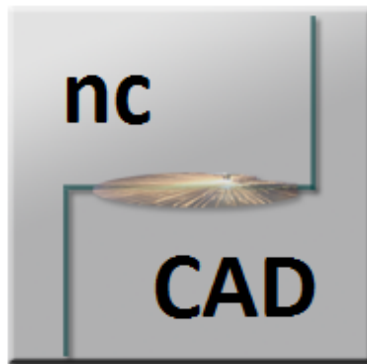




ncCAD32



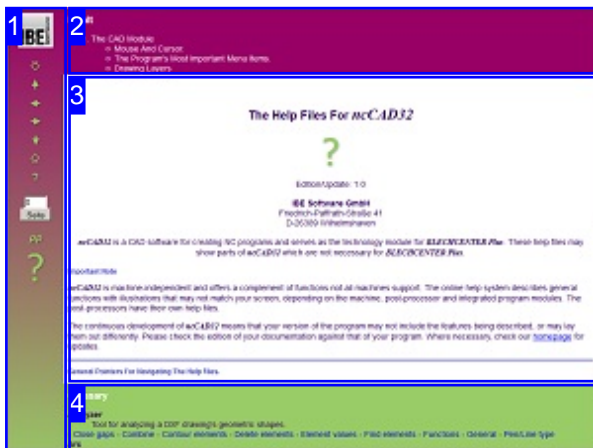
Online help

ncCAD32



Welcome to the help for ncCAD32

General Notes on Navigation



Select Navigation Method

There are two methods to navigate *ncCAD32* 's help files:

1. Interactive Navigation.

This allows you to control which texts you view by hovering the mouse over the relevant hotspots in the illustrations.

2. Consecutive Navigation.

This mode leads you sequentially through each help page. All available details are shown one after the other. Click the 'Next page' button (right arrow) in the left navigation panel [1] to move one step onwards.

Important Note

ncCAD32 is machine-independent software. It offers a multitude of functions, some of which may not be supported by your machine. This online help document describes the general functions of the program using illustration which - dependent on your machine, post-processor and purchased options - may vary from the display on your system. The continuous development of *ncCAD32* can also cause differences between the screenshots shown, and what is shown on your workstation. Please first compare the revision number of your help documentation to that of your software, and check our [Website](#) to see if there is a newer version available.

The screen is divided into four areas: the menu to the left with navigation buttons and the page number, which you can type in to jump to a specific side.

The content list [2] is at the top, divided into chapters. The glossary [4] is at the bottom, with key words and phrases linked throughout the help files. You can swap the glossary for an index list, as preferred. Simply click a keyword in either case to link straight through to a relevant help page.

The middle frame shows the current document. There are two methods to navigate the help files:

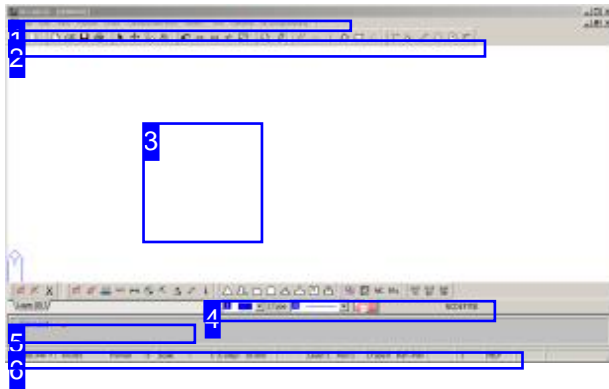
Interactive

or

Consecutive

The CAD Module

Drawing Your Own Components.



This is the interface of *ncCAD32*. The middle frame is a drawing board, the header and footer areas are menu, command and status bars.

The program is dongle-protected and must be authorized for use.

ncCAD32 represents a tailor-made tool for the creation of cut-/punch parts. You can create geometric figures and have all possible options for constructing a drawing.

Note:

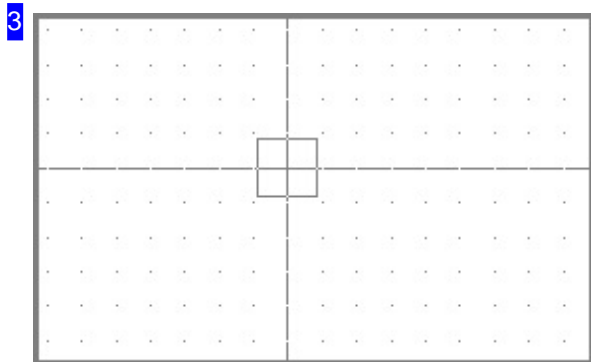
- o If in the next pages you see 'click' this means left click, unless specifically noted otherwise.
- o The cursor can be moved with the mouse, via the command line or with the arrow keys.
- o Letters in angled brackets (<Esc> <Enter> <A>) describe buttons on your keyboard.



The menu bar allows you to start new drawings or import existing ones, set formats for colors, lines and pens, launch drawing commands, edit your drawing and set general settings. The menu items are described in the 'The program's most important menu items' chapter.



This icon bar offers a selection of most commonly used drawing commands available at the click of an icon (quick-key). You can configure this panel; there is a description of the basic settings in the 'Entering commands via the symbol bar' chapter.

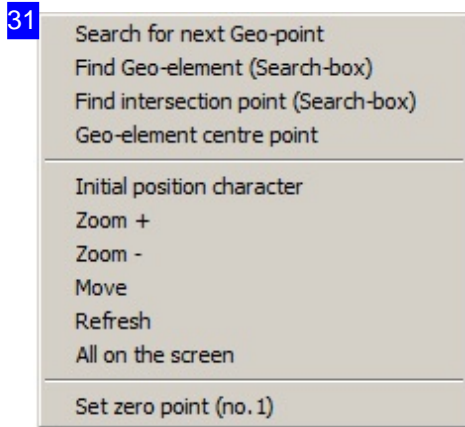


This is the 'Drawing Board' with a cursor in the middle, shown by a hairline cross. Right-click in the drawing area to access a Pop-up menu, via which you can give instructions to the program for searching elements or displaying the drawing.

You can move the drawing in any direction, holding the right mouse key.

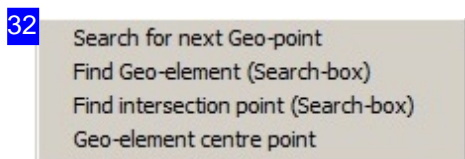
If you pressed the symbol for displaying 'information about geo-elements' in the format bar [4], the information will be shown in the status bar when you place the cursor over an element in your drawing. A symbol corresponding to the geo point will be displayed at the cursor when these are touched.

Use the mouse wheel to zoom in and out around the cursor position.



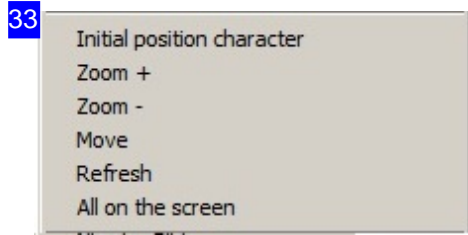
The top area gives instructions for Searching for elements around the cursor. The menu items in the middle allow you to control the display and the last menu item sets a new zero-point.

Alternatively, you can zoom in and out by using the left and right mouse buttons while holding <Ctrl>, or also use the <+> and <-> buttons.



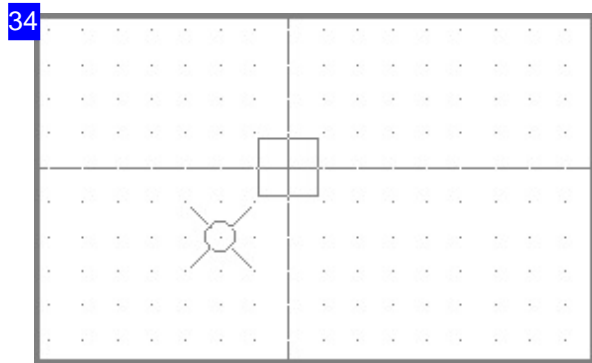
The search menu items in the Pop-up menu mean, from top to bottom:

1. Find the nearest geo point to the cursor.
2. Find the nearest element that is touched by the cursor rectangle.
3. Find the nearest intersection inside the cursor rectangle.
4. Find the mid-point of the element inside the cursor rectangle.



The menu items for display settings in the Pop-Up menu mean, from top to bottom:

1. Go to the origin.
2. Zoom in.
3. Zoom out.
4. Return to the original zoom size.
5. Move around the drawing board.
6. Re-draw the drawing, fragmented parts will be swept clear.
7. Fit the workpiece to the screen.



A drawing has a zero-point in the bottom left corner. This is declared as 'zero-point 0'. You can move the zero-point at any time, the new zero-point is then identified as 'zero-point 1' (here to the left below the cursor). For a description of this procedure, see the 'zero-point' chapter.



The 'Format line' shows the defined formats for line type and style, and offers options for switching the display of elements on and off. The penultimate icon enables the display of information about the drawing element the cursor is resting on.

Use the furthest icon right to show or hide the grid; important when starting a new drawing. Tip: The grid can obstruct when trying to 'capture' geometry points. In this case, simply hide it.

You can abort any command sequence by pressing <Esc> or right-clicking.



This is the program's 'Command line'. This line allows you to enter commands and their codes more accurately than with the mouse. The commands can be found in the descriptions of the draw functions as a stand-alone document (DE only).

The command '499 114' allows you to merge various commands to a batch, with its own symbol (shortcut). Then you can call this shortcut (with a point as prefix) in the command line.

The next step in the editing sequence will be shown when using text mode. If you wish to abort an editing sequence, press the <Esc> key.

The command line can also appear in an input format, whereby the content of the line changes according to the instructions you gave the program.

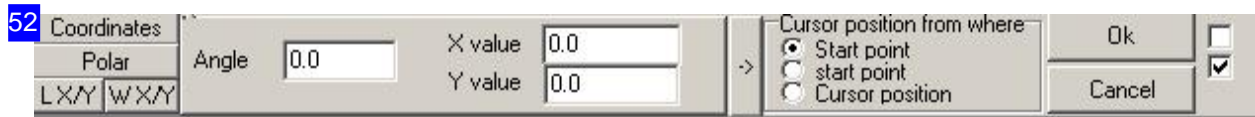
The input format can be activated directly by pressing <F2>. You will find a description of the content in the functions' corresponding pages.



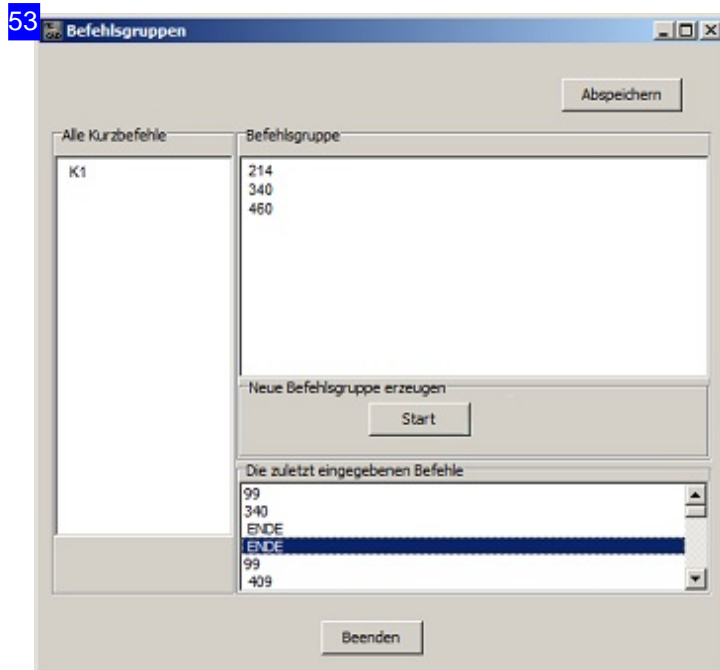
You can enter the position of the cursor in input format, either as X/Y coordinates, or polar coordinates (angle, offset). Press the corresponding button in the left panel and enter the required position. Click the arrow button next to the position field to move a marker to the new position. Click the 'Apply' button to move the cursor to the marked position.

The bottom buttons on the left panel allow you to move the cursor to a new position using a triangular calculation.

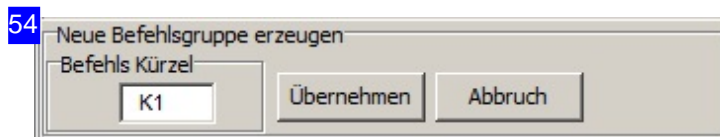
To keep the input format active during further editing, mark the checkbox at the far top right. The second selection box allows you to set whether the inputs are absolute or relative to the origin.



To enter a new cursor position using a triangular calculation, choose first between the side lengths and the angle of the side, and enter a value under X or Y. The cursor will jump directly to the newly calculated position.



You open this dialog with the command '499 114'. Here, you can build sequences of commands and give them an own symbol (shortcut), that can be called either in the geometrie or in the technologie with this symbol. All available shortcuts are listed on the left. Selecting an entry from the list, the sequence of commands is shown on the right. Click on 'Start', to enter a new symbol and the belonging commands.

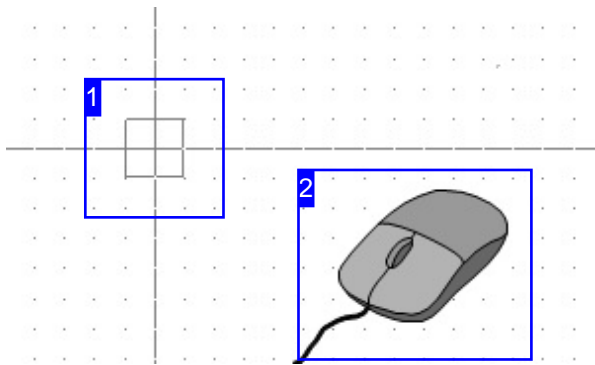


Enter a new symbol (shortcut) for a sequence of commands here. You can than call this symbol with a dot as prefix (.K1 in this example). This function allows you to build sequences using commands either from the geometrie or (with the command 340) from the technologie.



Status bar with information about the cursor position and the drawing. In the text fields to the right are the current layer number, pen number, line type and zero-point. The current zero-point is important for positioning a start-point when drawing new elements.

Mouse And Cursor.



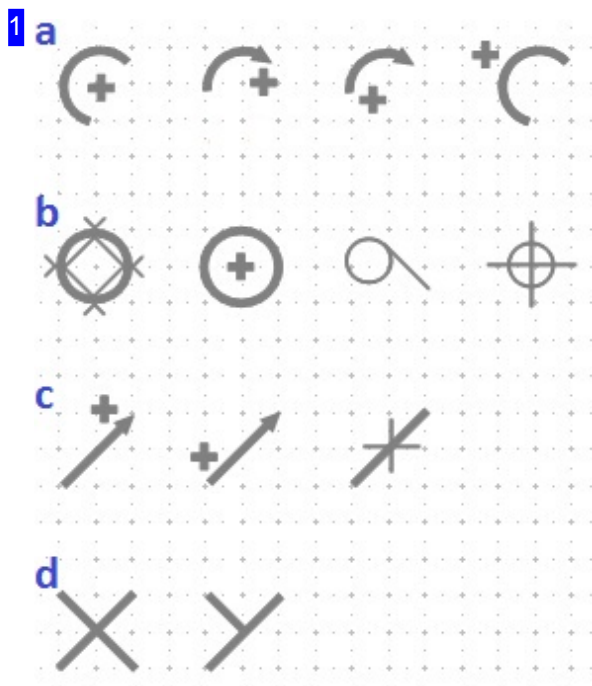
To adapt the display of the cursor to the points of a contour element, first select that function. Click the toggle button 'Format panel'. When the function is activated the cursor will jump to the next geometric point.

The left and right mouse buttons together with the control-keys <Shift> <Alt> and <Ctrl> can be used as quick-keys to help work progress more quickly.

ncCAD32 has some special functions to ease creating and editing drawings.

The mouse buttons can be used in conjunction with the control-keys <Shift> <Alt> and <Ctrl> to access various functions.

The display of the cursor can adapt to the appearance of the next contour element's point.



When the function for displaying information on the geometric points is activated, the cursor display adapts and snaps to the next point. This makes it easy to find specific points.

The symbols represent:

- o a) Arc points: center, start, end and middle.
- o b) Circle points: quadrant, center, tangent point, reference point.
- o c) Vector points: start, end, center.
- o d) Intersection and perpendicular point.

2



The mouse buttons' functions combined with the control keys are:

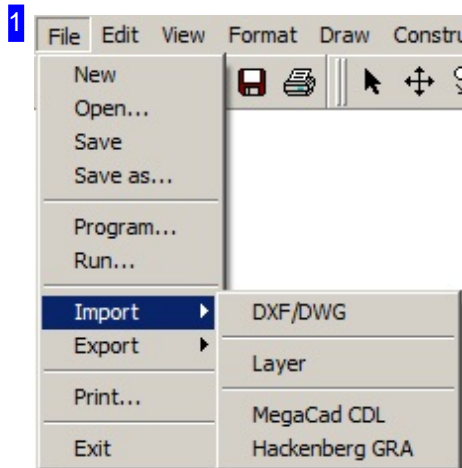
- o Shift + left-click: redraw the drawing layer.
- o Shift + right-click: redraw the drawing layer placing the clicked point in the center of the view.
- o Ctrl + left-click: zooms in on the current layer by a factor of 2 centered on the clicked point.
- o Ctrl + right-click: zooms out on the current layer by a factor of 2 centered on the clicked point.
- o Alt + left-click: moves the cursor to the next geometric point.
- o Alt + right-click: moves the cursor to the center of the view.

The Program's Most Important Menu Items.

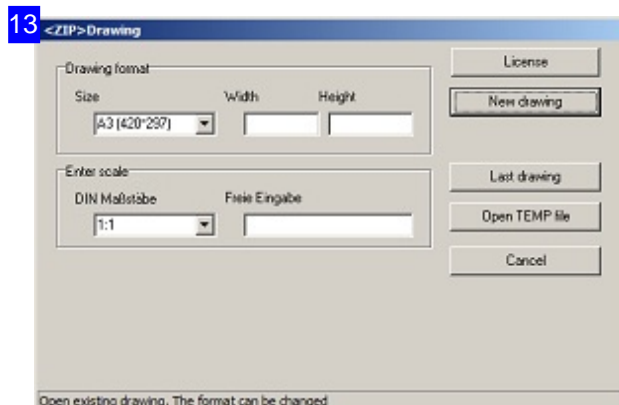


The menu bar in *ncCAD32* .

The menu bar allows you to start new drawings or import existing ones, set formats for colors, lines and pens, launch drawing commands, edit your drawing and set general settings. Some menu items are available directly from the symbol bar.

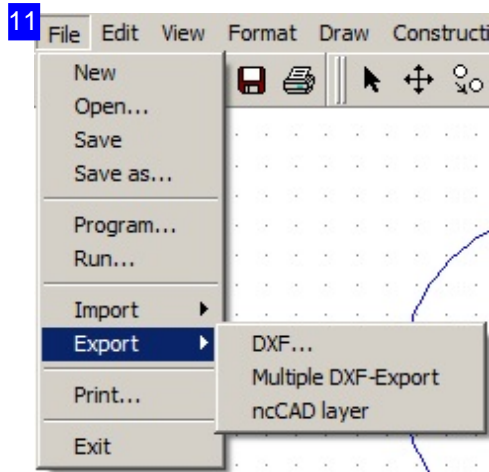


The 'File' menu allows you to create drawings, load drawings, and import and export drawings and layers, define and execute programs, print and lastly 'Exit' the CAD module.



When creating a new drawing you can select a predefined size from the list or create your own format.

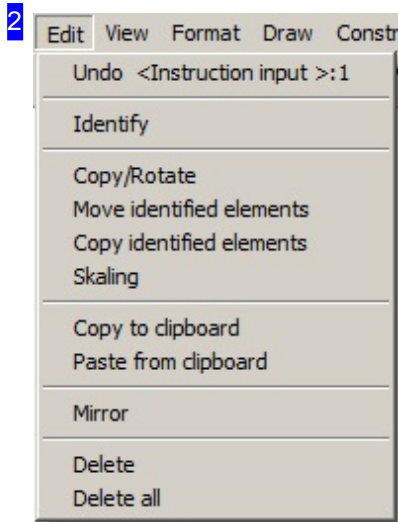
The bottom frame allows you to select or enter the scale of your sheet.



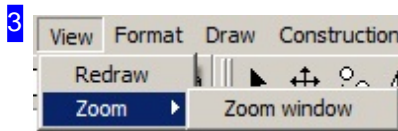
Drawings and layers can be exported from the module. More in the chapter on 'Exporting drawings and layers'.



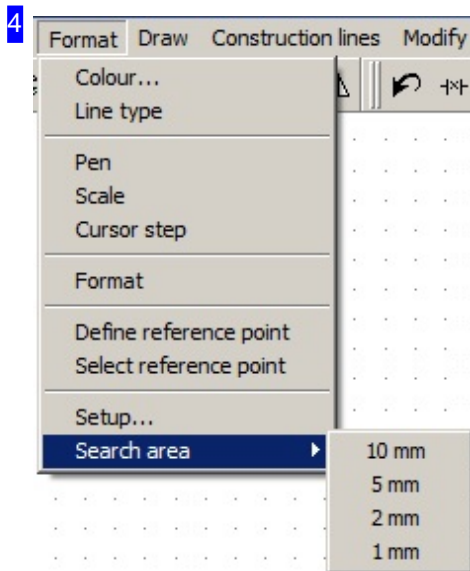
This dialog allows you to connect with external programs which you may wish to use while using *ncCAD32*. The programs listed here are added to a submenu under 'Execute...'. Click 'Insert' to add a program to the list.



Under 'Edit' you'll find the options for selecting and editing drawing elements. To edit an element you must first select it, then click the relevant edit command. The editing options are described in their own help page.

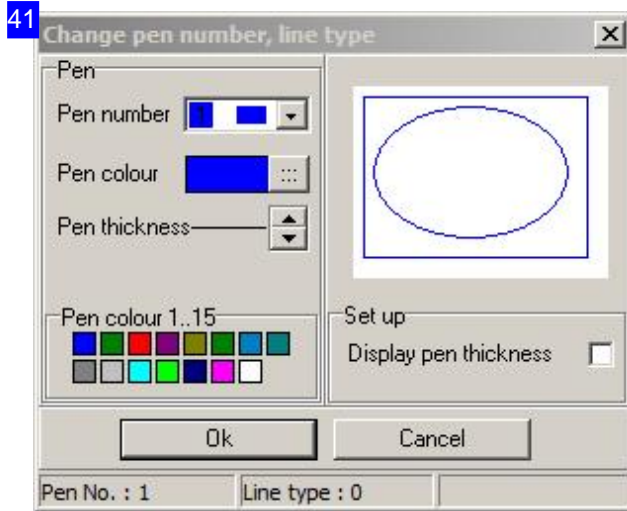


'Display' allows you to redisplay your whole drawing, or zoom on a particular region.



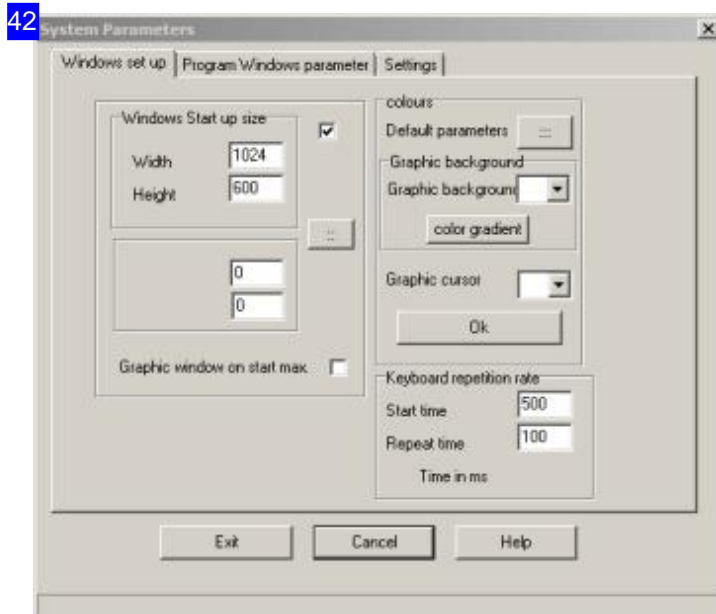
Commands in this menu will sometimes call a command which requires further input, this will be prompted in the command line.

- o 'Format' allows you to set rules for colors, lines and pens, and manipulate the origin. Entry is made in a dialog. Rules allow you to group elements for better control and structure when editing.
- o 'Scale' allows you to set the scale for your screen. The scale can be set only for an empty layer or new drawing.
- o 'Cursor step' defines the cursor's X/Y step size in mm when using the cursor keys.
- o 'Format' allows you to set the DIN paper size [0..4] of your drawing.
- o The 'Settings' menu item allows you to change parameters for Windows® as well as *ncCAD32*.
- o The last item 'Search area' defines the search area size around the cursor. This is shown visually by a larger or smaller cursor size.

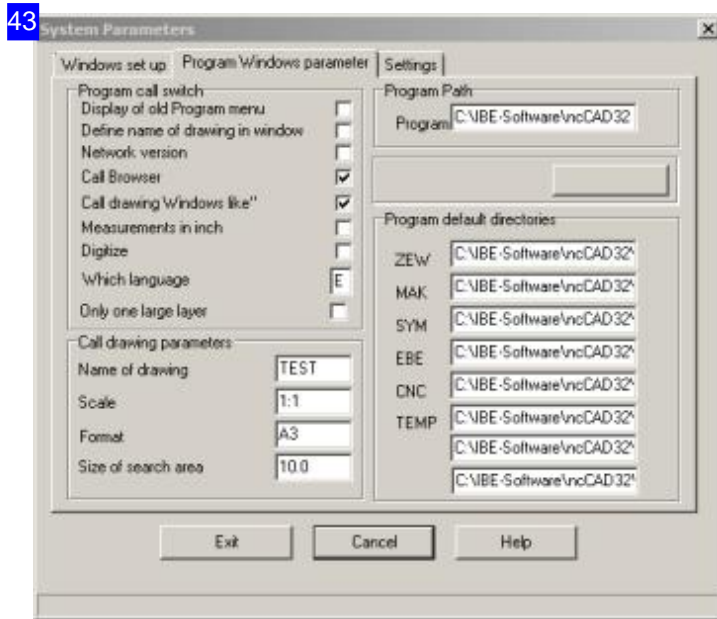


Set the pen types and allocate types, thicknesses and colors to your lines. Tip: the pen thickness is only shown when the selection box 'Show pen thickness' is checked.

The line types are given certain functions when editing. Line type 0 means cut lines, 2 and 3 are markings and 5 is an open contour. The other numbers are post-processor dependent, more information can be found in your post-processor help.



Enter window size and program color settings in this dialog. More program settings in the second tab.



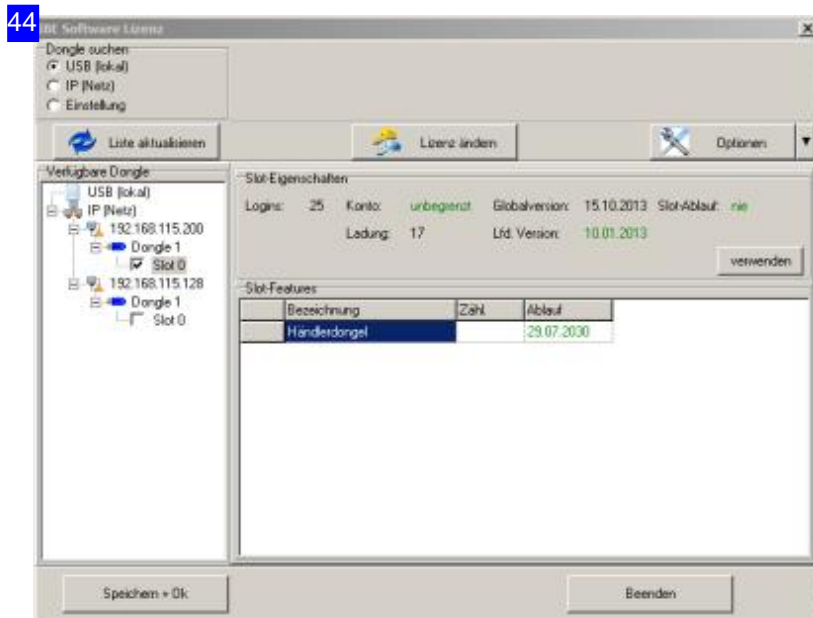
Program settings for *ncCAD32* can be made in this dialog. Select the language and mark the relevant checkboxes. Under language enter 'D' for German and 'E' for English. To join all layers to one large layer, mark the bottom checkbox.

The frame to the bottom left allows you to enter a default drawing name, scale and paper size.

The 'Program starter' frame allows you to open the license manager with the 'settings' button. Here you can add modules and permissions to your license. The program and optional modules are dongle protected and can be accessed only with an authorized dongle.

The dongle can be used locally or a network dongle can be used from your server.

Authorization of a dongle is carried out by IBE Software GmbH or our dealers; click the 'license window' to learn more.



Authorization is done by swapping 8-digit authorization codes, or using a license program from IBE Software GmbH or an authorized dealer. The dongle will keep your codes up to date and insure secure licensed use of the software. If you reach the end of your licensed use of the program then your dongle will need to be reauthorized by IBE Software GmbH or an authorized dealer.

Authorization will be made through a dialog:

1. Call your maintenance partner.
2. Select the program starter.
3. Tell your maintenance partner your customer code as displayed in the license window.
4. Your partner will give the new license code, enter this here to reauthorize your system.

When using a IBE dongle, authorization/maintenance is carried out using a license file which is updated online simply by clicking 'check Internet'. The license files are individually created for each dongle, and the dongle must be identified by the license server before a new license file is issued. The license file can also be sent to you offline and copied into your system this way. To do this click 'Install license file'.

Another option would be an extension using an authorization code which can be entered in the 'Manual entry' frame.



'Draw' contains the commands for adding elements to your drawing.

- o Circle
- o Arc
- o Ellipse
- o Tangents

have sub-menus for detailed user entry.

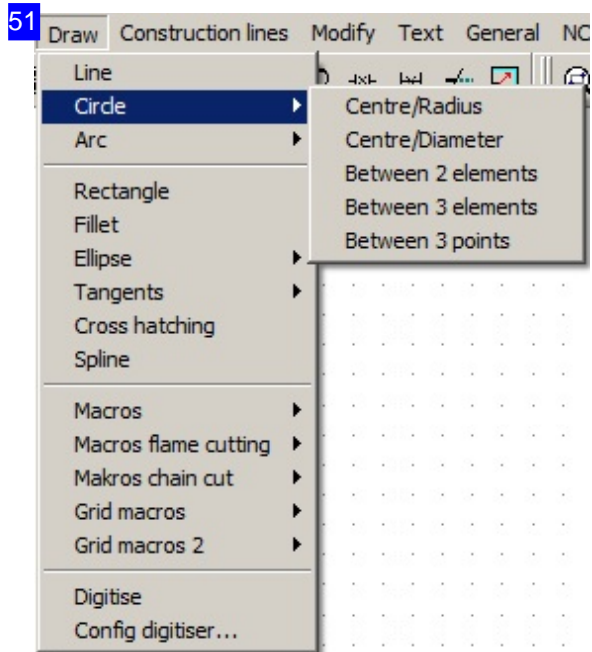
These commands are started with one click:

- o Line
- o Rectangle
- o Rounding
- o Spline

'Engraving' begins the command sequence for adding an engraving to your part.

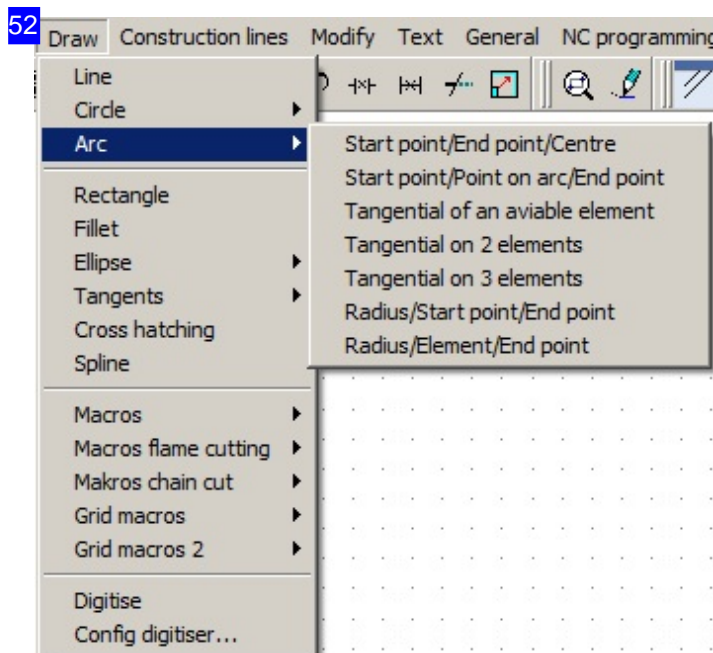
The macro block contains predefined macros sorted by function.

If a digitizer board is connected to your system, the bottom menu item will open the interface and begin Digitizing. Menu items are displayed only if the relevant optional program module is installed.



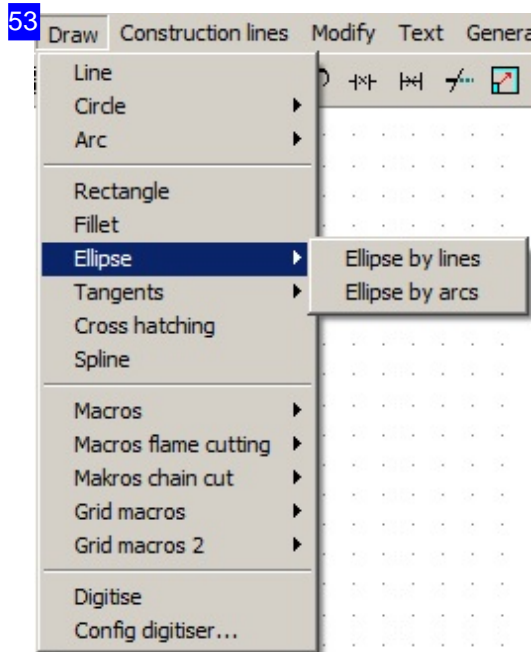
Drawing circles with various start points.
Examples in the relevant chapters:

- o Circle using radius.
- o Circle at two elements.
- o Circle at three elements.
- o Circle through three points.



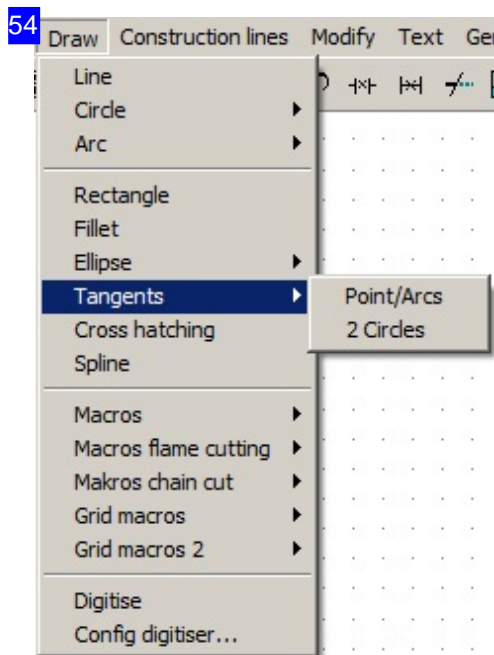
Drawing arcs with various starting information. Examples in the relevant chapters:

- o Arc through arc points and center.
- o Arc through three arc points.
- o Tangential arc from an element.
- o Tangential arc from two elements.



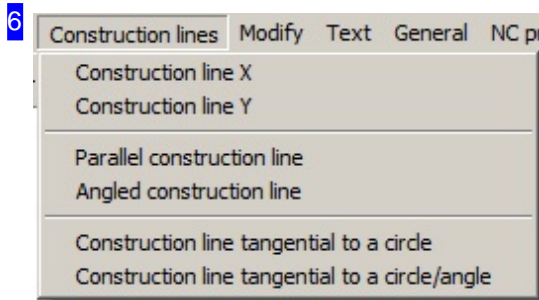
Drawing ellipses with various data. Examples in the relevant chapters:

- o Full ellipse with all data.
- o Draw full ellipse from lines.
- o Draw full ellipse from arcs.

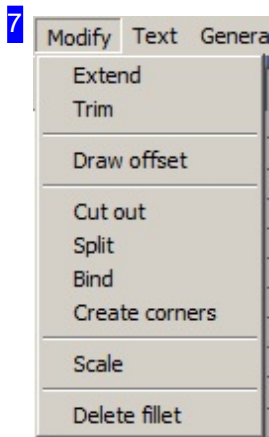


Lay tangents on one or two circles. Examples in the relevant chapters:

- o Tangents on one arc/circle.
- o Tangents on two circles.

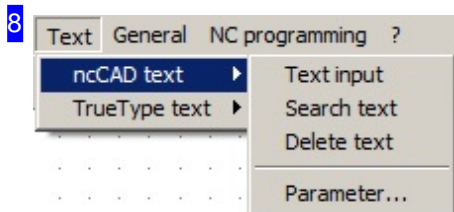


This menu allows you to add construction lines freehand or from existing drawing points. More information in the chapter on 'construction lines'.



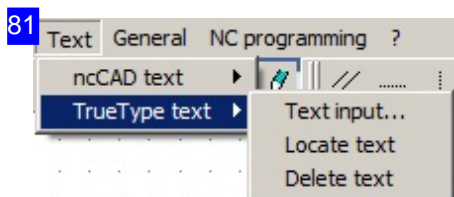
Modify drawing elements as you require. Select the relevant menu item and identify the required elements as prompted in the command line.

- o Trim - Allows you to stretch an element to meet a bordering element.
- o Adapt - shortens an element to the next drawing point.
- o Draw equidistants (see chapter on 'equidistants').
- o Separate - Removes a section of an element between two other elements.
- o Divide - Divides an element into n equal parts. The command is executed in the command line (selection 3) and you will be prompted for the number of parts.
- o Create corner - Joins two elements to form a closed corner.
- o Join - Joins two elements.
- o Area scale - Adds a scale to the drawing; release using 'Redraw'.
- o Remove rounding - Removes a fillet from a corner.

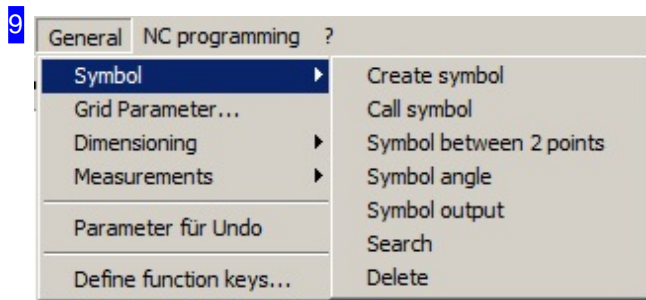


Find and edit simple texts for *ncCAD32*. See the chapter on 'entering simple texts'.

'True-type texts' have their own submenu.



Find and edit true-type texts. See the chapter on 'entering true-type texts'.

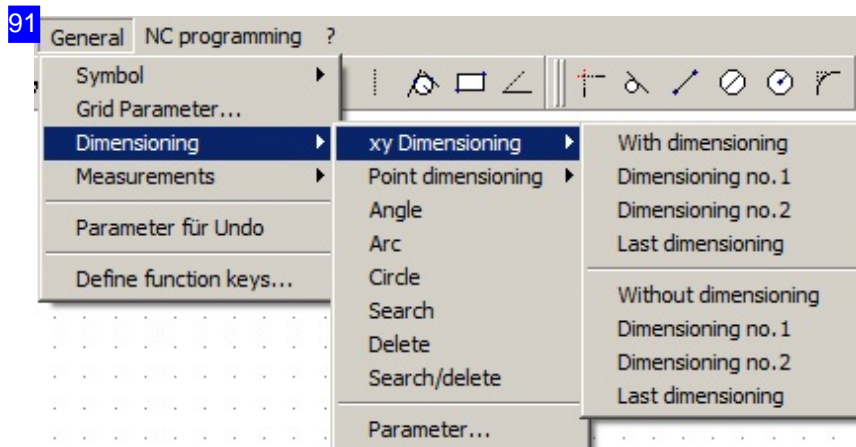


'General' settings and definition of measuring methods.

This menu allows access to dialogs for setting the grid parameters, undo parameters and function key usage.

The options for dimensioning are offered in their own submenu.

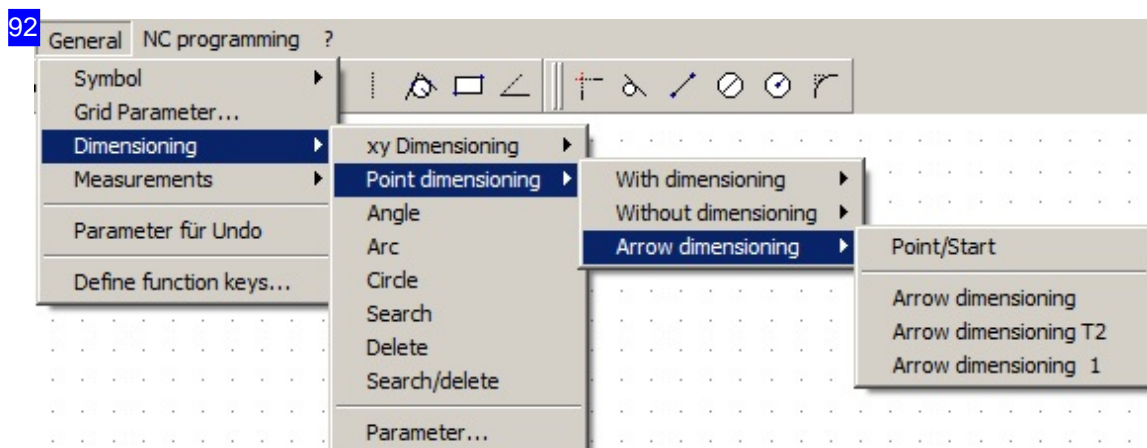
The use of symbols is described in its own chapter, as is measuring.



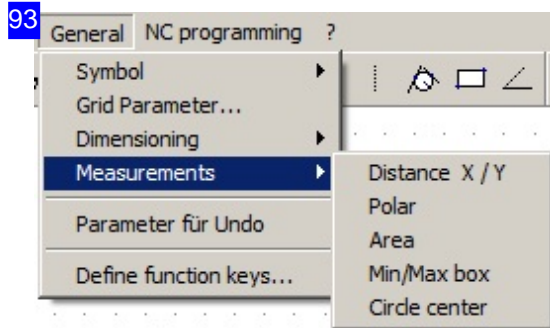
'XY Dimensioning' allows lines and construction lines to be dimensioned with cascading lines, with or without dimension numbering. More information can be found in the 'Dimensioning' chapter. For point dimensioning, select the corresponding submenu.

There are separate dimensioning functions for angles, arcs and circles; these are called via the input field.

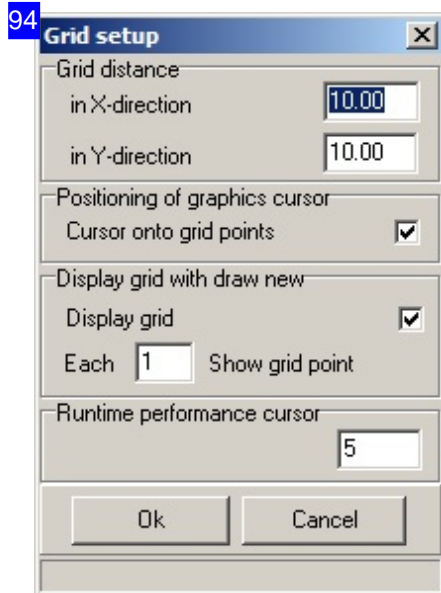
Settings for dimensioning can be made under 'Parameters'.



'Point dimensioning' allows you to mark a measurement between points. More information in the chapter 'Dimensioning between two points'.



The menu item 'Measure' allows you to precisely check all sizes. Notice and follow the prompts in the command line, to reach your required result.

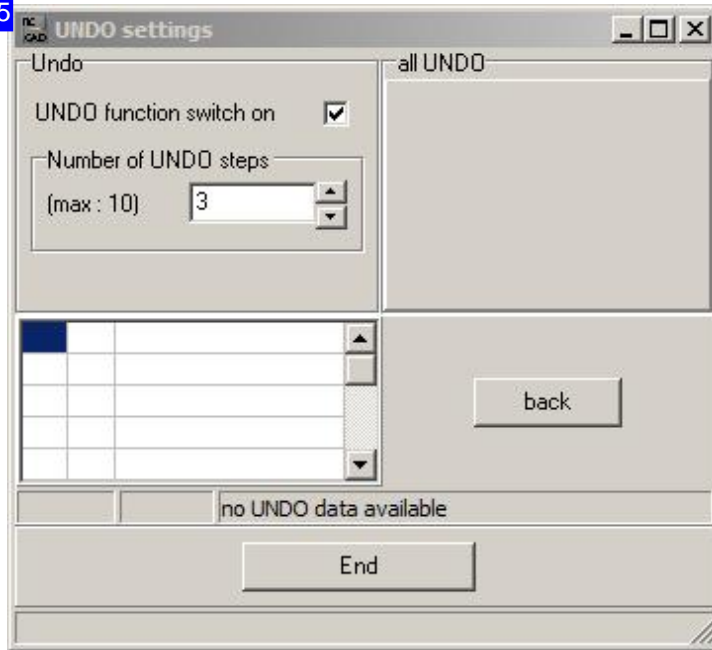


This dialog allows you to setup the grid parameters. Enter the grid size and mark the checkbox for snap to grid. The bottom frame allows you to set a snap to grid delay to allow you to also click between grid points.

Enter the conditions to be used when redrawing.

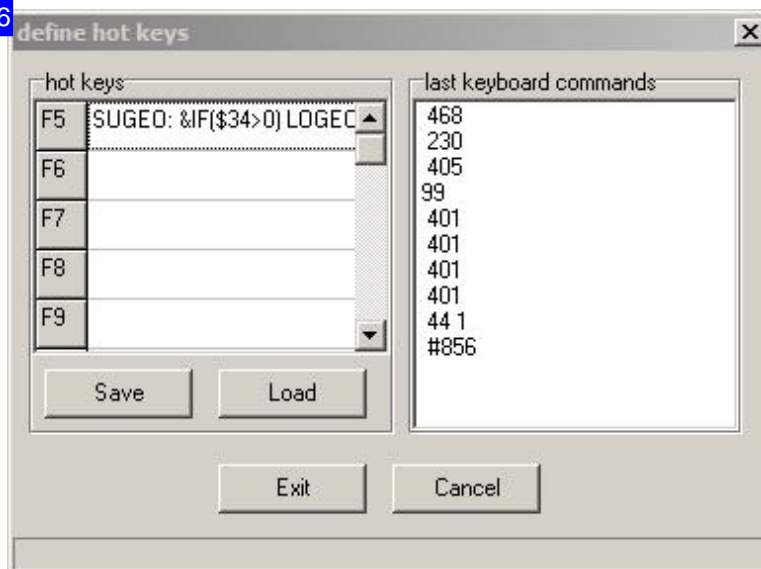
The grid display can be activated and deactivated using a button in the format line.

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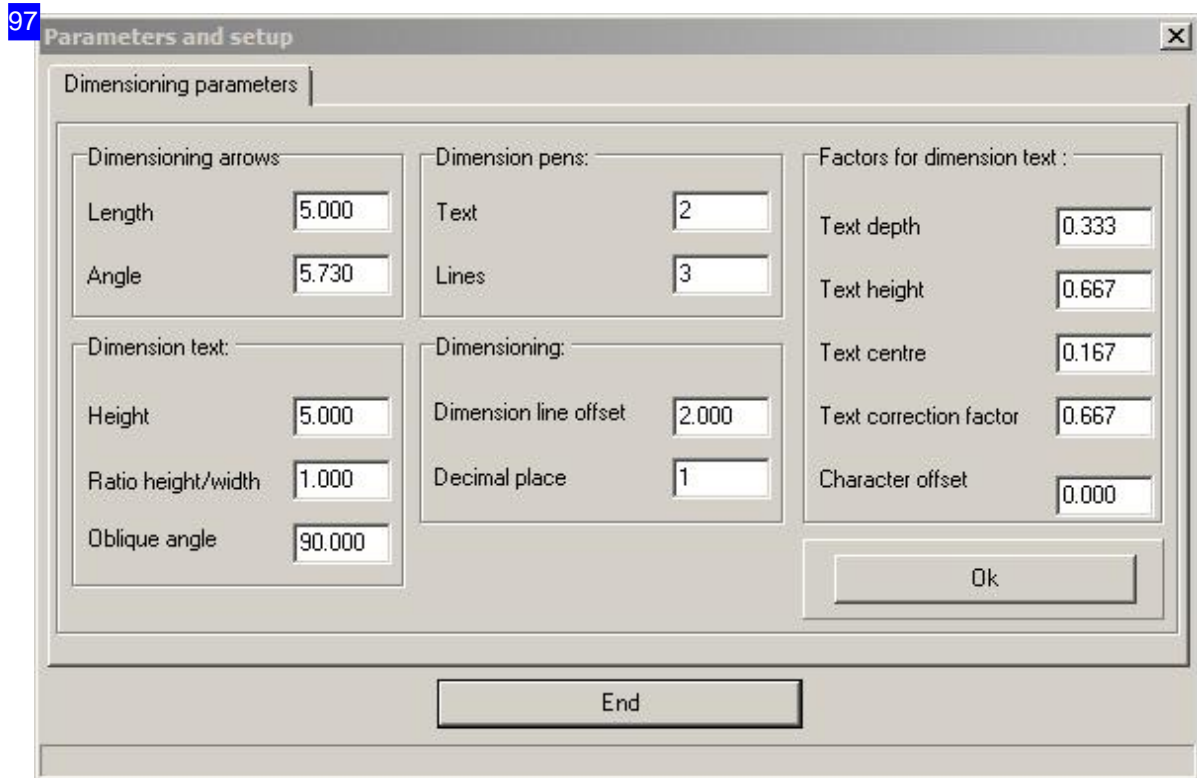


The 'Undo settings' dialog allows you to switch the undo function on or off, and set the number of undo steps that should be retained.

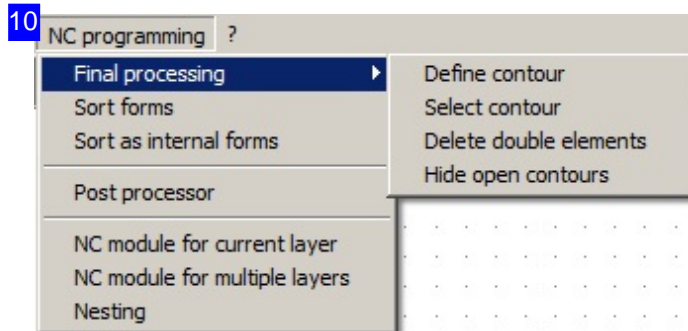
96



The 'Function key usage' dialog allows you to define commands that should be executed when you press the f keys from <f5> to <f12>. Enter the command code here, or drag-drop a command from the list to the right. Remember to confirm and save your selections before leaving this menu. Click 'Load' to access saved favorites; you can use different files for different selections / job types.



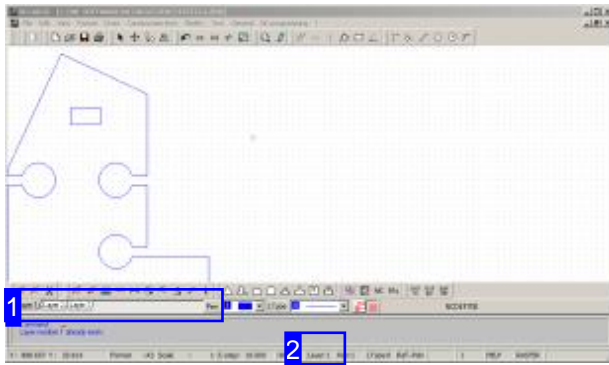
The 'Parameters and Settings' frame allows you to enter the margin conditions for dimensioning. Set the dimensioning line, numbering and arrow sizes. Click 'confirm' to save your settings.



'NC processing' allows you to make all settings required to turn a drawing into a machineable part. Processing modules can be stamping or cutting modules which create NC code in cooperation with your post-processor.

These functions are optional, see the chapter 'NC processing'.

Drawing Layers



Drawings can be made up of multiple layers. Layers allow you to group various functions and give your drawing structure. Thus you can keep dimensioning texts separate from holes, separate from etched texts, or keep all drilled holes separate from cut holes. *ncCAD32* can work with up to 100 layers.

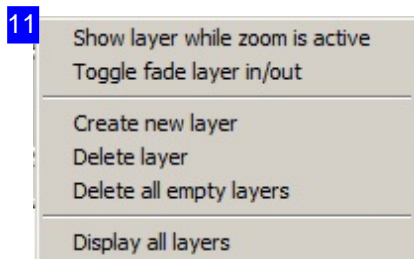
Layers are automatically applied when importing a drawing. A tab will be added to the drawing [1] for each layer, allowing quick switching between layers.

A layer can also be empty! Management is completely left to the user, thus there are two commands 'delete' (i.e. empty content) and 'remove'. You can also add layers at any time.

The element count for a layer is limited to 4000, If you need to create larger drawings with more elements than this, you can join two layers together and edit them this both at the same time in this 'one' very large layer. The layer setting is then superfluous.



A tab is shown here for each layer used. Click a tab to switch to the corresponding layer. Right-click for a context menu with options for layer management.



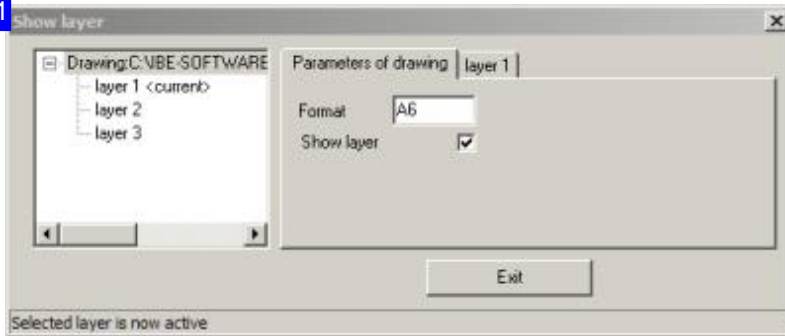
This menu allows you to manage your layers. You can create layers, delete their contents, and remove empty layers.

2



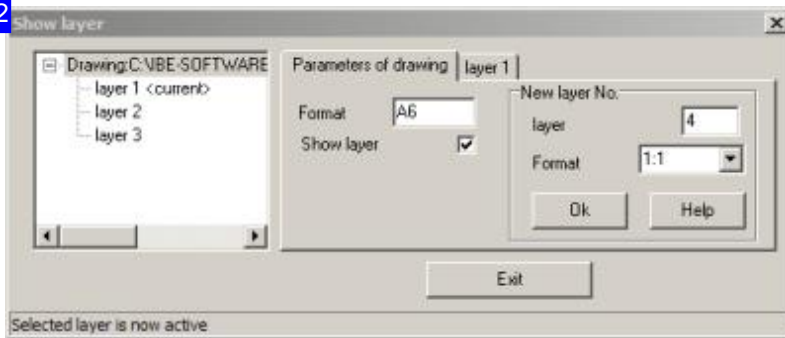
The status line shows the current layer number. Click this field to open a context menu with options for layer management.

21

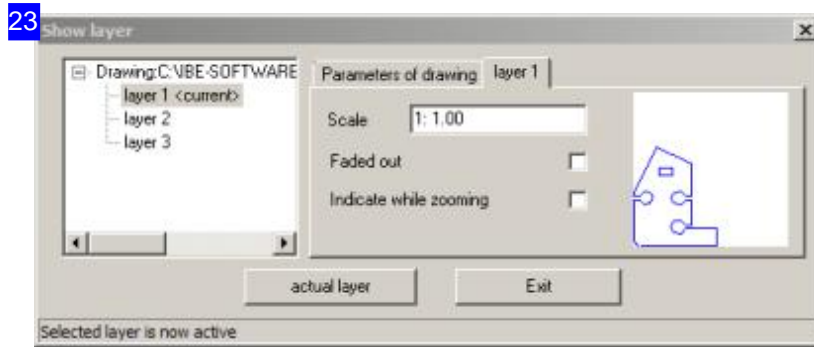


The list to the left shows the drawing layers. Click a list entry to open the corresponding info tab to the right. The 'Drawing parameters' tab allows you to enter the format and create 'New layers'. To edit a layer, select it in the list to the left, this opens the layer tab. Right-click a list entry to open a drawing menu or layer menu.

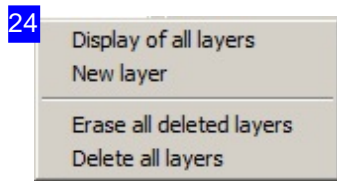
22



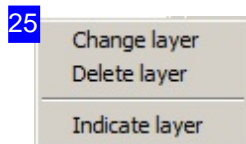
The frame to the right allows you to change a layer's format.



This tab gives information on a selected layer and a preview in the right-hand frame. To edit the drawing, mark the corresponding line in the list to the left, the drawing tab will be displayed.

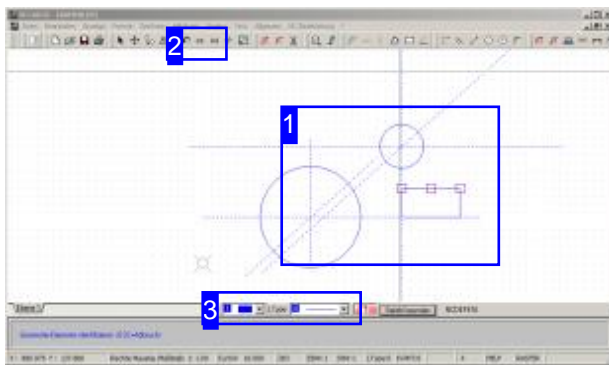


This menu allows you to manage your drawing layers. You can create layers, empty their contents (delete) and remove empty layers.



This menu allows you to manage your drawing layers.

Construction Lines and Identification



Construction lines are an important step in creating you parts. Construction lines give a reference point while building your workpiece. The usefulness in *ncCAD32* is stronger still through precise recognition of points including centers and intersections.

Lines can be identified individually or in blocks, *ncCAD32* offers several options.

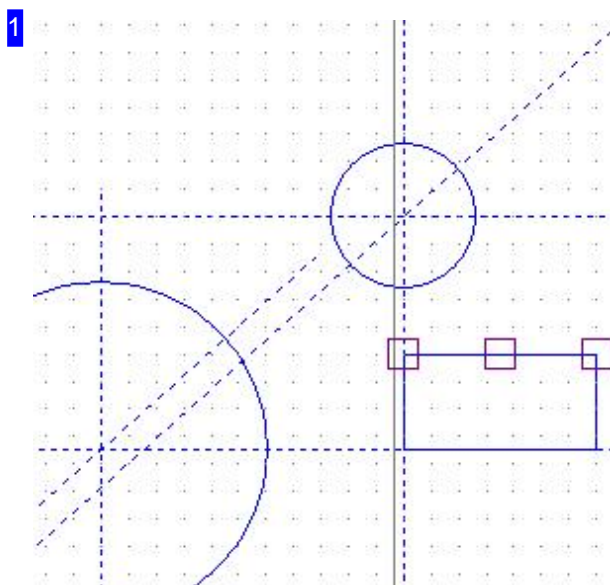
Construction lines can be generated using the menu panel or the Icon bar .

Construction lines can become part of the workpiece (note the trim functions) or be deleted when the drawing is completed. In this case it is useful to use a separate pen number or line type for easy identification later on.

Simple construction lines are always drawn across the displayed area; choose from:

- o Parallel construction line.
- o Vertical construction line.
- o Horizontal construction line.
- o Construction circle from two elements.
- o Construction line at a given angle.

You can also use special construction lines such as a tangential construction line or Equidistant. For drawing you can may also find the 'parts' function useful, whereby a section is separated into equal parts.



In this example the construction lines are shown as dashed lines. The upper side of the right-hand rectangle is selected, visible by the marking of the end points and center. Right-click to access a menu to identify lines in a block.



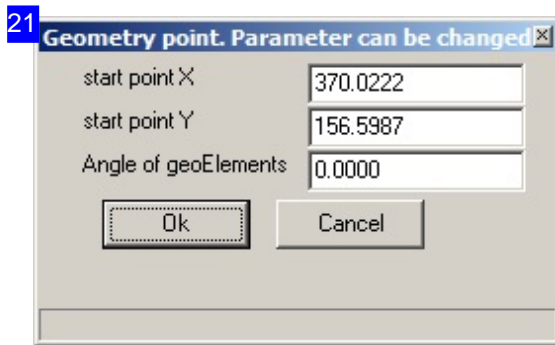
Click the left-most button for selection mode. Next select a contour element or stretch a box to selection more than one element.

Click an element to mark its drawing points. You may then drag and drop individual drawing points. Press the space bar to select a whole contour group. Right-click a drawing point to open a info dialog on that point.

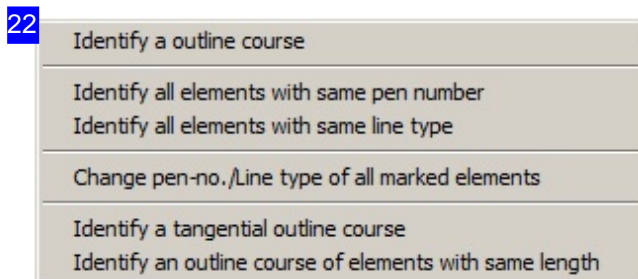
Right-click on the drawing board to access a selection menu for identifying whole contour groups.

Press <Esc> to exit the function.

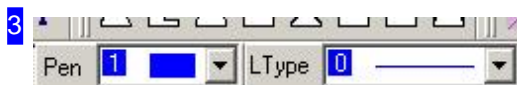
<Alt>-click to snap to the nearest drawing point.



In this dialog allows you to change a drawing point's parameters. The dialog's content is context-sensitive.

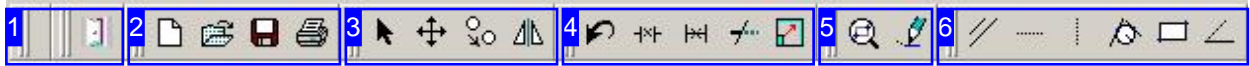


This menu allows direct access to various functions. You can identify groups of contours or transform to another form. Skilled use of transformation allows noticeable reduction of the overall element count.



Select a specific pen number or line type in connection with the selection menu to select groups of elements according to their next purpose.

Entering Commands Using The Icon Bar



The icon bar allows you to access your most frequently used commands with one click.

The icon bar is dynamic and can be adapted to suit each user. The individual sections can be moved in one or more rows as well as by the left and bottom edges, and can be shown or hidden.

There is always one empty element on the bar to access the context menu. This cannot be removed. Right-click on an icon to open a menu which shows all packets. Use this menu to alter the panel as required.

The individual menu sections can also be separated from the icon bar and left as floating tool windows anywhere on the screen.

The icon bar functions can be moved, and in this case grouped in the following work packets:

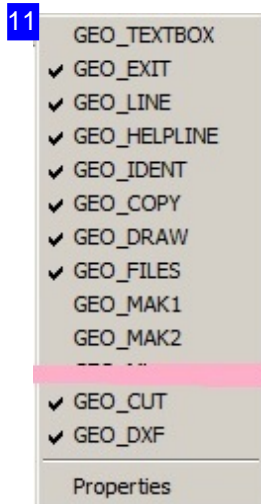
- o Empty element and exit program [1].
- o Open, save and print files [2].
- o Select, modify [3].
- o Undo, adapt elements [4].
- o Zoom and redraw [5].
- o Draw construction lines [6].
- o Draw elements.
- o Search and delete.
- o Search and measure.
- o NC preparation and post-processor.
- o DXF selection and Checker
- o Macros

There is a tool tip available for each icon by hovering over it.

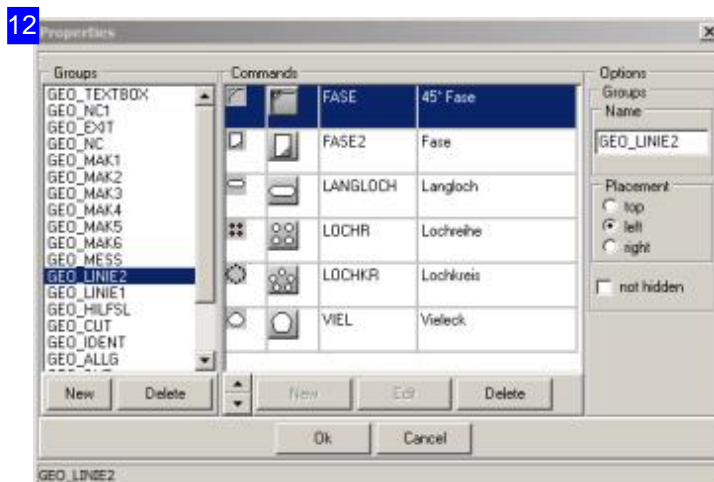


The empty block to the left is the icon bar's anchor and cannot be deleted. You can use this section to access the selection menu to show or hide the individual menu blocks.

The right-hand symbol exits the program.



In this menu you can see all menu sections which can be shown or hidden. Click 'Properties' to open a dialog for editing the menu points.



Select a menu item to view and edit its properties.



In this block you'll find the icons for file management:

- o Create new drawing .
- o Load existing drawing. In this case it's a ZEW file in IBE's proprietary format. A file manager opens to select the file.
- o Save drawing . If the file has been saved it will be saved with the same name, overwriting the old version. If this is a new drawing, a file manager will open to set the save path.
- o Print drawing.

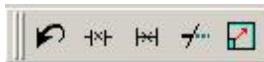
3



The buttons from left to right:

- o Activates the 'select an element' function.
- o Move elements
- o Copy elements
- o Mirror elements

4



The left button undoes the last action.

The next three buttons allow you to adapt elements. The commands are identical to those in the menu panel under 'Edit'.

- o Delete part element between two elements. Mark the part to be deleted. The element will be separated and fitted to the bordering elements. Mark only one side to fit only one side.
- o Fit element between two elements. The overlaps with bordering elements are removed. Mark the required side, or the middle to trim both sides.
- o Direct trimming of two geometric elements. Mark two converging elements sequentially to close the gap between them.

5



These buttons mean:

- o Show zoom window.
- o Redraw. Alternatively via this pop-up as 'Reassemble'

6



These buttons allow you to add construction lines.

- o Draw parallel construction line.
- o Draw horizontal construction line.
- o Draw vertical construction line.
- o Draw construction circle on two elements.
- o Draw construction rectangle.
- o Draw construction line under an angle.

Tip: select a separate line style for construction lines to aid drawing structure

and management.

13



These icons are for drawing elements. Note the tips in the Command line ; you will be prompted through the selected function.

- o Create corner.
- o Draw tangential construction line.
- o Enter line by start and end points.
- o Draw circle with diameter.
- o Enter circle by start and end points.
- o Add rounding.

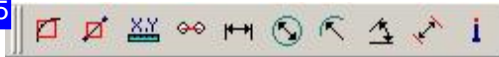
14



These buttons call delete functions:

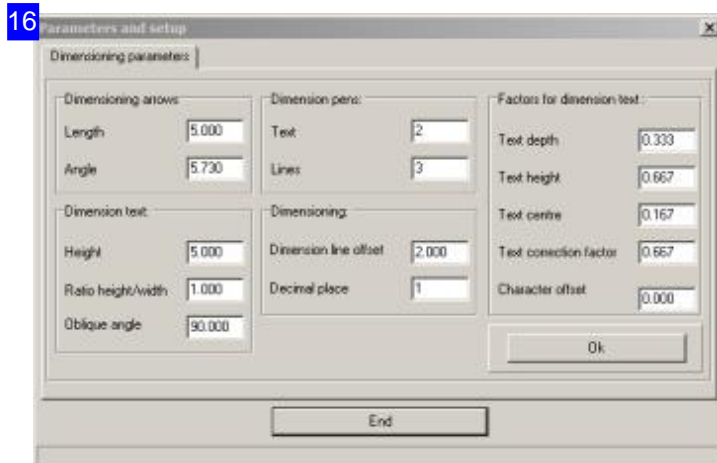
- o Delete element.
- o Delete circle or arc.
- o Find and delete element.

15

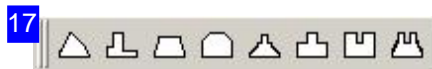


This block offers functions for dimensioning workpieces:

- o Find arc.
- o Find line.
- o Display X/Y offset.
- o Display circle/arc offset.
- o X/Y dimensioning with text.
- o Circle dimensioning.
- o Radius dimensioning.
- o Angle dimensioning
- o Point dimensioning with text.
- o Display dimensioning parameters.



In this dialog you can see and set general information for dimensioning lines, arrows and texts. Click 'Confirm' to save your changes.



Here you'll find functions for last minute adjustments of your drawing before processing takes place.

- o Delete duplicate elements.
- o Sort elements.
- o Call NC module.
- o Choose post-processor.



Quick-keys for using DXF files:

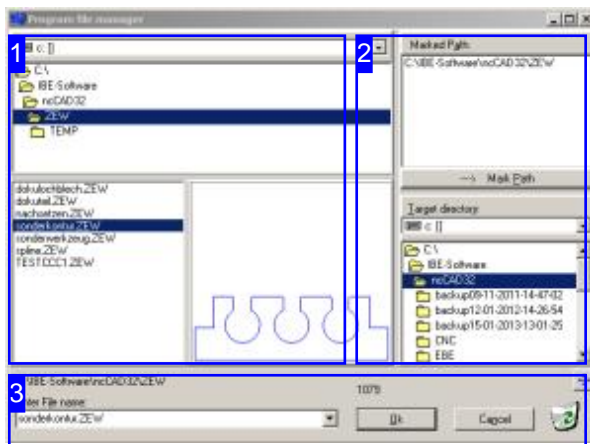
- o Import DXF drawing.
- o Save DXF drawing.
- o Open DXF Checker .



To quicken and ease recurring jobs, *ncCAD32* offers a selection of macros. These can make drawing various essential parts much quicker and easier. More information in the 'Macros' chapter.

File Management

A Special Dialog

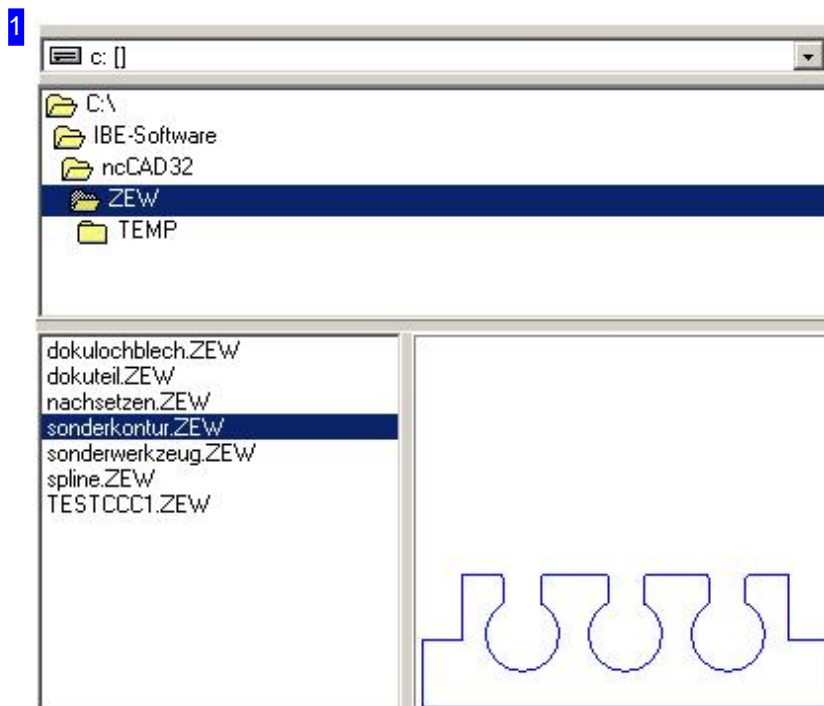


ncCAD32 calls this dialog for various purposes (load, save, import, export, etc.) and the file selection is adapted accordingly.

The system save format for drawings in ncCAD32 is 'ZEW' - a format which allows values such as technologies to be added to a regular drawing. This format is always used as a filter when opening drawings. Other formats can be used via the Import/Export options.

In the case you want to transfer the program to an other computer, the command #467 will help you to save and retrieve your system data.

ncCAD32 uses a special dialog to load files which contains a file manager with various management functions and also a preview of the files being viewed.



This screen shows the general navigation of your directories in the top pane, the contents of that directory are shown in the lower left pane and a preview of the selected file in the lower right pane. A preview can only be generated for the correct file types. You can zoom the drawing (mouse-wheel), drag (mouse-key) or display everything (double-click).



Here you can also remember preferred path names in the top pane to help find your drawings quicker.

The lower pane is for navigating files' save paths. You can drag and drop files from the file overview [1] to folders displayed here.

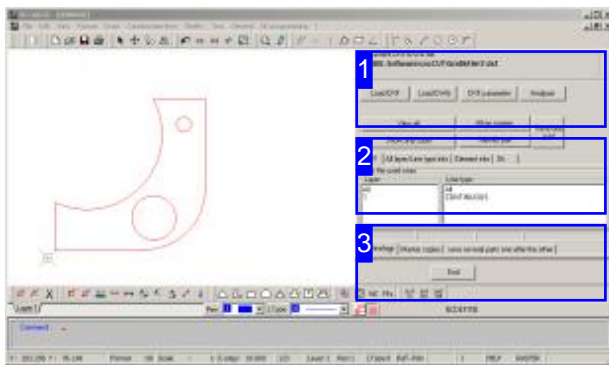


This pane shows the name of a file selected to be edited, or the entered name of a file to be saved.

The recycle bin on the right allows you to delete files; drag the file to the recycle bin.

The button at the top right allows you to fix the position and size of the dialog for the next time you call it.

Import Drawing And Layers



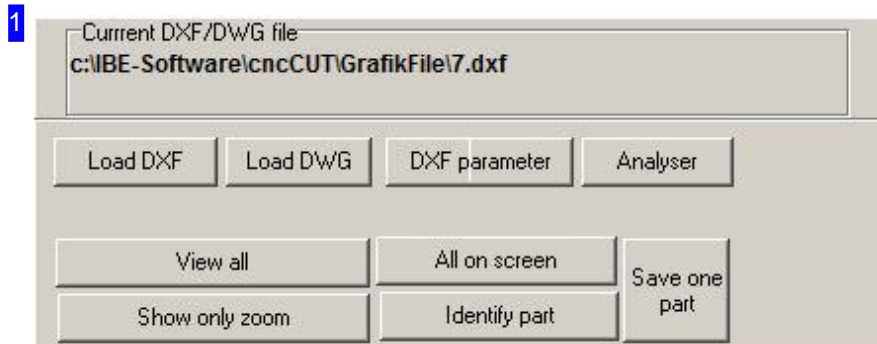
The menu item 'File' allows you to import DXF, DWG and EBE files as drawings.

1. Define the drawing import parameters in the 'Settings'.
2. Import drawings in the following formats:

- o DXF/DWG
- o EBE
- o Mega CAD using the CDL interface

ncCAD32 allows you to import drawings of various types from multiple sources. For processing, many drawing types must be adapted; i.e. the data quantities reduced or categorized as required by ncCAD32 and your machine. Apt settings can be used to remove excess data and unneeded layers.

You can also create rules for allocation of technology data to your new contours upon import.



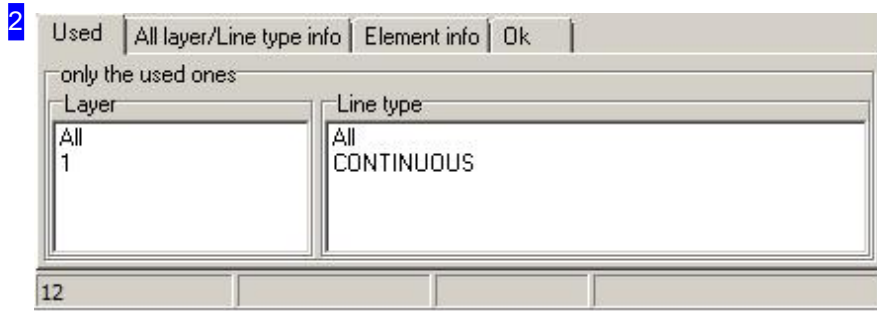
This screen allows you to load DXF and DWG files into the system.

To set rules, select 'DXF parameters' .

Click the corresponding button to select a drawing. A 'File management' dialog will open to select and import drawings.

Imported drawings can be analyzed to check for and correct potential errors or inconsistencies. Switch to the 'Checker' for this functionality.

The bottom-left buttons help you to bring the drawing to a clear overview, complete or sorted by layer; and identify a part of your drawing. A selected part can be saved using the bottom-right button.



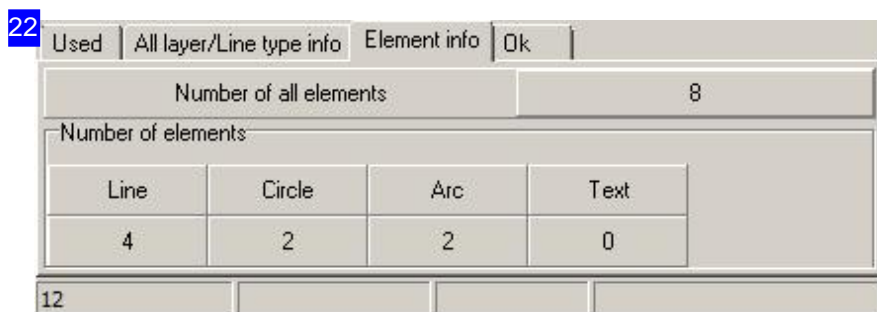
After loading a drawing, all its layers will be displayed here. Select a layer from the tabs below the drawing to show its contours.

For a complete view of all layer types switch to the 'All layers' tab.

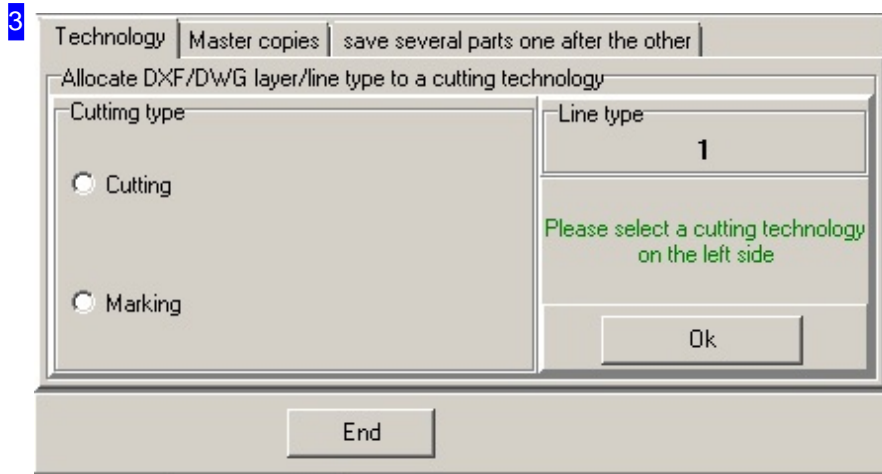
You can view an overview of all contour elements in the 'Element info' tab.



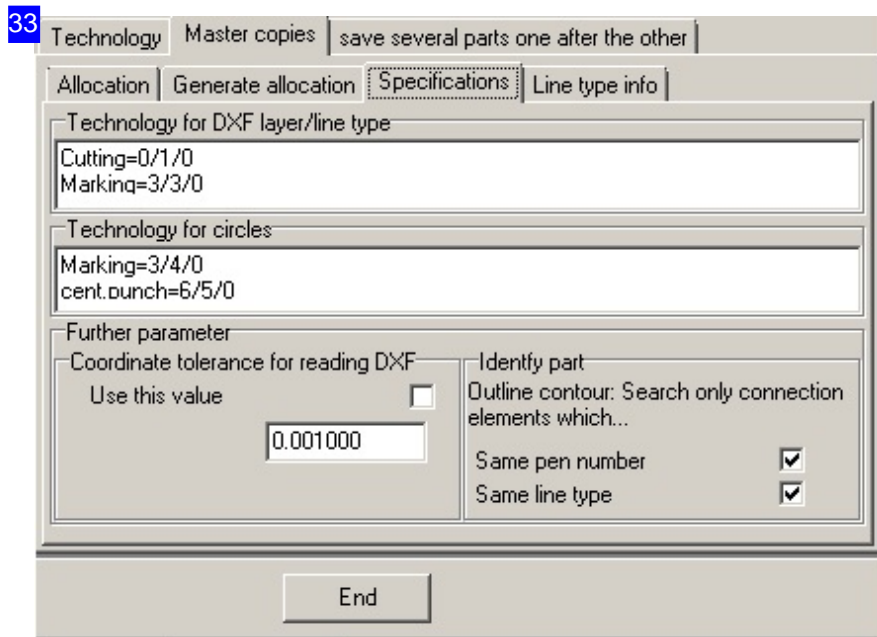
All available layers and line types are displayed here. Select a layer using the tabs below the drawing and its contours will be displayed.



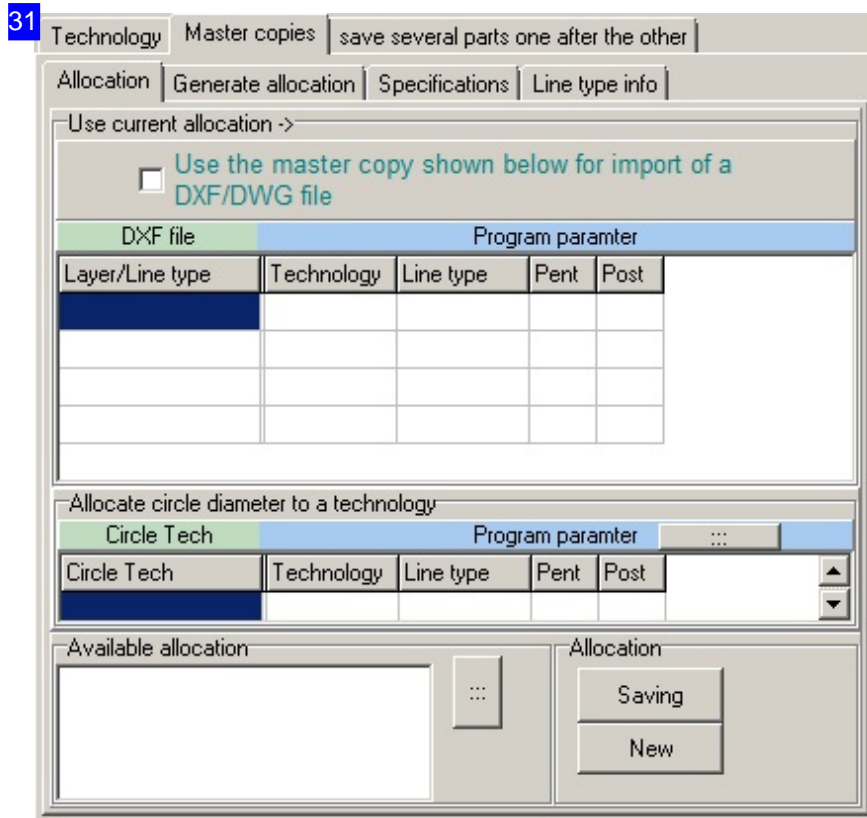
This tab gives an overview of all contour elements in the drawing.



To allocate technology data to a contour, select the contour type in the Overview [2]. Define the technology information for the contour, mark the selection box on the left and click 'confirm'. You can also create a template for allocating technology data in order to achieve a corresponding clean-up of the drawing at import. Switch to the 'Templates' tab. In the right-hand tab you can save multiple parts separately.



Presets and advanced parameters for the individual technologies can be entered here.

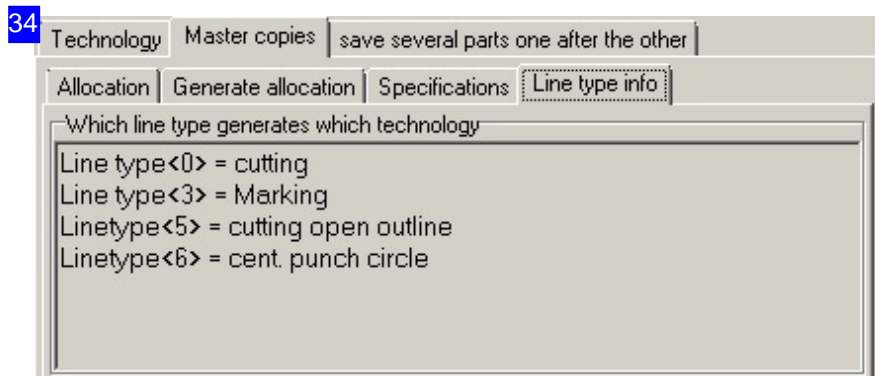


Here you can see the allocation of technology data to the contours. To allow ongoing use of this allocation mark the top selection box.

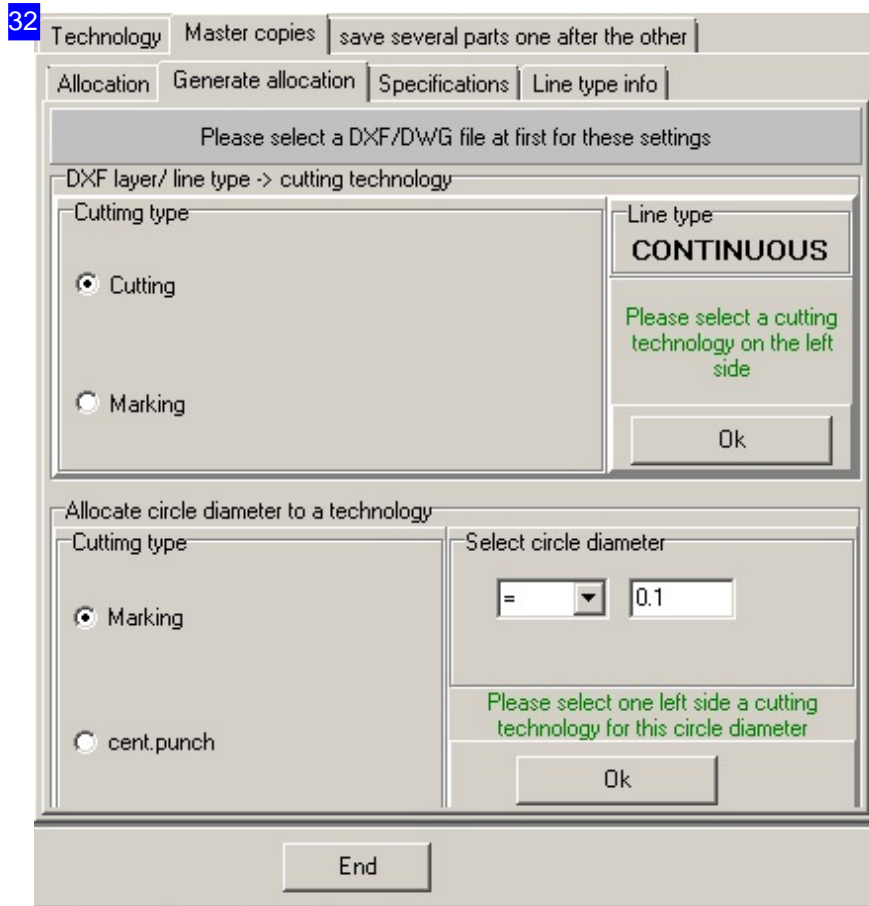
The allocations can be manipulated here. Mark a row and right-click to remove. A new allocation can be made in the next tab 'Create allocation'. Allocations for circle can be made direct on the drawing, to do this click the button next to 'Program parameters; in the second frame.

Don't forget to save the table after making changes.

To make a complete new allocations table select 'New' in the 'Allocation' frame at the bottom right.

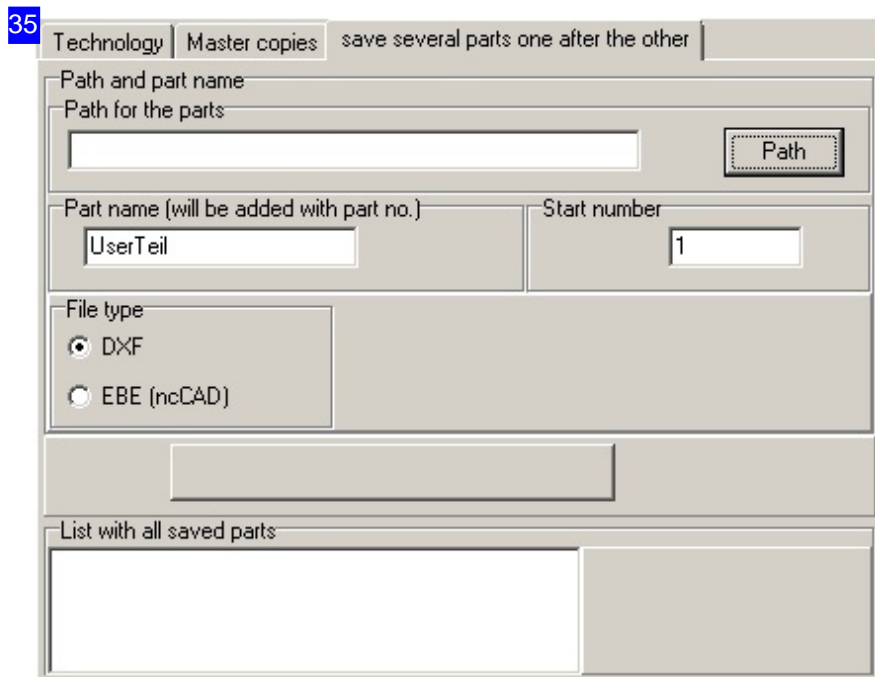


Here you have an overview of the allocation of line types to the cutting technologies.



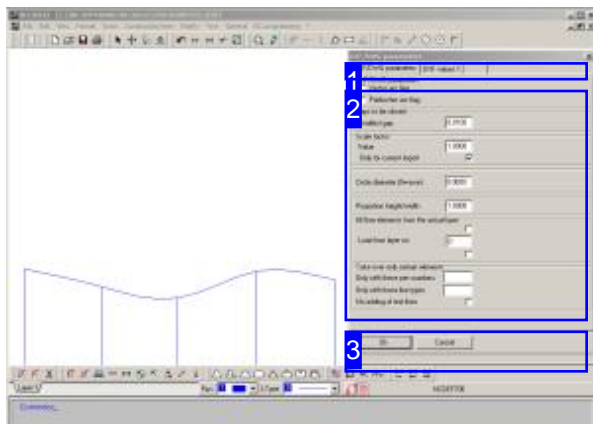
This tab allows you to create a new allocation of cutting technology data for a contour. Mark the contour type in the Overview [2]. Select a technology in the 'Cutting type' frame and click 'Confirm'. The allocation can now be seen in the list under the 'Allocation' tab.

The presets / parameters for individual technologies can be set in the 'Presets' tab. Information about the allocations can be viewed in the last tab.



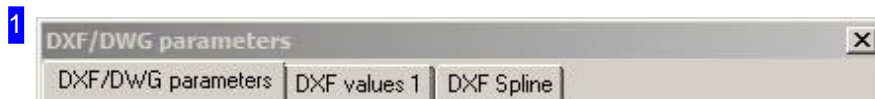
If a drawing has multiple parts, you can save them in individual files. Select a path for the save location, enter a part name and enter the number for the first part in the frame labeled 'Start number'. The parts will then be saved individually with incremental numbering. You can also decide which format to save in, select the corresponding button.

DXF/DWG Parameters



Enter the parameters for converting external drawing file types to the proprietary format. Check the values carefully before clicking on 'apply' to save.

Drawings from many sources have different file formats which must first be converted before they can be used in *ncCAD32* . This screen allows you to use exact parameters to trim drawings and import only set layers.



The setting options are sorted thematically in multiple tabs:

- o Arcs, holes, scaling, text pens and layers.
- o Commands, combining and ellipses.
- o Poly lines and splines.

2

DXF/DWG parameters

Vector arc flag

Publisher arc flag

Gaps to be closed

Smallest gap

Scale factor

Value

Only for current import

Circle diameter (0=none)

Proportion height/width

All Geo-elements from the actual layer

Load from layer no.

Take over only certain elements

Only with these pen numbers

Only with these line types

No adding of text lines

Enter the values and mark the relevant checkboxes:

- o Convert arcs from programs including VectorCAD and Publisher.
- o Close gaps up to a given size.
- o Scale the drawing to a given factor, e.g. from cm to inch using a factor of 2.54.
- o Display circles with the given diameter (when entered).
- o Text-line factor height:width. If text marking lines are too long you can compress them by entering a scaling factor of less than 1 here.
- o Layer number from which the drawing should be loaded.
- o Load only elements of a given type.

21

After import of a DXF/DWG file

Actual layer : first not empty layer

Connect parallel geometric elements

Execute ncCAD command

AV32 input

DXF/DWG-values

Settings for ellipses

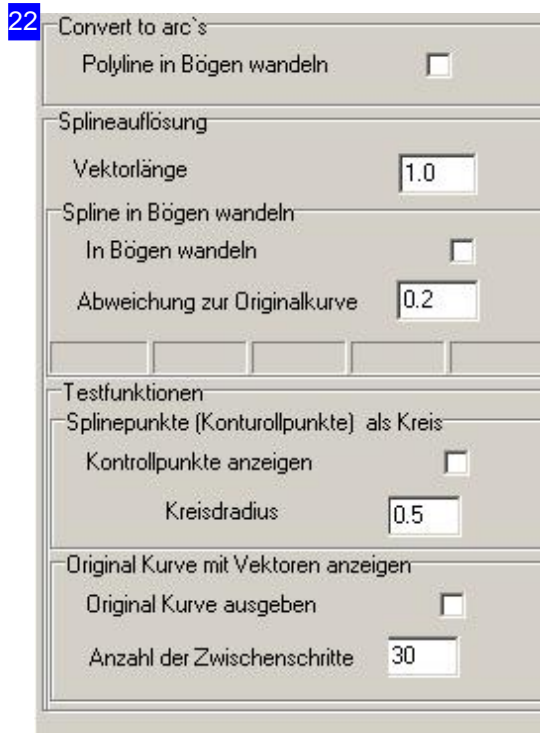
Convert ellipses to arcs

No. of arcs. (value<0 = than the correct no. of arcs) (value>0 = at the best spreading)

Ellipse vector length

Enter the values here and mark the relevant selection boxes:

- o Call *ncCAD32* commands after loading.
- o Read DXF/DWG values from the job management module.
- o Convert ellipses to arcs.



Spline conversion:

Use limiting values to replace an existing spline with an approximation. The dissolution of a spline can be done by entering new vector lengths and converting to arcs.

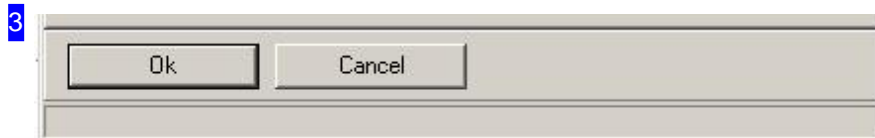
For dissolution into vectors, enter the maximum vector length. The default is 1.

For dissolution into arcs, enter the maximum deviation from the original curve. The default is 0.2.

You can display the original curve to check the change caused by your settings. Mark the selection box to convert the spline into arcs.

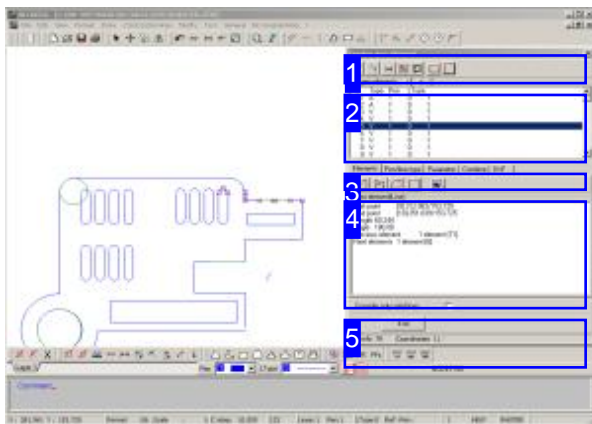
In the top frame you can choose to convert polygon lines into arcs.

Splines can be converted into lines or arcs in the bottom frame. You can display checkpoints or the original curve to check the deviation from the original.



After making these settings click confirm to save.

Repairing Errors with the Drawing Checker.



- o The upper panel [2] shows all elements present in the drawing. Those exhibiting errors are marked with a '>' symbol.
- o Errors can be repaired quickly using the buttons in the header panel [1].
- o The 'Elements' [3] tab in the bottom panel [4] shows the details of an element selected in the top panel [2], or a list of all elements exhibiting the same type of flaw. The 'Pen/line type' tabs [3] allow you to make settings to search by.
- o You can use the buttons in panel [3] to display elements by type.

The Drawing Checker offers you a tool to check the drawing of a component for errors and repair any you find.



From left to right, the buttons mean:

1. Shows the selected element in the overview.
2. Undo.
3. Delete the selected element.
4. Delete all duplicate elements.
5. Check if all contours are closed.
6. Close all gaps up to the given size.
7. Join all vectors/arcs with tangential intersections.

2

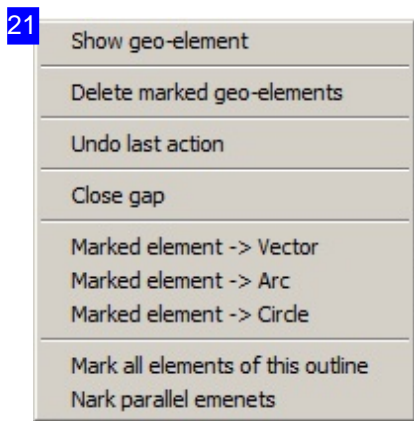
No.	Type	Pen	LType
1	A	1	0 1
2	A	1	0 1
3	V	1	0 1
4	V	1	0 1
5	V	1	0 1
6	V	1	0 1
7	V	1	0 1
8	V	1	0 1
9	V	1	0 1

All elements that make up your drawing are listed here. Those which exhibit errors are marked with a '>':

Click on an element to show details about it in the bottom panel [4].

Flawed elements will be marked in the drawing with a cursor.

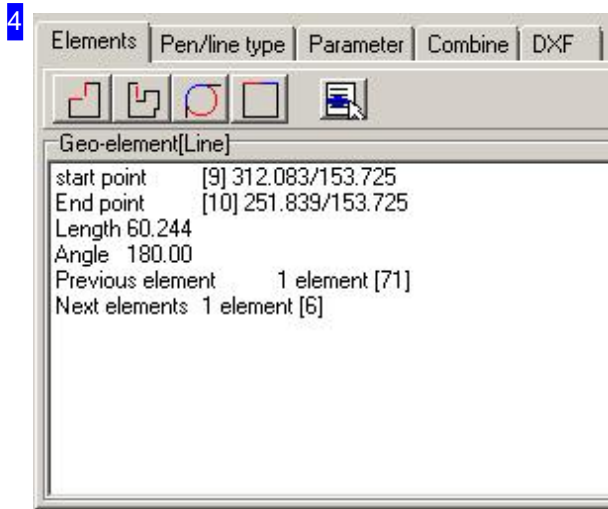
Right-click to open a context menu with options to repair errors.



This menu allows you to directly access commands for repairing errors.



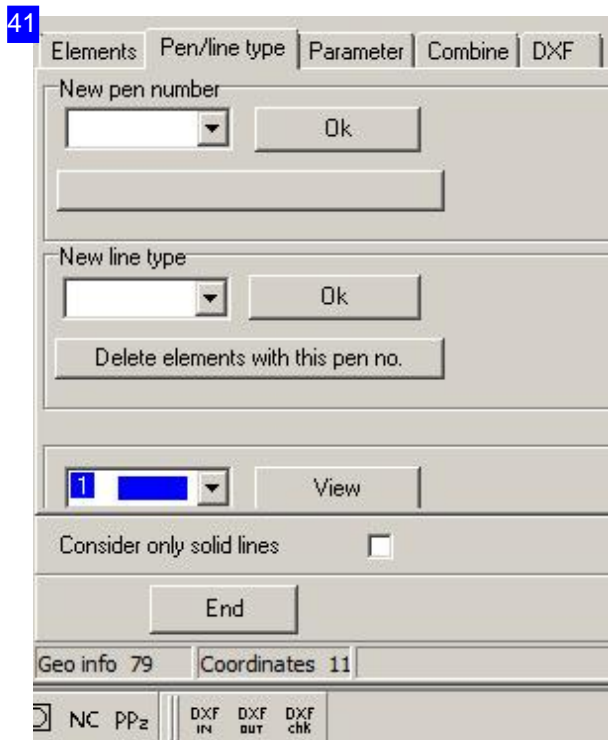
- o The 'Elements' tab shows the values for the element selected in the element list [2].
- o The 'Pen/line type' tab offers the option to search more accurately by a line's color or consistency.
- o The 'Settings' tab allows you to join tangential vectors.
- o The 'Join' tab allows you to join vectors and arcs according to presets such as length and angle difference.



This frame describes the vectors of an element in a drawing, or lists all errors of one Error type if you are searching using the buttons in panel.

The four top buttons allow you to seek for elements with a certain type of error. From left to right, the buttons mean:

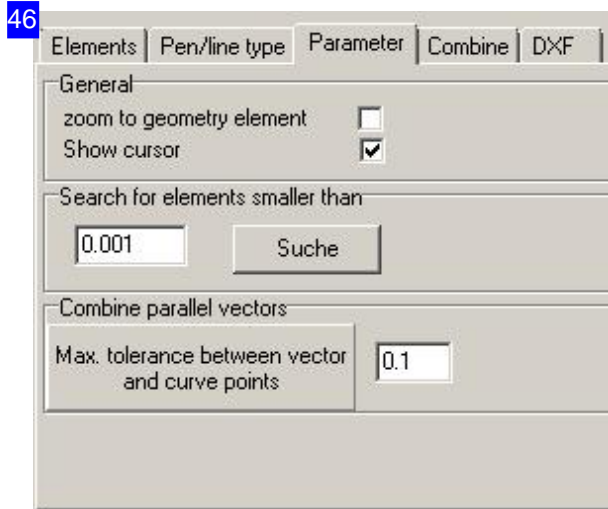
1. Find all elements without a connection to another element; one open edge.
2. Find all elements without any connection to another element; two open edges.
3. Find all circles whose centers and radii are the same as an arc's.
4. Find all vectors/arcs whose transitions are 180°.



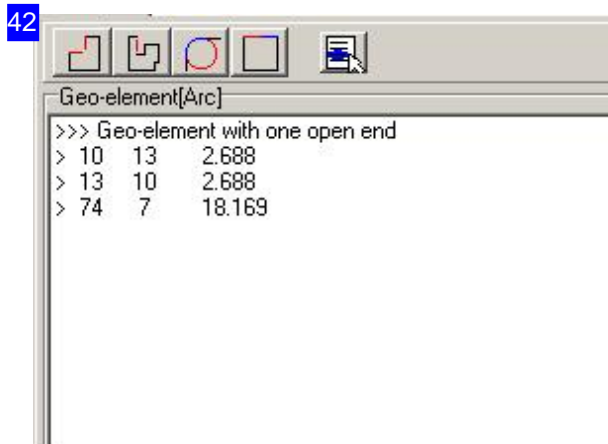
This frame allows individual elements to be allocated pen or line numbers. This 'marks' the element, allowing elements to be removed as a group, or searched within that group.

It is important to give open contours a special line type when cutting them. This line type (5) can then be cut together as an 'open contour'.

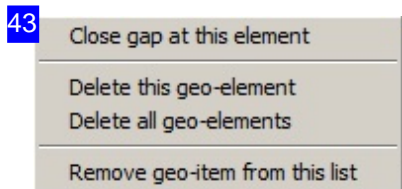
Line-type 0 signifies 'cut', while line-types 2 and 3 are preset for 'marking'. Other types are post-processor dependant, whereby types 2 and 3 must also be activated from the post-processor.



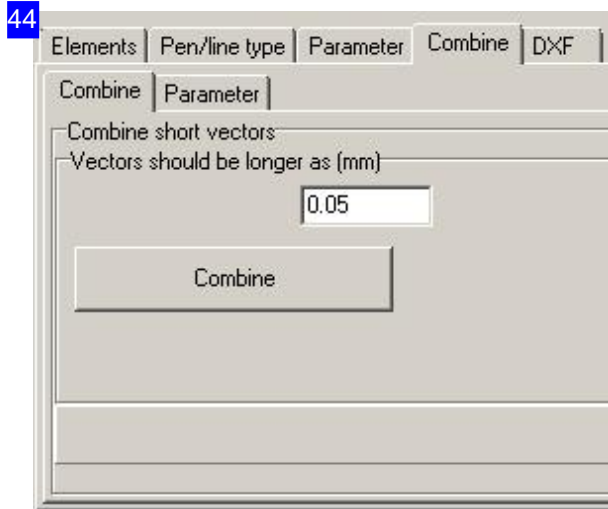
This frame allows you to seek the smallest of elements (faulty elements), or close gaps up to the given size.



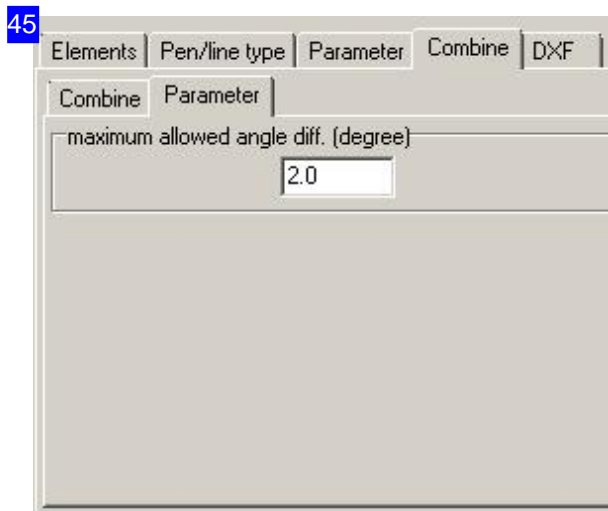
When searching specifically by error category using the buttons in panel [5], a list of the elements found under each category will be displayed. Clicking on an entry in the list switches back to the Vectors display. Right-click to access a context menu for repairing errors.



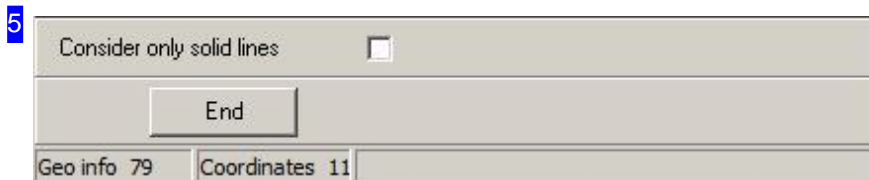
This menu allows you to directly access commands for repairing errors.



Contour groups which consist of multiple short vectors can be joined to one vector from this tab. Enter the limiting value for the maximum length of vector which should be joined. If you wish to join vectors together according to angle difference, switch to the 'Parameter' tab.

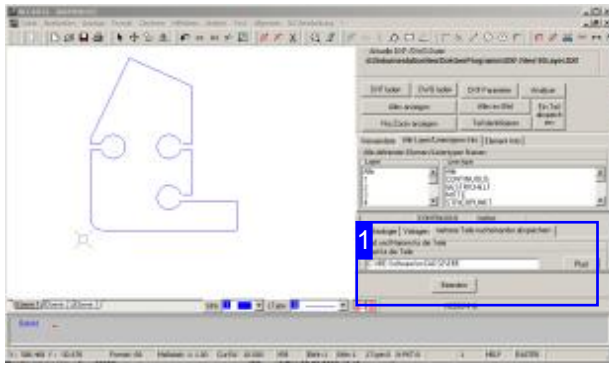


This frame allows you to set an angle difference limit for joining vectors. Enter the limiting value for the maximum angle difference which should be joined. If you wish to join vectors together according to vector length, switch to the previous tab.



Click on 'Exit' to return to the program.

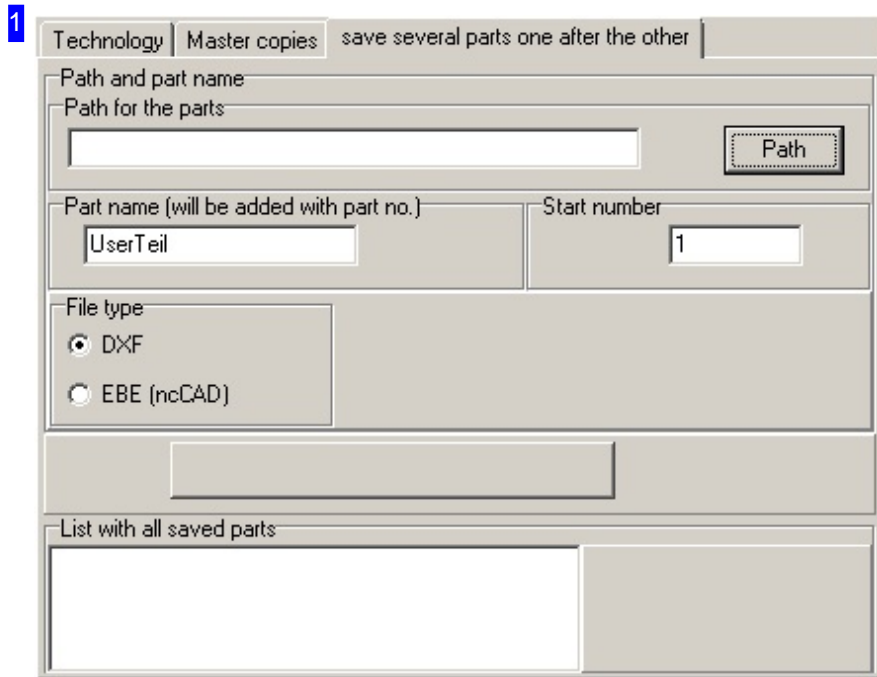
Export Drawings/ Layers



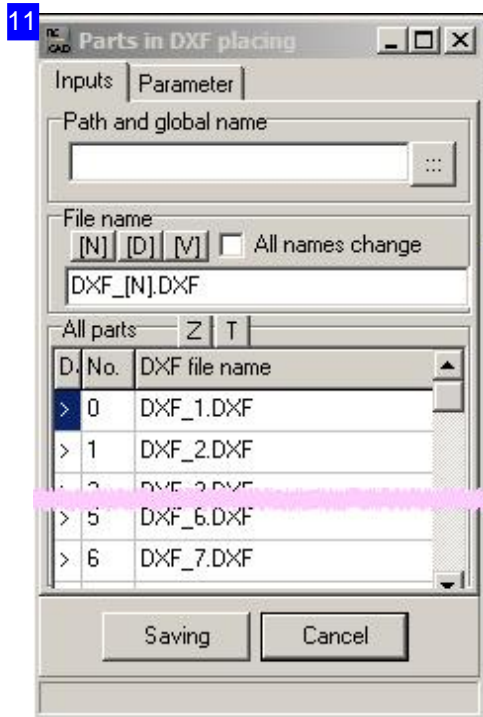
The 'File' menu allows you to export drawings and layers as DXF or EBE files.

- o Open a file manager to export as DXF files. Select the save path at the bottom-right and enter the filename bottom-left.
- o Select 'Multiple DXF export' to open a dialog to set the naming conditions for individual exported files.
- o To save a layer switch to the right hand pane [1].

Drawings and layers created using *ncCAD32* can be saved as complete drawings or individual layers. You can save as DXF or EBE files.



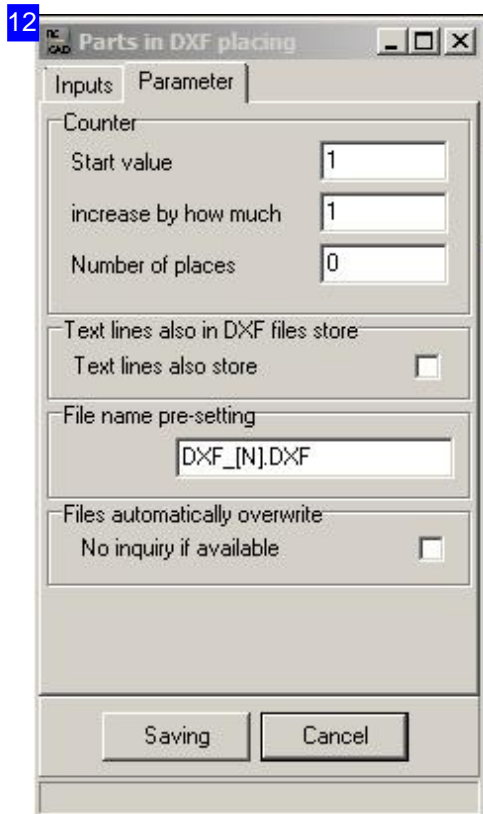
This pane allows you to save individual contours to various files. Enter the part name for all parts, and the index number for the first part. Select the save path and click the button to the left to confirm.



Multiple parts from one drawing can be saved to different DXF files. Enter the part name for all parts, and the index number for the first part. Select the save path and click to save.

The filename is created dynamically. The buttons in the 'file name' pane allow you to include the part name, index number and date. Index number settings can be made in the 'Parameter' tab.

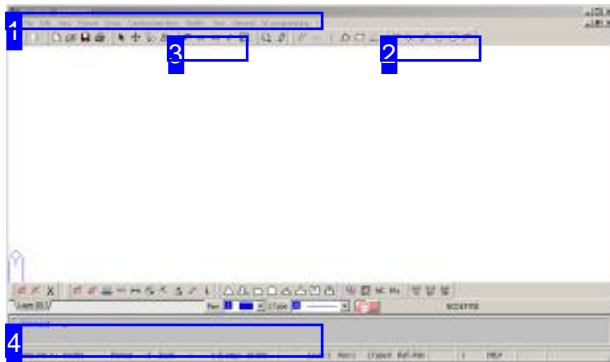
In the bottom pane you can set whether all parts should be output or only the selected part.



This tab is for making settings regarding the sequential naming of files. The 'Filename setting' pane allows you to enter a template which is expanded upon using your entries in the other sections to make the complete file name.

The Draw Functions.

Various Ways to Draw Elements.



Elements can be drawn in different ways. You can select the function via the menu, symbol bar, or command line.

The descriptions in the following pages will follow a tabular format.

Input via the command line is always shown as such, with the given example dimensions as an example. The code always stands for the corresponding function and is listed as command code or numerical code.

A drawing consists of various elements which, when combined, give a workpiece. When drawing, all elements are related to the origin (0 or 1).

Note: If you read 'click' in the following descriptions then, unless specifically stated otherwise, this means 'Left Click'.

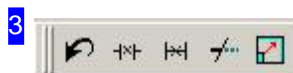
The cursor can be moved with the mouse, via the command line, or with the arrow keys.



The options for entry via the menu panel are given in the 'menu panel' chapter.



The most regularly used commands are easily accessible via the corresponding icon in the Symbol bar.

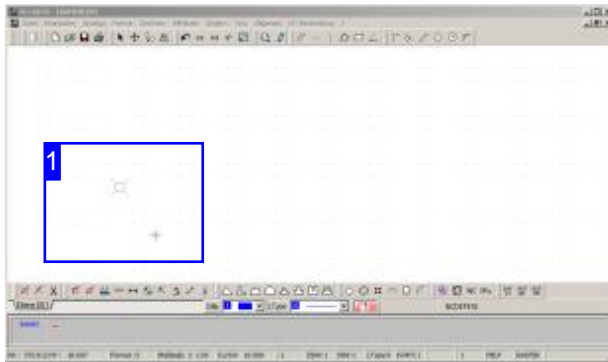


There is also a quick access route for drawing construction lines; click the corresponding icon in the Symbol panel.



You can make direct, exact inputs about an element's dimensions in the Command Line . There are examples listed with the required codes in the individual descriptions of each drawing command. You can gain access to the direct value entry form by pressing <F2>.

Set the Zero-point as Reference Point.



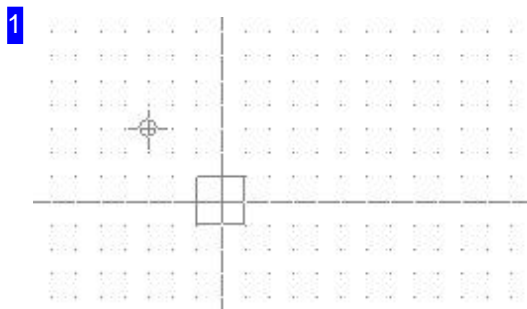
Function:
Set a zero-point, or select a zero-point as a future reference point.

Define the Zero-point.
Command: NULLPD
Code: 00
Parameter: Zero-point number
Example: 00 1

Menu: yes
Symbol: no

-
Select Zero-point.
Command: NULLPW
Code: 01
Parameter: Zero-point number
Example: 01 1

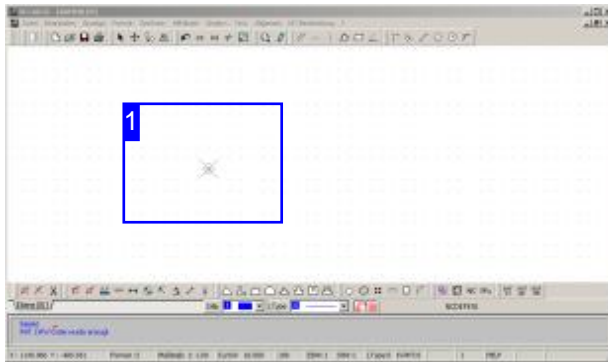
Menu: yes
Symbol: no



Set or select a zero-point via:

1. Command Line
 1. Enter the command or code and parameter into the command line.
2. PopUp menu
 1. Right-click in the drawing window.
 2. Click the bottom menu point.

Setting the Start-Point for a Draw Function.



Function:
Positions the cursor at a start-point for further drawing commands.

Start-point relative to the cursor position.

Command: ST

Code: 1

Parameter: x, y

Example: 1 100 50

-

Start point relative to the current origin.

Command: STN

Code: 11

Parameter: x, y

Example: 11 100 50

-

Start point Radius, Angle

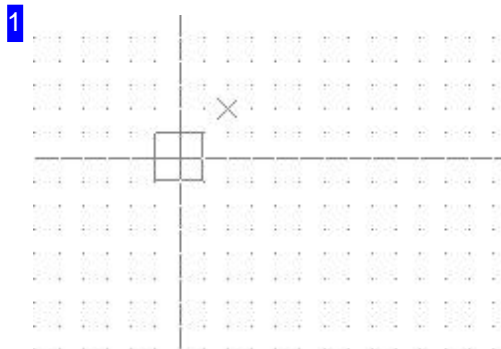
Command: STP

Code: 12

Parameter: Angle, Radius

Example: 12 30 100

-

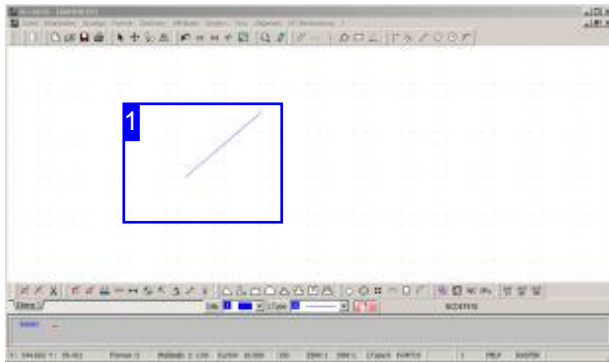


Set the start-point via:

1. Command Line

1. Enter the command or code and parameters in the command line.

Drawing a line



Function: Draws a line from the given start point to the given end point.

The VE command allows a line to be drawn from the current cursor position to the coordinates entered as parameters. By continually entering coordinates you can create unbroken outlines.

Command: VEKTOR

Code: 150

Parameter: XS,YS; XE,YE- Input is made sequentially.

Example: 150; Cursor to start point; <CR> Cursor to end point; <CR>.

Menu: yes

Symbol: yes

-

Command: VE

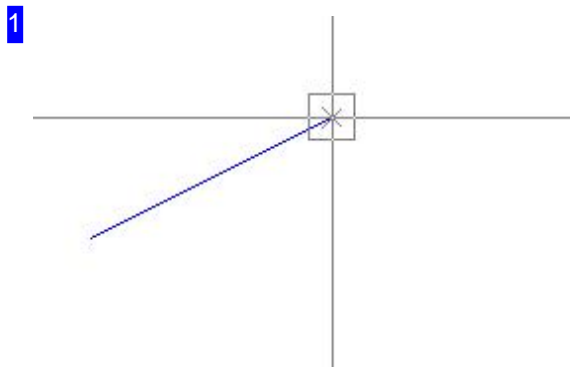
Code: 2

Parameter: XE,YE

Example: 2 100 50

-

Abort the function with <Esc>!



Drawing a line via:

1. Menu

1. Select the function 'Draw > Line'.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

2. Symbol

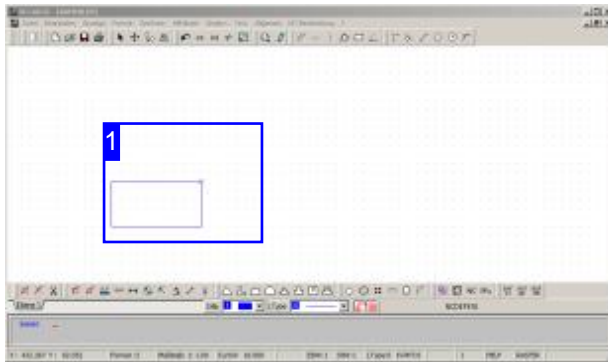
1. Click the 'Line' icon.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

3. Command line

1. Enter the command or its code in the command line.
2. Move to the start point and press <CR> or LC.
3. Move to the end point and press <CR> or LC.

You can create whole unbroken groups of lines by continually adding lines from the last cursor position using Code 2. Abort the function with <Esc>.

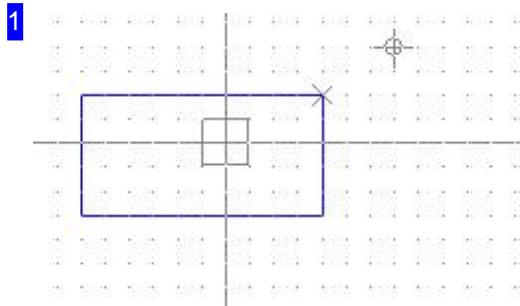
Drawing a Rectangle.



Command: RECHT
Code: 39
Parameter: x, y, Angle
Example: 39 100 50 0

Menu: yes
Symbol: yes

Function: Draws a rectangle from the current cursor position to the coordinates given as a parameter at the given angle.



Zeichnen eines Rechtecks über: Draw a rectangle via:

1. Menu

1. Select the 'Draw > Rectangle' function.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

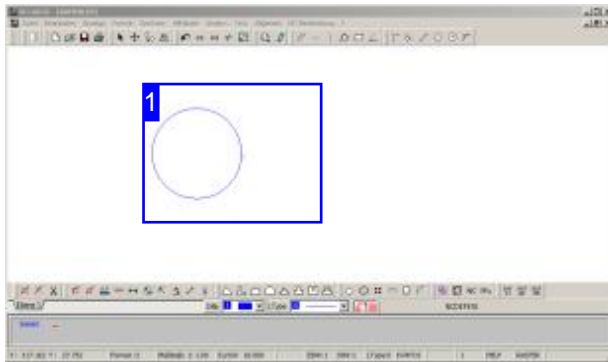
2. Symbol

1. Click the 'Rectangle' symbol.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

3. Command line

1. Place the cursor at the start point.
2. Enter the command or code in the command line.

Drawing a Circle by its Radius.

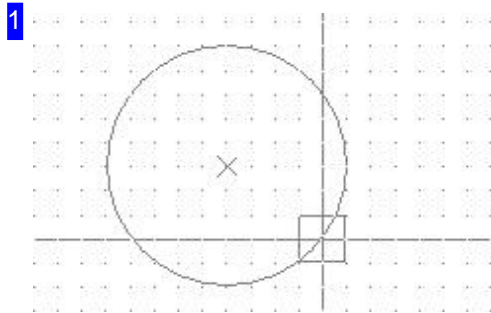


Command: KR
Code: 3
Parameter: Radius
Example: 3 50

Menu: yes
Symbol: yes

-
Abort function with <Esc>!

Function: Draws a circle from the current cursor position with the radius entered as a parameter.



Draw a circle by its radius via:

1. Menu

1. Select the function 'Draw > Circle > Center/Radius'.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

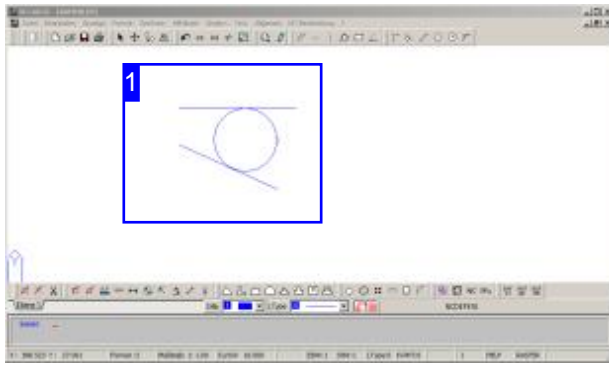
2. Symbol

1. Click the 'Circle/Radius' symbol.
2. Move the cursor to the start point, press LC.
3. Move the cursor to the end point, press LC.

3. Command line

1. Place the cursor at the start point.
2. Enter the command or code and the radius in the command line.

Drawing a Circle Against Two Elements.



Command: HLKR

Code: 227

Parameter: GE1; GE2; Radius - Inputs are made sequentiall

Beispiel: 227 - Enter parameters as prompted in the command line

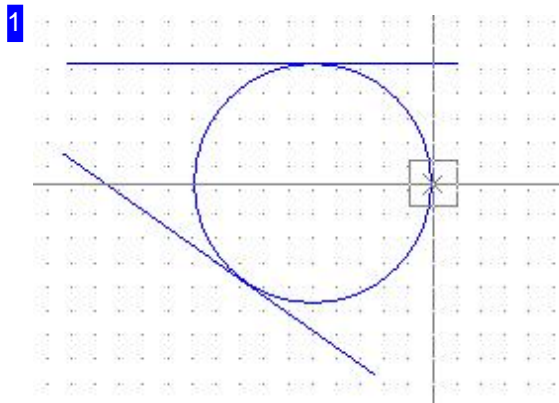
Menu: yes

Symbol: yes

-

Abort the function with <Esc>!

Function: Draws a circle with the given radius tangentially touching two identified elements (GE).



Draw a circle against 2 elements via:

1. Menü

1. Select the function 'Draw > Circle > Circle on 2 elements'.
2. Move the cursor to the first element and press LC.
3. Move the cursor to the second element and press LC.
4. Enter the radius in the command line.

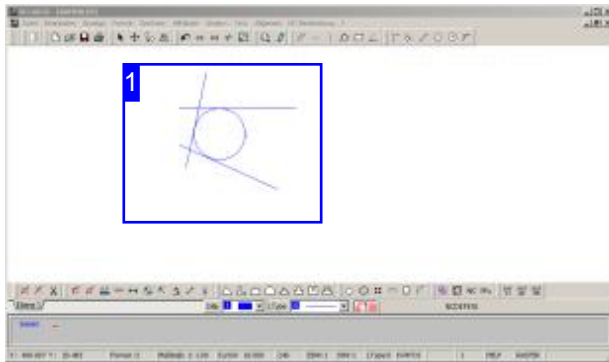
2. Symbol

1. Click the symbol for 'Circle on 2 elements'.
2. Move the cursor to the first element and press LC.
3. Move the cursor to the second element and press LC.
4. Enter the radius in the command line.

3. Befehlszeile

1. Enter the command or code in the command line.
2. Move the cursor to the first element and press <CR>.
3. Move the cursor to the second element and press <CR>.
4. Enter the radius in the command line.

Drawing a Circle Against Three Elements.



Command: BO3GO

Code: 246

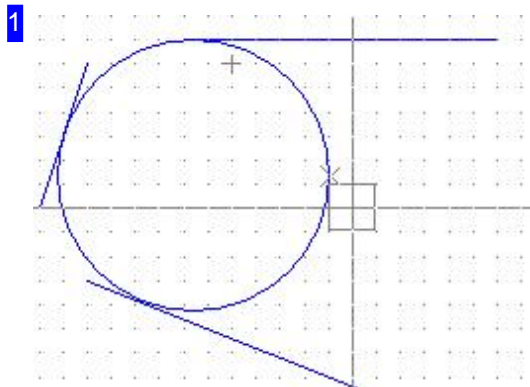
Parameter: GE1; GE2; GE3 - Inputs are made sequentially

Example: 246 - Enter parameters as prompted in the command line

Menu: yes

Symbol: no

Function: Draws a circle tangentially touching three identified elements (GE)(inscribed circle).



Draw a circle against 3 elements via:

1. Menu

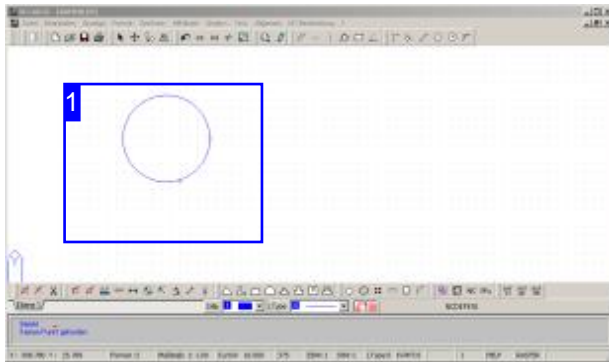
1. Select the function 'Draw > Circle > Circle on 3 elements'.
2. Move the cursor to the first element and press LC.
3. Move the cursor to the second element and press LC.
4. Move the cursor to the third element and press LC.

The circle will be calculated automatically and placed against all three elements.

2. Command line

1. Enter the command or code in the command line.
2. Move the cursor to the first element and press <CR>.
3. Move the cursor to the second element and press <CR>.
4. Move the cursor to the third element and press <CR>.

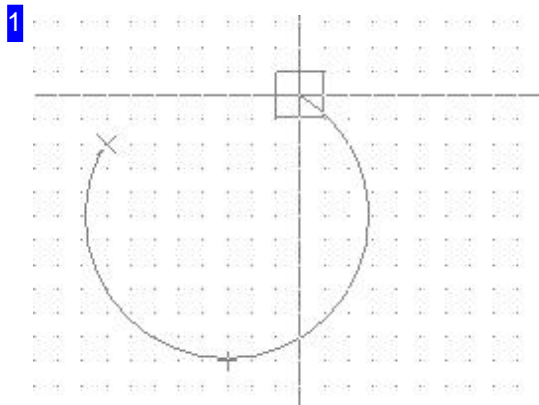
Draw a Circle Through Three Points (circumcircle).



Command: no
 Code: 375
 Parameters: SP; BP; EP - Inputs are made sequentially
 Example: 246 - Enter parameters as prompted in the command line

menu: yes
 Symbol: no

Function: Draws a circle through 3 points, perimeter to lie on the the start point (SP), second point (Arc point BP), end point (EP).



Draw a circle through three points via:

1. Menu

1. Select the function 'Draw > Circle > Circle through 3 points'.
2. Move the cursor to the start point and press LC.
3. Move the cursor to the second point and press LC.
4. Move the cursor to the end point and press LC.

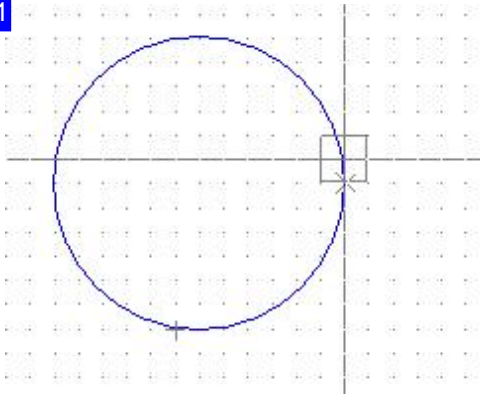
The Circle will be calculated automatically and located on the three points.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the start point and press <CR>.
3. Move the cursor to the second point and press <CR>.
4. Move the cursor to the end point and press <CR>.

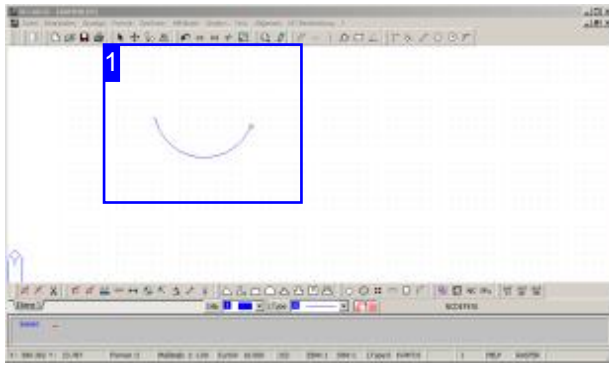
The Circle will be calculated automatically and located on the three points.

11



A typical circle with the perimeter through three points.

Draw An Arc Using Arc-Points and Mid-Point.



Command: BOGEN1

Code: 152

Parameter: XS,YS; XE,YE; XM,YM - Inputs are made sequentially.

Example: 152 - Enter parameters as prompted in the command line.

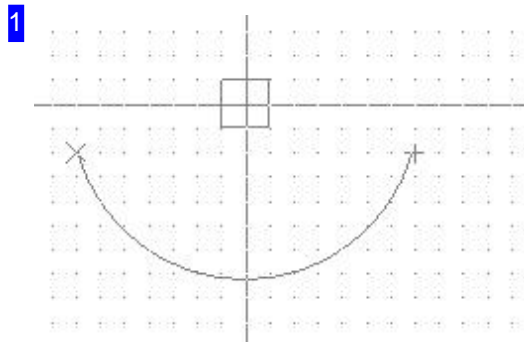
Menu: yes

Symbol: no

-

Abort function with <Esc>!

Function: Draws an arc with (S)tart-, (E)nd- and (M)id-points.



Draw an arc from its start-, end- and mid-points via:

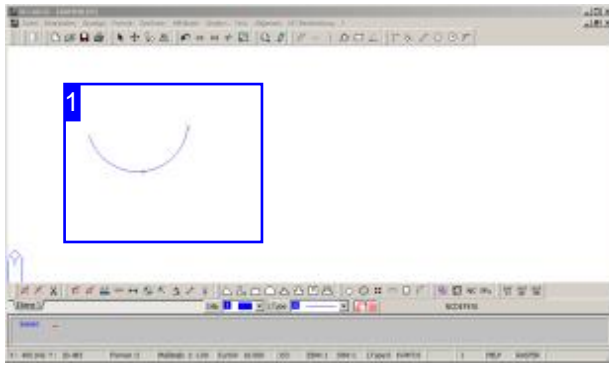
1. Menu

1. Select the function 'Draw > Arc > Startpoint, Endpoint, Midpoint'
2. Move the cursor to the start point and press LC.
3. Move the cursor to the end point and press LC.
4. Move the cursor to the mid point and press LC.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the start point and press <CR>.
3. Move the cursor to the end point and press <CR>.
4. Move the cursor to the mid point and press <CR>.

Draw an Arc Using Three Arc-Points.



Command: BOGEN2

Code: 153

Parameter: XS,YS; XB,YB; XE,YE - Inputs are made sequentially.

Example: 153 - Enter parameters as prompted in the command line.

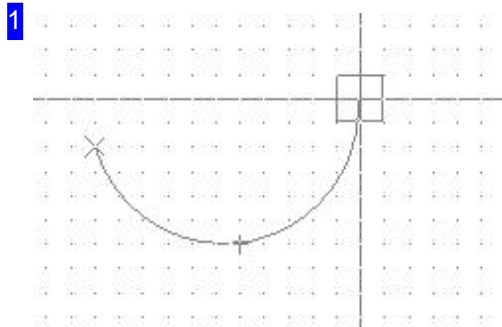
menu: yes

Symbol: no

-

Abort the function with <Esc>!

Function: Draws an arc from its (S)tart-, (B)Arc and (E)ndpoint.



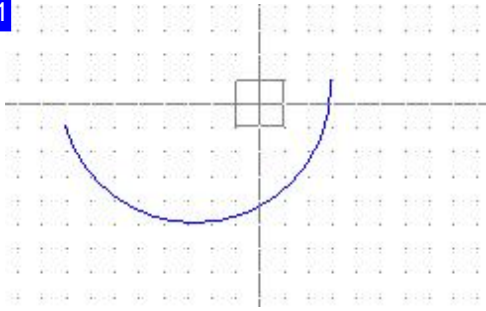
1. Menu

1. Select the function 'Draw > Arc > Start point, Point on arc, Endpoint'.
2. Move the cursor to the start point and click.
3. Move the cursor to the second point and click.
4. Move the cursor to the end point and click.

2. Command Line

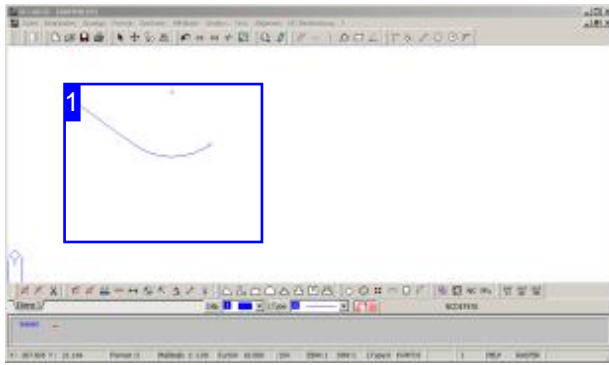
1. Enter the command or code in the command line.
 2. Move the cursor to the start point and press <CR>.
 3. Move the cursor to the second point and press <CR>.
 4. Move the cursor to the end point and press <CR>.
- The Arc will be calculated automatically and laid across the three points.

11



The completed arc through three points. The function allows you to chain any number of arcs together. Use <Esc> to abort the function.

Draw an Arc on a Tangent from an Element.



Command: BOGEN3

Code: 154

Parameter: XE,YE - Inputs are made sequentially.

Example: 154 - Enter parameters as prompted in the command line.

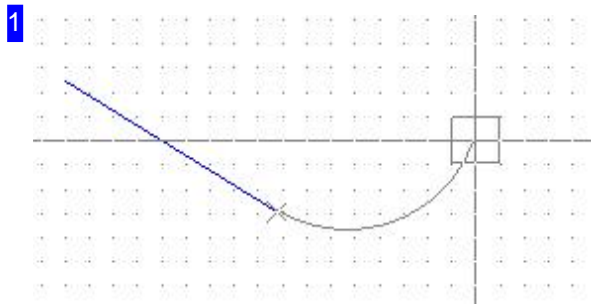
Menu: yes

Symbol: no

-

Abort the function with <Esc>!

Function: Draws an arc from the last element to the given (E)ndpoint.



Draw an arc tangentially from an element via:

1. Menu

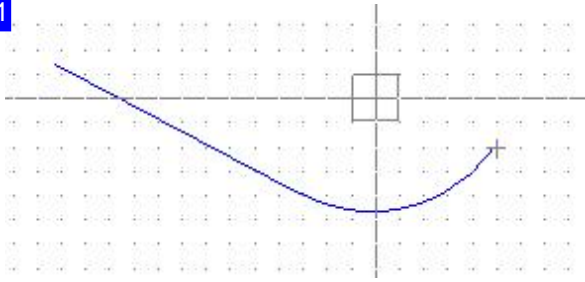
1. Select the function 'Draw > Arc > Tangential from an existing element'.
2. Move the cursor to the end point and click.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the end point and click <Enter>.

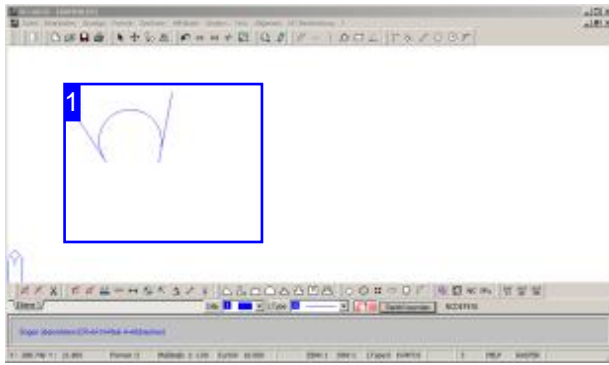
The Arc will be calculated automatically and tacked onto the existing element.

11



The completed arc from an existing element to a given (E)ndpoint. The function allows you to chain any number of arcs together. Use <Esc> to abort the function.

Draw an Arc Tangential to Two Elements.



Function: Draws an arc tangential to two drawing elements (1, 2).

Command: BO2GO

Code: 66

Parameter: GE1; GE2; Radius; <CR>, <N>, <A> - the inputs are made sequentially.

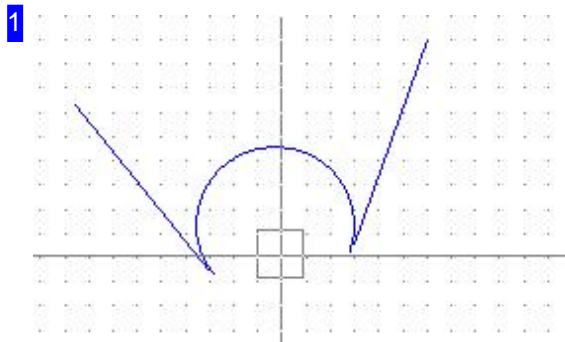
Example: 66 - Enter parameters as prompted in the command line.

Menu: yes

Symbol: no

-

Abort the function with <Esc>!



Draw an arc tangential to two elements via:

1. Menu

1. Select the function 'Draw > Arcs > Tangential to two elements'.
2. Move the cursor to the first element and press LC.
3. Move the cursor to the second element and press LC. - Cursor will go to the mid-point.
4. Enter the radius in the command line or confirm the suggested mid-point.
5. Accept the arc with <CR> or reject the arc with <N> - the opposite arc will be offered.
6. Accept the arc with <CR> or abort the function with <A>.

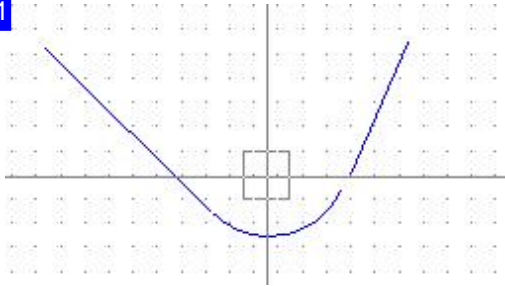
2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the first element and press <CR>.
3. Move the cursor to the second element and press <CR>.

4. Enter the radius in the command line.
5. Reply to the prompts with <CR>, <N> or <A> (as above).

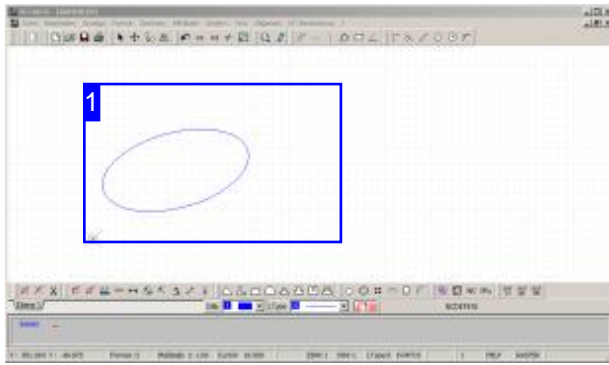
The opposite arc laid tangentially on the existing elements.

11



The opposite arc laid tangentially on two elements. Use <Esc> to abort the function.

Draw a Full Ellipse with all Inputs.



Function: Draws a full ellipse.

Note: Dependant on your machine, ellipse may have to be cut as Lines (Vectors) or Arcs. For this reason *ncCAD32* offers both construction options.

Command: ELLIPSV

Code: 165

Parameter: Large Radius; Small Radius; Angle of Inclination for Large Axis; - Inputs are made sequentially.

Example: 66 50,50 100 50 15.

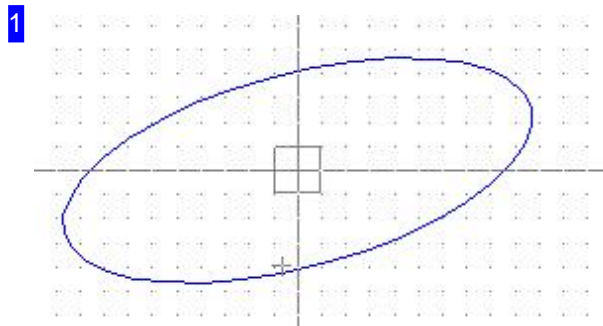
Menu: no

Symbol: no

-

Once an ellipse has been defined, it can be copied to multiple mid-points simply by clicking.

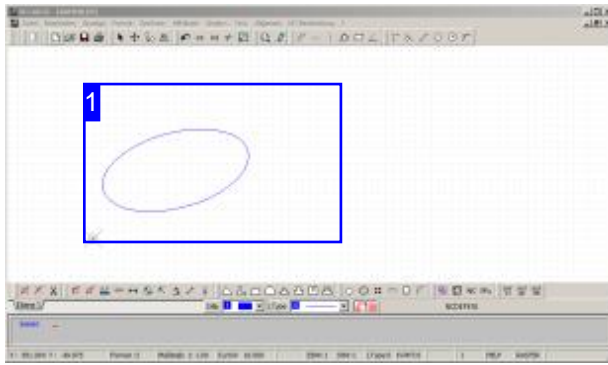
Abort the function with <Esc>!



Draw an ellipse from the command line:

1. Enter the command or code in the command line.
2. Move the cursor to the mid-point and press <CR>.
3. Enter the size of the large axis.
4. Enter the size of the small axis.
5. Enter the angle of inclination for the large axis.

Drawing Full-Ellipses with Lines.



Function: Draws a full ellipse with lines.

Command: VELLIP

Code: no

Parameter: Mid-point; Start-point; Dialog input ; -
Inputs are made sequentially.

Example: VELLIP; Start-point; Dialog.

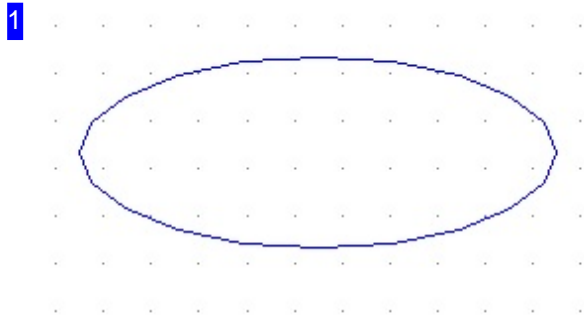
Menu: yes

Symbol: no

-

Once an ellipse has been defined, it can be copied to multiple mid-points simply by clicking.

Abort the function with <Esc>!



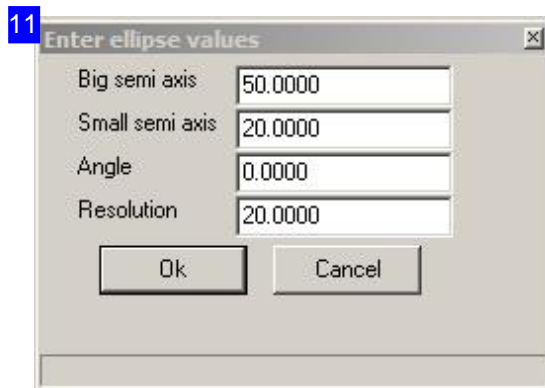
Draw an ellipse via:

1. Menu

1. Select the function 'Draw > Ellipse > Ellipse from vectors'.
2. Move the cursor to the mid-point and press LC.
3. Enter the ellipse's values in the dialog.
4. Click 'Apply'.

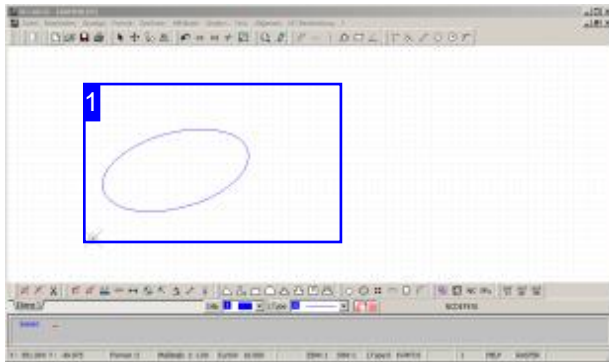
2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the mid-point and press <CR>.
3. Enter the ellipse's values in the dialog.
4. Click 'Apply'.



This dialog is for defining the ellipse. Click the 'apply' button to draw the ellipse.

Drawing a Full Ellipse from Arcs.



Function: Draws a full ellipse with arcs.

Command: BELLIP

Code: no

Parameter: Mid-point; Start-point; Dialog input ; -
Inputs are made sequentially.

Example: VELLIP; Start-point; Dialog.

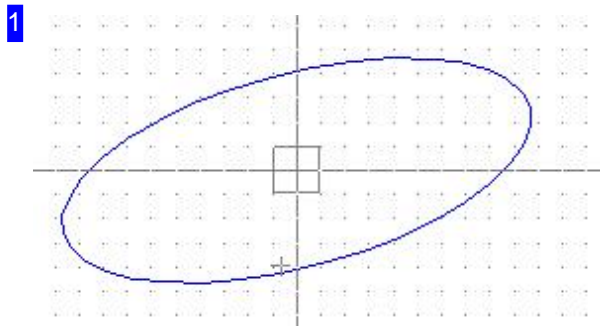
Menu: yes

Symbol: no

-

Once an ellipse has been defined, it can be copied to multiple mid-points simply by clicking.

Abort the function with <Esc>!



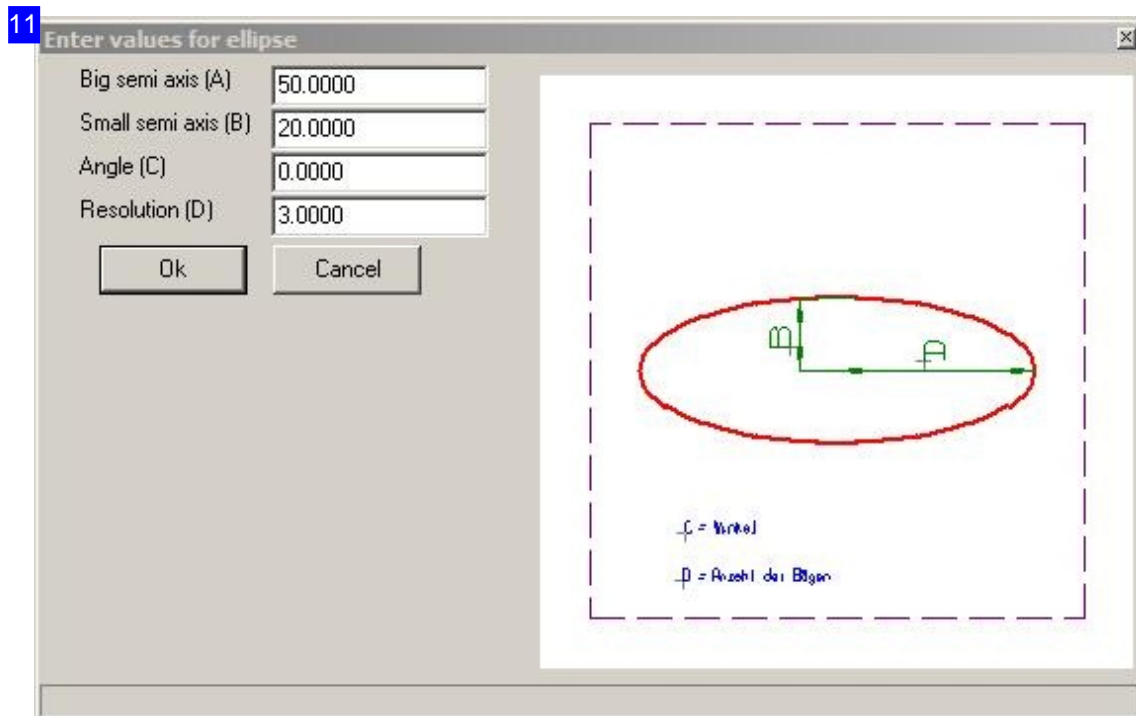
Draw an ellipse via:

1. menu

1. Select the function 'Draw > Ellipse > Ellipse from arcs.
2. Move the cursor to the mid-point and press LC.
3. Enter the ellipse's values in the dialog.
4. Click 'Apply'.

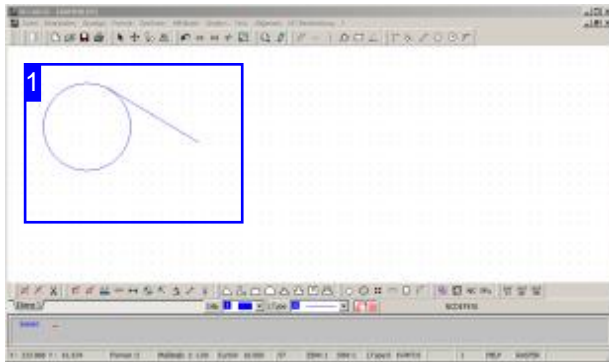
2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the mid-point and press <CR>.
3. Enter the ellipse's values in the dialog.
4. Click 'Apply'.



This dialog is for defining the ellipse. Click the 'apply' button to draw the ellipse.

Draws Tangents to an Arc or Circle.



Command: TANPK

Code: 57 1

Parameter: XP,YP; Element - Inputs are made sequentially.

Example: 57 1; Cursor to Startpoint; <CR> Cursor to Element; <CR>.

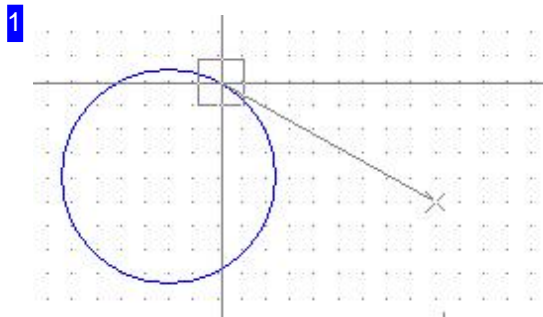
Menu: yes

Symbol: no

-

Abort the function with <Esc>!

Function: Draws a tangent from a (P)oint on a circle or an arc.



Draw a tangent via:

1. Menu

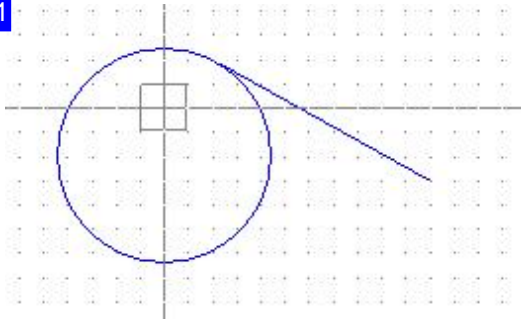
1. Select the function 'Draw > Tangents > Point/Circle'.
2. Move the cursor to the start point and press LC.
3. Move the cursor to the circle/arc and press LC.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the start point and press LC or <CR>.
3. Move the cursor to the circle/arc and press LC or <CR>.

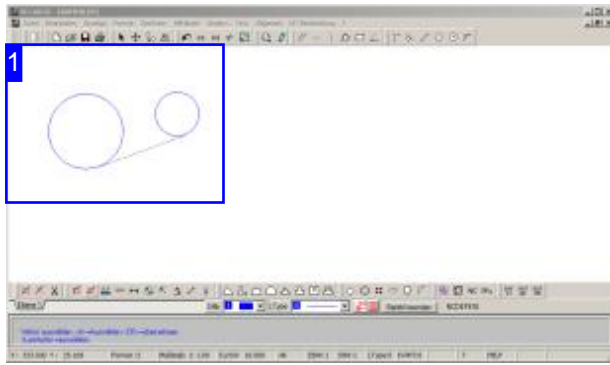
The Tangent will be laid against the circle.

11



The tangent will be laid from the start point to the circle/arc.

Draw Tangents Between Two Circles.



Command: TAN2K

Code: 46

Parameter: E1; E2 - Inputs are made sequentially.

Example: 46; Cursor to first circle; <CR> Cursor to second circle; <CR>.

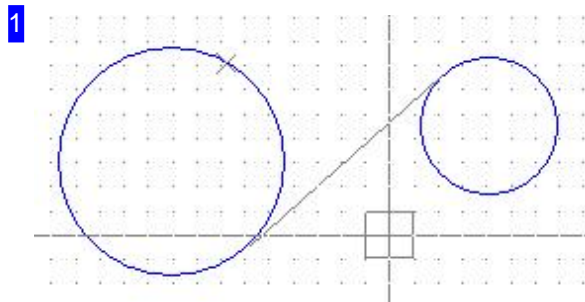
Menu: yes

Symbol: no

-

Abort the function with <Esc>!

Function: Draws a tangent from one circle (E1) to another circle (E2). A selection allows you to choose between four different tangents.



Draw a tangente via:

1. Menu

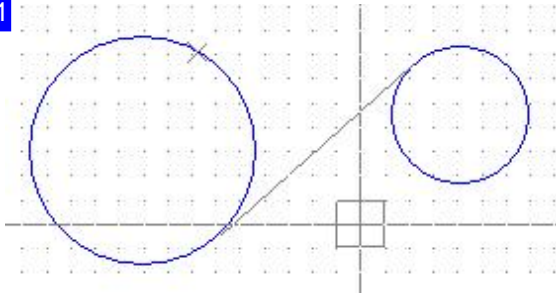
1. Select the function 'Draw > Tangents > 2 Circles'.
2. Move the cursor to the first circle and press LC.
3. Move the cursor to the second circle and press LC.
4. Select tangent with <A> (or spacebar) - press LC.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the first circle and press <CR>.
3. Move the cursor to the second circle and press <CR>.
4. Select tangent with <A> - press <CR>.

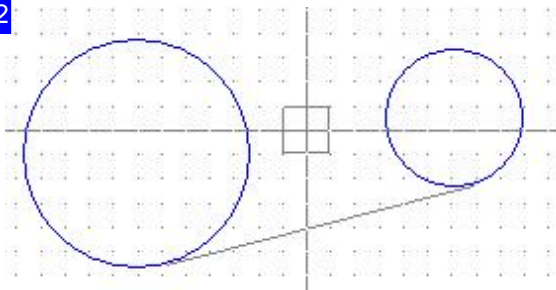
Use <A> or the spacebar to toggle between the four available options for setting the tangent on the circles.

11



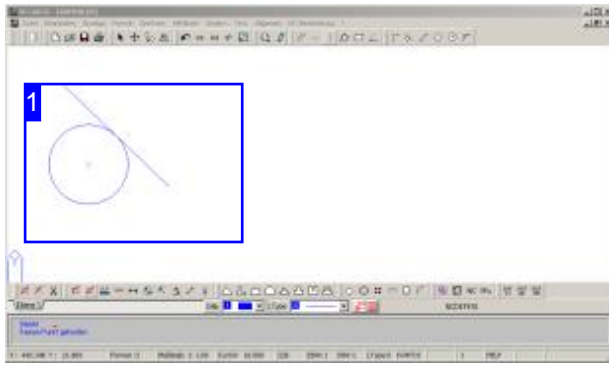
The tangent can be laid inside or outside. Press <CR> or click the mouse and the Tangent will be drawn.

12



The tangent can be laid inside or outside. This is the outside variant. You can switch between the variants with <A>.

Drawing Tangential Construction Lines.



Function: Draws a construction line tangential to a circle or an arc.

Command: TANPK

Code: 57 2

Parameter: Element; Angle - Inputs are made sequentially.

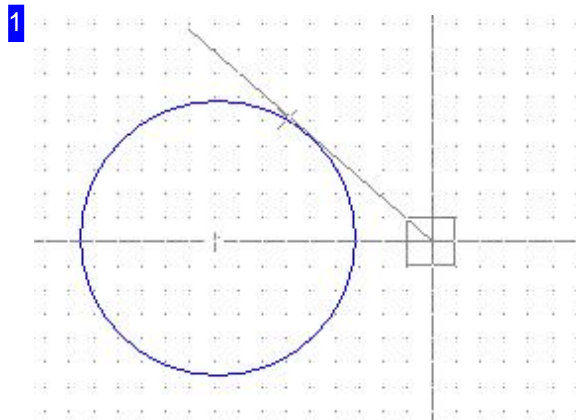
Example: 57 2; Cursor to element; <CR>; 60; <CR>.

Menu: yes

Symbol: yes

-

Abort the function with <Esc>!



Draw a tangential construction line via:

1. Menu

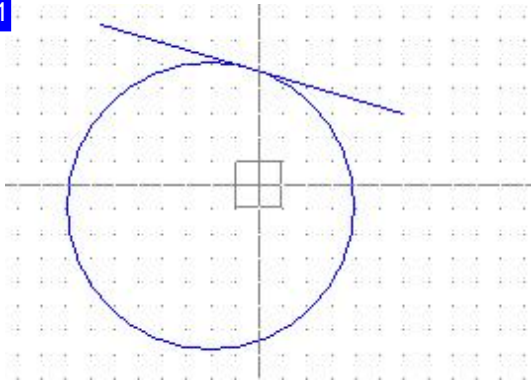
1. Select the function 'Draw > Tangents > Point/Circle'.
2. Move the cursor to the circle/arc and press LC
3. Use the mouse to move the tangent to the correct position (Rotational); confirm with a click.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the circle/arc and press <CR>.
3. Enter the tangent's angle and press <CR>.

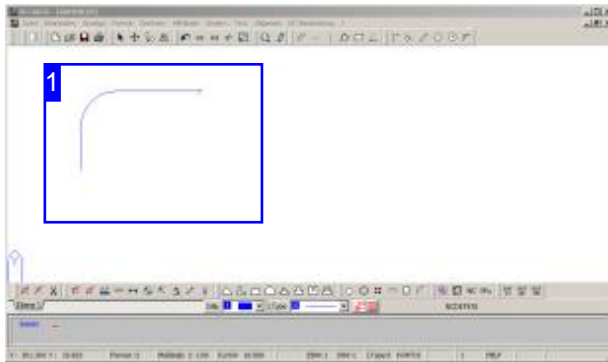
The Tangent will be laid on the circle.

11



The tangential construction line will be laid on the circle/arc at the given angle.

Filleting the Corner of Two Elements.



Command: VER

Code: 76

Parameter: Corner point; Radius - Inputs are made sequentially.

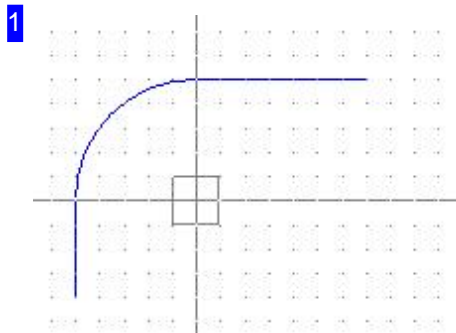
Example: 76; Cursor to corner point; <CR>; 60; <CR>.

Menu: yes

Symbol: yes

Function: Draws a fillet arc on the corner of two elements.

A fillet can be undone via the menu.



Draw a fillet via:

1. Menu

1. Select the function 'Draw > Fillet'.
2. Move the cursor to the corner and press LC.
3. Enter the radius and confirm with a mouse click.

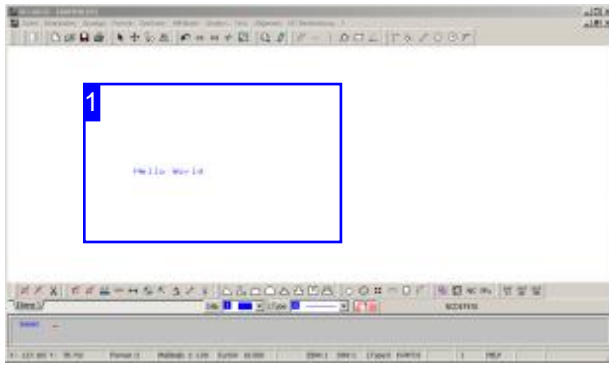
2. Symbol

1. Select 'fillet'.
2. Move the cursor to the corner and press LC
3. Enter the radius and confirm with a mouse click.

3. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the corner and press <CR>.
3. Enter the radius and confirm with <CR>.

Entering Simple Text.



Function: Draws simple text in a drawing.

- *ncCAD32* also offers the option to use 'True Type' texts to create layout texts.

Note: texts should always be written with an external contour. Internal elements should be held with cutting bridges.



Draw a text via:

1. Menu

1. Select the function 'Text > ncCAD Text > input'.
2. Move the cursor to the start point and LC.
3. Enter the text and press <CR>.

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the start point and press <CR>.
3. Enter the text and press <CR>.

Command: TEXT

Code: 75

Parameter: Start point; Text - Inputs are made sequentially.

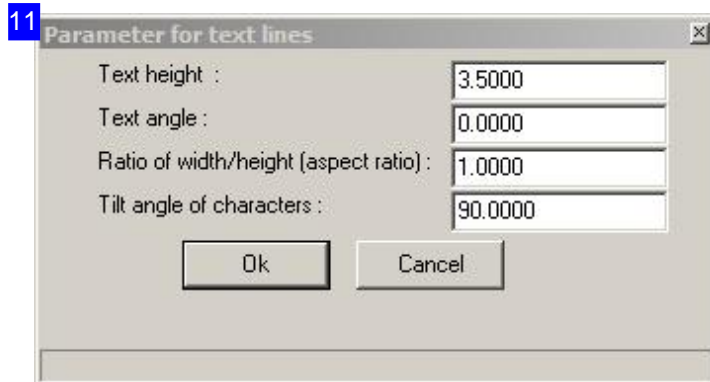
Example: 75; Start point; Example text.

Menu: yes

Symbol: no

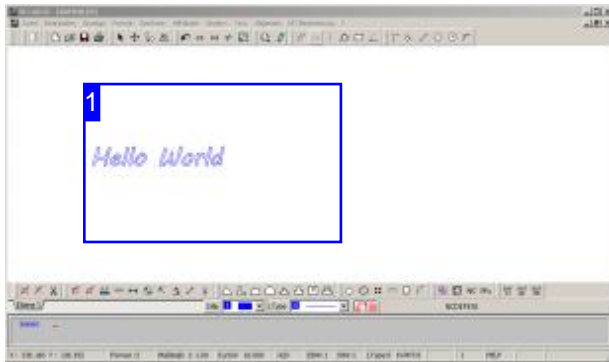
-

Texts are drawn based upon pre-defined parameters. These parameters are accessible via a Dialog, which you can reach through the menu item 'Text > ncCAD Text > Parameter ..'.



This dialog allows the presets for entering text to be changed. Call the dialog through the menu or the command line with the command (TEXTP), or code (75).

Entering True Type Texts.



Function: Draws layout texts in your drawing using true-type fonts.

Note: texts should always be written with an external contour. Internal elements should be held with cutting bridges.

Command: no
Code: 420
Parameter: Dialog; Start point - Inputs are made sequentially.
Example: 420; Dialog; 40,40.

Menu: yes
Symbol: no

-

Texts are entered into a dialog and drawn at the given location.



Draw a text via:

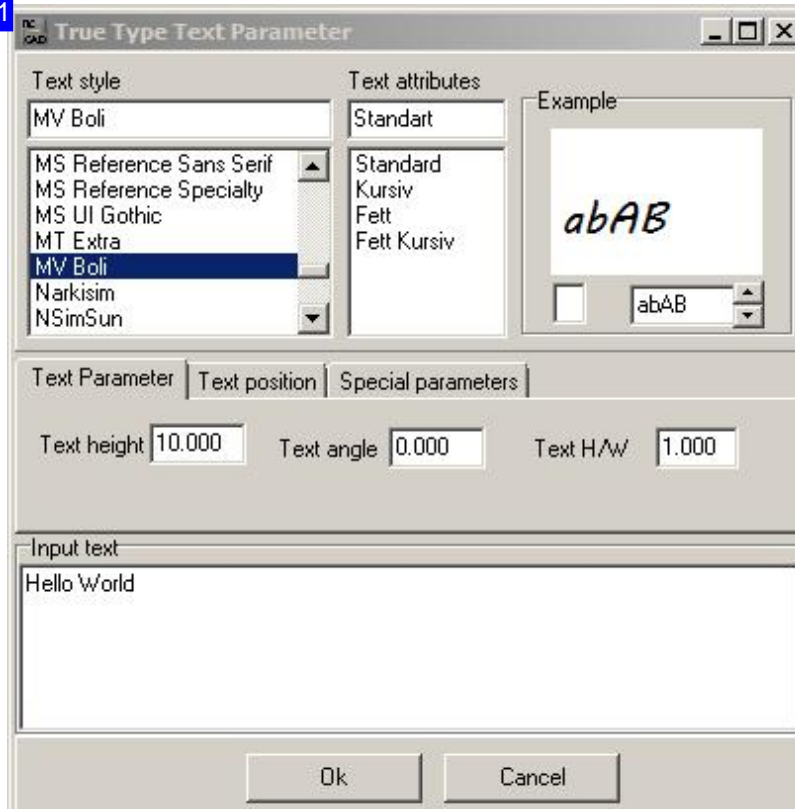
1. Menu

1. Select the function 'Text > TrueType Text > Text input'.
2. Fill out the dialog and click 'apply'.
3. Move the cursor to the start point and LC.

2. Command Line

1. Enter the command or code in the command line.
2. Fill out the dialog and click 'apply'.
3. Move the cursor to the start point and <CR>.

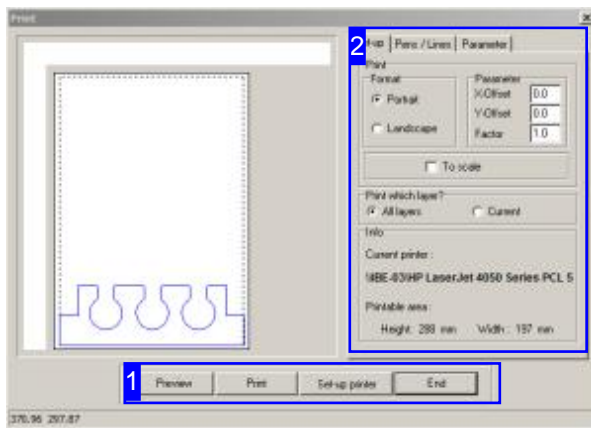
11



This dialog allows the presets for entering text to be changed. Call the dialog through the menu or the command line with the command (TEXTP), or code (75).

Printing

Printing the Drawing?



There is a dialogue box with options with various options for setting-up your printer when printing your drawing.

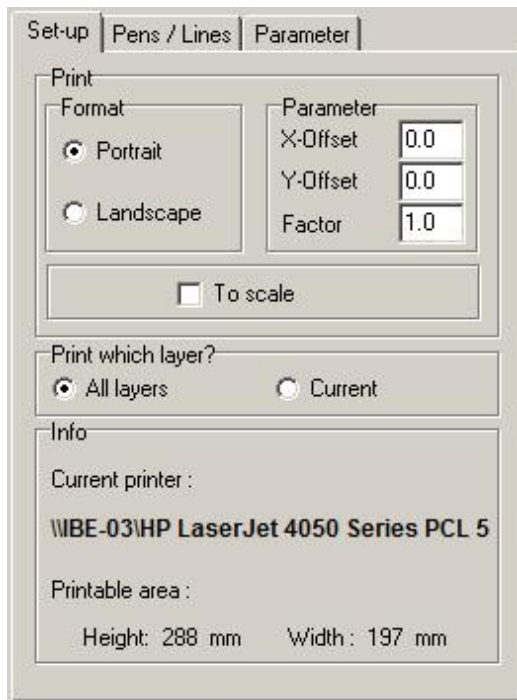
There is a preview of your drawing to the left. If there is no preview here, check the 'current' layer checkbox.

Make all settings for your printer and print your drawing out.



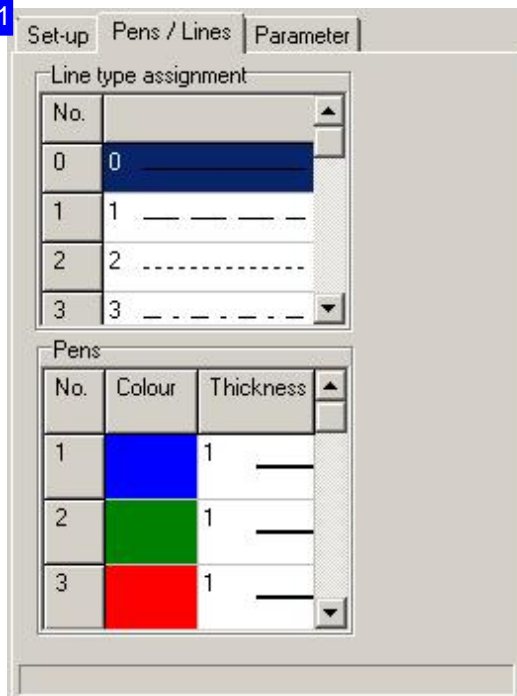
Use the buttons to access the printer setting for your system's printer. The 'Preview' button re-draws the drawing. Right-click to leave the dialogue box.

2



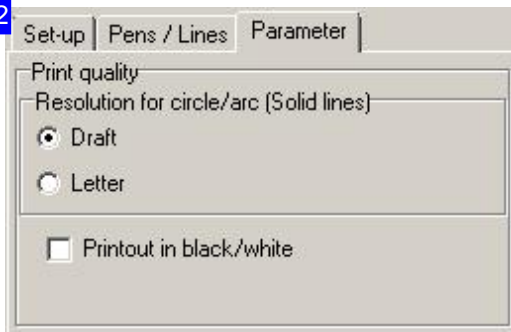
As well as settings for the print format, you can also select the Line types and colours and determine the Print quality.

21



Make settings about the line types and colours here. The print quality can be set in another frame.

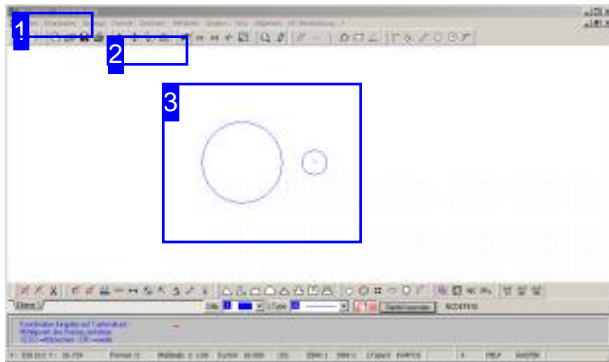
22



Make settings about the print quality here.

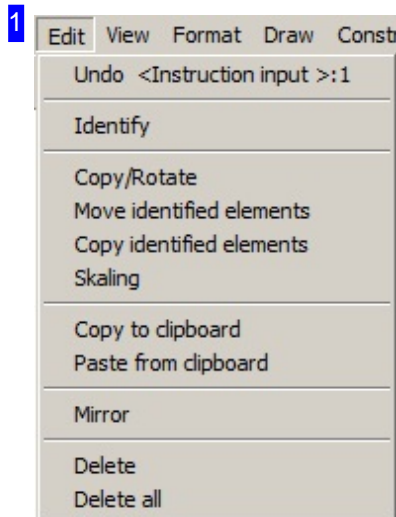
The Editing Functions.

Different Ways of Drawing Elements.



Elements can be edited in different ways. You can access the function via the menu, entering the code in the command line, or in part using the icon bar. Before you attempt to edit an elements, you must mark, or identify, one or more elements.

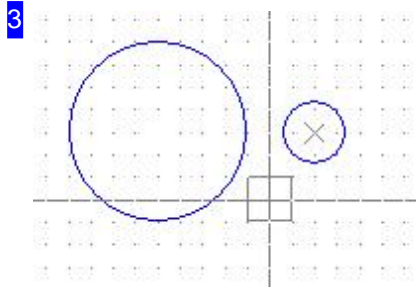
The editing functions can be accessed from the menu panel [1] and in part using the icon bar [2].



Use the menu to access an edit function. The relevant elements must be identified first.

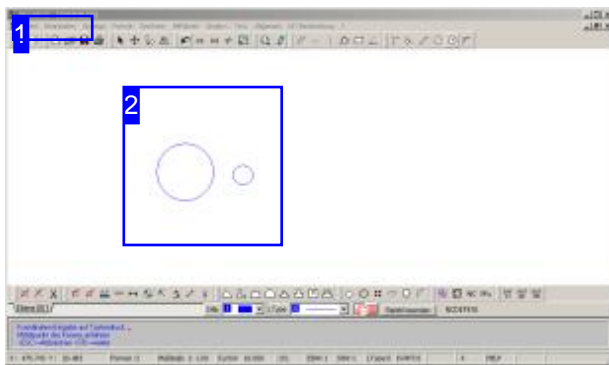


Some functions can be activated using the icon bar. The required elements must be selected as before.



These elements represent the foundation for describing the edit functions. The mode of operation shall be explained through them.

Copy/Turn Elements.



This function allows elements to be copied in rows, columns or a grid, or around a reference (center) point.

The element(s) to be copied must be identified before copying.

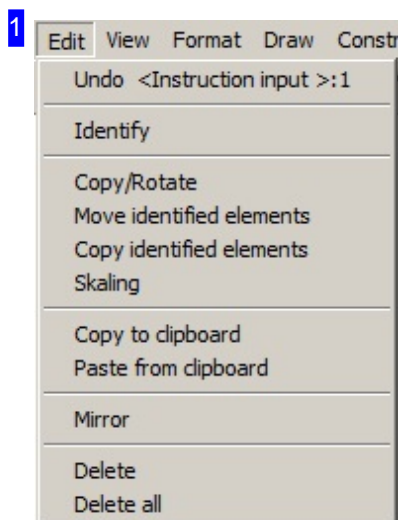
Copy in a row/column:

- o Select menu point [1] 'Copy/Turn'
- o LC in the drawing window; select 'Row/Column' in the selection dialog.
- o Select the 'Row' or 'Column' tab.
- o Enter the number of rows or columns and the offsets between the parts.
- o Click 'Apply'.

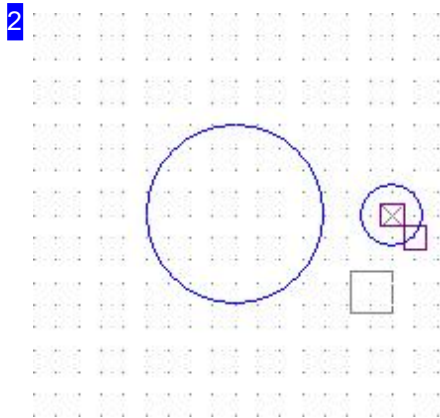
Copy around a center:

- o Select menu item [1] 'Copy/ Turn'.
- o LC on the center; select the 'Turn' tab in the selection dialog.
- o Select the 'Quantity/Diff' or 'Quantity/Angle' tab.
- o Enter the quantity of parts and either the difference angle or the total angle.
- o Click Apply.

The program also allows for freehand copying. This function is described in the 'Freehand copying' chapter.

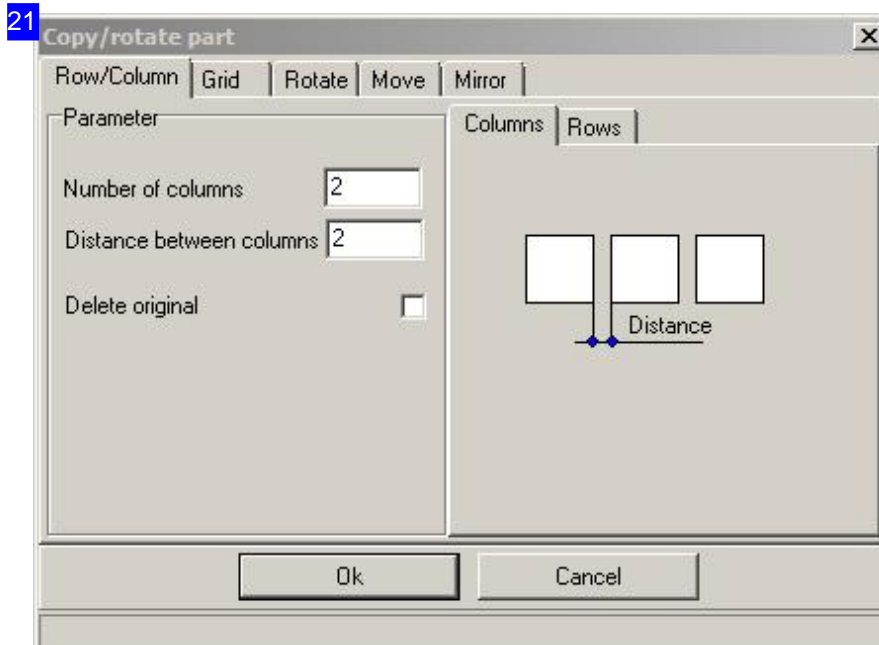


Select the menu item 'Copy/Turn'. The relevant elements must be identified first.

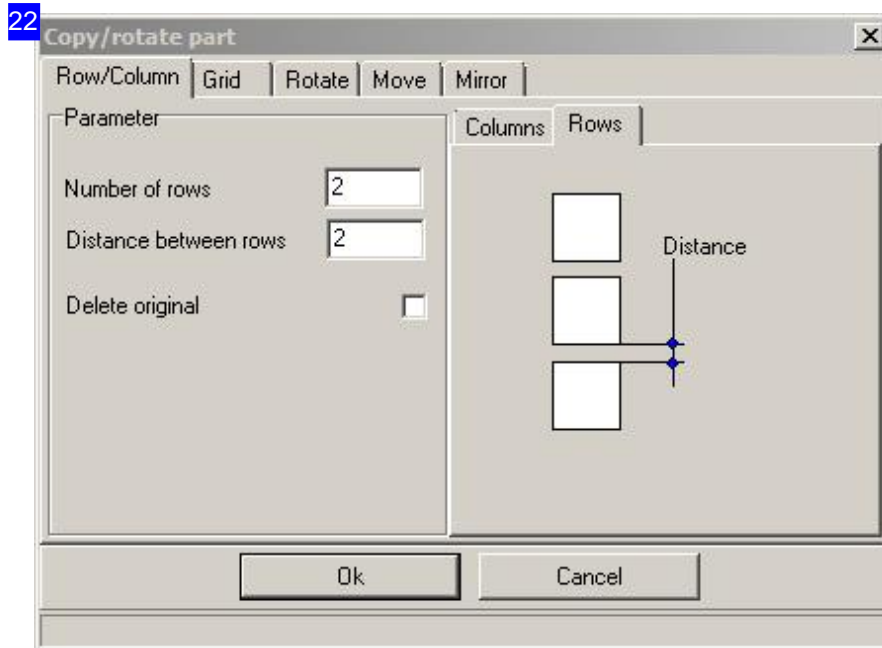


In this picture the small circle to the right has been identified, and is to be copied in multiple rows. To do this, select 'Copy/Turn' and LC the reference point.

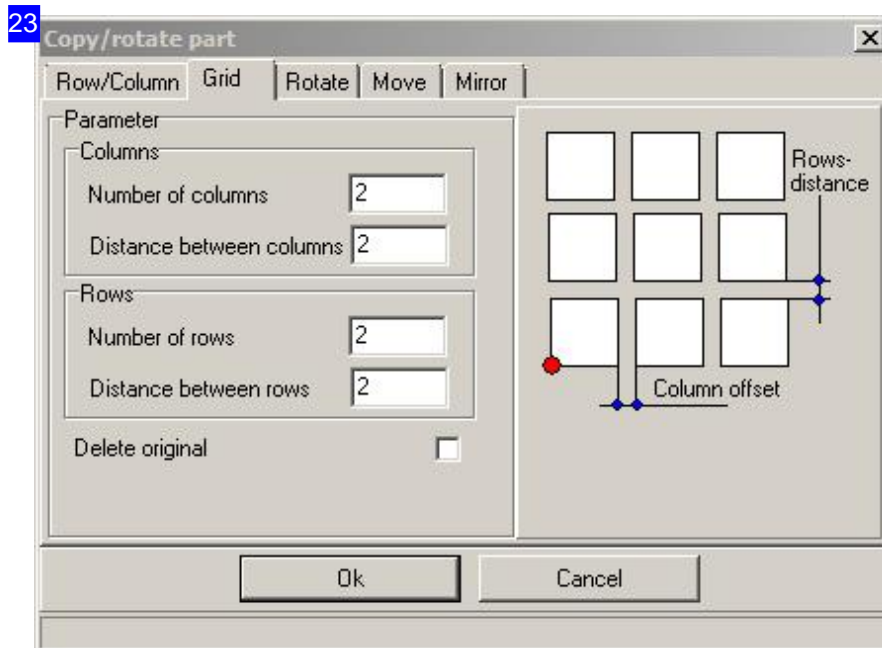
Select the copying method by choosing the tab 'Row/Column' or 'Turn'.



Select the 'Rows' tab and enter the number of rows, and the offset between parts. Click 'Apply' to copy the circle the given number of times into rows. There are separate options for 'Columns' and 'Grid'.

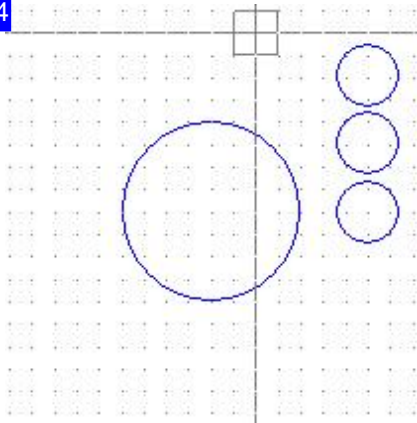


This is the selection dialog for columns. There are separate options for 'Rows' and 'Grid'.



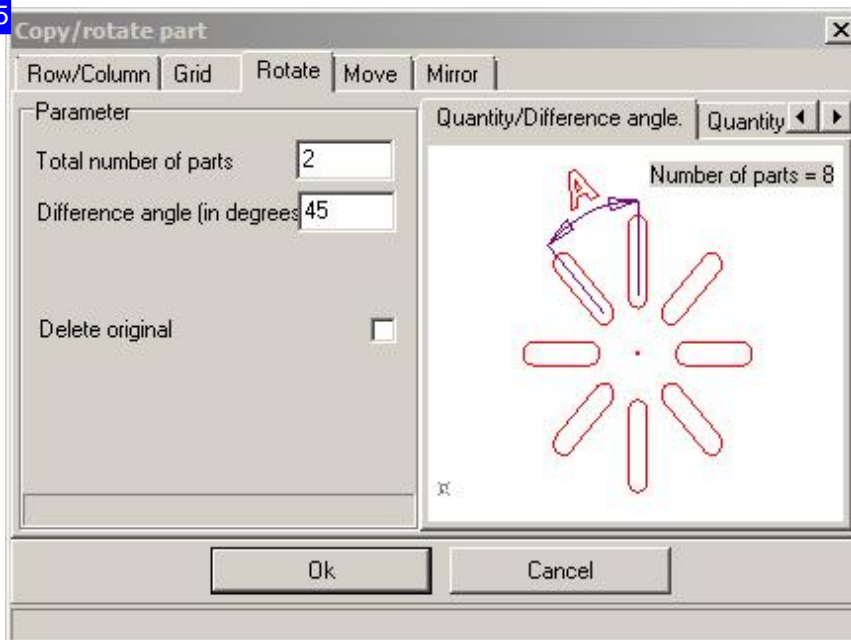
This is the selection dialog for a grid. There are separate options for 'Rows' and 'Columns'.

24



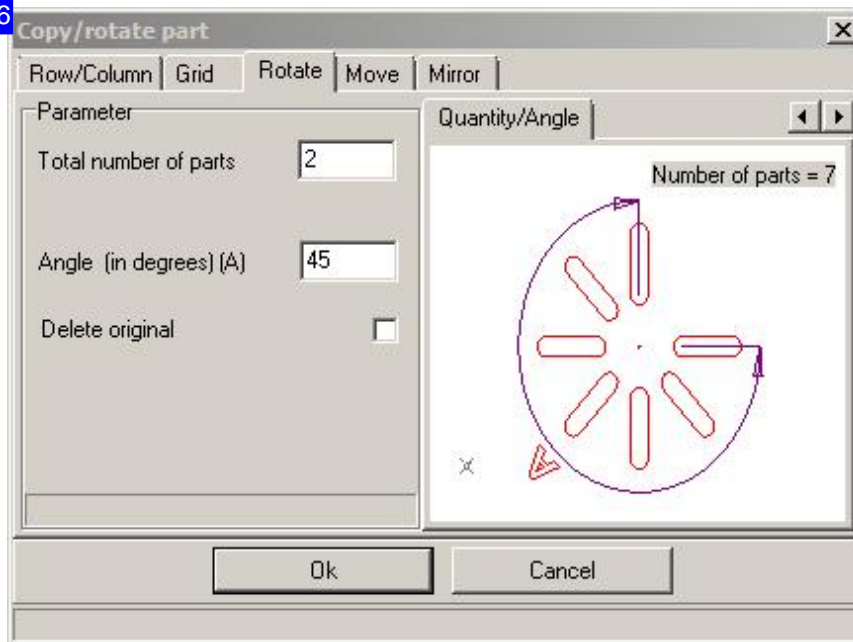
The result of copying in multiple rows. To copy into columns, work in the same manner, but use the 'Columns' option.

25



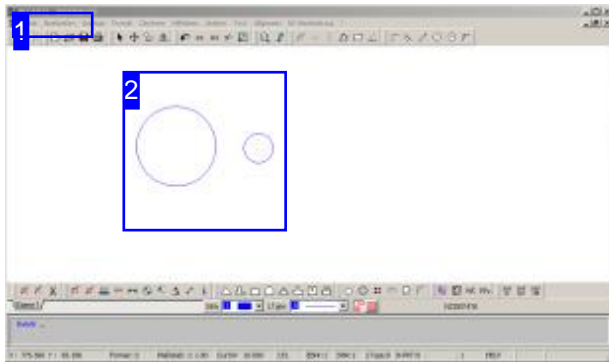
This is the selection dialog for copying around a center with an offset angle. Enter the number of elements and the difference angle between each copy. There is a separate option for 'Total angle'.

26



This is the selection dialog for copying around a center with a total angle. Enter the number of elements and the total angle. There is a separate option for 'Difference angle'.

Free Move.



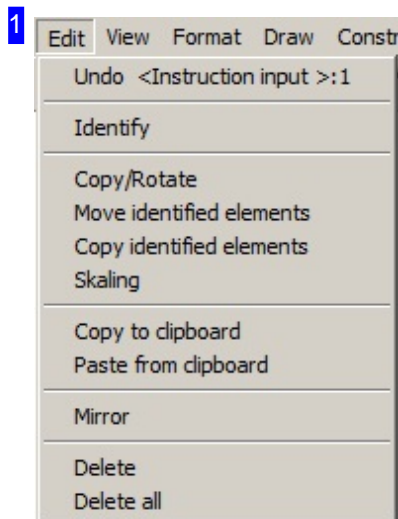
This function allows you to move elements around the drawing. You must first mark the element; it must be identified.

Free-moving drawing elements:

- o Select menu item [1] 'free move'
- o LC in the drawing window; the element will be 'tied' to the cursor.
- o Move the element to the required new position.
- o LC to 'release' the element, placing it in its new location.

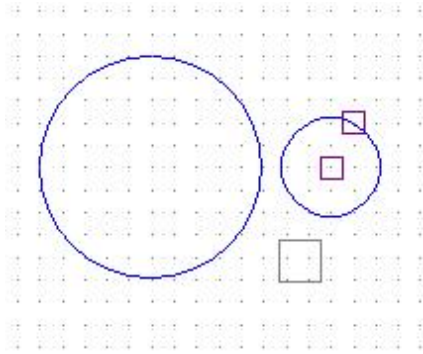
Move via Dialog:

- o Select menu item [1] 'Copy/Turn'
- o LC in the drawing window; select 'Move/Copy' from the selection dialog.
- o Select the 'Move' button.
- o enter X/Y values.
- o Click apply.



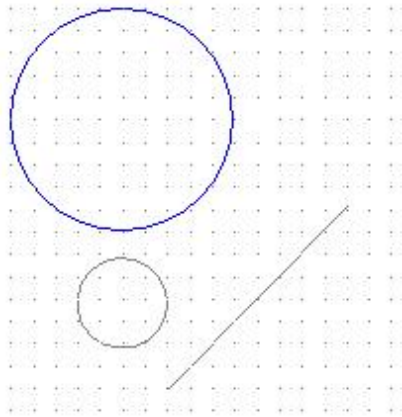
Select the 'Free Move' menu item to move with the mouse, or 'Copy/Turn' to move using a dialog. The relevant elements must be identified first.

2



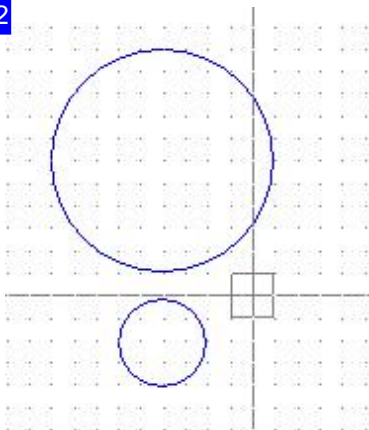
The small circle to the right has been identified and is to be moved. Select the 'Free Move' option and LC the reference point. Move the circle to the required position and LC to place it.

21



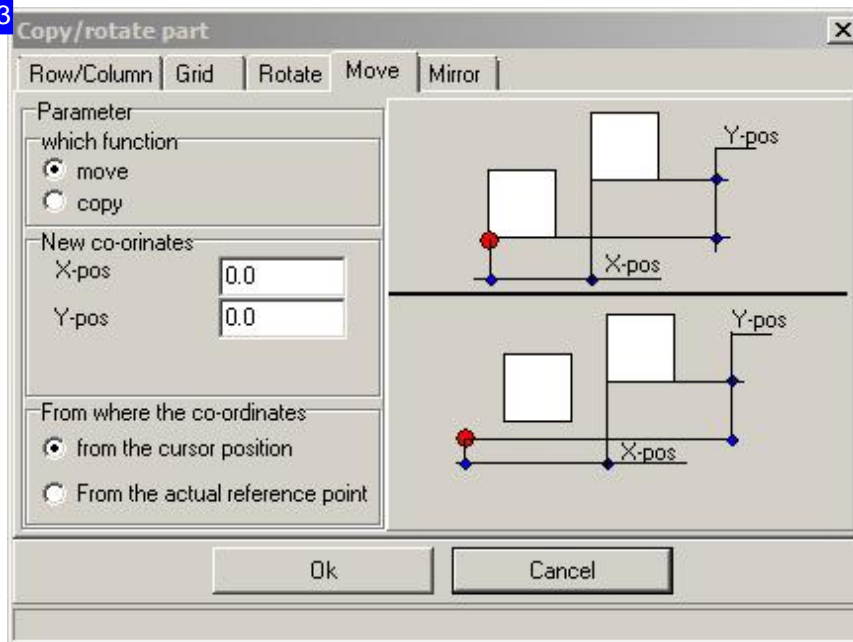
When moving, an offset line between the reference point and the new position will be shown as a check. LC to position the element.

22



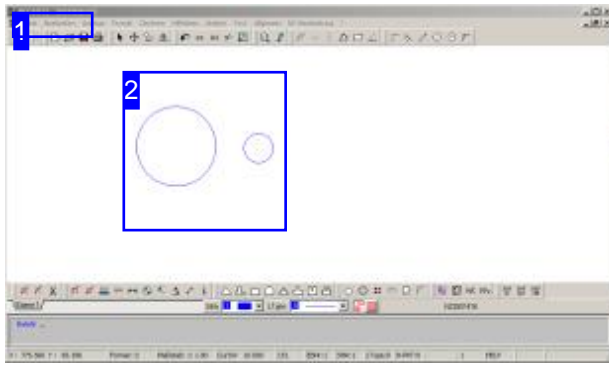
The result of moving.

23



Select the 'Move/Copy' tab, mark the 'Move' button and enter the X/Y position. Click 'Apply' to move the circle to the given coordinates.

Free Copy.



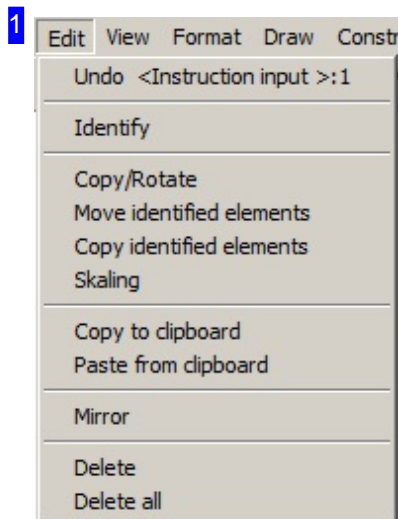
This function allows you to copy geometric elements in the drawing. You must first mark the element; it must be identified.

Free-copying drawing elements:

- o Select menu item [1] 'Free copy
- o LC in the drawing window; the element will be 'tied' to the cursor.
- o Move the element to the required new position.
- o LC to 'release' the element, copying it to its new location.

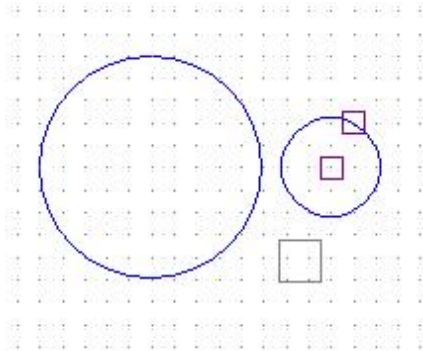
Free copy via Dialog :

- o Select menu item [1] 'Copy/Turn'
- o LC in the drawing window; select 'Move/Copy' from the selection dialog.
- o Select the 'Copy' button.
- o enter X/Y values.
- o Click apply.



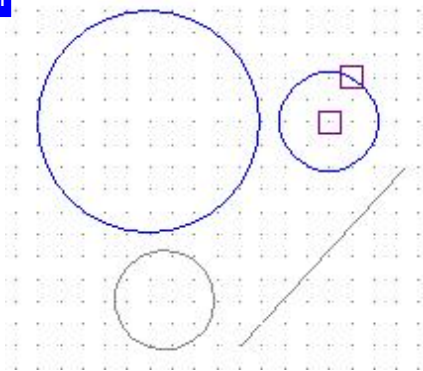
Select the 'Free Copy' menu item to move with the mouse, or 'Copy/Turn' to move using a dialog. The relevant elements must be identified first.

2



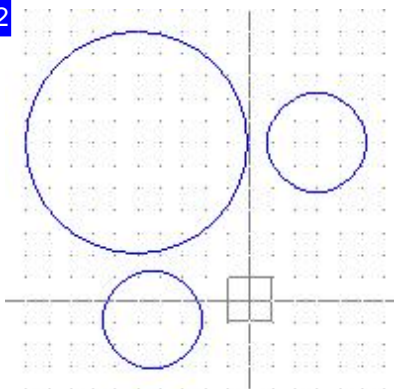
The small circle to the right has been identified and is to be copied. Select the 'Free Copy' option and LC the reference point. Move the circle to the required position and LC to copy it.

21



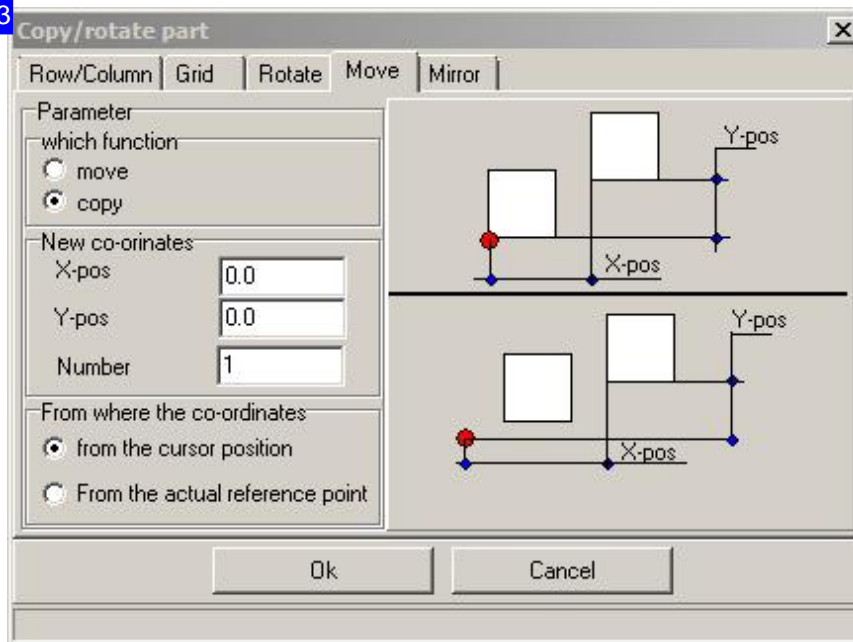
When copying, an offset line between the reference point and the new position will be shown as a check. LC to place the element.

22



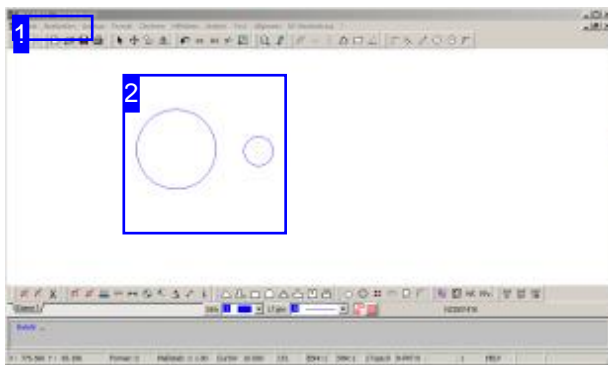
The result of copying.

23



Select the 'Move/Copy' tab, mark the 'Copy' button and enter the X/Y position. Click 'Apply' to copy the circle to the given coordinates.

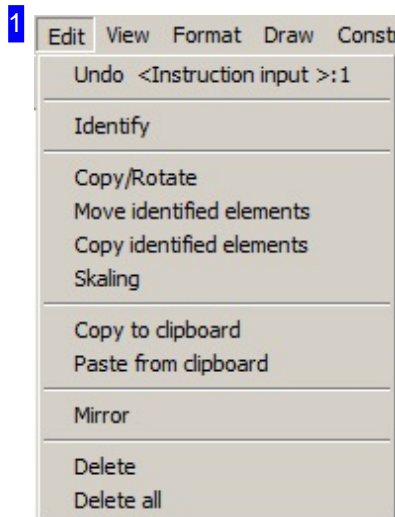
Scaling



Scaling geometric elements:

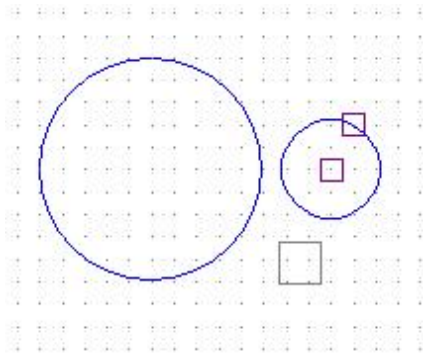
- o Select menu item [1] 'Scaling'.
- o LC on the drawing window to call a dialog box.
- o Enter the scaling factor.
- o Click 'Apply'; the elements is rescaled.

The function allows an element's size to be increased or reduced. The relevant element must be identified first.



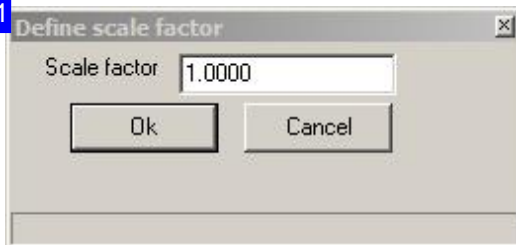
Select the 'Scaling' menu item.
The relevant elements must be identified first.

2



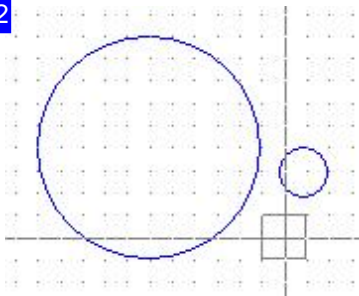
The small circle to the right has been identified and is to be rescaled. Select the 'Scaling' option and LC the element. a dialog box appears for you to enter the scaling factor. Enter the value and click 'Apply'.

21



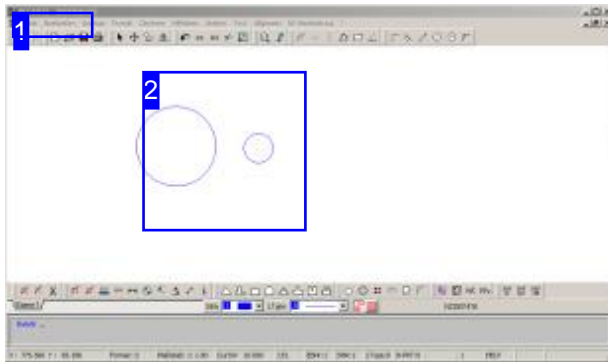
Enter the scaling factor for the increase or reduction in size; the element will be re-drawn.

22



The result of scaling.

Mirror



This function allows you to mirror geometric elements as required. First give a mirroring axis by defining its start and end point. A reference point is given as a construction point which can be useful when mirroring horizontally or vertically. The requirement for this is that an element has been selected first.

Mirroring geometric elements:

- o Select the menu item [1] 'Mirror'
- o Click the reference point.
- o Click the start point of the mirroring axis.
- o Click the end point of the mirroring axis.

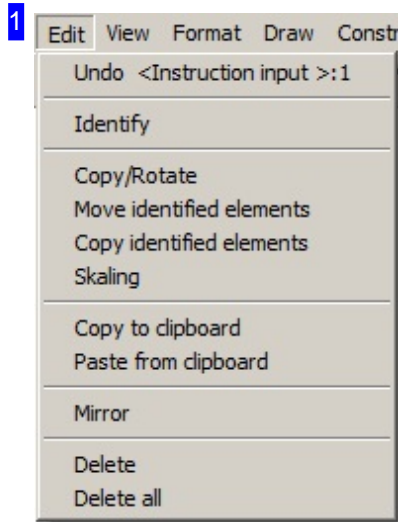
The function can also be accessed via the quick-keys.

Free copy via Dialog :

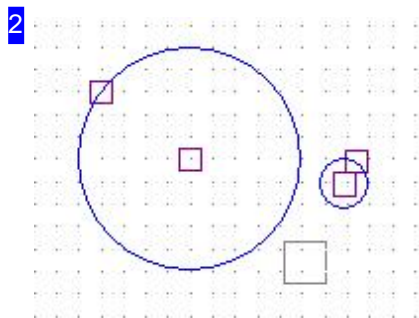
- o Select menu item [1] 'Copy/Turn'
- o LC in the drawing window; select 'Mirroring' from the selection dialog.
- o Select the axis.
- o Click apply.

Freies Spiegeln über Dialog [31]:

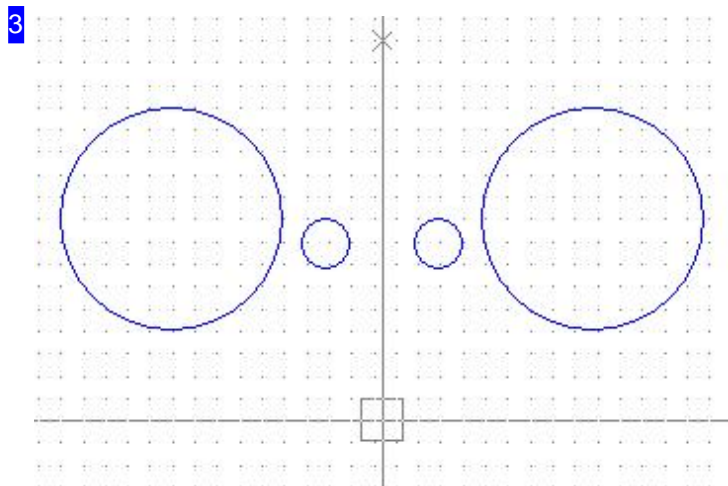
- o Menüpunkt [1] 'Kopieren/drehen' wählen.
- o MT auf Zeichenfeld; im Auswahldialog Tab 'Spiegeln' wählen.
- o Achse wählen.
- o Übernehmen drücken.



Select the 'Mirror' menu item.
The relevant elements must be identified first.

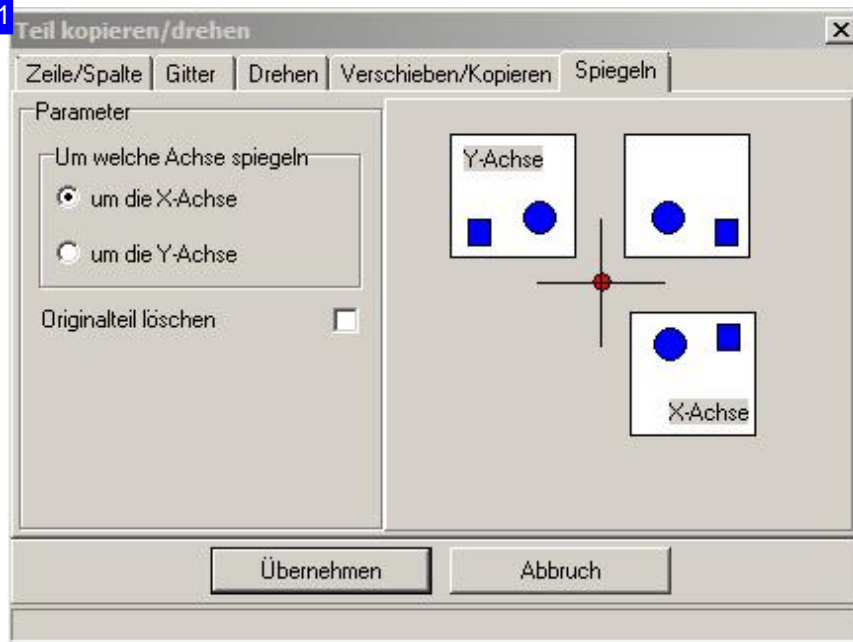


Both circles have now been identified and are to be mirrored. To do this, select the 'Mirror' menu item and click the reference point. The cursor 'runs' along the horizontal or vertical axis. Then click the start and end point. The elements are shown mirrored along the mirroring axis. The start and end points are freely selectable and are not bound to the reference point!



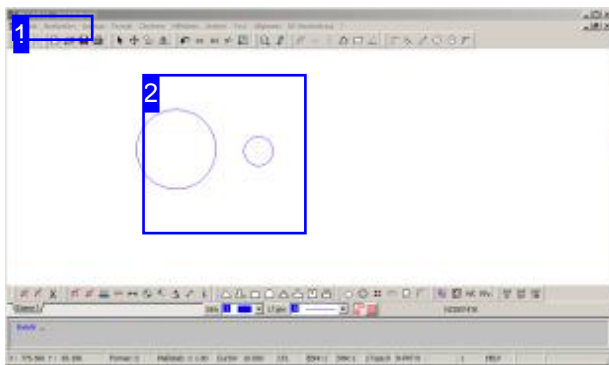
The mirrored element.

31



Select the tab 'mirror', select the axis. Clicking on 'apply' will copy the part at the marked position.

Clipboard



An element can be copied to the clipboard for use any number of times in the drawing. The copy is related to a reference point you set when you save to the clipboard.

You must first mark the element; it must be identified.

Copying to the clipboard:

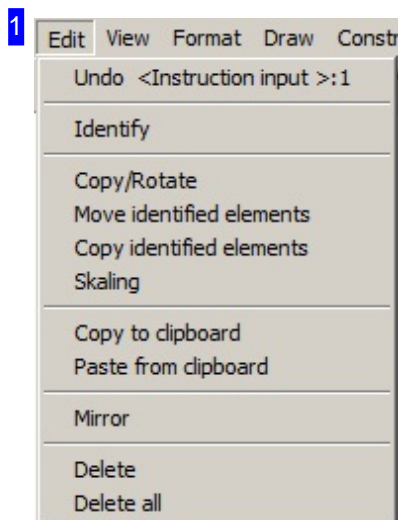
- o Select menu item [1] 'Copy to Clipboard'.
- o Click at reference point.

The function can also be called using the quick-keys <Ctrl><Insert> or <Ctrl><C>.

Pasting from the clipboard:

- o Select menu item [1] 'Paste from Clipboard'.
- o Click the reference point.

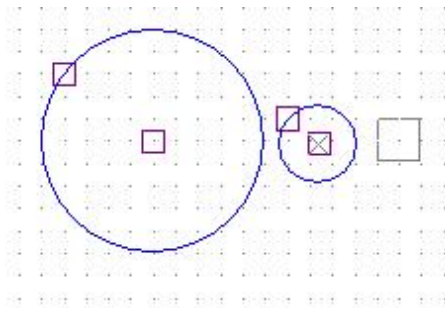
The function can also be called using the quick-keys <Ctrl><Insert> or <Ctrl><V>.



Select the menu item 'Copy to the Clipboard', or 'Paste from the Clipboard' respectively.

The relevant elements must be identified first.

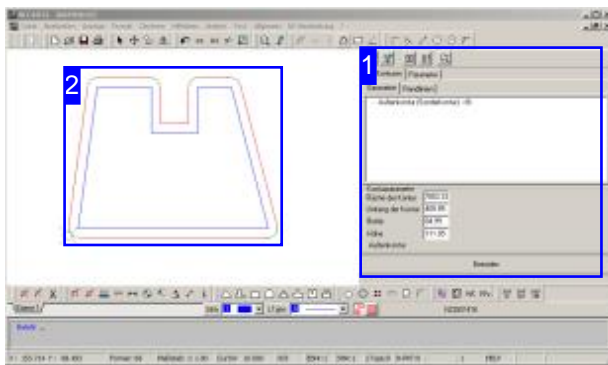
2



Both circles have been identified for copying to the clipboard here. Select the menu item 'Copy to clipboard' [1] and click the reference point.

To paste, select the 'Paste from clipboard' menu item [1] and click to copy the element, from its reference point, at the cursor position.

Display Equidistants.

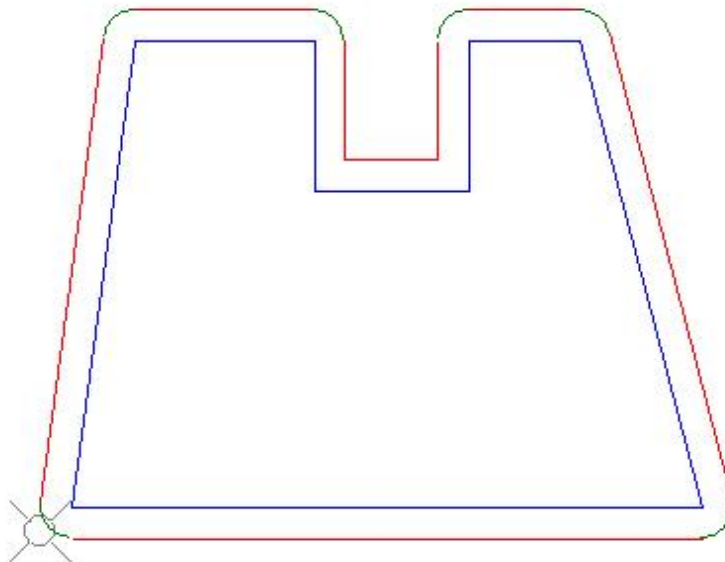


This screen is accessed via the Menu selection 'Edit' - 'Draw equidistants'.

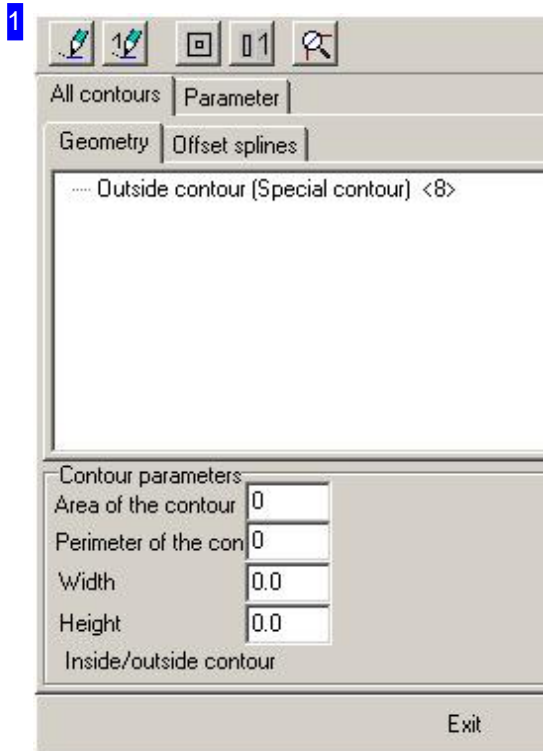
After creating an equidistant, check the contours for any overlaps in close spaces, and correct manually.

Equidistants are construction lines with an encircling clearance to a contour. This clearance is usually for the cutting tool. When displayed you receive extra information about the amount of material being used productively.

2



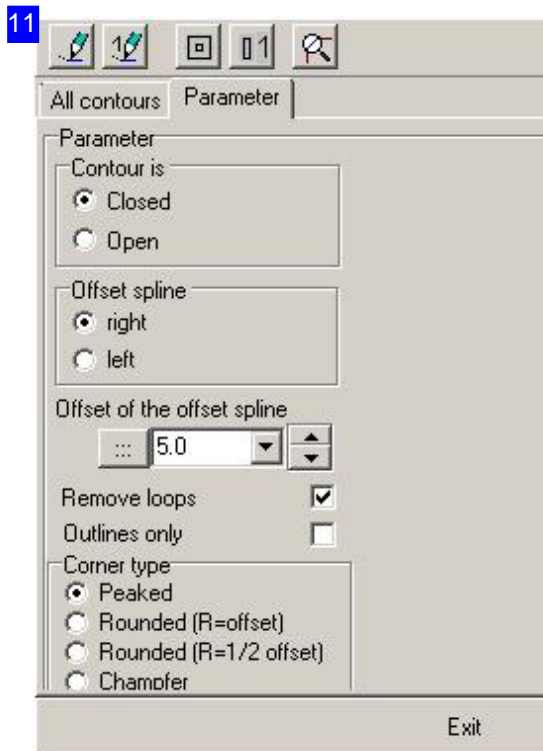
In this example equidistants have been placed on the external contour. The corners are rounded.



The 'All Contours' tab allows you to check the drawings. The bottom pane shows the calculated equidistants' values. Switch to the 'Parameter' tab to make settings for the calculation.

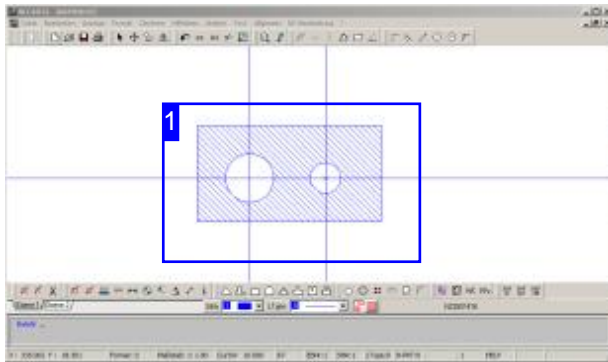
The functions of the buttons in the header panel are:

- o Redraw all.
- o Display only the current contour.
- o Create equidistants on all contours.
- o Create equidistants on only the current contours.
- o Display all geometric elements of the current contour.



This tab allows you to quickly enter the settings for equidistants.

Hatch An Area.



Command: SCHRAF

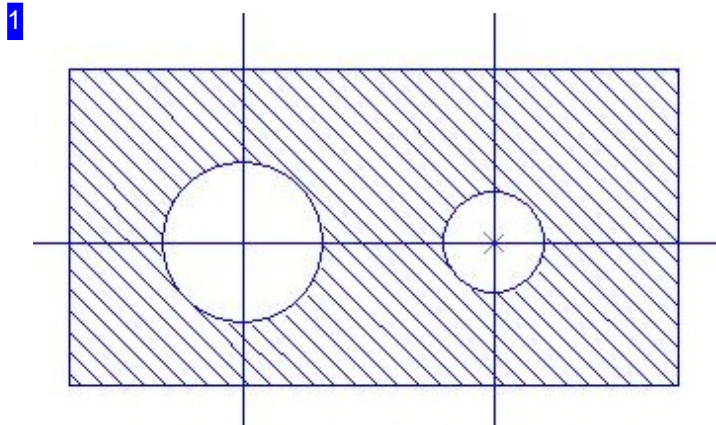
Code: 28

Parameters: Hatch offset in mm; hatch angle +/-; layer number; area identifier (area=2); start point; end point; hatch area; end - entry is sequential
Example: 28 5 -45 2 2 ...

Menu: yes

Icon: no

Function: The selected area is hatched on the entered layer.



Hatch a selected area:

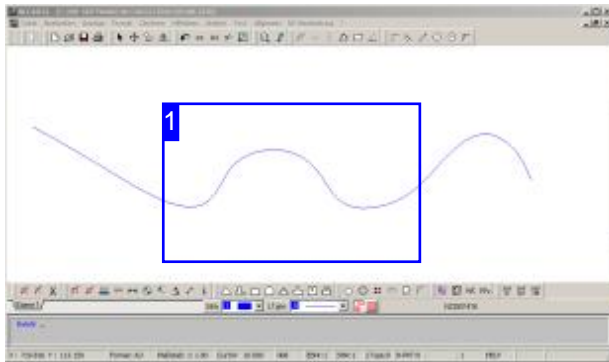
1. Menu

1. Function 'Draw > Hatch > Enter values in the command line.
2. Cursor to start point, click.
3. Cursor to end point, click.
4. Cursor to hatch area, click.

2. Command line

1. Enter command or code and parameters in the command line.
2. define area.
3. Finalize area selection 'E'.
4. Move to hatch area.
5. Finalize function 'E'.

Drawing And Resolution Of Splines.



Command: SPLINE

Code: 77

Parameters: Start point; support points 'E', resolution, 'E', save - entry is sequential

Example: 77, 1, <click, click,...>, E, 7, 10, E, <Enter>

Menu: yes

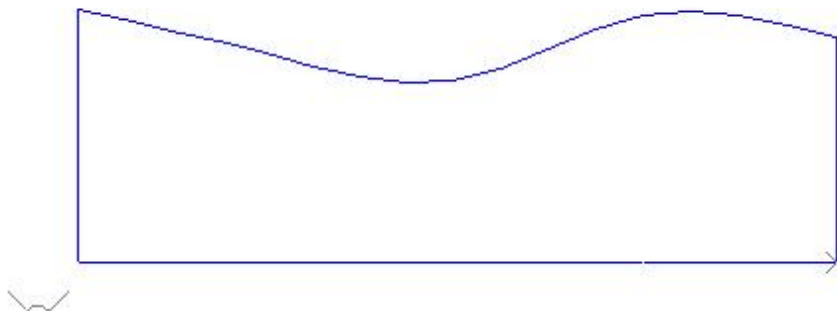
Icon: no

Function: A spline is drawn using the input or definition of support points and can be dissolved into any number of vectors and saved as a DXF file.

When entering the curves the spline points are identified through circles which can then be moved using entries in the command line.

If the drawing is saved as a ZEW file the spline remains in its entirety; single points are however then unidentifiable.

1



A spline curve as a partial contour in a component. Resolved into lines and saved as a DXF, the individual points can then be used as geometric drawing points for further editing.

Drawing a spline:

1. Menu

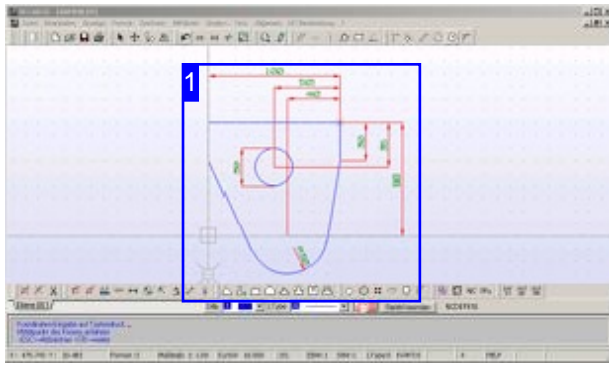
-
1. Function 'Draw > Spline > Enter values in the command line.
 2. Cursor to start point, click.
 3. Cursor to support points, click at each.
 4. End spline with 'E' in the command line.
 5. Edit spline; set resolution '7' in the command line.
 6. Save spline <Enter>.

2. Command line

1. Enter command or code and parameters in the command line.
2. Cursor to support points, click at each.
3. End spline with 'E' in the command line.
4. Edit spline; set resolution '7' in the command line.
5. Save spline <Enter>.

Dimensioning

Giving Dimensions in your Drawing.



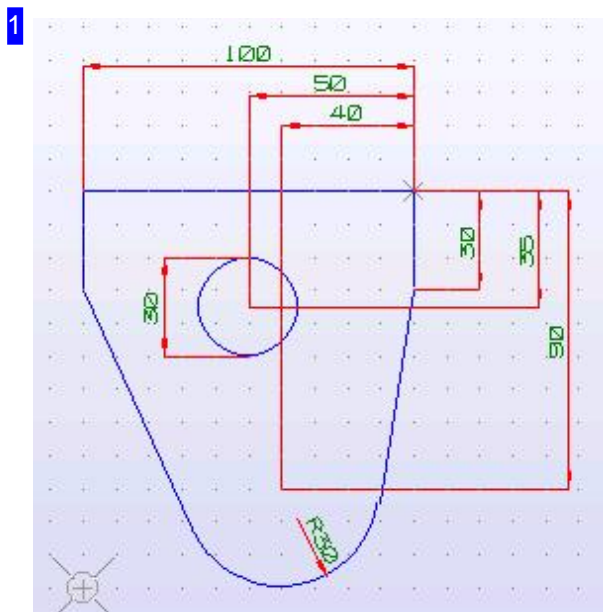
ncCAD32 allows you to dimension your drawings according to the standard rules. The programs menus will help you dimension your drawing correctly.

When dimensioning your workpiece, you should consider the arrangement of the dimensions carefully and then use the relevant menu items to locate the dimensions on your drawing in the correct order.

When placing the first dimensioning lines, leave yourself enough space to place any others you will need.

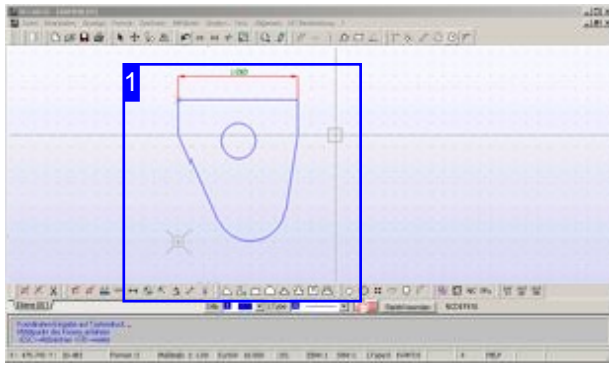
Important: After aligning a dimension, you must press <CR> or LC to place the dimension's label!

Dimensioning with and without numbers is done in exactly the same way, and so this procedure will only be described once (for dimensioning with numbers).



This workpiece is to be dimensioned between two reference lines. Menu items 1. Dimensioning and 2. Dimensioning, last Dim., Circle and Arc will be required to align the dimensioning.

Dimensioning a Length.



Function: Lays a dimensioning line (ML) across a length between a first dimensioning limit point (MBP) and a second dimensioning limit point.

Command: BEM2T (BEM2)

Code: 54 (34)

Parameter: 1.MBP; 2.MBP; ML - Inputs are made sequentially.

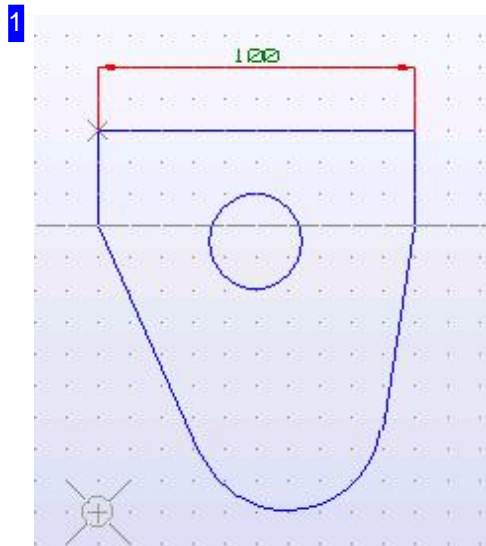
Example: 54; 1.MBP; 2.MBP; ML; <CR>; <CR>.

Menu: yes

Symbol: yes

-

The codes in brackets apply for dimensioning without measurements.



Lay a dimensioning line via:

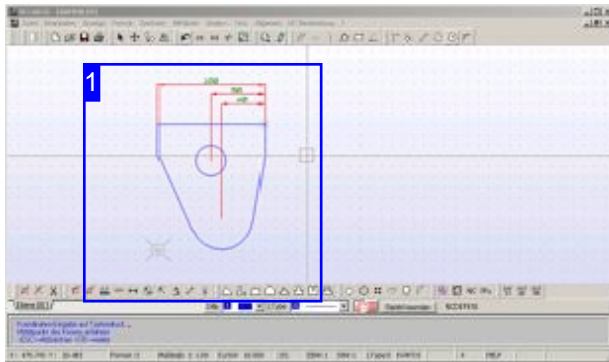
1. Menu

1. Select the function 'General > Dimensioning > XY Dimensioning > With Measurements'.
2. Move the cursor to the first dimensioning limit point and LC.
3. Move the cursor to the second dimensioning limit point and LC.
4. Move the cursor to the dimensioning line's position and LC.
5. LC!

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the first dimensioning limit point and press <CR>.
3. Move the cursor to the second dimensioning limit point and press <CR>.
4. Move the cursor to the dimensioning line's position and press <CR>.
5. Press <CR>!

Dimensioning Using a Reference Line.



Function: Lays dimensioning lines (ML) across a length between two limiting points, whereby a reference line is drawn from the first limiting point. The sequence must be adhered to when adding a dimensioning line in order to retain the relation to the reference line.

First set a first dimensioning between two limiting points. A reference line will be laid on the second point and (virtually to begin with) on the first limiting point.

Command: no

Code: no

Parameter: 1.MBP; 2.MBP; ML - Inputs are made sequentially.

Menu: yes

Symbol: no

-

Set a second dimensioning line relating to the same first limiting point.

Command: no

Code: no

Parameter: 2.MBP

Menu: yes

Symbol: no

-

Place last dimensioning line; the reference line on the first limiting point will finally be displayed along with all dimensioning lines related to it.

Command: no

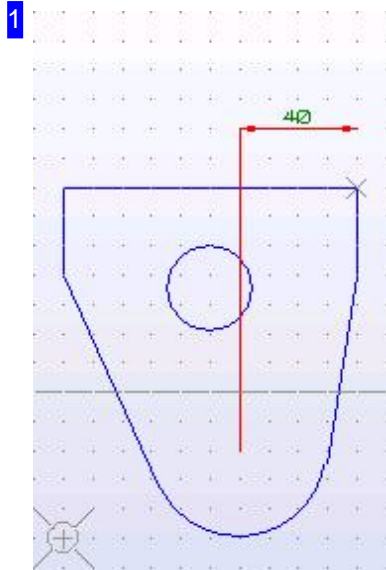
Code: no

Parameter: 2.MBP

Menu: yes

Symbol: no

-

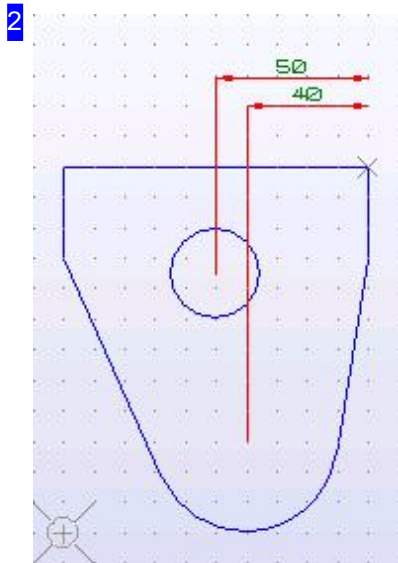


Laying a dimension on a reference line.
Set the first dimension via:

1. menu:

1. Select the function 'General > Dimensioning > XY Dimensioning > First Dimensioning'.
2. Move the cursor to the first dimensioning limit point and LC.
3. Move the cursor to the second dimensioning limit point and LC.
4. Move the cursor to the dimensioning line's position and LC.
5. LC!

After the first dimensioning line, a second dimensioning line can be related to the same reference line.



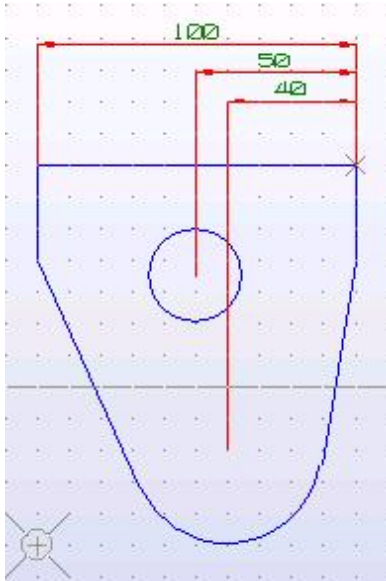
Setting a dimensioning line on a reference line.
Add a second dimensioning line via:

1. Menu:

1. Select the function 'General > Dimensioning > XY Dimensioning > Second Dimensioning'.
2. Move the cursor to the second dimensioning limit point and LC.
3. LC!

After the second dimensioning line, a final dimensioning line must be added to the reference line to close it.

3



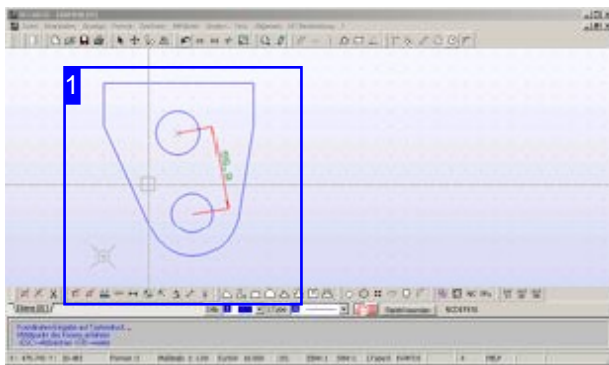
Laying a dimensioning line on a reference line.

Lay a final dimensioning line via:

1. Menu:

1. Select the function 'General > Dimensioning > XY Dimensioning > Final Dimensioning'.
2. Move the cursor to the final dimensioning limit point and LC.
3. LC!

Dimensioning Between Two Points.



Function: Lays a dimensioning line (ML) between a first limiting point (MBP) and a second.

Command: BEM1T (BEM1)

Code: 53 (33)

Parameter: 1.MBP; 2.MBP; ML - Inputs are made sequentially.

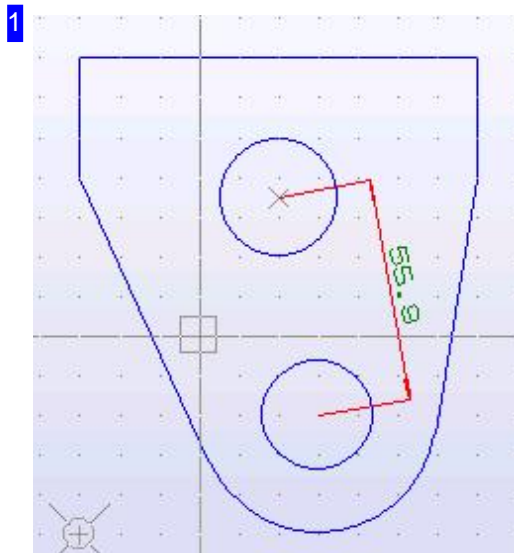
Example: 53; 1.MBP; 2.MBP; ML; <CR>; <CR>.

Menu: yes

Symbol: yes

-

The codes in brackets apply for dimensioning without measurements.



Lay a dimensioning line via:

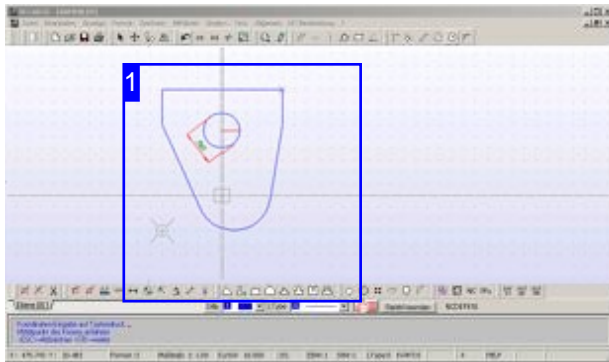
1. menu

1. Select the function 'General > Dimensioning > Point Dimensioning > With Measurements'.
2. Move the cursor to the first dimensioning limit point and LC.
3. Move the cursor to the second dimensioning limit point and LC.
4. Move the cursor to the dimensioning line's position and LC.
5. LC!

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the first dimensioning limit point and press <CR>.
3. Move the cursor to the second dimensioning limit point and press <CR>.
4. Move the cursor to the dimensioning line's position and press <CR>.
5. Press <CR>!

Dimensioning a Circle.



Command: BEMK

Code: 43

Parameter: Circle; ML - Inputs are made sequentially.

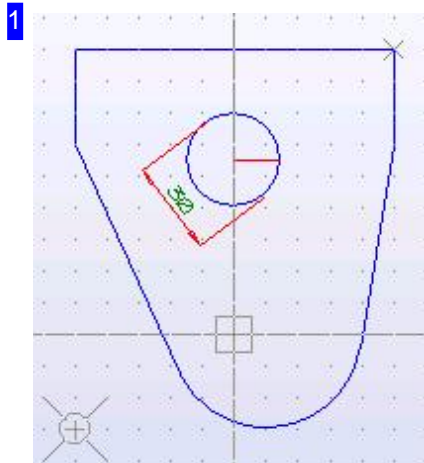
Example: 43; Identify circle; ML; <CR>; <CR>.

Menu: yes

Symbol: yes

-

Function: Lays a dimensioning line (ML) on a circle.



Lay a dimensioning line via:

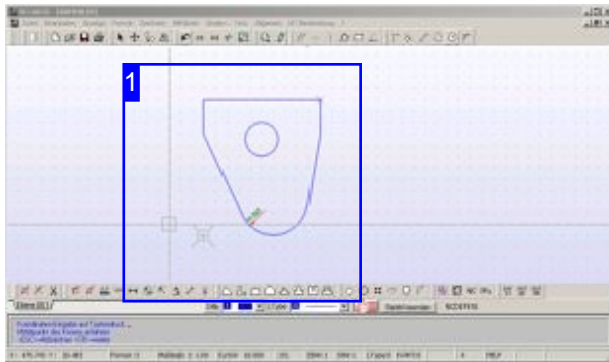
1. Menu

1. Select the function 'General > Dimensioning > Circles'.
2. Move the cursor to the circle and LC.
3. Move the cursor to the dimensioning line's position and LC.
4. LC!

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the circle and press <CR>.
3. Move the cursor to the dimensioning line's position and press <CR>.
4. Press <CR>!

Dimensioning an Arc.



Command: BEMB

Code: 63

Parameter: Arc; ML - Inputs are made sequentially.

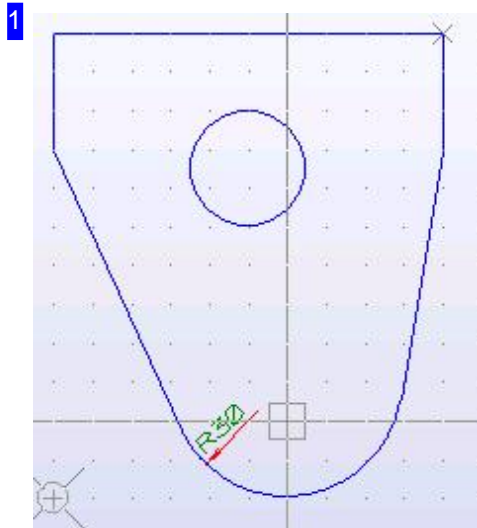
Example: 63; Identify Arc; ML; <CR>; <CR>.

Menu: yes

Symbol: yes

-

Function: Sets a dimensioning line (ML) on an arc.



Lay a dimensioning line via:

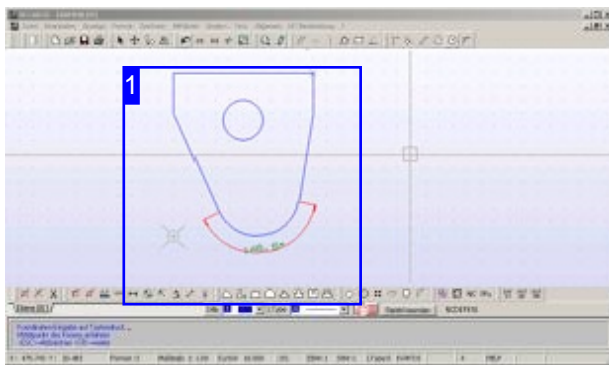
1. Menu

1. Select the function 'General > Dimensioning > 'Arcs'.
2. Move the cursor to the circle and LC.
3. Move the cursor to the dimensioning line's position and LC.
4. LC!

2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the arc and press <CR>.
3. Move the cursor to the dimensioning line's position and press <CR>.
4. Press <CR>!

Dimensioning an Angle.



Command: BEMW

Code: 64

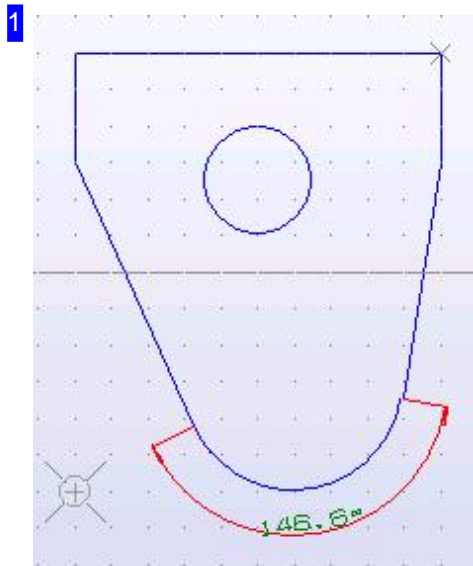
Parameter: 1.HL,1.GP; 1.HL,2.GP; 2.HL,1.GP;
2.HL,2.GP; ML - Inputs are made sequentially.

Example: 64; 1.HL,1.GP; 1.HL,2.GP; 2.HL,1.GP;
2.HL,2.GP; ML; <CR>; <CR>.

Menu: yes

Symbol: yes

Function: Sets a dimensioning line (ML) on an angle. The angle is limited by two reference construction lines (HL) which are defined by a first and second point in the drawing (GP). The ML can be located inside or outside the arc, or be placed as an opposite angle.



Lay a dimensioning line via:

1. menu

1. Select the function 'General > Dimensioning > 'Angle'.
2. Move the cursor to the first HL, first GP and LC.
3. Move the cursor to the first HL, second GP and LC.
4. Move the cursor to the second HL, first GP and LC.
5. Move the cursor to the second HL, second GP and LC.
6. Move the cursor to the dimensioning line's position and LC.
7. LC!

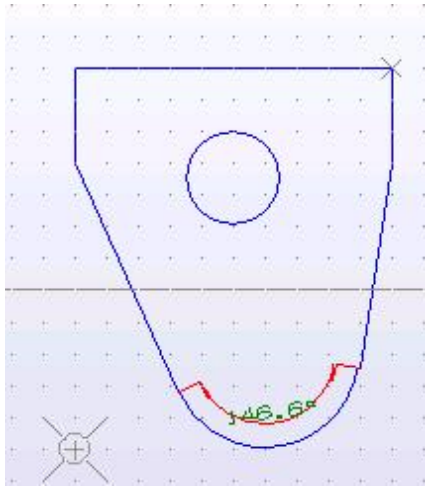
2. Command Line

1. Enter the command or code in the command line.
2. Move the cursor to the first HL, first GP and Press <CR>!.
3. Move the cursor to the first HL, second GP and Press <CR>!.
4. Move the cursor to the second HL, first GP and Press <CR>!.
5. Move the cursor to the second HL, second GP and Press <CR>!.

6. Move the cursor to the dimensioning line's position and Press <CR>!.
7. Press <CR>!

The dimensioning line can also be placed internally on the angle.

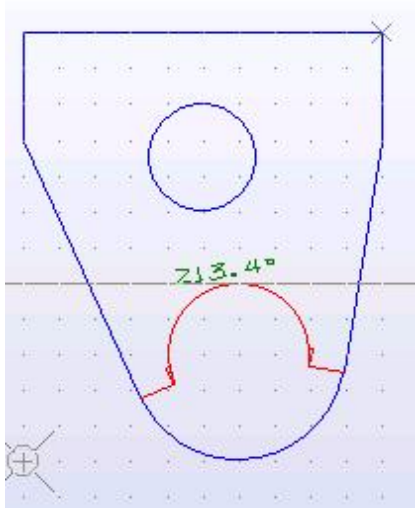
2



Now for an internal dimensioning line.

An opposite angle will also be shown to give the full picture.

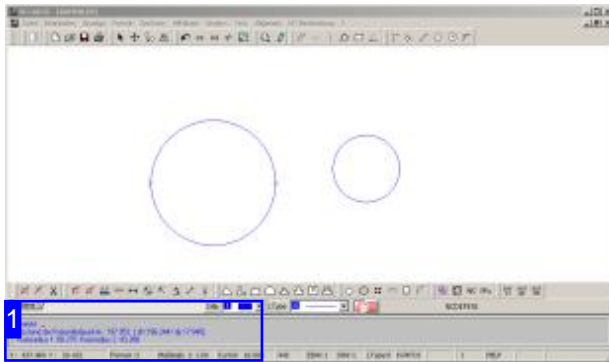
3



We have used the same subject here to illustrate the use of an opposite angle, despite the fact that it is not a sensible place to use it.

Measuring

Use drawing points to determine a dimension.

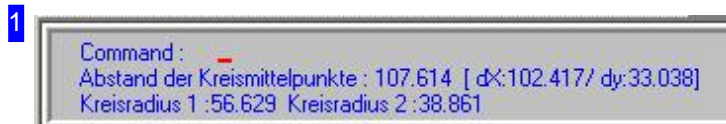


By defining drawing points you can use existing information to calculate certain dimensions of contour elements in your drawing..

In the submenu 'General', 'Measure' you will find the following functions for determining sizes:

- o Distance between two points in X and Y direction (69, ABM).
- o Polar distance between two points (79, ABP).
- o Surface area of a contour (89, FLB).
- o Size of an enclosing min/max box (219, MINMAX).
- o Offsets and sizes of one or two circles.

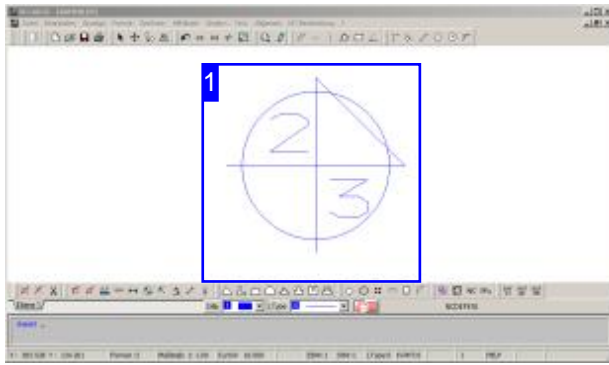
The functions use codes (in brackets) which when expanded with drawing points allow calculation of the values, and display the results in the command line.



When measuring circles you can select one or two circles. If you select two circles then the distance between them will be displayed as well as the circle parameters for both. If both points are on the same circle then only the data for this circle will be displayed.

Symbols

Create And Use Symbols.



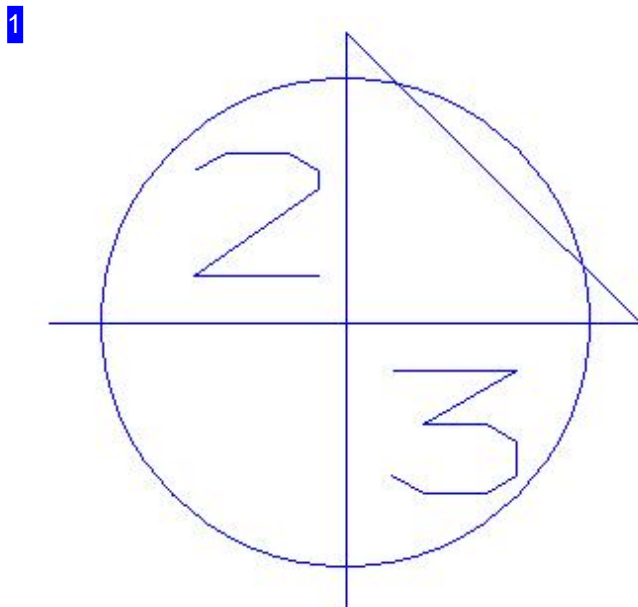
In the submenu 'General', 'Symbols' you will find the following functions for using symbols:

- o Create symbol (37, SYMD).
- o Call symbol (38, SYMA).
- o Symbol 2 points (48, SYMP).
- o Symbol angle (58, SYMMW).
- o Output: places a symbol in a drawing.
- o Search: finds the symbol nearest the cursor.
- o Delete: delete a symbol identified using the (search) command.

ncCAD32 gives you the option to use symbols as markings in your drawings. Symbols can be drawn to your requirements in the program, and saved separately as .SYM files. Symbols can then be scaled and added anywhere on your drawing.

The functions use codes (in brackets) which when expanded with positions create or display symbols.

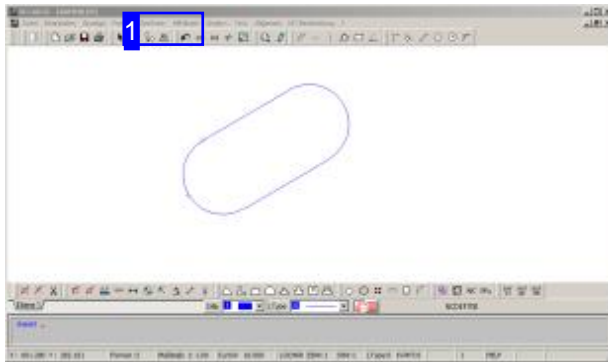
Symbols are saved independently in ZEW files and are not considered during processing. Save drawings with symbols as DXF files and the symbols will be deconstructed to identifiable elements.



Draw a symbol to your needs and labeling requirements, then save the symbol using the menu item 'General', 'Symbols', 'Create symbol'. Define the symbol using a min/max box defined by start and end points, then name and save your symbol as a separate .SYM file.

Macros

Macros As Predefined Drawings Or Drawing Elements.



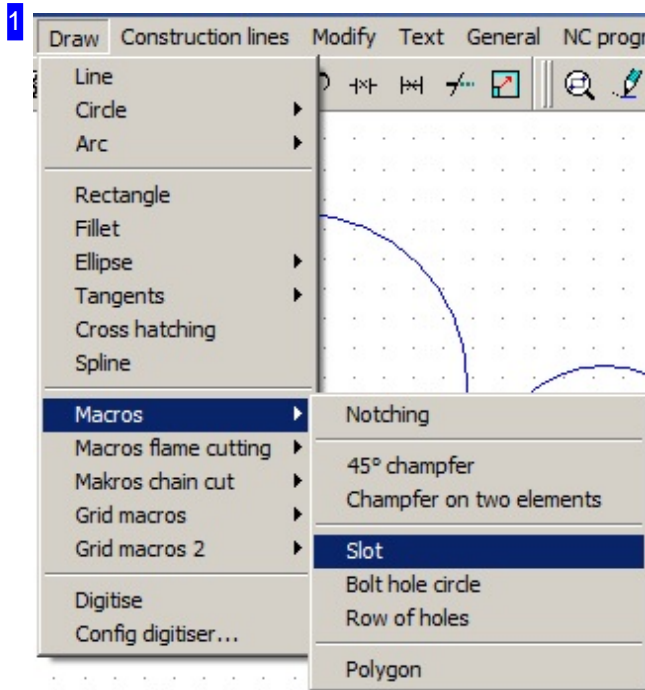
Macros are predefined drawings or drawing elements which can be adjusted to suit your recurring needs. They allow you to create and insert complex shapes like long holes, bolt hole circles, hole rows, notches, etc. quickly and easily.

Macros can also be executed by entering their names in the command line. The required values or steps will be prompted sequentially in the command line. It should be noted that an element's position or the start point for the macro must first be set using the cursor. For example to add a beveled edge, the cursor must first be set to the relevant corner.

The 'Draw' menu contains a separate block for macros, in which macros are sorted by purpose ready for direct selection.

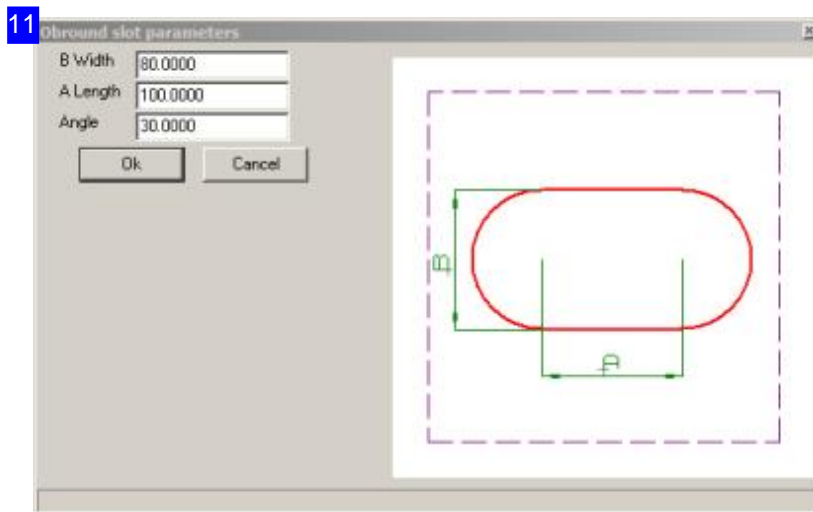
- o General macros
- o Macros for torch cutting
- o Chain cuts
- o Grid
- o Macros for perforated sheets

Macros can also be added to the icon bar so that frequently used macros can be called instantly. Right-click the blank element in the menu panel to access a selection menu to manipulate the icon bar.



The 'Draw' menu contains the macros grouped by function, these can be called in submenus.

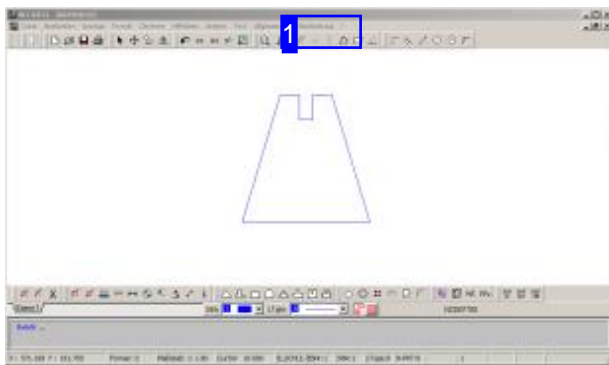
After selecting a macro you should select its start point then enter its parameters in the pop-up dialog which appears.



This dialog shows the selected macro. The drawing to the right shows the general form with the variable parameters identified. The drawing to the left shows a preview based on the current parameters. Enter the values you require and 'confirm'.

NC Processing

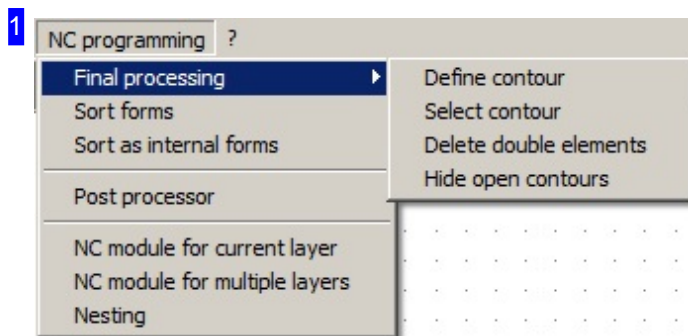
Prepare Your Drawing For NC Processing.



'NC Processing' allows you to take all necessary steps to prepare your drawing for passing to a production module:

- o Edit contour - select contours and contour elements and move them to different layers.
 - o Define contour.
 - o Select contour.
 - o Delete duplicate elements.
 - o Hide open contours.
- o Sort contours.
- o Sort inner contours.
- o Post-processor
- o Call NC module with current layer.
- o Call NC module with multiple layers.
- o Nest

After drawing a contour you may need to take further steps before it's ready to be processed. Production modules could be stamping or cutting modules which work with the post-processor to create the required NC code.

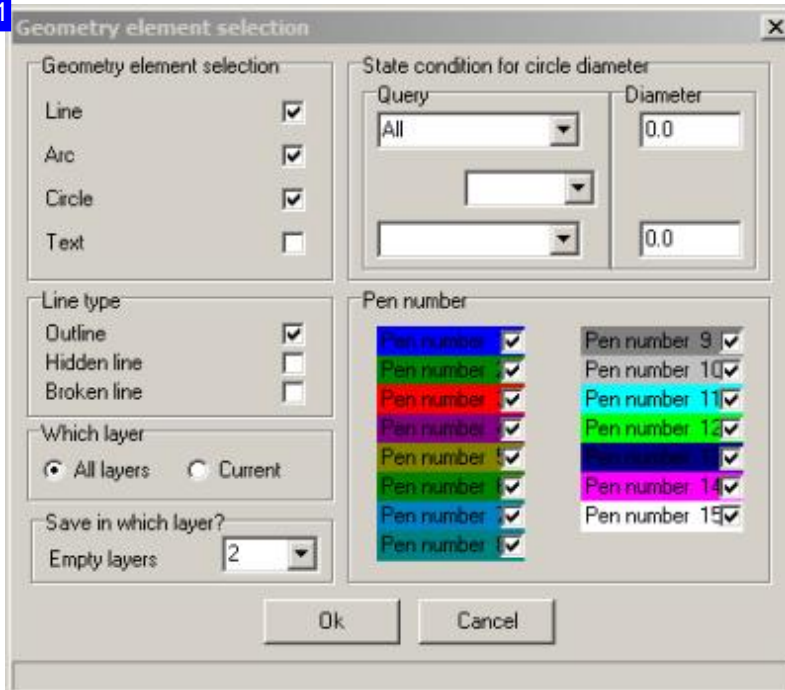


The 'NC Processing' menu allows you to take all necessary steps to prepare your drawing for passing to a production module:

The submenu 'Select contour' allows you to select an area containing multiple contours to be moved to a shared layer.

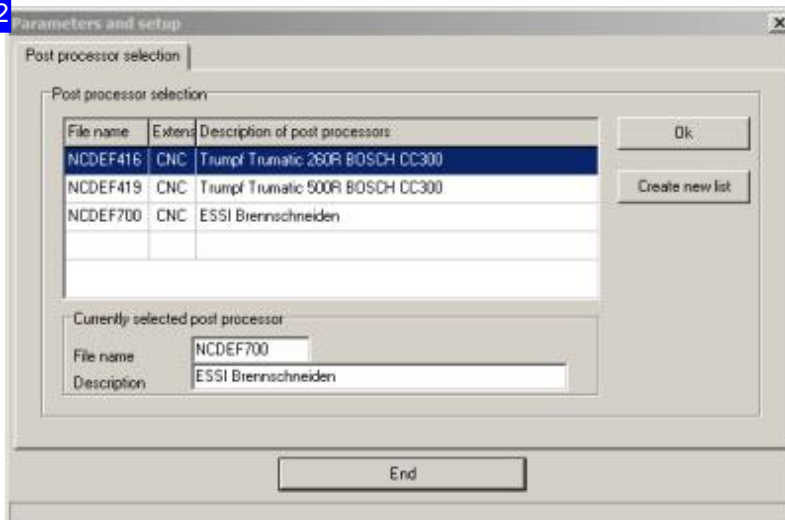
Sort contours: *ncCAD32* automatically sorts your workpiece's contours into outer and inner contours. If contours should be cut as inner contours because the outer contour is defined by a fitting, then make the relevant setting under 'Sort as inner contours'.

11



You can set the selection criteria for transferring the contours in the selected area in this dialog. Mark the selection boxes and enter the layer number in the bottom-left field, then click 'confirm'.

12



All post-processors installed on your system are listed in this dialog. Select a post-processor from the list and click 'confirm'. The subsequently created NC program will be suited to a specific cutting machine according to your post-processor selection.

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