positioned according to the *Optimal* option
Optimal - dimension lines are placed on this element side which is closer to a greater number of dimension points

- *part dimensions* – the option enables presentation only of overall dimensions of subparts; dimensions are generated following the same rules as in case of the whole assembly; these lines are always generated as internal dimension lines; orientation of dimension lines is determined in the local coordinate system of a dimensioned subpart or in the coordinate system of the main part of an assembly depending on the settings: part CS (Coordinate System) or assembly CS (Coordinate System); if any of the dimensions generated by the algorithm is already shown on the external dimension line as an overall dimension, then it is not repeated
- *openings in view* - the options are used for dimensioning of opening positions with respect to the element's main part; dimensions in the direction parallel to the main axis are placed on the external dimension line, whereas transverse dimensions are positioned on external or internal dimension lines depending on an opening position in a part; the options *Axes* and *Contour* are taken into account in case of dimensions transverse with respect to the main element's axis

  Dimensions along the axis: all the dimensions are projected onto common external dimension line which is parallel to the main element's axis
  Dimensions transverse with respect to the axis: these dimensions are placed on external or internal dimension lines depending on opening positions;
  dimensions on external dimension lines are generated for openings positioned on transverse edges of an assembly (in endplates); external dimension lines are positioned parallel to the

direction specified for overall dimensions; external dimension lines are always attached to the element's overall dimension
internal dimension lines are generated for openings positioned along the main element's axis; they are always placed perpendicularly to the main element's axis and as close as possible to dimensioned openings (in such a way so that they do not overlap with other elements in a drawing); internal dimension lines are attached to the main element's axis or to one of external edges of the main part that is parallel to the axis (an edge positioned closer to a greater number of dimensioned openings is selected)

- *perpendicular openings* – the option works identically as the *Openings in view* option; NOTE: if both options (Openings in view and Perpendicular openings) are switched on, then all the points are projected onto a common dimension line
- *angular* – the following parameters can be specified:
  external – the dimensioning concerns external edges of the main part and of subparts placed outside the main part's contour, positioned unorthogonally with respect to the coordinate system of the main part; an acute angle is always shown: the base is a vertical line
  internal - the dimensioning concerns edges of subparts placed within the main part's contour, positioned unorthogonally with respect to the coordinate system of the main part; an acute angle is always shown: the base is an element's edge parallel to the axis of the main part
  both – dimensions are generated on the basis of both above options.

The options of arc dimensioning operate identically as in the case of single parts.
There is also the *Depth* option available; it is used to determine the 'depth' of a structure for which assemblies will be presented in drawings. The depth may be defined as:

- *relative* – with respect to a value of the 'depth' of a whole structure (100% - total depth)
- *in absolute units* – in current units.

14.13. Geometrical dimensions (group)

This part of the dialog box consists of three identical tabs which allow the user to determine the following parameters:

- *overall* - the option operates identically as in the case of single parts
- *axial dimensions* - dimensions are generated as a dimension chain showing axial dimensions of a presented group; dimension points are generated at points of axis intersection (Note: for groups axes of main assembly parts are presented; therefore, axial dimensions are determined only for the visible axes); there are three options available:
  orthogonal - dimension chains are positioned along vertical or horizontal edges of the view
  aligned - dimensions are positioned as parallel to the dimensioned segment; dimensions are not extended to the most distant points of a group; additional dimension points are generated at the beginning and end of the group where there are no axis intersections
  both - both methods described above are applied here
- *assembly dimensions* - dimensions present the entire dimensions of assemblies included in a group; they are generated identically as overall dimensions for each assembly separately; there are the following options available:
  chain - all dimensions are projected on a common external dimension line (chain) positioned vertically or horizontally
  from edge - each dimension is presented on a separate dimension line; dimension lines are parallel to the axes of the local coordinate system; they are located on this side of a dimensioned element, which is positioned closer to dimension points
  aligned - dimensions are set in a parallel position with respect to the main assembly part (in the local coordinate system of the assembly); they are presented on the external side of the assembly (in relation to the group center).
- *angular* - the option operates identically as in the case of single parts.

The options of arc dimensioning operate identically as in the case of single parts.

14.14. Geometrical dimensions (3D view)

All dimensions generated automatically are positioned in one of the (orthogonal) planes belonging the local coordinate system of a part or group.
The following rules apply here:
For an object being dimensioned (part, assembly, group) the program defines a rectangular prism circumscribed on the selected part (the rectangular prism is perpendicular to the local coordinate system of the object). Dimension lines are positioned in one of the planes of the rectangular prism, so that they do not conceal the object. The options: *Distance to the first dimension line* and *Distance between dimension lines* refer to these planes in which dimension lines are positioned.

Single part, **Assembly**
For a single part and assembly the *Dimensions* field includes options enabling selection of dimensions and direction to be generated automatically.

Group
For groups the following options are available in the dialog box:
Presentation
There is a list available, on which the user may choose a method of presenting a part in the drawing. The list contains the following options:
Real – a full drawing of a part with all details is presented
Simplified – profile fillets are ignored
Scheme – profiles are presented as lines.
Below there are lists including settings of color, type and thickness of a line presenting a profile.
Dimensions
For groups the *Dimensions* field comprises options enabling selection of dimensions and direction to be generated automatically (for total dimensions, element axes and workframe axes).

14.15. Additional texts

If the *Comment* option is switched on, then automatically, to the views of a dimensioned object a title (name) is added, which may be a text (comprising one or several lines) containing variables or any character string defined by the user. In the bottom part of this field accessible variables are provided - if they are pressed, then a variable is added to the defined comment to the edit field where the cursor is positioned - a whole text of the comment is displayed in the edit field.
A comment may include the following variables:
%%Pos - if this option is turned on, then a position description is added to a text
%%Name - if this option is turned on, then a profile or plate name is added to a text, e.g. IPE 100
%%Scale - if this option is turned on, then a scale is added to a text, e.g. 1:20
%%Vname - if this option is turned on, then an automatic name of a view or section is added to a text
%%Length - if this option is turned on, then profile or plate length expressed in the current project units is added to a text
%%Mat - if this option is turned on, then name of a part material is added to a text
%%Quant - if this option is turned on, then a number of elements of a presented position is added to a text
%% Weight - if this option is turned on, then a calculated part weight is added to a text
%%Family - if this option is turned on, then the name of a family the part belongs to is added to the text
%%Surf - if this option is turned on, then a type of surface finishing of a structure element is added to a text.
The user may set a comment in a drawing; the following can be determined: text style, position of a text in a view and text aligning (align right, align left, center) as well as color of the comment text.

14.16. Styles of descriptions and symbols

In this part of the dialog box the user may select styles for the following descriptions and symbols (designations) presented in final drawings:

- Openings/bolts
- Parts
- Assemblies
- Elevation marks
- Weld symbols
- Workframe axes.

NOTE:
The number of symbols listed above depends on the selected category of dimensioning style (single part, assembly, group).

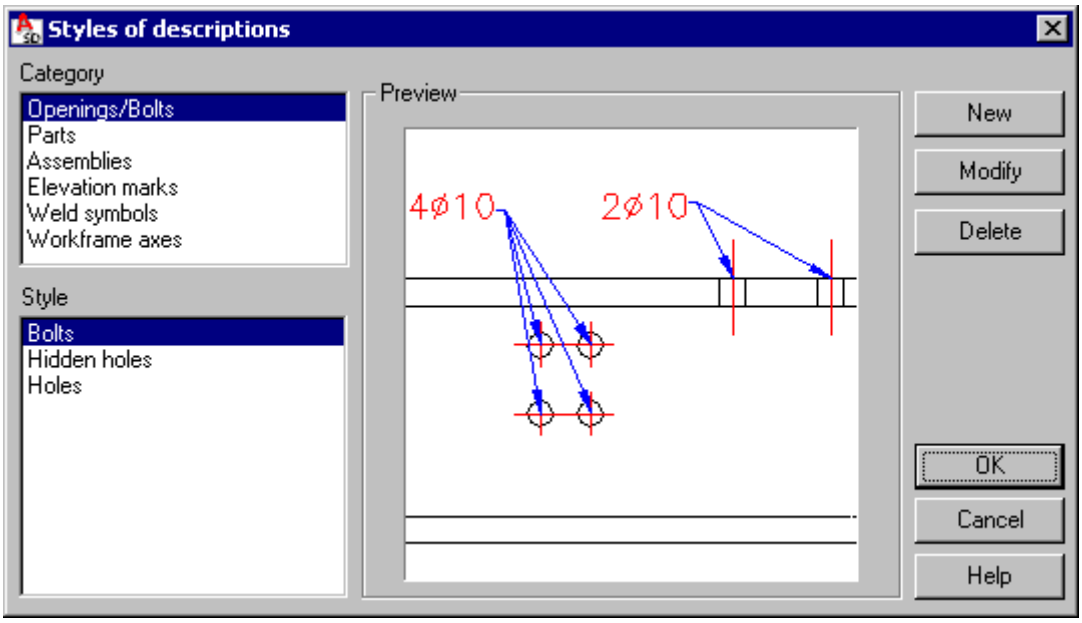
Pressing one of the (**…**) buttons on the right of the symbols listed opens the Styles of descriptions dialog box with the symbol category selected.

15. DESCRIPTION STYLES

15.1.Description styles

The dialog box shown in the figure below is displayed on the screen after pressing one of the (**...**) buttons located in the Dimensioning style settings dialog box.
The option enables definition of description styles for elements. The option is also available from:

- menu by selecting the *Steel / Styles / Description styles* option
- command line: RBCS_DESCRIPTION.



The above dialog box operates in the similar manner as the Dimensioning styles dialog box. A set of description styles is saved to a DWG format file or in a template (a DWT file) .

In the above dialog box selecting one of the categories in the *Category* field enables presenting styles defined for this category type; after highlighting one of the categories in the *Style* field, all the styles defined for the selected category are displayed. A preview of a selected style is shown in the *Preview* field. The following categories are available:

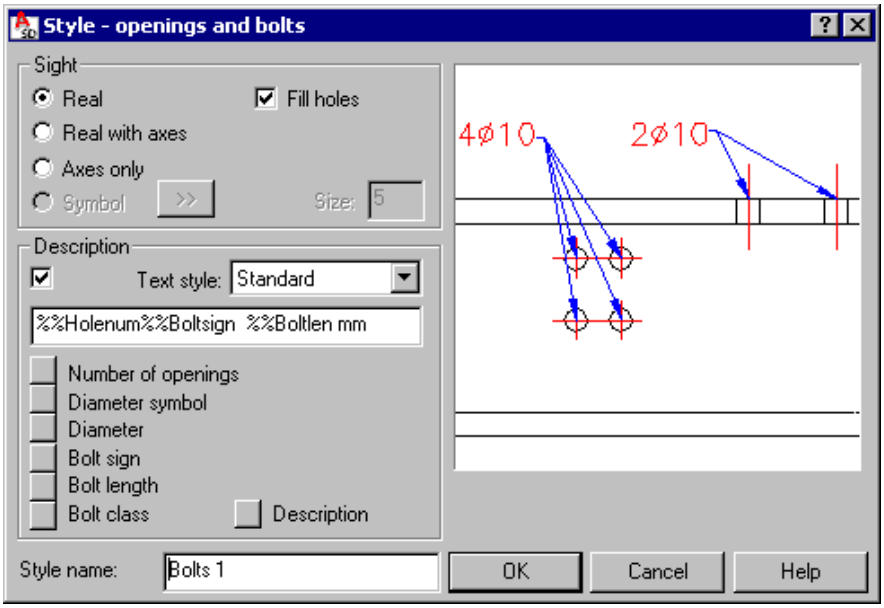
- Openings/bolts
- Parts / Assemblies
- Elevation marks
- Weld symbols
- Workframe axes.

One of the styles defined for each category is selected as a default style; it means that this style will be applied as a default one during manual (not automatic!) definition of symbols in a drawing.
The are also the following buttons provided in the above dialog box (apart from the standard ones):

- **New** - pressing this button opens the dialog box where the user may define a description style for a selected category; name of a new style and all its settings are inherited from the style currently selected
- **Modify** - pressing this button opens the dialog box where the user may modify a description style of a selected category
- **Delete** - pressing this button results in deleting the style currently selected from the list of available styles for a given category.

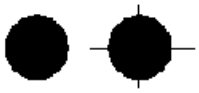
15.2. Styles of descriptions - Openings/Bolts

The dialog box is used to define a new style or modify an existing style of opening or bolt description.
The dialog box opens after pressing the **New** or **Modify** button in the Styles of descriptions dialog box when the *Openings/Bolts* category is selected.



The following parameters may be determined in the above dialog box:

- the bottom part of the dialog box contains an edit field where the user may specify a style name
- the *Sight* view enables selection of a method of opening generation in a drawing:
  Real - real opening edges are presented (openings in view and perpendicular openings)
  Real with axes - real opening edges are presented (openings in view and perpendicular openings) with opening axes
  Symbol - once this option is selected, openings in view and perpendicular openings are presented as symbols; pressing the **>>** button extends the dialog box; an additional part of the dialog box contains a library of symbols available in the current file
  Fill holes – if this option is switched on, then an opening for the *Real* or *Real with axes* option will be presented as shown in the drawing below:

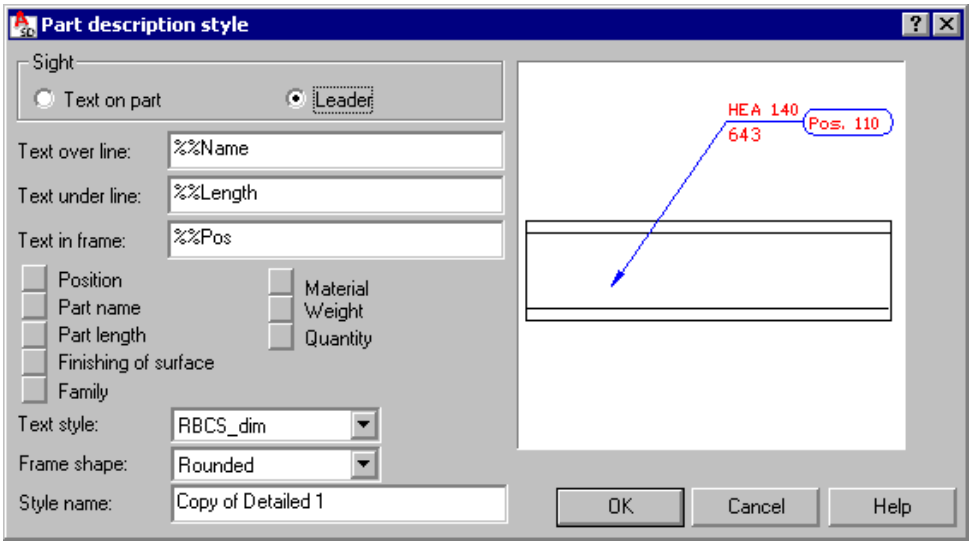
  

  There are three buttons provided in the additional part of the dialog box:
  Edit - opens the dialog box in which the user may modify a symbol selected on the list of available symbols
  Add - opens the dialog box used for defining a symbol; symbols are defined as AutoCAD blocks; an insertion point for a defined block is treated as a center of a drilled opening (top view) or as an end of the opening axis (side view)
  Delete - deletes a symbol selected on the list of available symbols
  Opening symbols must be defined as a pair: opening in view and perpendicular opening; the *Size* field determines the size of a symbol in a final drawing, expressed in the currently applied project units (the size refers to a symbol presented in a top view; a symbol in a side view is scaled proportionally to obtain a specified size, however, symbol length is adjusted to the thickness of a drilled part)
- the *Preview* field shows an opening as opening in view and perpendicular opening

- in the *Description* field the user may define a manner of opening description; descriptions of openings are generated if the *Description* option is switched on;
  The *Text style* list contains all text styles defined in a DWG format file; the edit field located below presents a description (text) syntax; a text may comprise any character string defined by the user - variables available under that field may also be applied - when they are pressed, it results in adding a variable to the defined comment to the edit field where the cursor is positioned; the following variables may be description components:
  %%Holenum - if this option is turned on, a number of openings in the opening group is added to the text
  %%Diasym - if this option is turned on, then ∅ symbol is added to the text
  %%Dia - if this option is turned on, then a value of opening diameter is added to the text
  %%Boltsign - if this option is turned on, then a bolt symbol is added to the text
  %%Bltlen - if this option is turned on, then bolt length is added to the text
  %% Boltclass - if this option is turned on, then a bolt grade is added to the text
  %%Desc - if this option is turned on, then a bolt description is added to the text
- if the *Groups on common leader* option is activated, then while dimensioning a group of openings, one description (one leader) is generated for all openings in a group; if this option is turned off, descriptions (leaders) are generated for each opening.

15.3.Styles of descriptions - Parts/Assemblies

The dialog box is used to define a new style or modify an existing style of part or assembly description.
The dialog box opens after pressing the **New** or **Modify** button in the Styles of descriptions dialog box when the *Parts* or *Assemblies* category is selected.



The following parameters may be determined in the above dialog box:

- the bottom part of the dialog box contains an edit field where the user may specify a style name
- the *Sight* view enables selection of a method of generating a description in a drawing:
  Text on part
  Only one edit field: *Text* is available in which a description may be defined - the method of its definition is identical to that described below for the *Leader* option. A text may be put in a frame, whose shape may be chosen in the *Frame shape* field; a text is placed within the contour of a dimensioned element (however, it is positioned in such a manner so that overlapping of elements presented in a drawing is avoided).

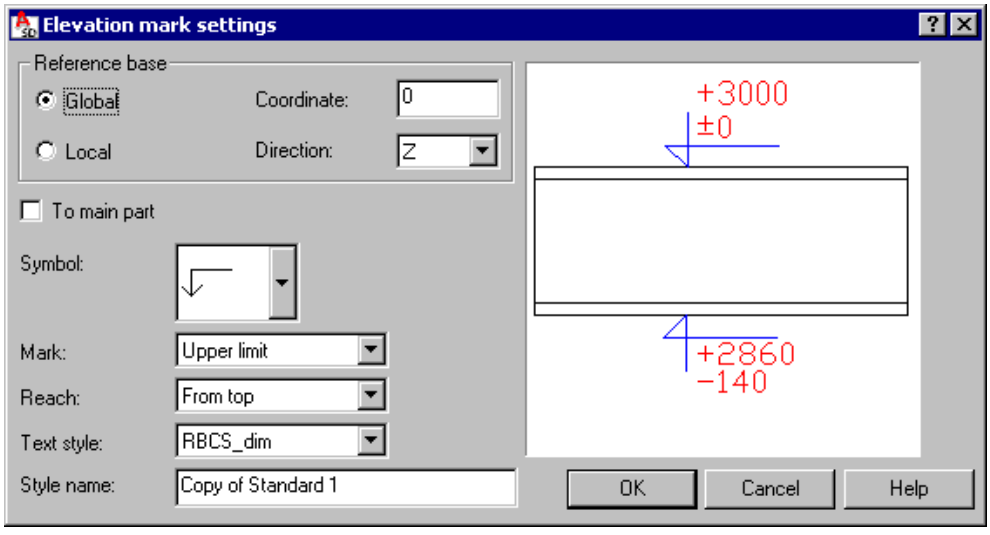
Leader
There are three edit fields available: *Text over line*, *Text under line*, *Text in frame*; each of them may be freely defined; the texts listed may comprise any character string defined by the user - variables available under these fields may also be applied - when they are pressed, it results in adding a variable to the defined comment to the edit field where the cursor is positioned; the following variables may be description components:
%%Pos - if this option is turned on, then description of a position is added to the text
%%Name - if this option is turned on, then a profile or plate name is added to the text
%%Length - if this option is turned on, then profile or plate length is added to the text
%%Mat - if this option is turned on, then a name of a part material is added to the text
%% Weight - if this option is turned on, then a calculated weight of a part is added to the text
%%Quant - if this option is turned on, then a number of parts is added to the text
%%Surf - if this option is turned on, then a type of surface finishing of a structure element is added to the text (available in the ***Part description style*** dialog box; it is not available in the dialog box for the assembly description style)
%%Family - if this option is turned on, then the name of a family the part belongs to is added to the text

- *Text style* - enables selecting a font to be applied in descriptions
- *Frame shape* - enables selecting a frame shape for a leader; available shapes include: rectangular, rounded or none (i.e. without frame).

Texts in leaders are positioned outside dimension lines in such a manner so that overlapping of leaders and other drawing elements and leaders with other leaders is avoided.

15.4. Styles of descriptions - Elevation marks

The dialog box is used to define a new style or modify an existing style of description of elevation marks; the elevation mark consists of a symbol and a coordinate.
The dialog box opens after pressing the **New** or **Modify** button in the Styles of descriptions dialog box when the *Elevation marks* category is selected.



The following parameters may be determined in the above dialog box:

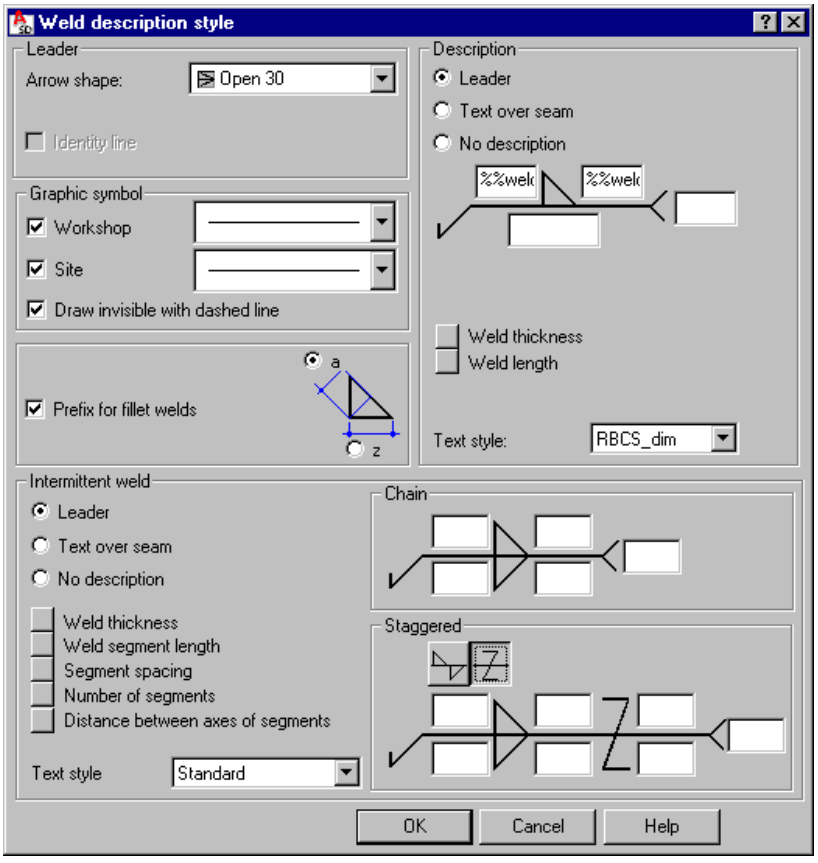
- the bottom part of the dialog box contains an edit field where the user may specify a style name

- in the *Reference base* field the user may choose the coordinate system (global coordinate system of a model or local coordinate system of an element) being the basis for definition of: elevation reference (coordinate) and direction
- if the *To main part* option is switched on for assemblies and groups, then limits are determined for a main part of the group and not for a group as a whole
- the *Symbol* field allows selecting one of the elevation symbols provided together with the program
- the *Mark* field allows selecting a part of the element edge on which an elevation mark is to be generated; symbols are generated automatically at the most distant point along the selected direction; the list contains the following options: upper limit, lower limit or both
- in the *Reach* field the user may determine a point of connection between a symbol and an edge of a dimensioned element: from top, from bottom, aligned
- *Text style* - enables selecting a font to be applied in descriptions.

Symbols are located near edges of a dimensioned element (inside dimension lines).

15.5. Styles of descriptions - Weld symbols

The dialog box is used to define a new style or modify an existing style of weld symbol description. The dialog box opens after pressing the **New** or **Modify** button in the Styles of descriptions dialog box when the *Weld symbols* category is selected.



The following parameters may be determined in the above dialog box:

- the bottom part of the dialog box contains an edit field where the user may specify a style name

- on the *Arrow shape* list the user may choose shape of the arrow from the list of available arrows; a user-defined shape is available, as well
- *Text style* - enables selecting a font to be applied in descriptions.

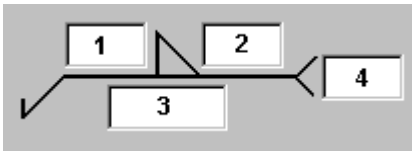
The weld symbol is composed of the following elements:
- leader ended with an arrow
- line
- weld symbol
- buttons that automatically place in the active text field – the variables corresponding to the following weld parameters: t (thickness) and l (length) expressed in mm
- additional designations.

The following rules apply to generation of weld descriptions:
- all the visible welds are given a description; an arrowhead indicating an edge is attached to the center of gravity; the arrow line may intersect an element edge or a dimension line
- weld description and extension line may not overlap with the element edge or a dimension line.

The *Description* field comprises options enabling configuration of a weld description; three possibilities of description are available:
- as a leader
- as a text over seam
- no description (only a weld is drawn).

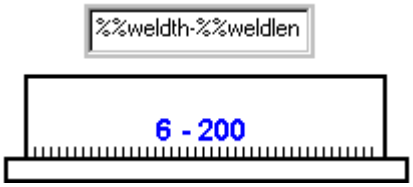
LEADER

The description may be positioned in the following locations (numbering of the description locations corresponds to the numbering in the drawing below):
1. above the extension line before the weld symbol
2. above the extension line behind the weld symbol
3. under the extension line
4. behind the extension line.



TEXT OVER SEAM

A description specified in the edit field will be placed above the weld seam (any character string with the possibility to use variables, e.g. %%weldth-%%weldlen) – see the drawing below.



In all the locations variables may be used (as in bolt descriptions); the following two variables of weld description are available in the current program version:
%%weldth – weld thickness
%%weldlen – weld length.

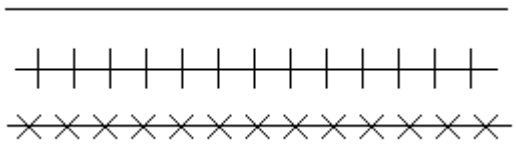
Descriptions are generated for all the welds presented in a selected projection of an assembly or group. If there are several welds of the same parameters located close to each other, then it is possible to generate for them one description on several leaders.

If the option *Draw invisible with dashed line* is switched on, then invisible weld edges (concealed by other elements) are presented in a drawing by means of a dashed line; if this option is switched off, then invisible edges are not drawn.

As regards circumferential welds, if in a model a weld forms a closed contour, then the program identifies automatically such welds in all drawings; to the weld symbol an additional symbol in the form of a circle is given in the same place as the site weld symbol.
There is a possibility to provide a line identifying a weld face in 2D drawings (the *Identity line* option is switched on); the identity line may be placed under or above an extension line. It is denoted with a dashed line of the same thickness as the extension line.
If the *Prefix for fillet welds* option is switched on, it enables as follows:

- selection of the *a* option – for weld description the program reads the thickness specified in the weld definition in a model and enters a prefix before the weld thickness (a5)
- selection of the *z* option - for weld description the program calculates the value z= a√2 and enters e.g. z7 as the weld thickness

If the *Prefix for fillet welds* option is switched off, then only the weld thickness (from a model definition) is provided.
For 2D drawings there is a possibility to present a graphical symbol of a weld in a form of one of the lines shown below (the options are available on the selection lists in the *Graphic symbol* field):



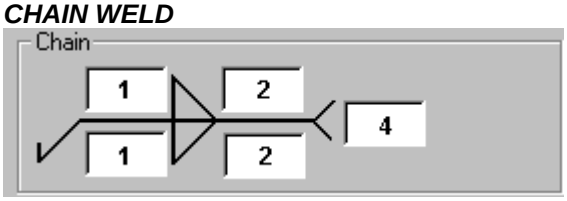
These lines can be configured separately for workshop welds and site welds. Line thickness and color are assumed identically as for the extension line.

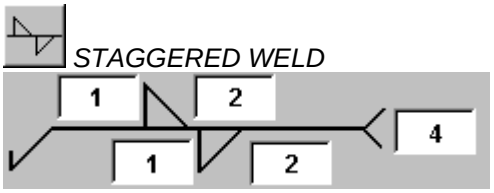
INTERMITTENT WELDS
The lower part of the dialog box holds parameters of description of intermittent welds; three possibilities of description are available here:
- as a leader
- as a text over the weld seam
- no text (only a weld is presented).

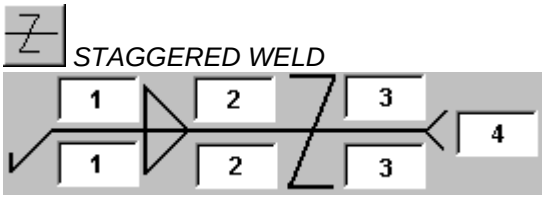
In all locations it is possible to use variables (as in bolt descriptions); the following variables of description of intermittent welds are available in the current program version:
%%weldth – weld thickness
%%lengthofseg – length of a weld segment
%%lengthofspace – spacing between segments of a weld
%%distbetwaxis – distance between axes of weld segments
%%numofseg – number of segments.

LEADER
A description may be positioned in the following locations (numbering of the description locations corresponds to the numbering in the drawing below):
1 = %%weldth
2 = metric units: %%NumOfSeg x %%LengthOfSeg
imperial units (USA) %%LengthOfSeg - %%DistBeetwAxis
3 = (%%LengthOfSpace)
4 = any character string.

CHAIN WELD







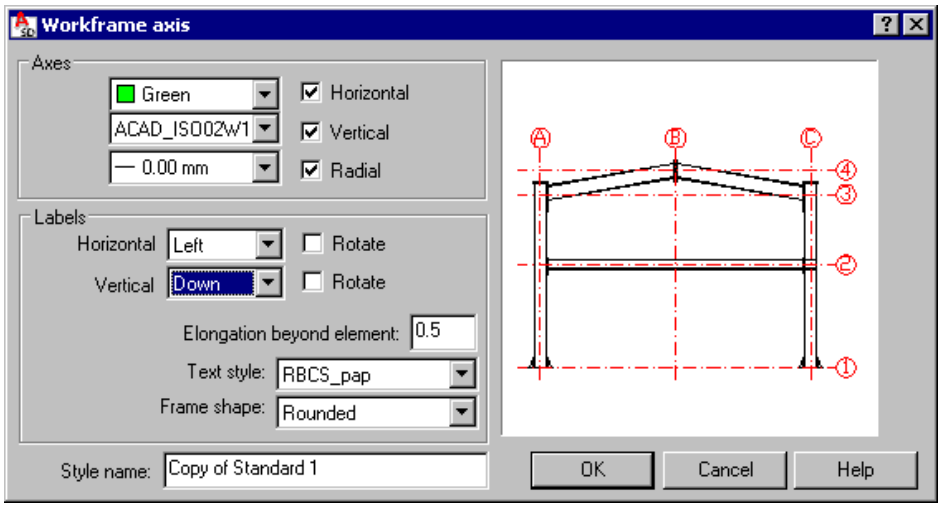
TEXT OVER SEAM

A description (any character string) specified in the edit field will be positioned over the weld seam.

15.6.Styles of descriptions - Workframe axes

The dialog box is used to define a new style or modify an existing style of describing workframe axes.

The dialog box opens after pressing the **New** or **Modify** button in the Styles of descriptions dialog box when the *Workframe axes* category is selected.



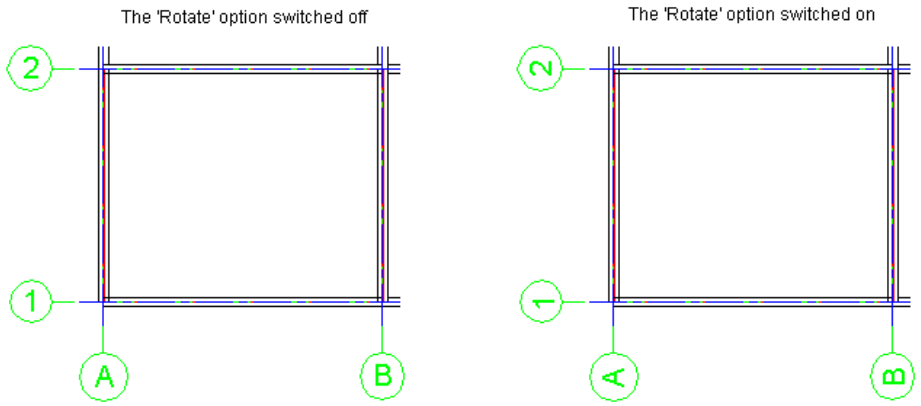
The following parameters may be determined in the above dialog box:

- *Axes* field includes options which enable setting parameters of workframe axes: color, style, line thickness; if the options: *Horizontal, Vertical, Radial* are switched on, then workframe axes (horizontal, vertical and radial, respectively) will be presented in drawings; if the options are switched off, a selected axis type will not be shown in drawings

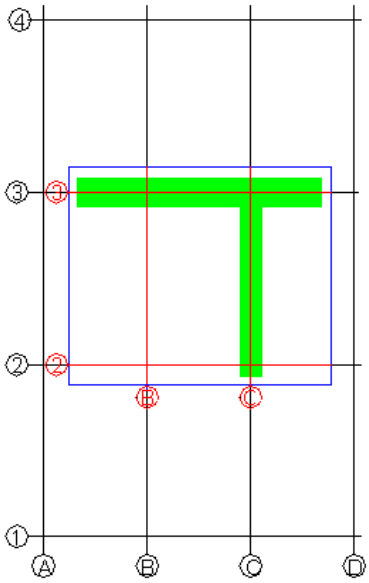
- the *Labels* field contains options which enable setting parameters of descriptions of workframe axes:

  vertical - axes may be presented with labels (above the axis, under the axis or on both sides) or without labels (the *none* option is selected); apart from that, to the right of the selection field there is the *Rotate* option – if it is switched on, then axis labels are rotated by 90 degrees (see the drawing below)

  horizontal - axes may be presented with labels (on the left side of the axis, on the right side of the axis or on both sides) or without labels (the *none* option is selected); apart from that, to the right of the selection field there is the *Rotate* option – if it is switched on, then axis labels are rotated by 90 degrees (see the drawing below)



elongation beyond element - workframe axes are contained within a rectangular contour, in which a dimensioned group is inscribed; the size of this contour is determined by a rectangle circumscribed on the group contour increased by the value defined in this field (this value is added to each side); rectangle sides are parallel to view edges; it is illustrated in the drawing below:
- green color is used to present an element scheme
- blue color is used to present element borders with a margin added on each side (a value of elongation beyond element defined in the above dialog box)
- black color is used to present the entire workframe
- red color is used to indicate which part of the workframe will be shown and where labels will be placed



text style – a list containing styles (settings) of fonts used for workframe description; the list presents all the text styles of the AutoCAD ® program available in the current project

frame shape – enables selecting the shape of a label of workframe axis description; the options available are: rounded, rectangular or none

- the bottom part of the dialog box contains an edit field where a style name may be specified.

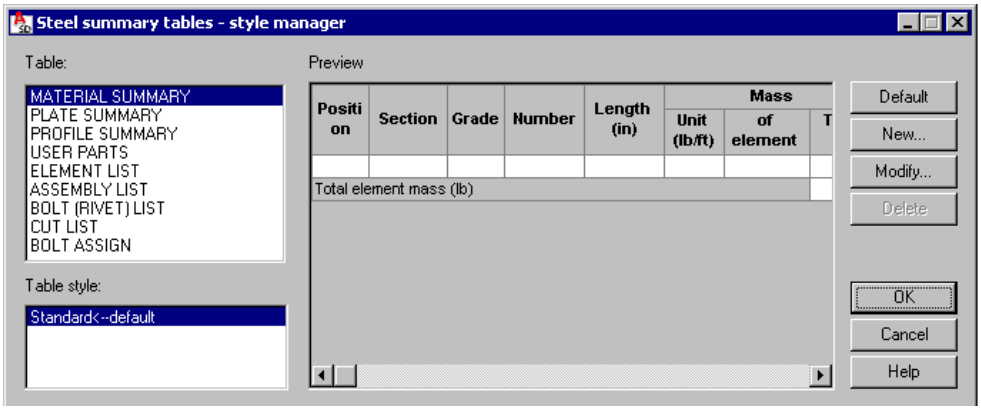
16. STEEL SUMMARY TABLES

16.1.Steel summary tables - style manager

The option allows defining/modifying tables used to prepare steel tables. The option is available from:

- the menu by selecting the *Steel* / *Reports* / *Styles* option
- the toolbar by pressing the icon
- the command line: RBCS_LISTTEMPL.

Once the option is selected, the ***Steel summary tables - style manager*** dialog box shown in the drawing below, is displayed on the screen.



The following types of steel summary tables (presented in the *Table* field in the above dialog box) are available in the program:

- Material summary
- Plate summary
- Profile summary
- User parts
- Element list
- Assembly list
- Bolt (rivet) list
- Cut list
- Boly list.

For each table type the standard table style (it is presented in the *Table style* field) has been defined in the program. Once the table type and table style are highlighted, the current table view is presented in the middle part of the dialog box (the *Preview* field).
The right part of the dialog box (apart from the standard buttons **OK**, **Cancel** and **Help**) contains the following buttons:

- **Default** - pressing this button restores the default (standard) table layout (description style)
- **New** - pressing this button opens the Definition of new steel summary table style dialog box where a new style of a selected table type may be defined (based on the style that already exists)
- **Modify** - pressing this button opens the Modification of steel summary table style dialog box where changes can be made in a selected table type and table style
- **Delete** - pressing this button deletes a highlighted table style from the list of styles available in the *Table style* field.

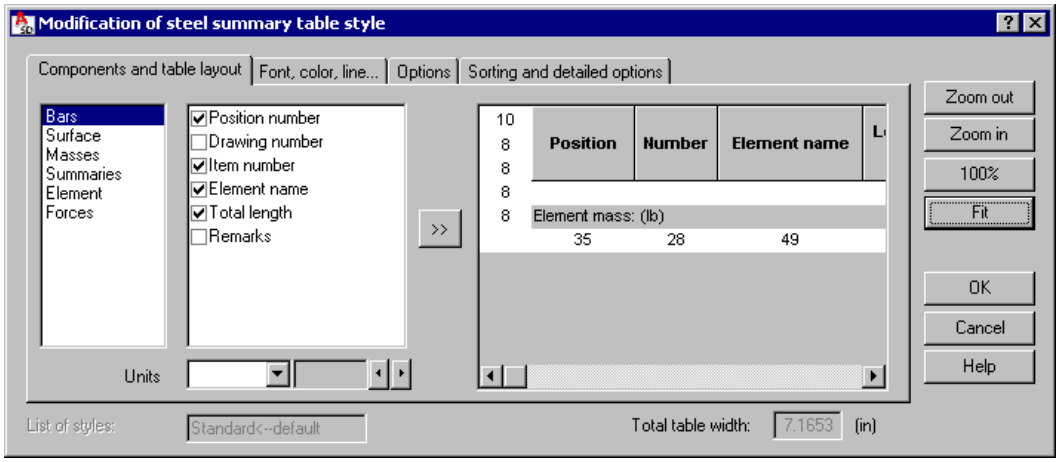
16.2. Definition/modification of steel summary table style

The dialog box is used to define a new style or to modify an existing style of a table type.
The ***Definition of new steel summary table style*** dialog box can be opened after pressing the **New** button in the Steel summary tables dialog box (the ***Modification of steel summary table style*** dialog box can be opened after pressing the **Modify** button).
The dialog box consists of four tabs:
Components and table layout
Font, color, line
Options
Sorting and detailed options.

NOTE:
If one of the steel summary table types (Assembly list) is selected, then there is also another tab Syntax of summary line available.

16.3. Components and table layout

The drawing below presents the dialog box that appears once the *Components and table layout* tab is selected in the Definition/modification of steel summary table style dialog box.



NOTE:
The options located on this tab depend on a table type selected in the ***Steel summary tables - style manager*** *dialog box. The drawing above shows the options available after selecting the element list.*

The bottom part of the dialog box contains the *Style name* edit field; a name of the table style defined should be entered there (when modifying the table style, the *Style name* field is inaccessible).
To define/modify a table style, a user should:

- select a set of table components (e.g. in the dialog box shown above they include: *Bars, Area, Masses, Summaries*)
- in the next field switch on the components to be included in the table (the option is switched on when √ symbol appears)
- press the **>>** button.

The right part of the dialog box presents the defined table layout.

The following options are also available in the above dialog box:

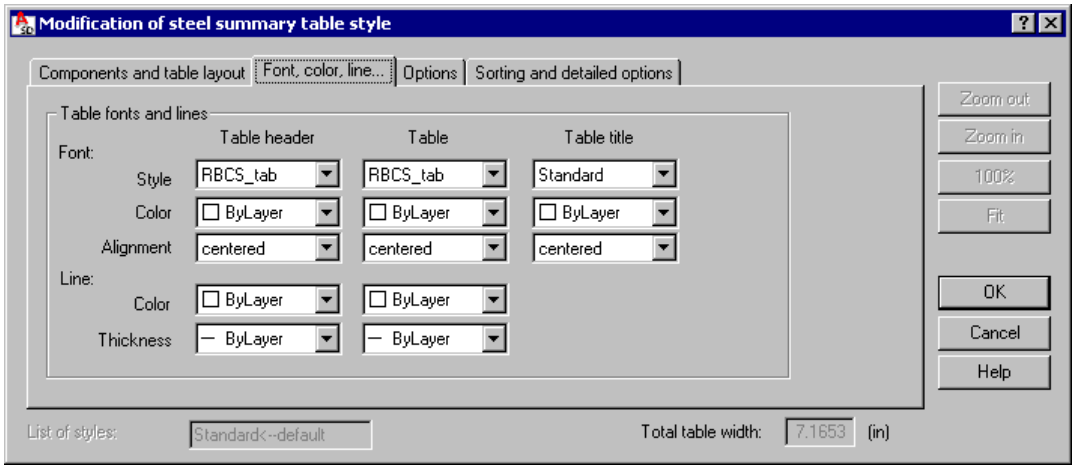
- *total table width* - inaccessible edit field which displays width of the defined table - determined in the program
- buttons: **Zoom in**, **Zoom out**, **100%**, which allow zooming in or zooming out, respectively, the table presented.

The tables enable the user:

- adding blank lines at the beginning and end of a table
- adding blank columns in tables
- exact definition of table dimensions (column width and line height) – dimensions of the column width and the line height are presented under the table and on the left of the table, respectively.

16.4.Font, color, line

The drawing below presents the dialog box that appears once the *Font, color, line* tab is selected in the Definition/modification of steel summary table style dialog box.

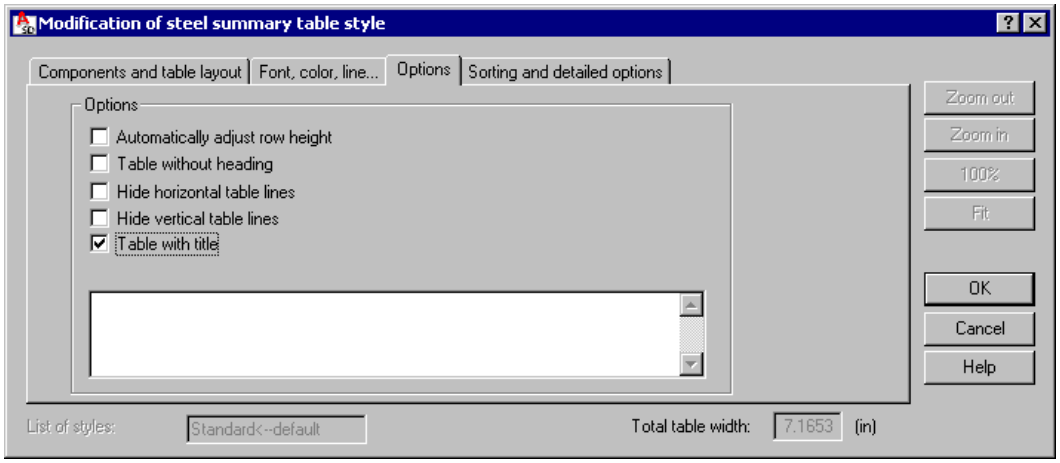


The options available in the above dialog box enable the user to:

- determine a font used in the table (in the table header and all table cells): style, color and alignment of table header and texts in table cells
- select table lines: thickness and color
- define a font used in the table title (if the *Table with title* option is activated on the *Options* tab): style, color and alignment of table title.

16.5.Options

The drawing below presents the dialog box that appears once the *Options* tab is selected in the Definition/modification of steel summary table style dialog box.

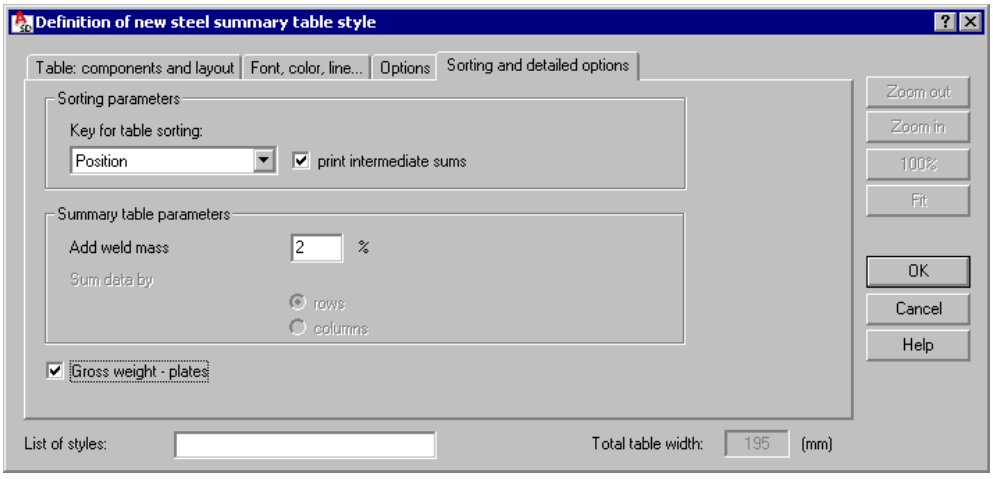


The following options are available in the *Options* field:

- *automatically adjust row height* - if this option is on (√ symbol appears), then height of table rows will be automatically adjusted to the size of symbols of element shapes presented in the table
- *table without heading* - if this option is on (√ symbol appears), then the table generated will not have a header
- *hide horizontal table lines* - if this option is on (√ symbol appears), then horizontal lines will not be shown in a table
- *hide vertical table lines* - if this option is on (√ symbol appears), then vertical lines will not be shown in a table
- *table with title* - if this option is on (√ symbol appears), then the edit field in the lower part of the dialog box becomes accessible and a table title may be typed there; a font used in the table title may be defined on the *Font, color, line* tab.

16.6.Sorting and detailed options

The drawing below presents the dialog box that appears once the *Sorting and detailed options* tab is selected in the Definition/modification of steel summary table style dialog box.



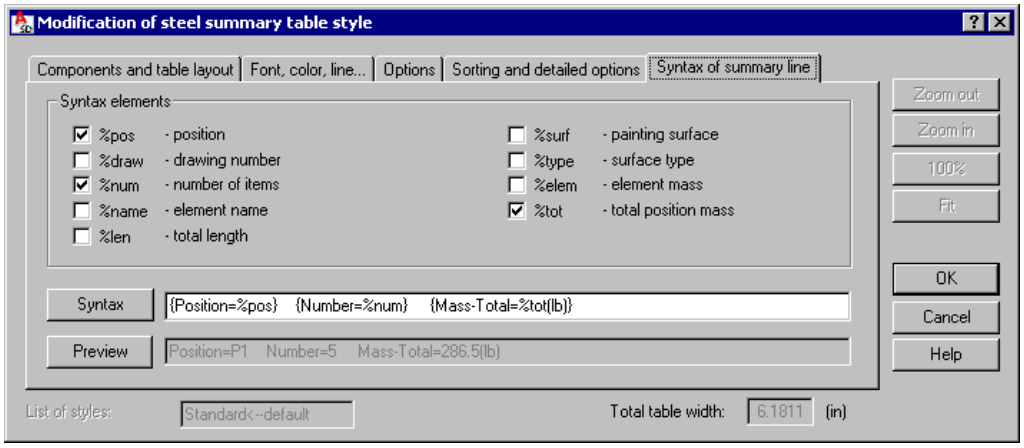
When the *Steel Summary* table is selected, the *Key for table sorting* option becomes accessible on the above tab (in the *Sorting parameters* field). It enables sorting the steel summary table by: position, steel grade or section (by standard the table is sorted by position numbers). Moreover, if

the *Print intermediate sums* option is turned on, then for individual elements of a steel structure (e.g. assemblies) their individual masses are specified (at the end of a table the total mass of all elements is given).
In the *Add weld mass* field the user may define a value (expressed in percentage) that will be added to the total mass (for example, if 2% is given in the *Add weld mass* field, then the total mass will be multiplied by the coefficient 1.02).
At the bottom of the dialog box is the *Gross weight - plates* option. If this option is switched on, then the weight of plates will be calculated as a weight of a plate sheet from which the plate is made (without considering machinings). If this option is switched off, then the weight of plates will be a net weight which means that when calculating the weight, all machinings of plates (chamfers, drilled openings, etc.) will be taken into account.

16.7.Syntax of summary line

The drawing below presents the dialog box that appears once the *Syntax of summary line* tab is selected in the Definition/modification of steel summary table style dialog box.

NOTE:
The tab is available only for one type of steel summary table (Assembly list).

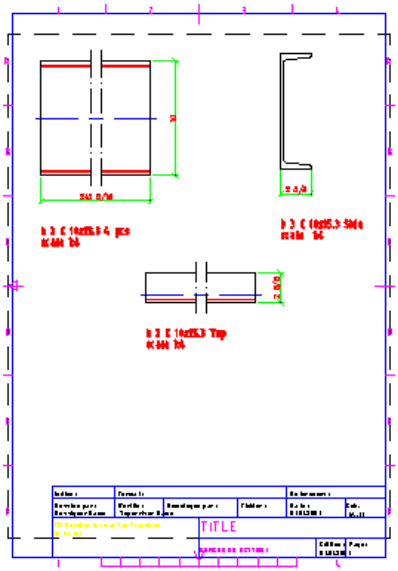


The options provided on this tab are used to define a syntax of the first line (heading) of the assembly list table containing the summary of assemblies; apart from the defined general description style, the above dialog box offers access to the mechanism of the arbitrary composition of syntax and contents of the summary line.
By switching on the relevant description components in the *Syntax elements* field they can be moved to the *Syntax* field. The *Preview* field presents description of the summary line resulting from the syntax defined. This description is based on fixed numeric values saved and responds to changes in the preferences (modification of unit, precision).
The list of variables included in the summary line is as follows:

%pos	- position
%draw	- drawing number
%num	- number of items
%name	- element name
%len	- total length
%surf	- painting surface
%type	- surface type
%elem	- element mass
%tot	- total position mass.

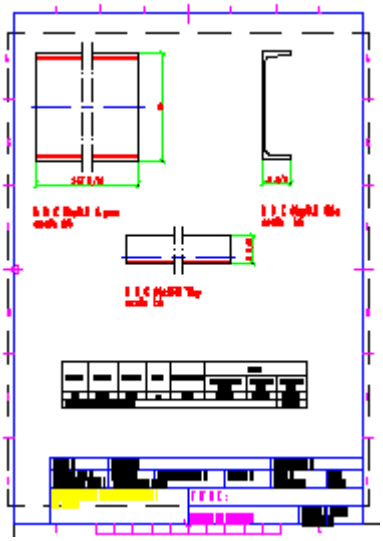
16.8.Example of generating and setting a table style

The example of creating tables of the bill of materials will be illustrated in the drawing for the example of automatic positioning and generating printouts. After positioning and generating printouts for the position *b2,* the drawing presented in the figure below has been obtained.



To generate a table of the bill of materials in the presented drawing, follow the steps below:

- select the menu command: *Steel / Reports / Bill of materials* or press the icon 
- determine the point of table insertion in the drawing; once the point is indicated, the table of the bill of materials is inserted in the drawing



- indicate the table of the bill of materials in the drawing (the table is highlighted), and after pressing the right mouse button select the *Object properties* option from the context menu
- in the ***Modification of selected table*** dialog box modify the table, e.g.: on the *Selection of table style* tab change the height of table rows and on the *Options* tab switch on the option *Table with title,* next, in the edit field in the lower part of the dialog box enter e.g.: *Steel table – position b2*
- press the **OK** button in the ***Modification of selected table*** dialog box; this will result in updating the table.

17. PRINTOUT

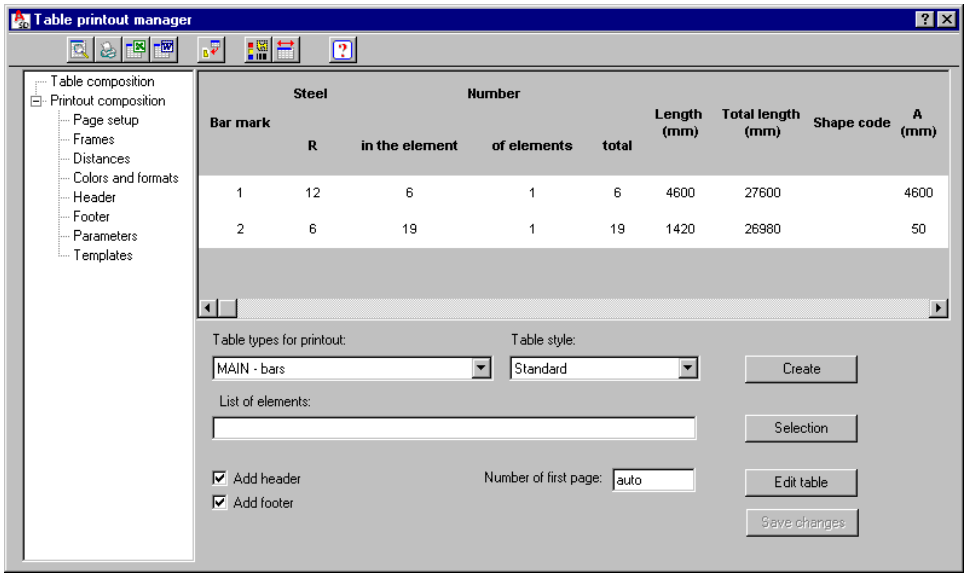
17.1.Table printout manager

The option allows defining/modifying the printout layout for tables used to prepare steel summary tables. The option is available from:

- the menu by selecting the option: *Steel / Reports / Printout* (steel part) or *Reinforcement / Reinforcement table / Table Printout/Export /Edit* (reinforcement part)
- the toolbar by pressing the icon (reinforcement part) or the icon (steel part)
- the command line: RBCS_LISTPRINT (steel part) or RBCR_LIST_EXP (reinforcement part).

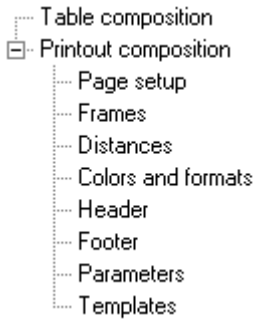
NOTE:
Reinforcement tables in ***AutoCAD Structural Detailing - Reinforcement*** *will be updated automatically after changes are made in a drawing, if in the* ***Options*** *dialog box (Structural Detailing tab) the Automatic table update option is switched on.*

Once the option is selected, the ***Table printout manager*** dialog box, presented in the drawing below, appears on the screen (the dialog box is shown for RC structure elements).



The ***Table printout manager*** dialog box may be divided into two primary parts:

- the left part of the dialog box contains the selection tree (see the drawing below) from which the user, by means of the mouse, selects one of the printout manager options



- to the right of the selection tree there is the part of the dialog box which contains appropriate parameters for the option selected by the user in the selection tree; the dialog box is updated after selecting the option by the user. The top part of the dialog box shows the layout of a selected table type.
- The top part of the dialog box contains a few icons:
  -  *Printout preview* - pressing this icon opens a preview of a table printout; the user may return to the dialog box by pressing the **Close** button
  -  *Print table* - pressing this icon starts printing a table
  -  *Save table* - pressing this icon opens the dialog box in which a table may be saved in the format of MS Excel © program. A table may be saved to:
    ⇒ *.CSV (Comma Separated Values) format files which are text files
    ⇒ *.XLS format files – this saving method fully reflects table settings that can be seen in a preview window
  -  *Save table (MS Word)* - pressing this icon opens the ***Save As*** dialog box which enables saving a table in an MS Word file with the specified name
  -  *Save graphical settings* - pressing this icon enables saving current settings of the printout manager
  -  *Automatic adjust of column width to header text* - pressing this icon results in adjusting the width of table columns to the length of table column names.
  -  *Help* - pressing this icon opens Help.

 *NOTE:*

*For **AutoCAD Structural Detailing - Reinforcement** there is another icon*  *available; when pressed, it enables inserting a table into an edited drawing.*

It should be mentioned here that, although a table shape (cell height, column width) depends directly on a defined table style, the user may freely determine it (i.e. cell height or column width).

In addition, there is a context menu available in the table; it contains the following options:

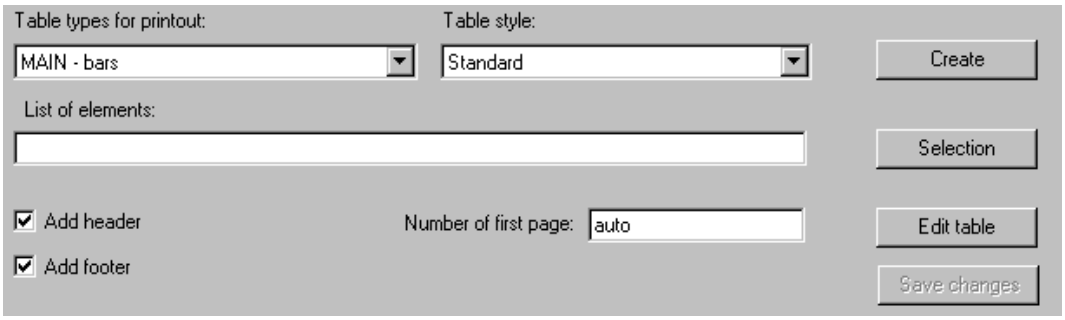
- table cells – the options: group (merges several table cells into one cell) and ungroup
- text orientation – vertical, horizontal
- adding or deleting a column in the table
- adding or deleting a row in the table.

An example of table layout after adding a new column at the beginning of the table, merging several table cells and changing text orientation to vertical, is illustrated in the figure below; the table presents data for a few positions of structure element reinforcement.

Element	Bar mark	Steel		Number			Length (mm)	Total length (mm)
		Grade 36	Grade 60	in the element	of elements	total		
beam 1	1		12	1	1	1	2680	2680
	2		12	1	1	1	2330	2330
column 1	3	6		1	1	1	3190	3190
	4	6		1	1	1	3390	3390

17.2. Table composition

Once the *Table composition* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



For steel structure elements the user may choose one of the following table types:

- Material summary
- Plate summary
- Profile summary
- User parts
- Element list
- Assembly list
- Bolt (rivet) list
- Cut list
- Bolt assign.

For RC structure elements the user may choose one of three table types:

- Main (reinforcing bars)
- Summary (reinforcing bars)
- Detailed (reinforcing bars) - it is necessary to select graphically the distribution varying linearly / surface bar distribution or to enter a number of reinforcement position
- Main (elements) – see the description of the ***Element manager*** dialog box
- Main (wire fabrics)
- Summary (wire fabrics).

For each table type the user may choose a table style defined previously for the selected table type.

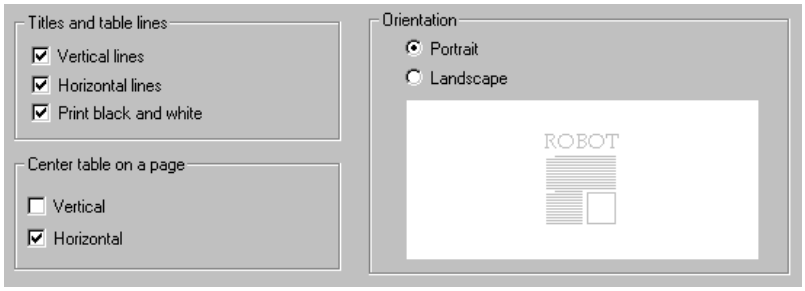
Pressing the **Create** button generates a table of a given type based on the current element list (if the list is empty, then a printout of a whole structure is made) in the format of the active table style (see: Styles applied in tables).

In the *List of elements* field numbers of the elements included in a table are displayed. Pressing the **Selection** button closes the dialog boxes and enables the user to select graphically elements to be included in the table.
Pressing the **Edit table** button enables the user to indicate graphically the table to be edited. Changes made in a table may be saved after pressing the **Save changes** button.
The bottom part of the dialog box contains the following options:

- *Number of first page* - the field in which the user may specify a number of the first page to be printed
- *Add header* - if this option is turned on, then a defined header will appear on a printout
- *Add footer* - if this option is turned on, then a defined footer will appear on a printout.

17.3.Page setup

Once the *Page setup* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

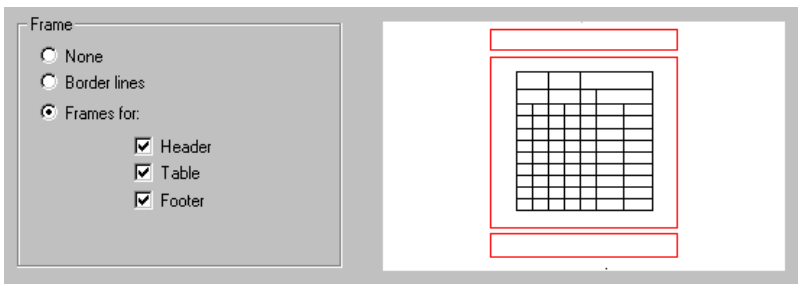


In the above dialog box the user may determine the manner of table presentation:

- in the *Titles and table lines* field:
  - if the *Vertical lines* option is turned off, then no vertical lines are displayed in a table
  - if the *Horizontal lines* option is turned off, then no horizontal lines are displayed in a table
  - if the *Print black and white* option is turned off, then a table is printed with the defined colors applied
- the options in the *Center table on a page* field determine how a table is to be centered (horizontally, vertically or both vertically and horizontally)
- in the *Orientation* field the user may define paper orientation (horizontal - the longer side of a paper sheet is horizontal, vertical - the longer side of a paper sheet is vertical).

17.4.Frames

Once the *Frames* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

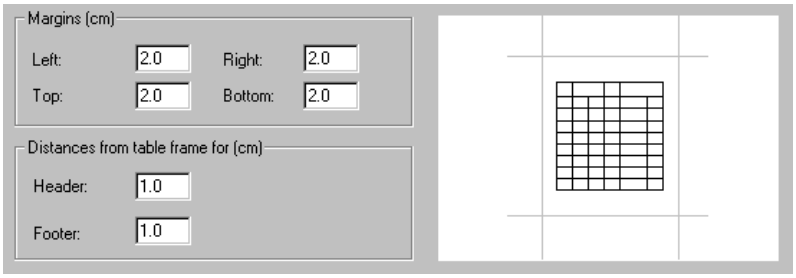


In the above dialog box the user may determine the manner of table presentation on the page:

- no separation between a header/footer and table
- with a border line that separates header and footer from the table
- headers and footers presented in frames (the user may select a frame only for a footer, only for a header, only for a table or combine frames of the elements listed).

17.5.Distances

Once the *Distances* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

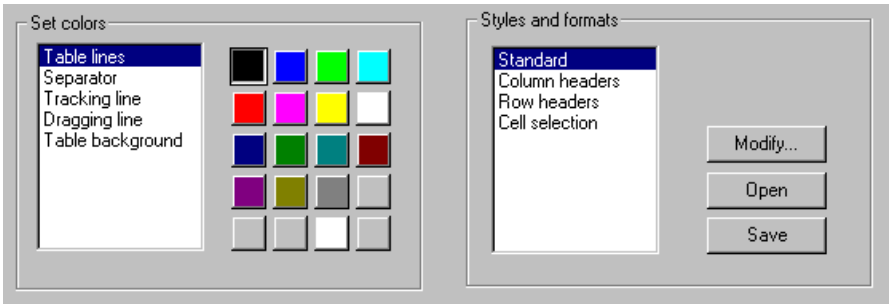


The above dialog box allows determining (identically as in each text editor) page margins: left, right, top and bottom. Moreover, the user may define distances between the table frame and header or footer.

The size of a header and footer is calculated automatically in the program; the parameters mentioned depend on a size of the applied font, size of a drawing with the company logo and number of lines required in a header or footer.

17.6.Colors and formats

Once the *Colors and formats* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.

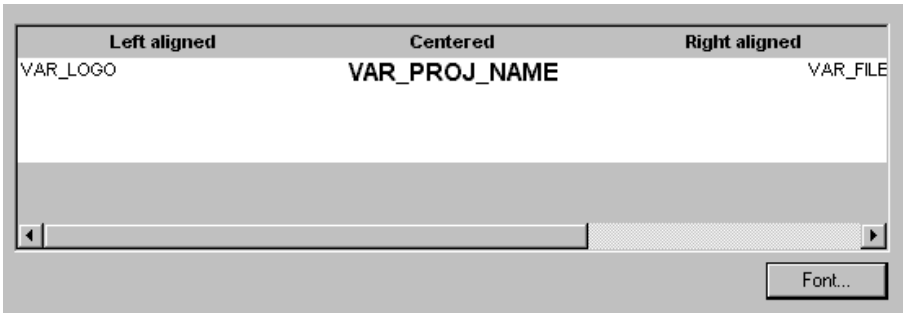


In the *Set colors* field the color of the following table elements may be chosen: table lines, separators, tracking lines, dragging lines and table background.

The *Styles and formats* field allows selection of formats and styles applied in the following table elements: table column headers, table row headers and text contained within a table. Pressing the **Modify** button opens the dialog box where the format (font, font color, alignment method) for the enumerated table elements may be chosen.

17.7.Header

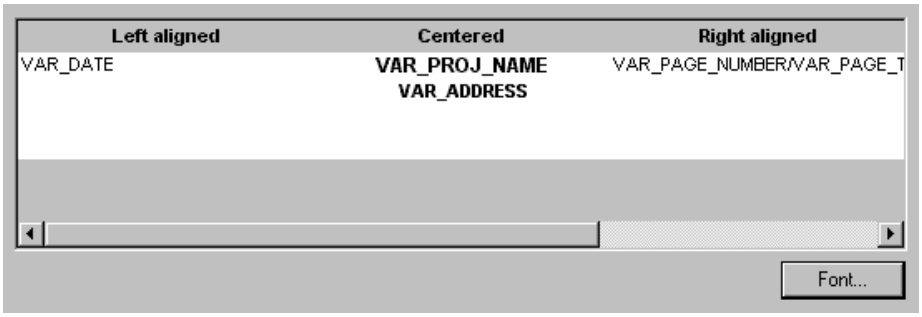
Once the *Header* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box shows the layout of the printout header.
In order to change the header layout, the user should press the table field presenting the header layout; then the list of available variables unfolds, on which an appropriate variable may be selected. When the cursor is positioned in a field of the table presenting the header layout, then pressing the **Font** button opens the dialog box where the user may choose the font to be applied in a selected field.

17.8.Footer

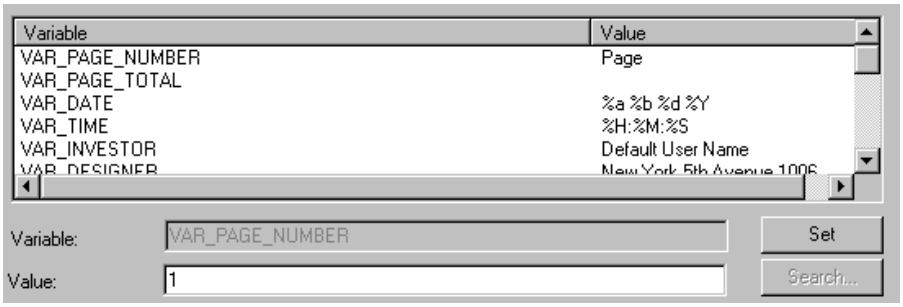
Once the *Footer* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box shows the layout of the printout footer.
In order to change the footer layout, the user should press a field of the table presenting the footer layout; then the list of available variables unfolds, on which an appropriate variable may be selected. When the cursor is positioned in a field of the table presenting the footer layout, then pressing the **Font** button opens the dialog box where the user may choose the font to be applied in a selected field.

17.9.Parameters

Once the *Parameters* option is chosen from the selection tree in the left part of the Table printout manager dialog box, the options presented in the drawing below are provided in the right part of the dialog box.



The above dialog box contains all the variables defined in the system and their names.

NOTE:
*Setting of a variable and next, its modifying **must** be confirmed by pressing the **Set** button.*

The variables are used for formatting header and footer of the printout:
VAR_PAGE_NUMBER - variable that allows assigning a current printout page. A text assigned to the variable will be preceding the total number of pages, if the VAR_PAGE_TOTAL variable is used (e.g. if the "Page **VAR_PAGE_NUMBER**" value is ascribed to the variable, then on the printout each page will be printed in the following form: Page 1, Page 2, etc.)
VAR_PAGE_TOTAL - value of this variable indicates the total number of printout pages. A text assigned to it may be preceded by a current page number, if the VAR_PAGE_NUMBER variable is used.
VAR_DATE, **VAR_TIME -** these variables may be assigned any text and combination of the key words presented below (it enables printing the current date/time on a printout); allowable formats include:
%A - full week day name (Monday)
%a - abbreviated week day name (Mon)
%B - full month name (January)
%b - abbreviated month name (Jan)
%c - standard presentation of date and time
%d - month day (01-31)
%H - time (24-hour clock) (00-23)
%I - time (12-hour clock) (01-12)
%j - successive day of the year (001-366)
%M - minute (00-59)
%m - month (01-12)
%p - local equivalent of the English abbreviations AM / PM
%S - second (00-59)
%U - successive week of the year (first day - Sunday) (00-53)
%W - successive week of the year (first day - Monday) (00-53)
%w - day of the week (0-6, Sunday is denoted by 0)
%X - standard time representation
%x - standard date representation
%Y - year and century
%y - year without the century specified (00-99)
%Z - time zone name
%% - percent mark.
Standard date representation is the following string of variables: %a %b %d %Y
Standard time representation is the following string of variables: %H:%M:%S
Standard date and time representation is the following string of variables: %a %b %d %H:%M:%S %Y.

The remaining variables listed below, do not contain other values than texts ascribed to them by the user. Their names serve only for the purpose of convenient classification while formatting.

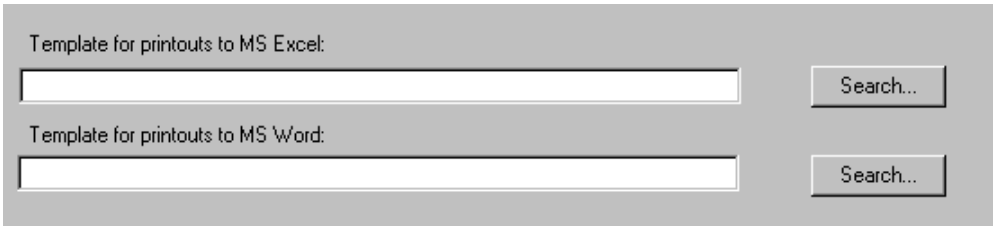
The following variables linked with the printout may be used in ***AutoCAD Structural Detailing - Reinforcement***:
VAR_INV_NAME – investor name
VAR_INV_ADDRESS – investor address
VAR_INV_PHONE – investor phone
VAR_INV_FAX – investor fax
VAR_INV_EMAIL – investor e-mail address
VAR_OFF_NAME – design office name
VAR_OFF_ADDRESS - design office address
VAR_OFF_PHONE - design office phone
VAR_OFF_FAX - design office fax
VAR_OFF_EMAIL - design office e-mail address.
VAR_SCALE – drawing scale
VAR_DRAW_NAME – drawing name
VAR_FILE – name of a DWG file including a drawing
VAR_DESIGNER - designer
VAR_VERIF - verification
VAR_PROJ_NAME, **VAR_PROJ_NUM**
VAR_REV_NAME, **VAR_REV_NUM**
VAR_LOGO – access path to an *.bmp file.
These variables may be also inserted when creating user's own printout layouts. When inserting such a layout the program will automatically fill out variables with values set in the table printout manager.

Moreover, the following variables linked with the printout may be used in ***AutoCAD Structural Detailing - Steel***:

VAR_POS - position name
VAR_SCALE - drawing scale
VAR_PART - part name
VAR_MATERIAL - part material
VAR_LENGTH - part length
VAR_WEIGHT - part weight
VAR_PAINTAREA - painting area
VAR_QUANTITY - number of items of a given position in the project.

17.10. Templates

After selecting the *Templates* option from the selection tree located in the left part of the ***Table printout manager*** dialog box, the right part of the dialog box includes the options shown in the drawing below.

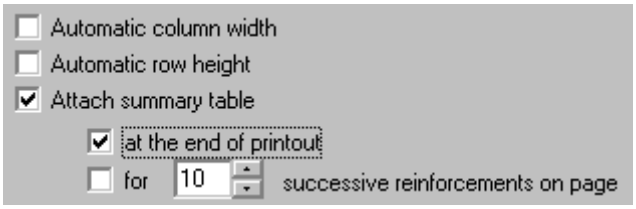


The dialog box above allows selecting templates for printing tables to the following two programs:
- MS Word ©: *.dot format files
- MS Excel ©: *.xlt format files.

In the edit fields the user may specify file names with a full access path; after pressing the **Search** button, it is possible to indicate the template file on the computer hard disk.

In ***AutoCAD Structural Detailing - Reinforcement***, at the bottom of the dialog box there are additional options available for printouts to MS Word©. They allow inserting an additional summary table including information about the reinforcement (the *Attach summary table* option is switched on); it is possible to set the table either at the end of the entire printout, or for several successive reinforcements on a page.
Switching on the following option:

- *Automatic column width* results in widths of table columns being automatically adjusted to fit the length of names of table columns
- *Automatic row height* results in heights of table rows being automatically adjusted to contents of table rows.



17.11. List of commands available in the printout module

In the module that enables printouts the following commands are available:

RBCT_ACTDOC
Available for a selected document located in the option tree; the option's task is to activate - in the edition layout - a selected document (for its edition); there may be many documents created, while only one of them is active, i.e. may be edited - in the edition layout.

RBCT_ADDTOPRINT
Available for a selected view located in the position tree; the layout to which the view is to be added, must be active; the option is intended for composing a final printout; the command adds a selected view to the current layout (printout); NOTE: a view may be contained only on one printout.

RBCT_ADDALLTOPRINT
Available for a selected document provided in the position tree; the layout to which the view is to be added, must be active; the option is intended for composing a final printout; the command adds all the views belonging to a given document (not added to other printout, yet) to the current layout.

RBCT_FITVIEWS
Available for an active document - the edition layout must be active, as well; its task is to provide greater work convenience to the user; the command results in adjusting dimensions of views included in the edition layout to the current size of the AutoCAD ® program window.

RBCT_DELPRINTOUT
Available for a selected printout in the printout tree; the aim of the option is to delete a printout; the command also causes deletion of the layout corresponding to it.

RBCT_REMFROMRINT
Available for a selected view provided in the printout tree; the aim of the option is to remove a view from a printout; the command removes the view only from a printout - it remains in a document and may be reused later on (e.g. by adding it to other printout).

RBCT_DELALLFROMPRINT
Available for a selected document located in the printout tree; it is used to remove views belonging to a selected document from the printout; the command removes the views only from the printout - they remain in a document and may be reused later on (e.g. by adding them to other printout).

RBCT_DELDOC
Available for a selected document located in the position tree; the option's task is to delete a document from the list of documents created for a given position; the command deletes all the drawings belonging to the document.

RBCT_EDITVIEW
Available for a selected view provided in the printout tree; it is used to switch to the edition mode for a selected view; the command enables the user to switch in a simple manner from edition of a printout to edition of a view (drawing) included in it; the command activates the edition layout and a document whose component is the selected view.

RBCT_EDITDOC
Available for a selected document provided in the printout tree; it is used to switch to the edition mode of a selected document; the command enables the user to switch in a simple manner from edition of a printout to edition of views (drawings) included in it; the command activates the edition layout and the selected document.

RBCT_MEDIT_ON
Available for an active document containing one view; it allows editing a document in the model layout.

RBCT_MEDIT_OFF
The command restores the standard functionality of the model layout.

RBCT_REG_LAYER
Available for the active document and view; after adding a layer, the user may change its name by means of the options available in the AutoCAD ® program; the program manages layers within the available views and documents; a layer must be added using the ***AutoCAD Structural Detailing*** options (the problem concerns only the layers used in the edition layout).

RBCT_DELETEPOS
Available for a position selected in the position tree; the option's task is to delete a position; NOTE: only positions defined by the user may be deleted - positions created on the basis of model elements cannot be deleted.

RBCT_RENAMEPOS
Available for a position selected in the position tree; the command enables changing the name of a position.

RBCT_RENAMEVIEW
Available for a view selected in the position tree; the command enables changing the name of a view.

RBCT_RENAMEDOC
Available for a document selected in the position tree; the command enables changing the name of a document.

RBCT_RENAMEPRINTOUT
Available for a printout selected in the printout tree; the command enables changing the name of a printout; a printout name is always identical to the name of a layout (printout) corresponding to it.

RBCT_ACTVIEW
Available for a view selected in the position tree; the command makes the view of the AutoCAD ® program which corresponds to the selected view, become active; NOTE: a view must be a component of the active document.

RBCT_ADDTEMPLATE
The command results in adding a new template; in the dialog box which opens once this option is selected, the user may specify the name of a new template and choose one of the registered template types.

RBCT_DELTEMPLATE
Available for a template selected in the template tree; the command deletes a selected template.

RBCT_RENAMETEMPLATE
Available for a template selected in the template tree; the command allows changing a name of a selected template.

RBCT_ACTTEMPLATE
Available for a template selected in the template tree; the command activates a selected template in the template layout (it enables its edition).

RBCT_ADDVIEWPORT
Accessible for an active template in the template layout; the command adds a view to the active template; in the dialog box which opens once this option is selected, the user may choose view name, scale and type; name and scale of the view may be changed at any time, as regards the type, it cannot be modified.

RBCT_DELVIEWPORT
Accessible for a view selected in the template tree; the command enables deleting a view.

RBCT_APPENDDOC
Available for a position selected in the position tree; the command allows adding a document to the selected position; in the dialog box which opens once this option is selected, the user may specify the name of a document and select a template based on which the document is to be created.

RBCT_CNGSCALE
Accessible for the active view located in the edition layout; the command allows changing the view scale; the scale is expressed as the natural number ***n***, which denotes the scale 1 : ***n***.

RBCT_SETVIEWRANGE
Available for the active view located in the edition layout; the command allows setting the view area visible on the printout; the user selects with a rectangle a part of the view which is to be visible on the printout; it should be remembered that edit operations in the edition layout do not result in modification of a defined print area.

RBCT_REFRESHDOC
Accessible for a document selected in the position tree; the command enables refreshing a selected document; if the element of a structure model for which the position has been defined, changes, then the documents created for this position are refreshed; a document that needs to be refreshed is marked with a red diagonal.

RBCT_REFRESHALLDOC
Available for a position selected in the position tree; the command enables refreshing all the documents belonging to a selected position; if the element of a structure model for which the position has been defined, changes, then the documents created for this position are refreshed; a document that needs to be refreshed is marked with a red diagonal.

RBCT_PRINTVFRAMEON
Available for all defined printout views.

RBCT_PRINTVFRAMEOFF
Available for all defined printout views; areas marked with rectangles (they determine views of the AutoCAD ® program) will not be printed.

RBCT_ADDDETAILVIEW
Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view; once the command is selected, the user should select with a rectangle a part of the active view that is to become a new view; a view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout; a new view is added to the position tree; the new view contains copies of the elements included in the selected rectangle; thus, the created view may be edited independently of the initial view; the command is useful in the situation when it is necessary to have another drawing of the same part to edit it independently or two drawings of different scale.

RBCT_ADDVIEW
Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view; once the command is selected, the user should select with a rectangle a part of the active view that is to become a new view; a view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout; a new view is added to the position tree; the new view contains the same elements as those included in the selected rectangle; if the created view is edited, it also makes changes in the initial view; the command is useful in the situation when several independent views is to be created out of one large view (drawing) which is not added to the printout; all the views created are assigned the same scale.

RBCT_DELVIEW
Available for a view selected in the position tree; the command enables deleting a view that has been added by means of the following commands: RBCT_ADDDETAILVIEW or RBCT_ADDVIEW; NOTE: the original view contained in a template, from which the document has originated, cannot be deleted.

RBCT_SHOWIEW
Accessible for a view selected in the position tree; the command allows edition in the active view; if additional views have been added to the document during edition, then not all the views are visible in the edition template.

RBCT_REGMODELLAYERS
Available for the whole project; the command enables work optimization; once this command is selected, in the dialog box that is displayed on the screen, the user may declare the layers intended for work in the model; while working in the edition layout, layout (printout) or template layout, the layers are frozen - it brings about optimization of the **REGENALL** command operation.

18. AUTOMATIC DRAWING GENERATION

18.1. Automatic drawing generation

The option enables automatic generation of a set of workshop drawings (1 part in one drawing); after completing definition of a steel structure model, the option allows generation of workshop drawings for selected positions. After choosing a position, the user may obtain automatically drawings for single parts or assemblies.

The option is available from:

- the menu by selecting the option: *Steel / Automatic printouts*
- the toolbar by pressing the  icon
- the command line: RBCS_AUTOPRINT.

Once generation of a steel structure model is completed and elements are ascribed position numbers, the stage of generating final drawings takes place. The steps to follow are presented below:

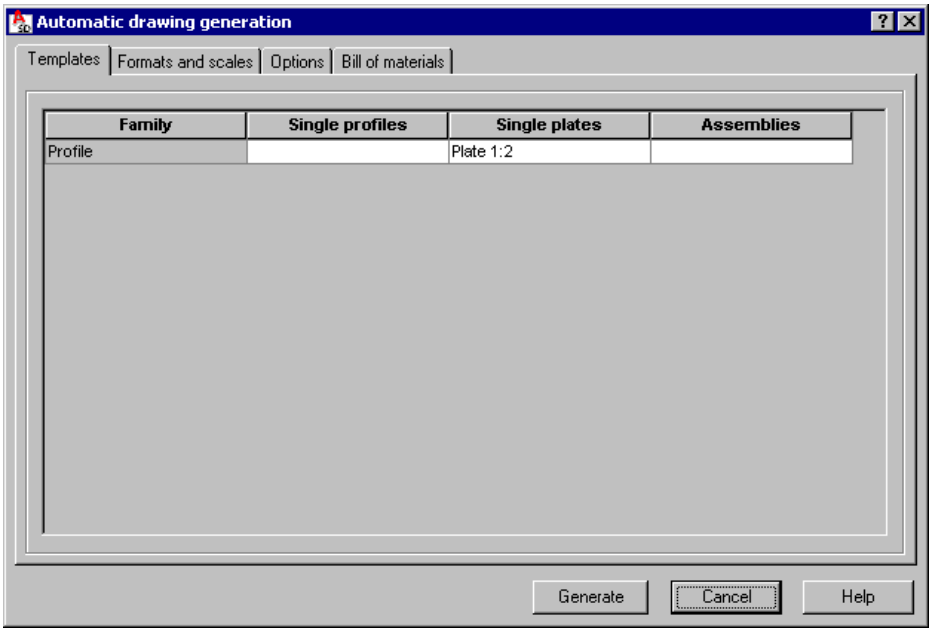
1. select positions for which drawings are to be generated
2. select templates for drawings of single parts
3. select templates for drawings of assemblies
4. select formats and scales
5. determine rules of assigning names to files
6. run automatic generation of final drawings
7. generate a final table.

After selecting structure elements, the program displays the ***Automatic drawing generation*** dialog box that consists of four tabs:
Templates
Formats and scales
Options
Bill of materials.

Pressing the **OK** button starts operation of automatic drawing generation.

18.2. Templates

The dialog box assumes the shape shown in the figure below once the *Templates* tab is chosen in the Automatic drawing generation dialog box.



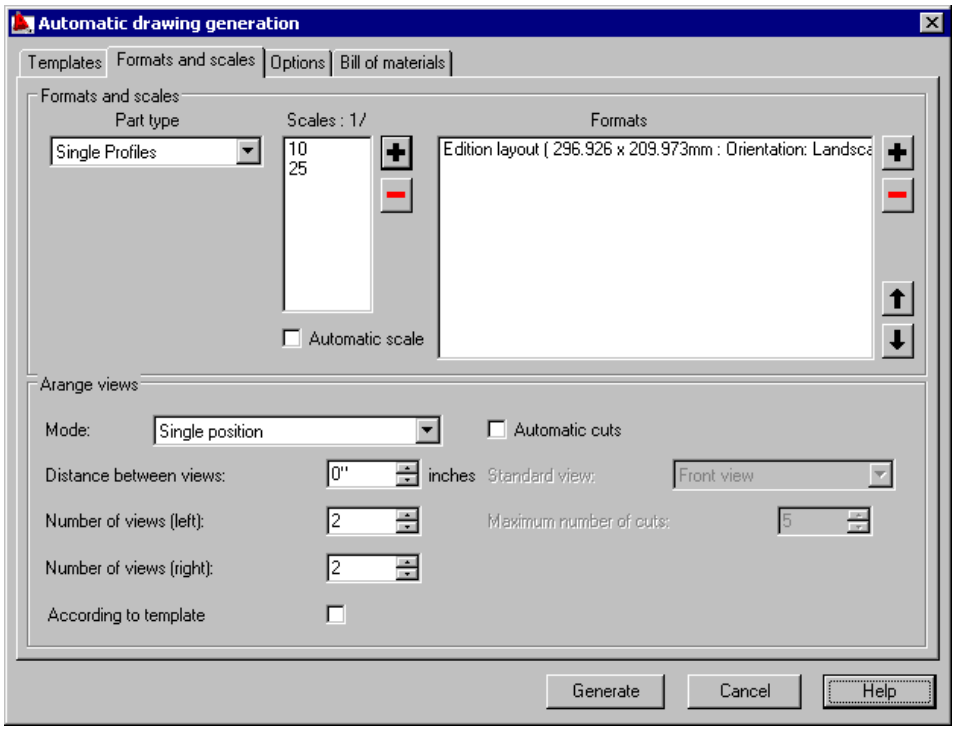
On the above tab the type of a part and a family to which a part belongs are defined.
The first step is to assign a single part template or an assembly template to each family. The user has to assign:

- default single part template for profiles or bent profiles
- default single part template for plates
- default assembly template.

AutoCAD Structural Detailing templates define a set of parameters for each view + set of views and scales.
For automatic generation of drawings a scale for views is not taken from a template, but defined in the dialog box.

18.3.Formats and scales

The dialog box assumes the shape shown in the figure below once the *Formats and scales* tab is chosen in the Automatic drawing generation dialog box.



Fields containing scales and formats refer to an option (object) selected in the *Part type* field.
In the *Scales* field any number of values can be defined: admissible scale values are the values from the interval 0.1 and 500. A new scale value can be defined after pressing the **+** button; pressing the **–** button deletes the currently selected scale value from the list. Under this field the *Automatic scale* option is provided; if it is switched on, then the above *Scales: 1/* field becomes inaccessible and the scale is calculated by the program in such a manner so that the printout layout is filled with drawings optimally.
In the *Formats* field any number of template files can be determined. A new template may be added to the list after pressing the **+** button (then an additional dialog box opens, which enables selection of a *.dwt format file); pressing the **–** button deletes the currently selected template from the list. After choosing a template on the list and pressing one of the ↑ ↓ buttons, the template chosen is moved up and down the list, respectively. While generating views of parts, templates are viewed from top to bottom of the list.

NOTE:
For templates used for automatic printouts it is necessary to set the 'Layout' option in the definition of the plot area in the plot settings.

The *Arrange views* field includes the following options:

- *Distance between views* – distance between views provided on the printout
- *Number of views (left)* and *number of views (right)* – number of views (sections) located in the drawing to the left or to the right of the front view.
- *Mode* – the following options are available:
  - single position – after selecting this option each position is placed in a separate drawing
  - assembly and parts - after selecting this option all the components of an assembly are provided in one printout
  - many positions – after selecting a format, positions are arranged in drawings in such a way so that each drawing contains many positions and all the positions are included in the drawings

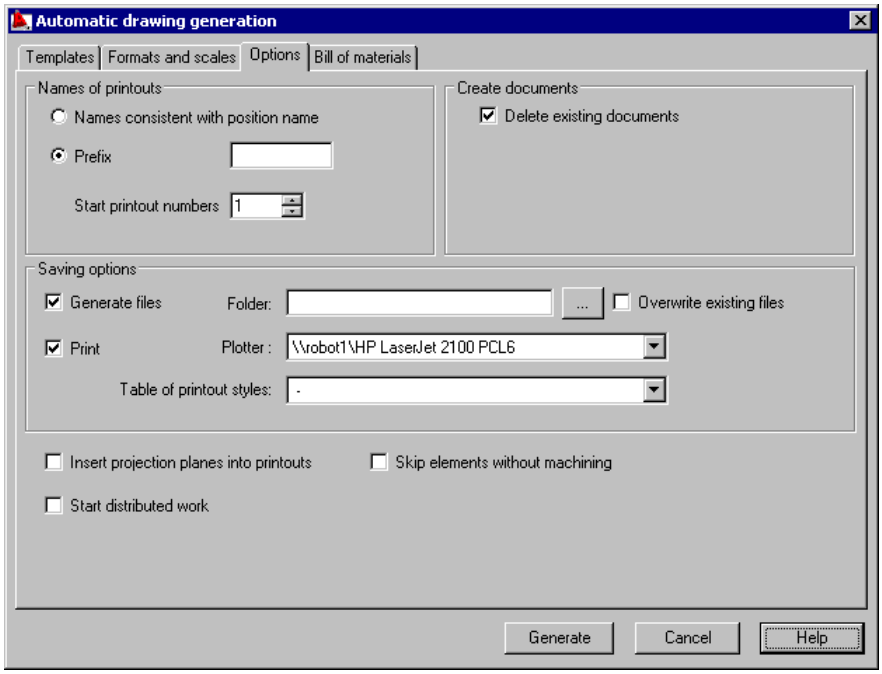
- templates of plate marking (T-line) – in this mode plate drawings will be made in the scale 1:1 and arranged in layouts of a selected format so that the maximum use of drawing space is ensured (positions not corresponding to plates are ignored in this work mode).

After selecting the *According to template* option, the previous options become unavailable. Views are generated based on the template settings.

If the *Automatic cuts* option is selected, cuts are created in views based on the user-specified parameters (standard view and maximum number of cuts).

18.4. Options

The dialog box assumes the shape shown in the figure below once the *Options* tab is chosen in the Automatic drawing generation dialog box.



The *Names of printouts* field enables the user to select a method of defining names of files and layouts; a printout name may be identical to the position name or the user may determine a prefix of a printout name. Below there is an option which allows specifying a page number from which the printout numbering will start.
If the *Delete existing documents* option is switched on, then during generation of drawings all the documents with the specified name included in the existing printouts will be deleted.
In the *Saving options* field the following parameters may be determined:

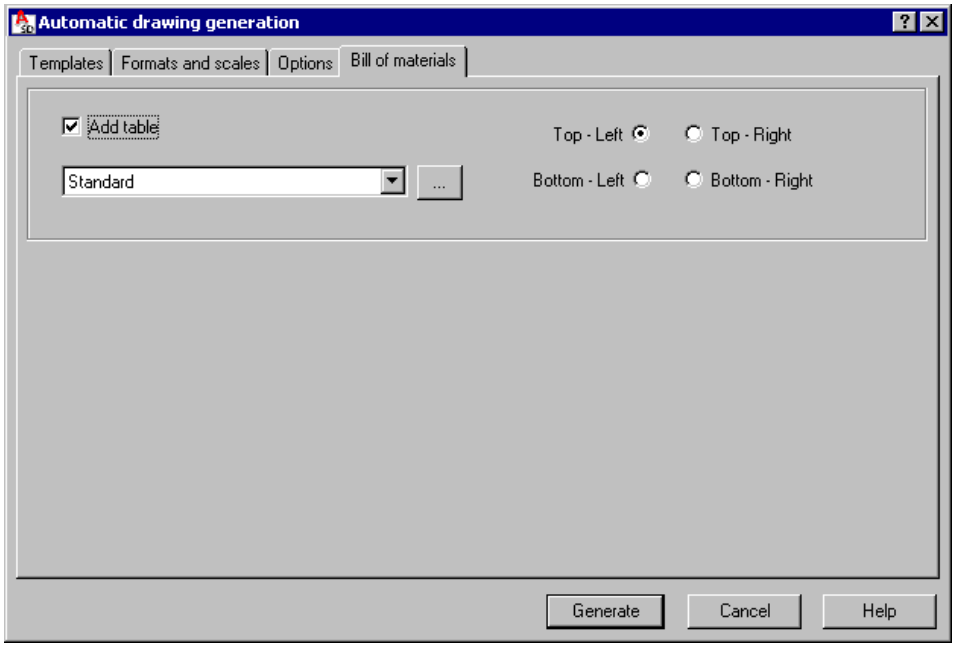
- switching on the *Folder* option enables selection of a folder where DWG files containing single printouts will be located (if the *Overwrite existing files* option is activated, then during generation of drawings all the existing files will be replaced)
- switching on the *Print* option enables obtaining plotter files
- selection list below specifies available printout styles.

If the option *Overwrite existing files without confirmation* is switched on, then files located in a selected folder will be replaced with newly-generated files.
If the option *Insert projection planes into printouts* is switched on, viewports will be inserted into printouts; if the option is switched off, then blocks will be inserted into printouts.

Switching on the *Start distributed work* option allows division of a project into several parts (distributed work on a document); it ensures reduction of size of a file containing project data (quicker loading of a *.dwg file).
If the *Skip elements without machining* option is selected, elements without machining are not presented on printouts.

18.5. Bill of materials

The dialog box assumes the shape shown in the figure below once the *Bill of materials* tab is chosen in the Automatic drawing generation dialog box.



If the *Add table* option is switched on, then a steel summary table will be added to the printout; table location in the printout can be determined by means of the options located to the right:

- top right corner of the printout
- top left corner of the printout
- bottom right corner of the printout
- bottom left corner of the printout.

Pressing the (**...**) button opens an additional dialog box where template of a steel summary table can be chosen.

18.6. Printout templates

Printouts in ***AutoCAD Structural Detailing - Steel*** (printout layouts) can be created based on user-created DWT files which are adjusted to typical formats and include templates of borders and drawing data tables according to user standards. Drawing info tables can be filled automatically with appropriate information specified in the project preferences on the Project Info tab. To do this, provide the following text variables on the Printout layout in appropriate places:

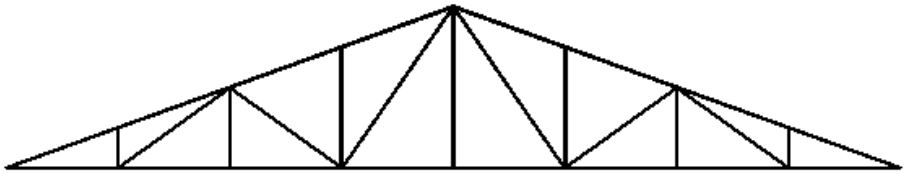
var_office - *Office* field
var_investor - *Investor* field
var_proj_name - *Project name* field
var_address - *Address* field

var_designer - *Designer* field
var_verif - *Verified* field
var_date - *Date* field
var_rev - *Revision* field
var_note - *Note* field.

For the 'automatic printouts' mode you can also use the following variables associated with the printout:
var_pos - position name
var_scale - drawing scale
var_part - name of a part
var_material - material of a part
var_length – length of a part
var_weight - weight of a part
var_paintarea - painting area
var_quantity - number of a given type of items in the project
var_lengthnu - length of a part without specified units
var_prndate - date of generating the printout
var_prndate - time of generating the printout
var_namenu - element name without specified units for plates
var_family - element family
var_surf - surface finishing
var_piecexp - specifies if a given part is included in an assembly.

18.7.Example of creating drawings (printouts)

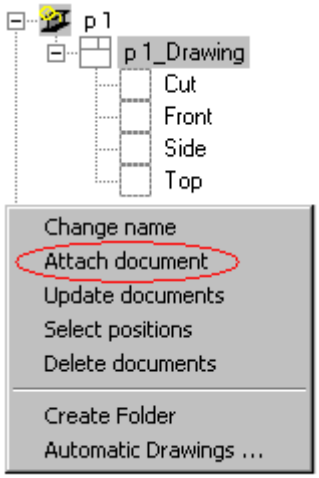
Below is presented the example of creating drawings of a simple truss.



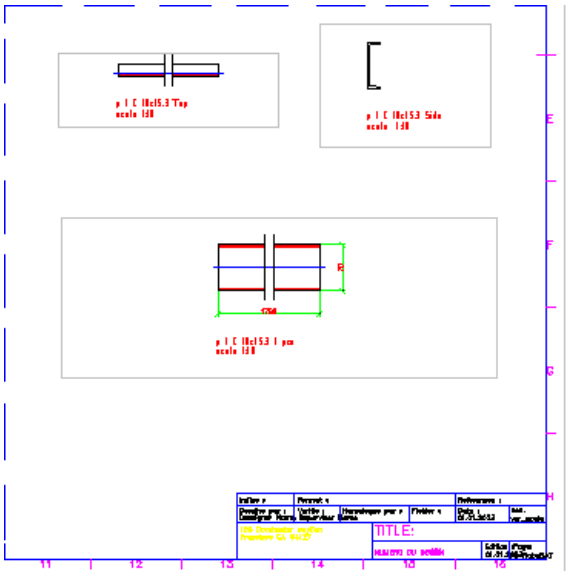
To create a drawing of structure elements, do as follows:

- in the ***Object Inspector*** dialog box, on the *Model* tab select all the truss profiles (the profiles will be highlighted)
- while indicating – with the mouse cursor – the profiles located in the ***Object Inspector*** dialog box, press the right mouse button and select the *Auto positioning* option from the context menu
- in the ***Automatic positioning*** dialog box choose the parameters as follows:
  on the *General* tab
  Positioning level: *single part*
  Prefix: the *By family* option switched on
  Number: numerical format, start from 1, step = 1
  on the *Additional* tab
  all the options switched on
  on the *Numbering* tab
  Sort by: *element shape*
  Sorting criteria: *family*
- press the **Run** button; all the truss elements will be assigned positions according to the parameters determined in the ***Automatic positioning*** dialog box
- move on to the *Positions* tab in the ***Object Inspector*** dialog box

- in the ***Object inspector*** dialog box, on the *Positions* tab select the first position p1 (the position will be highlighted)
- while indicating the first position with the mouse cursor, press the right mouse button; select the *Attach document* option from the context menu (see the drawing below)



- in the ***Select template*** dialog box box select the template *Profile 1:10* that will be used to create documents (projections) and press the **OK** button; in the edition layout the program has generated projections / views for the selected position
- move on to the *Printouts* tab in the ***Object Inspector*** dialog box; add a new printout tab, e.g. *A1 ASD*
- in the ***Object Inspector*** dialog box, on the *Printouts* tab activate *A1 ASD* (indicate the name *A1 ASD,* next, after pressing the right mouse button, select the *Activate* option form the context menu)
- move on to the *Positions* tab in the ***Object Inspector*** dialog box
- while indicating the drawing for the first position with the mouse cursor, press the right mouse button; select the *Add to current Printout* option from the context menu
- indicate the location of the document (projections of elements) in the printout layout on the *A1 ASD* layout; the program has added the selected drawing of the position to *A1 ASD* on the *Printouts* tab in the ***Object Inspector*** dialog box; a part of this drawing is presented below.



18.8.Engineering drawing

18.8.1. Engineering drawing

Use this option to define and modify structure schemes. Access the option using one of the following methods:

- select the Steel menu > Engineering drawings > Engineering drawings
- click  on the toolbar
- type RBCS_SCHEME in the command line.

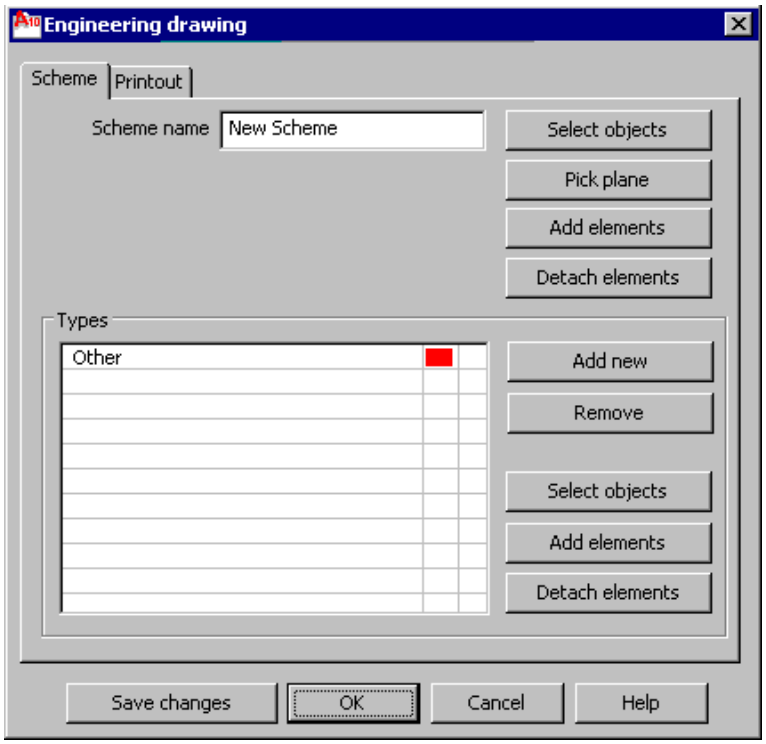
After selecting the option, select elements belonging to the scheme.
After selecting the structure elements, the ***Engineering drawing*** dialog displays; it has the following tabs:

- Scheme
- Printout.

Click **Save changes** to save changes in a defined scheme without closing this dialog.
Click **OK** to save changes in a defined scheme and close this dialog.

18.8.2. Scheme

Select the *Scheme* tab in the Engineering drawing dialog to display the following options.



The *Scheme* tab has the following options:

- *Scheme name* - use it to specify a scheme name; after defining or changing the name, click **Save changes**
- **Select objects** - clicking this button defines or modifies a selection of elements for a scheme; click this button to display the whole structure where you can select elements of the scheme; selected elements are displayed in red
- **Pick plane** - clicking this button defines or modifies the XY plane of the scheme; click this button, next select the main part (it aligns the coordinate system) using one of the options:

UCS - the XY plane is set with respect to the current UCS
WCS - the XY plane is set with respect to WCS
3 points - the XY plane is defined by selecting 3 points

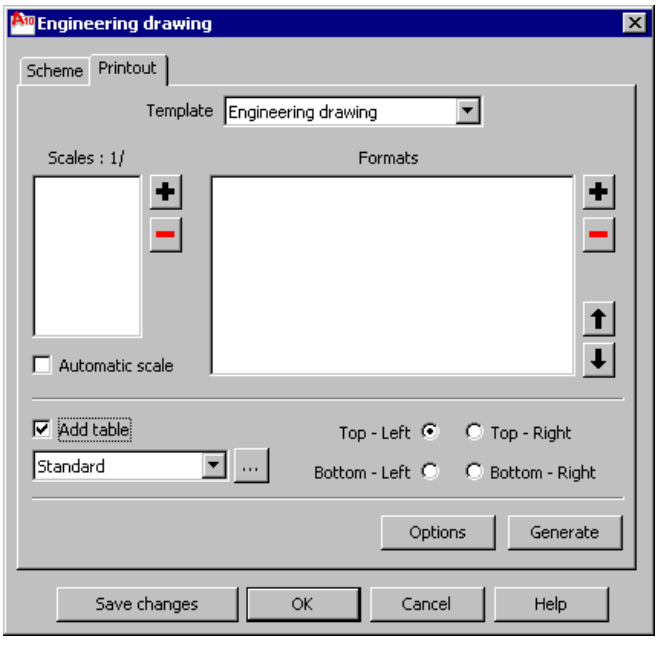
- **Add elements** - clicking this button adds elements to the current element selection; click this button to display the whole structure where you can select elements to be added to the scheme; selected elements are displayed in red
- **Detach elements** - clicking this button removes elements from the current element selection; click this button to display only elements belonging to the scheme; then select elements to be removed from the scheme.

Use the options in the *Types* field to define or modify subgroups in a selected scheme:

- **Add new** - clicking this button defines new types; click **Add new** to define a new type with a default name (which can be changed)
- **Remove** - clicking this button removes a selected type from the list
- **Select objects** - clicking this button defines or modifies a set of elements that belong to a selected type; click this button to display only elements belonging to the scheme; then select elements belonging to a type (available types are: selection, family, position or mark)
- **Add elements** - clicking this button adds elements to a selected type; click this button to display only elements belonging to the scheme; then select elements to be added to the type
- **Detach elements** - clicking this button removes elements from a selected type; click this button to display only elements belonging to the scheme; then select elements to be removed from the type.

18.8.3. Printout

Select the *Printout* tab in the Engineering drawing dialog to display the following options.



At the top of the dialog select a template; a printout is generated based on this template.
You can define any number of values in the *Scales* field; the allowable scale values are those from the interval 0.1 and 500. Click **+** to define a new scale value; click **–** to remove a selected scale value from the list. Under this field is the *Automatic scale* options; if selected, the Scales *1:n*

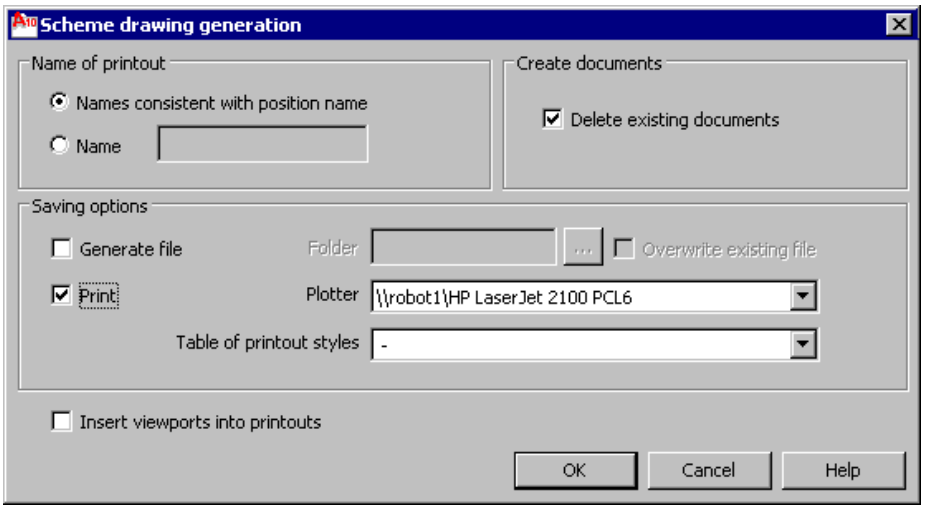
field is not available and the software calculates the scale as to ensure that the printout layout if optimally filled with drawings.
You can define any number of template files in the *Formats* field. Click **+** to add a new template to the list (it opens an additional dialog where you can select a *.dwt file); click **–** to remove a selected template from the list. To move a template up or down on the list, select a template and click ↑ or ↓ , respectively. When views of parts are created, templates are viewed from top of the list to the bottom.
If the *Add table* option is selected, a steel summary table is added to the printout; to specify the position of the table on the printout, select one of the following options:

- Top - Right (corner of the printout)
- Top - Left
- Bottom - Right
- Bottom - Left.

Click (**…**) to open the ***Steel summary tables – style manager*** dialog where you can select a steel table template.
Click **Options** to open a dialog with options used for generating a scheme drawing.
Click **Generate** to start generating a scheme drawing.

18.8.4. Generating a scheme drawing

Click **Options** in the Engineering drawing dialog to display the following dialog.



The *Name of printout* field allows you specify the way of defining names of files and tabs; you can define a printout name identical to the position name or specify a prefix for a printout name. Below specify a number from which to start numbering the printout pages.
If the *Delete existing documents* option is selected, all documents with the specified name are deleted from the existing printouts when generating drawings.
Specify the following parameters in the *Saving options* field:

- *Folder* – select this option to select a folder where DWG files with single printouts are located (if *Overwrite existing file* is selected, all existing files are replaced when generating drawings)
- *Print* – select this option to select a plotter file type
- the selection list includes available printout styles.

After selecting *Insert viewports into printouts*, viewports are inserted on printouts; if this option is cleared, blocks are inserted on printouts.

19. LINK WITH OTHER PROGRAMS

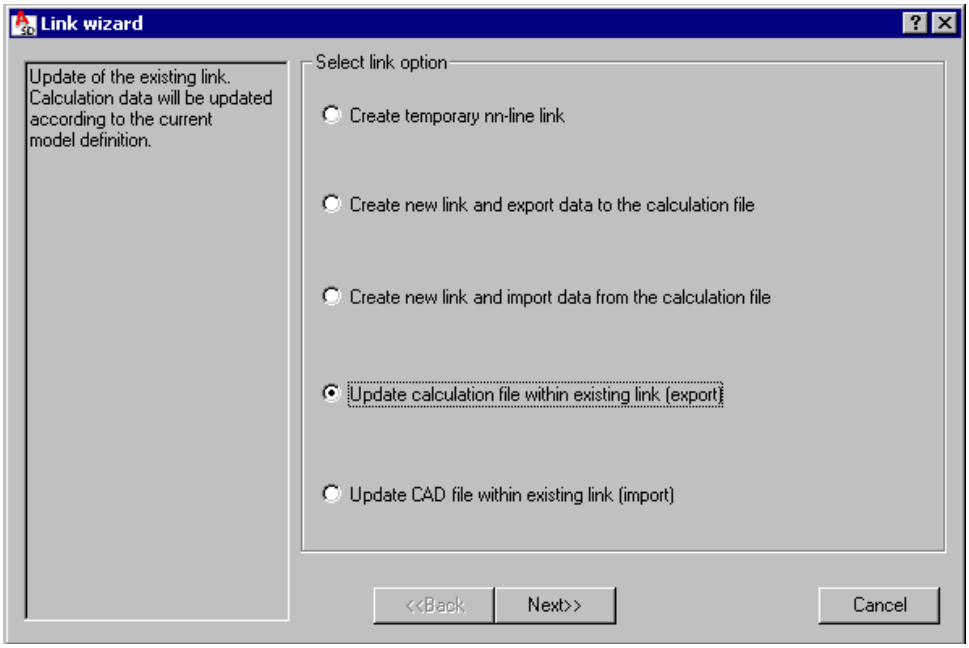
19.1. AutoCAD Structural Detailing - Robot link

The link between two programs (***Autodesk Robot Structural Analysis*** - the calculating program and ***AutoCAD Structural Detailing*** - the program used to create drawings) has been designed in order to enable the user to:

- perform calculations of a defined CAD 3D model by means of ***Autodesk Robot Structural Analysis*** at any work stage; calculations may be performed both for a whole structure and for a structure part selected by the user
- apply a calculation model created in ***Autodesk Robot Structural Analysis*** as the basis for creation of a detailed CAD model and documentation of a designed structure
- perform full synchronization of changes made in a structure calculation model as well as in its CAD model; synchronization pertains to both directions (from ***Autodesk Robot Structural Analysis*** to ***AutoCAD Structural Detailing*** and vice versa).

The operation of updating data of the CAD file and file containing the structure calculation model is carried out in the ***Link Wizard*** dialog box shown in the drawing below. The option is available from:

- the menu by selecting the *Steel / Static analysis - Robot* option
- the toolbar by pressing the  icon
- the command line: RBCS_R2R.



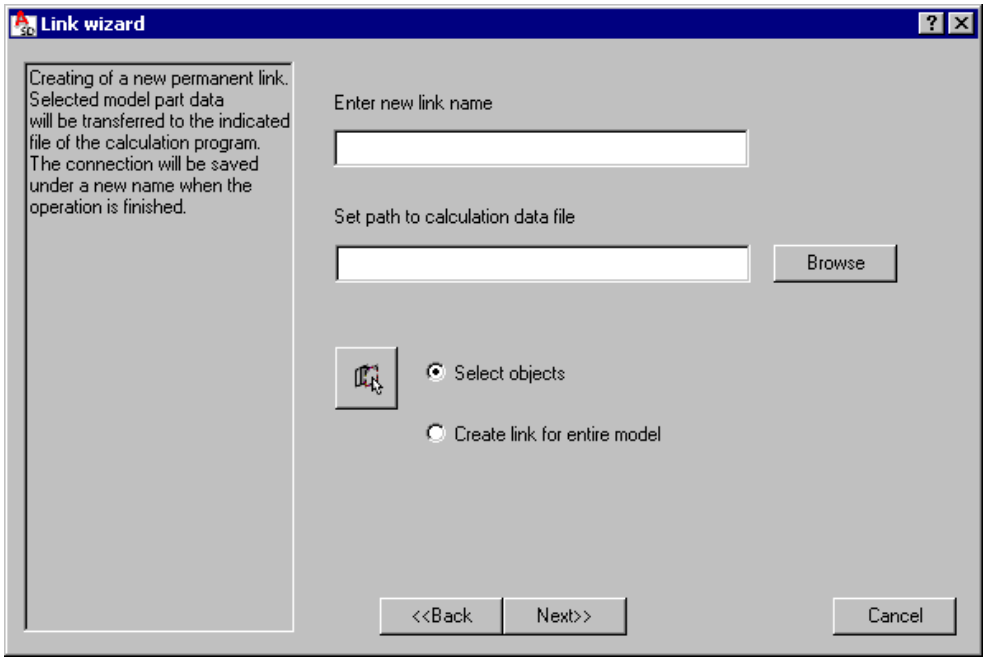
In this dialog box the data update is conducted for a selected ***ROBOT*** - ***AutoCAD Structural Detailing*** connection defined in the current CAD file. To start with, the user should choose one of five operations available in the dialog box:

- first three options provided in the above dialog box are used to create a new link (permanent or temporary one)
- last two options are used for data update for a link that already exists (update of a structure calculation model based on changes made in ***AutoCAD Structural Detailing*** or vice versa).

The appearance of the above dialog box depends on a selected option; in the left part of the dialog box there appears a short description of the option selected in the above dialog box.

After pressing the **Next** button, the ***Link wizard*** dialog box (see the drawing beneath) changes, thus allowing the user to:

- assign a name of the defined link or select a name of the existing link
- create or select a file with an RTD extension (a file of the ***Autodesk Robot Structural Analysis*** calculation program) in which calculation results will be saved
- select objects that will be considered while creating a link.



Creation of a new link

In the *Enter new link name* field the user may determine a link name - this field is accessible only while creating a permanent link. The link is saved under a specified name in the current DWG file. The name identifies links, therefore, it has to be unique in a given file.

In the *Set path to calculation data file* field the user should indicate a calculation file of an RTD extension (the ***Autodesk Robot Structural Analysis*** file) to which calculation results for the created link will be saved. This field is accessible only when a new permanent link is being created. While data is being updated, the path is presented in the field, however, it is displayed only for informative purposes, since the path cannot be changed. The file may be indicated, as well, after pressing the **Browse** button. Then one of the below-listed standard dialog boxes of the Windows system appears on the screen:

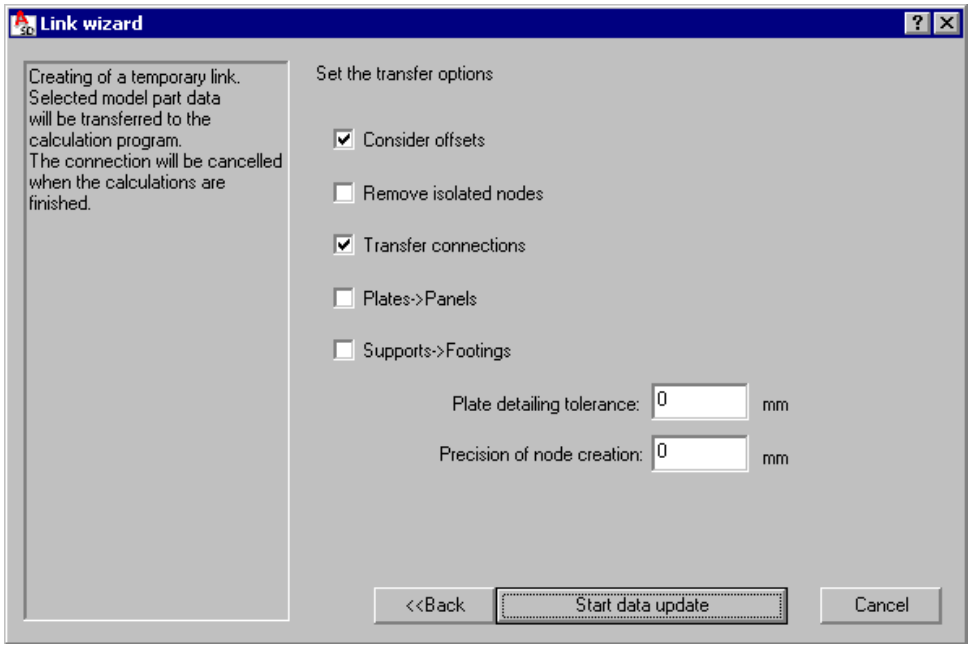
- the ***Open*** dialog box - when importing data from an existing file that contains a calculation model (RTD file)
- the ***Save*** dialog box - when exporting data to a new file which is being created at this stage.

The options provided in the bottom part of the ***Link wizard*** dialog box enable selection of objects that are to be link components: it is either the entire structure (the *Create link for entire model* option) or structure part (the options are available only when a new link is being created). If the *Select objects* option is chosen, then the button becomes accessible which allows on-screen graphical selection of objects that are to be included in a link.

Data update for the existing link

From the *Choose link* field the user may select one of the permanent links available in the current DWG file; once selected, the link become active. The list of all permanent links defined in the current DWG file may be viewed (and configured, if needed) in the additional dialog box that opens after pressing the **See list** button.

After pressing the **Next** button, the dialog box changes (see the drawing below), thus allowing the user to set parameters of data transfer. The options contained in this dialog box depend on the option (direction of data update) chosen in the first dialog box.



In the above dialog box:

- for a new link the default option set is activated
- for the link already existing, these options, that were applied for the selected link during the last operation of data update, are activated.

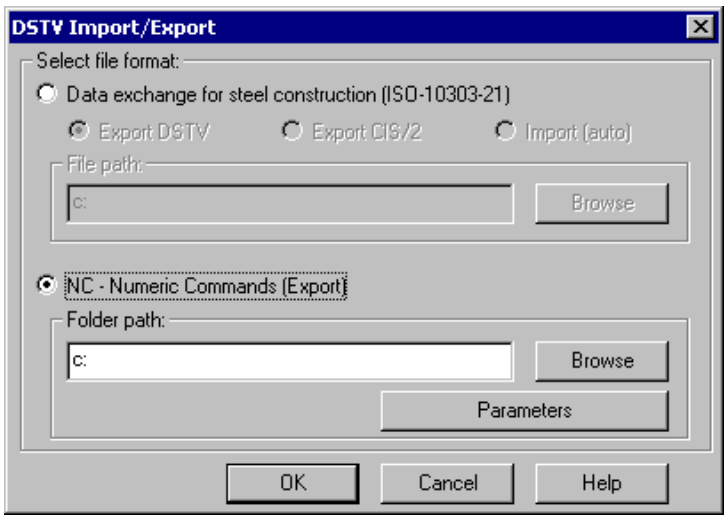
Naturally, before the data update is started, the options located in the above dialog box may be changed freely.
The bottom part of the dialog box contains the **Start data update** button. Its pressing results in creating a link and starting data update based on the options selected in the above dialog boxes. Data update is an obligatory operation while creating a new link. Pressing the **Start data update** button closes the above dialog box; then the dialog box containing a report on the performed operation of data update is displayed on the screen.

This dialog box presents the run of the data update process. In the bottom part of the dialog box the bar is displayed which shows the progress of the data update operation and the messages appearing during the data update process are presented. Information concerning the update includes: date of the update process, file names (source file and target file), direction of the data update process (from ***AutoCAD Structural Detailing*** or to ***AutoCAD Structural Detailing***) as well as all warnings that may be displayed during the data update process.
The contents of the dialog box with a report on the data update process may be saved in a text file; it may be done after pressing the **Save log file** button. A text file may be corrected manually by the user.
The **OK** and **Save log file** buttons are available only after the data update process is completed.

19.2. DSTV import/export

The option enables reading or saving a DSTV format file. It is accessible from:

- the menu by selecting the *Steel / DSTV data export* option
- the toolbar by pressing the  icon
- the command line: RBCS_DSTV.

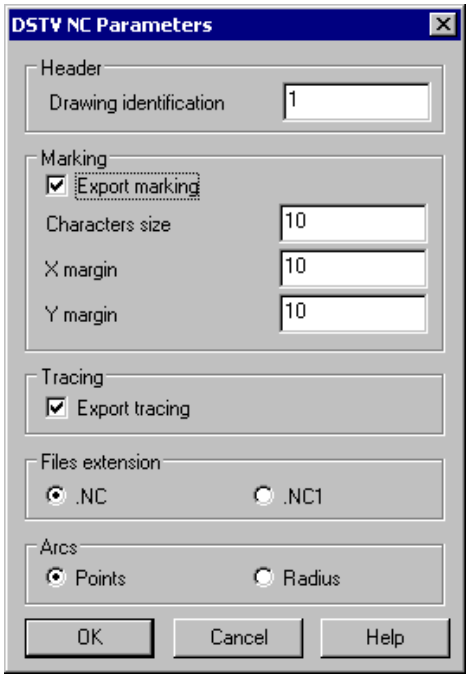


In the current program version two possibilities of importing or exporting DSTV or CIS/2 format file are available:

- a file containing description of a steel structure model (a file that is compliant with the ISO-10303-21 standard can be both imported and exported); in the *File path* field the user should specify the file location on disk (file location can be indicated, as well, after pressing the **Browse** button); this import/export mode results in generation of one file with *.stp extension including description of a structure model
- files containing descriptions of steel structure parts to be applied when numeric commands are used (the files can only be exported); in the *Directory path* field the user should specify location of the folder in which files with description of structure parts will be saved (folder location can be indicated, as well, after pressing the **Browse** button); in this mode - for each part of a steel structure that has been assigned a position – a separate file containing description of parts is created in the indicated folder. Pressing the **Parameters** button opens the DSTV NC Parameters dialog box.

19.3. DSTV NC Parameters

The ***DSTV NC Parameters*** dialog box is used for definition of additional parameters of pointing and marking. It opens after pressing the **Parameters** button in the DSTV Import / Export dialog box.



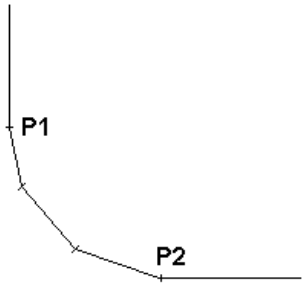
In descriptions of steel structure parts, used in the numeric control of machine tools, it is possble to provide the following blocks which are concerned with:

- header – a drawing will be identified by means of a specified text / number
- marking – if this option is switched on, the marking block will be included in the exported file; apart from that, the following parameters of marking a steel element may be defined: size of characters and position of the point where a description will be inserted (X margin and Y margin)
- pointing - if this option is switched on, the pointing block will be included in the exported file.

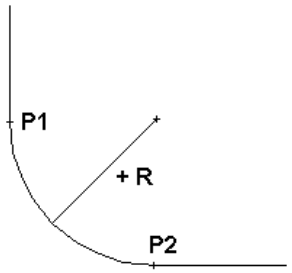
In the lower part of the dialog box the user may select the following parameters:

- file extension (NC or NC1)
- method of arc definition in a file for the numeric control of machine tools:

  by means of ponts positioned on the arc; the arc will be approximated between the beginning point P1 and the end point P2 by segments between points on the arc

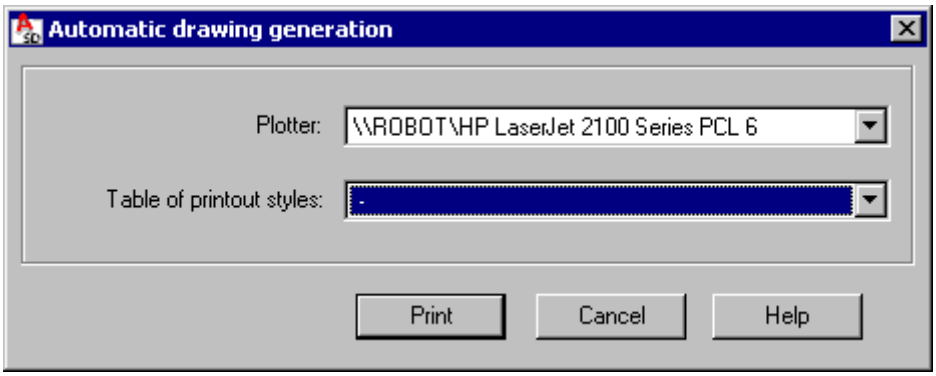


by means of the arc radius; between the beginning point P1 and the end point P2 the arc will be defined based on the radius value.



19.4. Automatic drawing generation - saving to plotter

The option enables automatic generation of workshop plotter drawings. The option is accessible from the toolbar by pressing the *Automatic plotting* icon.



The following parameters can be determined in the dialog box above:

- the *Plotter* selection list enables choosing a type of plotter file
- the selection list below contains a list of available printout styles.

Pressing the **Print** button starts generation of drawings by a selected plotter type.