ncSchneid

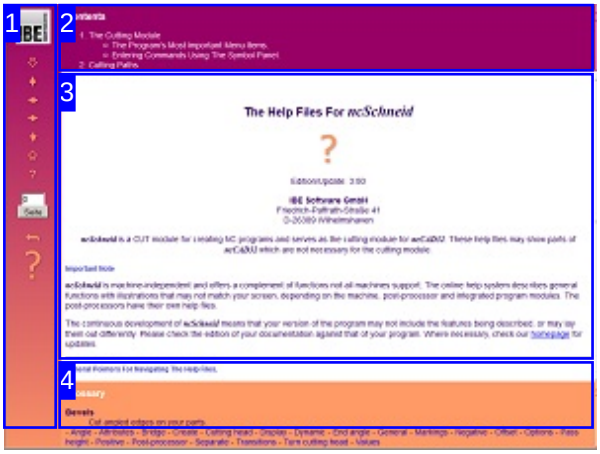
Online help SHEET METAL CENTER Plus

Cutting Modul for ncCAD32

IBE Software GmbH; Friedrich-Paffrath-Str. 41; D-26389 Wilhelmshaven
Fon: 04421-994357; Fax 04421-994371; www.ibe-software.de; info@ibe-software.de

Welcome to ncSchneid's help files

General Tips for Navigation of the Program



Choosing the Control Method:

There are two ways to navigate the help pages for *ncSchneid* .

1. Interactive Navigation
   This allows you to decide for yourself which texts you need to read. Place the mouse cursor over the rectangles drawn onto the screenshots to get a more detailed description.

2. Sequential Navigation
   This mode leads through the document sequentially. All details will be shown one after the other, simply press the 'Next' button (right arrow in the Navigation Panel [1]) to move through the document one step at a time.

The screen is split into four frames: The Menu to the left [1] with the navigation arrows for navigating the whole help document, and a box for entering a page number.

The top shows the Contents List [2] divided into chapters. The Glossary [4] of the most important technical terms is shown at the bottom. You can switch between the Glossary and an Index in this frame. Whichever you choose, clicking the catchwords will link you directly to the most relevant part of the corresponding help page.

The middle frame shows you the current help page [3].

Choose your preferred method of navigating the document

interactive

or

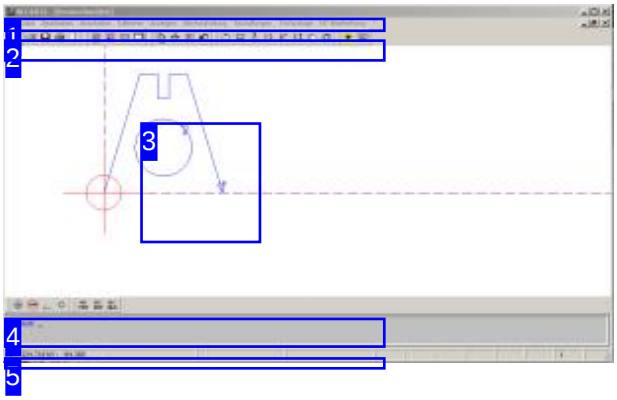
continuous

Important Note
The *ncSchneid* program is not machine dependant. It offers an abundance of features, some of which will not be supported by your machine. This online help document describes the general functions based upon GUIs which, dependant on your machine, the modules you have installed and your post-processor(s) may appear differently on your copy. The post-processors have their own independent help files.

The continuous development of *ncSchneid* can also lead to differences between the program's actual GUI and the GUI shown in the online help documents. If this occurs, please check the version and date of the help file against that of the program. If necessary, check our Internetseite for a newer version, or contact us.

The Cutting Module

Preparing and Cutting Components.



This is the working screen of *ncSchneid* . The middle area is the current work area, while the header and footer panel contain menu, command and status panels.

ncSchneid is a module which is called from *ncCAD32* . When the program is closed you will be returned to *ncCAD32* .

If *ncSchneid* is called from the job management module then many supplementary data will be drawn from this database, and updated data returned there.

The program is protected by a license dongle and must be authorized for use. You can learn more about dongles in the online help for *ncCAD32* .

Important:

ncSchneid is tool manufacturer independent software and thus requires the correct post-processor to generate NC code suitable for your machine. Don't forget to select the correct post-processor in *ncCAD32* .

ncSchneid allows you to prepare your parts and generate the NC code required to produce them using your cutting machine.

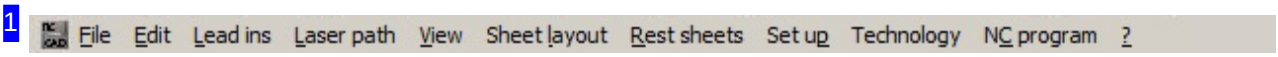
Preparation involves creating the cutting tool paths, adding lead-ins and lead-outs, sorting contours into internal (holes) and external (parts) contours, etc. In short, all processes which are required so that your machine's post-processor can make a part from your drawing.
The final cutting paths can be saved as a '.LAW' file for reuse later.

Multiple components can be placed together on a sheet to form a nesting plan, which can be saved as an '.SPL' file. Optional nesting modules allow you to place parts automatically on a sheet.

Properly sorted closed contours are prerequisite to obtaining correct nesting results. Internal and external contours are usually sorted in opposite directions. Contour elements must be lines, arcs or circles.

Tips:

- If in these papers reference is made to 'clicking' an object, this means left-click with your mouse, unless specifically stated otherwise.
- The cursor can be moved using the mouse, the command line and also the arrow keys.
- Terms in angle brackets (<Esc> <Enter> <A>) refer to keys on your keyboard.



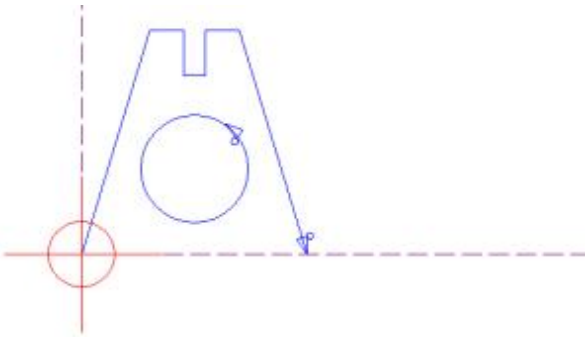
The 'menu panel' allows you to import or create new cutting paths, make modifications or change the

software's settings. The main menu items are described in the chapter titled 'The program's most important menu items'.

2

This 'icon bar' makes a selection of often-used commands available with a single click. This panel is user-configurable - more details in the chapter 'Entering commands using the icon bar'.

3

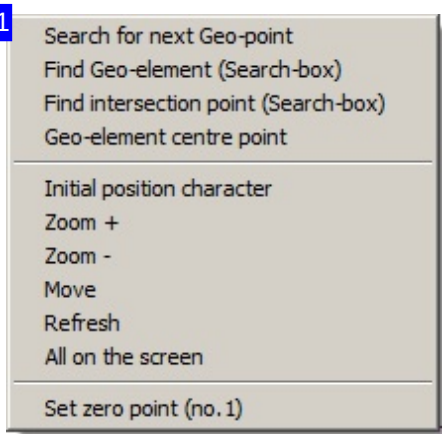


This area is the 'work screen' with its cursor highlighted by a crosshair. Right-click in the main area to open a pop-up menu with various options for finding elements and displaying your drawing.

Use the mouse wheel to zoom the drawing centered about the cursor position.

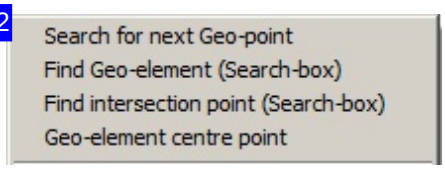
At the bottom left corner of the preview a zero point is displayed. This is defined as 'zero point 0' and is the reference point for your cutting machine.You can set a new zero point at any time which will then be known as 'zero point 1'. Zero points can be defined in the 'configuration screen'.

31



The upper area contains commands for searching centered around the cursor. The middle menu items allow control over the display of your drawing; you can also zoom in and out by <Ctrl>+clicking or <Ctrl>+right-clicking the drawing.

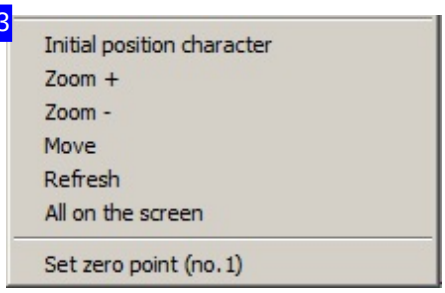
32



These are the functions of the pop-up menu 's search commands :

1. Find the nearest geometry point to the cursor.
2. Find the nearest element in the selection box.
3. Find the nearest intersection in the selection box.
4. Find the center point of the element in the selection box.

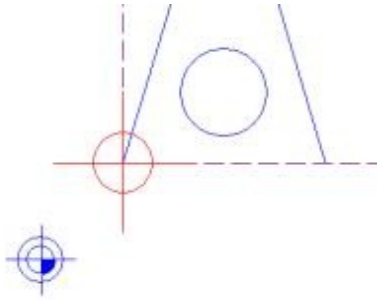
33



These are the functions of the pop-up menu 's display commands :

1. Go back to the start point when drawing.
2. Zoom in.
3. Zoom out.
4. Zoom back in both directions.
5. Move the view around the drawing board.
6. Redraw. Any fragments will be cleared.
7. Show the whole drawing.

34



The (automatic) machine zero point is shown as a fixed reference in the bottom left corner. This zero point is defined as 'zero point 0'. You may set a new zero point at any time, this will be labeled 'zero point 1' (shown here down and left from zero point 0). Select the corresponding menu item to move the null point.

4

This is the program's 'command line'. This line allows you to enter precise text commands as a sequence of numbers and actions.
In the text mode the next processing step is displayed. To exit a processing sequence press <Esc>.
The command line can also be used as an input mask where the line's content can be changed according to your instructions.
The input mask can be accessed directly by pressing <F2>.

The cutting module will accept only relevant commands.

41

The entry mask allows you to enter cursor positions as polar coordinates (angle and offset) or X/Y positions. Click the corresponding button on the left side and enter the desired coordinates. Click the arrow button next to the position entry fields to place a guide mark at the entered position. Click 'apply' to move the actual cursor to this position.
The bottom buttons on the left side allow you to enter a new cursor position using triangulation.
To keep the input mask open for continued use, mark the checkbox at the top right. The checkbox below defines whether coordinates are relative or absolute from the zero point.

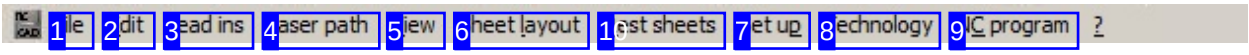
42

To enter the cursor position using triangulation, select either the side length or angle and enter a value under X or Y. The cursor moves directly to the newly calculated point.

5

Status panel with information about the cursor position and the drawing. The second field shows the current lead-in selected from the lead-in management. Click this field to place a lead-in on any contour. The field is divided into two halves for this purpose - click left to select a lead-in and click right to place a lead-in.

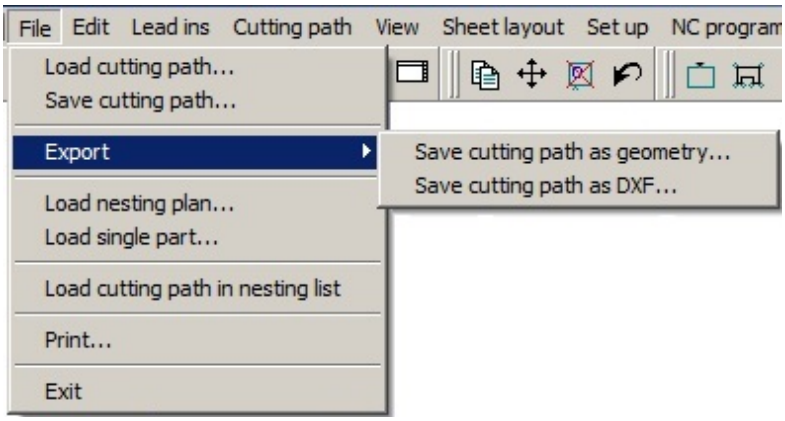
The Program's Most Important Menu Items.



The menu panel of *ncSchneid* . Each menu item's validity is decided by the source (cutting path, nesting plan or individual part) and the selected post-processor.
Many of the menu items can also be selected from the icon panel.

The 'Menu Panel' allows you to create and edit cutting paths, import existing ones, allocate technology data, define cutting routes and make general settings.

1

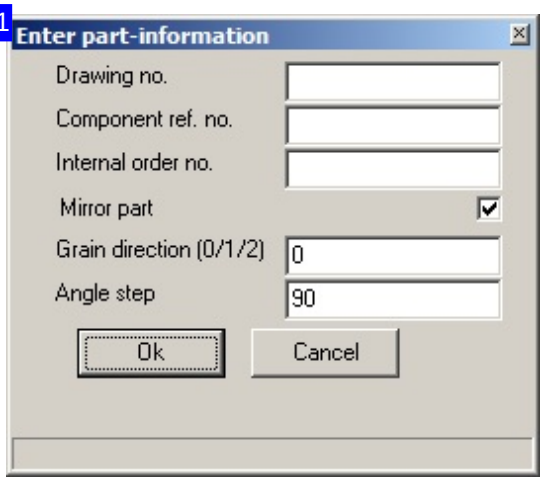


'File' allows you to import and save cutting paths; import nesting plans and single parts; export cutting paths as contours or DXF drawings; and exit the CUT module.
You can find more information on 'Printing' in the online help for *ncCAD32* . There you will also find more details about the 'File manager'.

Switch to the file manager to import a cutting path; here you can select from your .LAW file database. When saving cutting paths a dialog will open where you can enter extra information about the file as a descriptor, then a file manager appears to enter the filename and save path.

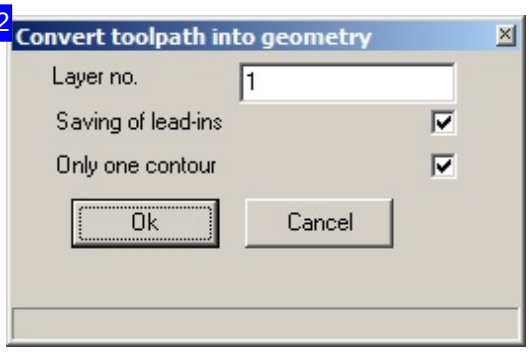
Cutting paths can also be saved in other formats. Use the menu item 'Export' to save files as Geometric drawings or DXF files.
To add multiple cutting paths to a nesting list, select 'Load cutting path to nesting list'. A screen will open to collate files and create a nesting list.

11



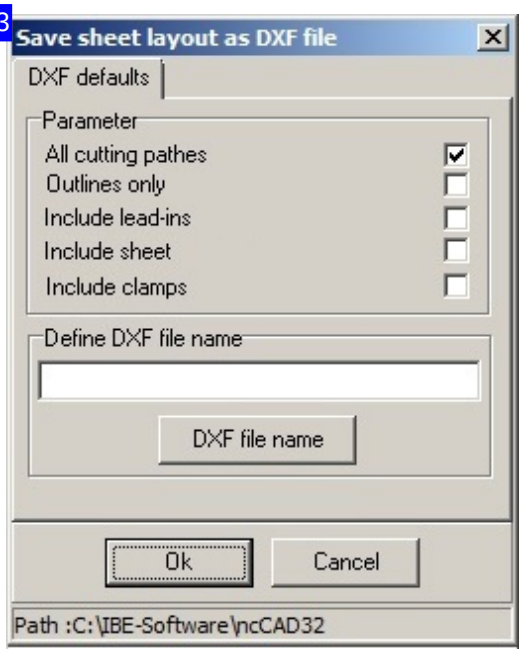
This dialog allows you to enter part information which can be saved as a drawing descriptor. Drawing number, component descriptor and job number are free text strings available to help organize your work.
The bottom fields are for use with variables which must be considered when nesting your parts. Mark the 'mirror part' box to allow the part to be mirrored for better material utilization. The rolling direction can be free or in the X or Y axis. The step angle sets the angle the part should be turned by at each fitting step. Use 90 if the part is oblong!

12



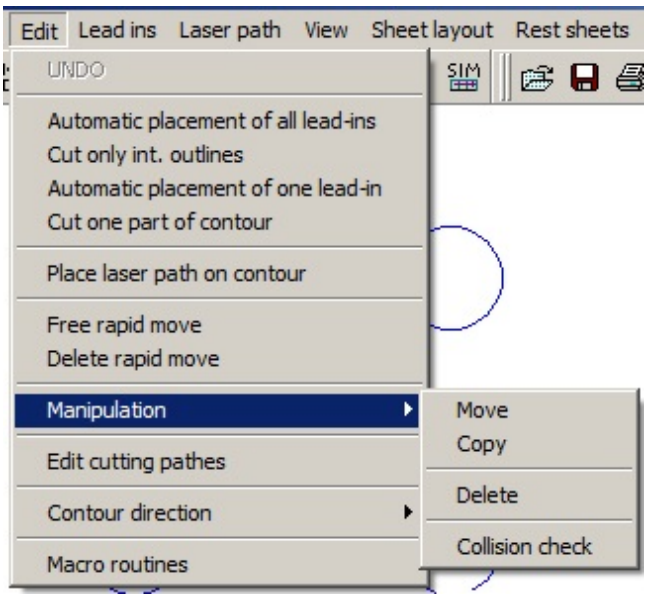
To convert a cutting path to a geometric drawing for *ncCAD32* , you must enter the layer number the contour can be placed in. Mark the corresponding selection boxes to copy lead-ins as drawing elements, and to transfer only one part.

13



This dialog allows you to save a cutting path to a DXF file. Mark the required selection boxes, enter the file path and click 'Apply'. To browse for the correct file path, click DXF file name.

2



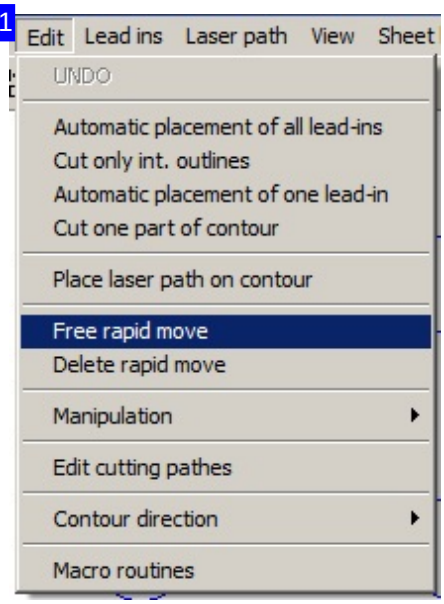
'Edit' contains the options for converting contours into cutting paths, and manipulating cutting paths. Cutting paths are the fundament of *ncSchneid* ; a description of your options here can be found in the 'cutting paths' chapter.

To change the processing direction select 'Geometric contours'.

How to edit individual contours is described in the chapter 'Edit cutting paths'.

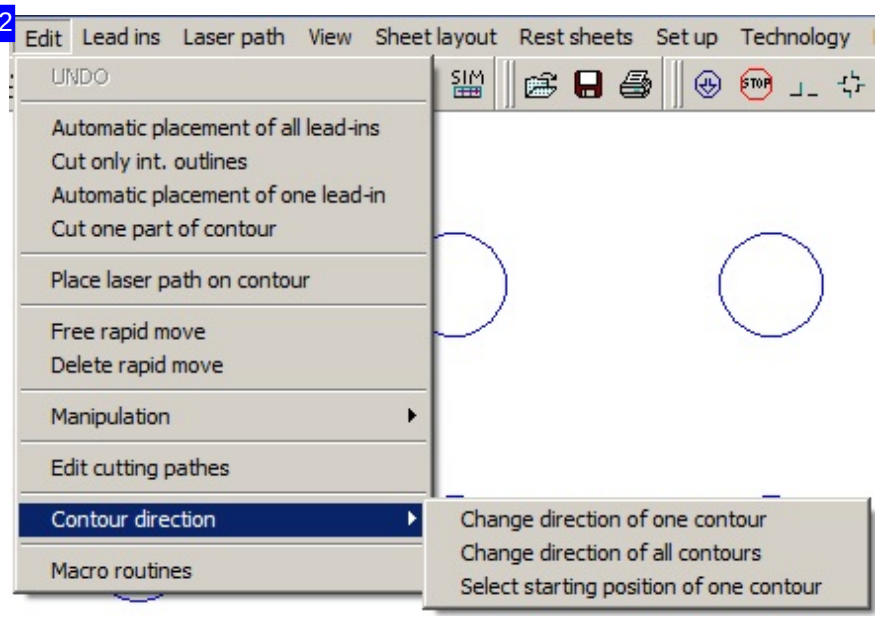
Individual commands can be grouped and saved as macros. More on this in the chapter 'Macros'.

21



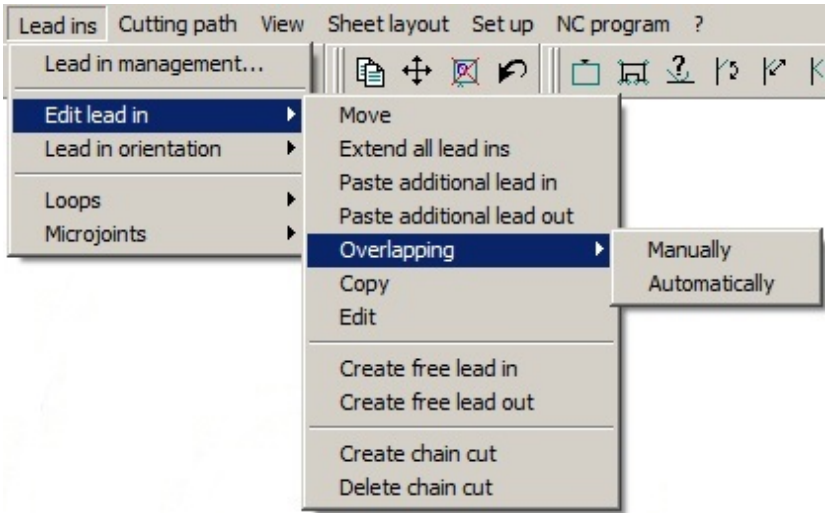
The functions 'Free cutting path' and 'Free idle path' allow you to add extra moves and cuts as required. This allows you to cut up sheets and internal contours as required.

22



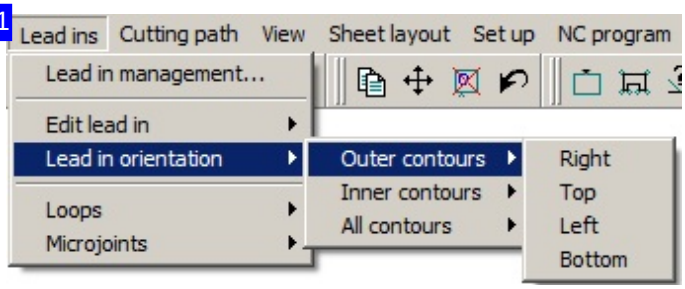
Geometric contours - the processing directions of individual parts can be changed here, giving you full control of the cutting direction.

3



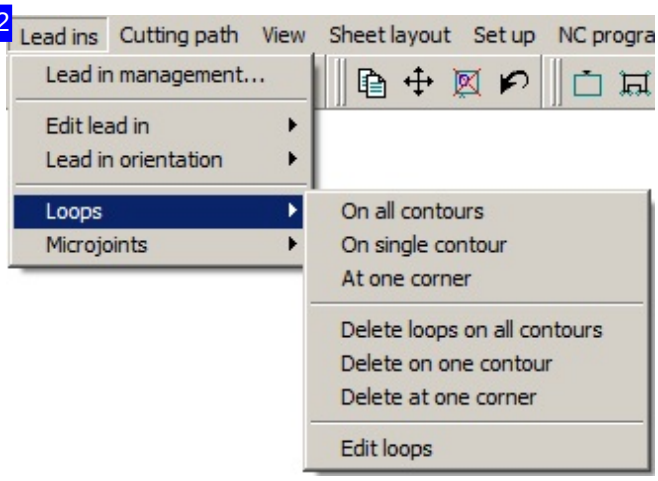
'Lead-ins' allows you to manipulate existing lead-ins and lead-outs (see the Edit Lead-ins chapter for more detail) and place freehand lead-ins on your contours.

31



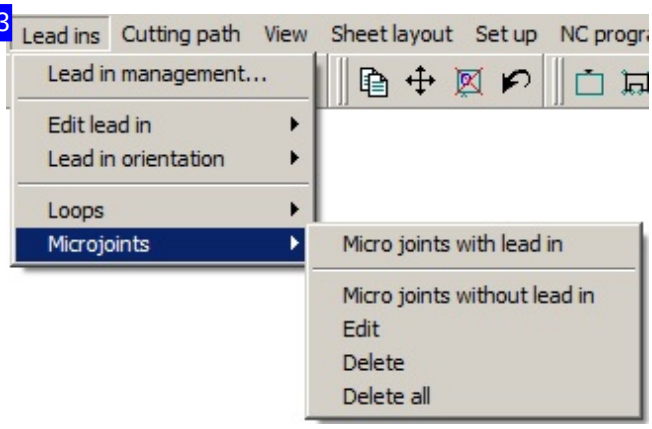
Lead-in Alignment - a simple click sets the global default location for automatic lead-ins.

32



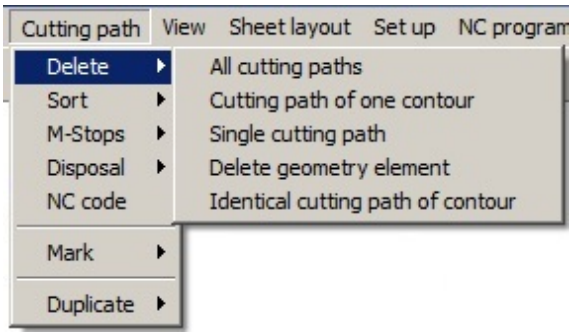
Loops - use the menu items to add and remove loops at your contours' corners. More information in the chapter 'Roundings and Loops'.
The last menu item allows editing of individual loops. Select the required loop and stretch to the size you want.

33



Micro-bridges - to hold parts in a sheet instead of cutting them completely free, add micro-bridges (webbing) to your contours. More information in the chapter 'Cut Parts With Micro-Bridges'.

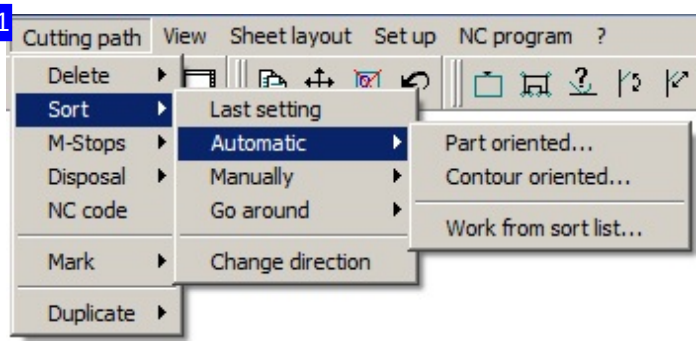
4



Edit - offers many functions for manipulating contour elements and cutting paths.

- Delete - remove single or all elements or cutting paths.
- Sort - choose between: 'Automatic', 'Manual' und 'Avoid'
- 'Machine Stops'
- 'Disposal'
- 'Marking'
- 'Duplicate'

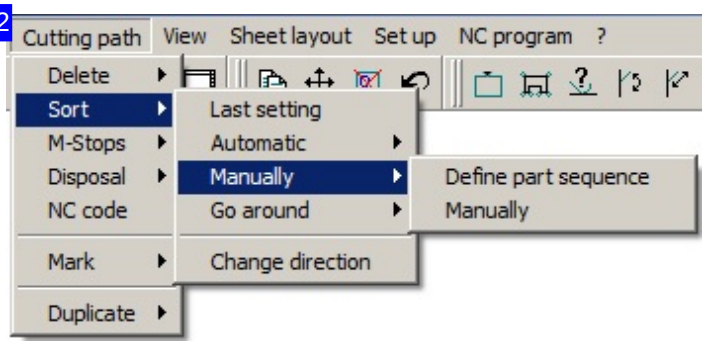
41



Automatic sorting - decide in which order the contours should be cut. More information in the chapter 'Sorting and Combining'.
The menu item 'Last setting' applies the last used sorting settings to save time if you often use similar settings. The function can also be accessed via the icon panel.

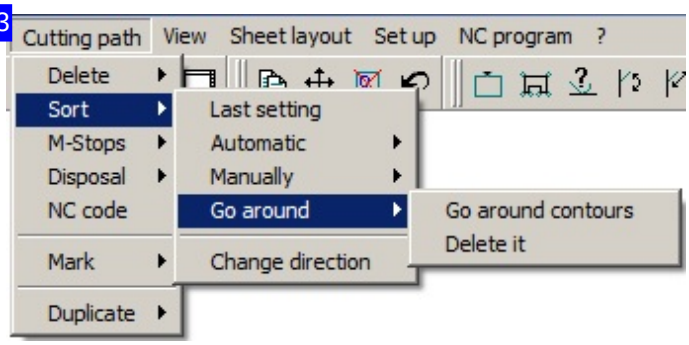
'Change turning direction' allows you to change a contour's cutting direction.

42



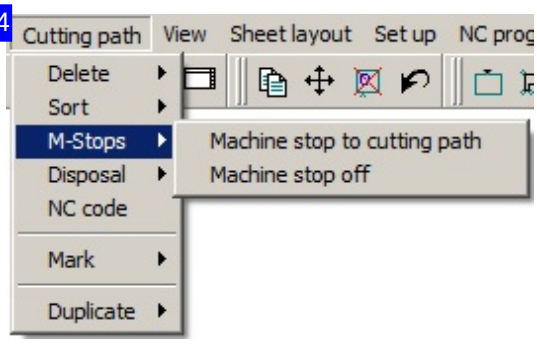
Sort manually - set the cutting sequence manually here. More information in the chapter 'Manually Sort And Avoid Parts'.

43



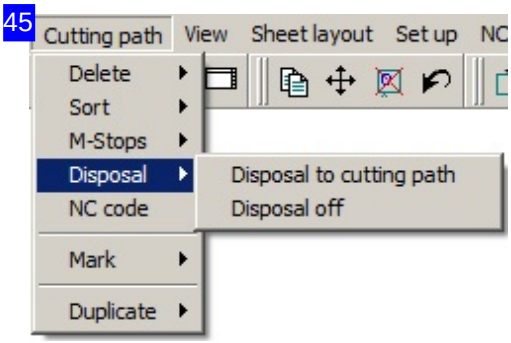
Avoid - This function allows you to avoid parts which are already cut.More information in the chapter 'Manually Sort And Avoid Parts'.

44



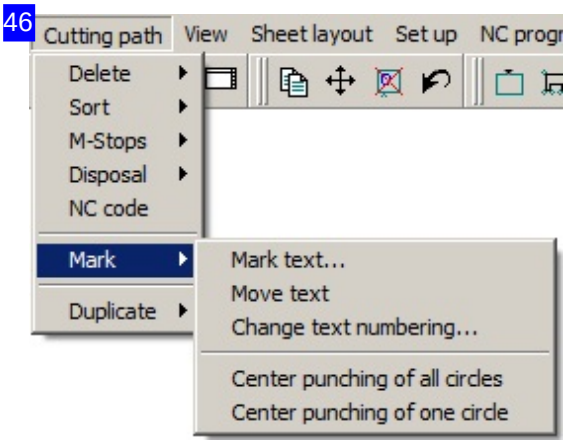
M-Stops - This functions allows you to pause the machine as many times as you require.More information in the chapter 'Changing The Cutting Direction And Adding Machine Stops'.

45



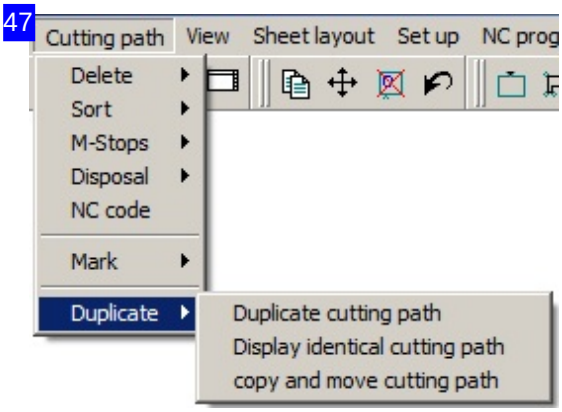
Disposal - Part disposal using chutes.

46



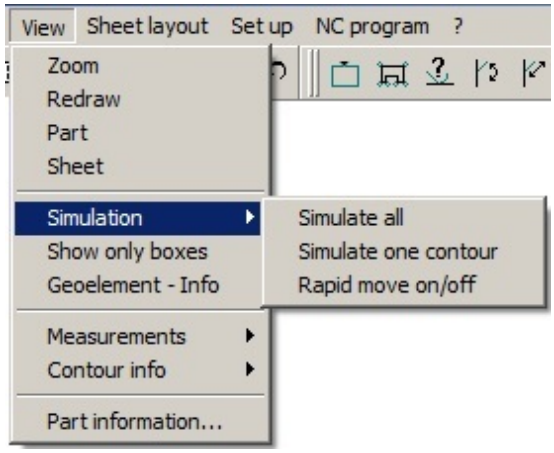
Marking - parts and circles can be marked separately. More information in the chapter 'Marking Parts'.

47



Duplicate - these menu items allow you to manually convert contours to cutting paths, singly or for multiple identical contours. You can also copy and move contours. When moving the axes can be limited by pressing <X> or <Y>. Select the command then the contour. The next steps are prompted in the command line.

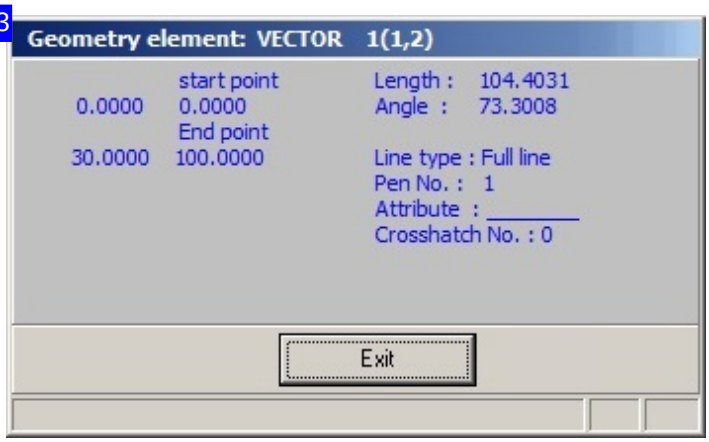
5



Display - settings regarding what's shown on the screen, and which extra information is visible.

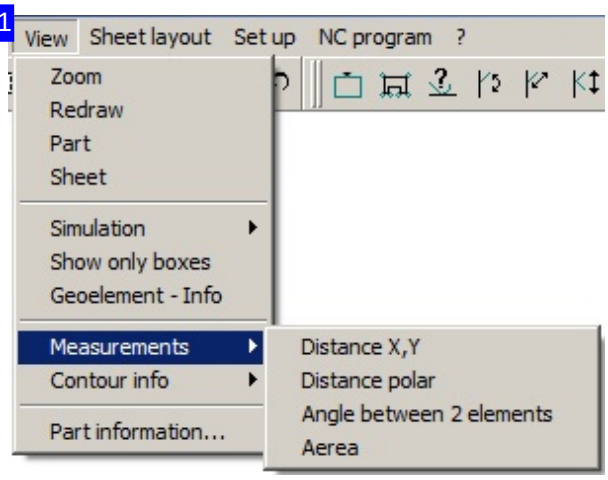
- 'Show only boxes'
- 'Geo. element - Info'
- Submenu 'Measure'
- Submenu 'Contour - Info'
- 'Part information'

53



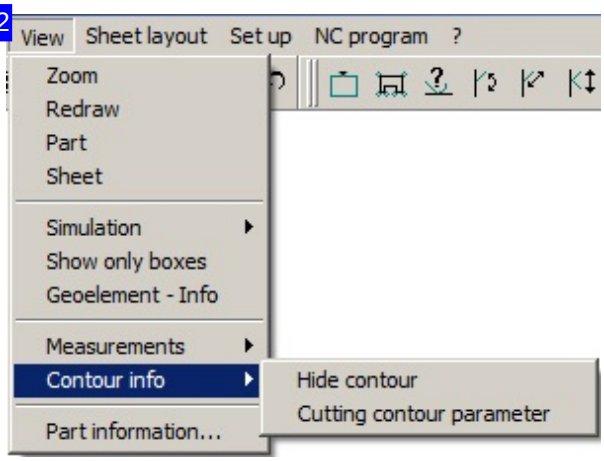
When the 'Geo-Info' function is activated, click an element to see a dialog with its characteristics.

51



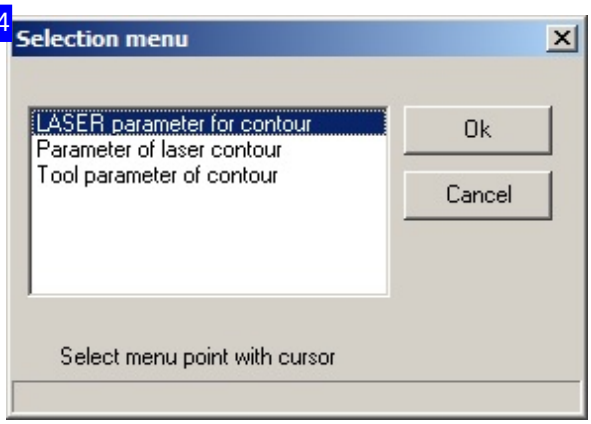
Measure - More information on measuring in the help files for *ncCAD32* .

52



Contour Info - click 'Cutting contour parameter' to open a selection dialog to edit contour attributes. Click 'Hide contour' to identify individual contours which should be hidden when converting to cutting paths.

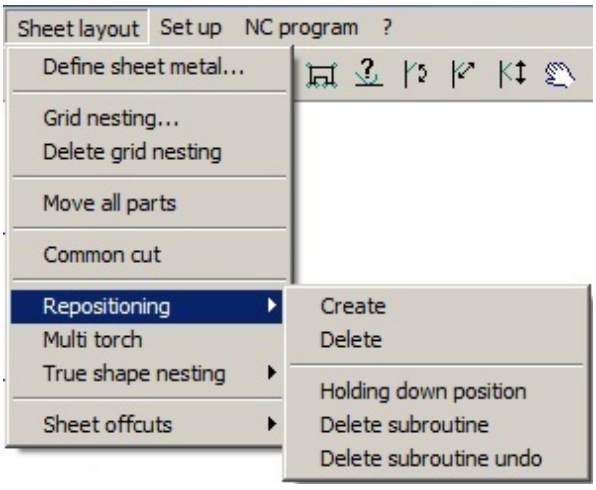
54



This dialog allows you to change the attributes of the selected contour element.

- Contour's Laser Parameters - opens a dialog for more detailed settings.
- Laser Contour's Parameters - adjust speeds here.

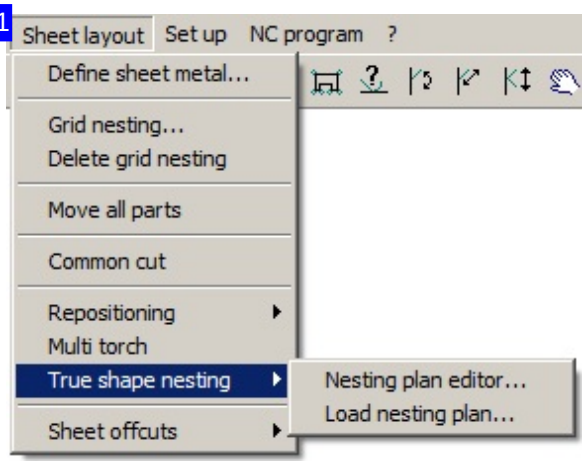
6



Sheet division - this menu contains settings for the size of a virtual sheet and functions for allocating parts to a sheet.

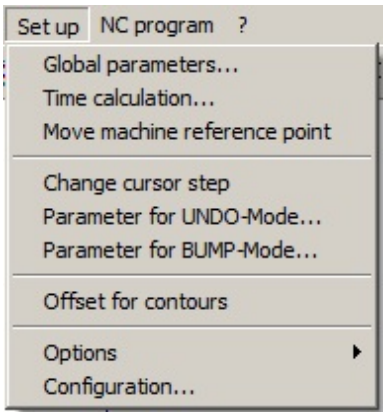
- 'Sheet size'
- 'True-shape nesting'
- 'Multi-torch cutting'

61



True-shape nesting - Call the optional true-shape nesting module. More information in the chapter 'Automatic Nesting'.

7

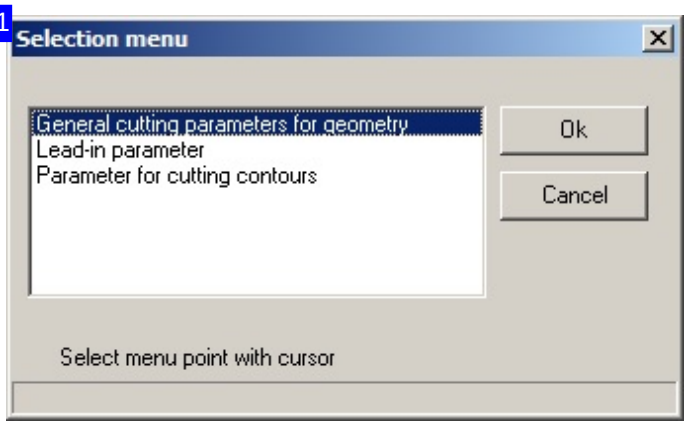


Settings - this menu gives direct access to several dialogs containing settings for various work steps.

- 'Global settings'
- 'Time calculation'
- 'Settings for undo'
- 'Settings for bump mode'

Simply click the mouse to move the machine's zero reference as required.
The menu items 'Options' and 'Configuration' are post-processor dependent.

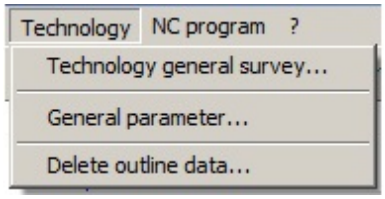
71



This dialog gives access to the various input dialogs for:

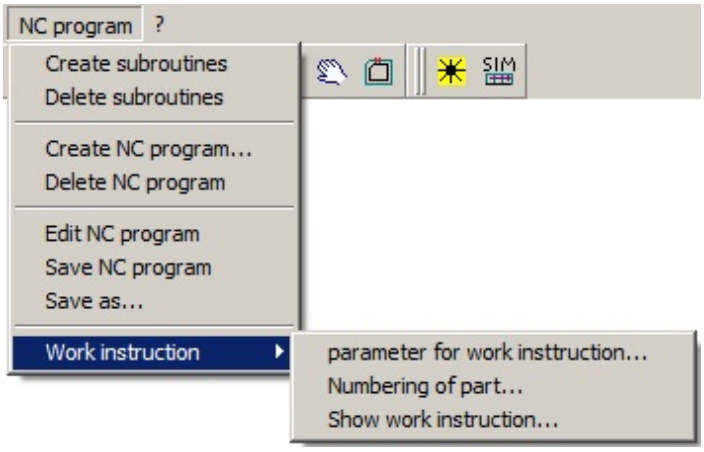
- General cutting parameters
- Lead-in parameters
- Parameters for cutting contours

8



Technologies - This menu is post-processor dependent. More information in the chapter 'Technologies'.

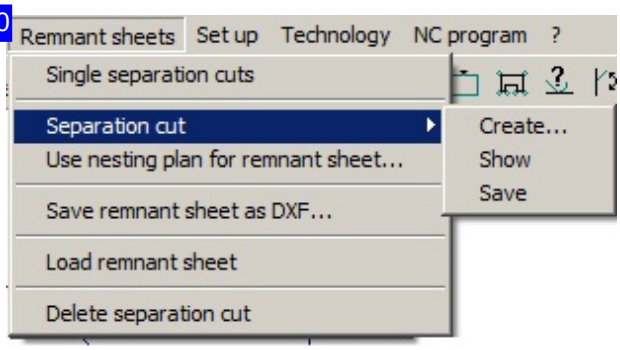
9



NC Edit - all menu items required for creating your NC code are listed here. More information in the chapter 'Convert Cutting Paths To Nc Programs'.

The menu item 'Setup plan' allows you to operate an interface to an optional job management module. You can add customer and part information for a setup plan to help organize your workflow.

10



Remnant sheet - This menu shows you all functions for working with remnant sheets. The menu item 'Single separation cuts' allows you to make all settings for separation cuts in a special dialog.
The menu item 'Separation cut', 'Create...' opens a dialog and allows you setting separation cuts, also automatic separation at existing workpiece outlines. Take care to set the distance between separating cut and pieces cutting lines.

You can use an existing nesting plan as remnant sheet completely. Click on 'Use nesting plan for remnant sheet..'. You have to acknowledge the function in an appearing dialog.

Entering Commands Using The Symbol Panel.

The 'icon bar' allows you to activate commands with a single click on the corresponding icon (quick key).

The 'icon bar' is dynamic and can be configured by the user. This process is as in *ncCAD32* .

The functions within the icon bar can be moved. In this case they are organized as follows:

- Spare element [1].
- Open, save and print files; exit program [2].
- Sheets [3].
- Manipulate contours [4].
- Lead-ins and loops [5].
- Create cutting paths [6].

- Micro bridges and breaks.
- NC code.

For ease of use, the meaning of each icon is displayed as a 'tool tip' when hovering the cursor over it.

1

The spare block to the left is the icon bar's anchor point and cannot be deleted. This is used to access the selection menu to activate or deactivate the various icon groups.

The selection menu is fully customizable - select the 'properties' menu item, exactly as you would in *ncCAD32* .

11

This menu allows you to activate and deactivate individual menu groups.

The selection menu is fully customizable - select the 'properties' menu item, exactly as you would in *ncCAD32* to access an edit dialog for the menu items.

2

This block contains the icons for file manipulation. The items function exactly the same as those in the 'file' menu:

- Load existing cutting path.
- Save cutting path.
- Print cutting path.
- Exit this module and return to *ncCAD32* .

3

The buttons from left to right:

- Automatic nesting
- Show only boxes
- Grid production
- Sheet dimensions

4

The first three buttons allow you to manipulate your contours. The items function exactly the same as those in the 'edit'-'manipulate'. menu:

- copy
- move
- delete

The right hand button means undo the previous action.

5

These buttons mean:

- Create micro-bridges.
- Rounding corners
- Create lead-ins
- Edit lead-in
- Extend lead-in
- Move lead-in
- Last setting - apply the previously made sorting settings for the processing sequence.
- Delete all processing paths. This command deletes all cutting paths. Note: this command cannot be undone!

6

These keys create the cutting paths and allow you to check the processing sequence in a simulation.

7

Supplementary functions:

- Small part slides
- Machine stop
- Place micro bridge
- Copy micro bridge multiple times

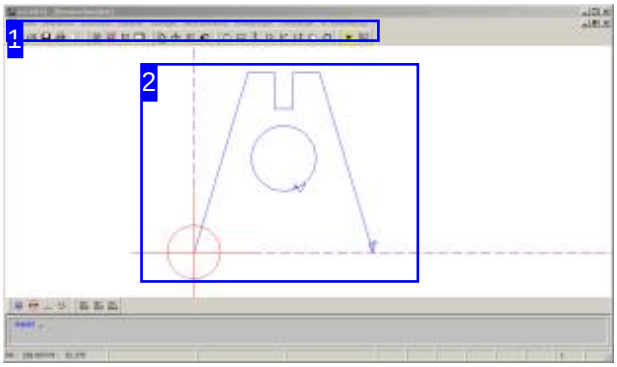
8

These buttons manipulate NC code:

- Create
- Edit
- Save

Cutting Paths

Creating Cutting Paths.



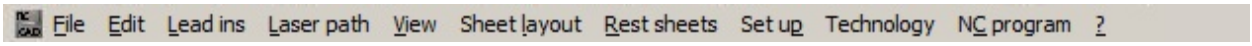
Cutting paths are groups of individual contours which, together, describe a cutting part. By setting cutting directions and adding lead-ins and loops you can define the individual cutting path.
An imaginary box which can also be shown in the drawing is overlaid on the part for editing and checking of areas of validity.
For some jobs it is useful to show only these boxes, as this can seriously reduce computing and redraw time.
The first thing required when making a part is a suitable sheet. If the sheet is not an existing one defined in the system then you can define a virtual sheet to get started with. The virtual sheet allows you set material and thickness.

A component which has been transferred from *ncCAD32* to the cutting module must be converted to cutting paths. Select 'create cutting paths' from the 'edit' menu to put your component in the correct format. Insure that the contours have first been properly sorted in *ncCAD32* so that automatic conversion can be used successfully.
Conversion can also be started from the icon bar.

Lead-in and out elements (lead-ins) will be added when converting, using your global settings as previously defined in the lead-in manager. If default lead-ins are defined then conversion takes place completely automatically. Current settings can be viewed in the 'lead-in settings' dialog.

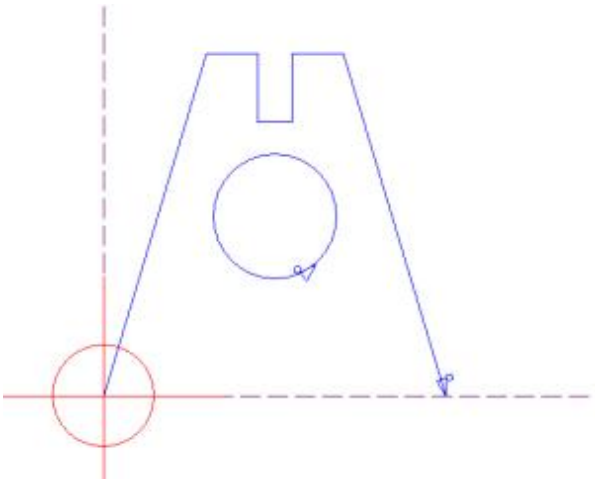
You can define further contour management parameters in the 'contour management settings' dialog. Cutting speeds are controlled in a submenu of this dialog.

Sheet settings can be accessed via the menu 'sheet division', 'sheet dimensions'; a dialog opens for entering the sheet information.

1 

The 'edit' menu item allows you to edit and create new cutting paths. You can also create a cutting path from the icon bar or using command '430'.

2

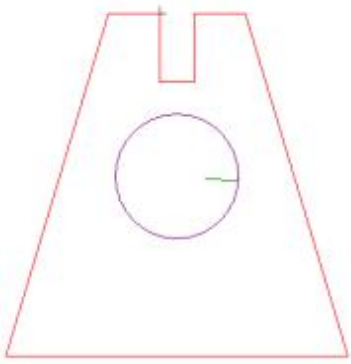


A component transferred to the cutting module from *ncCAD32* showing the cutting directions and sorted into internal and external contours. In order to edit the part in *ncSchneid* , it must first be converted into cutting paths. Select the corresponding menu item to begin automatic conversion.

When converting manually you can 'create single contours' by clicking contour individually or selecting contour groups by defining a selection area with a rectangular or polygonal selection box.

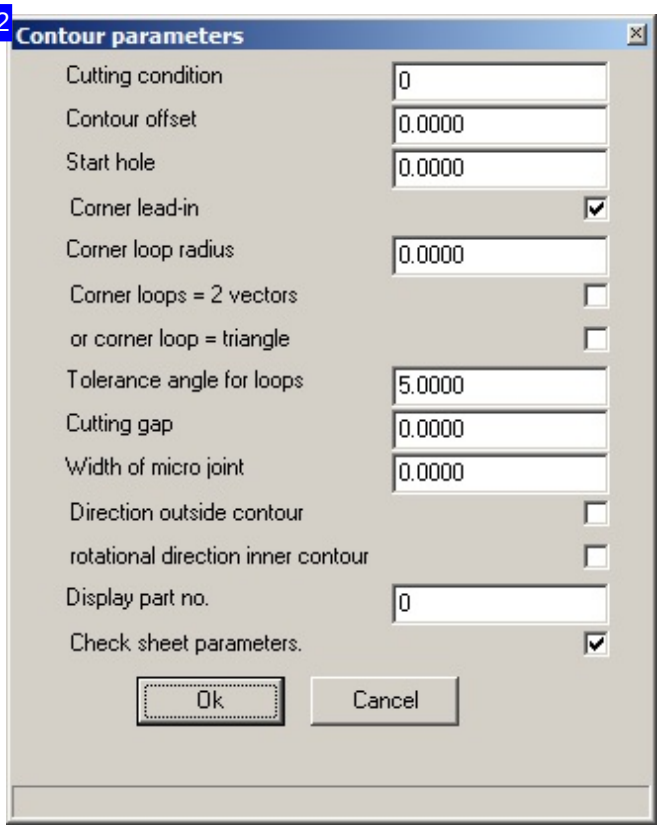
If a contour's cutting direction is not as you want it, then you can change it using the menu item 'geometric contours'. This must be done before converting to cutting paths!

21



After conversion you will see your part as cutting paths with lead-ins.

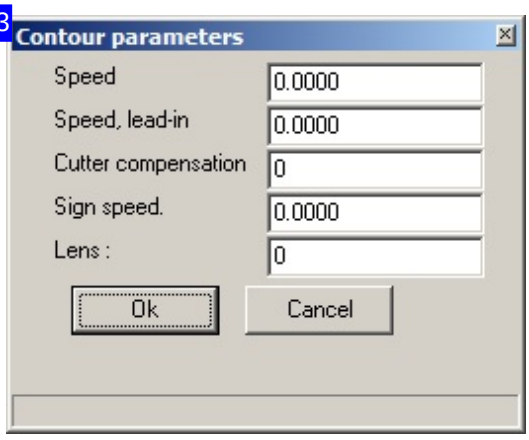
12



This dialog is accessed via menu point 'display', 'contour - info'.

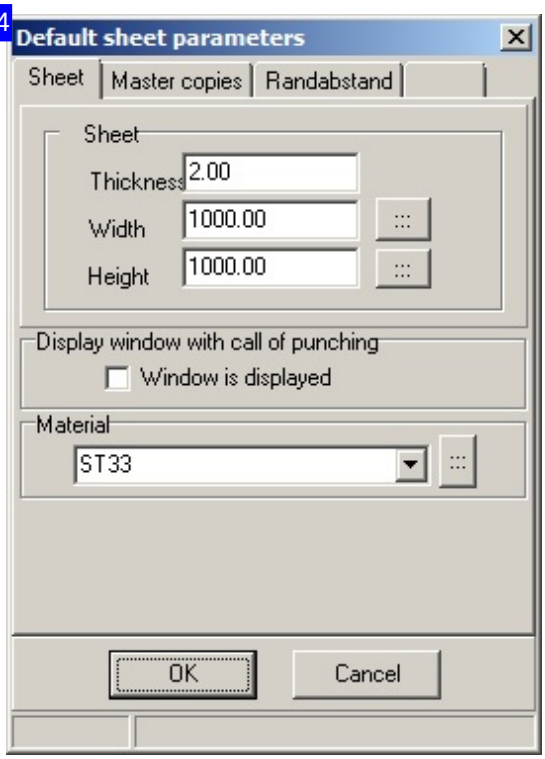
- Cutting conditions: 1 - values from the database.
- Contour offset: offset of the cutting path from the drawn contour.
- Piercing: > 0 - drill diameter; pre-piercing a lead-in.
- Corner lead-in: marked - lead-ins only at corners.
- Corner loop radius: arc radius.
- Corner loops: marked when a) bisected or b) triangulated.
- Tolerance angle: angle deviance for a corner before a loop is required.
- Cutting width: with of the cutting beam for shared separating cuts.
- Micro bridge width: enter the value for micro bridges.
- Conversion: mark the checkbox to change the turning direction of external and/or internal contours.
- Show part number: 1 - show
- Sheet parameters: marked - sheet parameters are loaded with your sheet.

13



This dialog is accessed via menu point 'display', 'contour - info'. Speeds are entered in mm/min, where -1 means none.
Kerf speeds are entered in 1/10mm.

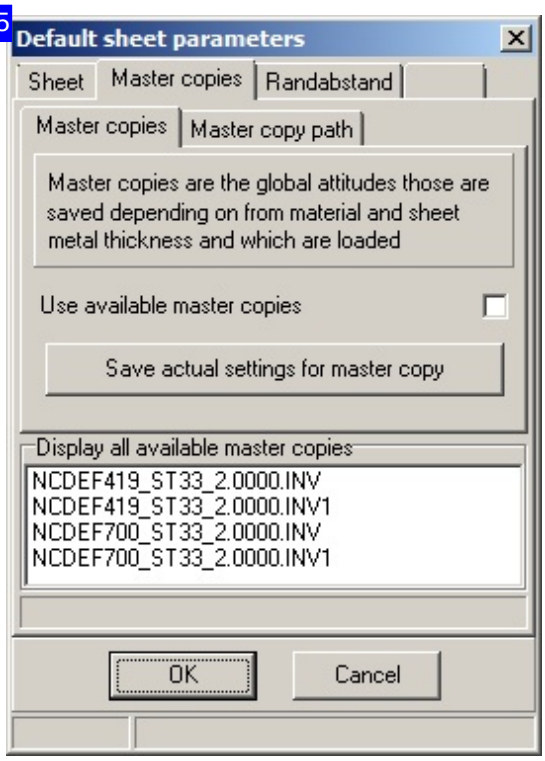
14



This dialog allows you to define a sheet for continued editing.
Click the button to the right of the 'material' field to access a dialog containing your known materials.

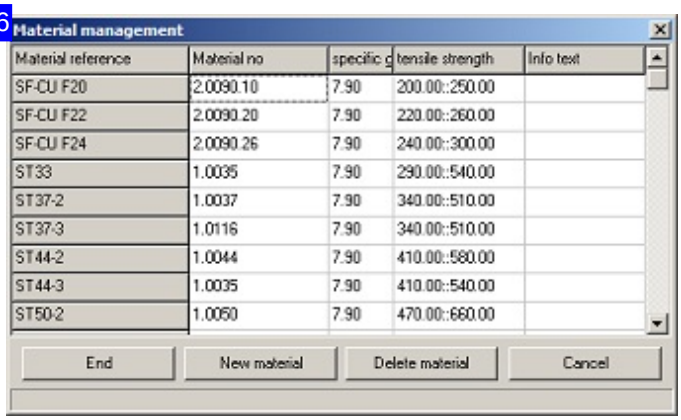
The 'templates' tab allows you to save global machine settings corresponding to sheet material and thickness to a predefined path.

15



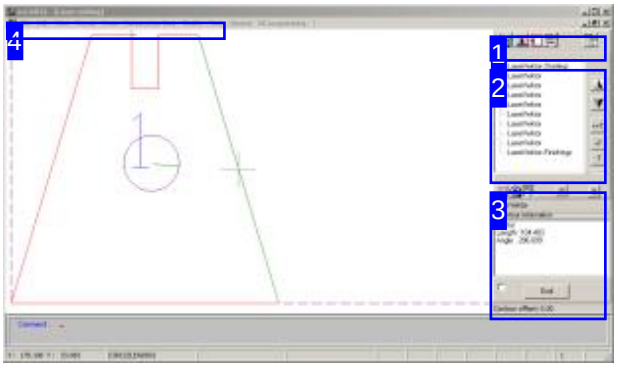
Settings for sheet material and thickness can be saved as templates. Set the file location and type under the tab 'template path' (.INV and .INV1).

16



This material information is important for some cutting machines and must be considered by the post-processor. The material manager allows you to save as many materials as you need and enter all values your post-processor(s) require.

Editing A Cutting Path's Elements.



The frame to the right lists all elements of a cutting path. Every single element can be selected and edited individually. Right-click an element for a context menu with edit commands.

The menu items can largely also be found through the menu panel. The menu is context sensitive depending on the menu selection in middle button panel.

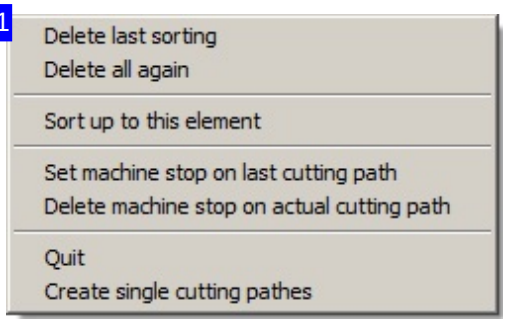
The individual sections of cutting paths can be edited to fit your requirements. You have an overview of all individual elements and can access and edit every one.

1

These quick-keys allow access to more commonly used commands. The button's function from left to right:

- Sort cutting paths manually. You select the start element and the subsequent elements in sequence. Run the cursor over an element and the direction will be shown with an arrow. Fix the direction in this way, and click to confirm. To leave the function press <ESC>, <Q> or right-click to access a menu.
- Join tangential elements. All elements with a tangential transition will be combined as one element. If your contour has lots of small elements, this is useful to reduce processing time and data quantities. When allocating parts for common separating cuts straight cutting paths which run parallel will be joined to make one path.
- Change to another contour. This allows you to switch to another cutting path without leaving the current function.
- Start the simulation. Once all elements are sorted you'll be able to check the cutting sequence. A dialog will appear for controlling the simulation.
- Undo.
- Enter parameters. A dialog will open to enter values for editing elements..

11

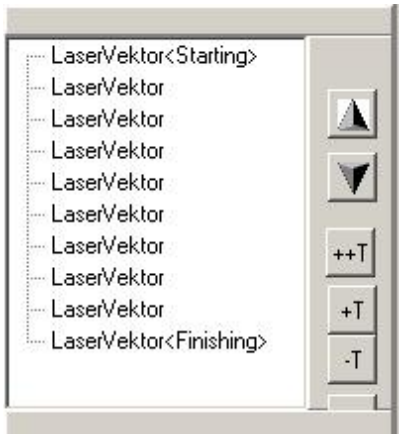


You can undo your cutting path element sorting in the first section.
The penultimate section allows you to add or remove machine stops.
The penultimate item exits sorting.

12

When starting the simulation this dialog will appear for you to set the virtual cutting speed. Use the slide bar for speed, and mark the check box to show a virtual cutting tool. The right-hand button exits the simulation.

2

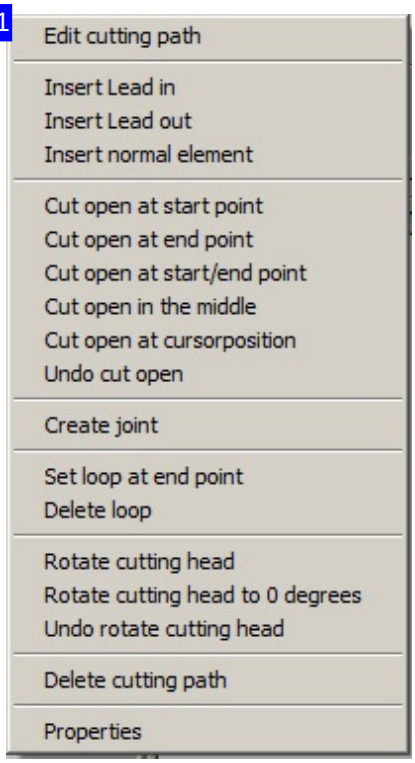


This list shows all contour elements. The arrow button to the right allow for easy navigation.

The buttons below allow you to highlight or add 'T'ags to the elements, which you can then treat as element groups. The tags can be added to or removed from one or all contours. When you select a function from the selection menu it will be applied to all tagged elements.

The selection menu is dependent on the selected menu form in the quick selection in the middle of the window. If the selection is changed a further selection menu will open.

21

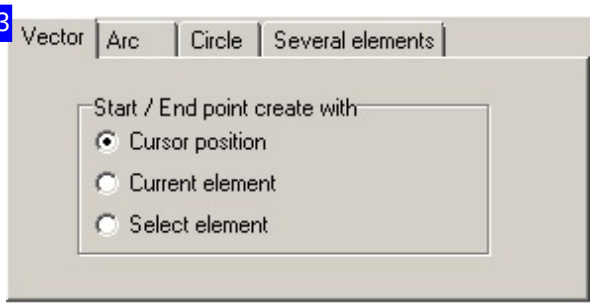


This menu allows access to the cutting path manipulation and expansion functions. The functions affect the marked element. The 'separate' block allows you to divide elements in order to turn the cutting head. This is important when beveling.

Click 'Insert normal element' to open a dialog for adding one or more elements to your cutting path.

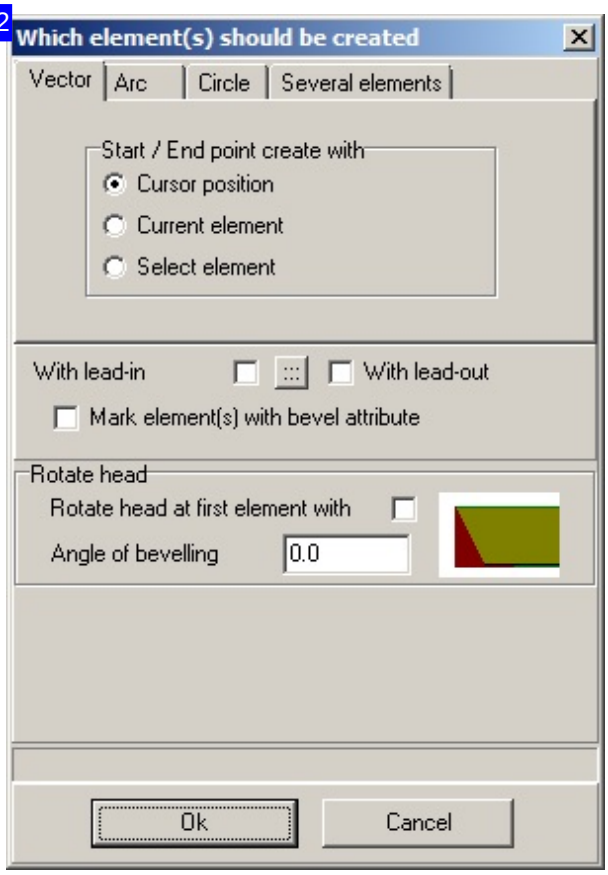
The last menu item 'Properties' opens a dialog to edit a cutting path's attributes.

23



Set how vectors should be created here. Mark the corresponding selection.

22



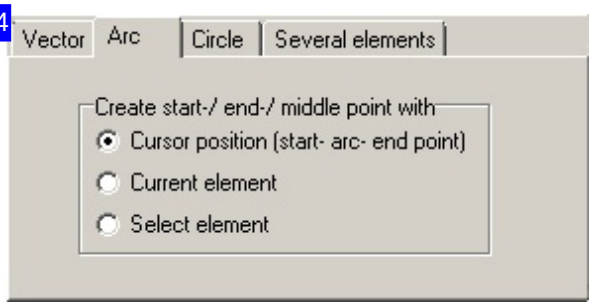
Select the element's corresponding tab at the top:

- Vector
- Arc
- Circle
- Multiple elements - with center point setting.

The second section allows you to define lead-ins elements; mark the checkbox and adjust the parameters using the button in the middle if required.
The 'Turn head' pane allows you to set up values for beveling. For multiple elements use the 'Offset' pane to set the behavior at transitions. You can use the 'Offset over bevel' button to set up the values for the bevel.

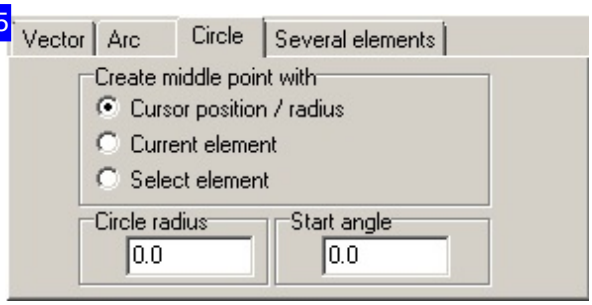
More details can be found in the chapter 'Adding Beveled Edges to Cutting Paths'.

24



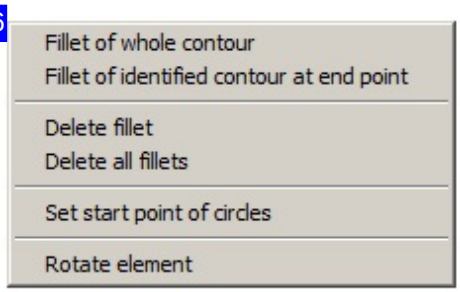
Set how arcs should be created here. Mark the corresponding selection.

25



When creating circles you can define the radius and start angle and creation type. Mark the corresponding selection.

26



This menu is called by right-clicking an element in the selection list [2], where the menu selection for this display is activated in the middle button panel [3].

3

The button panel above allows you to make display settings. The buttons' functions from left to right:

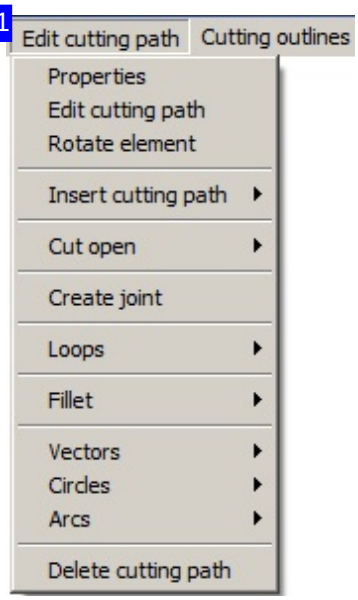
- Show only the selected cutting contour.
- Show cutting paths with directional arrows.
- Toggle the parameter window .
- Change the menu for the list selection [2].
- Mark the cutting contour as closed.

4

This screen has several menu panels:

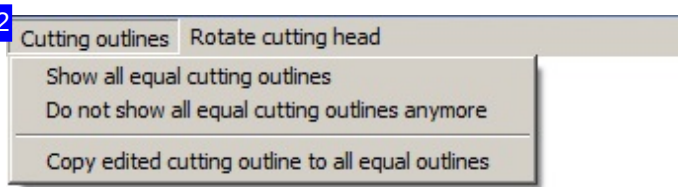
- Edit cutting path
- Cutting contour
- Turn cutting head

41



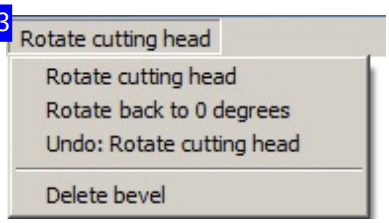
This menu gives access to the functions for manipulating and expanding cutting paths. The commands can largely also be accessed through the selection list [2] in the window to the right.

42



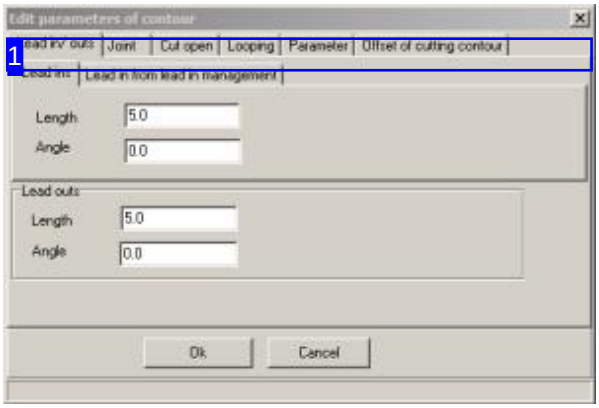
This menu allows navigation of identical contours.

43



Turn cutting head - this menu allows you to tilt the cutting head for beveling, if your machine supports this functionality. More details can be found in the chapter 'Adding Beveled Edges to Cutting Paths'.

Settings for Cutting Paths.



This dialog allows you to set the parameters for each individual cutting path edit function.

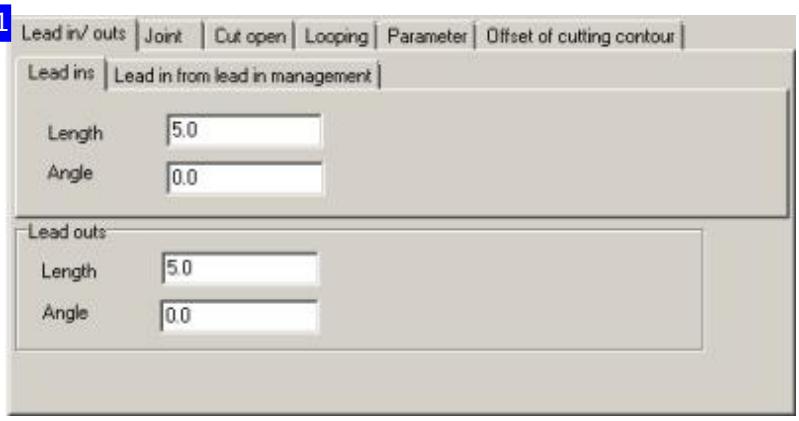
The tabs allow you to select parameter groups to easily access the required settings:

- Lead-in and out elements
- Bridge
- Separate
- Loops
- Settings
- Offset cutting contour

1

Select the tab corresponding to the function whose settings you wish to change.

11

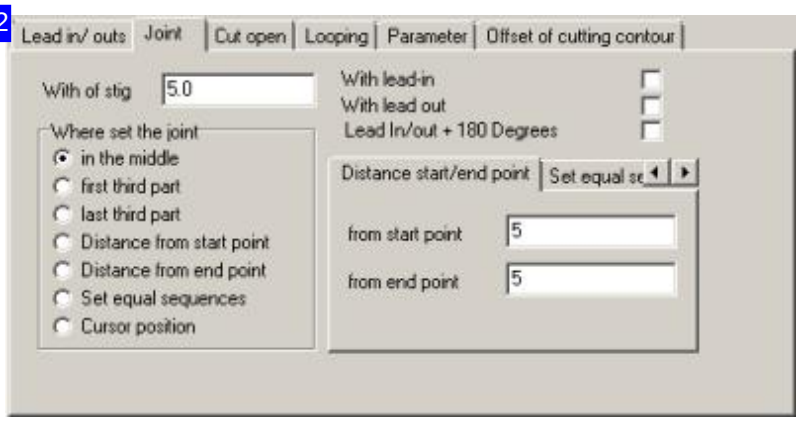


Lead-in and out elements - set the lengths of lead-ins and outs.
The second tab allows you to apply lead-ins from the lead-in manager.

18

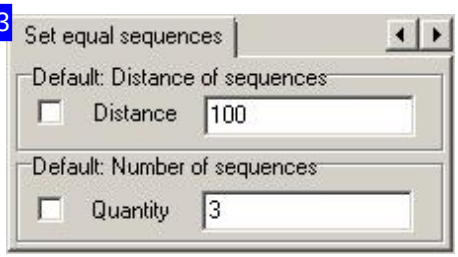
Lead-in from the manager - to reuse previously defined lead-ins from the lead-in manager , mark the checkbox and select a lead-in in the next screen.

12



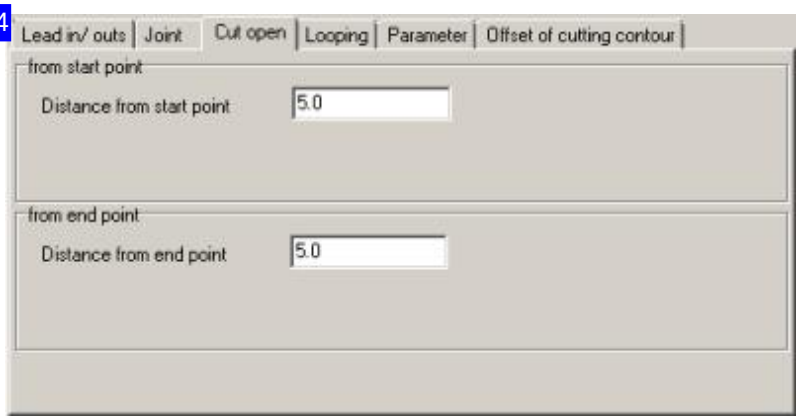
Bridge - this frame allows you to define and place a bridge. Enter the width and mark the checkbox for lead-ins. To place multiple bridges on a contour, you can divide it.

13



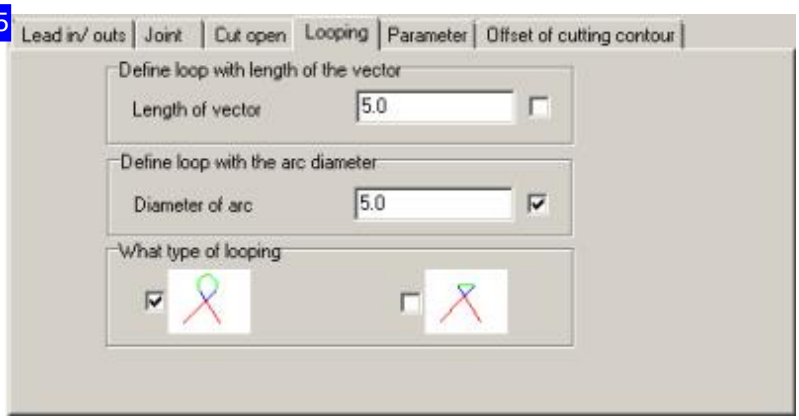
Divide - if you want to use multiple cutting bridges on a single part then enter the offset between, or the quantity of bridges here in this tab.

14



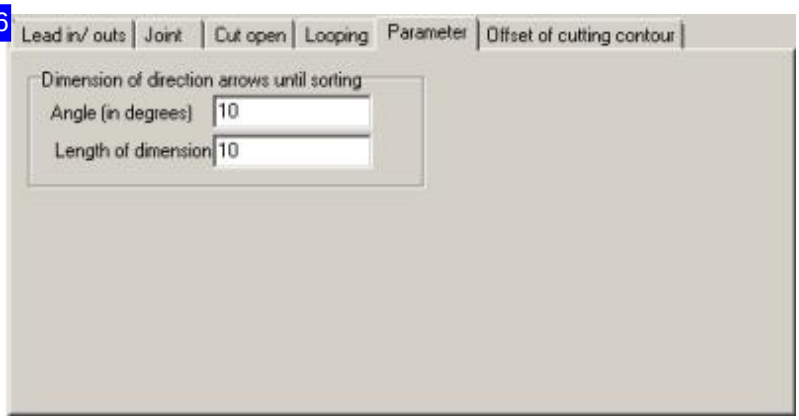
Separate - if you wish to separate a cutting contour at the beginning or end of an element to create the clearance for necessary machine functions such as turning the head, enter the offset here.

15



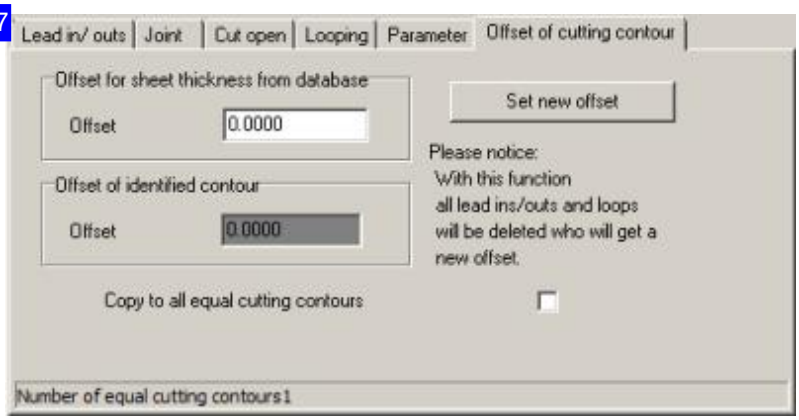
Loops - a contour's corners can be looped; more on this in the chapter entitled 'rounding and looping'.

16



Settings - enter the dimensioning values for the directional arrow when sorting manually.

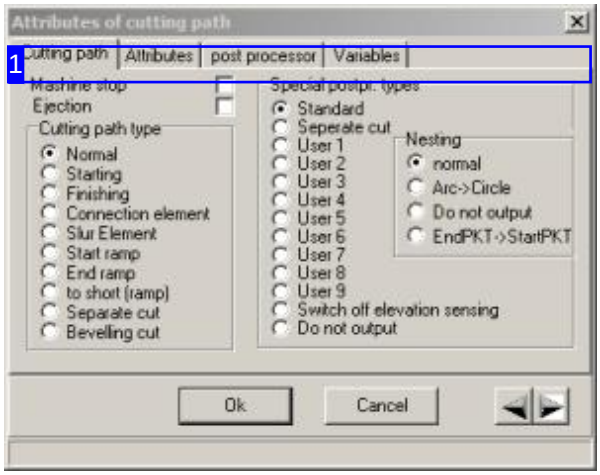
17



Offset cutting contour - you can put any offset on a cutting contour, or load this data from the sheet database according to sheet thickness.

This function allows you to define the offset when using common separating cuts, allocating an offset of half the cutting beam width, thus creating a common cut.

Edit Cutting Path Attributes.



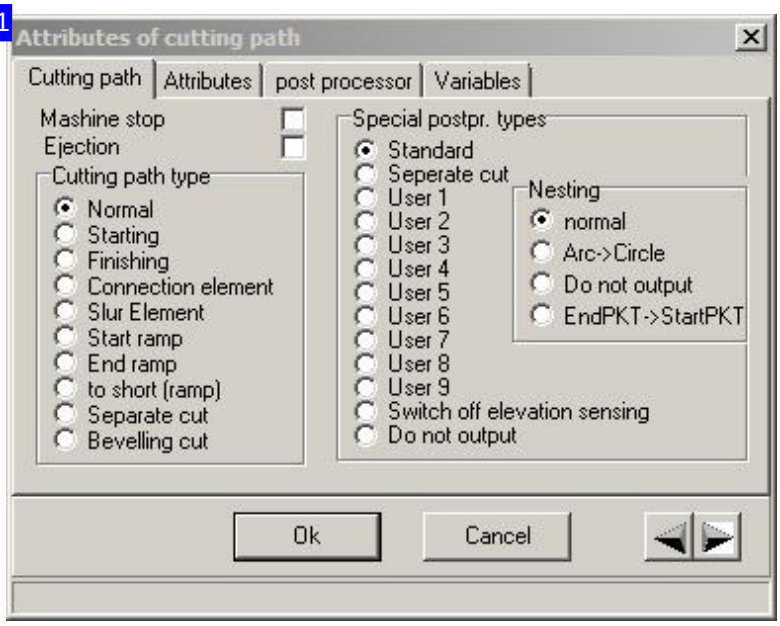
Use the arrow buttons at the bottom right to navigate the contour elements in the main window.

The cutting path type is set by the program and defines the processing type. This dialog allows you to adjust the attributes for each individual element. These settings are specific to and handled by your post-processor!

Select the tab corresponding to the settings you'd like to change:

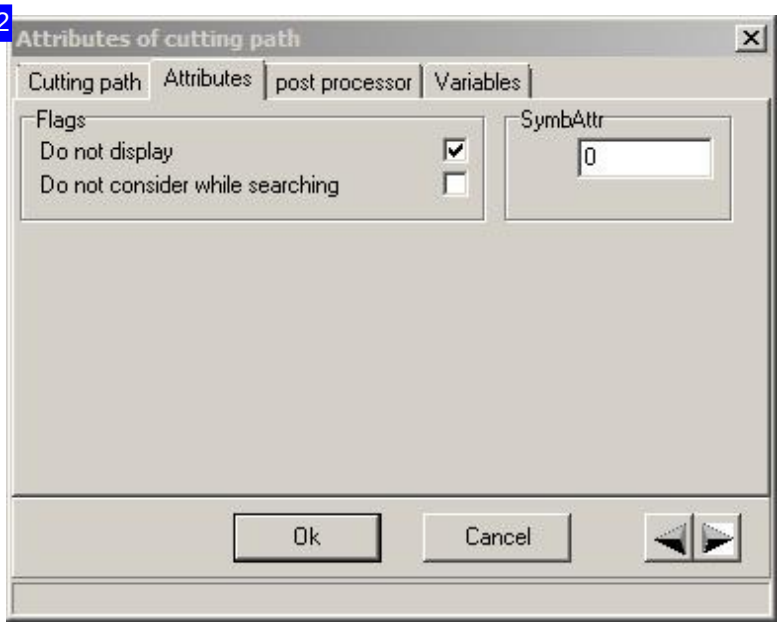
- Cutting paths
- Attributes
- Post-processor
- Variables

11



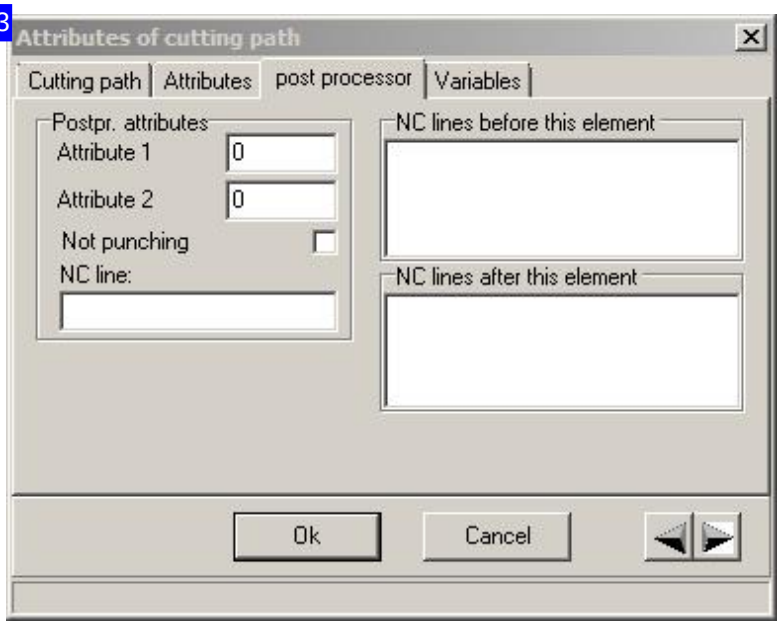
Cutting paths - mark the checkboxes to add machine stops or ejections. You can select from the offered cutting path types here; mark the standard checkbox or the relevant one offered from your post-processor.
The 'nesting' frame allows you to make settings to enable nesting of open contours which are not nested.

12



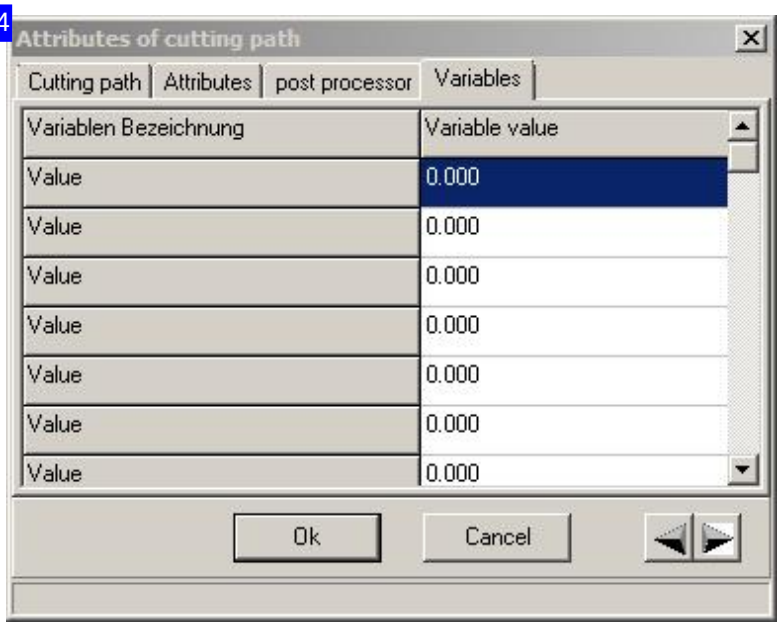
Attributes - use the checkboxes in this tab to decide if flags should be displayed or not, and whether these should be considered when searching.

13



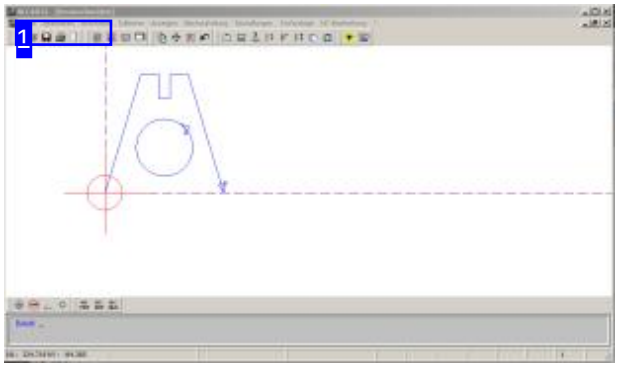
Post-processor - this tab allows you to enter code lines and attributes which will be processed in accordance with the post-processor. Entries and variables in these fields must meet the post-processor's requirements.

14



Variables - as well as the post-processor's code settings you can also define various post-processor variables here. Entries must be in the post-processor's expected syntax!

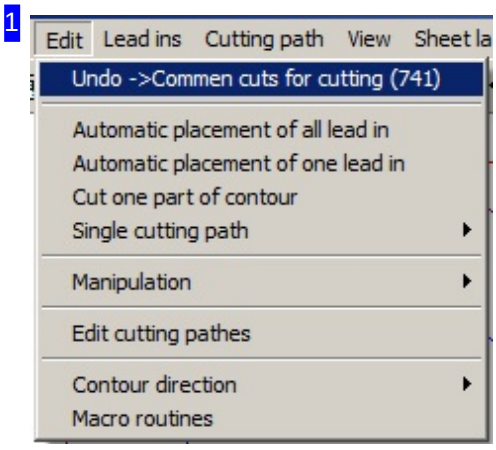
Undo Your Last Command.



This function can be accessed via the icon bar or the menu panel. Settings for the undo function can be made under 'Settings'. Clicking undo in the menu opens a dialog with various undoing options.

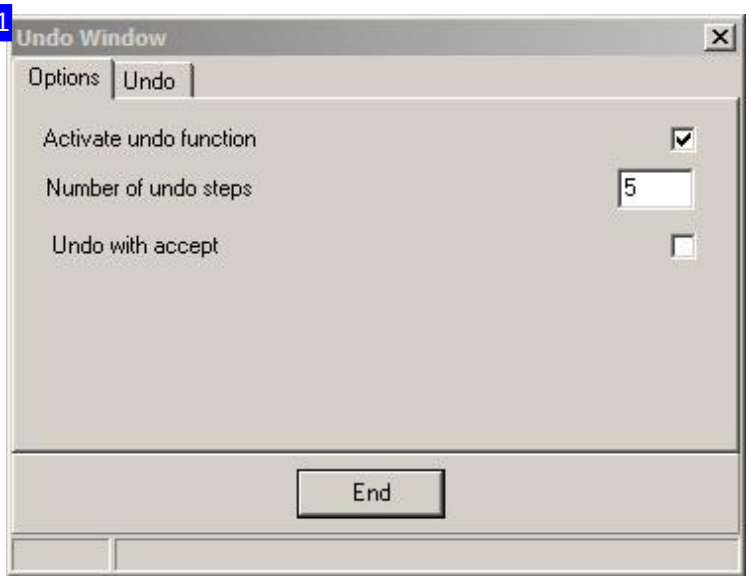
Undo - an important and helpful cutting path edit function.

All executed functions are stacked in the order they were executed, and by clicking 'Undo' you can reverse the executed sequence one step at a time. The <U> key allows you to reorganize undo levels in the stack and use them as restore points in case of error.



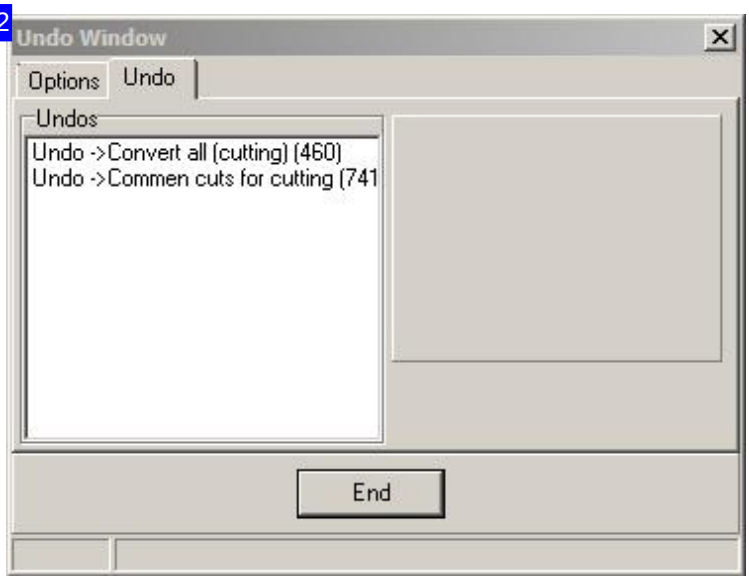
The 'Edit' menu contains the undo function at the top. Click here to undo the previous command. The corresponding command is shown in the menu to help you keep track.

11



This dialog is accessed through the 'Settings' menu.
Enter the number of commands to save as undoable steps; how many commands to stack. Check relevant selection boxes. The 'Undo' tab allows you to view the current stack.

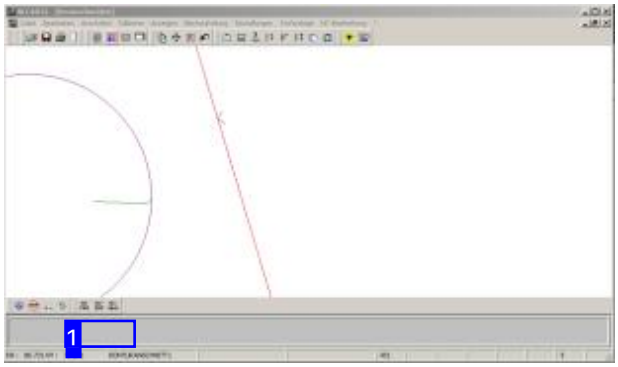
12



The list to the left shows the stack contents, the preview window to the right the status at the point that command was executed. Click each command to see which status you would jump back to.

Technologies

Define Your Machine's Technology Data.



Technology features are always dependent on the features of your cutting machine. Therefore, to be able to create useable NC codes requires a suitable post-processor. Post-processors are to cutting machines as printer drivers are to your printer, so to enable use of *ncSchneid* with your machine, you will need the corresponding post-processor!

Technology features can be enabled depending on your post-processor and cutting machine. The corresponding data may then need to be adapted to suit your machine.
The post-processor is selected in *ncCAD32* .

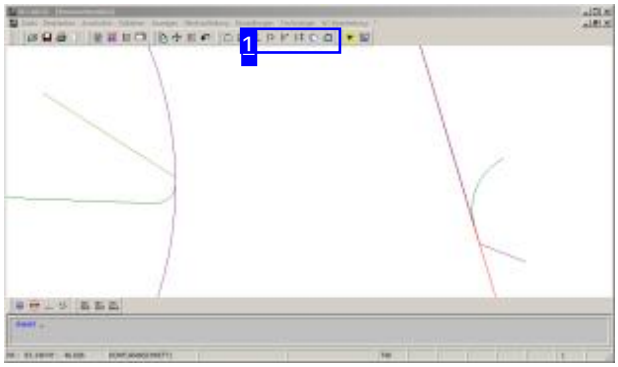
You can find more information on the technology settings in the following pages:

- Lead-ins
- Loops / rounding
- Bevels
- Attributes
- Edit cutting paths
- Marking, punching

1

To correctly make use of your cutting machine's technology features you will need a post-processor which can generate NC code your machine will understand. Insure you have installed and selected the correct post-processor in *ncCAD32* .

Edit Lead-Ins.



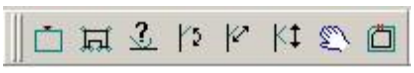
Technology cuts - lead-ins and lead-outs when cutting your parts, more generally referred to in this program as lead-ins - are allocated to your components automatically by the software. You can check the current values under settings.

The lead-in manager allows you to make global default settings about your lead-ins which will then be used when generating automatic lead-ins.
Lead-ins once created can still be edited and moved however you require. This screen offers all the possibilities of lead-in manipulation.

Lead-ins can also be placed globally on one side of a cutting path - select the corresponding point in the menu panel.

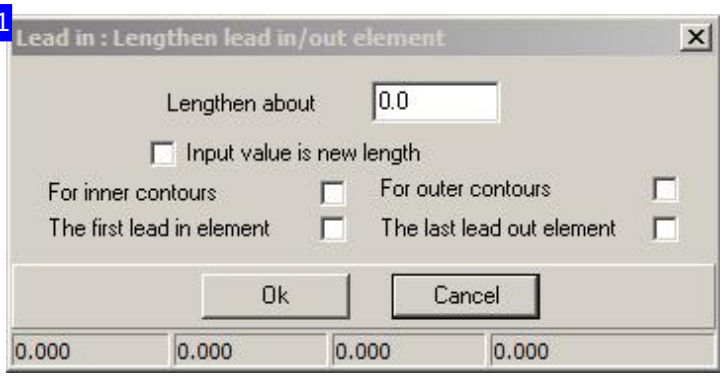
To manipulate lead-ins, select the menu item 'lead-ins', 'edit lead-ins'. Select the required function from the menu:

- Move lead-in: identify a lead-in and drag it wit the cursor to a new position on your cutting path.
- Extend lead-in: a dialog, opens to enter values for lengthening.
- Add to lead-in: identify the required lead-in and add as many elements as required in a polygon line group. Exit the function with <ESC>.
- Overlap: if you wish your lead-ins to overlap for a smooth join. Identify the required cut and drag the lead-out to the required length.
  Overlaps can also be placed automatically - a dialog will open for the corresponding settings.
- Copy: whole lead-ins are copied and can be placed at another point on the cutting path, to avoid overly long cutting paths.
- Edit: this function allows you to make general changes to a cutting path. Identify the required element and lengthen or shorten the cutting path using the mouse. Press <P> to change the angle.

1 

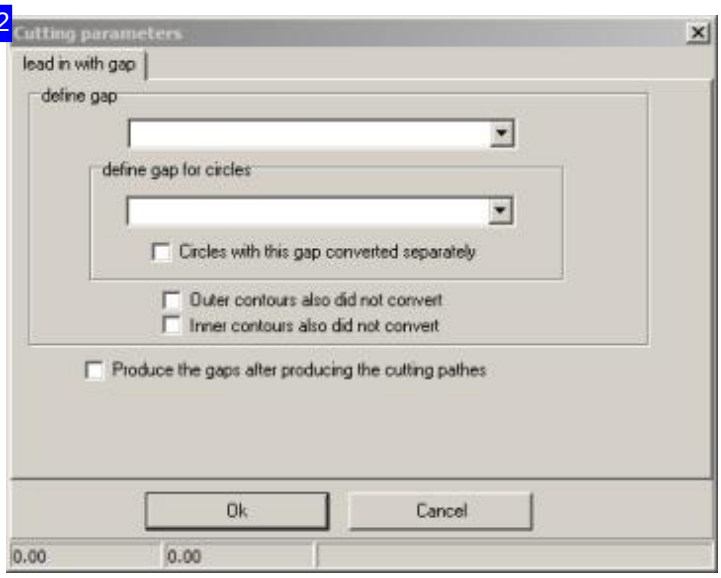
The most important lead-in manipulation functions are available as quick-keys in the icon bar. The same commands can also be accessed from menu panel.

11



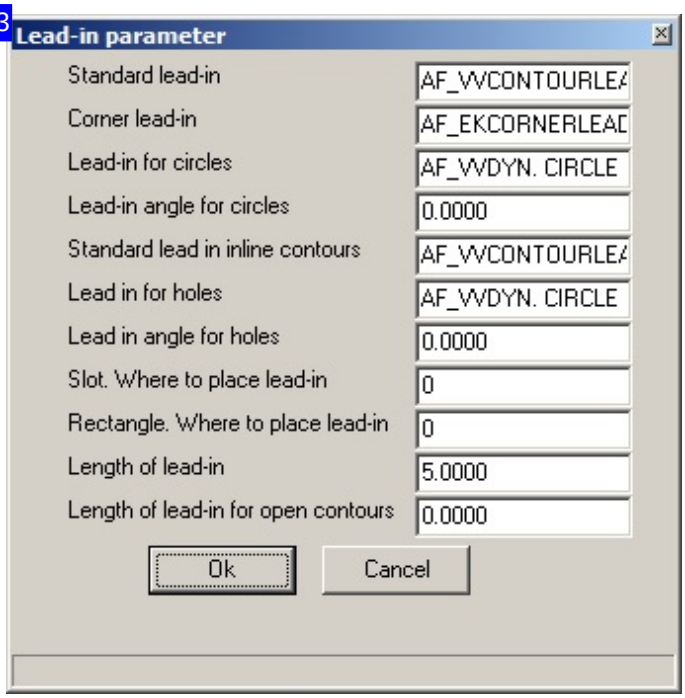
This dialog allows you to define the parameters of the extension of a contour's lead-in. Select the corresponding checkboxes> Note that the settings for all contours will be used for automatic lead-ins.

12



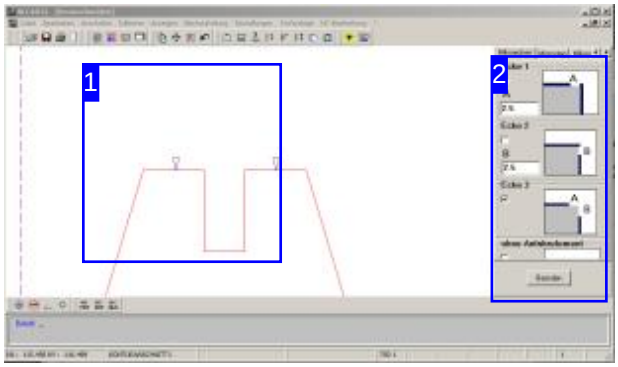
This dialog contains the cutting parameters for an overlap. The overlap gap can be entered as a concrete value or dependent on sheet thickness. The value can be prefixed with (+/-); thus you can create gaps as well as overlaps.

13



You can access this dialog through the context dialog under the menu point 'Settings', 'Global Settings', 'Lead-in Settings'.
This dialog shows the general lead-in parameters. The values can be modified directly in this window. Hover the cursor over a field to receive a tool tip giving the parameter's meaning, and in some cases the valid codes you can use.

Cut Parts With Micro-Bridges.

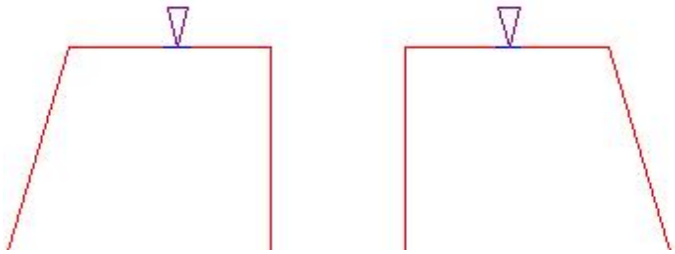


In order to be able to use micro-bridges, the bridge width must be known. You can define the width of manually placed micro-bridges [1] in the global defaults window.
To place micro-bridges, select the function in the menu panel or the corresponding quick-key in the icon bar.

Select 'Micro-bridge with lead-in' to switch to a screen with a tab [2] for lead-in definition.

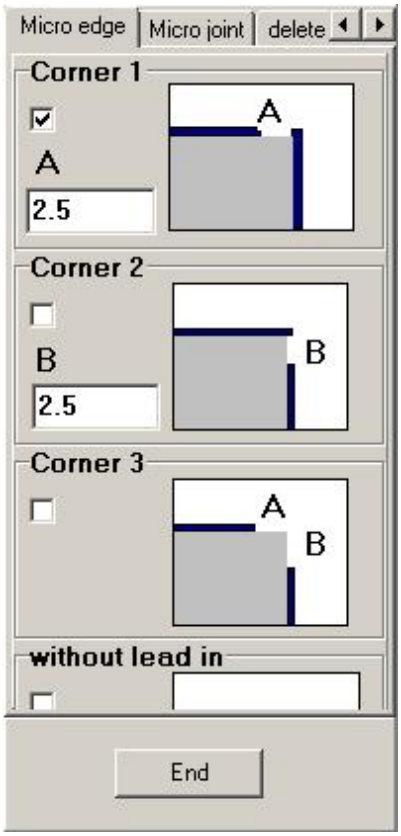
If cut parts should remain 'webbed' in the sheet and not be allowed to drop, you can add micro-bridges (cutting bridges) to hold the part in position.

1



To add micro-bridges, select the function and then the required contour element.
The micro-bridges can be moved around an element as required using the 'edit' function.
Select either one or all to delete micro-bridges, and identify as required.

2

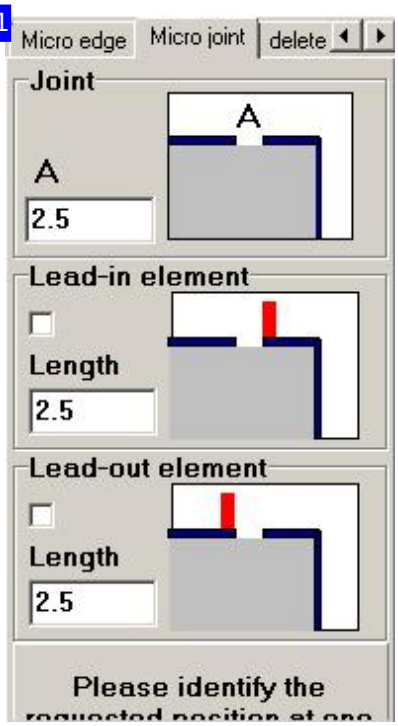


In the tab panel you can:

- Define micro-corners
- Define micro-bridges
- Delete one micro-bridge
- Delete all micro-bridges
- Define lead-ins

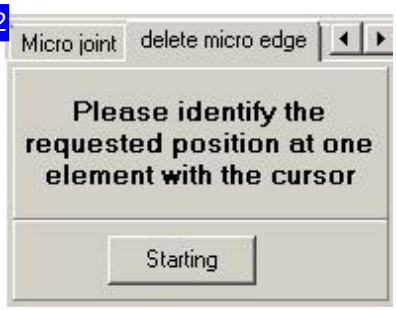
Enter the micro-corner lengths in the input fields and select the corresponding checkbox. The bottom pane selects the lead-in requirement.

21



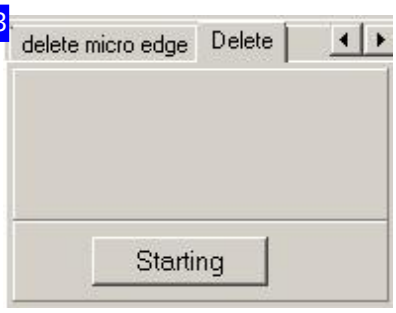
This tab is for defining micro-bridges. Enter the lengths of micro-bridges and their lead-ins in the input fields. Select the corresponding checkbox for the lead-ins and lead-outs requirement.

22



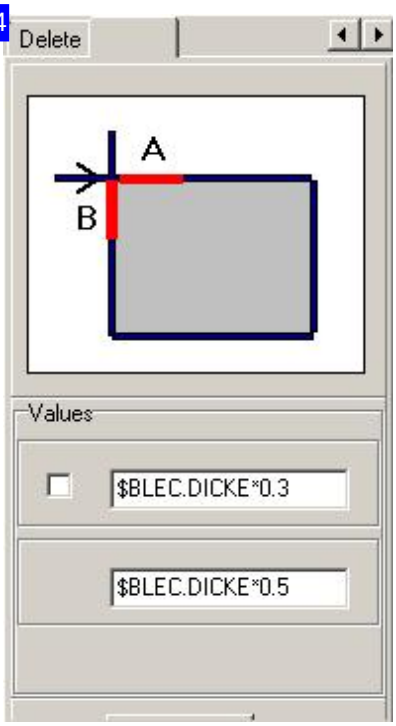
You can delete a specified micro-bridge here. Use the cursor to identify.

23



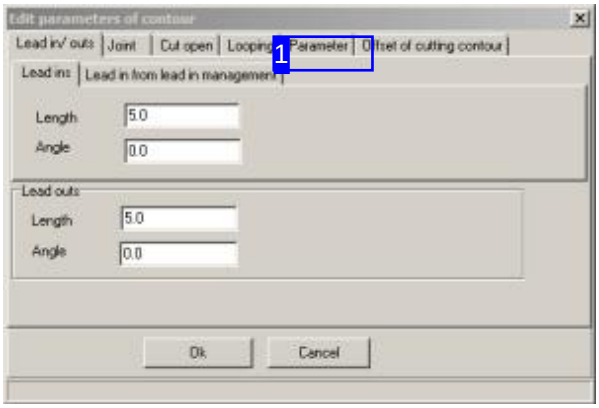
Click on 'start' in this tab to display all micro-bridges in a list. You can then select and delete any item from the list.

24



The lead-in lengths can be entered as concrete values, or formulae dependent on the sheet thickness or other values.

Roundings and Loops.



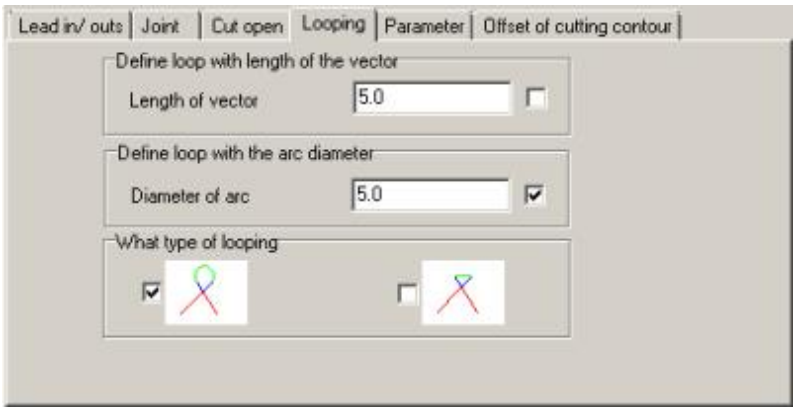
When forming a component's corners you can add loops or roundings to its cutting paths.

The 'lead-ins', 'loops' menu allows you to add computer-defined loops to the corners of your components. You define the what is considered a corner by entering the minimum angle of deviation from a tangential transition. The input dialog opens automatically.

The 'edit cutting path' menu in the 'edit cutting path' screen allows you to add and edit individual roundings or loops.

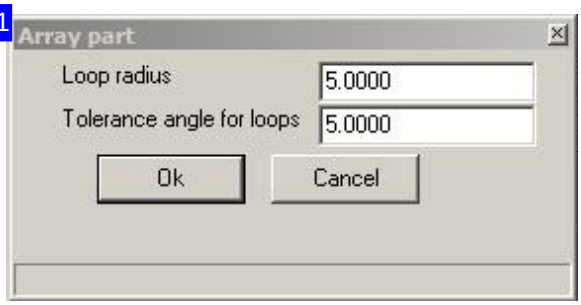
The 'parameters for editing cutting contours' dialog allows you to define the parameters for and form of loops.
Default settings can be made in the 'contour management parameters' dialog.

1



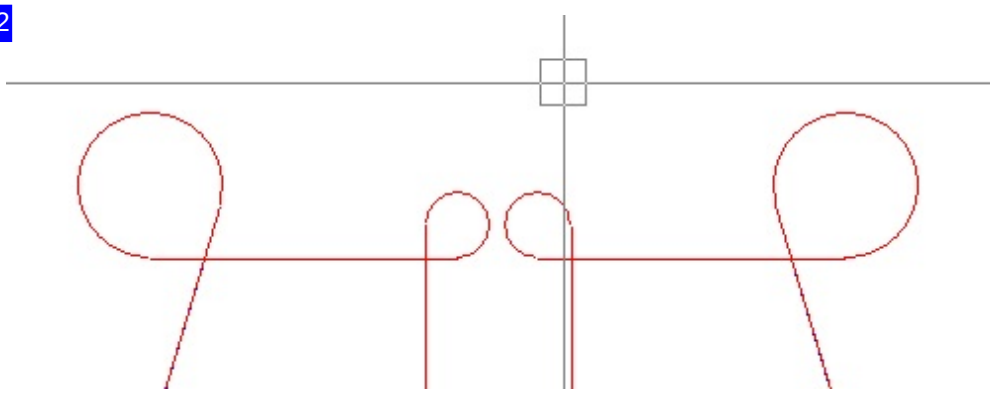
Select the 'loops' tab to access the settings for your loops' size and form.

11



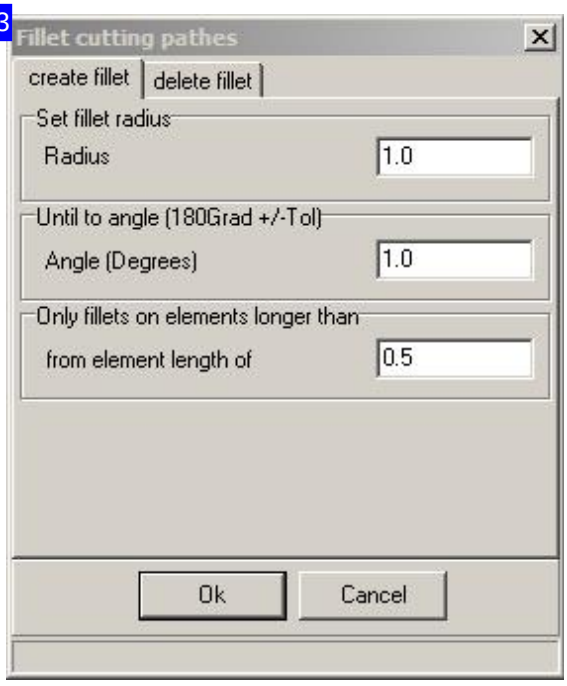
Use this dialog to define a corner which will require a loop. This is defined by the angle of variance from a tangential transition which you enter here.

12



Loops on a component's corners. Loops can be placed automatically at every corner, or placed and edited individually in case of conflicts.
Use the selection menu in the cutting path edit screen to delete individual loops.

13



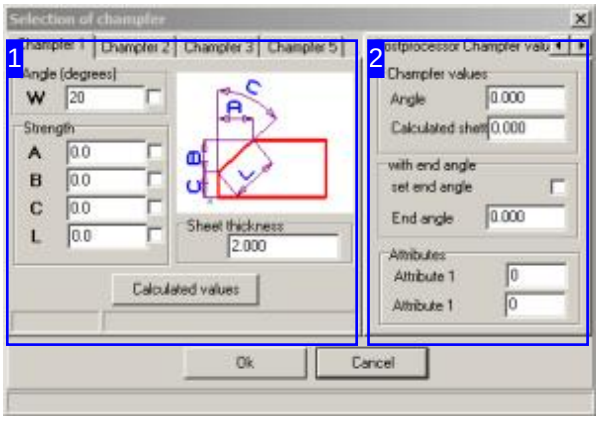
This frame is for setting the conditions for rounded corners. Define a corner in the same way as for a loop . Use the next tab to delete individual roundings.

Access this dialog from the cutting path edit screen.

14

You can delete one, all or only the roundings from a contour here. Access this function from the cutting path edit screen's selection menu.

Adding Beveled Edges to Cutting Paths.



If your cutting machine offers the option to tilt the cutting head and execute beveled cutting paths, then this module allows you to setup your contours to tilt the cutting head at any point you wish. This feature is optional.

You can alternatively implement the optional bevelling-module from *cncCUT* . You start this module with command 732 in the commandline, in case the post-processor does not show an menu-field. The online-help 'Technologies' for *cncCUT* gives you more information about bevelling in the chapter 'Cutting contour settings - Bevels'.

Note:
You must insure you have the necessary clearance to the next component when using bevels with a base height! It is also important to check the sheet is thick enough, i.e. the sheet thickness must be greater than the base height.

To cut parts with beveled edges requires the cutting head to be tilted at the beginning of the beveled edge. Edges are then beveled until the next lead-out, or until the head is tilted back. Should the bevel direction (positive - negative) or a contour's bevel values change, then the element must be separated at this point and the bevel redefined. To finish cutting a bevel, tilt the head back to 0° at the required point.

The left tab panel [1] contains various bevel types which have unlimited possibilities:

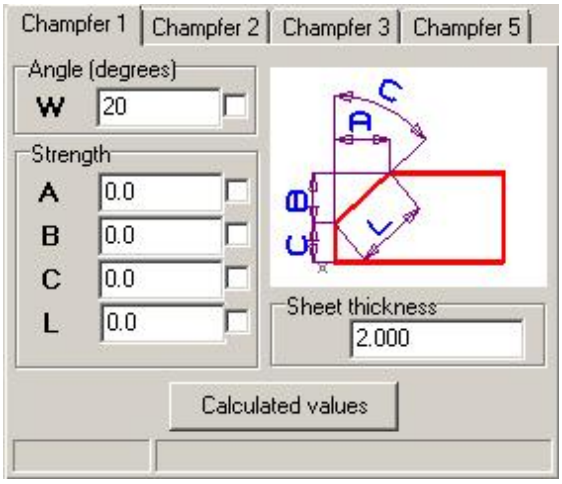
- Bevel 1 - Complex, positive bevel with calculations.
- Bevel 2 - Complex, negative bevel with calculations.
- Bevel 3 - Simple, positive bevel.
- Bevel 5 - Simple, negative bevel.

Two further tabs are available to access program settings for the automatic definition of bevel transitions for multiple elements. You can also enter the pass height here; the content of these tabs is therefore post-processor dependent.

- Bevel 1 - Complex, positive bevel with calculations.
- Bevel 2 - Complex, negative bevel with calculations.

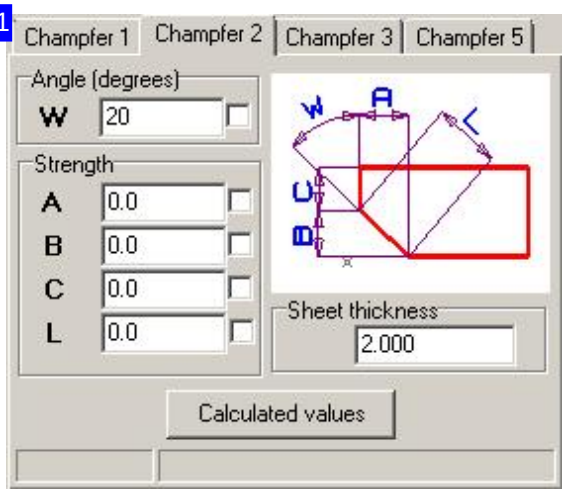
In the 'show bevel' tab in the right pane [2] you will see a preview of the bevel according to your current settings to help proof your calculations.

1



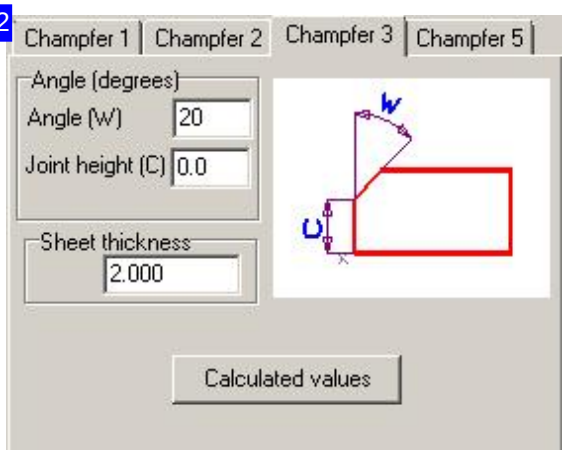
Complex, positive bevel with calculations - A positive bevel with a base height can be defined using the angle (W), base height (C), cutting width (A), cutting height (B) or cutting length (L). Enter the value, mark two values' checkboxes and click 'calculate values'. The current values will be displayed and can be allocated to your contour by clicking 'Apply'.

11



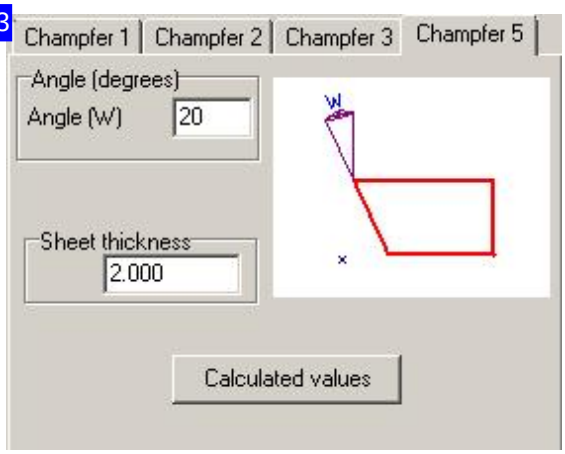
Complex, negative bevel with calculations - A negative bevel with a base height can be defined using the angle (W), base height (C), cutting width (A), cutting height (B) or cutting length (L). Enter the value, mark two values' checkboxes and click 'calculate values'. The current values will be displayed and can be allocated to your contour by clicking 'Apply'.

12



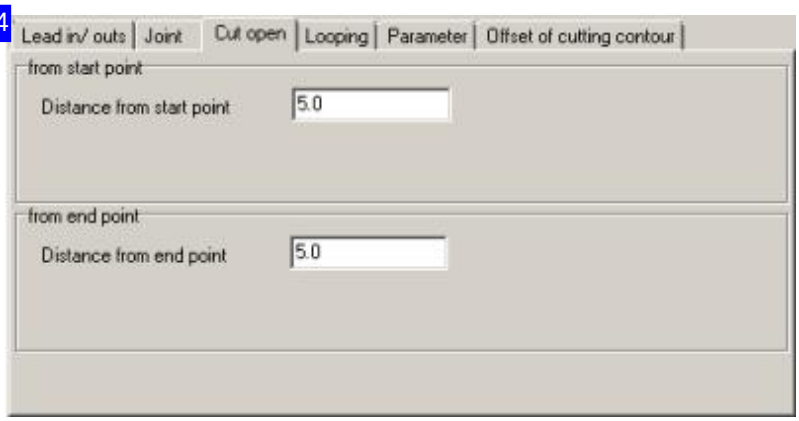
Simple, positive bevel - A positive bevel with a base height can be defined using the angle (W) and base height (C). Enter the values and click 'calculate values'. The current values will be displayed and can be allocated to your contour by clicking 'Apply'.

13



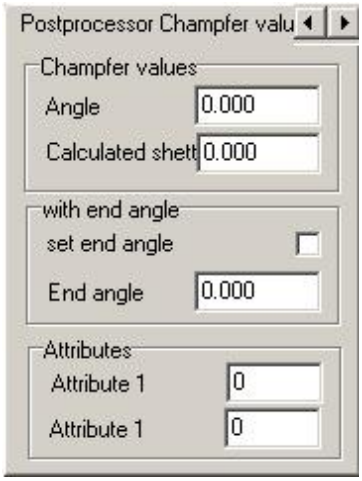
Simple, negative bevel - A negative bevel with a base height can be defined using the angle (W) and base height (C). Enter the values and click 'calculate values'. The current values will be displayed and can be allocated to your contour by clicking 'Apply'.

14



The 'separate' function allows you to split a contour and choose a specific point in the cutting path at which to change the bevel angle. When editing a cutting path you can also separate a contour element in the selection menu.

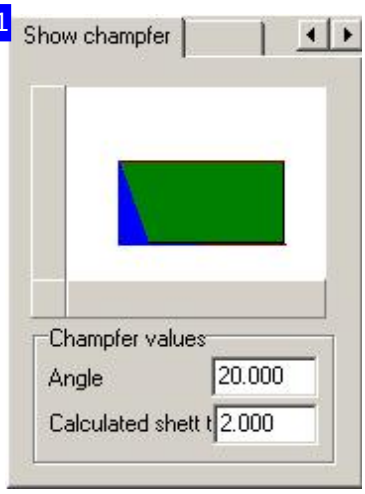
2



This frame allows you to enter further values for your post-processor. The values are considered by the post-processor and their functions must be defined there.
The 'with end angle' frame allows you to define the end angle of a dynamic bevel. Mark the checkbox to set an end angle and the cutting head will then steplessly rotate from the start angle to the end angle, if your machine has this capability.

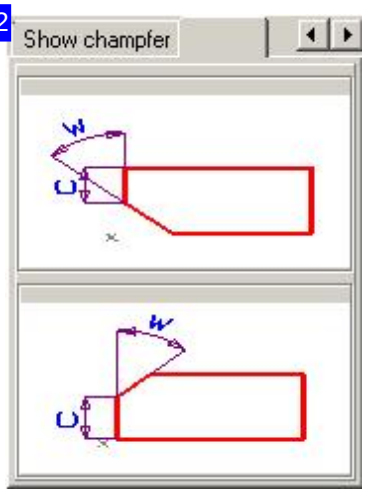
The 'display bevel' tab shows a preview of the bevel according to your variables. The bevel values are also shown on the cutting path using corresponding symbols.

21



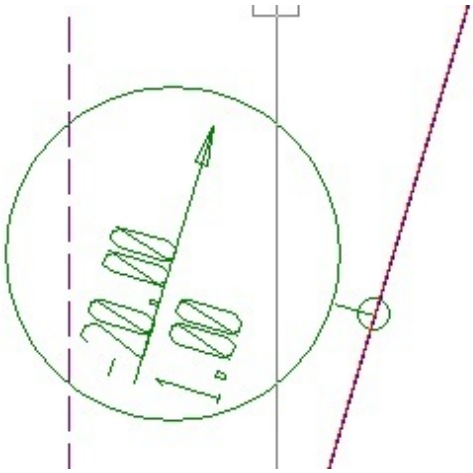
This tab shows a preview of the bevel according to your variables, allowing speedy visual error-checking of your work.

22



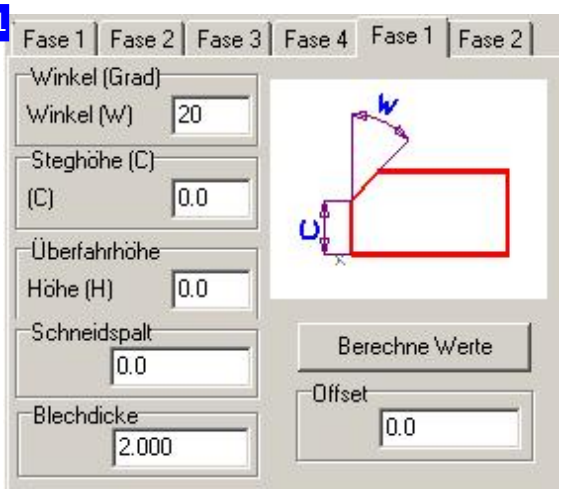
This display gives orientation assistance for multi-torch cutting and is therefore post-processor dependent; your machine must have a multi-torch cutting facility.

23



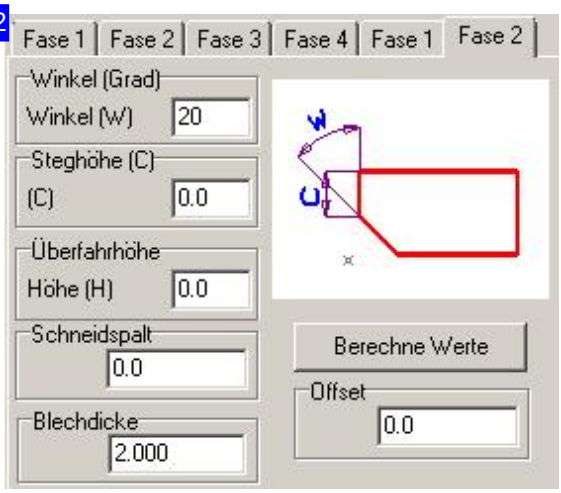
Bevel values are shown on your cutting paths with the angle, base height and cutting direction displayed at bevel transitions.

31



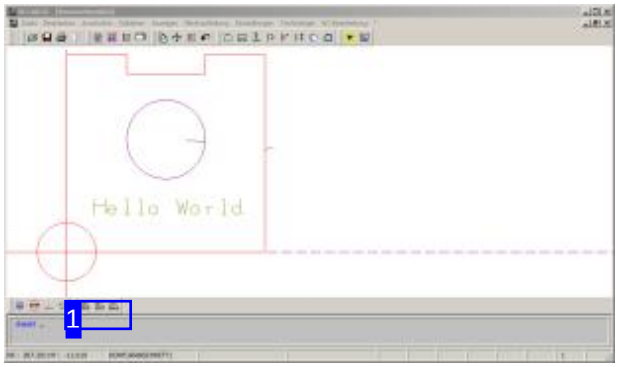
Where supported by your post-processor, you can enter a pass height (H) for a positive bevel here.

32



Where supported by your post-processor, you can enter a pass height (H) for a positive bevel here.

Marking Parts.

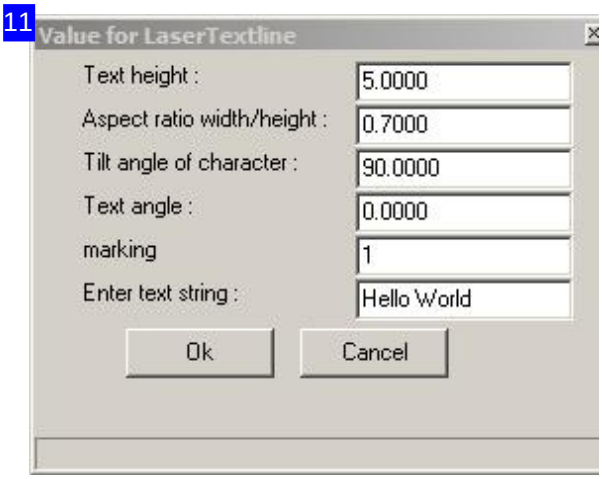


Parts or contours can be marked with various technologies. The marking types available are machine and post-processor dependent. You can add texts and punch circles ready for processing with your cutting tools.

Marking texts are created as elements which can subsequently be changed:

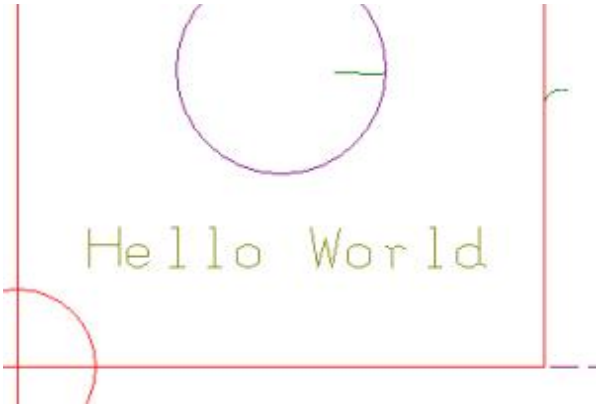
- Create text
- Move text
- Change text numbering

Circles can be punched individually or as types, i.e. all circles with the same diameter will be punched. Select the corresponding menu item and identify the required circle.



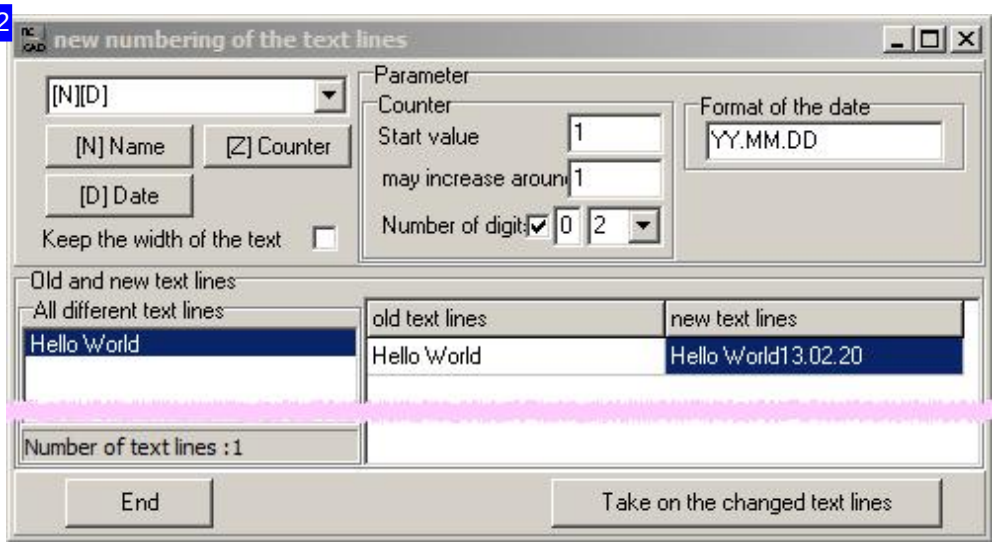
To enter a marking text, select the menu item 'Edit', 'Marking', 'Text Marking'.
A dialog will open to enter your text and text settings. Text variables available are: height; width:height ratio; text orientation angle; text lean angle. The processing type is set in the field 'Marking' <0:cut, 1:punch, 2:powder, 3:mark>. The text string is finally entered under 'Text line'.

1



Marking texts can be moved about the part. Select the menu item 'Edit', 'Marking', 'Move Text' and click the bottom left corner of the text to be moved. The text is picked up and can be dragged around the part. To move in one axis only press <X> or <Y>.

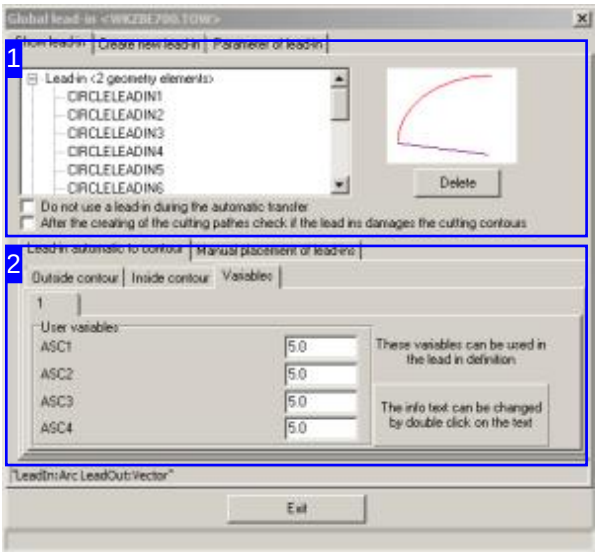
12



This dialog is accessed through the menu item 'Edit', 'Marking', 'Change Text Numbering' .
This dialog allows you to modify all created texts and add a sequential number and date stamp.
Enter the marking format in the top-left entry field. Select [N] for name, [Z] for a sequential counter and [D] for a date stamp. Enter the format using spaces to separate. The 'Parameter' pane allows you to set date format, using a period as the separator.
The bottom-left list shows all existing marking texts. Select the text you wish to modify according to the settings made and the new version will be previewed in the bottom-right section of this dialog.

Lead-in Manager

Prepare Lead-Ins For Use.

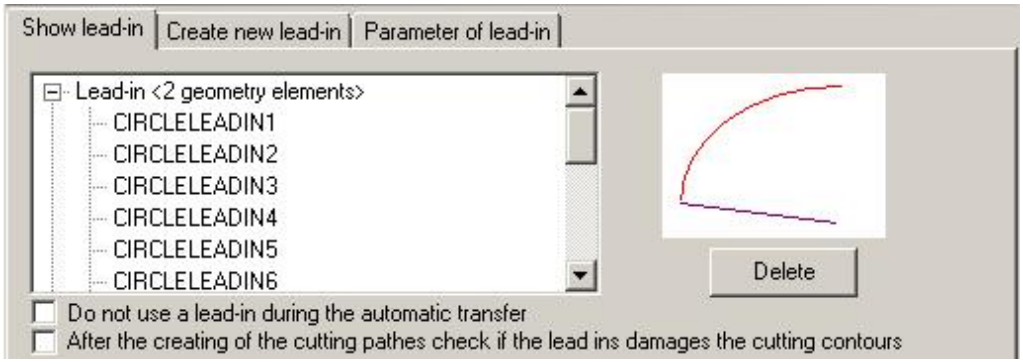


Tech cuts, i.e. the lead-ins and lead-outs when cutting your parts here referred to more generally as lead-ins, are automatically allocated by the program when creating nested sheets. Prerequisite for this is that your lead-ins are defined first. The lead-in manager of *ncSchneid* offers a selection of standard lead-in types which you can use as-is, or customize to your needs and machines.

Lead-ins are managed from a dialog which can be accessed from the menu under 'Lead-ins' , 'Lead-in Management'. The dialog shows all available lead-ins [1] and offers the option to create customized lead-ins as required and allocate them to certain job types.
The top tab row has the following options:

- Display and allocate lead-ins.
- Create new lead-ins.
- Display a lead-ins parameters.

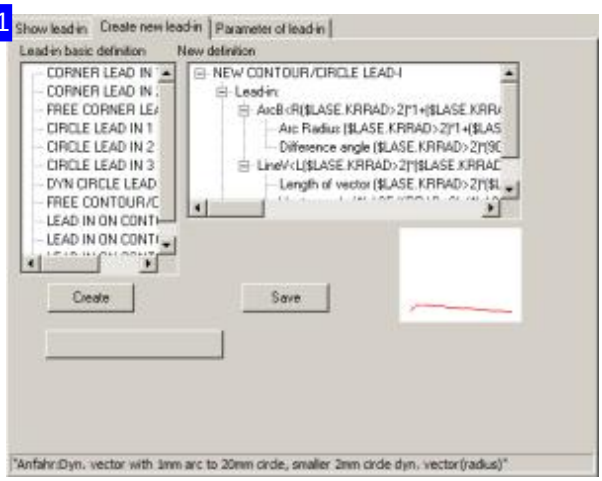
1



The 'Display lead-ins' tab has a list of all available lead-in types to the left. Select a list item to preview that lead-in to the right. You can also delete items from the selection in this tab. Right-click to open an edit menu.
Double-click to open the 'Lead-in Parameters' tab to review the parameters.

The selection boxes at the bottom allow you to choose whether to use automatic lead-in placement, and whether to check for collisions with existing cutting paths. Such collisions can occur e.g. on tightly nested sheets, in this case you can modify the lead-in manually.

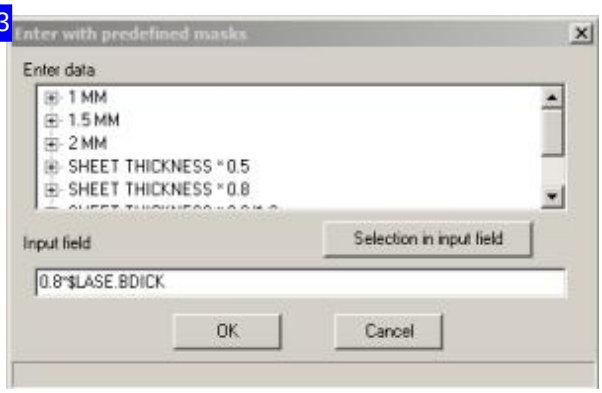
11



This tab allows you to define new lead-ins' values. Select the basic form from the list on the left, it will be shown in detail to the right, ready for further editing.
Select the required lead-in and choose the shape to be added using the buttons below the definition pane.
The selection will be added to the lead-in definition. Select an element and click 'Geo element values' to define the element.

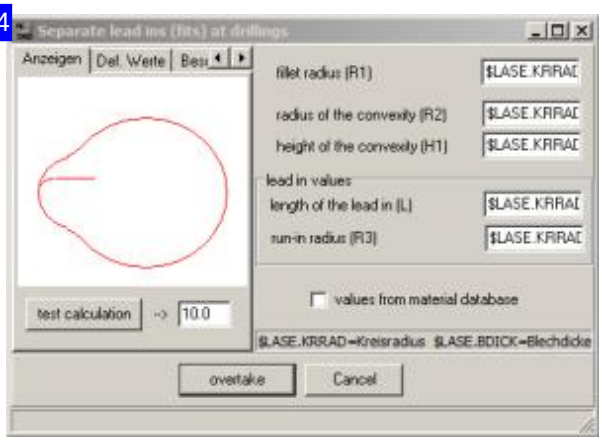
A special lead-in type is the hollow lead-in, click 'Create hollow lead-in' to use it.

13



You can enter concrete values or use formulae based on e.g. sheet thickness to define a lead-in element. The program offers a selection including useful variables in the top menu; make a choice and click to copy it to the entry field.

14



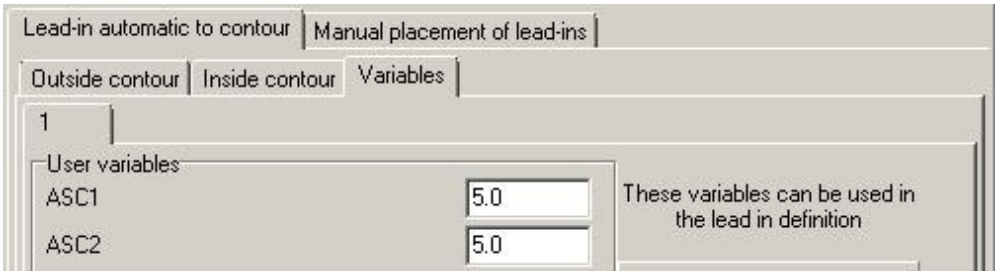
The hollow lead-in is a special type of lead-in consisting of various provisory elements which you can edit here. The individual parameters are described in the 'Description' tab, top-left.

12

The parameters of a selected lead-in are shown here. Double-click a lead-in in the 'Display lead-ins' tab to access this tab.

15

This menu allows you to delete, edit and save lead-ins.

2 

Decide here which lead-ins will be used for external contours when automatically converting drawings into cutting paths. Set the lead-in type by contour type. If you prefer corner lead-ins, mark the bottom-left checkbox. Click 'Insert' to confirm a lead-in type for automatic conversion.
Use the separate tab to define lead-ins for 'Internal contours'.
Lead-ins can be defined using variables, open the 'Variables' tab.

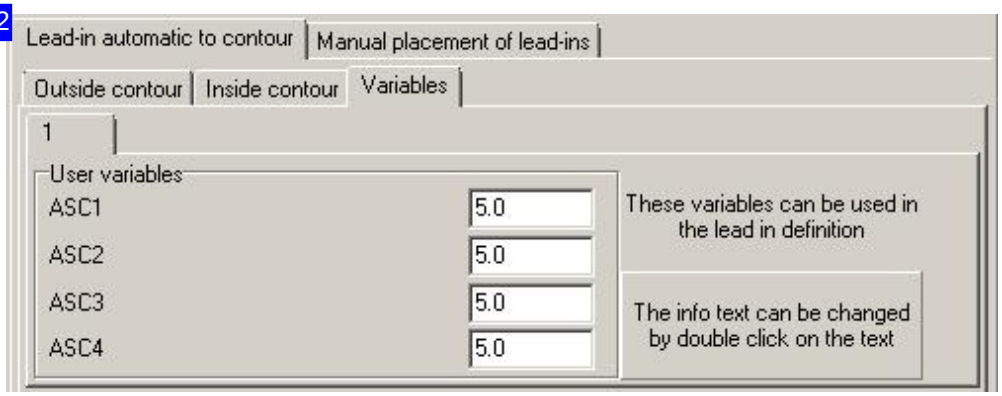
You are able to cut up inner circles. This function lets you cut areas in pieces, that are worked out as inner circles, to avoid obstruction or destruction of the tool. Open the 'Cut up inner circles' tab

If you prefer to add lead-ins manually, open the 'Lead-ins manually' tab.

21

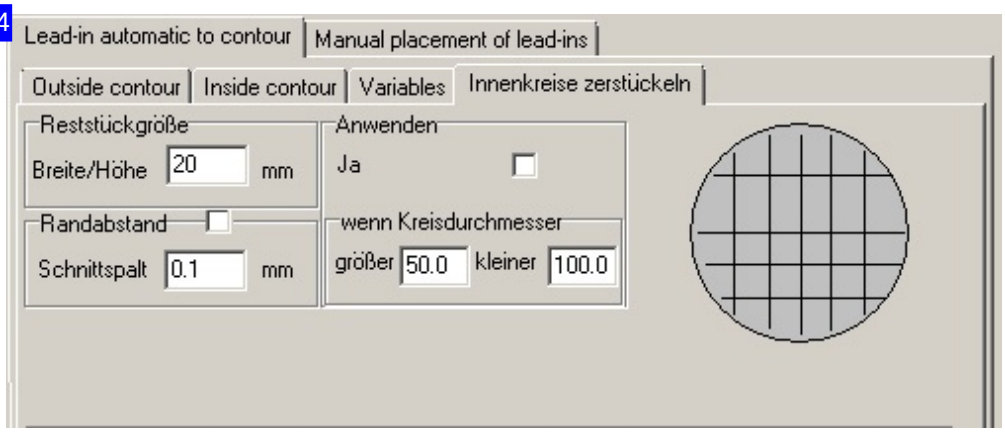
Decide here which lead-ins will be used for internal contours. The requirements are the same as for External contours.

22



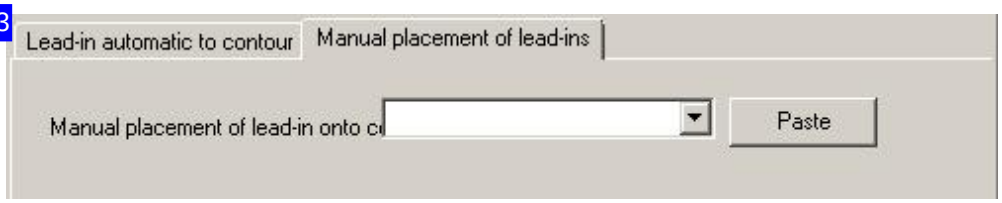
Lead-ins can be defined using variables; their identifiers and values can be set here.

24



Mark the checkbox 'Apply' for cutting inner circles in pieces. If you proceed with creating cutting pathes, the cuts are displayed in the drawing. The inner pieces cuts will be cut first and after it, the inner circle. This function is only applicable to inner circles!

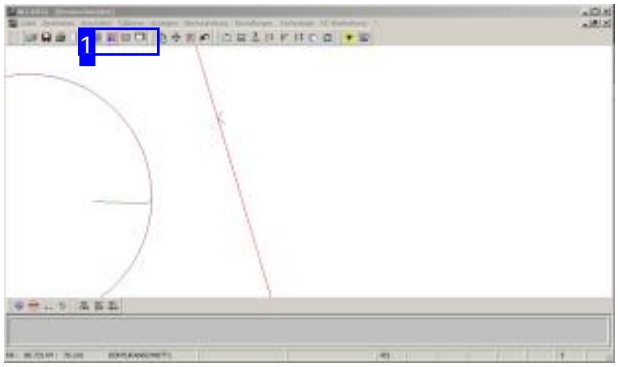
23



Enter settings about lead-ins which will be added manually to your contours.

Nesting Plans

Organize Parts On A Sheet.



Nesting plans contain multiple parts which each represent one component to produce. These parts are set out on a sheet in columns and rows to create a grid consisting of outline boxes around each component. Settings made in *ncSchneid* allow the parts to be sorted in such a way as to use the shortest possible cutting paths. There are however various reasons (e.g. heat dissipation) why the shortest path is not always the best solution. For this reason you can process columns and rows in whichever order works best for you.

Multiple, varying parts can also be nested together manually. More information on this process can be found in the chapter 'Collating Parts To A Nesting List'.

Nesting plans can also be created automatically by separate nesting software, please see the chapter 'Automatic Nesting'.

Completed cutting paths with inside and outside contours can be duplicated as required and the processing order set.

Individual parts can be:

- Copied
- Deleted
- Moved
- Checked for overlaps.
- Sorted

The following pre-production steps can be taken:

- Set cutting direction.
- Define offsets.
- Set loops.
- Setup multi-torches.
- Sort

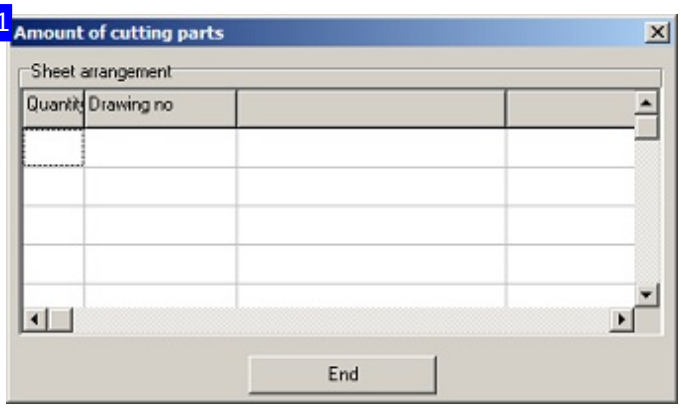
The basis for production is always your metal sheet. If the required sheet is not defined in your database then you can begin with a virtual sheet. The sheet properties can be accessed through the 'Sheet Segmentation', 'Sheet Size; menu. A dialog will open for you to enter the sheet information.

The menu item 'Display', 'Part Information' opens a dialog with various information about your part.

1

Sheet size, grid processing and automatic nesting can be accessed directly from the icon bar. Select the corresponding menu item to gather cutting paths to for a nesting list.

11



This dialog shows information about the parts on the sheet.

Produce Grid Layouts.

Your cutting paths and manual constructions can be placed on the sheet quickly and efficiently using your settings to generate a grid production layout.

Grid production is based on your requirements for the number of columns, rows and the offsets. The current layout will be expanded using your settings.

Use the command 'Delete Grid Production' to remove the grid layout.

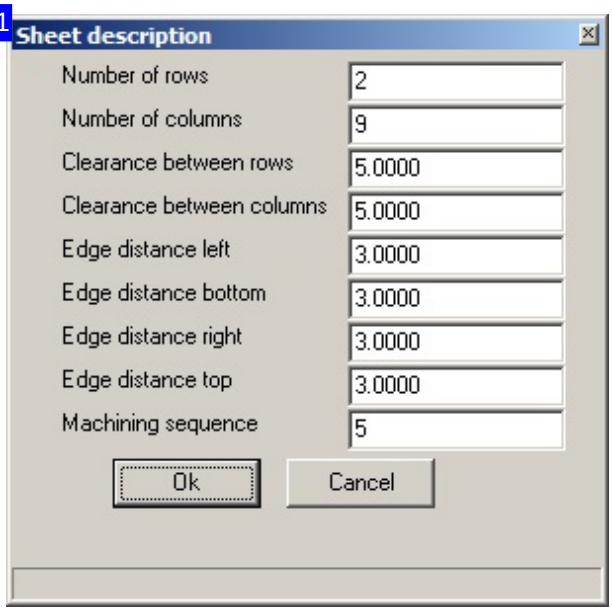
The columns and rows can also be sorted to your requirements. For more information please see the chapter 'Sorting the processing order'.

Use the menu item 'Move all parts' to reposition the entire nest on the sheet.

1

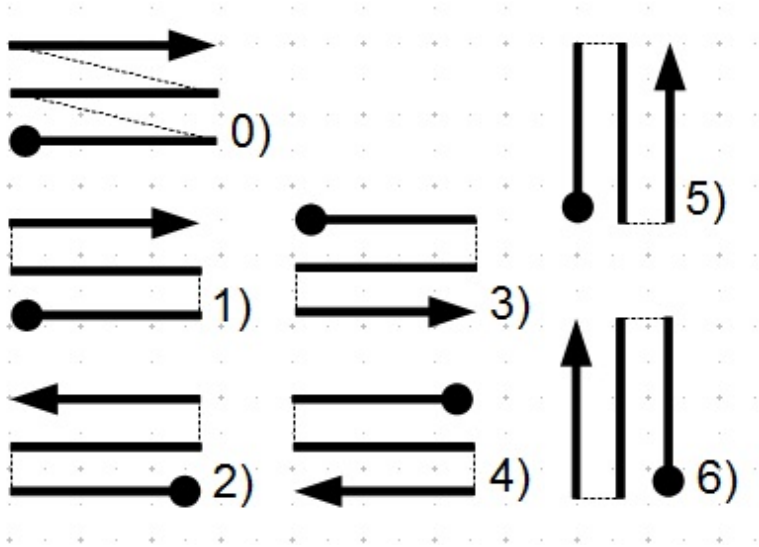
Grid production is accessed through the menu panel under 'Sheet allocation' or through the icon bar. A dialog opens for you to enter all grid specifications.

11



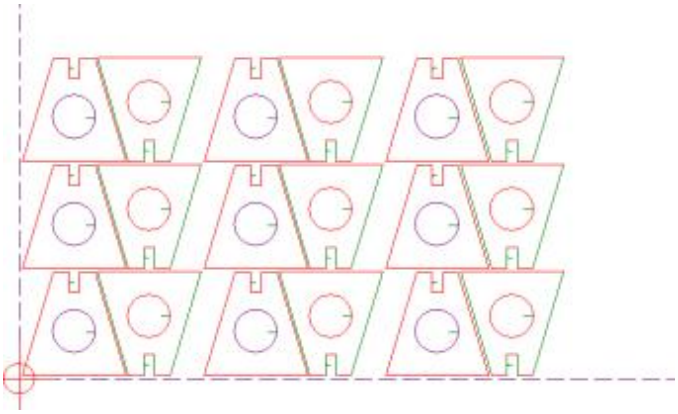
Enter the sheet specifications for your grid layout in this dialog. The top fields are for entering the number of rows and columns of parts required. Enter '0' here and the computer will take the maximum number that fit on your sheet.
The 'Sheet part positioning' variable defines the processing sequence.

12



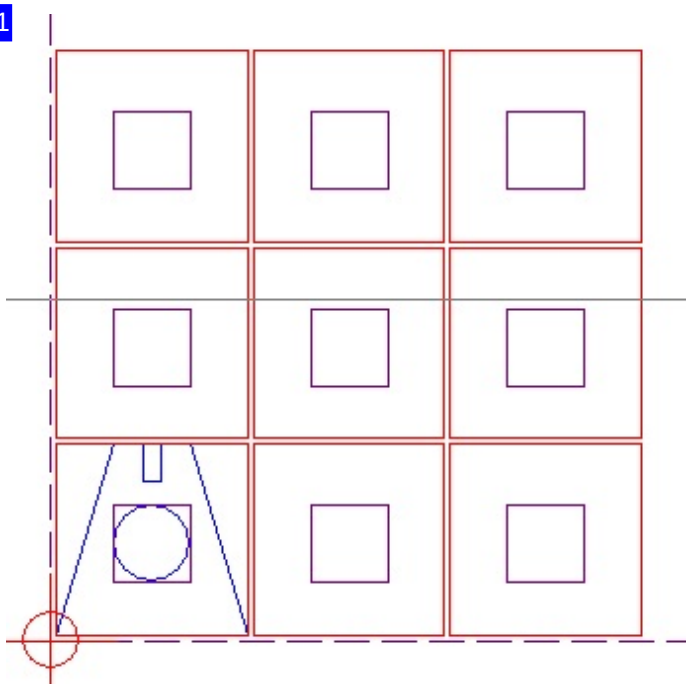
The processing sequence is defined using a code number. Processing begins at the point and follows the arrow. Dashed lines show necessitated idle travel paths. Enter the corresponding code number in the 'Sheet part positioning' field in the grid production settings dialog.

2



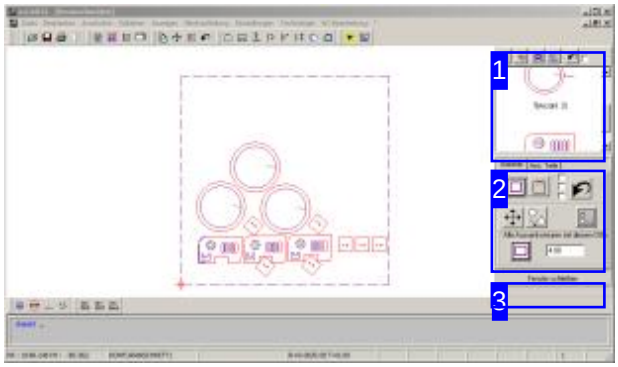
The example layout in rows and columns.
To check the processing order click to start the simulation .
If you're working on complex sheets with many parts of many contour elements, you may wish to simplify and speed up the display using outline boxes.

21



Use the menu item 'Show only boxes' or the corresponding selection in the icon bar to simplify the preview and thus shorten preparation calculation time and simulation run time.

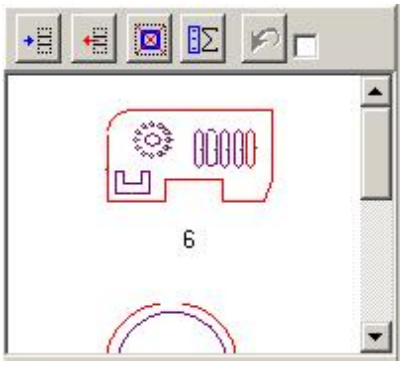
Collating Parts To A Nesting List.



The top right pane contains the collation list [1], and the bottom right pane the manipulation options [2]. All actions in this screen contribute to your final processing pattern, the screen remains open until you close it. Pay attention to the notes in the command window at the bottom left to know the editing status. A help window with the relevant commands will become available as required, open it by pressing <Ctrl><F1>.

This screen allows you to collate various cutting paths from your database to form a nesting list for manual nesting to your sheet. All manipulation commands are available such move, rotate, mirror, etc., so you can create a complete manual nesting plan, even allocating parts to non-permissible areas. The screen can be accessed through the menu under 'File', 'Load Cutting Path To A Nesting List'.

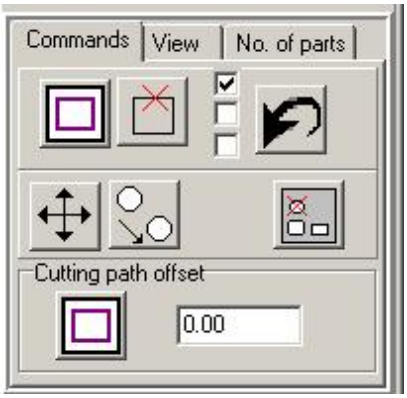
1



This list shows all loaded cutting paths, and allows them to be selected for processing. The list can be navigated using the buttons above the list and the parts on the sheet checked.
From left to right the buttons' functions are:

- Load a cutting path to the list. A 'file manager' opens for selection.
- Delete a part from the list.
- Delete a part from the sheet. You can identify one or more parts. Please check the command line.
- Count parts on the sheet.
- Undo last step.

2



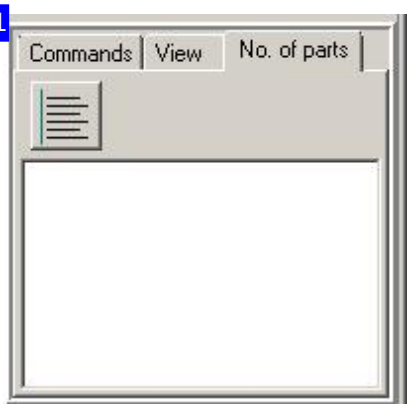
Use the buttons in the top of this pane to allocate an offset to a part or delete a lead-in.

The middle area allows you to move parts and copy or delete as required. Click the function, then identify the part on the sheet.

The bottom section allows you to overlay the outside contours with a value as required.

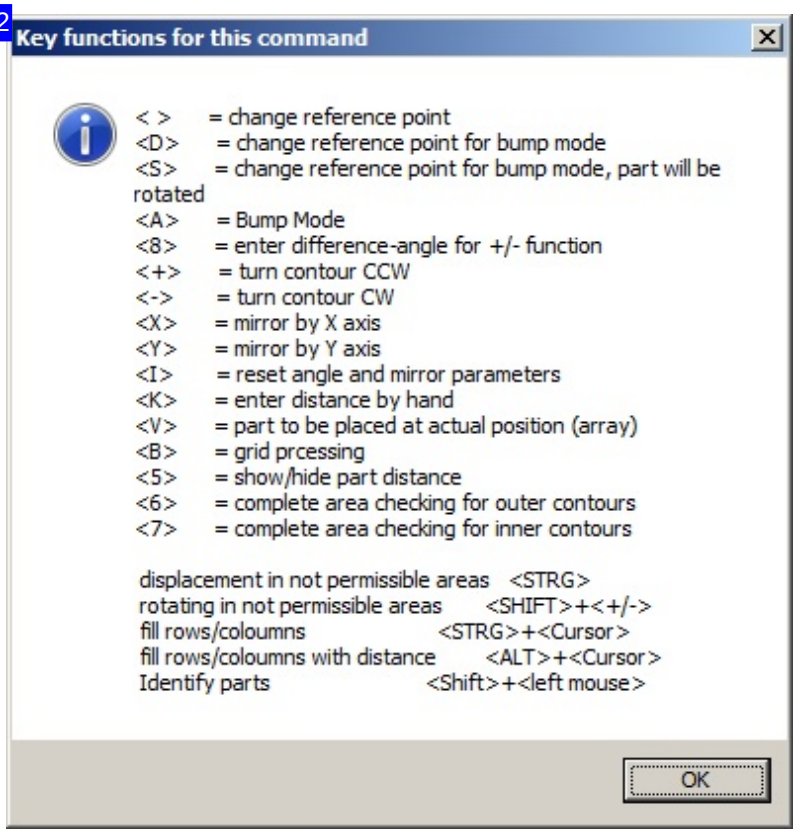
The 'Part Count' tab shows an exact checklist of the parts on the sheet.

21



This display shows a checklist of parts on the sheet. Click the button above the list to the left to refresh the display and check the effect of any changes.

22



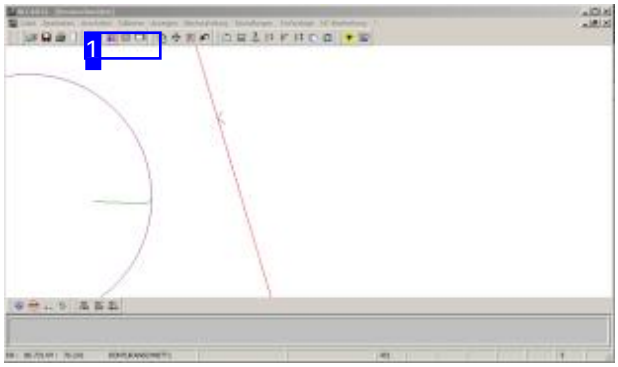
Keys are allocated different quick-key functions when executing command sequences. This dialog gives a summary of the corresponding functions and is accessed by pressing <Ctrl><F1> when shown in the command field.

The content of this dialog can also be accessed when manipulating cutting paths to remind you of useful quick-key functions.

3

Once all cutting paths have been placed on your sheet, close this screen and the cuttings paths will be transferred for further editing.

Automatic Nesting.



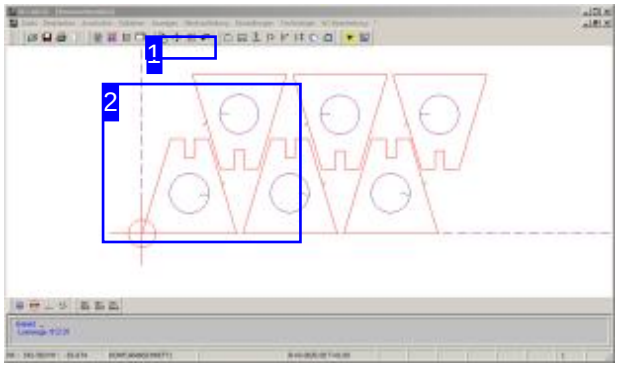
You can access an optional nesting module through the icon bar or the menu item 'Sheet Allocation', 'True-shape nesting'.

Automatic nesting allows optional software modules to use varying methods to create the most efficient sheet layouts. The nesting programs are individual modules which can be integrated into your software individually as options to meet your business needs. The optional components must be enabled in your license dongle.

1

Click the leftmost button to access the optional nesting modules. You can also gain access using the menu item 'Sheet Allocation', 'True-shape nesting'.

Sorting Parts Onto The Sheet Manually.



A whole cutting path can be copied, rotated, mirrored and moved around your sheet anywhere you require. This allows comprehensive nesting plans to be created by hand.

When placing a part it will be 'attached' to the cursor. You can select the grab point in the 'Cutting Settings' dialog [12].

Completed parts can be duplicated and positioned on your sheet as required. Under 'Edit', 'Manipulate' you will find a submenu with the commands for 'Copy', 'Delete', 'Move' and 'Check Overlaps'.

Other important functions include copying parts to the bump mode (<A>), which places parts at a preset offset to existing parts or the sheet margin; and copying parts to a grid (<B>), which places parts in rows and columns on your sheet according to an area you define with a stretch-box. These tools allow the minimum time to be used to create your manual nests.
You can also place parts using the following quick-key methods:

- <A> - Top-left corner
- <O> - Left sheet margin
- <0> - Top sheet margin
- <P> - Right sheet margin
- <L> - Bottom sheet margin

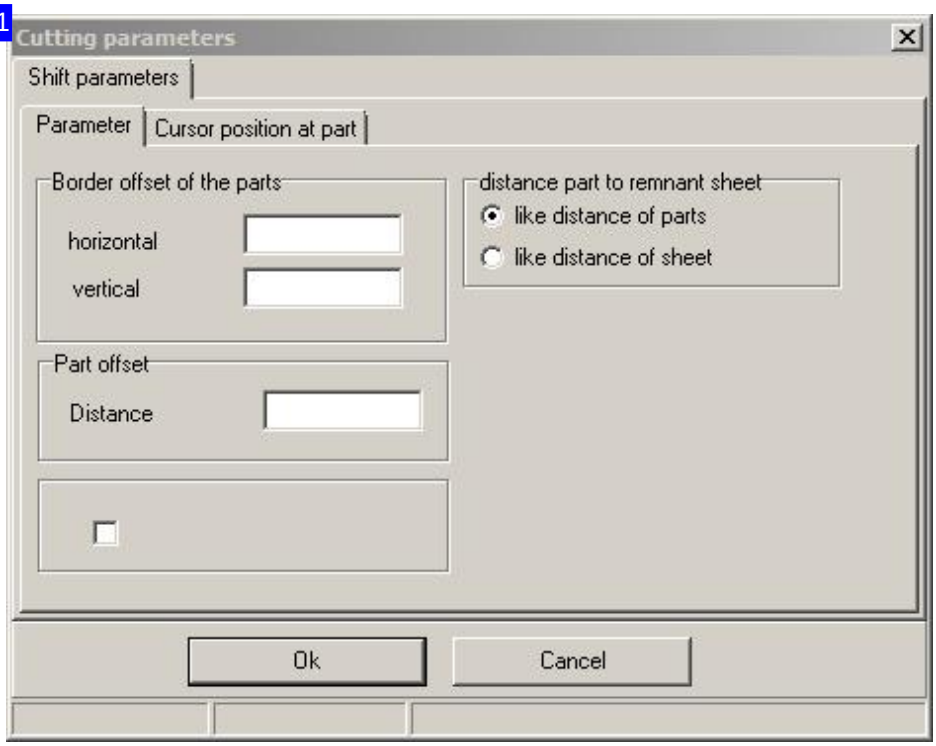
When deleting contours you can double-click to repeat the operation as required. Press <Esc> to end the function.

After placing parts you should check the nest for overlaps of contours or lead-ins. Solve problems as required - you may need to move lead-ins.

1

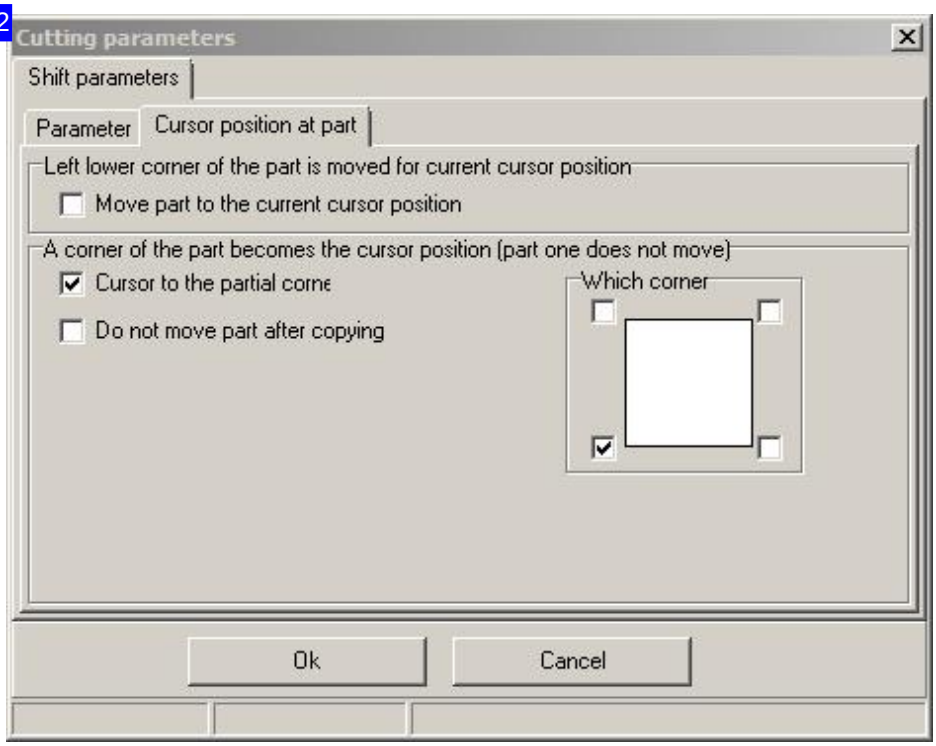
You can also reach most of the manipulation controls through the icon bar.

11



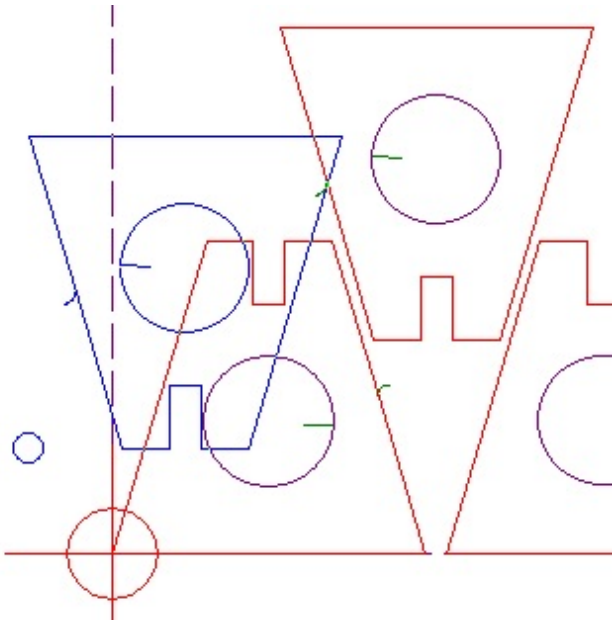
The dialog for the bump-mode's offset settings can be accessed through the menu panel.
For successful automatic part placement, set the required offset here. Set the margin offsets and clearance from one part to the next. For creation of offcut sheets you can choose whether to treat the separating cut as a sheet margin or a cut part edge.
To apply the values mark the checkbox at the bottom-left, otherwise the default settings in the database are used.
The part being placed will be hung from the cursor according to the offsets being used. The second tab contains the cursor settings.

12



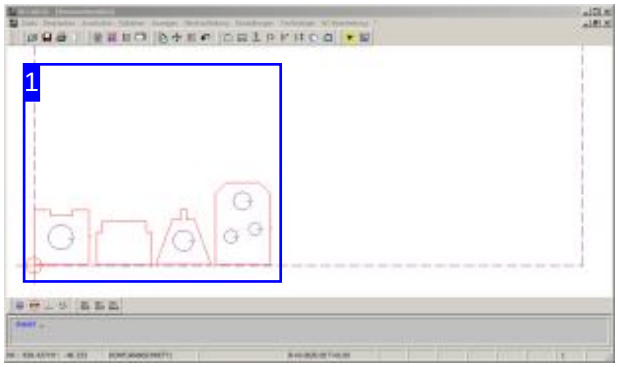
This dialog allows you to set where the cursor should grab a part when placing parts manually. This setting is important to maintain the offsets to the sheet margins and other parts.

2



When copying, the cutting path is 'hung' from the cursor as a template - shown here in blue - which can be replicated onto the sheet as many times as required simply by clicking. Right-click to open a context menu for manipulating the part template. This menu has the same content as the selection dialog in the nesting list. Use the corresponding keys to manipulate the template directly.
Exit the function using <Esc> or <Q>.
The part can be grabbed by different corners, more information here.

Cutting Paths With Multiple Torches.

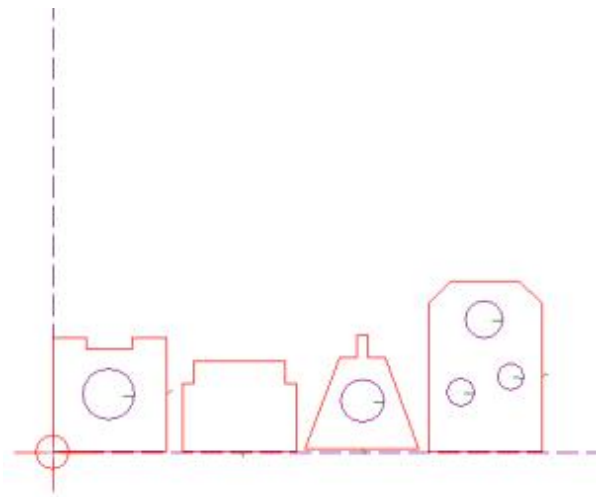


Nests on which it is useful to employ multiple torches are characterized by a quantity of cut parts arranged in identical rows on the sheet.
Place all parts which should be cut in a single strip. Keep a smooth top line in order to insure maximum material utilization.

Select the menu item 'Sheet Division', 'Multi-Torch'; to open a dialog for setting up multi-torch paths.

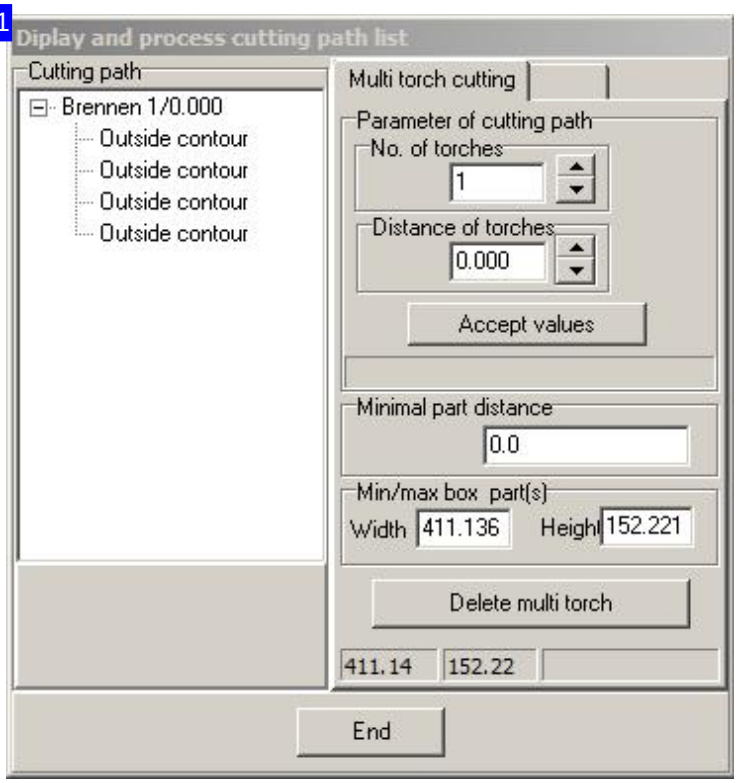
Machines which are equipped with multiple cutting torches can only use them when the nest is arranged with parts parallel. It is necessary to place parts on the sheet in 'strips' relative to the number of cutting heads, and the cutting head offset. If the cutting heads can be switched individually, this must be setup through your post-processor. Usage of multi-torch cutting is machine and post-processor dependent.

1



Here we see four parts in a strip. This parts can be set into strips as a group, or each part individually. The strips will be cut with the corresponding number of torches.

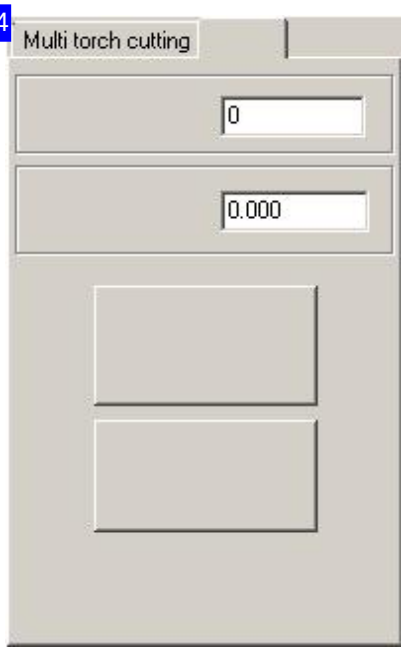
11



The table to the left lists all parts and the torch quantity (strip quantity) and offset. The 'Min max box' field shows the height and width of all parts in the list, or select a single list entry to see its max dimensions. The box height is the minimum torch offset. Enter a safety offset in the 'Minimum part offset' field and double-click the 'Torch offset' field to sum these values, giving the total torch offset.
Set the number of cutting heads in the field 'Torch quantity', and the preview will be updated correspondingly.

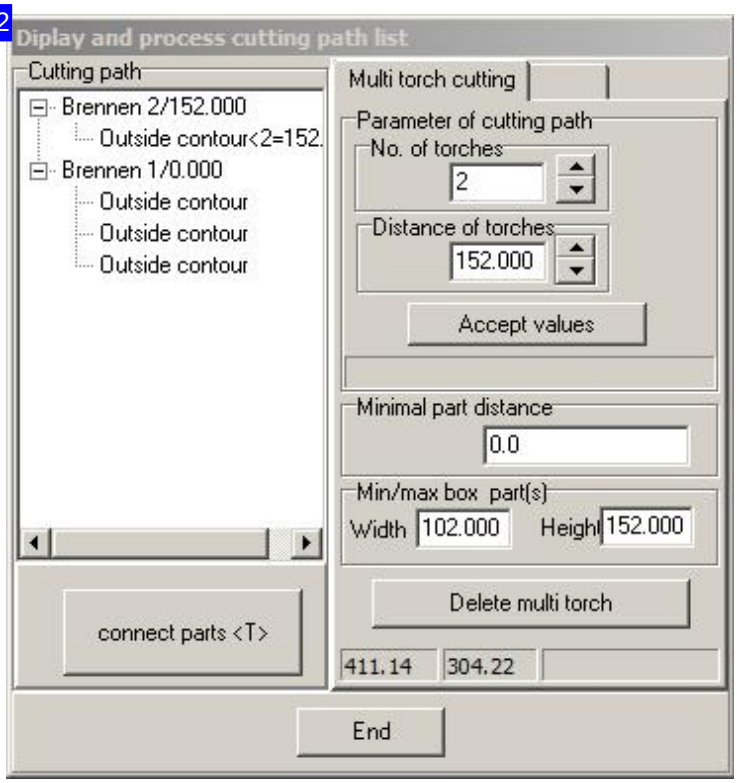
You can repeat this process for individual parts, or tag any parts and process them in a group. If you hold the Ctrl-key always the next item from the list is selected. The torch list will be updated accordingly.

14



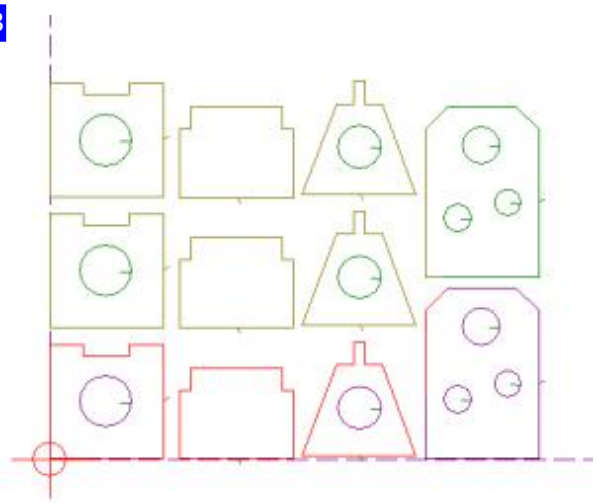
The 'Single torch' tab is required only for a small number of machines.

12



The overview to the left shows the list of torch paths with varying part counts. The right-hand part from the original layout was separated from the original list and given a new torch offset. This creates a new torch list for this part. The new allocation is shown in the preview.

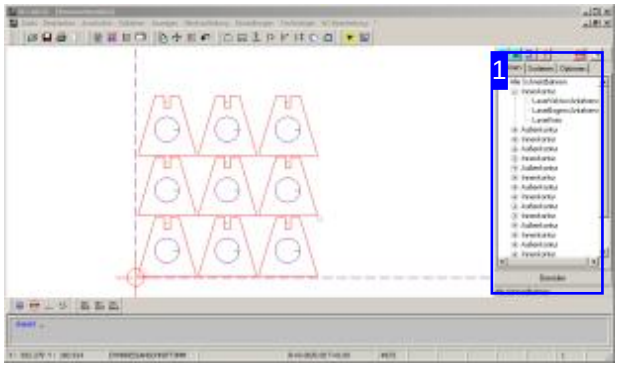
13



The preview shows the allocation from the torch settings with different torch counts. The right part will be cut in only two strips, the others in three.

Sorting and Combining

Sorting The Processing Sequence.



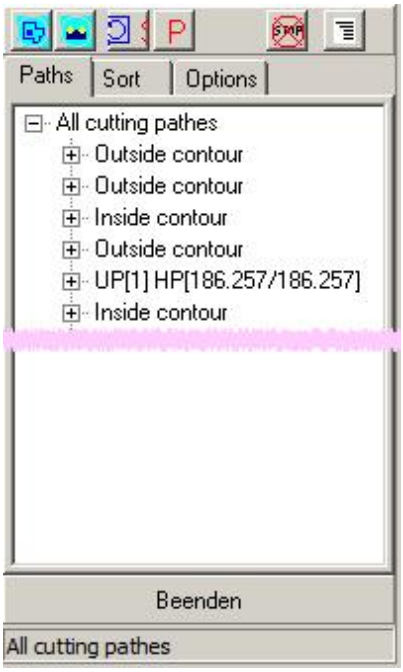
When cutting multiple parts from a sheet you may need to modify the processing sequence to allow for heat dissipation.

The menu item 'Edit', 'Sort', 'Automatic', Edit Sorting List...' contains all options needed to resequence the processing order for the parts on your sheet. The rows and columns can be processed in any order, i.e. the cutting sequence can be customized.

Right-click a cutting path in the list to open a menu with instructions for manual sorting. Click a contour element for a tailored menu.

After sorting, check the processing sequence in the simulation.

1



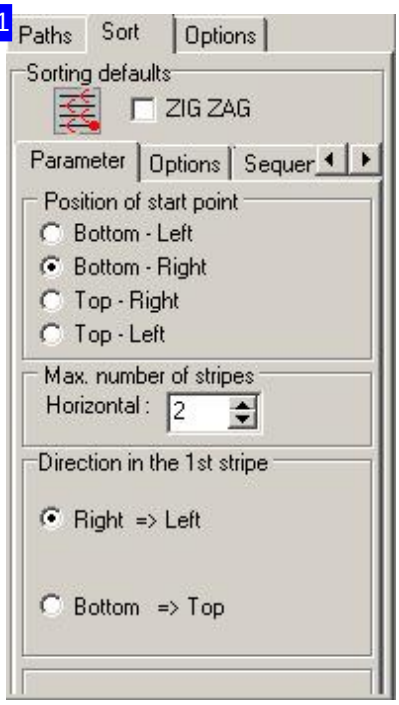
This pane shows the sorting list for each individual cutting contour. The buttons in the top row have the following functions:

- Show only drawing.
- Show all.
- Sort all cutting paths according to pre-selection.
- Delete all machine stops.
- Show only the top selection layer.

Use the tabs in the header panel to switch to the 'Sorting' window to set the cutting sequence or the 'Options' to make general settings.

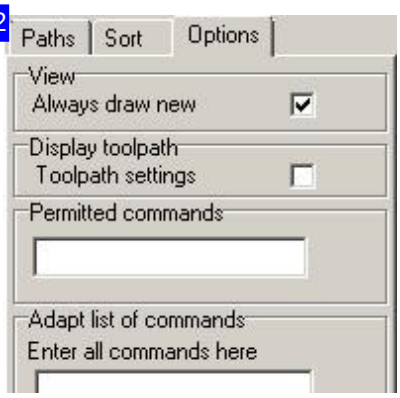
To move a contour element, select it in the list and drag it to the required new list position.

11



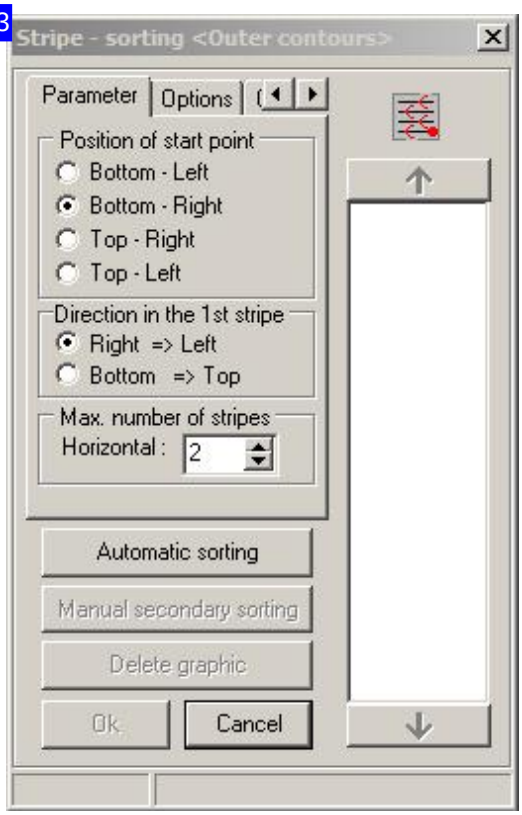
The sorting defaults tab allows you to set the sorting conditions for columns and rows. The icon in the top pane changes according to your selections and shows the start point and cutting route. For alternating cutting direction on each row, click 'Meander'; this saves time on idle paths.
The settings are the same as for direct sorting under menu item 'Automatic sorting'.

12



Make general sorting and display choices under the 'options' tab.
If you select the 'No cutting paths' option, only tech cuts (lead-in and lead-out) will be shown.

13



The dialog 'Strip sorting' shows the sorting of internal and external contours according to menu choice. Enter the start point location and select the direction for the first row. Enter the number of rows and click 'automatic sorting'. The icon at the top-right shows the selected sorting pattern according to your chosen options. The red point marks the location of the first part, the arrows show the continuing cutting direction. To cut rows in alternating direction just mark the 'meander' button in the 'Options' tab.

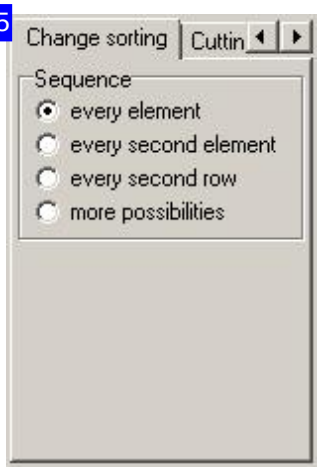
In the header panel tabs you can access sorting 'Options', select the processing 'Sequence' for rows and columns, and define the sorting of 'Markings'.
The sorting sequence can also be changed manually by drag-dropping items in the list to the right.

14

The 'Options' tab allows you to set the parts' reference point when sorting. This affects only the sorting sequence, not the parts' positions on the sheet.

- 'Start point' - the program uses the part's first cut and sorts so that these are sorted optimally.
- 'Center point' - the program takes the part's geometric center for sorting.
- 'min/max box' - the program places an imaginary box around each part and sorts based on that.

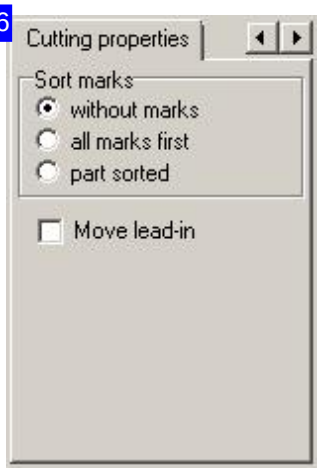
Mark 'Meander' to process strips in alternating direction.

15 

The 'Sequence' tab allows you to define the sorting sequence.

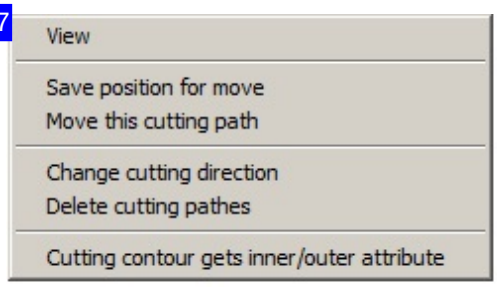
- 'Every part' - parts are cut one after the other, as nested.
- 'Every second part' - every second part is cut, skipping one at a time. This is useful when the sheet could otherwise overheat and deform.
- ''Every second strip' - this has all the advantages of cutting every second part and allows even more cooling time for the material. Idle paths are increased!

Select 'Further Options' for further options regarding sorting of strips and parts.

16 

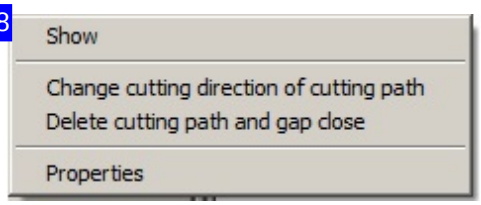
Settings for sorting your part markings. 'Part oriented' allows you to set if the sorting should start at an individual part, or be sheet oriented.

The selection box 'move lead-ins' informs the program that lead-ins should be moved in the cutting direction to avoid running over cut-out parts.

17 

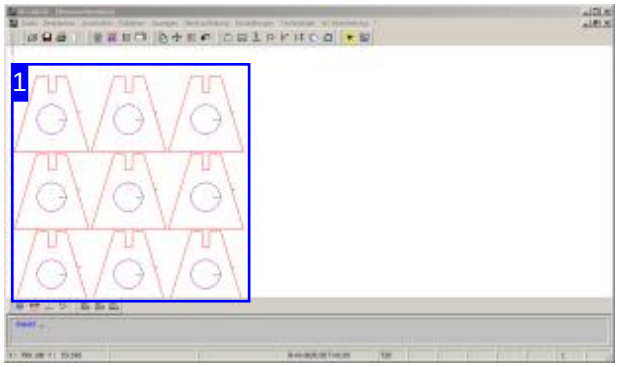
The menu items allow you to sort cutting paths manually and edit attributes.

18



Contour elements have a tailored menu for cutting path manipulation. Use the menu item 'Properties' to open the 'Cutting path attributes' dialog for adjusting your cutting paths.

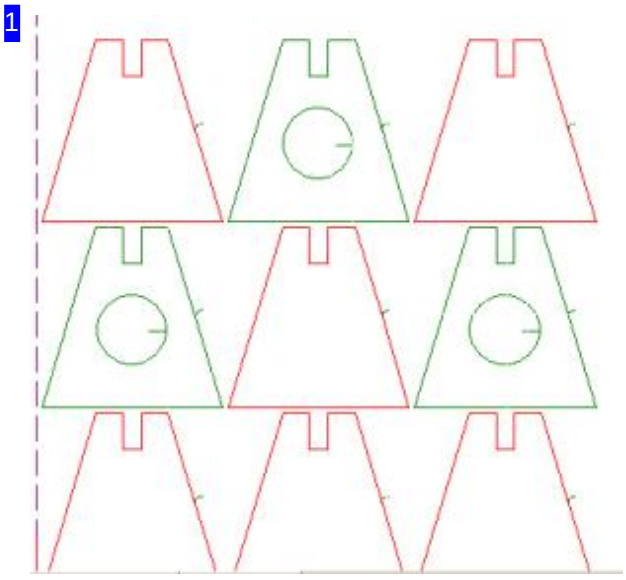
Manually Sort And Avoid Parts.



As well as automatic sorting of cutting paths, you can sort the processing sequence manually by parts, or by individual inside and outside contours.

When sorting manually a distinction is made between sorting parts and sorting cutting paths. The menu item 'Edit', 'Sort', 'Manual' allows you to choose between the two options.

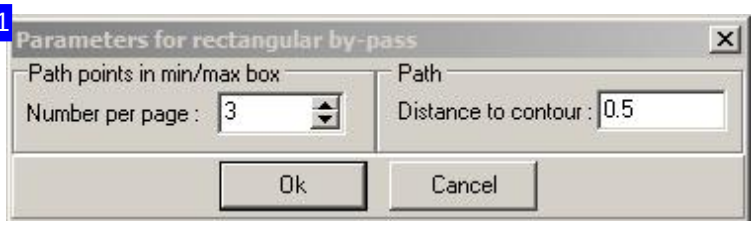
If you would like to avoid parts with potential collisions, select the menu item 'Edit', 'Sort', 'Avoid' . The part to be avoided will be enclosed in a box which shows the processing direction when cutting. Contours which have been cut cannot be collided with. When selecting the menu point a dialog opens for the corresponding settings.



Once you begin manual sorting, parts will be displayed only as outlines. Click the parts one after another in the order you want them cut. Selected parts are then shown in their entirety so you can see the status.

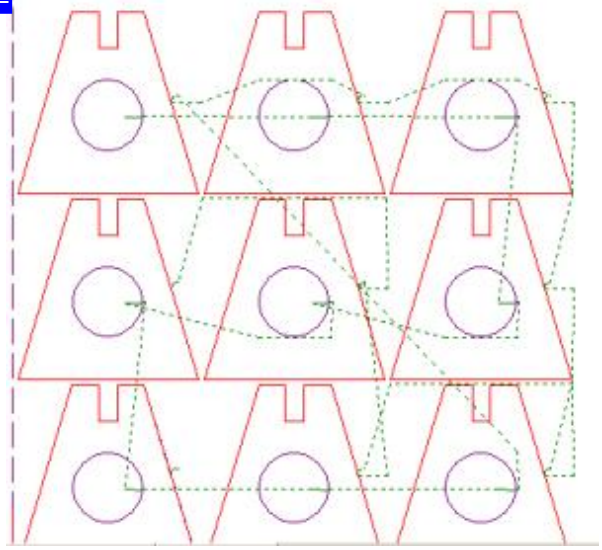
When sorting individual cutting paths a connecting line will be shown from the previous contour to the cursor position. Click the cutting paths sequentially as they should be processed.

11



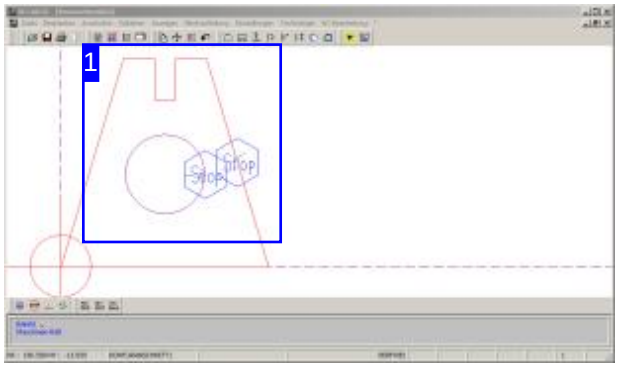
For avoiding cutting paths enter the number of waypoints the program may use for seeking free space in this dialog. A large number increases calculation time. In the right pane enter the offset of the processing path from contours. Finally click 'OK' to calculate the processing path.

12



When you have set 'Avoid' the processing path is shown in the drawing. Start the simulation to check the cutting order; you may need to increase the number of waypoints.

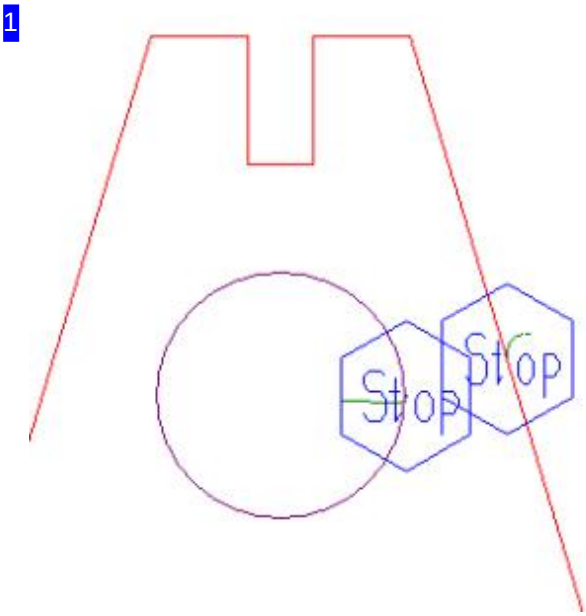
Changing The Cutting Direction And Adding Machine Stops.



for processing, *ncSchneid* offers the option to change a cutting path's cutting direction, and add as many machine stops as required.
When changing the direction, contour types are not changed, i.e. inside contours remain holes and outside contours remain outlines!

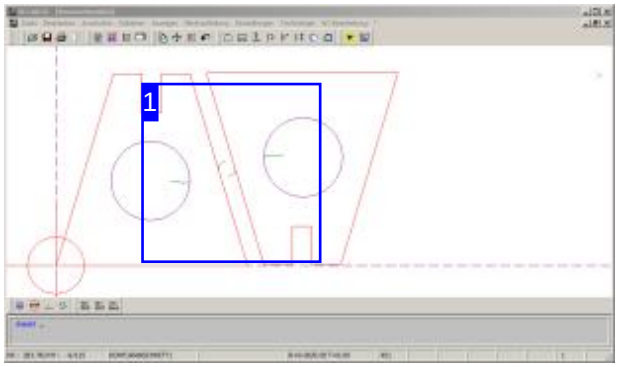
To add a machine stop to a cutting path, select the menu item 'M-Stops' or the corresponding quick-key in the icon bar .

To change the processing direction of a cutting path, select that function, then the required path. Double-click to repeat the command, and exit by pressing <ESC>.



You can use the 'M-Stop' function to add multiple stops on a cutting path, in case you need to take any other steps during production. Select the function then click the cutting path to add a stop to. Double-click to repeat the command, and exit by pressing <ESC>. The same process is used for deleting machine stops.

Connecting Parts Using A Chain Cut.

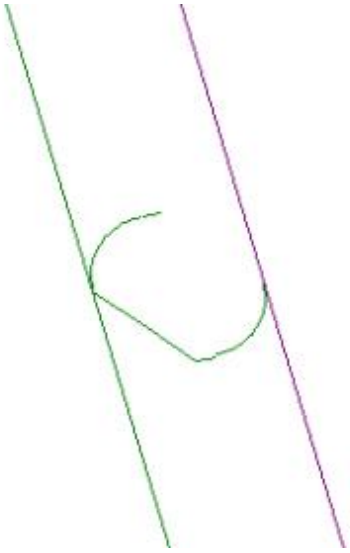


To add a chain cut, select the function then click the first part to be joined. Next mark the next part to be cut, their lead-ins will be joined and you can drag a polygon line across the sheet as the cutting torch should run. Click the mouse to add a new waypoint, and exit by pressing <ESC>.

To delete a connecting line, select the menu item 'Lead-ins', 'Delete chain cut'.

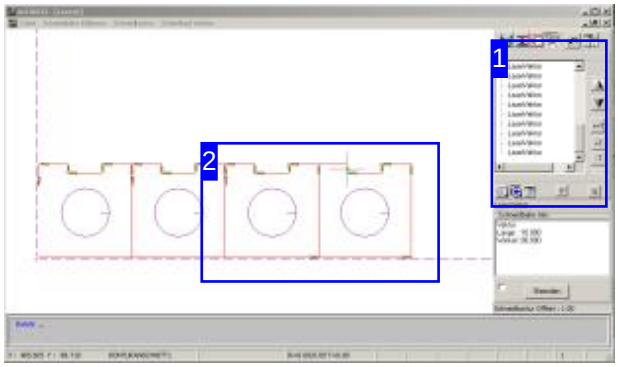
For some cutting tasks it is advantageous to keep the number of pre-cuts as low as possible. *ncSchneid* offers the functionality to cut multiple parts in a chain with one lead-in.

1



In this example the two parts are linked with a direct chain cut. The lead-out of the left hand part leads directly into the lead-in of the right hand part. Start the simulation to check the cutting path matches tour expectation. If you see in the simulation, that the chain is not cut in the correct sequence you can sort the chain new with the command '467 4'. Identify the starting workpiece after you have entered the command.

Optimal Material Usage With Common Separating Cuts.

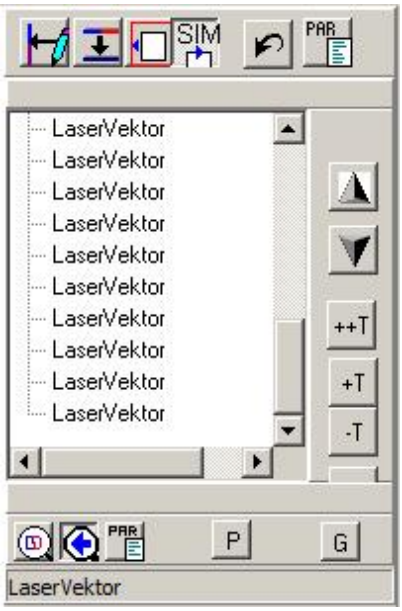


If you are cutting multiple identical parts with mostly straight edges then common separating cuts can help you achieve maximum material utilization. Common separating cuts require careful nest layout and entry of the cutting torch width, but give the best possible material usage.

Common separating cuts have certain prerequisites in order to arrange the parts optimally:

- Parts must contain combinable elements.
- Parts may not have any lead-ins. Common lead-ins are set for all parts.
- The parts' offset must be half of the cutting torch width. The offset is predefined when using common separating cuts; all lead-ins are automatically deleted.
- For optimal usage, mirroring must be permitted.

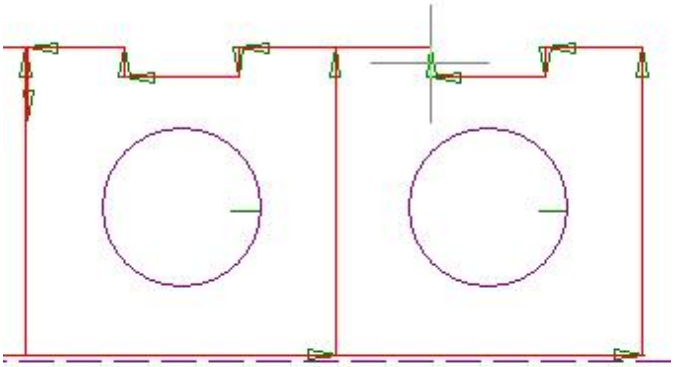
1



- First set the cutting contour offset for common separating cuts to half the cutting torch width. This function automatically deletes all outside contour lead-ins.
- Then copy the part as required and place it in the 'Bump' mode with as many flat edges against each other as possible. Rotate the parts as required, consider mirroring too! To 'snap' contours <ALT> + right-click.
- To allocate a common separating cut the shared cut lines, switch to the 'Edit cutting paths' screen to join the straight lines; click the second button in the button panel.
- Check the use of common separating cuts by clicking the fourth button in the simulation.
- Click the second button in the footer panel to overlay the cutting directions onto the preview. Select the contour in the list and

navigate using the arrows to the right.

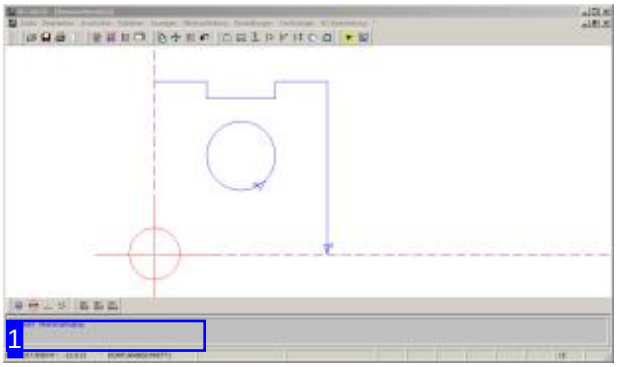
2



In this example the cutting directions are shown in the preview pane. The cutting sequences can be checked in the simulation.

Macros

Execute Command Sequences As Macros.



Select the menu item Macro routines to open the dialog for working with macros. You can then create and define a start macro and in assemble general cutting macros.

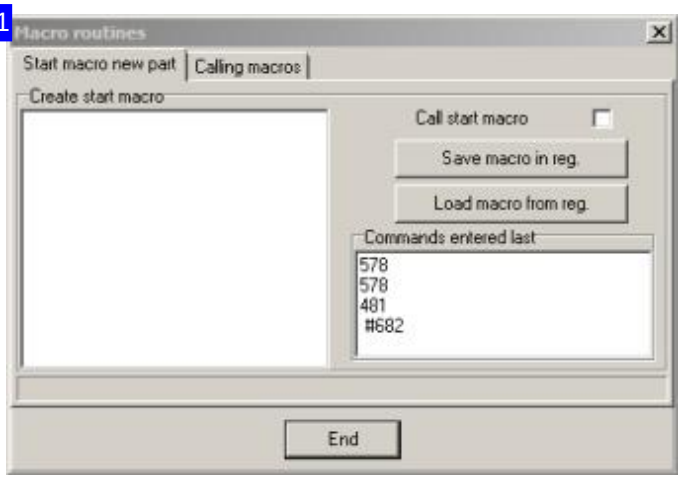
When creating a macro you must first give it a name to be saved under for future use. All cut part macros have the extension '.SAK' and are saved by default to the 'MAK' folder.

Processes entered as commands through the command line [1] which come up time and again can be grouped and saved under a macro name for simplified future use to improve productivity and reduce the opportunity for errors.

1

Functions can be called from the command line using the corresponding command name or number. Such commands can also be given using macro names which execute several preset commands sequentially.
Custom macros can be created however you require to optimize your workflow.

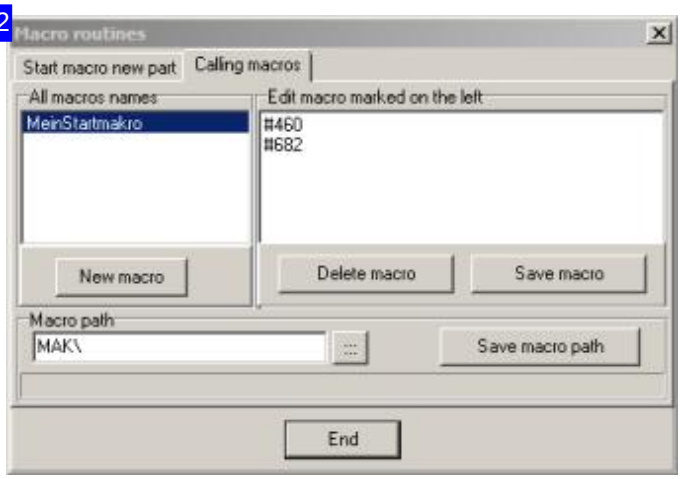
11



Creating a start macro:
Assemble the required commands for your start macro in the left text field. The macro is saved to the registry as a start macro in the right-hand pane. To check the registry's current start macro, load it through the text field. Mark the corresponding checkbox if the start macro should be called automatically.

The 'Last executed commands' pane lists all functions recently used. You can take these as the basis for your start macro in the left-hand pane.

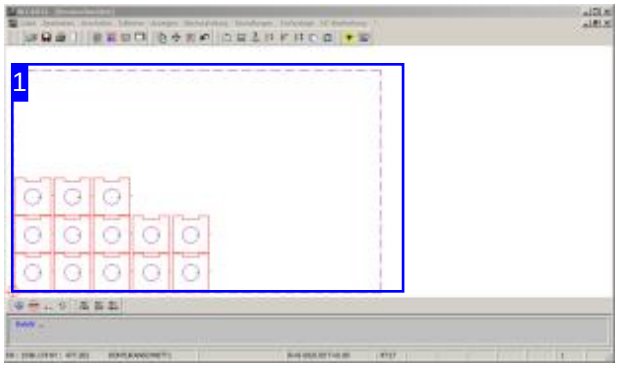
12



Use this tab to create as many macros as you require to simplify your workflow. First create a new macro in the list to the left by clicking 'New macro'. Enter the macro's name; it should be memorable and short enough that you will remember and use it!
Assemble the required commands in the right-hand pane and save the macro. The default save path is the 'MAK' folder, this can be edited if you wish.

Offcuts

Save Spare Material As An Offcut.



Once the required parts have been cut from a sheet, remaining material can be cut into one or more offcuts with one or more separating cuts.
Offcuts are automatically cut to fixed patterns, but can also be saved with complex shapes as required.

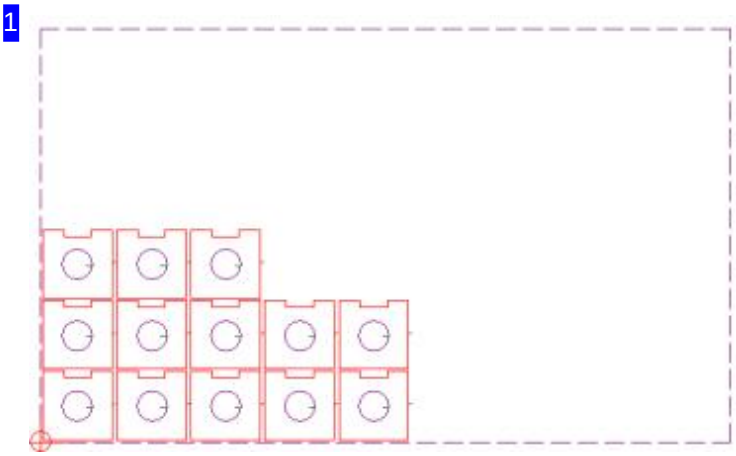
Create offcuts under the menu 'Rest Sheet'. Select 'Separeting Cut', 'Create' to open a dialog for automatic or manual definition of the required separating cuts.

The menu selection allows you to view the offcut and if the preview doesn't meet your requirements, delete it. Select the menu item and identify the separating cut.

If the result previewed is OK, save the offcut for future use.

If the job management module in *ncSchneid* is used, the offcut will automatically be added to your materials database upon completion.

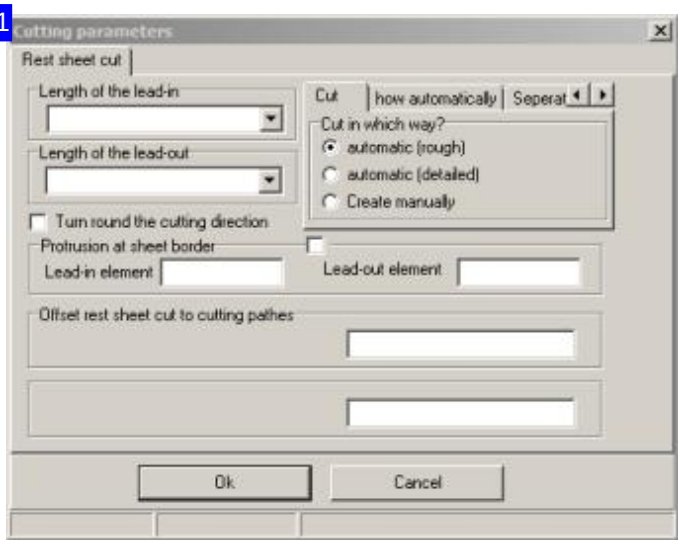
You can set any separation cuts for simple sheet metal blanks. Select 'Single separation cuts' in the 'Remnant sheet'-menu, to enter the settings in an special dialog.



The preview [1] shows the required parts laid out on the selected sheet. After cutting there will be leftover material which, dependent on the parts, will consist of an offcut grid and a margin. These spare areas can be removed using separating cuts.

You can use a nesting plan completely to make a remnant sheet. Select the menu-item 'Use nesting plan for remnant sheet..' and the contours will be inverted to sheet-outlines. A dialog appears, to acknowledge the usage.

11

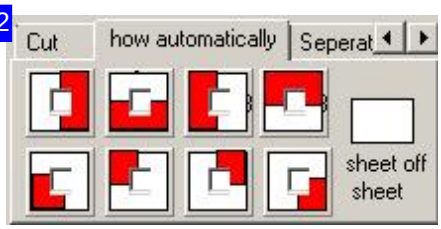


This dialog allows you to create offcut sheets automatically or manually. You have to enter the separating cut's lead-in and out lengths and set the offset from the existing cutting paths. Enter the cutting beam width. Lead-in excess can be outside the margins if required.

The tabs in the top-right pane are used to define the separation type; auto or manual. The second tab allows you to define the occupied space, the third tab the separating cut type if your machine requires this.

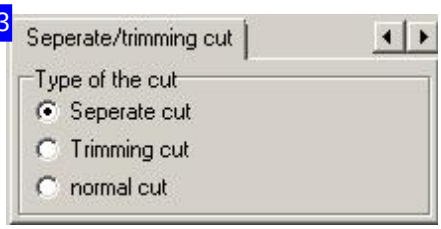
Click 'Apply' to show the separating cuts. If you selected 'Automatic coarse' as a continuous rectangle, or if 'Automatic fine' as a dashed line enclosing contour. Follow the subsequent prompts in the command line to create your offcut.

12



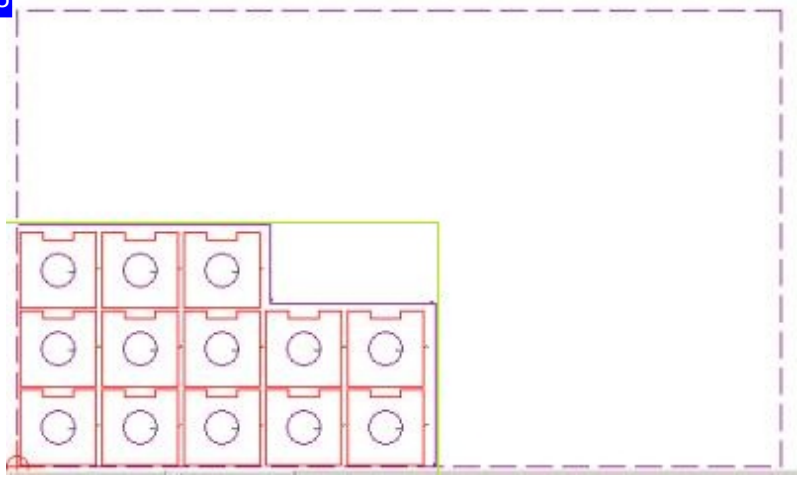
Mark the area occupied by your cutting paths in this tab, as highlighted in red.

13



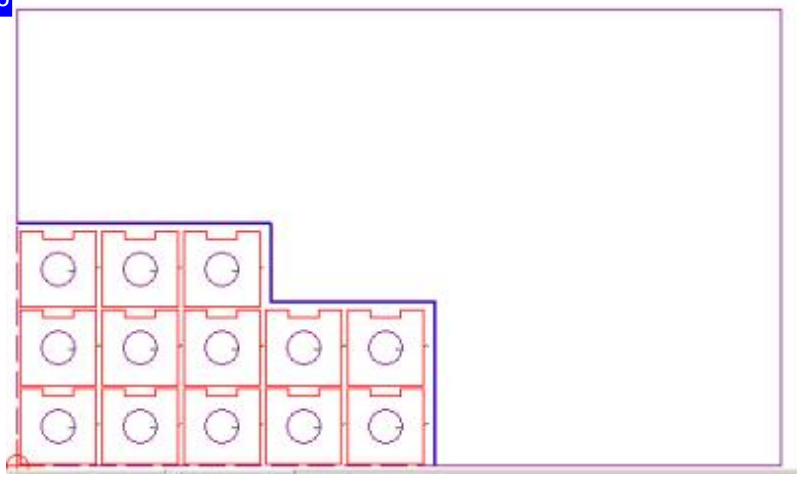
Depending on your post-processor you may need to specify the type of cut - mark the corresponding radio box.

15



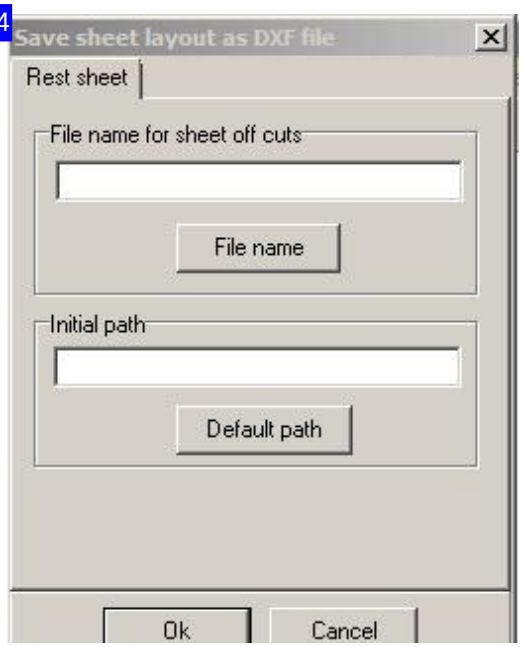
Once the definition of separating cuts is initialized, the next steps will be shown in the command line. Identify the elements to use as the start and end elements. These elements will then be extended to the sheet edges and that line set as the separating cut line. You can preview the offcut through the menu.
The inner line here shows the suggested cut using the method 'automatic fine' ; the green line shows the suggestion for 'automatic coarse'.

16



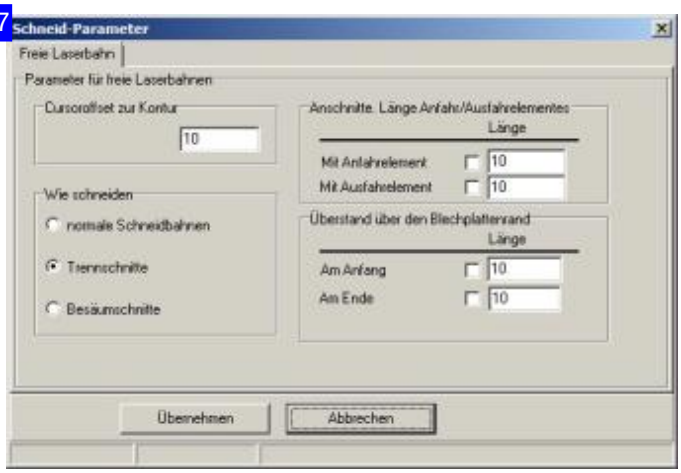
An offcut can be previewed [1] through the menu item 'Offcuts', 'Display' . To delete an offcut, select the corresponding menu item and click to identify the separating cut.

14



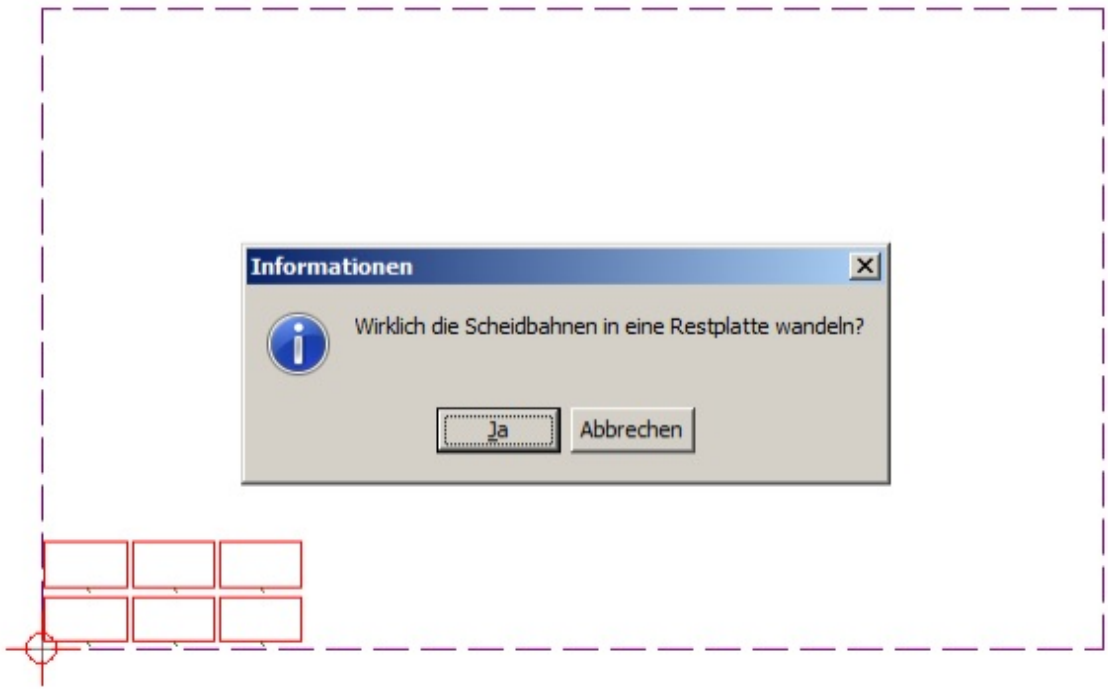
Completed offcuts are saved as DXF files for future use. Enter an identifiable filename and save path. Click 'Apply' to save your offcut sheet.

17



This dialog includes all settings for separating cuts. You define the lengths and the cutting type for lead-ins and outs in the frame 'Lead-ins' in the upper right corner. Select all the checkboxes you want to be executed. If you use overhangs for lead-ins, enter the values in the lower rigth frame. Mark the checkboxes here accordingly.
You can choose different cutting types, depending on your processor. The frame 'How to cut' shows you the different types for choice.
Enter an offset in the the upper left frame as safty distance to the next part's outline.
Click on 'apply' for starting the function and take care of the commands in the command-line for processing. You can apply separating cuts easily at the suggested ledger lines. Click on <ESC> to end the function.

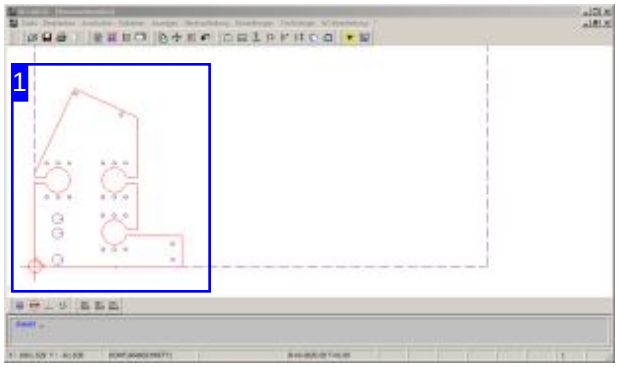
18



As a special feature, you can change a existing nesting plan completely in a remnant sheet. Using the menu-entry 'Rest Sheets', 'Use Nesting Plan for Rest Sheet..', will invert the outlines and take them for outcuts. A dialog opens where you have to confirm the change.
With confirmation the process closes and the outlines will change in outcuts.
You can use the procedure also for existing nesting plans (LAW-Files).
You can save and retrieve the generated remnant sheet arbitrary.

Don't forget to save the new remnant sheet as DXF-File for further use.

Use Freehand Separating Cuts To Create Polygonal Offcuts.

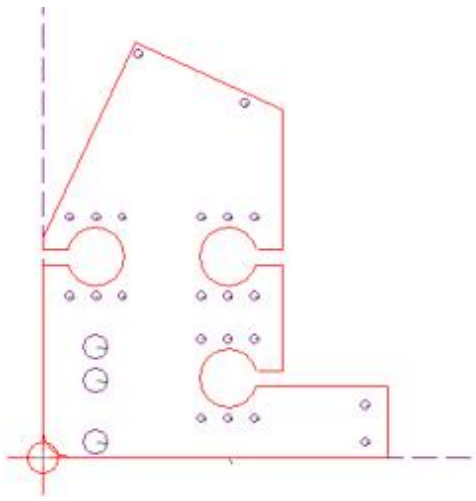


To create freehand separating cuts or idle paths, select the corresponding menu item under 'Edit', 'Individual editing' .
The process for idle paths and separating cuts is the same, only the selected menu item varies.

You can use freehand separating cuts to create a polygonal offcut from a sheet which has an uneven distribution of parts.

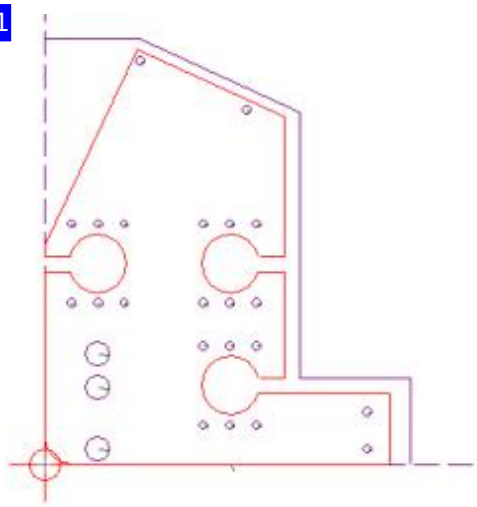
Freehand separating cuts can also be useful to divide oddly shaped internal holes, such that they fall from the sheet easier with less risk of deforming the part.

1



The preview [1] shows the cutting paths on the selected sheet. Select the required function and create a polygon line group using the mouse. A dialog will appear after your first line is drawn to choose if it should be a lead-in, then create your polygon group as normal, clicking at each waypoint. End the function by pressing <Q>. A dialog will appear to ask if the last element should be a lead-out. Tech cuts can also be created automatically by selecting the corresponding checkbox. The separation line is then added to the preview.

11



Here the separating cut has been added to the preview.

Computation

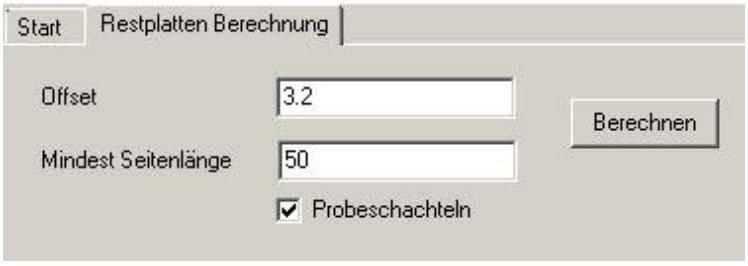
If you have entered the command, you get the shown window. Choose 'Remnant Sheet Computation' [1] and you will get another tab sheet with the same name.

Command <700 15>: computation of offcuts

This command allows you displaying all areas not used, entering settings for remnant sheets to cut remnant sheets.
You need the DLLs: 'IBEBOOP.dll' and 'CNCNEST.DLL' for execution and both have to be stored in the program folder.

Please notice, that remnant sheets doesn't make sense in ncCAD32. Remnant sheets are useful, when ncCAD32 works as technology module in SHEET METAL CENTER Plus. There, remnant sheets can be stored in the sheets database.

1

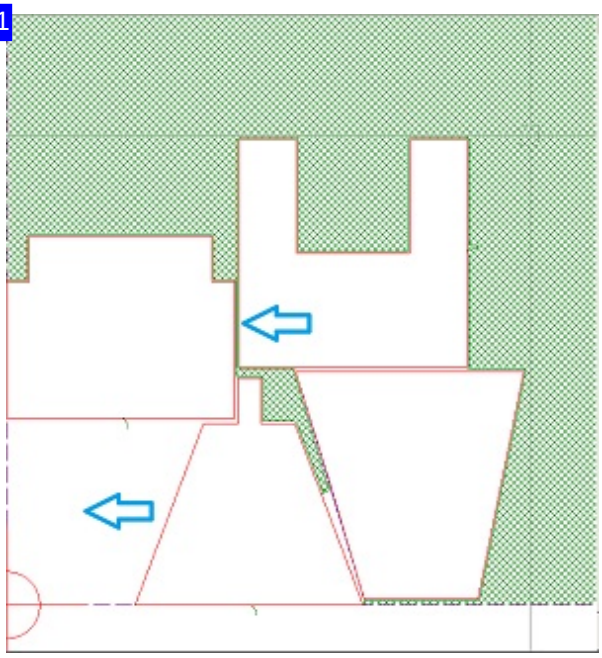


Click on tab [Remnant Sheet Computation]. Enter here the settings for remnant sheet computation. The offset determines the gap between remnant piece and workpiece. This value defines a kind of safety distance to the workpieces and maybe affects the number of remnant pieces e.g. reduces them.

The minimum length of an edge gives the smallest side of a remnant sheet, to be one. The comparison is done with the part's enclosing, rectangle box. If you mark the checkbox 'Test nesting', nesting is

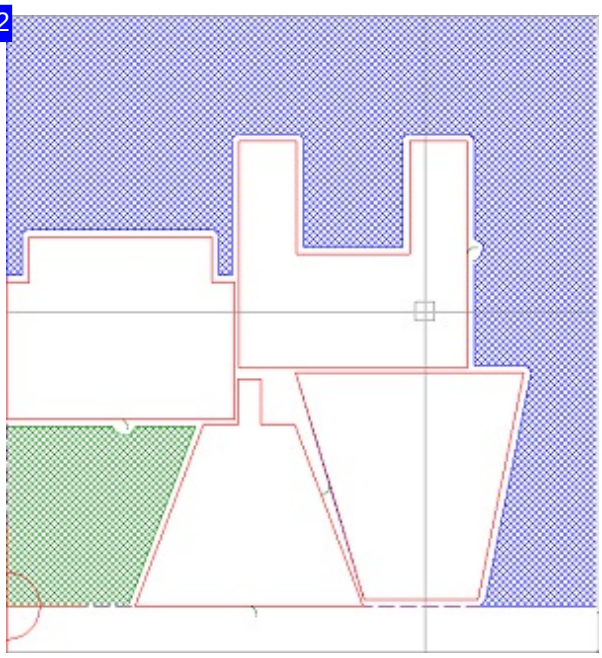
tryed with a virtual, square part with an edge of the minimum length into the remnant piece. The remnant sheet is determinated if this is possible. Click on [Computation] for execution.

11



This example shows you a remnant sheet (hatched area) with two small bridges (arrow on the top and bottom right). You can separate them with an additional cut, or simple suppress them by changing the offset value. The remnant part in the left corner (arrow) is a piece, that is not designated as a remnant part. If you change the minimum size, the result looks more different.

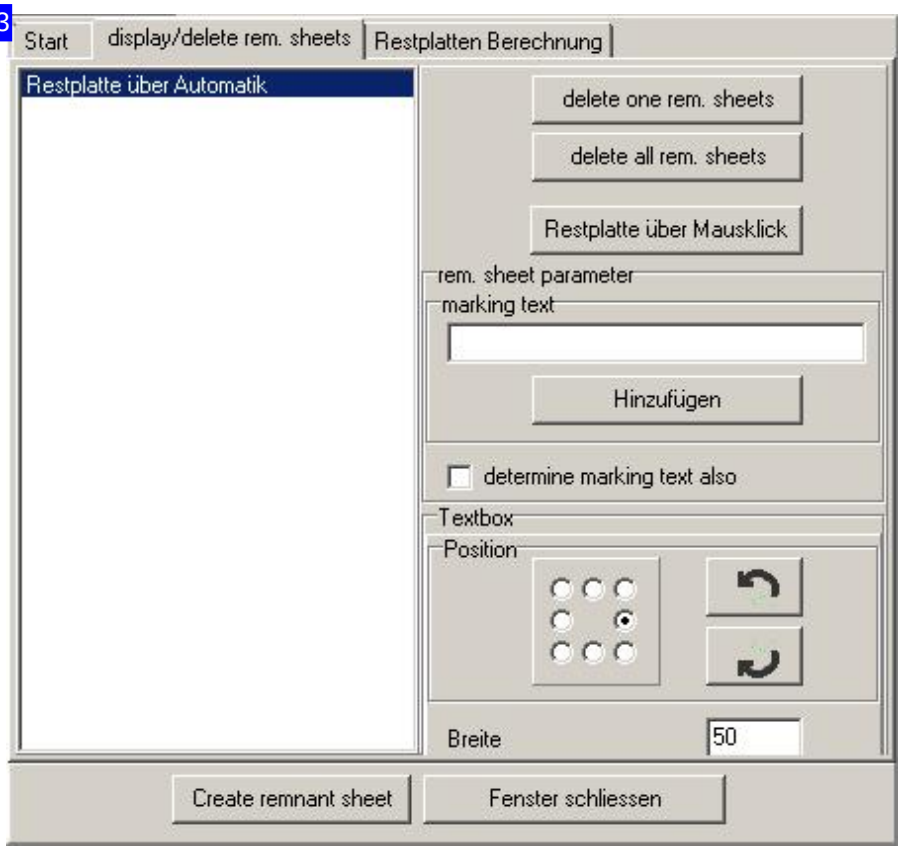
12



You get two proposals for remnant sheets, the green and blue hatched region.

The new tab sheet shows you the result of computation. You see all the remnant sheets in the list left for single selection and editing.

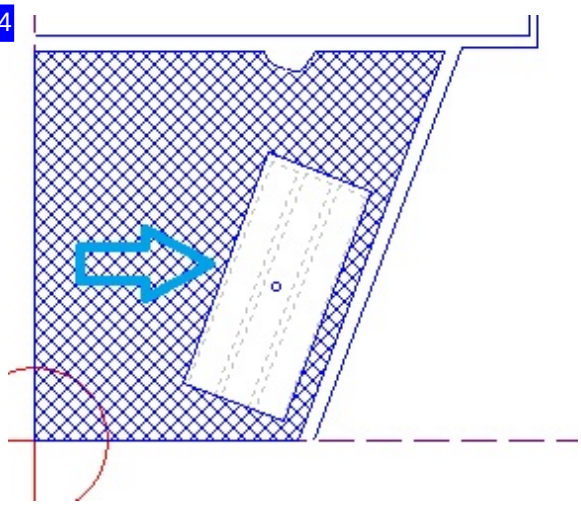
13



The list shows you all the remnant sheets. Use the buttons top right for deleting list entries or identify entries with a mouse click.

You can define text boxes at the bottom and position it on the remnant sheet. Use the rotate buttons to position the text box at the outline and rotate it along the outline.

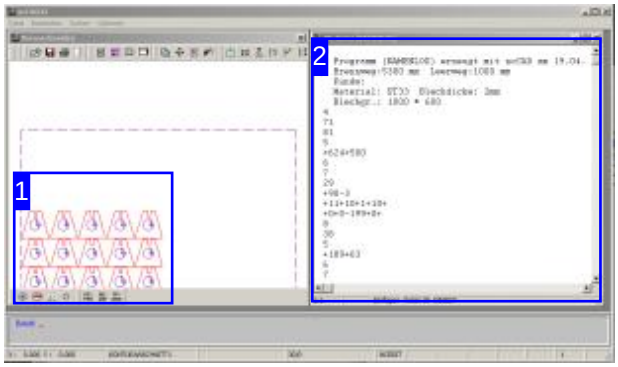
14



You can define a text box for any remnant sheet and position it directly or rotate it along the edge.
A text box looks like that, that's rotated with the buttons.

NC Programs

Convert Cutting Paths To Nc Programs.



The NC editor contains all the function required to generate an NC program. The process is dependent on and therefore may be limited by your post-processor. Subroutines are only possible if your machine supports them.

NC programs generally take the form of a succession of commands which are executed sequentially. If there are lots of parts on your sheet this can result in large NC programs. The file size for such sheets can be greatly reduced using subroutines. When creating subroutines, identical cutting paths are reduced to command packets which can be called repeatedly as required using a single command.

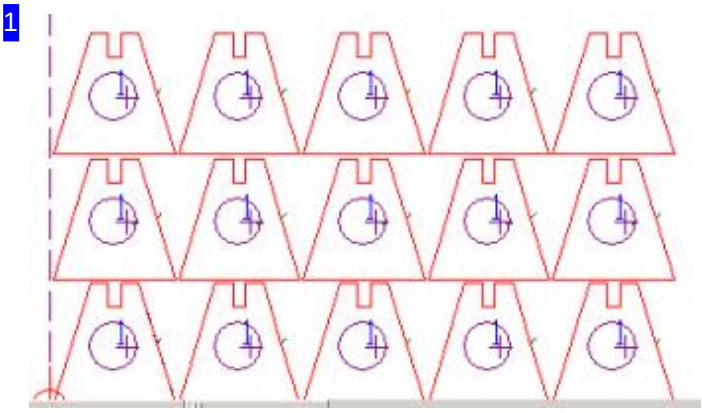
Your NC code is generated automatically with a simple menu command. Select 'Create NC Code' to open a dialog where you can enter your program number.

The NC program will be created automatically according to the settings made through your post-processor.

If your machine is able to process subroutines then create these first using the corresponding menu command.

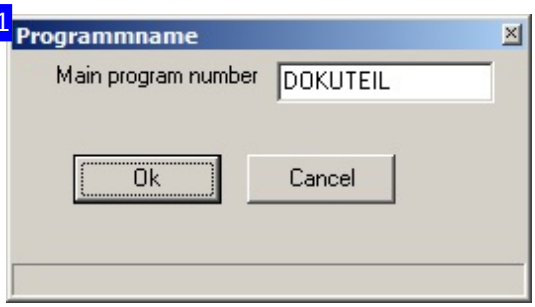
You can subsequently change your NC code, e.g. to add settings required by a specific machine. In most cases the NC code can be executed without further modification.
You can access the editor through the menu item 'Edit NC Code' - a dialog containing the NC code lines will open next to the preview [1].

Don't forget to save your NC program! The default file type is '.CNC' and the save path is the folder 'NCCAD32/CNC'.
You can use the menu item 'Save NC code as' to open a file manager and change the save path.



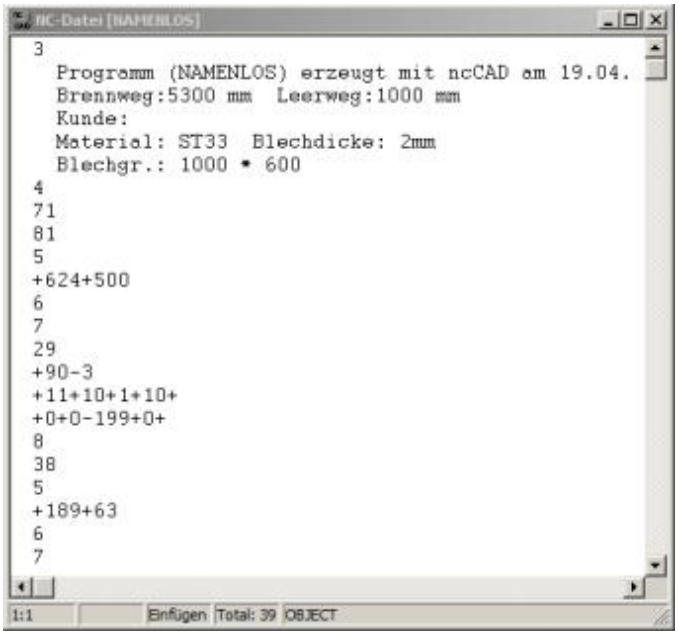
The preview [1] shows the cutting paths on the selected sheet. If you have created subroutines then identical cutting paths will be grouped numerically, here shown using the number '1'.

11



This dialog allows you to enter a name or number for your NC program. Further fields may be offered in conjunction with your post-processor, if this is the case you will find more information in the handbook for your machine or control panel.

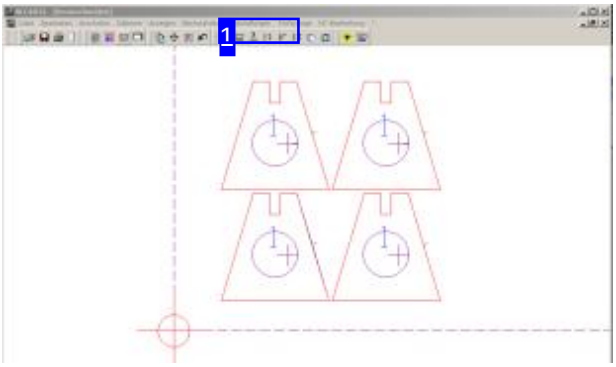
2



This dialog shows the NC code generated in conjunction with your post-processor in an editor window (here ESSI). You can make any changes required.

Configuration

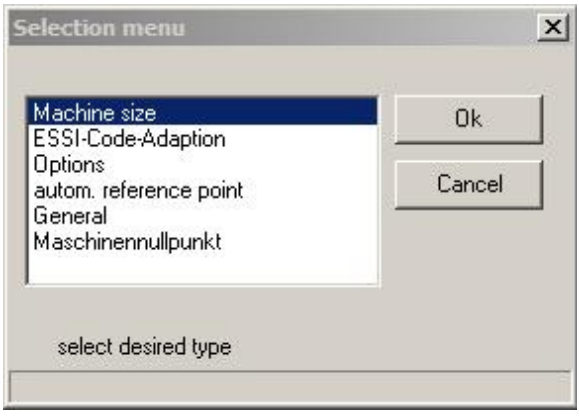
Basic Settings For Programs And Machines.



This screen allows you to make basic settings for the software's correct configuration so that it can operate your machine and work with your post-processor successfully. This data entered on this page is important towards obtaining properly formatted NC programs.

The configuration settings are accessed through the menu under 'Settings', 'Configuration'. When clicked, a dialog to select the configuration dialogs. The configuration dialogs' contents are post-processor dependent, thus settings demonstrated here may be missing from your setup, and you may have others present which are not mentioned. The technical values (machine parameters) which must be entered here can be found in the handbook for your machine or control panel.

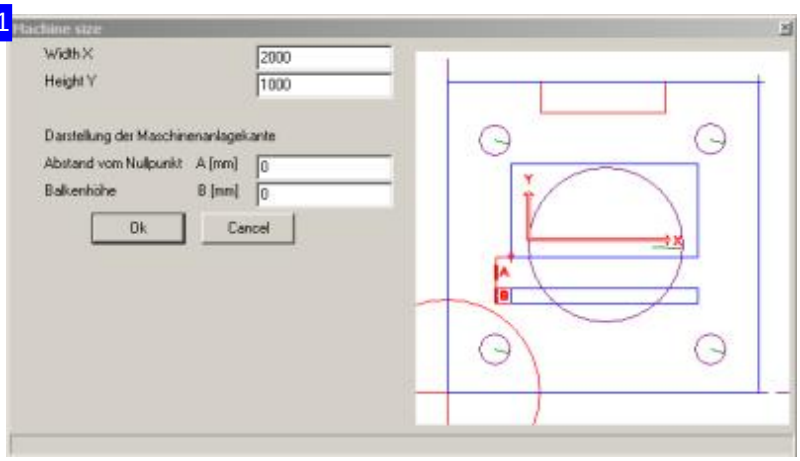
1



This dialog offers a selection of configuration fields, each of which open themed configuration submenus.

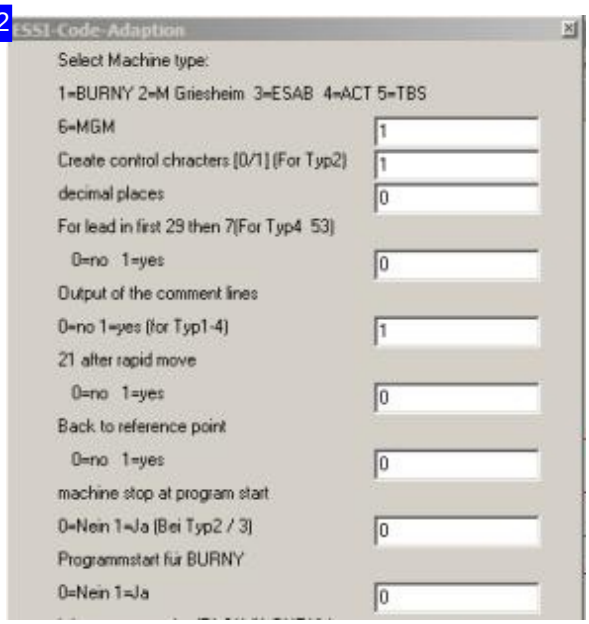
- Machine size
- Essi Code adaptation
- Options
- Auto zero point
- General
- Machine zero point

11



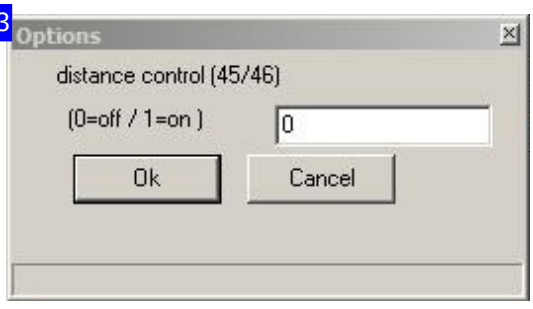
Machine size - enter the size of your machine's table and lay-on edge if present here.

12



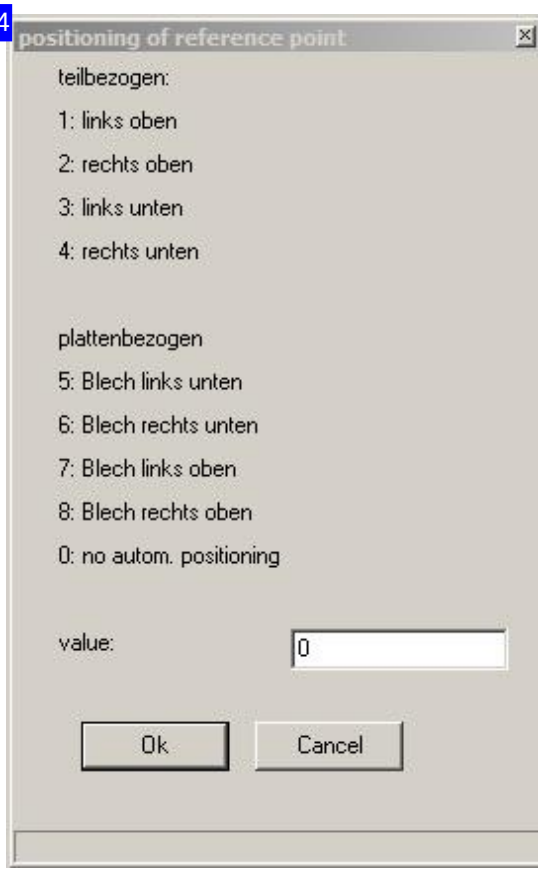
Essi Code adaptation - NC code is generated by default for an Essi post-processor. This dialog allows you to make default settings for some control panels.

13



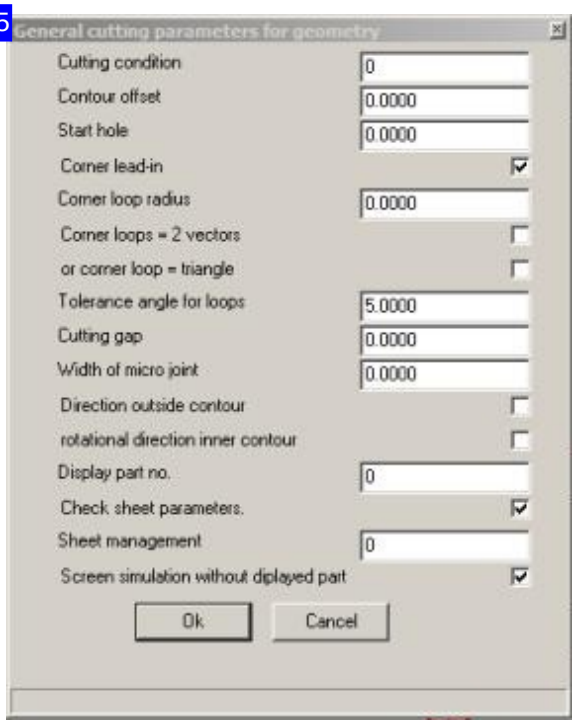
Options - activate the digital sampling control where fitted.

14



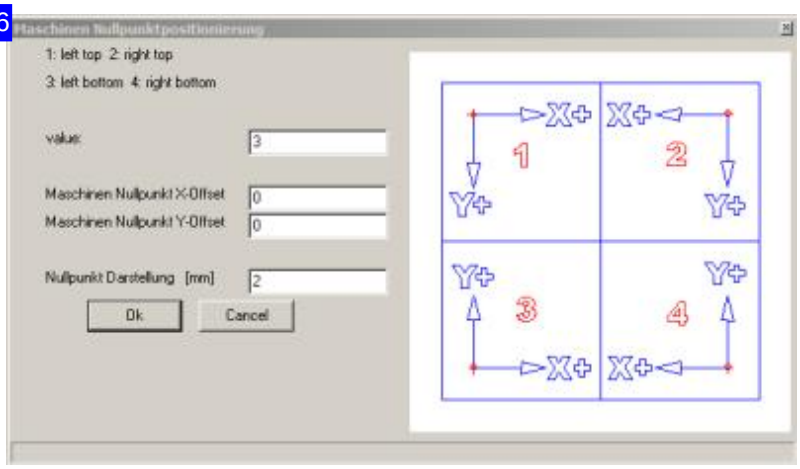
Auto zero point - enter the position of the zero reference point relative to parts or sheets as a code number.

15



General - use this dialog for settings relating to NC program generation and general program use.

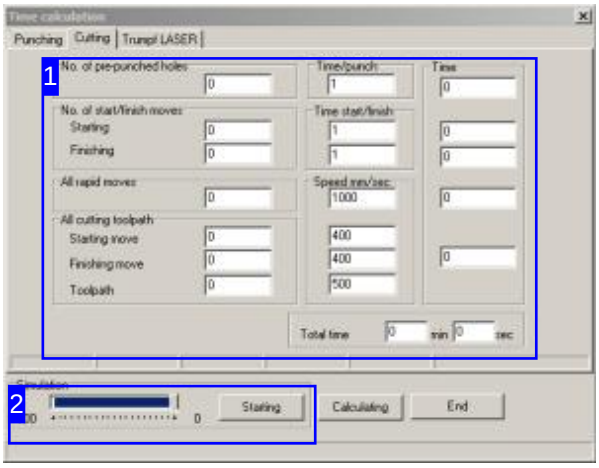
16



Machine zero point - Enter the position of the machine's zero reference point as a code number.

Time Calculation

Use The Simulation To Calculate The Cutting Time.



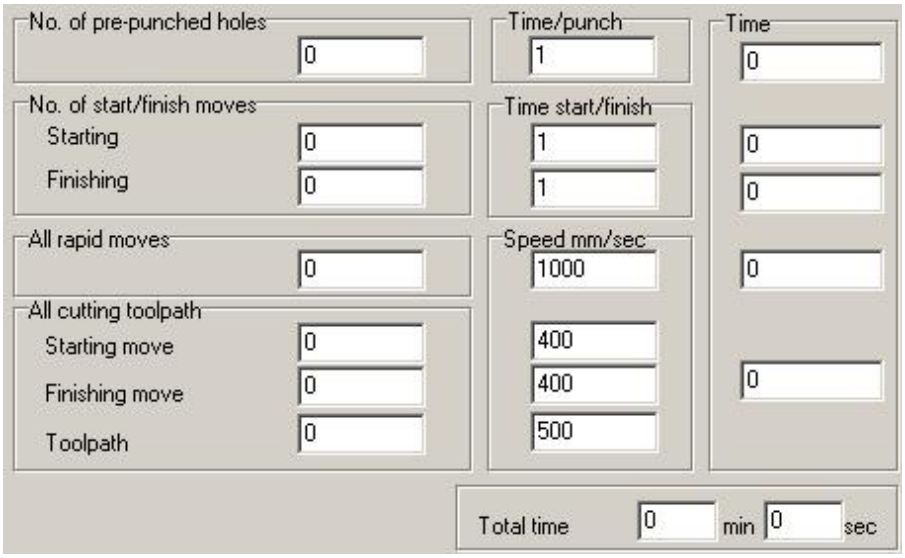
Select 'Settings','Time Calculation' to open a dialog for defining the speeds of various machine movements and fixed times for other processes in order to calculate the total processing time for your sheet.

Time calculation is dependent on many factors and can be only as precise as the machine information entered allows. The sheet is cut in simulation to obtain the data required to calculate the total time.

When cutting beveled edges, the process is more complex and it is not practical to obtain an exact time calculation since machine data such as hub times are not easily available.

More precise time calculations can be executed using the post-processor; results will then be displayed in a post-processor dialog.

1



Check the machine data in the 'Speed mm/sec' pane and correct as necessary. The times in the 'pre-drilling' and 'lead-ins' fields should be entered in seconds. The times in the left column are calculated by the simulated run-through.
Start the simulation; the total time is subsequently shown in the bottom-right pane.

2

Start the simulation here. The values calculated from this run-through are used for your time calculation. If everything looks right click 'Calculate' when the simulation is done and the values will be copied into the dialog's right-hand column.

Glossary

Bevels

Cut angled edges on your parts.

Angle 29
Attributes 53
Bridge 52
Create 50
Cutting head 28
Display 54
Dynamic 53
End angle 53
General 50
Markings 54
Negative 52
Offset 29
Options 50
Pass height 55
Positive 52
Post-processor 53
Separate 53
Transitions 29
Turn cutting head 31
Values 52

Circles

Round cutting paths.

Center punching 56
Create 30
Cutting 56
Marking 55
Punching 56
Radius 30
Start angle 30

Common Separating Cuts

PArquet layout of parts so more than one can be cut with one tool pass.

Cutting width 23
General 86
Joining 26
Lead-ins 87
Offset 35
Requirements 86

Components

The individual items on your nesting plan.

Convert 20
Identifier 7
Prepare 1

Cutting Bridges

Sections to be left out when cutting.

Create 11
General 44

Cutting Contours

The shapes which make up your parts.

Bevels 50
Closed 30
Cutting direction 11
Display 30
Info 15
Loops 47
Offset 23
Open 35
Parameters 31
Rounding 47
Selection 16
Separate 34
Sorting 78
Times 106
Turning direction 11

Cutting Contours

The cutting sequence for a part's contours.

Bevels 50
Cutting parameters 8
Kerf 23
Loops 47
Offset 87
Parameters 16
Processing direction 9
Roundings 47
Sorting 78

Cutting direction

Cutting by columns and rows.

General 62
Modify 84

Parameter 80
Sorting 79

Cutting Parameters
Parameters for individual contours.

Kerf 23
Lead-in speeds 23
Marking speeds 23
Overlaps 43
Selection 16
Speeds 23

Cutting Route
The route the machine should take when cutting your parts.

Create 20
Define 20
Delete all 20
Freehand 9
General 1
Load 19
Modify 25
Print 19
Save 19
Simulation 20
Time calculation 106

Cutting Sequence
The sequence in which true-shape nested parts should be cut.

Avoid 82
Cutting direction 87
General 77
Manual 82
Menu 12
Sorting 78

Cutting Technologies
The options for editing your part.

General 5

DXF File
DXF is a standardized format for saving CAD drawings.

Cutting paths as DXF 8
Export 2
Restplatten 17

Save 94

Estimates
Approximate processing time estimates for your job.

Time calculation 106

External contours
The outside shape of your part.

Avoid 82
Convert 22
Direction 1
Lead-ins 60
Machine stop 84
Manual sorting 82
Offset 69
Processing direction 84
Shared 86
Sorting 82

Geometric Points
Identifiable points on your contour's elements.

Search 4

Internal Contours
The shapes of holes in your parts.

Cutting direction 84
Cutting up 61
Lead-ins 61
Machine stop 84
Separating cuts 95
Shredding 9
Sorting 79

Lead-ins
Technology cuts - the lead-ins and lead-outs for your cutting tools.

Adapt 41
Add 43
Alignment 10
Apply 7
Automatic 20
Copy 41
Define global 59
Definition 59
Delete 60

Display global 58
Edit 41
Elements 59
Extend 43
Forms 59
Global Values 59
Hollow lead-in 59
Load 60
Manager 57
Manipulate 10
Move 41
Not automatic 58
On bridges 33
On corners 23
On Micro-bridges 47
Overlap 41
Parameter 44
Place 8
Quick-keys 20
Remove 87
Save 60
Split Cuts 94

Macro
Macros are command sequences which can be executed as groups with one command.

Automatic 90
Commands 90
Create 88
Execute 89
Names 88
Save 90
Start macro 90

NC Program
The code needed to operate your cutting machines.

Change 102
Create 100
Delete 17
Edit 102
General 100
Important 1
Menu 17
Names 102
Quick-keys 20
Save 100
Stops 85
Subroutines 100

Technologies 40

Nesting
Complex arrangements of parts to maximize your material utilization.

Auto nesting 19
Automatic nesting 70
Grid production 19
Manual 71
Menu 16
Nesting list 67
Open contours 37
Settings 7
True-shape nesting 70

Offuct Sheets
Spare material cut to size and saved in the database for future use.

Computation 97
General 90
Menu 17
Nesting plan 17
Offsets 73
Save 94
Separating cuts 93
Settings 98
Split cuts 94
Text boxes 100

Open Contours
Cutting paths which don't form a closed shape.

Nesting 37
Types 37

Parts
Components.

Allocation 62
Avoid 82
Bump mode 71
Chain cuts 85
Common separating cuts 86
Cursor position 74
Customer number 7
Cutting bridges 44
Delete 68
Disposal 13
General 1

Identifiers 7
Import 6
Information 7
Manipulate 70
Manual Save 71
Manual sorting 83
Marking 55
Micro bridges 44
Mirror 7
Move 69
Multi-torches 74
Nesting list 67
Nesting 70
Offset 35
Offsets 73
Only boxes 19
Part number 23
Positioning 71
Redraw 4
Removal 67
Rolling direction 7
Sequence 66
Sequence 77
Show all 4
Sorting 82
Step angle 7

Post-processor
Post-processors are the drivers for your cutting machines.

Attributes 35
Bevels 53
Functions 5
Important 1
NC program 100
Select 41
Technologies 40
User inputs 102

Program
List of individual commands worked sequentially to form a program.

NC program 100
Subroutines 100

Separating Cuts
Cuts for spare material and offcut sheets.

Common 86

Freehand 95
Offcut sheets 92

Settings
Configuration values.

Auto zero point 105
Machine size 104
NC program defaults 104
NC program extension 105
Post-processor data 104
Sampling control 104
Setup angle 104

Sheets
Sheets of metal stock from which your parts will be cut.

Define 20
Dimensions 24
Margin 73
Materials 25
Offcut offsets 73
Offcut sheets 90
Offcuts 17
Offsets 73
Part info 64
Separating cuts 95
Setup plan 17
Shredding 9

Simulation
A dry-run of your cutting tool's path around the sheet.

Checks 26
Cutting time 106
Speeds 27
Start 20
Time calculation 108

Texts
Text markings on parts and offcuts

Counter 57
Date 57
Entry 56
General 55
Move 57
Parameter 56

Time calculation
Program function for working out the machine's cutting time.

General 106
Idle paths 108
Lead-ins 108
Parameter 108
Pre-drilling 108
Speeds 108

True-shape Nesting
Collate and automatically nest variegated parts on one sheet.

Automatic 70
Introduction 16

Zero Points
Reference points for you machine when cutting.

Define new 4
Identify 3
Menu 3
Move 16
Reference point 0 3
Reference point 1 3
Reference 5

Index

A Adapt (Lead-ins) 41
Add (Lead-ins) 43
Alignment (Lead-ins) 10
Allocation (Parts) 62
Angle (Bevels) 29
Apply (Lead-ins) 7
Attributes (Bevels) 53
Attributes (Post-processor) 35
Auto nesting (Nesting) 19
Auto zero point (Settings) 105
Automatic (Lead-ins) 20
Automatic (Macro) 90
Automatic (True-shape Nesting) 70
Automatic nesting (Nesting) 70
Avoid (External contours) 82
Avoid (Cutting Sequence) 82
Avoid (Parts) 82

B Bevels (Post-processor) 53
Bevels (Cutting Contours) 50
Bevels (Cutting Contours) 50
Bridge (Bevels) 52
Bump mode (Parts) 71

C Center punching (Circles) 56
Chain cuts (Parts) 85
Change (NC Program) 102
Checks (Simulation) 26
Closed (Cutting Contours) 30
Commands (Macro) 90
Common (Separating Cuts) 86
Common separating cuts (Parts) 86
Computation (Offuct Sheets) 97
Convert (Components) 20
Convert (External contours) 22
Copy (Lead-ins) 41
Counter (Texts) 57
Create (Macro) 88
Create (Cutting Route) 20
Create (NC Program) 100
Create (Circles) 30
Create (Bevels) 50
Create (Cutting Bridges) 11
Cursor position (Parts) 74
Customer number (Parts) 7
Cutting (Circles) 56
Cutting bridges (Parts) 44

	Cutting direction (Cutting Contours)	11
	Cutting direction (Internal Contours)	84
	Cutting direction (Cutting Sequence)	87
	Cutting head (Bevels)	28
	Cutting parameters (Cutting Contours)	8
	Cutting paths as DXF (DXF File)	8
	Cutting time (Simulation)	106
	Cutting up (Internal Contours)	61
	Cutting width (Common Separating Cuts)	23
D	Date (Texts)	57
	Define (Sheets)	20
	Define (Cutting Route)	20
	Define global (Lead-ins)	59
	Define new (Zero Points)	4
	Definition (Lead-ins)	59
	Delete (Parts)	68
	Delete (NC Program)	17
	Delete (Lead-ins)	60
	Delete all (Cutting Route)	20
	Dimensions (Sheets)	24
	Direction (External contours)	1
	Display (Bevels)	54
	Display (Cutting Contours)	30
	Display global (Lead-ins)	58
	Disposal (Parts)	13
	Dynamic (Bevels)	53
E	Edit (NC Program)	102
	Edit (Lead-ins)	41
	Elements (Lead-ins)	59
	End angle (Bevels)	53
	Entry (Texts)	56
	Execute (Macro)	89
	Export (DXF File)	2
	Extend (Lead-ins)	43
F	Forms (Lead-ins)	59
	Freehand (Cutting Route)	9
	Freehand (Separating Cuts)	95
	Functions (Post-processor)	5
G	General (Common Separating Cuts)	86
	General (Cutting Technologies)	5
	General (Offuct Sheets)	90
	General (Cutting Route)	1
	General (Parts)	1
	General (Cutting Bridges)	44
	General (Cutting direction)	62
	General (Bevels)	50
	General (Texts)	55

	General (Time calculation)	106
	General (NC Program)	100
	General (Cutting Sequence)	77
	Global Values (Lead-ins)	59
	Grid production (Nesting)	19
H	Hollow lead-in (Lead-ins)	59
I	Identifier (Components)	7
	Identifiers (Parts)	7
	Identify (Zero Points)	3
	Idle paths (Time calculation)	108
	Import (Parts)	6
	Important (NC Program)	1
	Important (Post-processor)	1
	Info (Cutting Contours)	15
	Information (Parts)	7
	Introduction (True-shape Nesting)	16
J	Joining (Common Separating Cuts)	26
K	Kerf (Cutting Contours)	23
	Kerf (Cutting Parameters)	23
L	Lead-in speeds (Cutting Parameters)	23
	Lead-ins (Time calculation)	108
	Lead-ins (Internal Contours)	61
	Lead-ins (Common Separating Cuts)	87
	Lead-ins (External contours)	60
	Load (Lead-ins)	60
	Load (Cutting Route)	19
	Loops (Cutting Contours)	47
	Loops (Cutting Contours)	47
M	Machine size (Settings)	104
	Machine stop (External contours)	84
	Machine stop (Internal Contours)	84
	Manager (Lead-ins)	57
	Manipulate (Parts)	70
	Manipulate (Lead-ins)	10
	Manual (Cutting Sequence)	82
	Manual (Nesting)	71
	Manual Save (Parts)	71
	Manual sorting (Parts)	83
	Manual sorting (External contours)	82
	Margin (Sheets)	73
	Marking (Circles)	55
	Marking (Parts)	55
	Marking speeds (Cutting Parameters)	23
	Markings (Bevels)	54
	Materials (Sheets)	25

	Menu (Offuct Sheets)	17
	Menu (Nesting)	16
	Menu (Cutting Sequence)	12
	Menu (NC Program)	17
	Menu (Zero Points)	3
	Micro bridges (Parts)	44
	Mirror (Parts)	7
	Modify (Cutting Route)	25
	Modify (Cutting direction)	84
	Move (Texts)	57
	Move (Lead-ins)	41
	Move (Parts)	69
	Move (Zero Points)	16
	Multi-torches (Parts)	74
N	Names (Macro)	88
	Names (NC Program)	102
	NC program (Post-processor)	100
	NC program (Program)	100
	NC program defaults (Settings)	104
	NC program extension (Settings)	105
	Negative (Bevels)	52
	Nesting (Parts)	70
	Nesting (Open Contours)	37
	Nesting list (Parts)	67
	Nesting list (Nesting)	67
	Nesting plan (Offuct Sheets)	17
	Not automatic (Lead-ins)	58
O	Offcut offsets (Sheets)	73
	Offcut sheets (Separating Cuts)	92
	Offcut sheets (Sheets)	90
	Offcuts (Sheets)	17
	Offset (External contours)	69
	Offset (Bevels)	29
	Offset (Cutting Contours)	87
	Offset (Cutting Contours)	23
	Offset (Common Separating Cuts)	35
	Offset (Parts)	35
	Offsets (Parts)	73
	Offsets (Sheets)	73
	Offsets (Offuct Sheets)	73
	On bridges (Lead-ins)	33
	On corners (Lead-ins)	23
	On Micro-bridges (Lead-ins)	47
	Only boxes (Parts)	19
	Open (Cutting Contours)	35
	Open contours (Nesting)	37
	Options (Bevels)	50
	Overlap (Lead-ins)	41

	Overlaps (Cutting Parameters)	43
P	Parameter (Texts)	56
	Parameter (Time calculation)	108
	Parameter (Cutting direction)	80
	Parameter (Lead-ins)	44
	Parameters (Cutting Contours)	16
	Parameters (Cutting Contours)	31
	Part info (Sheets)	64
	Part number (Parts)	23
	Pass height (Bevels)	55
	Place (Lead-ins)	8
	Positioning (Parts)	71
	Positive (Bevels)	52
	Post-processor (Bevels)	53
	Post-processor data (Settings)	104
	Pre-drilling (Time calculation)	108
	Prepare (Components)	1
	Print (Cutting Route)	19
	Processing direction (Cutting Contours)	9
	Processing direction (External contours)	84
	Punching (Circles)	56
Q	Quick-keys (Lead-ins)	20
	Quick-keys (NC Program)	20
R	Radius (Circles)	30
	Redraw (Parts)	4
	Reference (Zero Points)	5
	Reference point 0 (Zero Points)	3
	Reference point 1 (Zero Points)	3
	Removal (Parts)	67
	Remove (Lead-ins)	87
	Requirements (Common Separating Cuts)	86
	Restplatten (DXF File)	17
	Rolling direction (Parts)	7
	Rounding (Cutting Contours)	47
	Roundings (Cutting Contours)	47
S	Sampling control (Settings)	104
	Save (Offuct Sheets)	94
	Save (NC Program)	100
	Save (Lead-ins)	60
	Save (Macro)	90
	Save (Cutting Route)	19
	Save (DXF File)	94
	Search (Geometric Points)	4
	Select (Post-processor)	41
	Selection (Cutting Contours)	16
	Selection (Cutting Parameters)	16
	Separate (Cutting Contours)	34

	Separate (Bevels)	53
	Separating cuts (Sheets)	95
	Separating cuts (Internal Contours)	95
	Separating cuts (Offuct Sheets)	93
	Sequence (Parts)	77
	Sequence (Parts)	66
	Settings (Nesting)	7
	Settings (Offuct Sheets)	98
	Setup angle (Settings)	104
	Setup plan (Sheets)	17
	Shared (External contours)	86
	Show all (Parts)	4
	Shredding (Internal Contours)	9
	Shredding (Sheets)	9
	Simulation (Cutting Route)	20
	Sorting (Cutting Contours)	78
	Sorting (Internal Contours)	79
	Sorting (External contours)	82
	Sorting (Cutting Contours)	78
	Sorting (Cutting Sequence)	78
	Sorting (Parts)	82
	Sorting (Cutting direction)	79
	Speeds (Simulation)	27
	Speeds (Cutting Parameters)	23
	Speeds (Time calculation)	108
	Split Cuts (Lead-ins)	94
	Split cuts (Offuct Sheets)	94
	Start (Simulation)	20
	Start angle (Circles)	30
	Start macro (Macro)	90
	Step angle (Parts)	7
	Stops (NC Program)	85
	Subroutines (Program)	100
	Subroutines (NC Program)	100
T	Technologies (NC Program)	40
	Technologies (Post-processor)	40
	Text boxes (Offuct Sheets)	100
	Time calculation (Estimates)	106
	Time calculation (Simulation)	108
	Time calculation (Cutting Route)	106
	Times (Cutting Contours)	106
	Transitions (Bevels)	29
	True-shape nesting (Nesting)	70
	Turn cutting head (Bevels)	31
	Turning direction (Cutting Contours)	11
	Types (Open Contours)	37
U	User inputs (Post-processor)	102
V	Values (Bevels)	52

Table of Contents

Welcome to ncSchneid's help files 1
- General Tips for Navigation of the Program 1

The Cutting Module 2
- Preparing and Cutting Components. 2
- The Program's Most Important Menu Items. 6
- Entering Commands Using The Symbol Panel. 18

Cutting Paths 21
- Creating Cutting Paths. 21
- Editing A Cutting Path's Elements. 26
- Settings for Cutting Paths. 32
- Edit Cutting Path Attributes. 36
- Undo Your Last Command. 39

Technologies 41
- Define Your Machine's Technology Data. 41
- Edit Lead-Ins. 42
- Cut Parts With Micro-Bridges. 45
- Roundings and Loops. 48
- Adding Beveled Edges to Cutting Paths. 51
- Marking Parts. 56

Lead-in Manager 58
- Prepare Lead-Ins For Use. 58

Nesting Plans 63
- Organize Parts On A Sheet. 63
- Produce Grid Layouts. 65
- Collating Parts To A Nesting List. 68
- Automatic Nesting. 71
- Sorting Parts Onto The Sheet Manually. 72
- Cutting Paths With Multiple Torches. 75

Sorting and Combining 78
- Sorting The Processing Sequence. 78
- Manually Sort And Avoid Parts. 83
- Changing The Cutting Direction And Adding Machine Stops. 85
- Connecting Parts Using A Chain Cut. 86
- Optimal Material Usage With Common Separating Cuts. 87

Macros 89
- Execute Command Sequences As Macros. 89

Offcuts 91

Save Spare Material As An Offcut. 91
Use Freehand Separating Cuts To Create Polygonal Offcuts. 96
Computation 98

NC Programs 101
Convert Cutting Paths To Nc Programs. 101

Configuration 103
Basic Settings For Programs And Machines. 103

Time Calculation 107
Use The Simulation To Calculate The Cutting Time. 107

Glossary 109

Index 118