

What's new in WiseMechanical 4

Speed

Speed in WiseMechanical has increased 1.5-2 times. All functions were optimised – from the file opening to the model update: speed of file loading, model regeneration, command execution for many objects.

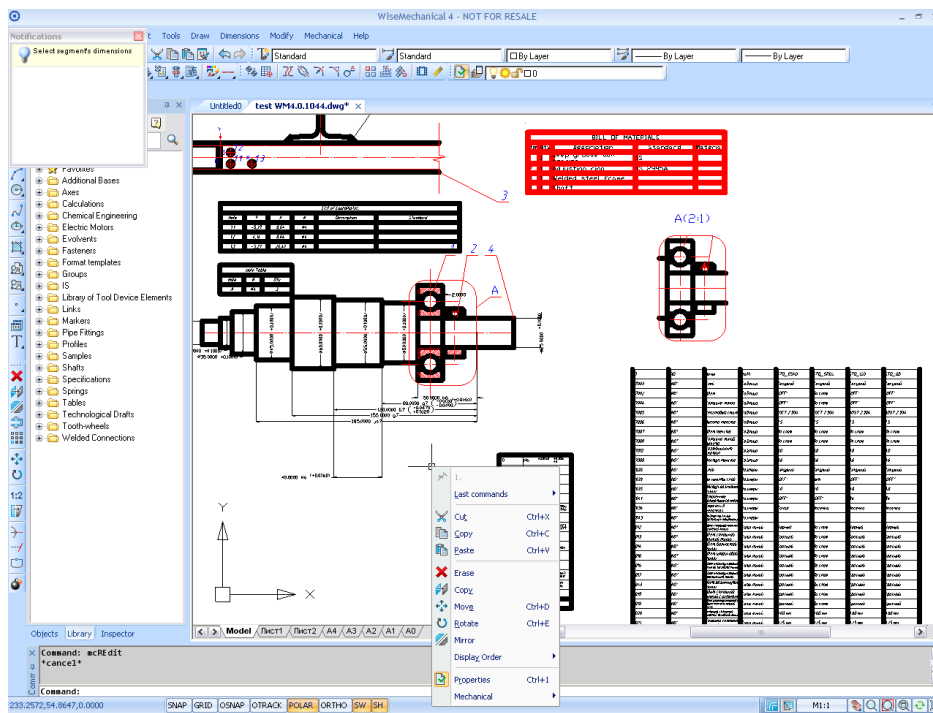
Support of .dwg 2010

WiseMechanical supports format of AutoCAD 2010 drawing files. This allows interaction with all modern solutions supporting the *.dwg files and also Autodesk solutions – AutoCAD 2010/2011/2012.

Using the clipboard to support AutoCAD, you can easily transport data from one environment to another.

Program interface

WiseMechanical was rebuilt and is close to the interface of classic CAD: the location of menu items and toolbars, commands in the command line.



WiseMechanical is a system for drafting; here is why its interface was totally reorganised to optimise everyday workflow. The height of the command line, the number of toolbars on the screen was decreased; the right button menu was optimised.

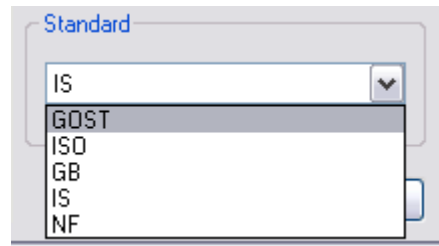
Selection of settings profile

At the installation of the program you can select corresponding standards for drawing preparation:

Furthermore, you can change a standard in the **Mechanical>Settings** dialogue.

Default drafting standard for WiseMechanical 4.0:

ESKD
GB
IS
ISO
NF



New commands and aliases

```
45 ARRAI=sARRAI
46 ATT=sATTDEF
47 ATTDEF=sATTDEF
48 ATTEDIT=sATTEDIT
49 ATTEXT=sATTEXT
50 B=sNEWBLOCK
51 BATMAN=sATTEDIT
52 BEDIT=sBEDIT
53 BHATCH=sHATCHCMD
54 BLOCK=sNEWBLOCK
55 BMOD=sNEWBLOCK
56 BC=sBOUNDARY
57 BOUNDARY=sBOUNDARY
58 BOX=s3D_BOX
59 BREAK=sVCBREAKCMD
60 BS=sSAVEDOCUMENT
61 C=sCIRCLE
62 CHA=sCHAMFER
63 CHAMFER=sCHAMFER
64 CIRCLE=sCIRCLE
65 CIRCLE2=sCIRCLEBYDIAMETER
```

There are more than 500 commands and aliases which allow functions of the program to be performed using the command line. You can find most of the familiar commands: RE, REC, LINE etc.

Toolbar «Object snap»

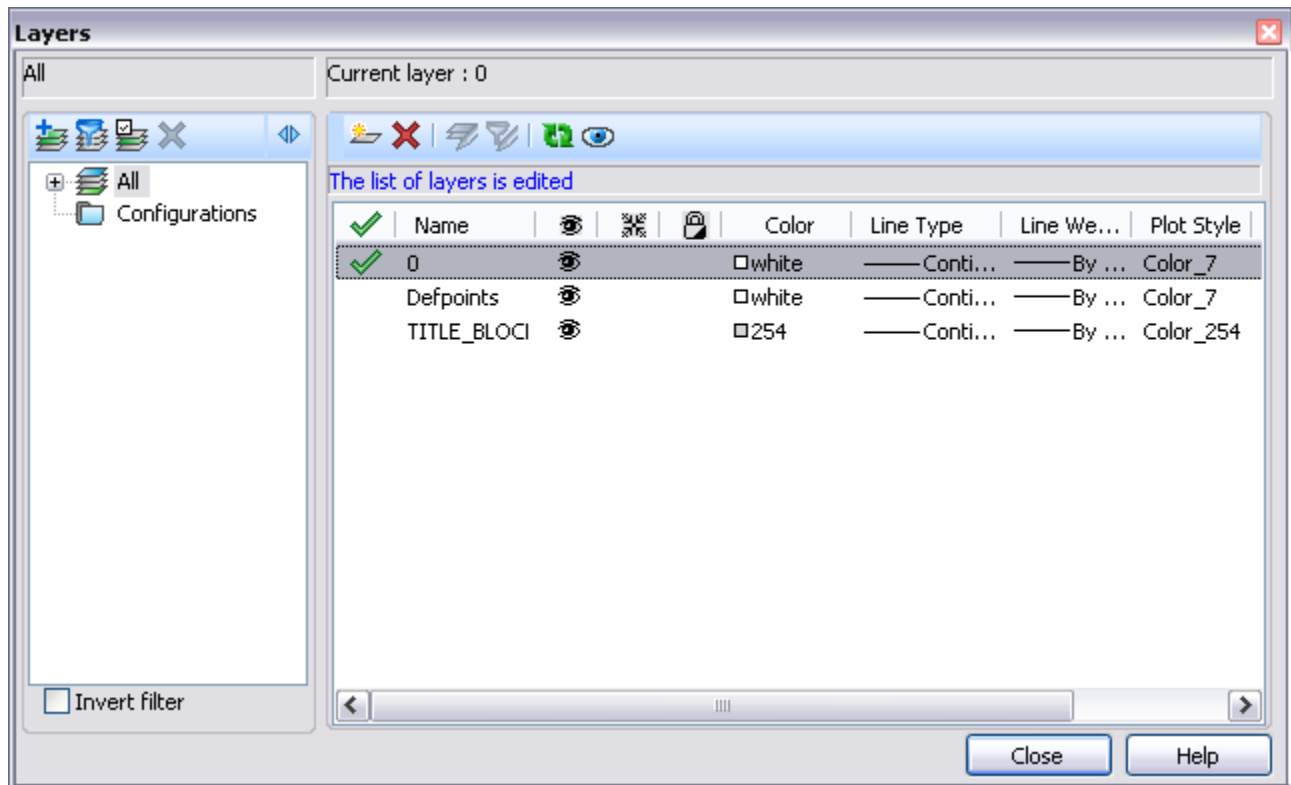
The **Object snap** toolbar allows the switching on/off of different types of snaps.



To open the toolbar, select **View > Main Toolbars > Snap** or from the context menu of the OSNAP button in the status bar.

Layer manager

The layer manager was seriously reworked. There is a category tree which allows the selection of all layers, used layers, layers of external reference, a group of layers, a filter of layers or a configuration of layers. There is a list of the selected element's layers in the right part of the manager.

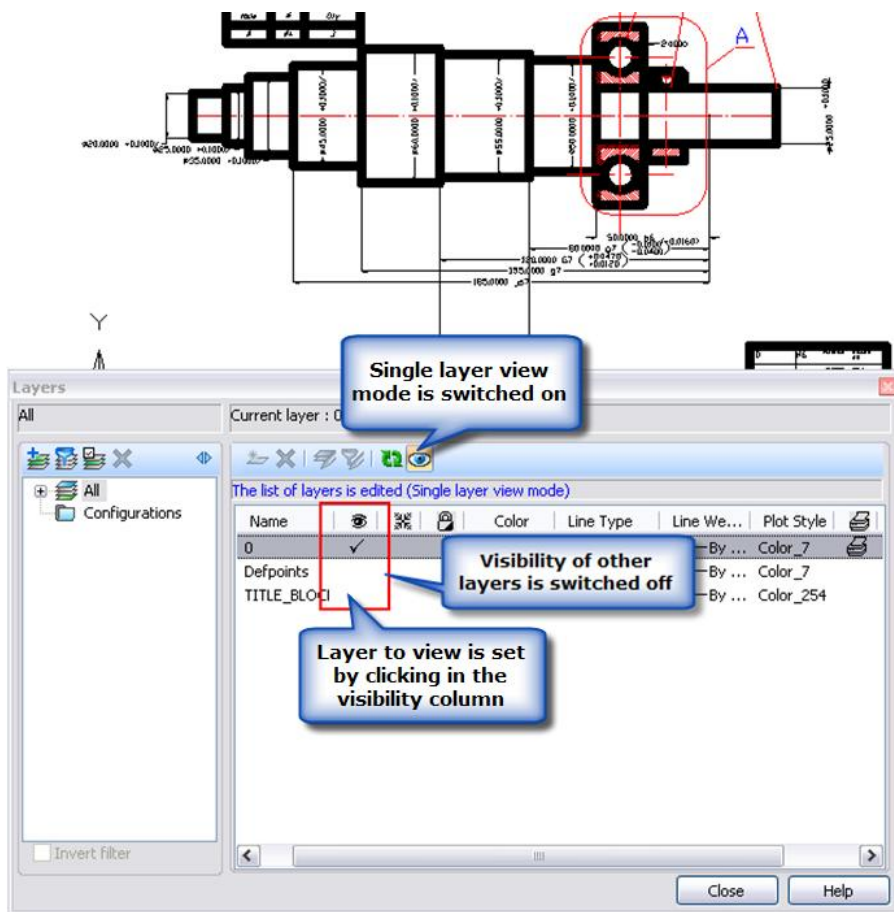


A drawing can be zoomed and panned when the layers manager is opened.

Single layer view mode

It is not always useful to switch off the visibility of the layers in the drawing (☑ icon). If there are a lot of hidden layers, it is difficult to display the content of each layer, because you have to set the previous state of visibility for all layers.


Single layer view mode addresses this problem. The visibility of all layers, except the one selected, is switched off in this mode, but is switched on when you close this mode.



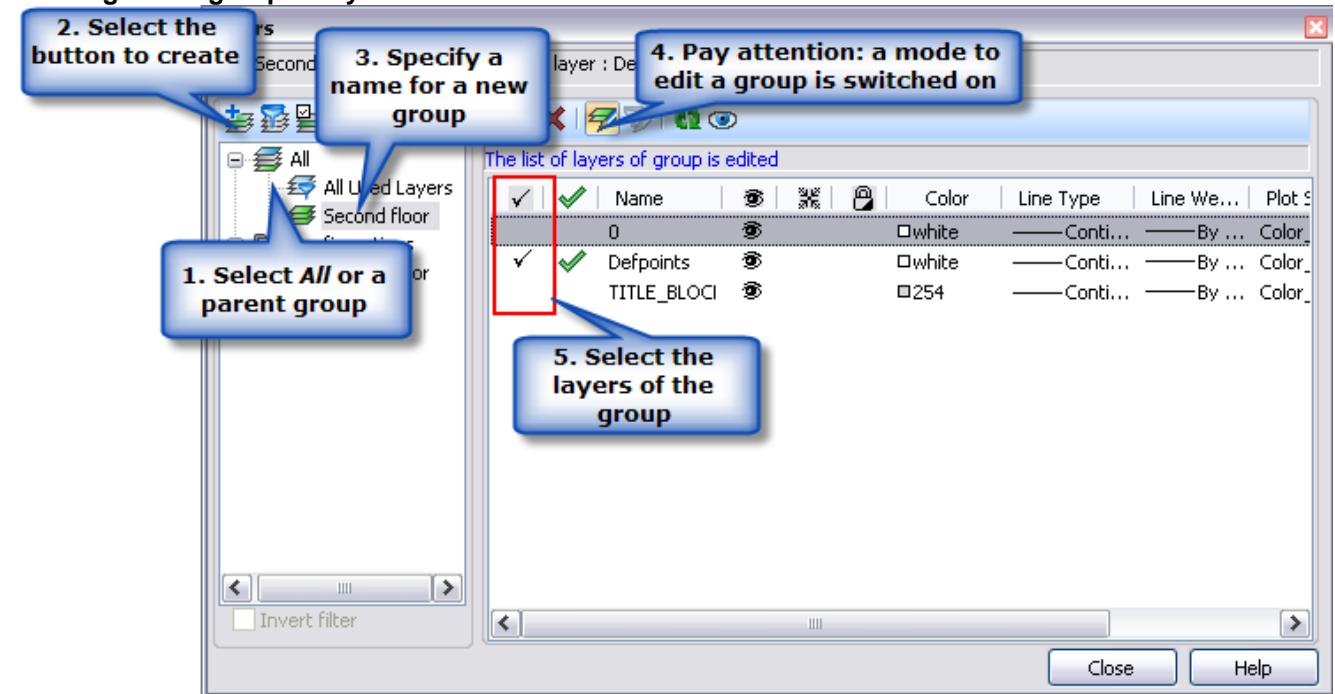
Grouping layers


The **Layer manager** allows grouping of layers. Grouping is worth doing when a document contains a lot of layers or it is required to change the settings of the specified layers often. For example, it is possible to show quickly a group of layers on the drawing or switch on/off their printing.

The tree of categories displays the groups, filters and configurations existing in the document. When you select an element in the tree of categories, the layers from the selected group, filter, configuration or external document link are shown in the right part of the manager.

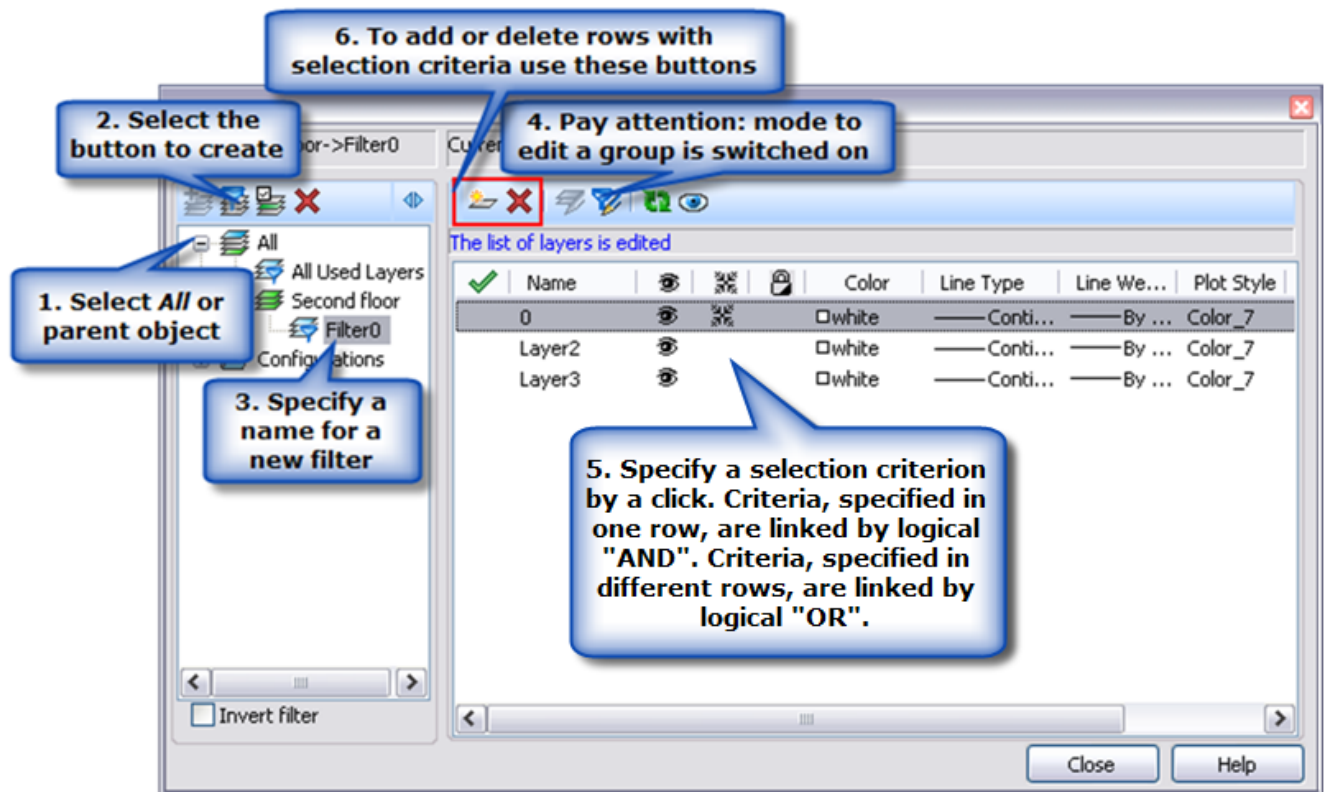
A group is a set of layers, specified by a user. One layer can belong to different groups. A group is marked with  sign in the tree of categories.

Creating a new group of layers:

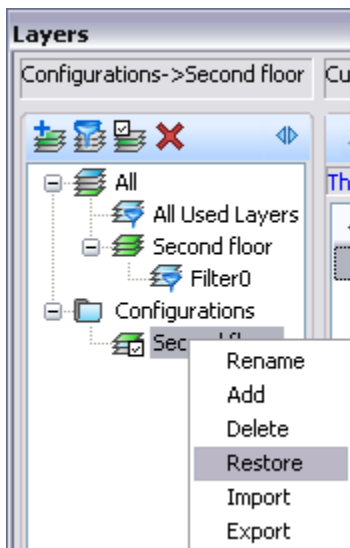



A filter forms the list of layers, satisfying the criterion of selection. Selection can be performed by one or several parameters of a layer. For example, selection of layers with line weight - 0.5 for printing. In the tree of categories a filter is marked with  sign.

Creating a new filter of layers:



Recovery of layer's parameters



The configuration of layers is contained within a named backup copy of the layers' parameters with the ability to edit and recover them. The layers' configurations are stored in a separate folder and marked with  sign in the tree of categories.

Recovery of configuration means copying the parameters of the same name layers from the configuration into the document. To recover the configuration, select it from the context menu of the configuration in the tree of categories.

A configuration can be exported in a .dwg or LAS-file and imported from them again.

Mode of insertions editing

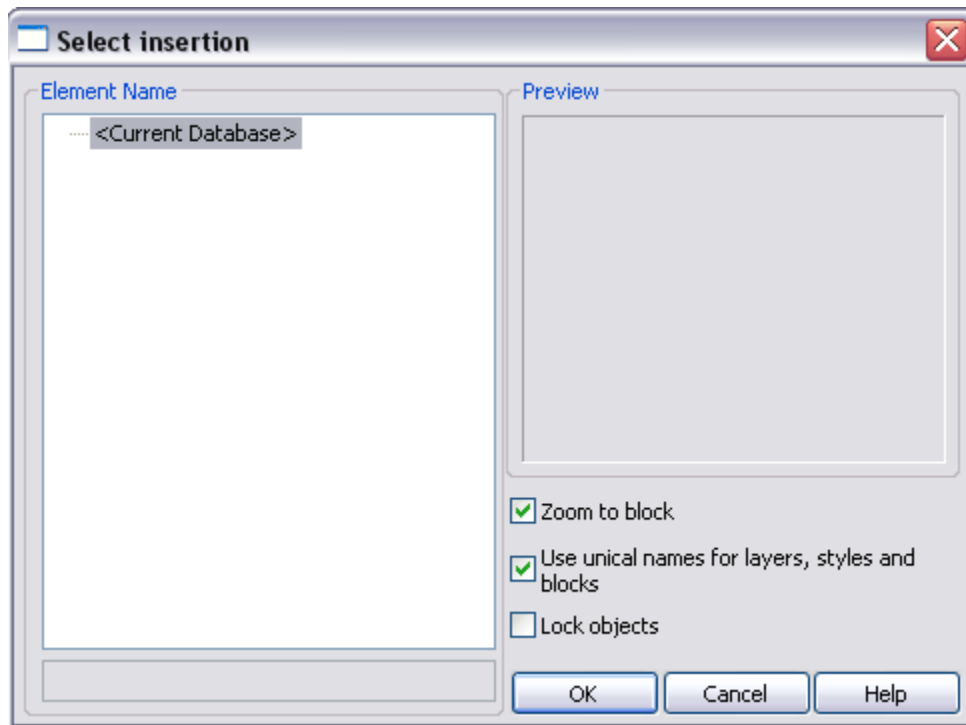
There is a new ability to edit external references (DWG insertions and blocks) in the mode of insertions editing.

In the editing mode a working set, which can distinguish objects belonging to an external reference or description of the block from the objects from the drawing, is made from the objects inserted as an external reference. You can add new objects to a working set and exclude any already added. New objects, created during the editing insertions, are almost always automatically added to a working set.

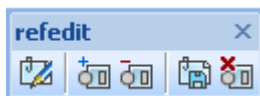
Tools > External reference > Edit reference.

Command line – REFEDIT.

The Select insertion dialogue contains information about the selected reference and nested objects which can be selected for editing.



After selecting objects for editing, the **refedit** toolbar appears. Using the buttons of this toolbar you can add and exclude objects from a working set and save or cancel changes of insertions.



Command line

Entry of relative coordinates

Relative coordinates are calculated from the last specified point. It is useful when the position of the point from the previous point is known. To set relative coordinates use the '@' symbol before a value. For example, @7, 8 marks a point placed in 7 units from the previous point in the positive direction along X axis and in 8 units from the previous point in the positive direction along Y axis.

Entry of polar coordinates

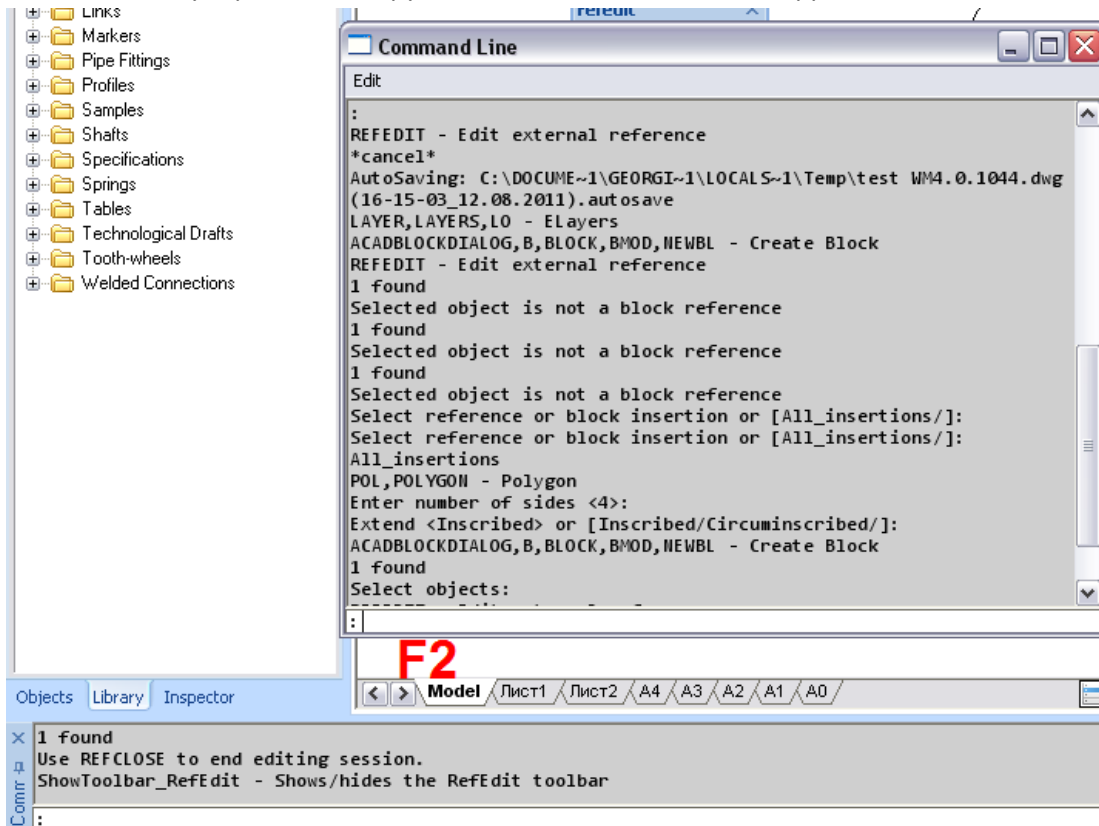
Polar coordinates are very useful to draw lines in the direction of something. The position of the point is set by the distance from the previous point and the specified angle. The values of distance and angle are separated by the < symbol. For example, to enter a point, set in 9 units from the previous point and with 35 degrees angle, type 9<35.

Calculation of mathematical expressions

You can calculate mathematical expressions in the command line at any time. To calculate a mathematical expression, type the '?'. A full list of registered mathematical functions and constants can be displayed in the command line when entering the '?' symbol. Variables, specified by a user, can be added to mathematical expressions. Such variables are defined with the following syntax: ?<variable>=<value>.

Copy of the command line using F2

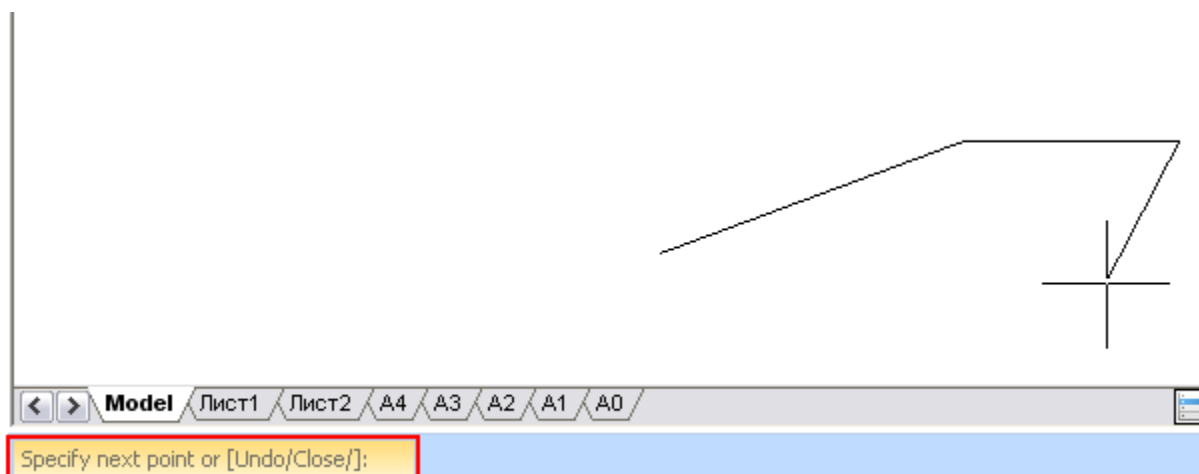
When the F2 key is pressed, a copy window of the command line appears.



You can close it at any time and open it again later. The next time it is launched, the window is at its previous size and position, but it is always over the main window, even if the cursor is moved in the drawing area or other windows.

Auto hide mode in the command line

Now the command line can be switched into auto hide mode. In this mode the command line is hidden in the tab. In the tab name you can always see the command line's message and entered values.

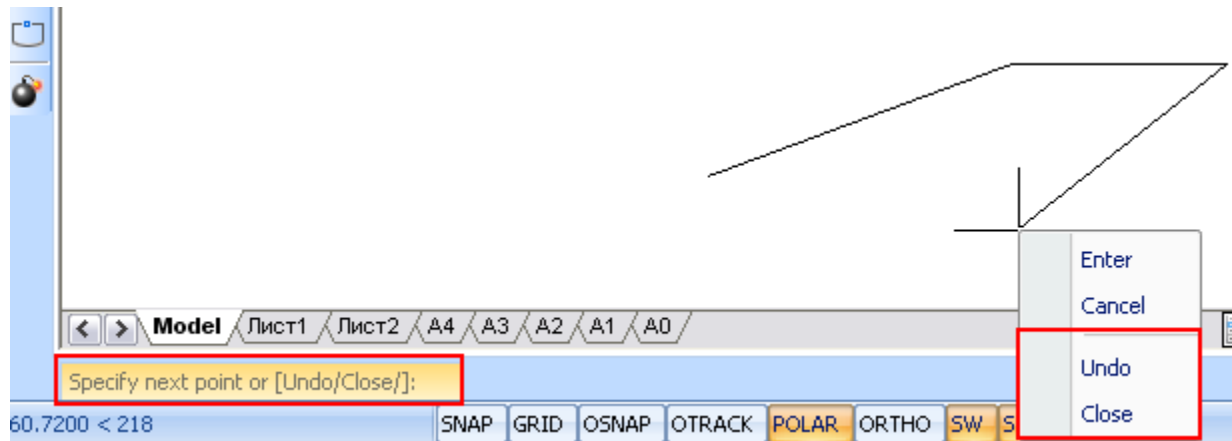


The entry of commands and required values are performed without showing the command line. When the cursor is moved over the tab, the command line expands to its full size; it shrinks again when the cursor is moved away.

To switch on auto hide mode, select **Autohide** mode from the context menu of the command line.

The command line is hidden when working with a drawing, but you can see the current messages of the command line. A command or values entry can be performed without expanding the command line.

To select a parameter of the current command, open the context menu in the drawing area:



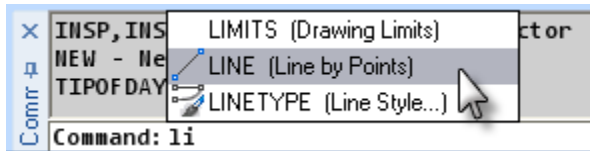
Programmatic access using ActiveX® API

Programmatic access to the application is achieved using ActiveX® API. The Programmers Guide about the WiseMechanical object model is in the **Help folder**.

Autofinishing commands and settings of the right button

WiseMechanical has a classic CAD interface. There is a command line where you can enter commands or their parameters in the bottom part of the window; there are numerous toolbars on the sides of the working area and the mouse is used for selection. In the 30 years since the development of this interface, a lot of users have got used to it. Autofinishing commands and the settings of the right button must be in a professional CAD solution. Let's look how this is achieved in WiseMechanical.

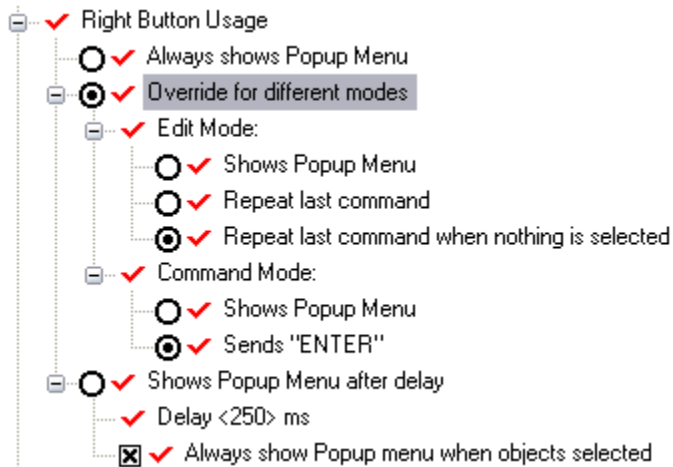
Autofinishing commands – this is a very easy to use mechanism. When you start typing a command in the command line, a new temporary window emerges over the command line with all the commands, containing typed symbols. For example, when you enter «li», LIMITS, LINE, LINETYPE and LINM commands are shown:



Adding a new symbol reduces the number of commands shown.

By pressing the TAB button there is round robin of commands in the pop up window, and you just have to press ENTER to start a selected command.

If you do not want to press ENTER you can set the right button parameters:



If the right button works as ENTER, you just have to press the right button. ENTER is used to confirm entry of data and repetition of operations, so such a setting of the right button is very useful.

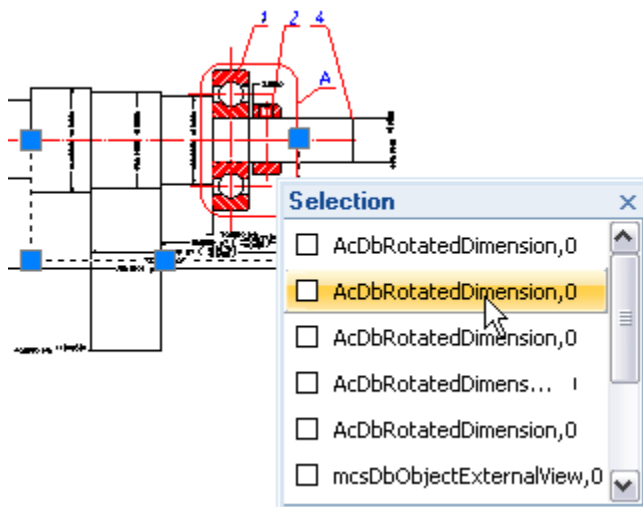
There is one exception: if any object is selected and you want to do something with it – copy, change its properties, replace or select similar objects, select these commands from the right-button menu. The “Repeat last command when nothing is selected” setting is important in this case.

Try to adjust WiseMechanical as shown in the picture and your productivity will increase!

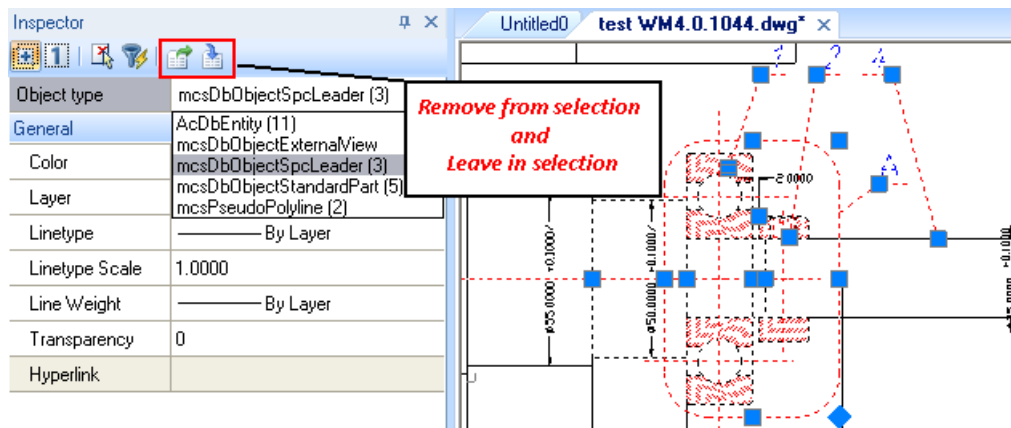
The improved object selection

Work with a dense drawing is complicated. When the drawing is full of objects, selection of the objects becomes difficult.

The object selection mechanism is improved in the new version of WiseMechanical. You can click the object group which is located close together or overlaid one on another and the *Selection* window appears. In this window you will see objects that are located under the cursor at the moment. You should specify which objects you want to select or, by one click, choose *All*, or cancel by clicking *Nothing*.



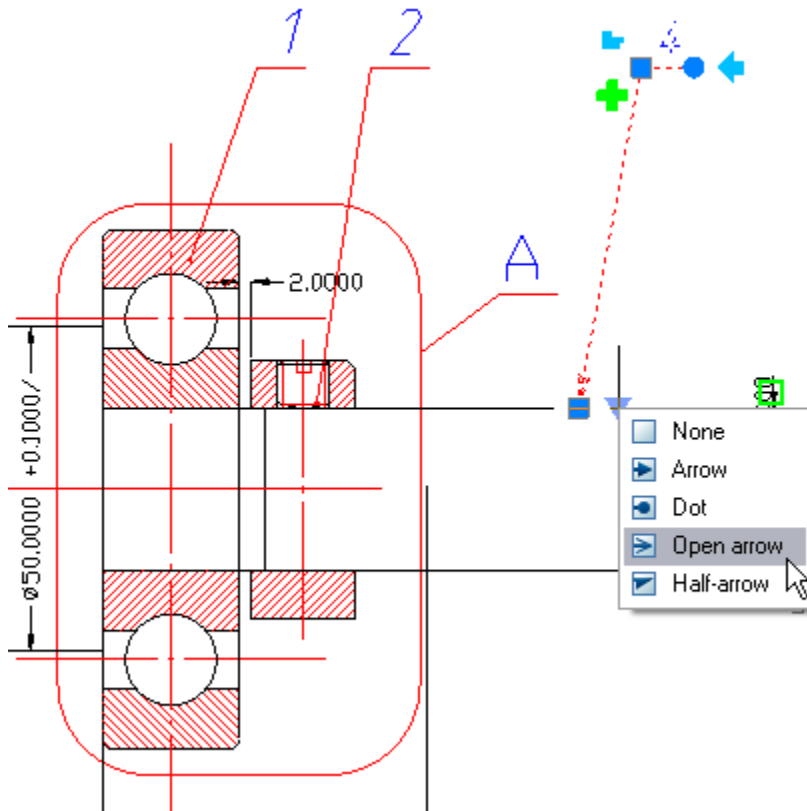
If you select a group of objects by frame and use the *Object type* filter in the *Inspector* window, the *Remove from selection* and *Leave in selection* icons will be active.



Advanced grips

The grips are often used to carry out object editing in the design system. If you select an object, you can see the blue markers (squares, circles, ellipses, etc.) in its node points. Use these grips to modify the object – to stretch it, move it, rotate it etc...

There are different methods of editing using the grips– for example, it is possible in the leader note to add extension lines, change text alignment or change the arrow type:

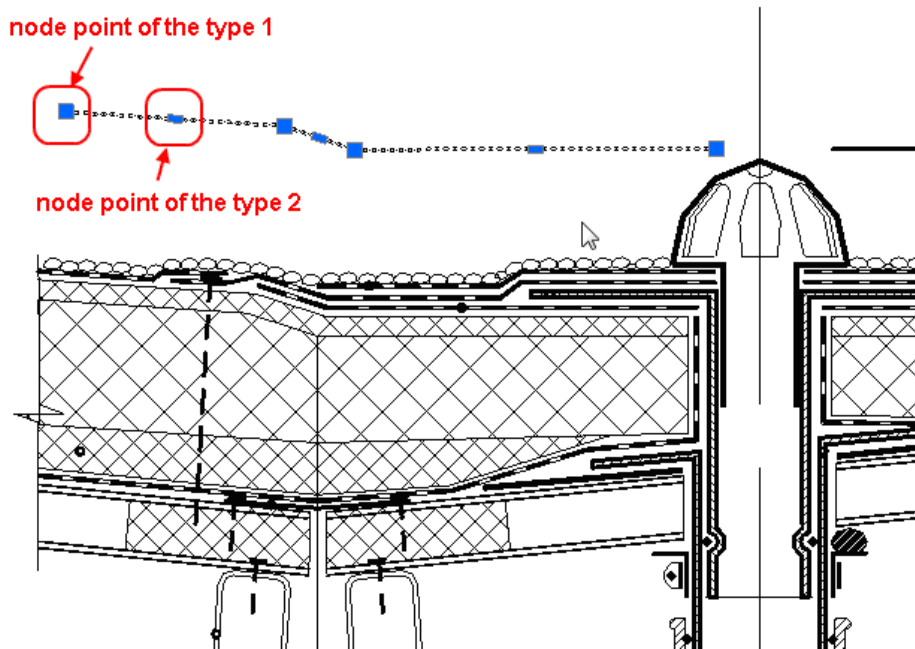


Advanced grips are very helpful in the designer's work – it is the most intuitive, fast and convenient method to edit objects.

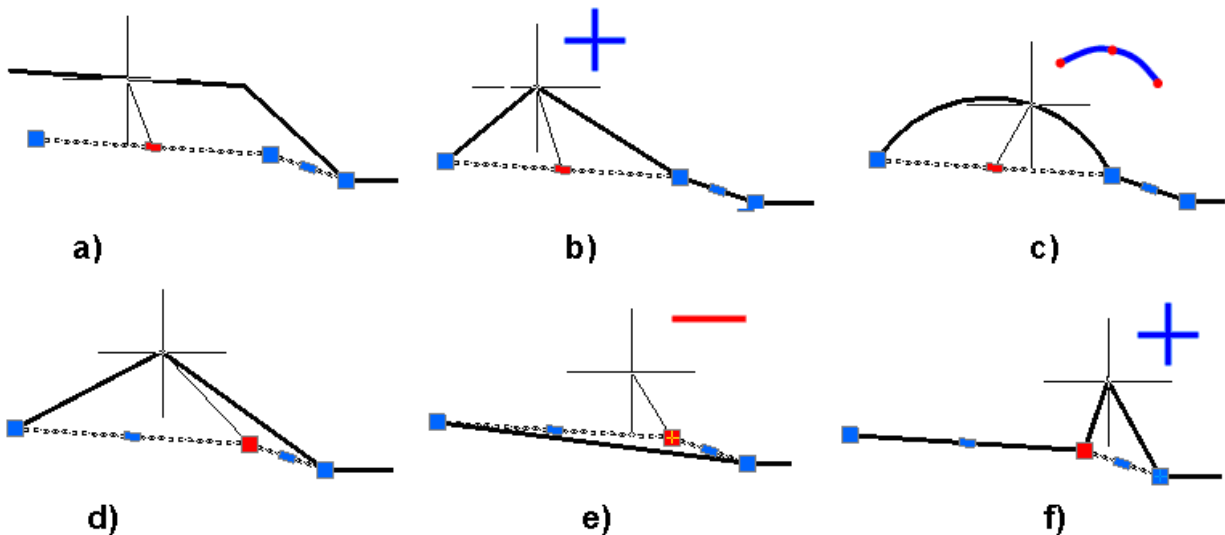
In the new version of WiseMechanical you can use advanced grips for polygonal objects: polylines, hatches and limits of polygonal viewports, as well as with lines, arcs, and splines.

Consider some of them in detail in the example of a polyline.

When you select the polyline, you can see two types of node points: in the vertex of the polyline (type 1 – look like a "classic" grips) and in the middle of segment (type 2 – flattened and look like rectangles):



If you click the node point in the middle of any segment, the segment can be moved right after the cursor (fig. 3-a). If the CTRL key is clicked once, you can add a new vertex (fig. 3-b). And if the CTRL key is clicked again, the line segment will be transformed to an arc (fig. 3-c).



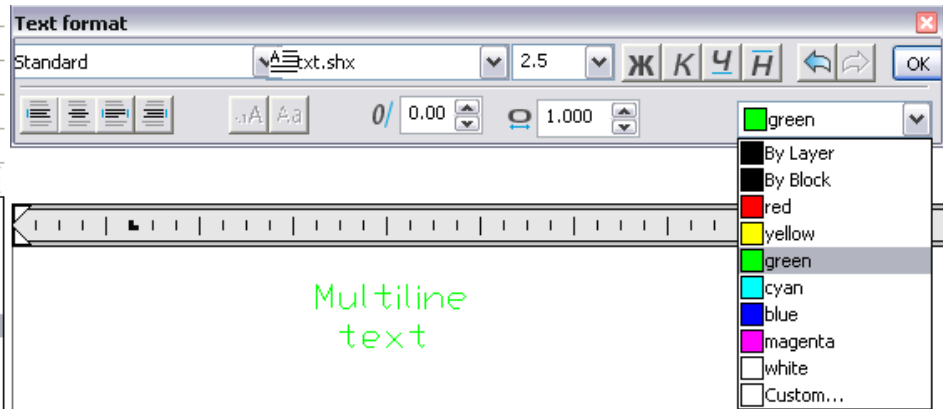
It is possible to apply the same action to polylines vertices. In that case, the vertex will be moved, removed and added (fig. 3-d, 3-e, 3-f).

Try it and see how convenient it is!

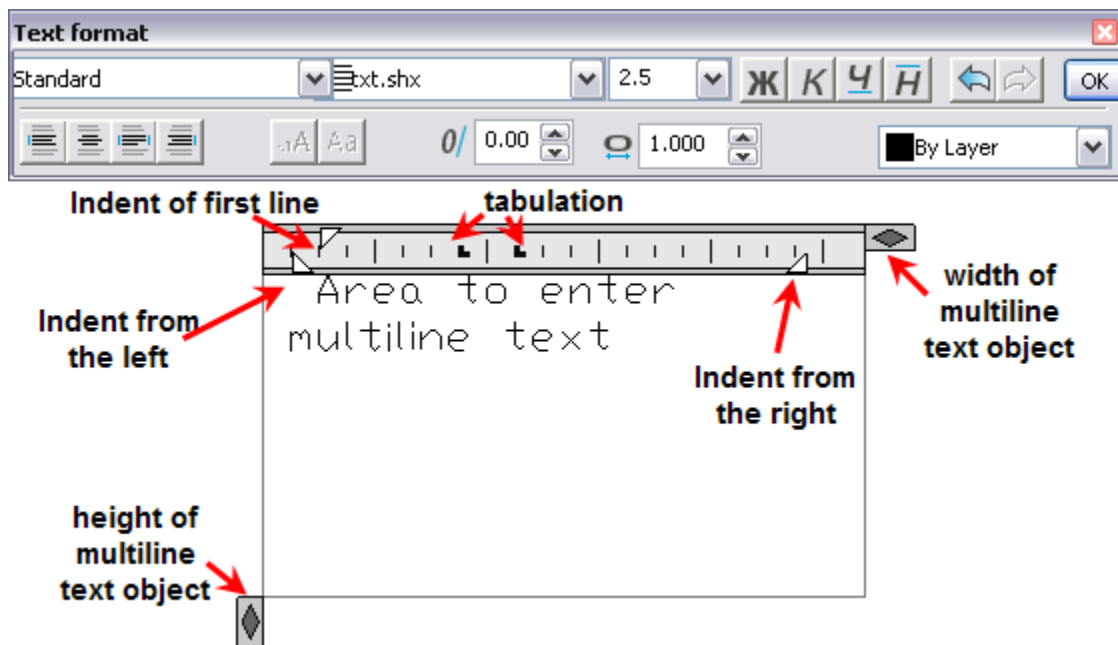
Multiline text

Using the *Multiline Text* command (MTEXT) you can set the colour of the symbols in the *Text format* dialogue at the editing process. You can set the background for the selected text block in the *Inspector* window. These options are useful when working with complex drawings:

Text	
Contents	\pnt5,7;\C3; ...
Style	Standard
Justify	Top left
Text height	2.5000
Rotation	0
Line space factor	1.0000
Line space dista...	4.0000
Defined width	208.0646
Defined heihgt	80.9797
Background Ma...	background color
Background Off...	No
Linespace style	background color
Direction	red
Geometry	yellow
Position X	green
Position Y	cyan
Position Z	blue
	magenta
	white
	More Colors...



The **Edit multiline text** command was improved. There are undo/redo buttons, indent from edge and tabulations:



 **Undo/Redo** buttons for multiline text.

You can select separate fragments of text and edit their parameters.

The command automatically zooms the text for easy editing.

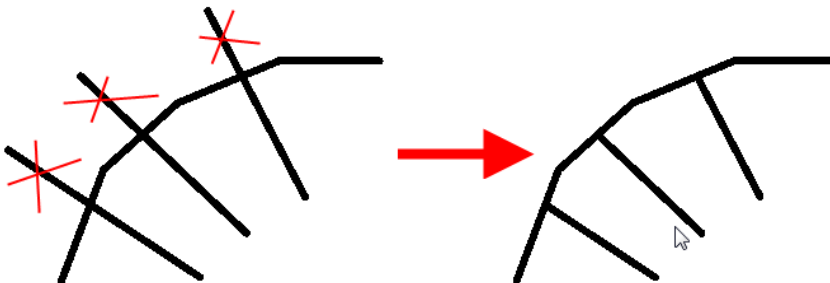
Select All	Ctrl+A
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Undo	Ctrl+Z
Redo	Ctrl+Y
Insert symbol	▶
Degrees	%%d
Plus/Minus	%%p
Diameter	%%c

To insert a special symbol, select the **Insert symbol command** from the context menu.

Smart Trim and Break vectors at point

In the process of objects editing, you can use one of two new commands – *Smart Trim* and *Break vectors at point*. If you need to break the line on two segments, use the *Break vectors at point* command. Just click the mouse at the required point on the line.

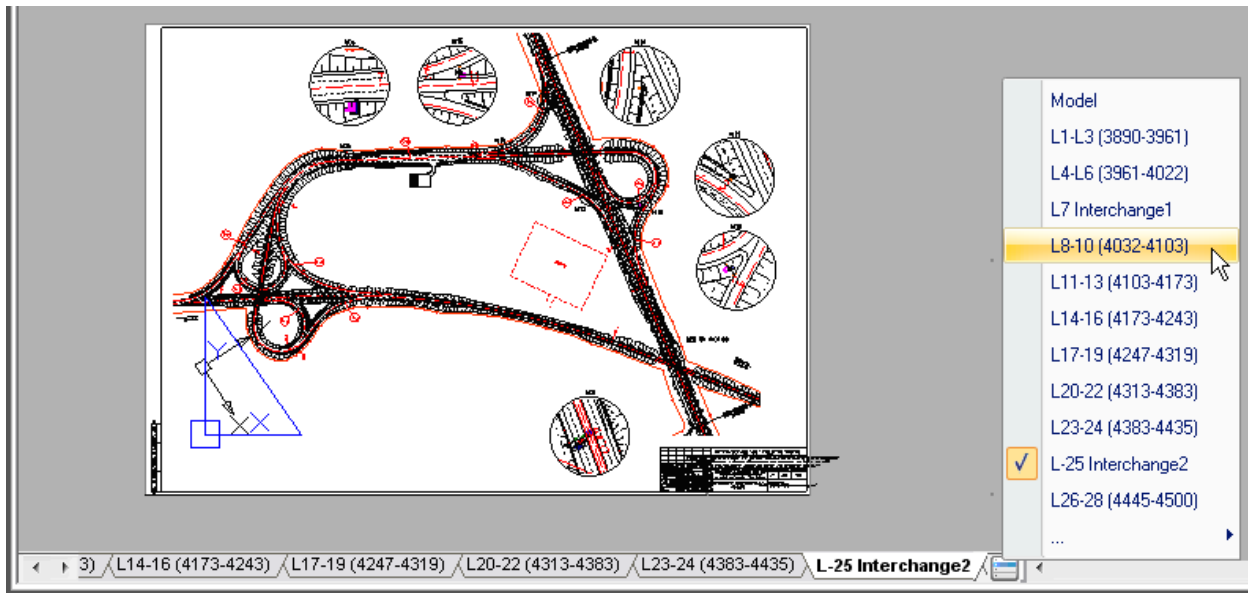
Using the *Smart Trim* command, you need to select the object to trim. Then, by analysis of the drawing geometry, the nearest intersection points of the trimmed object to other objects are found. One click and the object is trimmed:



Working with paper space and viewports

The modern technologies of design allow preparation of the drawings in the paper space. It is more convenient because it is possible, by using the viewports, that the same model can be prepared on multiple layouts in different scales. Much attention has been paid to updating this functionality in the new version of WiseMechanical.

First of all, the speed of operation when you switch from model space to paper space and back was increased. It is noticeable in drawings which contain a large number of viewports on the layouts. An additional control element was entered – the button between the layouts and named views:

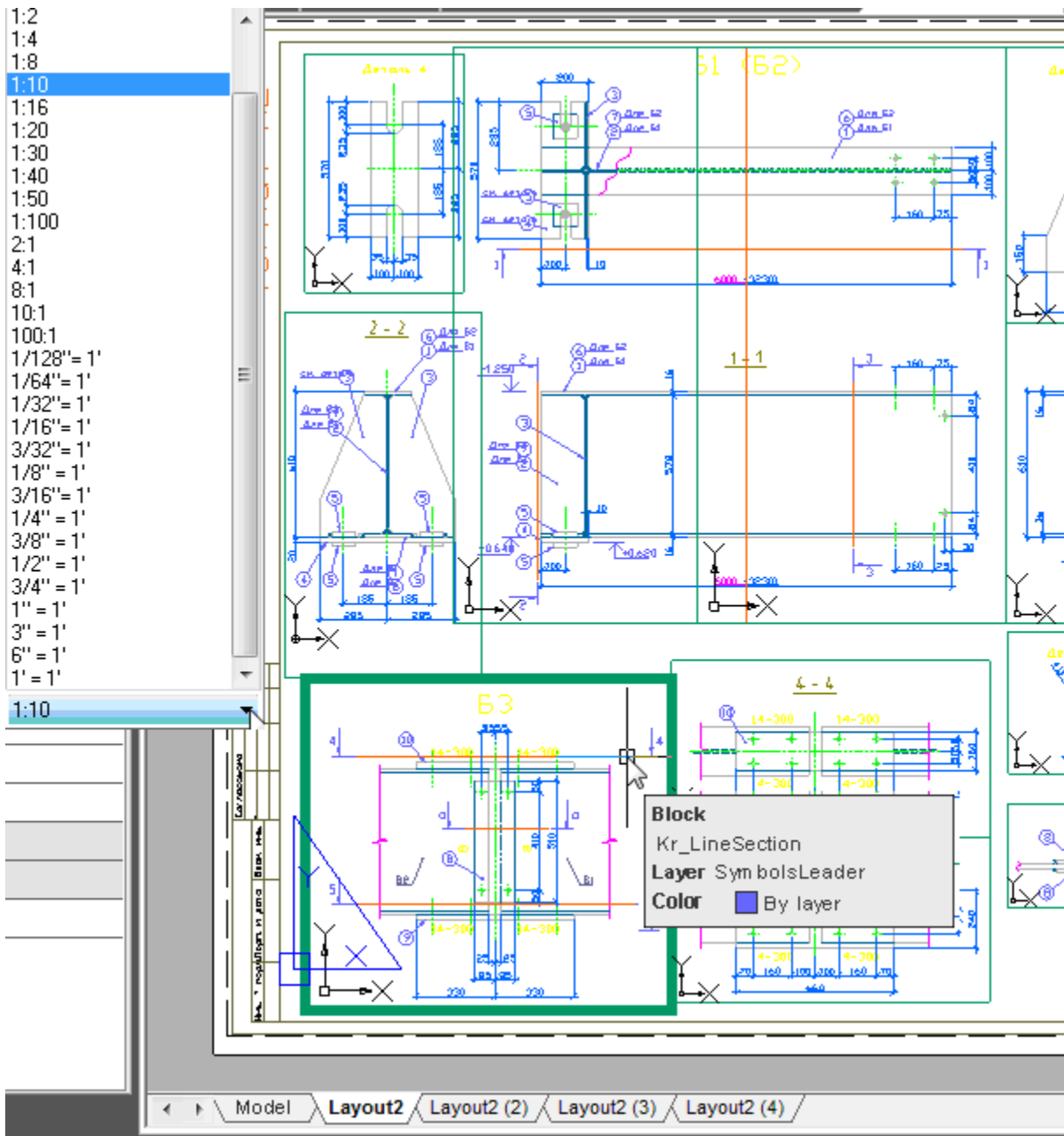


Typically, switching between layouts or model space was carried out by tabs at the bottom of the screen. It is convenient when there are only a small number of layouts. If there are a lot of layouts, then the tabs “escape” under the scroll bar. To find a tab, it is necessary to click in the last visible tab and wait whilst the program handles this transition, and then click again, and again and again... When the drawing is complicated, it can take a long time.

The additional button, located at the end of the tabs list, allows you to switch to whichever layout is required with just two clicks. The advantages of such a function are clear.

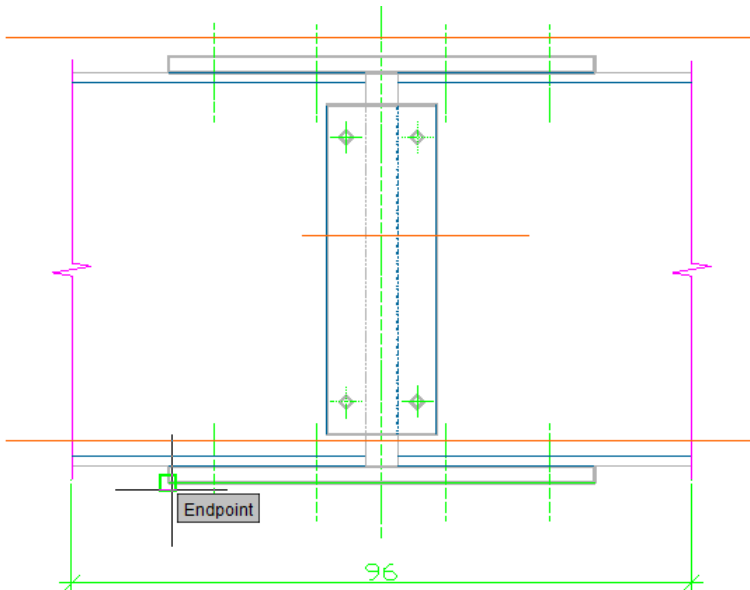
The preparation of drawings, implemented through the viewports, required future functional development. For example, in the previous version, the viewports were not blocked from the changes, it was difficult to set the scale, reference to the objects of viewport was absent, etc.

The new version took the most important comments of users into account. Working with viewports was much improved. Note the following figure:



The changes are important: for example, it is possible to block the viewport and select the scale from the list of standard scales (1:10, 1:100, etc.). The editable viewport is highlighted with a thick frame. To enter into the edit mode, double-click on the viewport and to switch to another viewport, click once.

The references to the objects of viewports have been added. This allows you to prepare the inserted sections of drawing directly into the paper space:

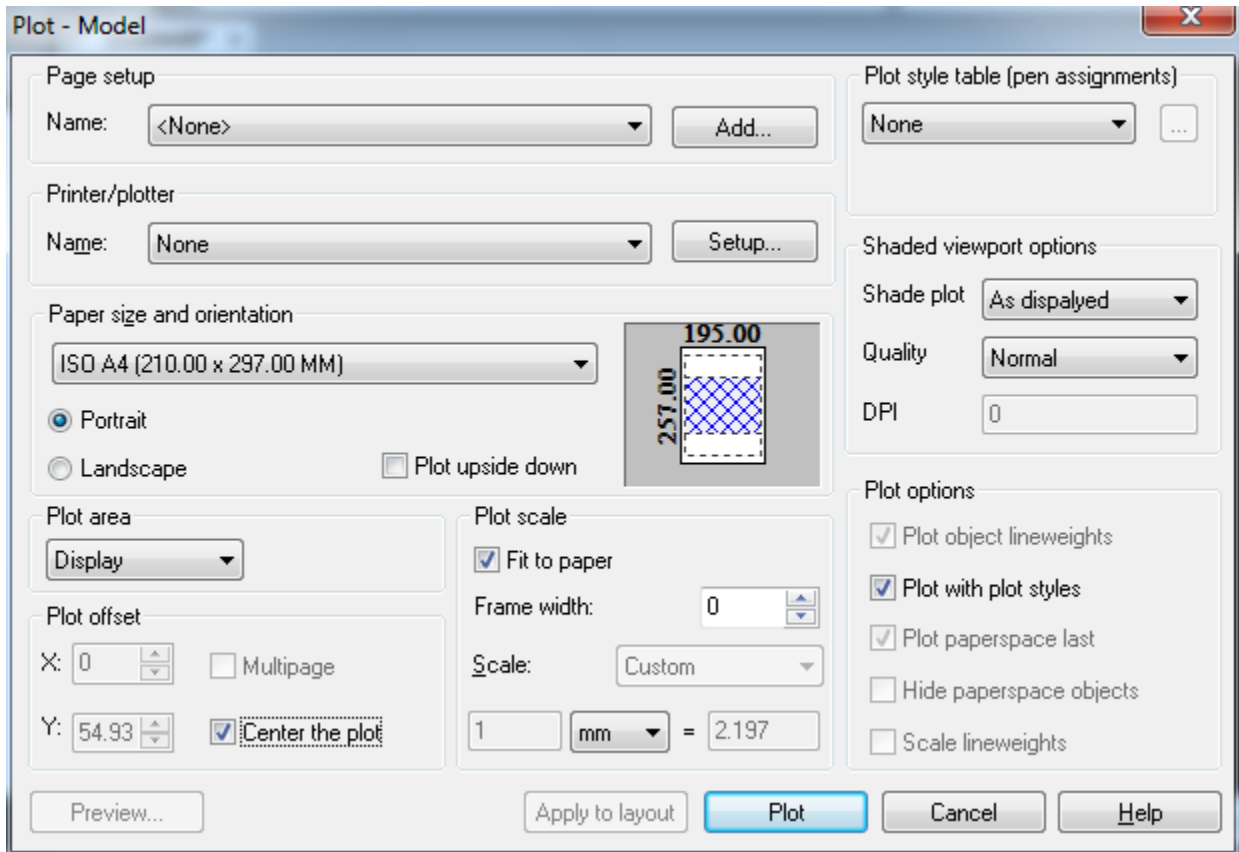


Plot in the WiseMechanical

Increased functionality of the drawings print was one of the main users' wishes. The ability to print from preview, print the custom-sized layouts, more convenient printing using one settings dialogue. These and many other requests have been incorporated into the new version of WiseMechanical.

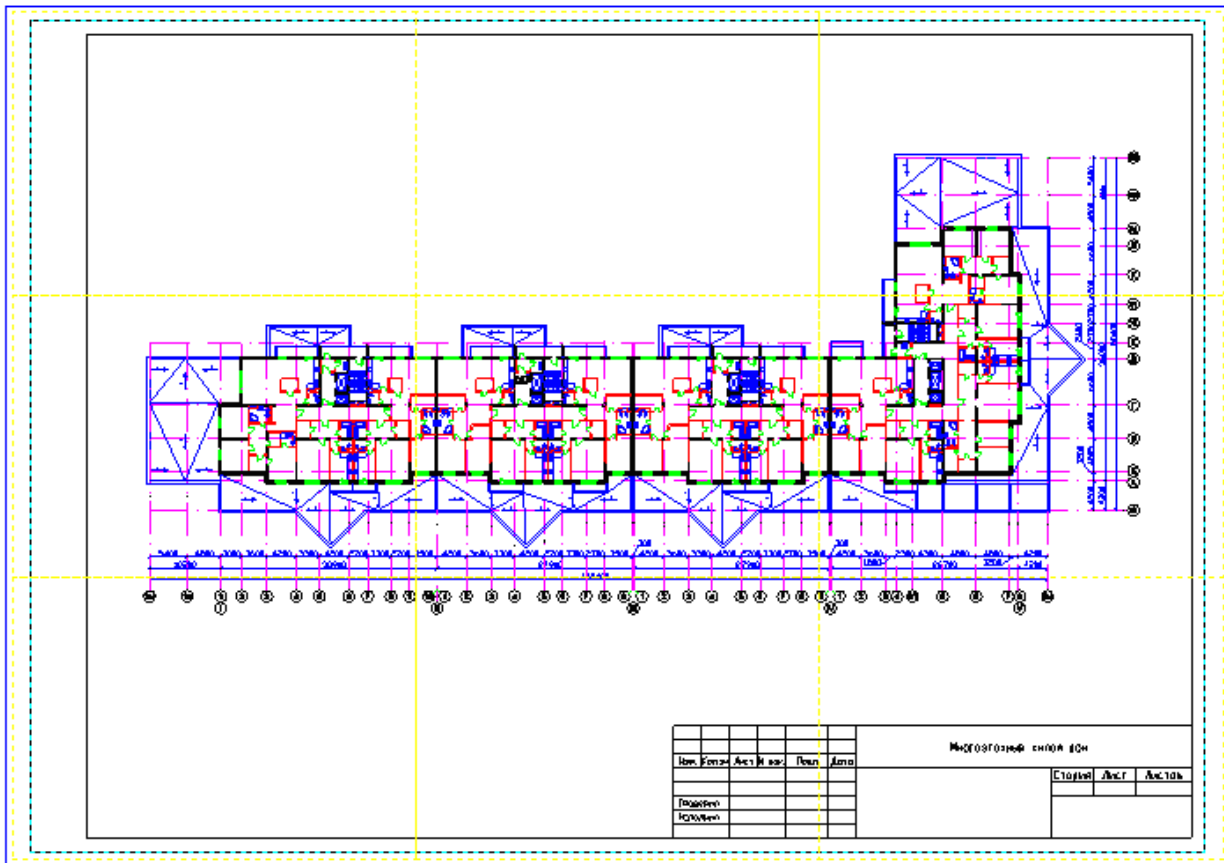
Now you can set the necessary parameters in the Plot dialogue, such as choose a printer/plotter, paper size and orientation, plot scale, plot style, etc.

Using the Plot dialogue, it is much easier and simpler to print the drawings:



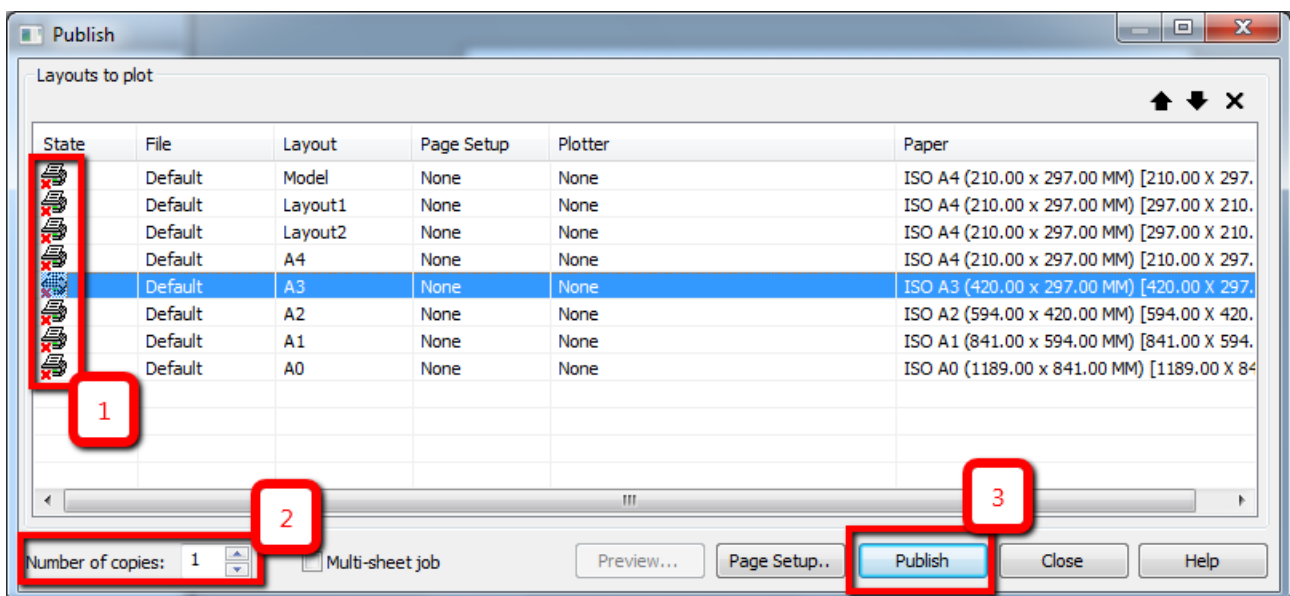
When you set the plot area, the paper size and orientation is displayed (blue frame), print area border of printer/plotter (blue dotted frame), and the paper dimensions (yellow lines).

Now it is easy to lay out drawings to print on smaller size papers:



Using the Print button located in the Plot Preview dialogue, you can print a customised worksheet immediately after viewing.

The new Batch Plot command will be useful and convenient when you print large size documents or multiple documents. In the Publish dialogue you can set: State of plotting documents (1), number of copies (2), and then send a batch of documents to the printer (3):



And some more new printing features:

- settings for custom paper;
- opportunity to specify the model of multiple plot areas;
- opportunity to save the plot settings as sets of parameters list.

More features

The finished work consists not only of the new and powerful features, but also of the optimisation of already familiar tools. Almost all of the tools were completed and optimised in the new version of WiseMechanical.

New features

- The new properties of Transparency can be set to any objects, such as hatch.
- The new commands of Shape and FillShapes.
- The hyperlinks insertion tool.
- The new snap type: the intersection of extension vector.
- The new Editing2 toolbar.
- The ability to set the background colour and font of the command line.
- The status bar now displays not only the absolute coordinates, but also the distance and angle.

The program optimisation


- Network licensing – more user-friendly work with licenses within an organisation.
- Expanded memory support – WiseMechanical can use up to 3GB of RAM on 32-bit operating systems.
- The on/off of the line weight or hatches showing no longer have to be forced to set the image redrawing (REGEN command). It was possible after optimising the speed display on the screen – now even on multi-megabyte drawings this operation doesn't take much time.
- Adjustments were made to the interface. Added the ability to create custom toolbars.
- Spline can be cutting edge for objects in the Trim Vectors and Extend Vectors commands.
- Work with splines was improved: now there are two methods of editing and graphical switch between these methods.
- Dynamic blocks display more control buttons: rotate, scale, moving, choice of sizes, etc.
- The Chamfer and Fillet commands work with zero values.
- The plot is performed faster.
- Displaying of a worksheet that contains many viewports is faster.
- Interface is improved. It makes the program more convenient and enjoyable.

- The way some commands work was redone and the memory usage was reduced for some intermediate operations.
- The **Undo** command was optimised: takes less RAM, undo commands was redone.
- Dimensions: when creating a diametric dimension, arrows, which are not close to the dimensioned arc, are hidden.

The stability and quality

- A “breakdown” of graphics on small scales has been corrected.
- The error of reset of properties when copying multiline text has been corrected.
- Addition of Z-height, when copying the objects, has been corrected.
- The “Zoom all” command has been corrected. Work with dense drawings has become more accurate.
- Work using the User Coordinates System (UCS) has become more accurate.
- Corrected snaps when working with large drawings (the entities have Z-heights).
- The Extend command with arcs and text has been corrected.
- The Copy Object Properties command (MATCHPROP) has been corrected.
- In the Save dialogue the version list of different .dwg formats has been added.
- Errors when working with tables were corrected.

Other changes

- A command to edit block insertions was added (**Tools > Edit block insertion**).
- The **Stretch** command (**Modify** menu > **Stretch**): the preview of stretch points was added, logic of work was corrected.
- The hatch (**Draw > Hatch**): the dialogue was corrected, search of contours, contour preview.
 - The Clipbook: Insertion of OLE objects in the drawing;
 - Because of copying data, it is possible to insert objects of BMP and Windows MetaFile format.
- Settings (**Tools > Settings** parameters):
 - Adjustment of layers, profiles and colours can be made using the  button to open the corresponding dialogues;



- A setting **Edit > Set associativity during insertion of objects** was added
- A setting **Symbols>Notes>Rotate rack on UCS** was added
- A setting **Notes> Show dialog before inserting the object** was added.
- Line width:
 - Showing line width in the paper space was corrected. This mistake could make work in the paper space difficult if the visibility of line width was switched on (SW button in the status bar);
 - Showing/hiding line width was corrected when the visibility of line widths is switched on/off (SW in the status bar);
 - The visibility of line widths was corrected in the printing preview ;
 - The printing of line widths was corrected using the tables of printing styles (ctb-, stb-files);
 - The saving of ctb, stb, pc3 and other utility files was corrected.