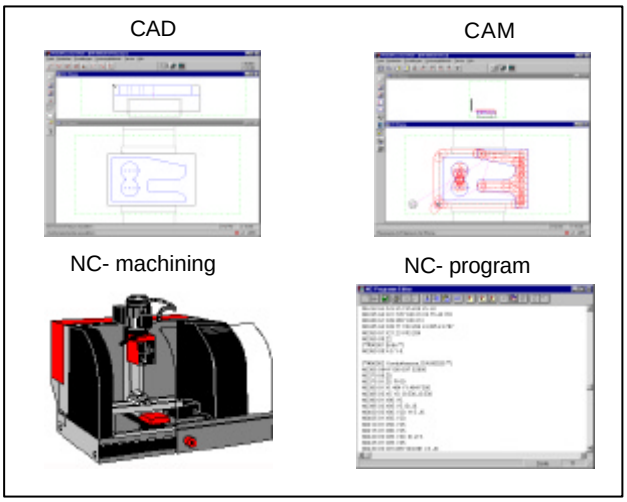
EMCO WinCAM Milling

Software description - CAD/CAM Programming system



Software description
EMCO WinCAM Milling

Ref.No. EN 1808 **Edition H2002-4**

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

Phone ++43-(0)62 45-891-0

Fax ++43-(0)62 45-86965

Internet: www.emco.at

E-Mail: service@emco.co.at

emco
industrial training systems

Preface

Hints for beginners

You can also start with the examples for copying:

CAD example	chapter C
CAM example	chapter D
NC example	chapter E

Note for language versions:

Please note that most of the screen shots shown are in German. The screens of the other language versions differ only in the text and not in the layout. In the English language version in the examples also the inputs are indicated in inch in brackets.

WinCAM concept

Today in practice programming of NC machines is carried out via automatic contour programming. However, comprehending the automatically generated NC program represents a necessity for the expert.

WinCAM offers a continuous didactic concept starting with the simple creation of workpiece contours in CAD mode via automatic interactive creation of the NC program in CAM mode up to the execution of the NC program at a connected NC machine. Due to the completely structured online help Win-CAM is suited optimally for training.

WinCAM scope of supply

- Simple graphic surface
- Creation of CAD workpiece contours
- Adjustable clamping devices and unmachined parts
- Automatic contour programming
- Cycle assistance
- Online-NC-program editor
- Status displays of the programmed machine status
- Tool library
- Import and export interfaces
- Measurement of clamping devices and tools
- Assistance of various control and machine types
- 3D simulation of machining
- Online NC-machine functions
- Online-help functions

Presupposed knowledge

For working with WinCAM the operation of MS Windows as well as handling and programming basic knowledge of the connected NC machine are a prerequisite. Therefore, if necessary, consult the respective manuals.

Training aims

WinCAM imparts the following training aims:

- Drawing and changing CAD contours
- Automatic interactive generating of NC programs
- Altering ready NC programs
- Comprehending connections of NC-machine settings and NC programming
- Remote control of an NC machine

Literary set-up

The WinCAM software description has the following structure:

- General bases of operation
- Examples CAD, CAM, NC
- Description of the menu lines
- CAD mode
- CAM mode
- NC mode

Due to its continuous user guidance WinCAM ist conceived in such a manner (online help and explanations in the status line) that the software description is to be used only in rare cases.

It would be best for you to start with looking through the general chapters A and B as well as the examples for copying. Then consult the other parts of this description, if necessary.

We, the team of the technical documentation of EMCO, hope that this manual will be a "real" help to you and that it will take as little of your time as possible.

Contents

Chapter A Bases

Screen set-up ... A1
Start of WinCAM ... A1
Window divisions ... A2
Menu lines ... A3
Erroneous inputs ... A3
Command symbols ... A4
Status messages ... A4

Chapter B Operating sequences

Definition of the CAD-window ... B1
Positioning the cursor ... B3
Command symbols ... B4
Selection of elements ... B4
Delete ... B5
Inputs in the status line ... B6
Inputs in the WinCAM window ... B6

Chapter C CAD example

Start ... C2
Settings of the CAD window ... C2
Drawing the workpiece contour ... C4
Save drawing ... C8
Convert drawing into DXF format ... C8
Print drawing ... C8

Chapter D CAM example

Specifications ... D1
CAD drawing ... D1
Technological data ... D1
Assemble toolholder ... D2
CAM settings ... D2
CAM machining ... D6
Saving the project ... D11
Printing the NC-program ... D11
3D- Simulation ... D11

Chapter E NC examples

Exercises for NC- manual operation ... E1
Exercises for NC- program operation ... E3
Exercise Define vice position ... E4
Exercise define unmachined part/clamping point ... E6
Exercise Tool measurement ... E7
Working off the CAD / CAM example ... E8

Chapter F Menu lines

Menu "File" ... F1
- New ... F1
- Open ... F1
- Save ... F1
- Save as ... F1
- Import ... F2
- Import drawing ... F2
- Import NC program ... F2
- Export ... F4
- Export drawing ... F4
- Export NC-program ... F4
- Print ... F6
- Information ... F7
- Quit ... F7

Menu "Edit" ... F8
- Undo ... F8
- CAD-Mark all ... F8
- Delete CAD element ... F8
- CAM program new ... F8
- Set CAM reference point ... F8
- Reset CAM reference point ... F9
- Start CAM line graphics ... F9
- Redraw ... F9
- Mark NC workpiece ... F9

Menu "Setup" ... F10
- Display ... F10
- Line ... F12
- Input ... F13
- Automatic contour recognition ... F13
- Define vice position ... F14
- Define unmachined part / clamping point ... F15
- Target control ... F16
- Target machine ... F16

Menu "Tool library" ... F17
- Tool library ... F17
- Tool holder ... F17
- Switch areas "Transfer" ... F18
- Switch area "Tool data" ... F18
- Tool data ... F18
- Switch area "Delete" ... F19
- Control area "3D-View" ... F19
- Switch area "Tool measurement" ... F20
- Tool measurement ... F20

Menu "Window" ... F21

Menu "Help" ... F22
- Using help ... F22
- Short instructions ... F22
- About WinCAM ... F22

Chapter G CAD commands

Point menu ... G1
- Set auxiliary point ... G1
- Set reference point ... G2
- Reset reference point ... G2
- Display design points of marked elements ... G3
- Suppress design point(s) ... G3

Line menu ... G4
- Draw line ... G4
- Poly line ... G4
- Parallel with point indication ... G5
- Parallel with distance ... G5
- Perpendicular ... G6
- Insert chamfer ... G6
- Tangent point - circle ... G7
- Tangent circle - circle ... G7

Circle menu ... G8
- Circle with center and radius ... G8
- Circle with circle point and center ... G8
- Circular arc with start, end and circle point ... G9
- Circular arc with start, end and radius ... G9
- Circular arc with start, end point and center ... G10
- Insert radius ... G10
- Rounding drawing elements ... G11

Modifying menu ... G13
- Shift drawing elements incrementally ... G13
- Shift drawing elements absolutely by means of reference point ... G14
- Copy drawing elements incrementally ... G15
- Rotate drawing elements ... G16
- Rotate and copy drawing elements ... G17
- Mirror drawing elements ... G18
- Trim corner ... G19
- Trim contour ... G20
- Cut elements ... G21

Text milling ... G22

Chapter H CAM Bases H1

WinCAM data transmission ... H1
Basic knowledge ... H1
Shift CAD - CAM ... H1
CAM Initialization ... H2
Command symbols ... H3
CAM machining ... H3
Defining the contour ... H4

Chapter I CAM commands

NC parameter on/off ... I1
Position display on/off ... I2
Change tool ... I2
Editor (change program) ... I3
- Operating the editor ... I3
- Editor ... I4
- Entries in the editor ... I5
- Significance of the editor ... I5
- Saving and loading the editor ... I6
- Deleting the whole editor ... I6
- Direct NC Input ... I6

Machining ... I7
- Pocket milling with islands ... I7
- WinCAM window "pocket milling with islands" ... I8
- Follow contour ... I9
- WinCAM window "contour milling" ... I9
- Circular pocket ... I11
- WinCAM window "circular pocket" ... I11
- Rectangular pocket ... I12
- WinCAM window "rectangular pocket" ... I12
- Drilling ... I13
- WinCAM window "Drilling cycle removal" ... I14
- WinCAM window "Deep hole drilling (chip breaking)" ... I14
- WinCAM window "Deep hole drilling (removal)" ... I15
- WinCAM window "Drilling - thread cutting" ... I15
- Movement in rapid motion in plane ... I16
- Movement in operating feed in plane ... I16
- Movement in rapid motion in feed direction ... I17
- Movement in operating feed in feed direction ... I17

Periphery ... I18
- Main spindle stop ... I18
- Main spindle on/left ... I18
- Main spindle on/right ... I19
- Clamping device close/open ... I19
- Chip blow-out on ... I19
- Automatic door open/close ... I19
- The way to ready simulation: ... I20

3D-simulation ... I20
- Operation of simulation ... I21
- Menu "File" ... I21
- Menu "Edit" ... I21
- Menu "View" ... I22
- Menu "Setup" ... I23
- Menu "Help" ... I26

Creation of Tool Models ... I27
- Structure of the Tool Library ... I27
- Structure of the Model File ... I28
- Model Commands ... I29
- 2D Modelling ... I30
- 3D Modelling ... I32
- Mathematic Functions ... I34
- Colour Constants ... I34
- Tool Definition ... I35
- Examples for Tool Modelling ... I36
- Settings in the File 3DVIEW.INI ... I43

Programmexport mit "exports840" ... I45

Chapter J NC basesNC data flow

Basic knowledge J1
Shift NC Mode J2
NC Initialization J2
Command symbols J3
NC error messages J3

Chapter K NC Online-commands

Periphery K1
Main spindle on/left K2
Main spindle stop K2
Main spindle on/right K2
Clamping device close/open K2
Chip blow-out on K2
Automatic door open/close K2
1maliger Werkzeugwechsel K2
AUX ON/OFF K2
Program execution K3
Start program (Cycle Start) K3
Stop program (NC-Stop) K3
Stop program (Reset) K4
Skip block (Skip) K4
Single block operation (Single) K4
Test run without main spindle(Dryrun) K4
Approach reference point K4
NC parameters on/off K5
Positions display on/off K5
Tool change K6
NC-keyboard assignment K7

Capter L Software installation

System Requirements L1
Software Installation L2
Notes for Network Installation L2
Settings of the Interface Board L3
PCCOM Master-Slave Setting L6
Networkcard L7

Chapter A Bases

WinCam Mill

Start icon

Start of WinCAM

In general at this point we will refer to the user philosophy of Windows NT 4.0 which will not be dealt with separately in this brochure. Please have a look at the respective handbooks of your operating system.
After effected windows installation of WinCAM lead the mouse pointer to the start icon of Win CAM and click twice (double click).

Menu line - help, short instructions

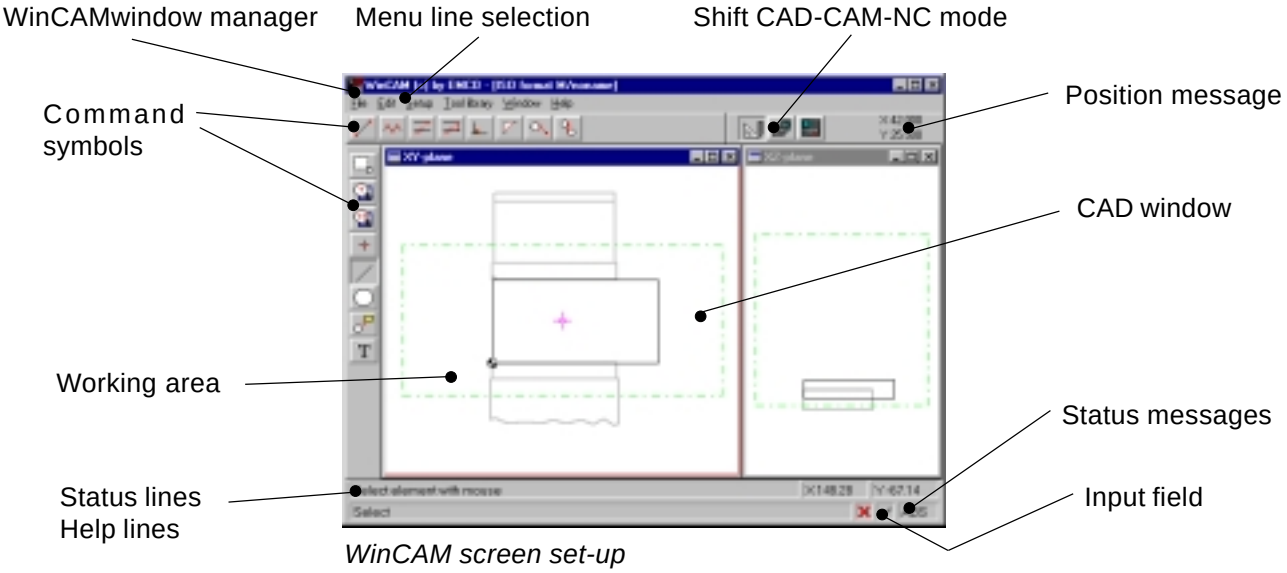
Help functions

WinCAM offers you the necessary assistance for every working step by various online help functions:

- The complete WinCAM help to be called up via the menu line. As usual in other windows programs, here, too, you can turn forward and backward in all help texts via an index.
- The WinCAM help field being displayed with certain functions which leads directly to the right help.
- Below at the left of the screen there is a help line of WinCAM which keeps you permanently informed. Here you see which inputs WinCAM expects from you or at which functions the mouse pointer is pointing.

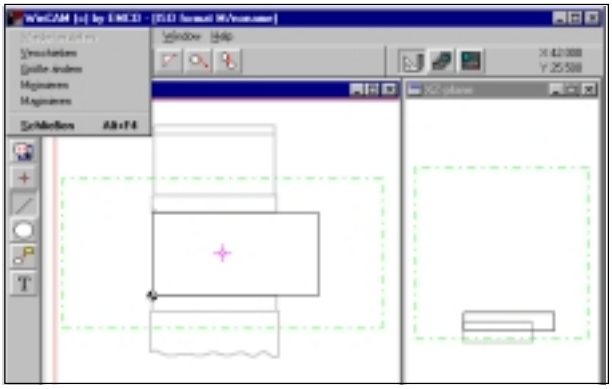
Screen set-up

The following picture shows the individual screen areas in the CAD part of WinCAM. These areas will be dealt with in detail on the next pages.



WinCAM screen set-up

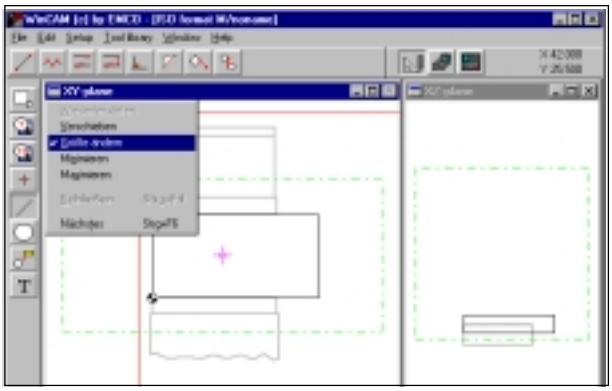
Window divisions



WinCAM main window

WinCAM main window

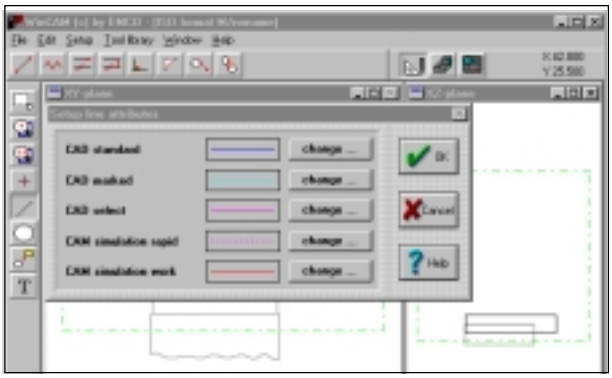
After the start WinCAM with its main window appears as full screen. In the working area of the main window additional active and passive windows are possible.



Window menu of the active window

Active windows

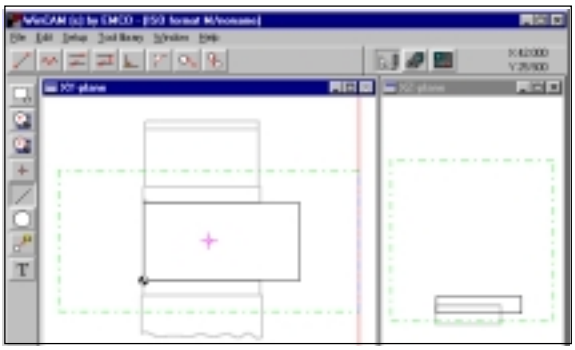
Active windows (e.g. XY-plane or XZ-plane) can be changed by means of the window system-menu field (frame top left in the window) and the mouse pointer like normal Windows windows (shift, enlarge, reduce, etc.).



Example Win CAM window

WinCAM windows

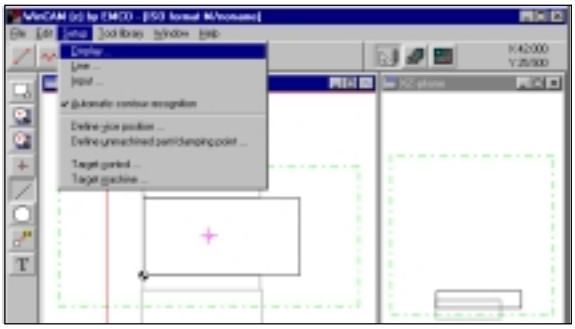
WinCAM windows are ready displayed windows which serve for information (e.g. info on WinCAM) or which can be opened for the input of determined parameters (e.g. line attributes). These windows can only be shifted.



Example menu name

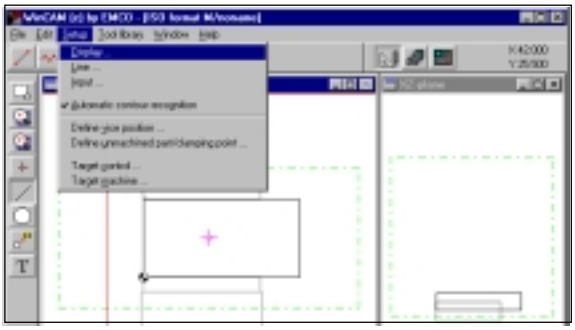
Menu lines

By clicking on a menu name a list of the commands which can be selected (standard font) and of those blocked at that time (diffuse, matrix font) is opened.



Example menu line

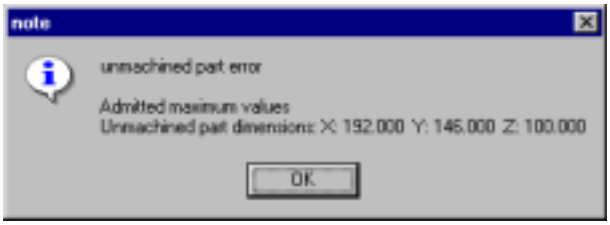
Menu names which have a direct effect as command get a hook as status display. The command is set with the hook.



Example commands with points

Commands marked by the following points open a WinCam window with further input possibilities.

Example WinCAM window



Example WinCAM - error message

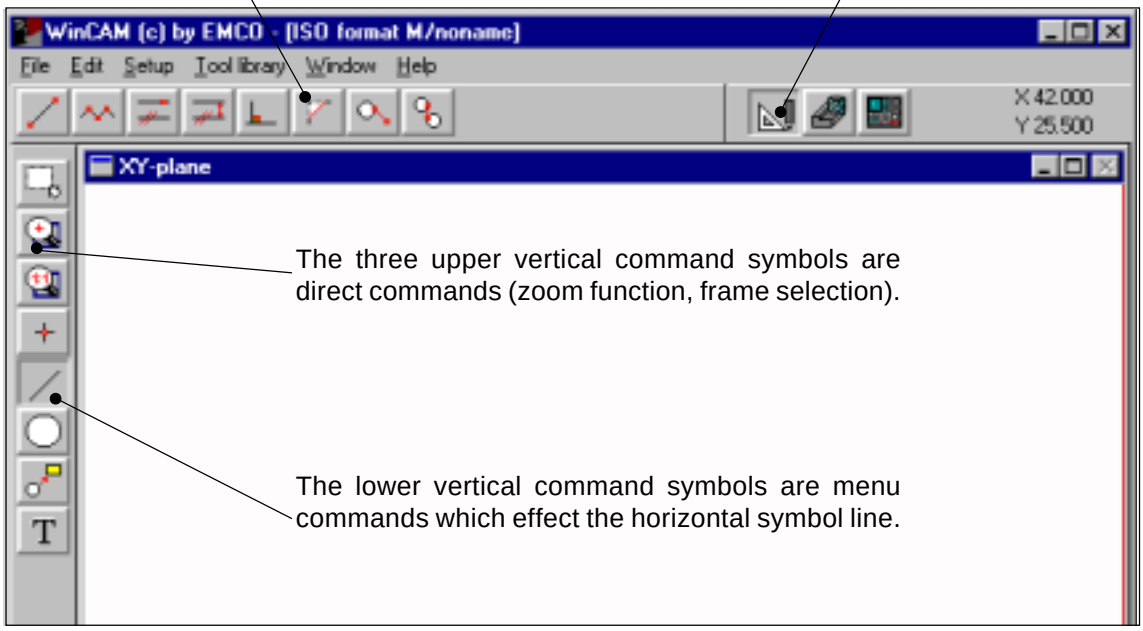
Erroneous inputs

Erroneous inputs are intercepted to a large extent by WinCAM. If WinCAM can make nothing with an input, an error message is displayed.

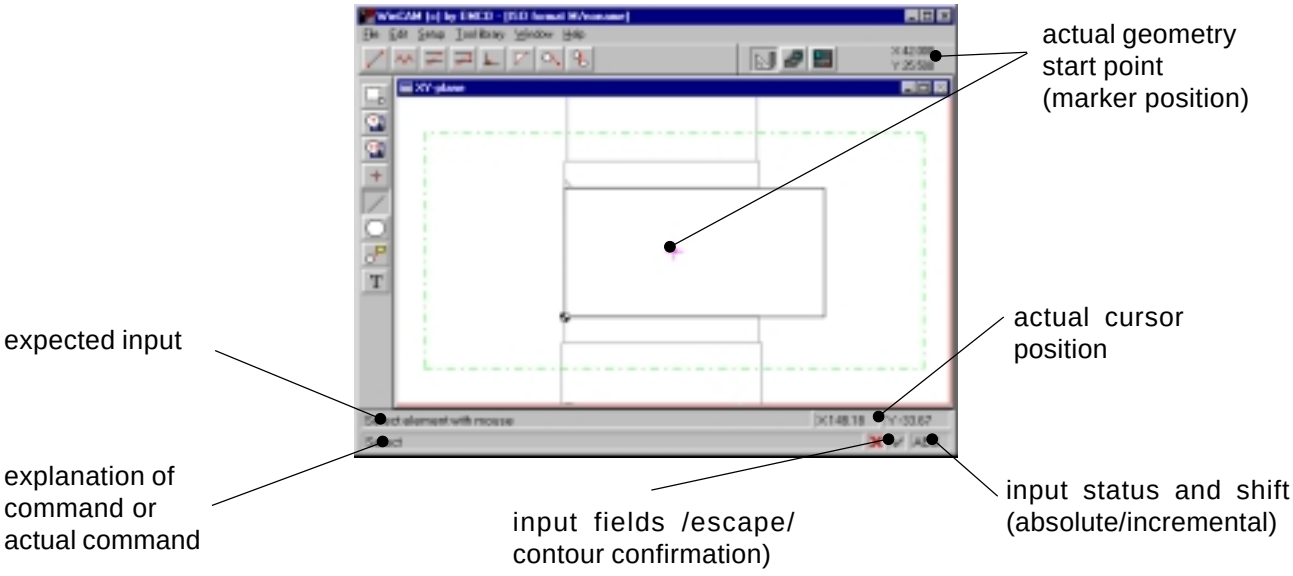
Command symbols

In the horizontal symbol line those command symbols are displayed which are called up via the vertical menu commands.

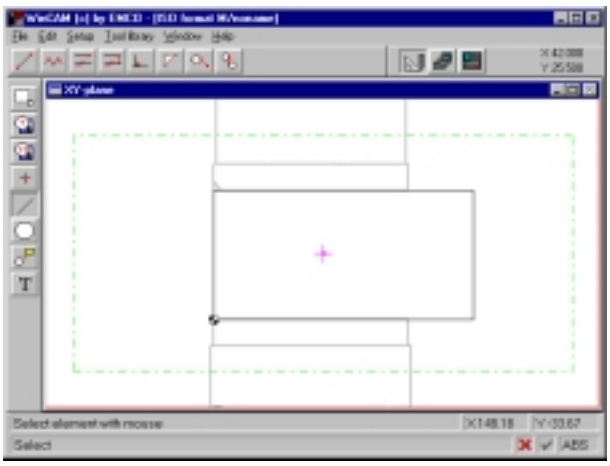
With the 3 shift symbols you can shift from CAD into CAM or into NC-mode. By this shift also all other command symbols are shifted. Here shift to CAD.



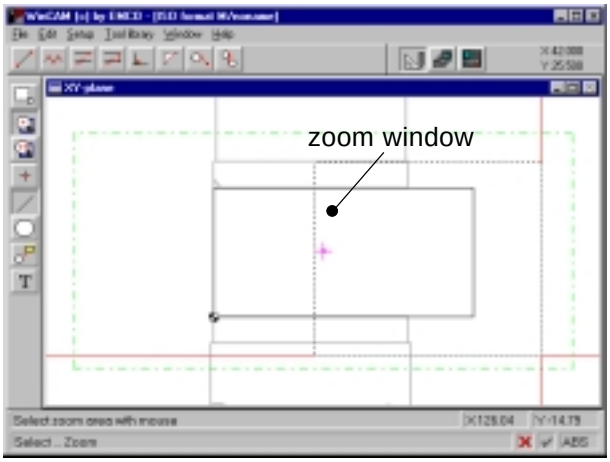
Status messages



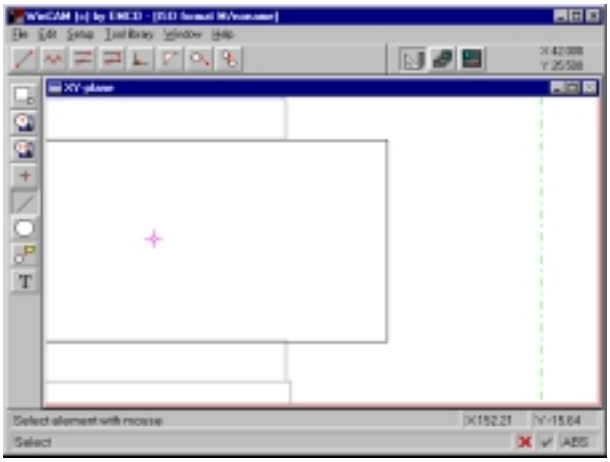
Chapter B Operating sequences



Full picture with clamping device and unmachined part



Defined zoom window



Zoomed window

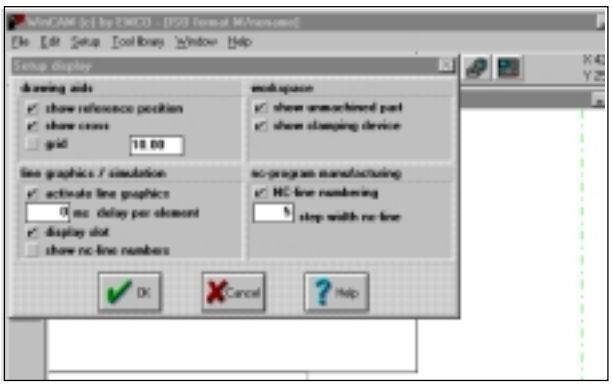
Definition of the CAD-window

Full picture 

At the beginning work with the greatest window representation possible. Switch the Win CAM window manager to full picture.

Clamping device and unmachined part

Por better imagination of space switch on the representation of clamping device (machine vice) and unmachined part. To do so call up the WinCAM window "Setup Display" with the menu line "Setup - display" and click into the respective square.



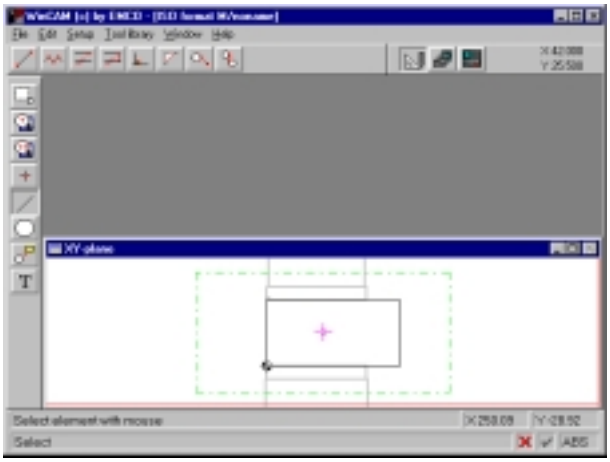
WinCAM window "Setup Display"

Zoom drawing elements 

By clicking to the command symbol "Zoom drawing elements" a rectangular section can be opened by means of the cursor and mouse click which corresponds to the new screen section. With the command symbol "Full picture" the entire CAD area is represented again.

Redraw

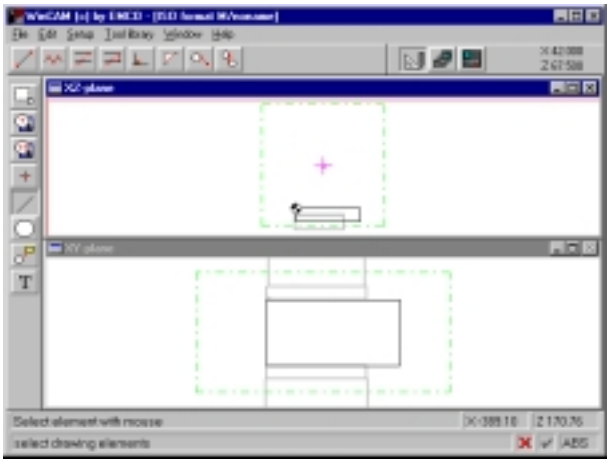
Due to modifications at the drawing it may happen that the representation is not faultless any more. Using the command "Redraw" in the menu line "Edit" the CAD screen area is set up newly.



Workpiece in XY-plane

Display of the workpiece

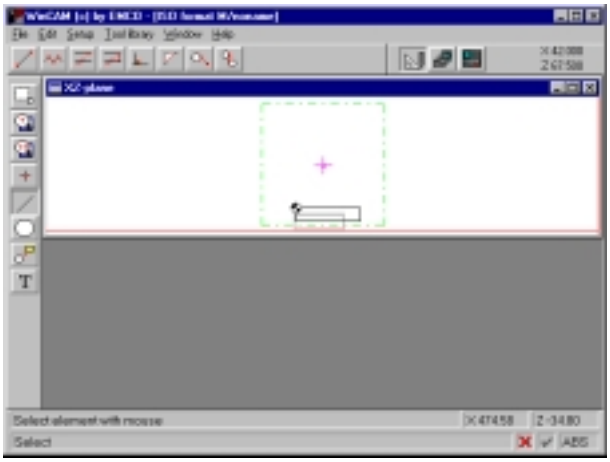
Workpiece displayed only in XY plane
Usually it is enough if the workpiece is displayed in XY-plane. This plane is also significant for the creation of the NC-program in the CAM part of WinCAM. Therefore, draw this plane in any case.



Workpiece in XZ-plane

Workpiece displayed only in XZ plane

Drawings displayed only in this plane cannot be machined any more in the CAM part of WinCAM.



Workpiece in both planes

Workpiece displayed in both planes
The display in both planes is useful if it is necessary for a better comprehension, e.g. with various machining planes.

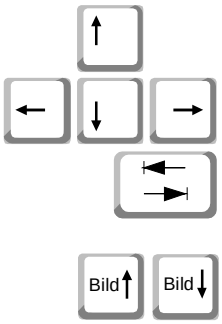
The coordinate zero point has to be defined separately for each window.

Each window can be zoomed separately, thus there is no allocation of size between the displays. Select the arrangement of the windows logically (e.g. the standardized normal view representation).

Positioning the cursor

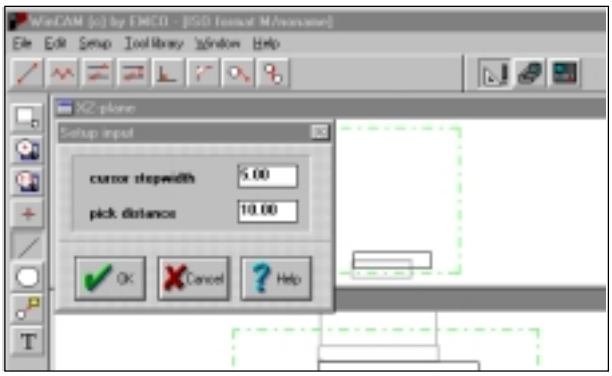
The cursor indicates that point on the screen to which the commands and actions refer.

Display of the cursor
The cursor can be shown in different ways (arrow, cross, etc.) depending in which screen section it is (input fields, CAD window, etc.) Please also mind that different mouse drivers can set or change the size and form of the cursor.



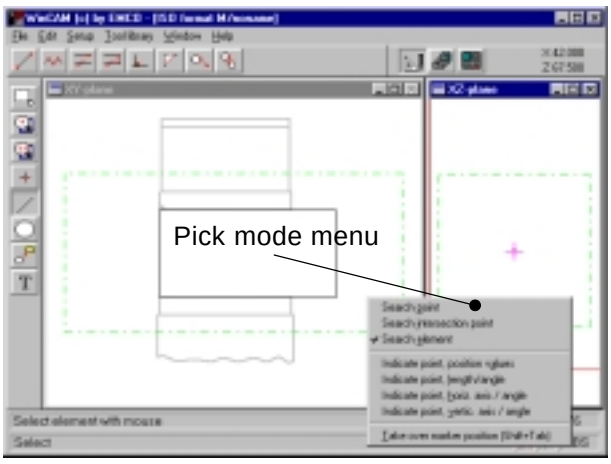
Moving the cursor
The cursor can be moved using the mouse or the arrow keys (in the actual window). You shift between these kinds of movement with the "TAB" key.
The actual position of the cursor can be read in the status window below at the right.

Cursor step width
With the arrow keys the movement is carried out with a determined step width. This step width can be changed from 0.01mm up to 20 mm by means of the keys "Page Up" and "Page Down". Setting using the menu line "Setup Input" and an input of numbers are possible, too.



Settings "Input - cursor stepwidth"

Drawing with the cursor
Select the CAD command (e.g. line drawing), shift to cursor drawing by means of the "TAB" key and move the cursor by means of the arrow keys to the selected point. Take over the actual position by means of "ENTER". Switch again to cursor drawing by using the "TAB" key and so on.



Pick mode menu

Positioning cursor
Three possibilities:

1. Moving the cursor to the selected point in the CAD window.
2. Input of the selected coordinate point in the status window. This input can be carried out absolutely or incrementally. Shift ABS/INC in the status field below at the right.
3. Setting a pick mode by clicking on the right mouse key and selection of positioning.

When and how the cursor is positioned depends on the actual CAD command. Observe the status line.

Command symbols


Symbol inactive


Symbol active

Symbol display
If a command symbol is selected by means of the mouse key (also active) the symbol frame is displayed mirrored (like pressed).

If a command of the horizontal symbol line is active the other commands of the symbol line are displayed gridded.

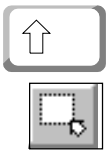
The symbol remains active until
- the command has been carried out (direct command symbols)
- the command is deselected by another one (menu commands and shift symbols)
- the command is interrupted by means of the "ESC" key or the input field CANCEL GENERAL or another click of the symbol by means of the mouse (horizontal symbol line).

Input field CANCEL GENERAL 

Selection of elements

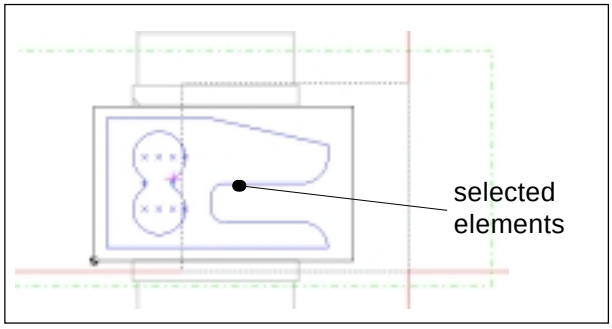
For some CAD commands and for the delete function it is necessary to select previously one or more elements. The selected elements change the colour and are thus discernible.

Selection of one element
Lead the mouse cursor near the element to be selected and click.



Selection of more elements
Two possibilities
1. Keep the "SHIFT" key pressed and click on various elements one after another.
2. Draw a frame around the elements to be selected by means of the command symbol "Select drawing elements". All elements being completely in the frame are selected.

Deselection of an element
Click again on the element or on one of the elements already selected.



Selection of elements

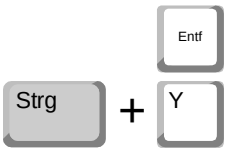
Delete

Undo the last action

If you have made an error you can cancel the last actions (e.g. delete line) by means of the command "Undo" in the menu "edit" or using the key combination "Ctrl + Z". The number of the stored actions is set on 10.

Function "Delete"

Three possibilities:



1. Delete in the menu "Edit" with "Delete CAD element" .
2. by means of the key "Delete"
3. by means of the key combination "Ctrl + Y"

Delete individual CAD elements

Select the element to be deleted and the function "Delete".

Delete more CAD elements

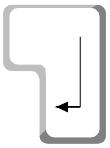
Select various elements to be deleted and the function "Delete".

Delete all

By means of the selection of the command in the menu"File":

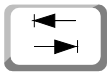
1. Command "New" deletes the CAD window for a drawing to be set up new as well as the NC-program.
2. Command "Open" loads a drawing into the CAD window, thus deletes the old drawing and overwrites the old NC-program with the new NC-program.
3. Command "Quit" (as well as all other possibilities of Windows to interrupt a task) terminates WinCAM.

A safety inquiry if the actual drawing shall be saved is obligatory.



Example: 15.83

Examples: 20,55 or 20 55
17.82,25.3 or 17.82 25.3



Inputs in the status line

If the program expects inputs, in the status line it is indicated which values have to be entered. Inputs have to be terminated with "ENTER".

Value input

Input is carried out in millimeters (or inch). A comma value is entered by a point.

Coordinate input

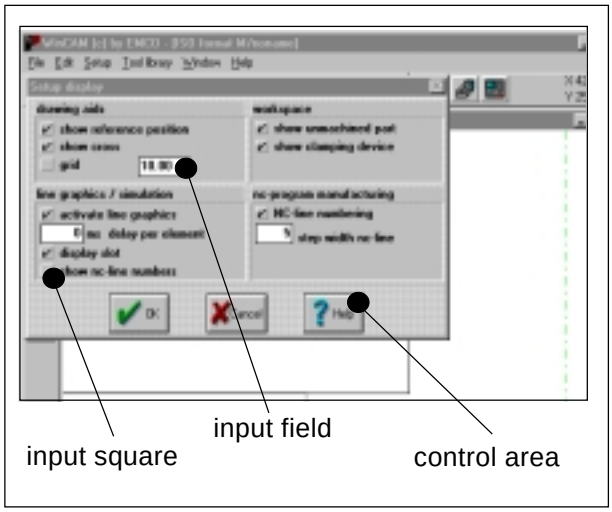
First enter the X, then the Y-coordinate, and/or X and Z according to the active window). Separation has to be carried out by means of a comma (,) or a space character.

Passing the actual cursor position

By pressing the "TAB" key twice (only if a coordinate input is expected) the actual cursor position (coordinates) can be passed into the status line.
By pressing the key combination "SHIFT" + "TAB" once (only if a coordinate input is expected) the actual marker position (coordinates) can be passed into the status line.

Pocket calculator

The status line in which you enter the numerical values can also be used as pocket calculator.
The 4 basic arithmetic operations (+ , - , * , /) are permitted, as well as the arithmetic principle "Point prior to line". Brackets are not assisted.



Example WinCAM window

Inputs in the WinCAM window

Different inputs by means of the mouse cursor and/or keyboard are possible according to the window.

1 Control areas:
- OK - the displays of the window are taken over and the window is closed.
- Cancel - the window is closed without change.
- Help -the WinCAM help to this window is overlayed.
- Various control areas marked by symbols effect functions.

2 Input squares:
The function indicated beside is activated or deactivated. The activated function is indicated by a hook.

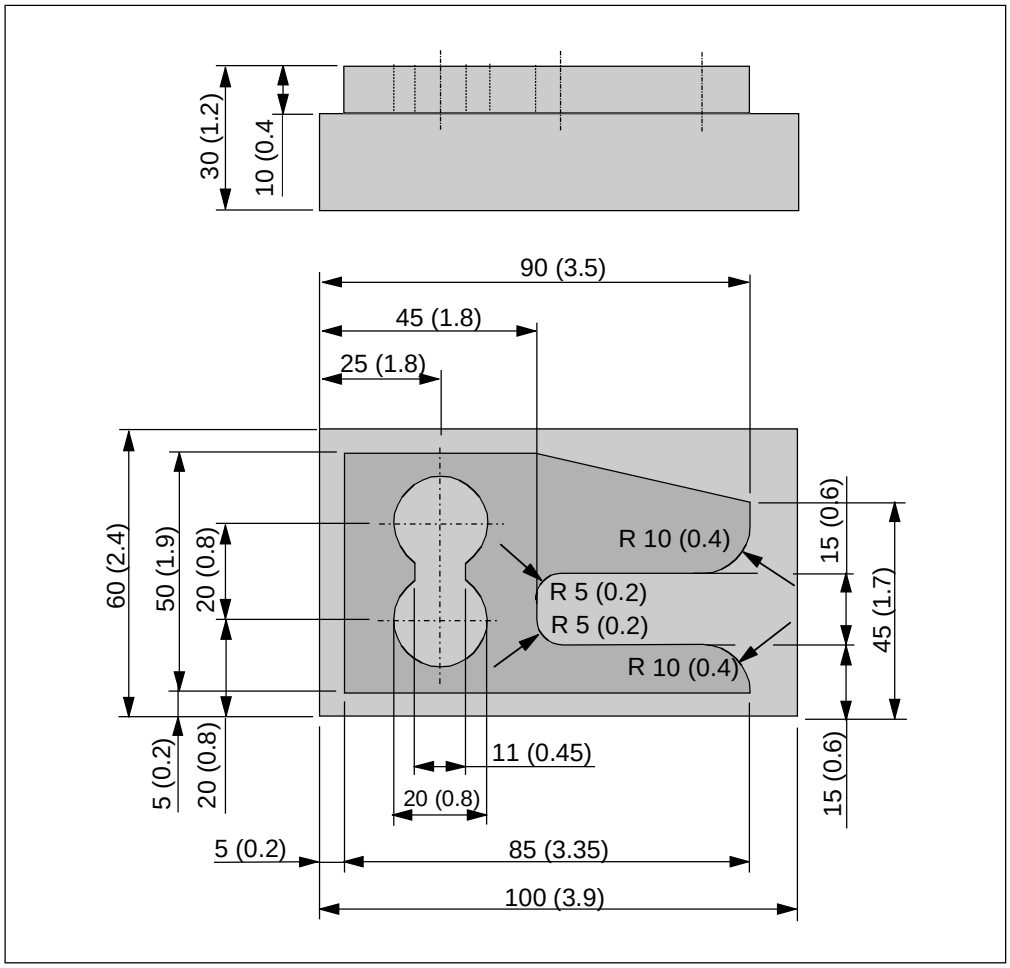
3 Input fields:
Lead the mouse cursor into the field and indicate the desired numerical value. You can also use the cancel keys and the tabulator key for switching further to the next area.

Chapter C CAD example

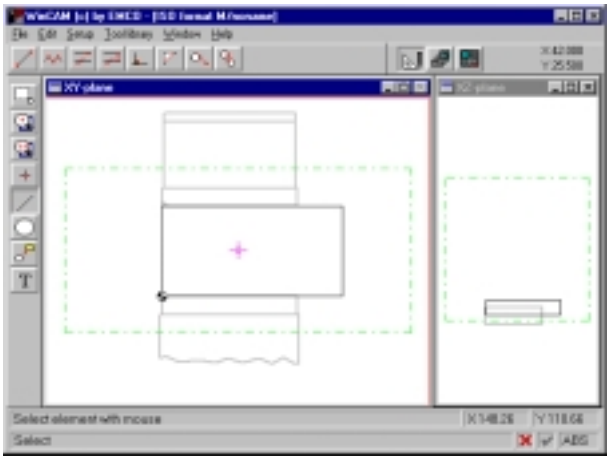
In the following 3 chapters the production of a workpiece starting from the drawing (CAD example) of the contours with WinCAM via the automatic generation of the NC-program (CAM example) with WinCAM, up to machining controlled by WinCAM at the NC machine is shown. Each of the 3 example chapters can be worked through separately in order.

To facilitate the introduction of the operation of WinCAM the following CAD example is shown. In the right column the necessary operating steps are indicated. In the left column the results at the screen are indicated, so you can check and compare immediately. You learn to configure the CAD screen and to draw the necessary contours of the workpiece shown below.

Note:
The numerical values in brackets are dimensions indicated in "inch" and refer to the inch installation of WinCAM.



Workpiece drawing for the CAD example

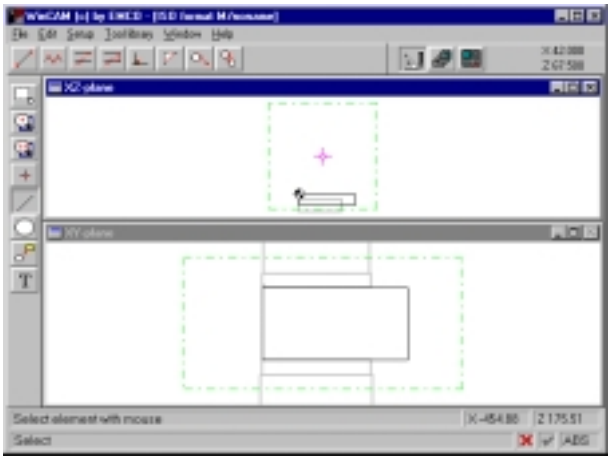


Windows arrangement at choice

Start

Load WinCAM starting from Windows.

Which CAD window you can see now and in which arrangement depends on the setting stored last. Your screen could look as shown left (XY- and XZ-window side by side).
If you should have problems with the arrangement of the windows on the following page please look up in your Windows manual. The operation is in conformity with Windows.

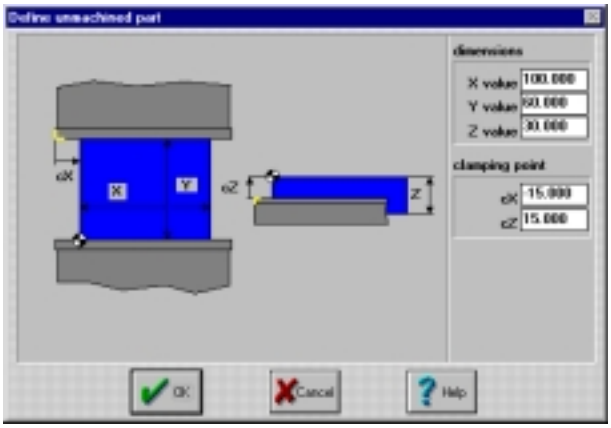


Window division for the example

Settings of the CAD window

Arrangement of the CAD window
The figure beside is to be understood as proposal for a window arrangement which assists the comprehension of the normal design projection (front view above top view). Any other arrangement of the windows including the full picture display of only one window (e.g. XY-level) is possible.

- Move and draw the window so that the window of the XZ-level fills the upper third and the window of the XY-level covers the lower two thirds.



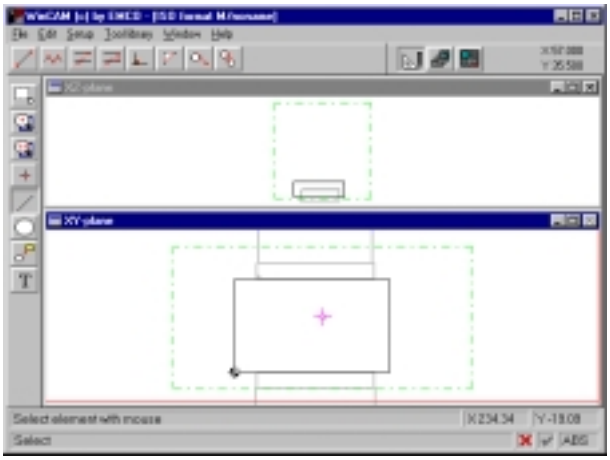
Define unmachined part/clamping point

Define unmachined part/clamping point
Define the unmachined part and the clamping point with the menu "Setup - Define unmachined part/ clamping point" in the WinCAM window "Define unmachined point".

- Enter the "dimensions":
  X-Value 100 mm (3.9 inch) (length)
  Y-Value 60 mm (2.4 inch) (width)
  Z-Value 30 mm (1.2 inch) (thickness)

- Enter the "clamping point":
  cY is automatically 0 mm (0 inch) (machine vice mounted in Y-direction)
  cX -15 mm (0.6 inch)
  cZ 15 mm (0.6 inch)

Confirm the inputs with OK.



Reference XY-level

Reference point
The reference point was automatically set by WinCAM. Check the position. The reference point is only indicated in the active window. Therefore activate subsequently the XY-window and the XZ-window.

The reference point is to be found in the XY-window at the left front corner and in the XZ-window at the left top corner of the unmachined part.

From now this CAD reference point position is used as workpiece zero point when drawing the workpiece contour.

Reference point XZ-level

Zoomed window

Zooming the workpiece
The CAD commands acts only in the actual window. Therefore the desired window has to be activated prior to the input of a CAD command. In our example the XZ-window is still active therefore we start with this window.
Click on the command "Zoom drawing elements" and draw a frame around the desired section by means of the mouse cursor.

- Call-up command "Zoom drawing elements"
  
- Draw frame (in the XZ-window)
- Activate XY-window
- Call-up command "Zoom drawing elements"
- Draw frame (in the XY-window)

Contour1

Drawing the workpiece contour

Drawing is only possible in an active window. We start in the actually active window in the XY-level. Enter the following by means of the menu commands (mouse click) and status line (coordinate input and ENTER):

Outside contour

- Line menu
- SelectPoly line

- Start point 5,5 (0.2,0.2)
- End point 5,55 (0.2,2.1)
- End point 45,55 (1.8,2.1)
- End point 90,45 (3.5,1.8)
- End point 90,40 (3.5,1.6)
- Deselect Poly line

(figure "contour 1")

- Circle menu
- Circular arc with start, end and radius
- Start point 90,40 (3.5,1.6)

- End point 80,30 (3.1,1.2)
- Radius 10 (0.4)
- Select result using "Space" and "ENTER"

Contour 2

- Line menu
- Draw line
- Start point 80,30 (3.1,1.2)
- End point 50,30 (2.0,1.2)

- Circle menu
- Circular arc with start, end and radius
- Start point 50,30 (2.0,1.2)
- End point 45,25 (1.8,1.0)
- Radius 5 (0.2)
- Select result using "Space" and "ENTER"

- Line menu
- Draw line
- Start point 45,25 (1.8,1.0)
- End point 45,20 (1.8,0.8)

(figure "contour 2")

- Circle menu
- Circular arc with start, end and radius
- Start point 45,20 (1.8,0.8)
- End point 50,15 (2.0,0.6)
- Radius 5 (0.2)
- Select result with "Space" and "ENTER"

- Line menu
- Draw line
- Start point 50,15 (2.0,0.6)
- End point 80,15 (3.1,0.6)

Contour 3

- Circle menu
- Circular arc with start, end and radius
- Start point 80,15 (3.1,0.6)
- End point 90,5 (3.5,0.2)
- Radius 10 (0.4)
- Select result using "Space" and "ENTER"

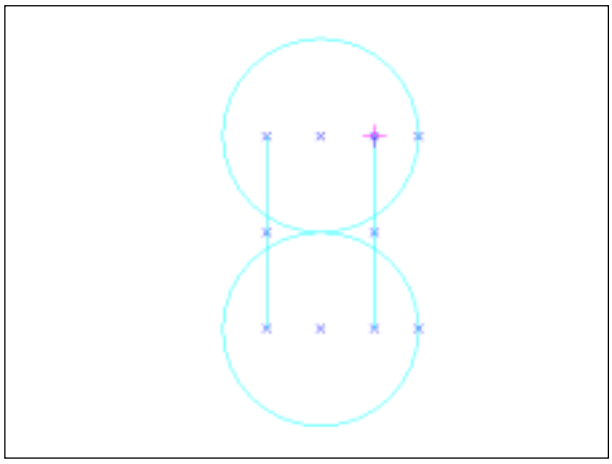
- Line menu
- Draw line
- Start point 90,5 (3.5,0.2)
- End point 5,5 (0.2,0.2)

(figure "contour" 3")

Contour of the bores

- Circle menu
- Circle with center and radius 
- Center 25,20 (1.0,0.8)
- Radius 10 (0.4)
- Circle with center and radius
- Center 25,40 (1.0,1.6)
- Radius 10 (0.4)

- Line menu
- Draw line
- Start point 19.5,20 (0.775,0.8)
- End point 19.5,40 (0.775,1.6)
- Draw line
- Start point 30.5,20 (1.225,0.8)
- End point 30.5,40 (1.225,1.6)



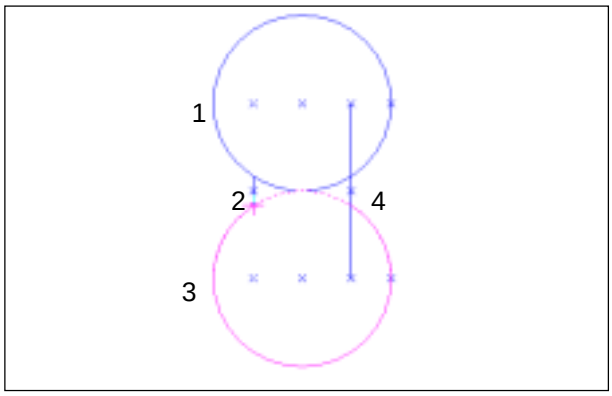
Contour 4

Displays of the construction points

- Point menu 
- Select drawing elements 
- Draw frame around the circles to be trimmed
- Display design points of marked elements

- Zoom drawing elements 
- Draw frames around the circles to be trimmed

(figure "contour 4")



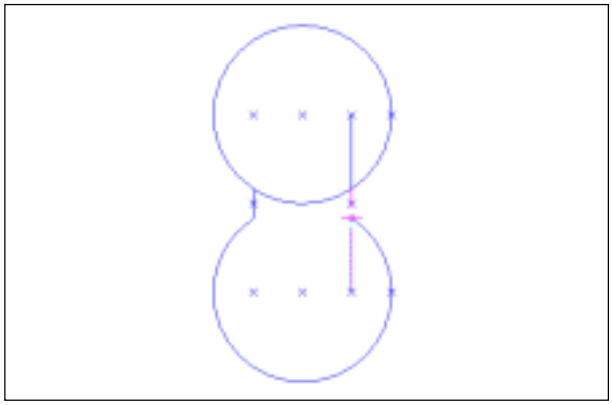
Contour 5

Trimming the contour

- Modifying menu 
- trim contour 
- Select all 4 elements (by means of "SHIFT" and mouse click in order 1 to 4)
- Green hook confirm with contour determination 
- First selection of the possible partial circle by means of "Space"

(figure "contour 5")

- Confirmation with "ENTER"

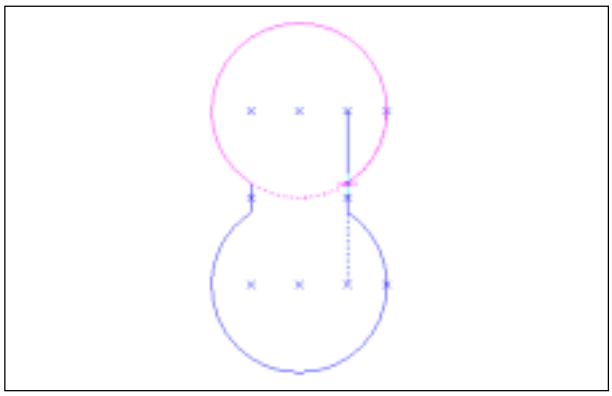


Contour 6

- Second selection of the possible partial line with "Space"

(figure "contour 6")

- Confirmation with "ENTER"

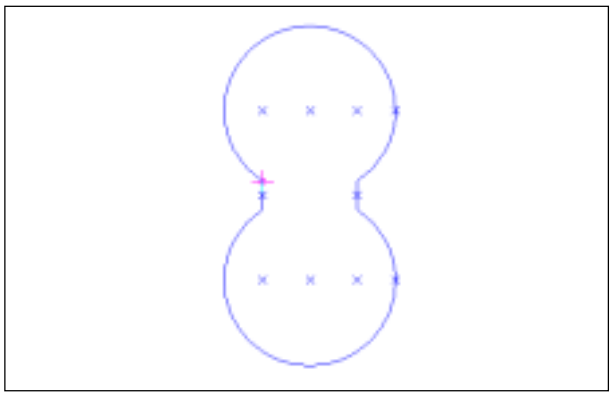


Contour 7

- Third selection of the other possible partial circle with "Space"

(figure "contour 7")

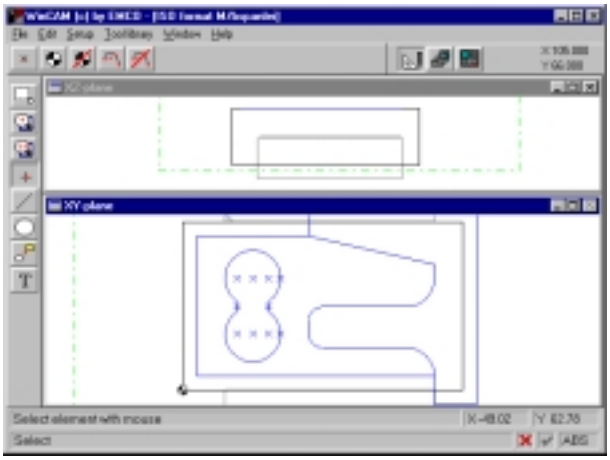
- Confirmation with "ENTER"



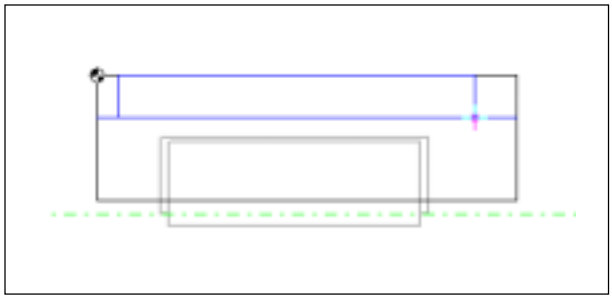
Contour 8

Now the contour is ready trimmed.

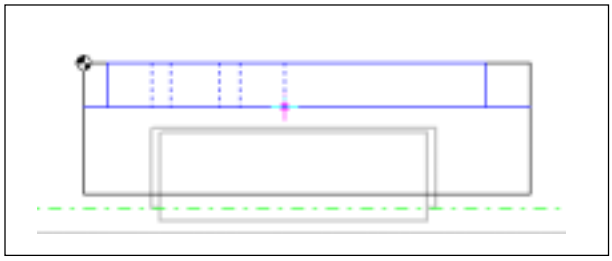
(figure "contour 8")



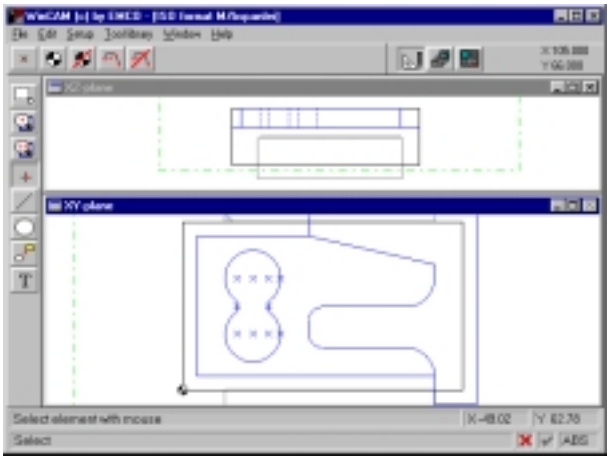
Contour 9



Contour 10



Contour 11



Contour 12

Change of window

- Select the window of the XZ-level

(figure "contour 9")

Contour of the front view

- Line menu
- Poly line
- Start point 5,-10 (0.2,-0.4)
- End point 5,0 (0.2,0)
- End point 90,0 (3.5,0)
- End point 90,-10 (3.5,-0.4)
- End point 100,-10 (3.9,-0.4)
- Ploy line (deselect)
- Draw line
- Start point 0,-10 (0,-0.4)
- End point 90,-10 (3.5,-0.4)

(fig. contour 10")

Draw hidden lines

- Menu "Setup"
- Menu "Line"
- CAD standard change
- Line style
- Broken line
- OK, OK
- Draw line
- Start point 15,0 (0.6,0)
- End point 15,-10 (0.6,-0.4)
- Draw line
- Start point 19.5,0 (0.775,0)
- End point 19.5,-10 (0.775,-0.4)
- Draw line
- Start point 30.5,0 (1.225,0)
- End point 30.5,-10 (1.225,-0.4)
- Draw line
- Start point 35,0 (1.4,0)
- End point 35,-10 (1.4,-0.4)
- Draw line
- Start point 45,0 (1.8,0)
- End point 45,-10 (1.8,-0.4)

(figure "contour 11")

Reset full line

- Menu "Setup"
- Menu "Line"
- CAD standard change
- Line style
- Full line
- OK, OK

Now drawing of the workpiece is terminated (see page "contour 12") and work at the workpiece can be continued in the CAM part of WinCAM. Store the drawing for safety reasons (see next page).

Save drawing

- Menu "File"
- Menu "Save as"
- Enter file name
- OK or ENTER

Convert drawing into DXF format

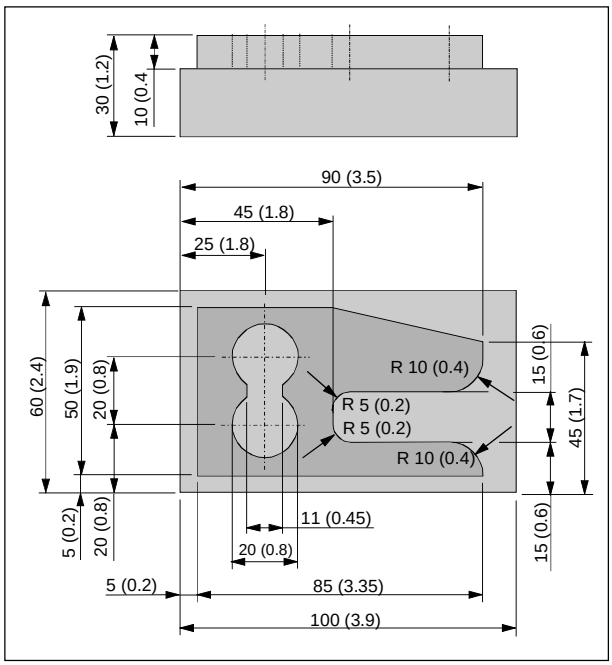
- Menu "File"
- Menu "Export"
- Menu "Drawing"
- Enter file name
- OK or ENTER

Print drawing

- Menu "File"
- Menu "Print"
- Menu "Drawing"
- OK or ENTER

Chapter D CAM example

To facilitate the introduction to the operation of WinCAM the following CAM example is shown. The CAD drawing from the CAD example of this manual is used for this purpose. In the right column the necessary operating steps are indicated. In the left column the results are shown on the screen so that you can check it immediately. You learn to configure WinCAM for automatic processing and to generate and simulate an NC-program.



Drawing of the workpiece (dim. see page 17)

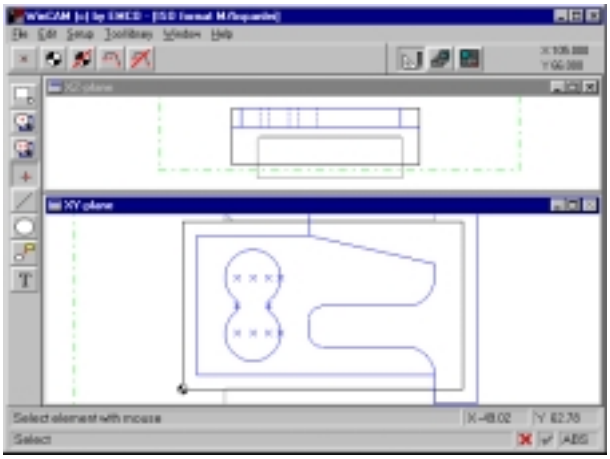
Specifications

- Workpiece
  The drawing of the workpiece which is to be machined.

- Unmachined part
  Length 100 mm (3.9 inch), width 60 mm (2.4 inch), thickness 30 mm (1.2 inch)

CAD drawing

- Generate the geometry of the workpiecein CAD mode of WinCAM. Thus, please look at chapter "CAD example", or load the stored drawing.

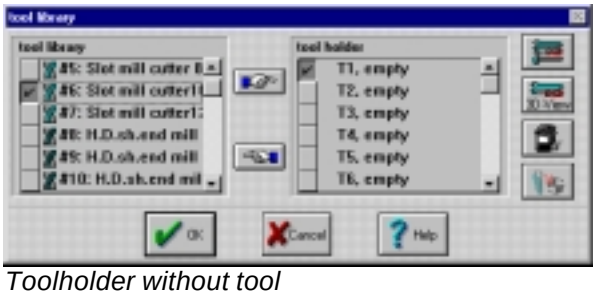


Finished CAD drawing

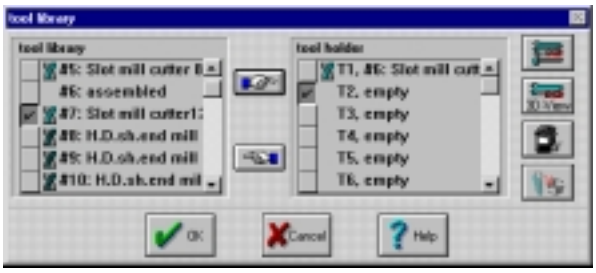
Technological data

- Consider the technological data for machining.
  1 the order of the machining steps
  2 which tools are used
  3 which cutting speed (speeds)
  4 which feeds (milling, immersion)

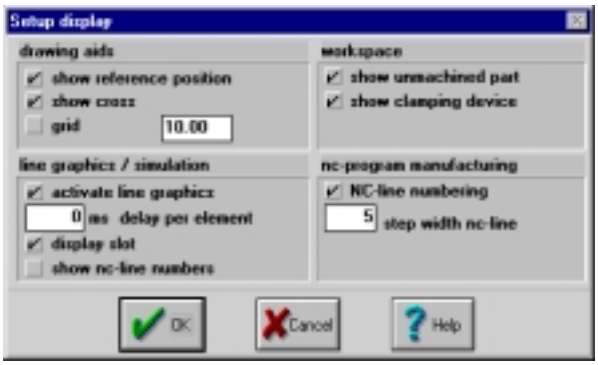
- The following has been selected:
  1 Milling the bores 20 mm (0.8 inch) milling the outer contour
  2 Slot mill cutter 10 mm diameter from the library
  3 Speed 2000 rpm
  4 Mill feed 300 mm/min (12 inch/min) immerse feed 150 mm/min (6 inch/min)



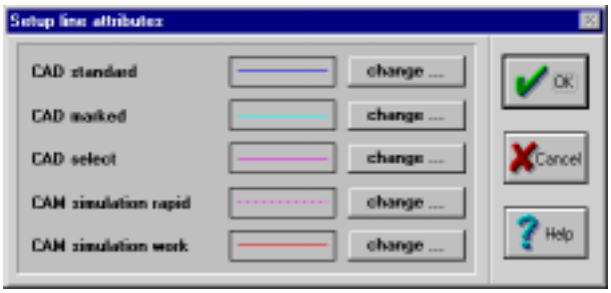
Toolholder without tool



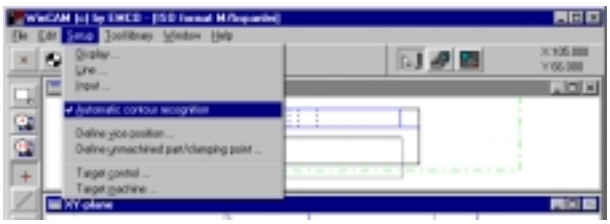
Toolholder assembled with tool



Inputs in the window "Setup - Display"



Inputs in the window "Setup line attributes"



Input "Automatic contour recognition"

Inputs "Target control and target machine"

Assemble toolholder

- Call up the tool library by means of the menu line.
- Select the tool slot mill cutter 10 mm in the library.
- Select the toolholder post number 1.
- Pass the tool into the holder by means of the command area.

- Close the library via the command area OK.

CAM settings

Displays

- Call up the WinCAM window "Setup - Display" and select:
  - Show referenz position
  - show cross
  - Activate line graphics
  - Display slot
  - Delay per element 200 ms
  - Show unmachined part
  - NC- line numbering
  - Show clamping device

Line attributes

- Have a look at the WinCAM window "Setup line attributes" if the CAM line displays are correctly set.
  - CAM simulation rapid motion with broken line
  - CAM simulation work with full line

Contour recognition

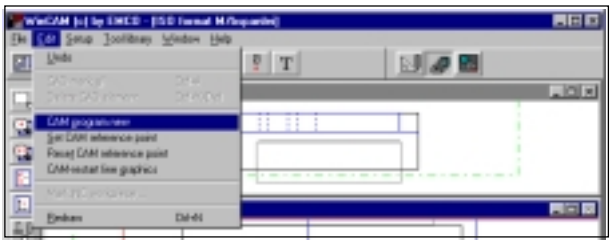
- Set the "*automatic contour recognition*" in the menu Setup by means of a click with the mouse cursor, if a hook is not yet indicated.

Target control

- Set the target control with the menu Setup in the WinCAM window "Select target control" to "DIN 66025".

Target machine

- Set the target machine with the menu Setup in the WinCAM window "Select target machine" to "PC Mill 50" (if used).



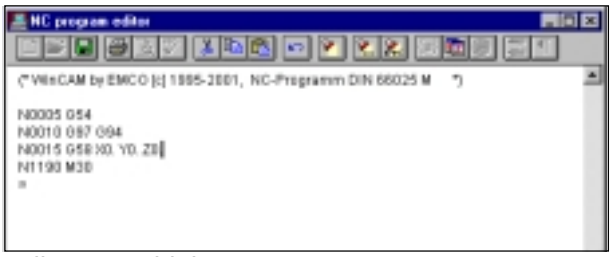
Input "CAM program new"

CAM - Mode

- Switch to the CAM mode by means of the shift symbol.

CAM Program new

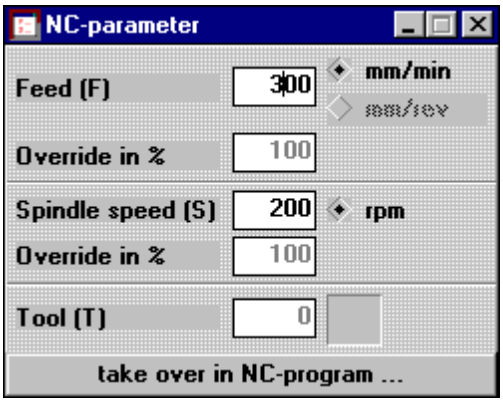
- Select "CAM program new" in the menu Edit so that all CAM parameters are reset.



Editor - machining step 1

NC-Program Editor

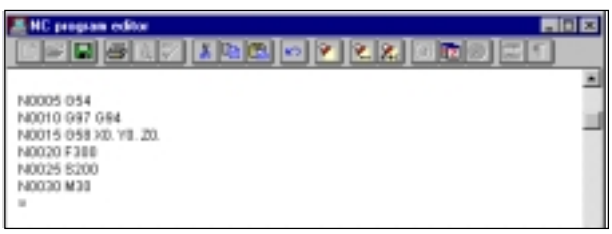
- Call up the window "NC program editor" by means of the command symbol "editor" and have a look at the program generated up to now. Repeat this procedure always when the command symbol editor is displayed in the example. So you can see how WinCAM writes down the NC-program during machining.
- Close the window by clicking again on the command symbol .



Inputs in the window "NC-parameters"

NC-parameters

- Call up the WinCAM window "NC parameters" by means of the command symbol and determine the NC parameters.
- Feed milling 300 mm/min (12 inch/min)
- Speed 2000 rpm
- Take over the inputs by means of a click into the command field "take over in NC-program".
- Close the window by another click on the command symbol "NC-parameters".



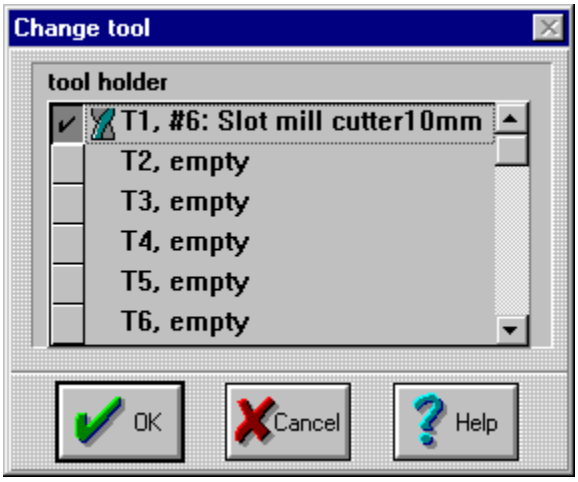
Editor - machining step 2

- Have another look into the editor. The NC parameters (F300 and S2000) have been entered.

- Close the editor window again.

Change tool

- Now define the tool with which you want to machine or on which it shall be changed. Open the WinCAM window "Change tool" by means of the command symbol "Change tool".



Inputs in the window "Change tool"

- Click on toolholder number 1.

  Check the presently actual data for feed and speed in the automatically opened window "NC-parameters".

- Close the window by a click on OK.

- Open the horizontal symbol line by a click on the symbol "periphery".

Switch on spindle

- Switch on the main spindle by a click on the command symbol "main spindle on / right".

- Look up again in the editor. The tool (T2 D3 M0) and the switch-on of the spindle (M03) have been entered.

- Close the editor window again.

Editor - machining step 3

- Open the horizontal symbol line by a click on the symbol "machining".

Move workpiece into start position

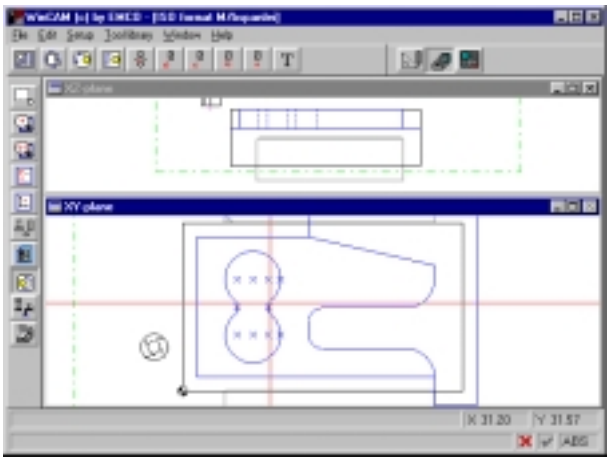
- Move the tool to the unmachined part by clicking on the command symbol "movement in rapid motion in plane".
- Enter the target coordinate
  -10,0 (0.4,0)
  (X,Y-plane) and acknowledge with "ENTER".

- Click on the command symbol "movement in rapid motion in feed direction".
- Enter the Z-target coordinate
  2
  (XZ-plane) and acknowledge with "ENTER".

- Have another look in the editor. The movements (G0) have been entered.
- Close the editor window again.

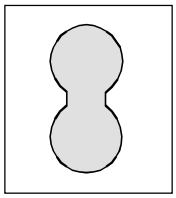
Editor machining step 4

In the CAM screen the miller has become visible now in both levels. The movement of the miller in rapid motion to this position is displayed with a broken line.

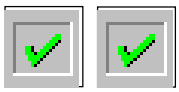


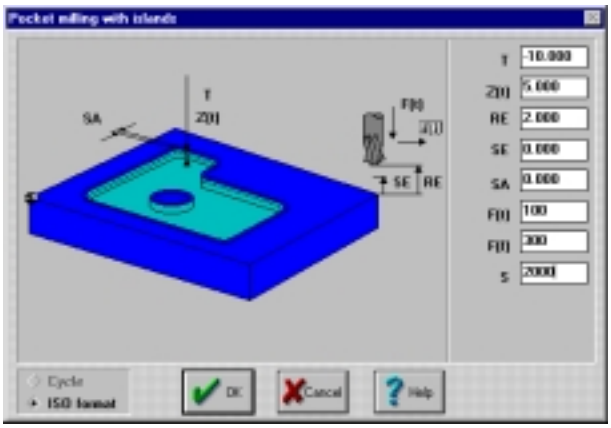
CAM screen - machining step 4

Thus, all fundamental CAM settings have been accomplished. According to the automatization of the machine further steps could be still necessary (automatic door, automatic clamping device, coolant, etc.). For this purpose have a look into the manual of the NC machine.

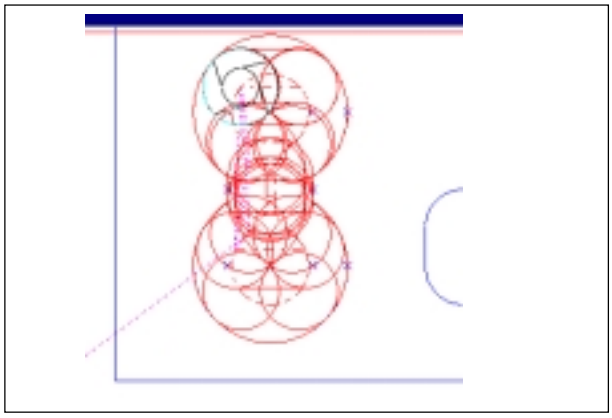


Contour of the circles





Inputs in the window "pocket milling with islands"



CAM screen - working step 5

CAM machining

Milling the contour of circles

- Start machining via a click on the command symbol "island milling".

- Select the contour of both circles (diameter 11 mm (0.45 inch)) using the mouse cursor.

- Confirm the displayed contour by means of the green hook "confirm with contour determination".
- Click again on the green hook.

The WinCAM window "pocket milling with islands" is displayed. Enter the following:

- T = -10 mm (-0.4 inch)
- Z(t) = 5 mm (0.2 inch)
- RE = 2 mm (0.08 inch)
- SE = 0 mm
- SA = 0 mm
- F(t) = 100 mm/min (4 inch/min)
- F(f) = 300 mm/min (12 inch/min)
- S = 2000 rpm

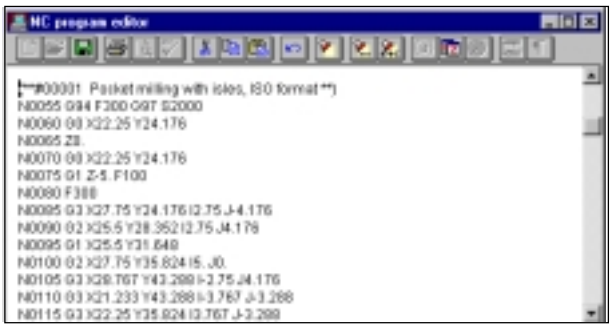
- OK

Machining is started.

In the CAM screen the miller has now worked out the circle contour in both levels. Not only the travel paths but also the envelopes are displayed.

- Have again a look in the editor. The movements the cycle has travelled have been entered between the comments "island milling" and "End".

- Close the editor window again.

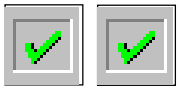


Editor machining step 5

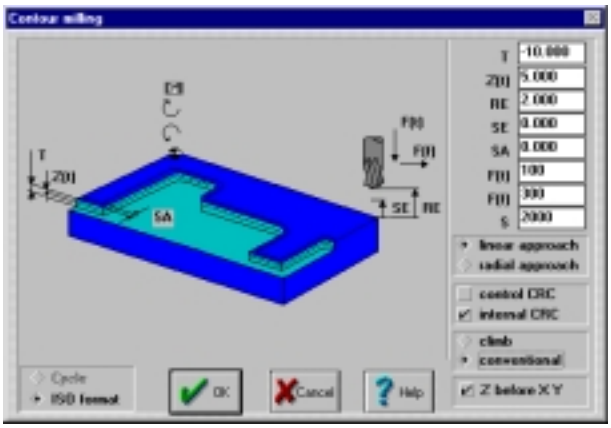
Milling the outside contour

- Move the tool again out of the contour by clicking on the command symbol "movement in rapid motion in plane".
- Enter the target coordinate -5,-5 (-0.2,-0.2) (X,Y-plane) and acknowledge with "ENTER".

- Then mill the outside contour. Click on the command symbol "follow contour".
- Select the contour. It is displayed in color.
- Confirm the contour by means of the green hook "confirm with contour determination" (2x).



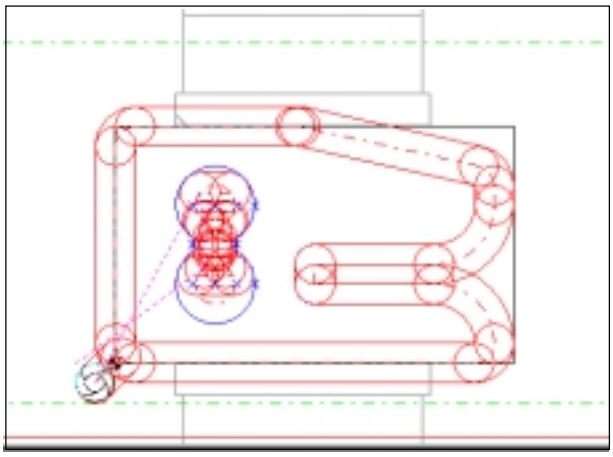
The WinCAM window "contour milling" is displayed. Enter the following:



Inputs in the window "contour milling"

- T = -10 mm (-0.4 inch)
- Z(t) = 5 mm (0.2 inch)
- RE = 2 mm (0.08 inch)
- SE = 0 mm
- SA = 0 mm
- F(t) = 100 mm/min (4 inch/min)
- F(f) = 300 mm/min (12 inch/min)
- S = 2000 rpm
- linear approach
- internal CRC
- conventional
- OK

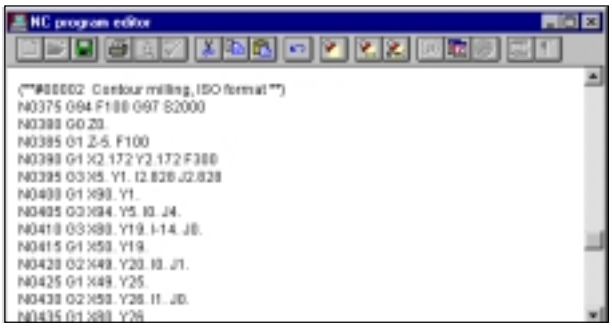
Machining starts.



CAM screen - machining step 6

In the CAM screen the miller has now worked out the outer contour. Not only the travel paths but also the envelopes are displayed.

- Have another look in the editor. The movements the cycle has travelled have been entered between the comments "contour milling" and "End".
- Close the editor window again.



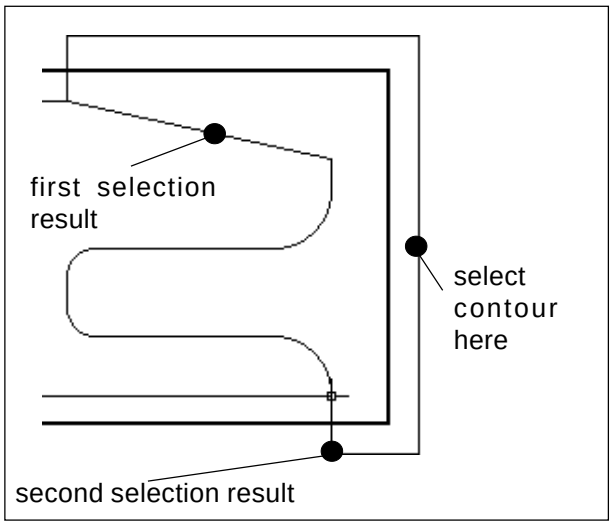
Editor machining step 6

Milling residual areas

Beside the contour right there are still unmachined areas left. These have to be reworked. The easiest way to do that is to draw an auxiliary contour around these areas and to use the cycle island milling.

- For drawing the auxiliary contour switch into the CAD mode.

- Select the command symbol "poly line".



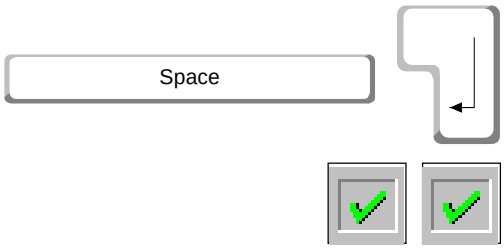
Auxiliary contour for machining step 7

- Start point 45,55 (1.8,2.1)
- Target point 45,66 (1.8,2.6)
- Target point 105,66 (4.2,2.6)
- Target point 105,-5 (4.2,-0.2)
- Target point 90,-5 (3.5,-0.2)
- Target point 90,5 (3.5,0.2)

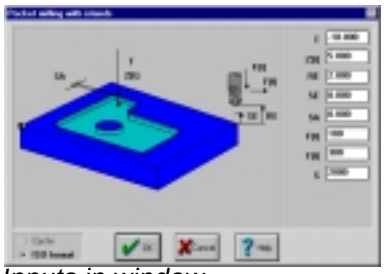
- poly line

- CAM mode

- pocket milling with islands

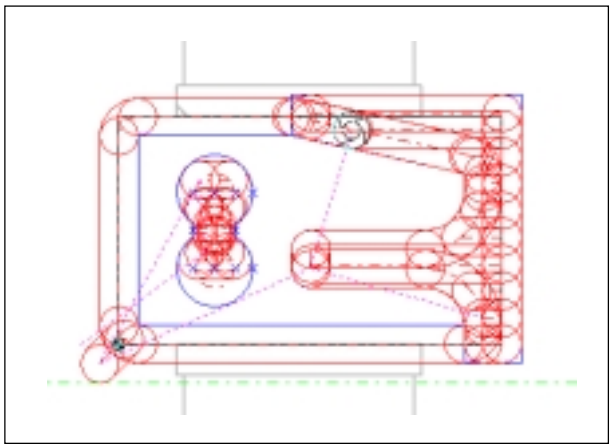


- Select contour (see picture above)
- Select the selection result using the space (see picture above).
- ENTER
- Select the second selection result using the space key (see picture above).
- ENTER
- Click hook for confirmation of the contour determination.
- Click hook for confirmation of the contour determination a second time.



- Take over WinCAM window "pocket milling with islands" with all values by clicking on OK.

Inputs in window "pocket milling with islands"



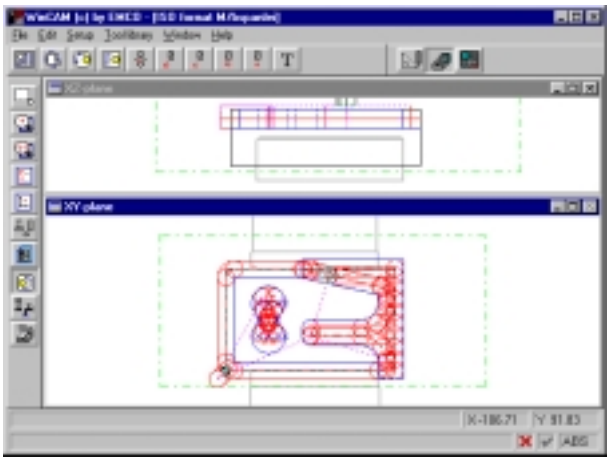
CAM screen - machining step 7

In the CAM screen the miller has now reworked the unmachined areas. Not only the travel paths but also the envelopes are displayed.

- Erase the auxiliary contour now not necessary any more (in WinCAM CAD mode) since otherwise the auxiliary contour is also saved when saving the project (like the normal drawing geometry of the workpiece).

- Move the tool with rapid motion away from the workpiece (Position value -30.0 mm or -1.2,0 inch).

Thus, the CAM example is terminated. At the opposite you can see the ready CAM screen.



CAM screen - workpiece machined

- Have another look in the editor.
  Now you can see and also print the entire NC-program.

Editor with program end

- Close the editor window again.

Saving the project

- Menu "File"
- Menü "Save as"
- Enter file name
- OK

Always the entire project consisting of drawing and NC program is saved.

If various NC-programs (acc. to DIN66025, Fanuc and Sinumerik) have been generated (just let machining run through 3 times with different target control), the 3 NC-programmes are automatically saved separately.

Printing the NC-program

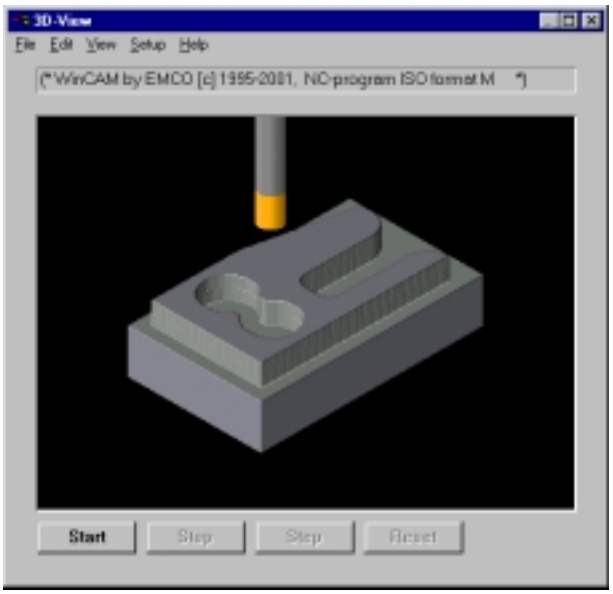
- Menu "File"
- Menu "Print"
- Menu "NC-program"
- OK

The NC-program is printed for the control (DIN66025, Fanuc or Sinumerik), defined in the WinCAM window "target control" .

3D- simulation

Everytime you can check your programmed contour with the 3D- simulation.

- Press the 3D- key
- Simulation are opened. With the start- icon you can start the simulation.



3D- Simulation

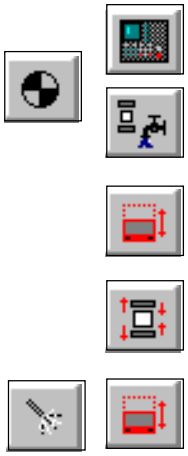
Chapter E NC examples

To facilitate the introduction into the operation of the NC-machine of WinCAM, exercises are shown. First some exercises to control the NC machine in NC manual operation and in NC program operation. Subsequently the clamping device and the tools are measured to prepare the NC-machine for machining the workpiece shown in the CAD and CAM example. The chapter ends with the work-off of the workpiece. Here the handling of the PC Mill 50 is described.

NC- manual operation
In manual operation the NC-machine is only controlled by online commands. Only one function is carried out at a time.

NC- program operation
In program operation an NC-program contained in the editor is worked off after entering the function "Start program".

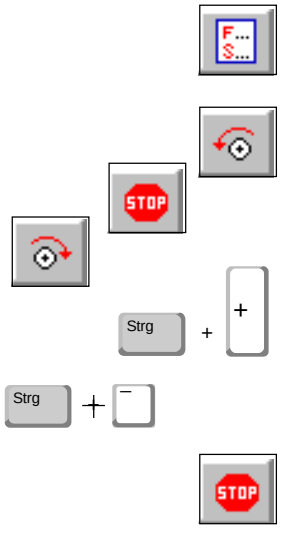
Exercises for NC- manual operation



- Switch to NC mode.
- Approach the reference point.
- Switch on the periphery symbols.

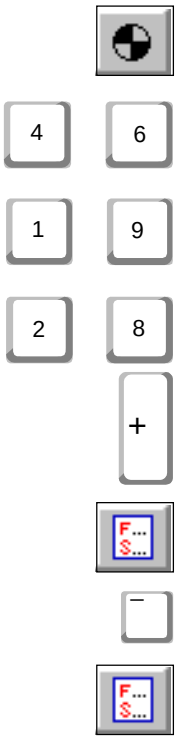
Periphery

- Open or close the automatic door by clicking the command symbol. Terminate the exercise with open door.
- Open or close the clamping device by clicking the command symbol. Terminate the exercise with closed clamping device.
- Switch on or off the blow-out device by clicking the command symbol. Terminate the exercise with switched off blowing device and close the door again.



Main spindle

- Open the WinCAM window "NC-parameters". Indicate the speed with 1000 rpm.
- Switch on the spindle by clicking the command symbol in counterclockwise direction.
- Switch off the spindle by clicking the command symbol.
- Switch on the spindle by clicking the command symbol in clockwise direction.
- Increase the speed with the PC keys "Increase speed override".
- Reduce speed with the PC keys "Reduce speed override ".
  Have a look at the displays in the WinCAM window "NC-parameter".
- Switch off the spindle again.



Feed

- Approach the reference point of the NC-machine.
- Traverse the longitudinal slide into
  +X direction and into
  - X direction.
- Traverse the cross slide into
  +Y direction and into
  - Y direction.
- Traverse the vertical slide into
  +Z direction and into
  - Z direction.
- Increase the feed speed by means of the PC keys "Increase feed override". Thereby traverse with the slides.
- Look at the displays in the WinCAM window "NC-parameters".
- Reduce the feed speed by means of the PC keys "Reduce feed override".
  Thereby traverse with the slides.
- Look at the displays in the WinCAM window "NC-parameters".

Position display

- Open the WinCAM window "position display". Look at the displays. Traverse the slide and compare the displays.

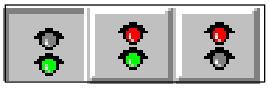
NC-test program

Exercises for NC- program operation

- Write a short exercise NC-program into the editor. Look at the listing at the left.
- The NC-machine must be initialized prior to an NC-program start (see NC basics).

- Click on the command symbol "machining".

- Start the NC-program with the command symbol "start program".

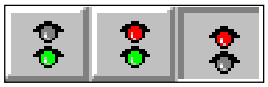


- Wait for the program end. Look at the status displays.

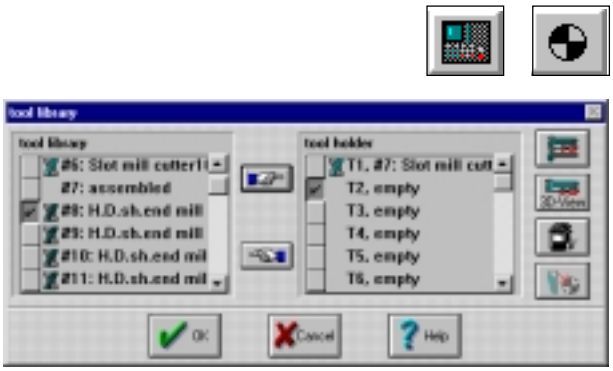
- Start the NC-program again.

- Set the NC-program into the intermediate hold with "stop program (NC stop)". Now you can also carry out various inputs in NC-manual operation. However, mind that the same state as prior to the hold has to be established.

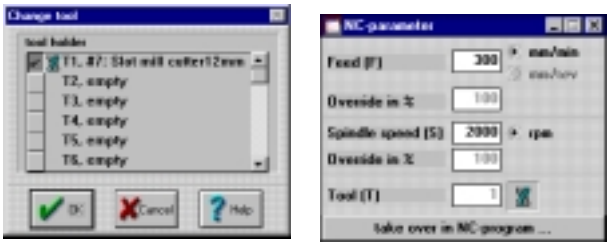
- Continue the program with "start program".
- When the NC-program is terminated the opposite status is to be seen.



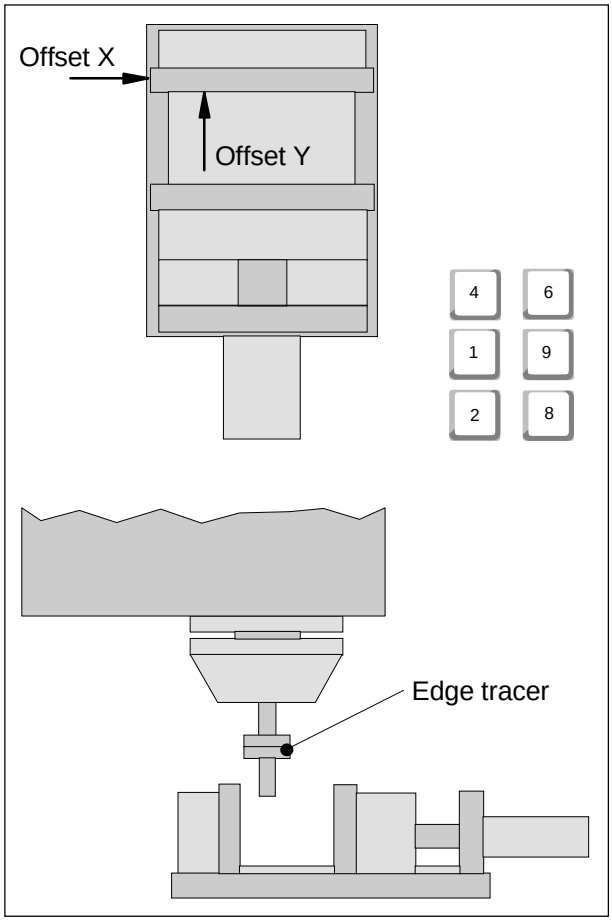
Train also the other functions of the command symbol "machining" (particularly the single step) as well as also the PC-program keys which are equal in function. Practice also with the override keys while the NC machine is travelling with the slides.



Menu "Tool library"



Tool change and NC parameter



Approach movement in X- and Y-direction

Exercise Define vice position

In the following the procedure for measuring the machine vice offset coordinates is described in detail.

- Select NC-mode
- Approach reference point.
- Insert edge tracer like a tool into the spindle.
- Assemble (in the menu "Tool library") the toolholder with a tool with the same cutting radius as the edge tracer (e.g. slot mill cutter diameter 4 mm).
- Exit from the menu with OK.
- Select this tool with the command symbol "Tool change".
- Select the window "NC-parameter" and enter the spindle speed with 400 rpm (The feed in the NC-manual is automatically 600 mm/min and is changed via the override keys. Therefore, a specification is not necessary).
- Leave the window with OK.

- Switch on the spindle by means of the command symbols "periphery" and "main spindle on / right".

Note: The spindle does not run with open door. Therefore close the chip door.

- Open the window "Setup position of clamping device" in the menu "Setup".
- Approach carefully the left fixed machine vice jaw edge (position 1 in X-direction) with the edge tracer (finally only step by step with the minimum feed) using the PC movement keys (keys "4", "6", "1", "9", "2", "8"). When approaching the jaw the edge tracer is turning always more circularly and shows a sudden restlessness during complete contact.
- Now this position is taken over into the input square "clamping device X" via a mouse click.
- Repeat this procedure by approaching the position 2 in Y-direction.
- Transfer this position into the input square "clamping device Y" by means of a mouse click.
- Close the window with OK.

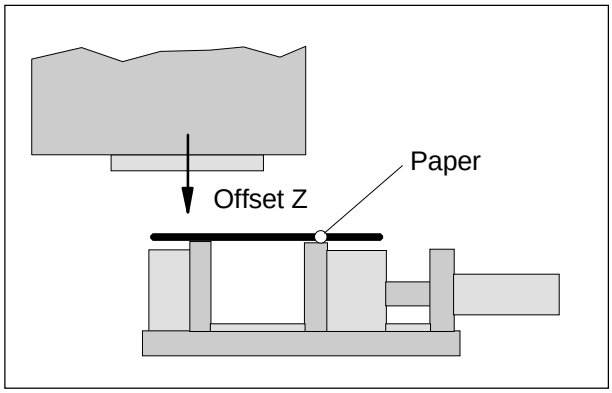
Vice position defined in X and Y

Hints:
Instead of an edge tracer also the method with the paper is possible (see below). If possible, use a round steel instead of a tool.
During manual traverse of the slide with open door an NC error message is emitted. Ignore it during the adjustment work.
By opening the position display window you can get a survey of the present slide position.
Take care that the slides do not start fully, so that the motors get blocked.

- Switch off the spindle
- Remove the edge tracer from the spindle.

Tool with offsets = 0

- Select a tool in which all tool offsets equal 0 (perhaps you might create such a reference tool for that case in the tool library,)
- Open the window "Define vice position" in the menu "Setup".



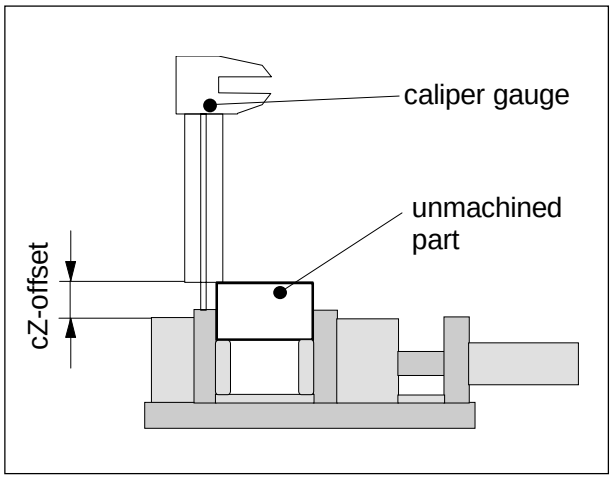
Approach movement in Z-direction

- Place a thin sheet of paper onto the vice. Traverse the milling head by means of the PC movement keys up to the upper vice jaw edge (position 3 in Z-direction). Approach carefully the spindle area (at the end only step by step with minimum feed) until the paper is clamped.

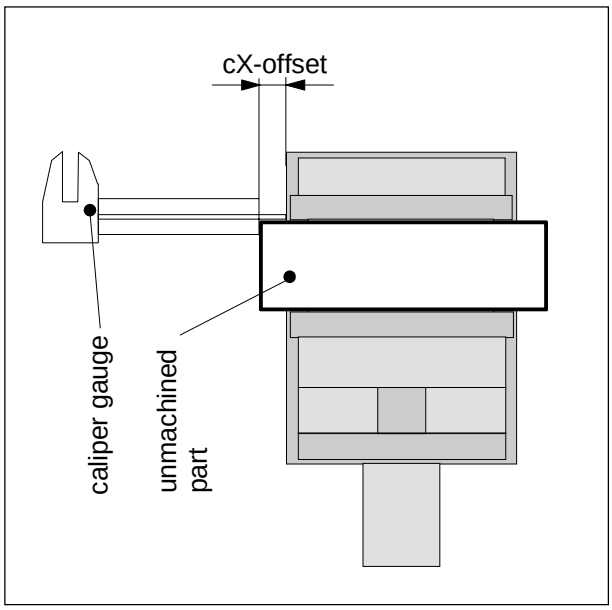
Vice position defined in X, Y and Z

- Now this position is taken over into the input square "clamping device Z".

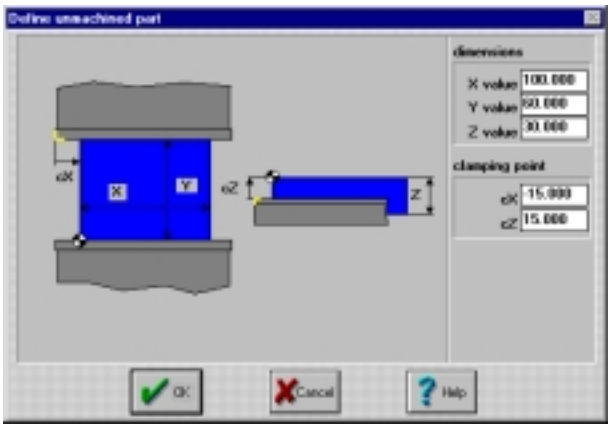
Now the vice is measured. The vice offset data are stored by WinCAM.



Measuring the cZ-offset



Measuring the cX-offset



Enter measured values

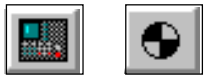
Exercise define unmachined part/clamping point

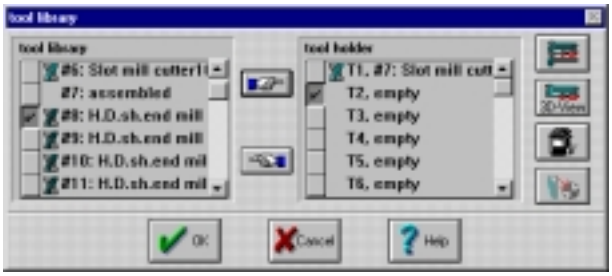
- Open the WinCAM window "Define unmachined part" by clicking on the menu item "define unmachined part / clamping point" in the "Setup" menu.
- Measure length, width and height of the unmachined part by means of a caliper gauge.
- Enter the length, width and height as X-, Y- und Z-value into the input fields of the window "Define unmachined part" .
- Clamp the unmachined part into the vice.
- Measure the clamping point with reference to the defined vice jaw areas in X- and Z-direction by means of the caliper gauge.
- Enter these clamping point values as cX and cZ into the input fields.
- Close the window with OK.

The values are saved by WinCAM. Thus, the CAM coordinate point in NC-mode is defined.

Exercise Tool measurement

In the following the procedure for measuring the tool offset coordinates is described in detail. Here no special reference tool is necessary but the spindle surface is used as reference.

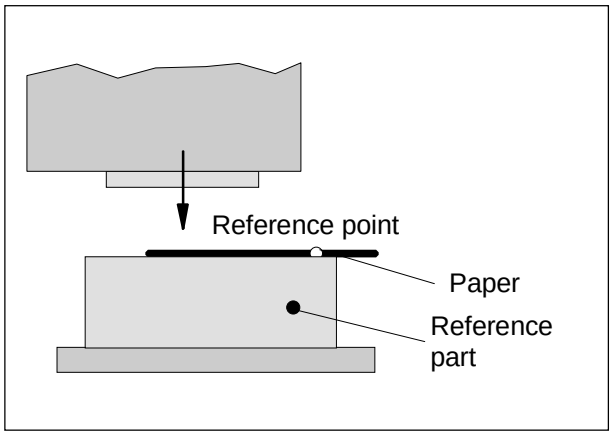




Assembled toolholder

- Select NC-mode
- Approach reference point.
- Assemble toolholder by call-up of the tool library (menu "Tool library", e.g. slot mill cutter 12 mm).

- Click on the assembled tool in the toolholder so that it is active (with hook).
- Click on the switch area "tool measurement".

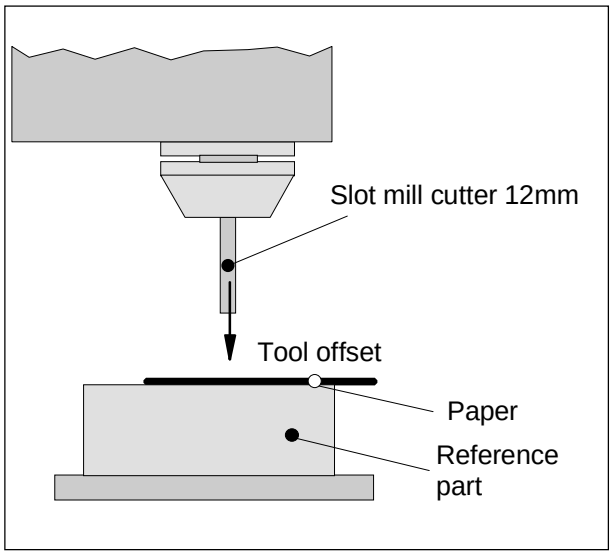


Approach movement reference point

- Empty the spindle so that no tool is assembled. Place a reference part (aluminium log, fluid gauge chamber) on the slide and on this part a sheet of thin paper. Traverse the milling head with the PC movement keys to the reference part in Z-direction. Approach carefully the spindle surface (finally only step by step with minimum feed) until the paper is clamped.

Transferred reference point

- Now this reference point is transferred by means of a mouse click into the input square "reference point Z".



Approach movement reference point

Tool shift taken over

- Lift the milling head again and insert the tool to be measured (slot mill cutter diameter 12 mm) into the spindle.
- Traverse the milling head again to the reference part in Z-direction. Approach carefully the lip of the tool (finally only step by step with minimum feed) until the paper is clamped.
- Now this tool position is transferred into the input square "tool offset Z" by means of a mouse click.
- Leave the window with OK. The tool offset data are saved by WinCAM.

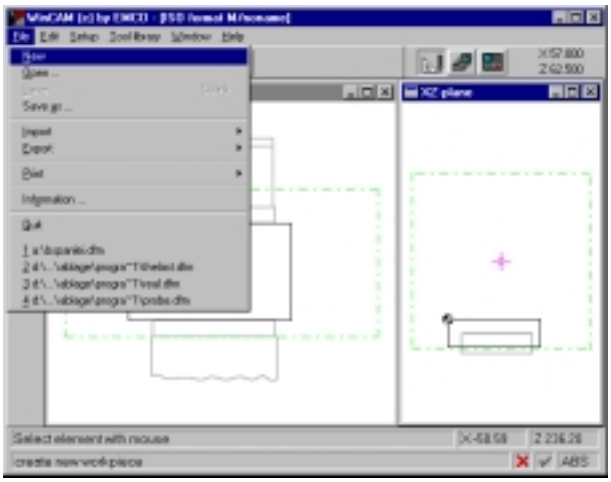
If you want to measure further tools you have to select another tool in the toolholder. Take over the reference point (it has not changed) and measure the new tool offset as described.

Working off the CAD / CAM example

To terminate the NC exercises now the NC-program generated by WinCAM in the CAM exercise part is to be worked off. The necessary settings of the NC-machine have already been explained on the previous pages of this chapter. Thus, the machine is accepted set and referenced.

- Load the example project and thus the NC-program.
- Check if:
  - the unmachined part is clamped tightly
  - the slot mill cutter 12 mm is clamped
  - the chip door is closed
  - WinCAM is in NC-mode
- Start the NC-program with the command symbol "start program".
- Observe the machining procedure. If problems might occur during machining you have 2 possibilities to stop the NC-program:
  - Press Emergency stop -key (danger)
  - Command symbol "stop program" (NC stop for specific interference)

Chapter F Menu lines

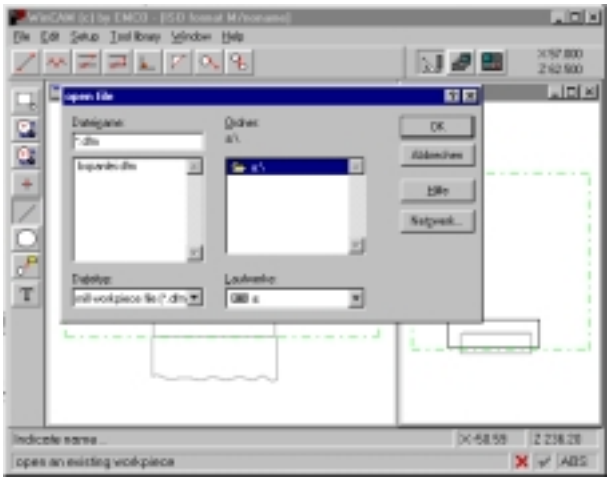


Menu "File"

Menu "File"

New

By means of it you open a new CAD screen and you can work immediately.
If there is already a drawing on the screen it is erased (after a safety inquiry).

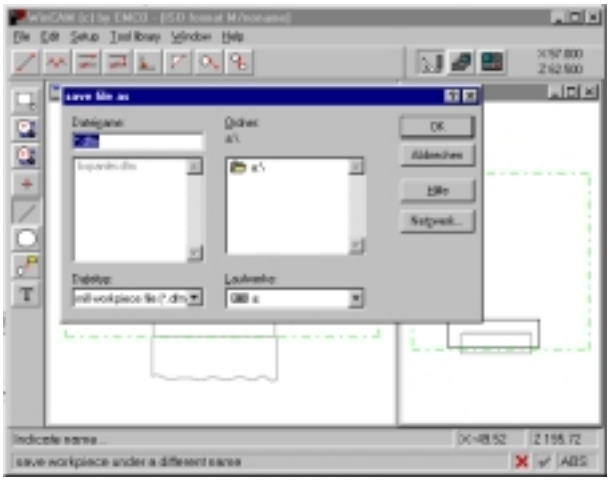


Menu "File open"

Open

With "open" an existing drawing file is loaded. The Windows file window appears for the selection of defined data.
If there is already a drawing on the screen it is erased (after a safety inquiry).

Note:
The last 4 file names stored are listed in the menu "File" on the bottom and can be clicked on directly.



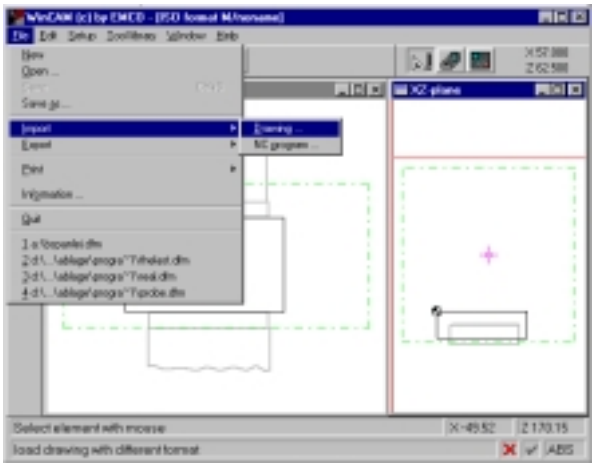
Menu "File save as"

Save

This menu point appears only when there is a drawing not yet saved (in this form) on the screen. The whole defined project is stored automatically under the file name with which it was opened.

Save as

This is the menu provided for saving the whole defined project under a defined file name and in a defined directory. The Windows file window for input and/or selection appears.

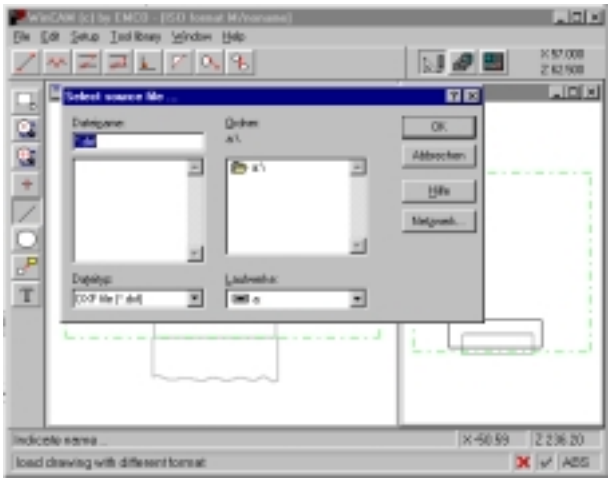


Menu "File, import"

Import

After the input of this menu another menu line is opened which permits the import of a drawing or of an NC-program.

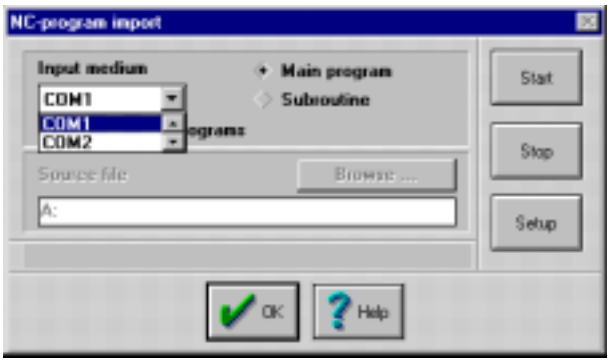
By means of import, data of a purchased program are taken over in a defined standardized format.



Menu "File, import, drawing"

Import drawing

The Windows file window "Select source file" is displayed. Here drawings in "DXF" format can be selected (can be recognized by the extension ".dxf"). Only up to AutoCad version 14.



WinCAM window "NC program import"

Input squares "Main progr. - subroutine"

1 Main program: During import a new project is created (function similar to file NEW).
2 Subroutine: The file is read to the open actual project.

Note: If a main program has various subroutines, you always read in the main program first (to open the project) and then only the subroutines.

Import NC program

The WinCAM window "NC program import" appears. At first make sure that the import interface is set correctly (control area "Setup").

Input medium

Select the connected serial source interface (COM1 to COM4) or the source drive. When selecting "Drive" you can enter a drive letter in the lower input field or perhaps, in addition to it, a path. Then the control area "Browse" is active by means of which the usual Windows file options are called up. Search and select the desired file here.

Input square "overwrite all programs"

1 Without hook: A check is carried out whether an imported file name already exists. An already existing file is not overwritten. A respective message is given and the connection is interrupted.
2 With hook: All programs are read in and possibly existing files are overwritten without message.

WinCAM window "RS232 setup imp."

Control area "Start"
Start of transmission. If a project is opened, a message is given whether the actual project may be overwritten by the import.

Control area "Stop"
The transmission is interrupted.

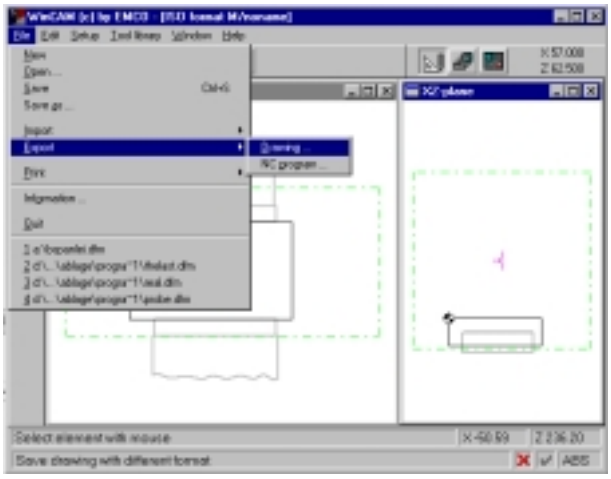
Control area "Setup"
A selection opens the Windows window "RS 232 setup". Here you can optimally set the interface parameters to the import source. If in the input square "default values" the hook is set, the parameters are reset to the default values of the manufacturer. In this case the input windows are screened.

Note:
Import is always carried out into the format of the actual control. Selection by means of the menu "Setup target control".

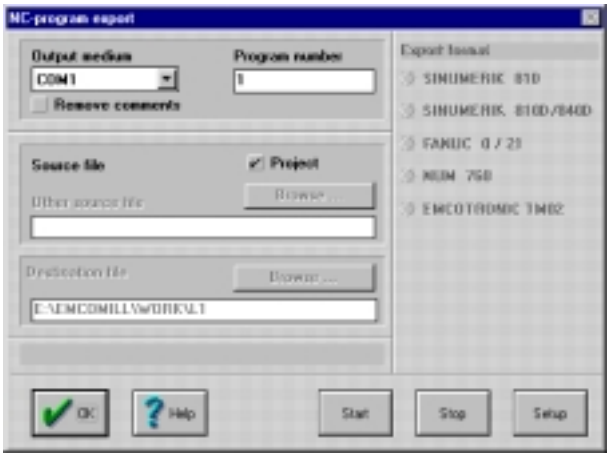
Note referring to the import of NC-programs (tool offset-parameter)
In addition to the correct assembly of the toolholder, also the WinCAM declaration for the tool offsets has to be adhered to. The D-parameter is calculated from the tool number. Figures other than the calculated ones are not admissible.

Calculation formula:
1st possibility
Parameter number = tool number x 2 - 1
2nd possibility
Parameter number = tool number x 2
The 1st and the 2nd possibility refer to the measured cutting tip of the tool.

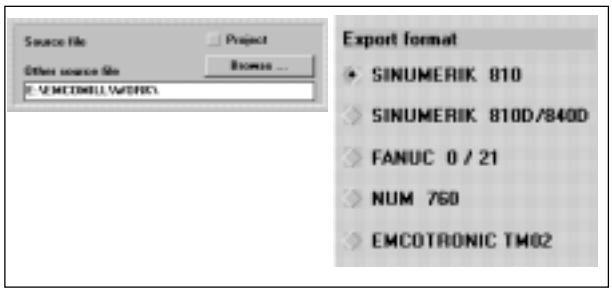
Calculation example:
Tool number T11, parameter D21 (11x2-1 for step miller cutting edge above) and D22 (11x2 for step miller cutting edge below).



Menu "File, export"



WinCAM window "NC-program export"



Exportformate bei deaktivierter Projekteinstellung

Export

After the input of this menu another menu line is opened which permits the export of a drawing or of an NC-program.

Export drawing

The file window "export file" appears. Here enter the desired name with the extension ".dxf". The drawing actually displayed on the screen is saved as DXF file with this file name.
The data of the WinCAM drawing are converted into the standardized transmission format DXF so that in the following these files can be taken over from a purchased program.

Export NC-program

The WinCAM window "NC-program export" is displayed. At first make sure that the intersecting point is set correctly.
Always the NC program of the current project will be exported, corresponding to the respective control.

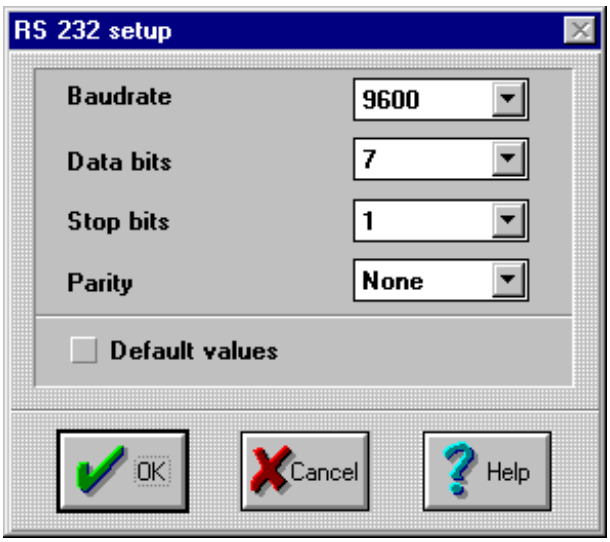
Output medium
Select the connected serial target interface (COM1 to COM4) or the target drive. With the selection "drive" you can indicate a drive letter in the lower input field and possibly, in addition to it, a path. In this case also the control area "Browse" is active with which the usual Windows file options can be called up.

Input square "Remove comments"
Set the hook here if the project to be exported has been created with a WinCAM version prior to 2.0. Reason: The target control does not understand interlaced comments.

When unselecting „Project“ in the input field „source file“, different export formats for your target controls are available.

Control area "Start"
Start of transmission

Control area "Stop"
Transmission is cancelled.



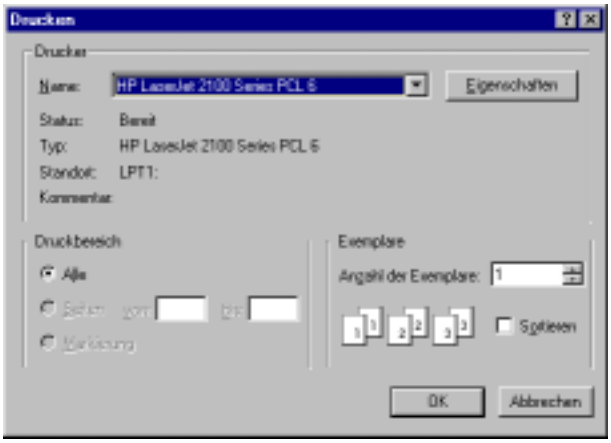
WinCAM window "RS232 setup exp."

Control area "Setup"
A selection opens the Windows window "RS 232 setup". Here you can set optimally the interface parameters to the target control. If in the input square "default values" the hook is set the parameters are reset to the default values of the manufacturer. In this case the input windows are screened.

Notes:

We advise to export the NC-programs into a separate directory since some turning cycles need subroutines which are edited together with them (set up "Project").

According to the target control an exported file may have another file name (required by the control) at the target destination. Thus, there is also compatibility with EMCO WinNC.



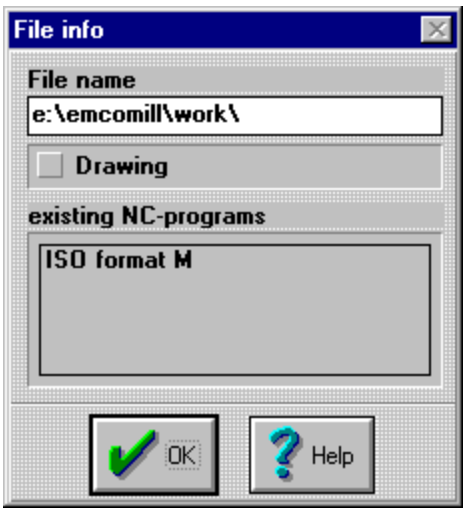
Menu "File, print"

Print

After the input of this menu another menu line is opened which permits printing a drawing or an NC-program.

Print of drawing
The Windows window "Print" appears. Click on OK if you agree with the preset printer setting. Otherwise you can open the Windows window "Printer setting". Please consult your Windows manual.
The actual drawing is transmitted to the Windows printer.

Print of NC-program
The Windows window "Print" appears. Click on OK if you agree with the preset printer setting. Otherwise you can open the Windows window "Printer setting". Please consult your Windows manual.
The actual NC-program is transmitted to the Windows printer.



Menu "File, information"

Information

After the input of this menu the WinCAM window "File, information" appears. This window gives you information about the project just in work.

Project file
A project file is created automatically as soon as a drawing or an NC program exists. The name of the project file is indicated in the window under "file name".

Drawing file
If there is a drawing for this project it is indicated by a hook at "drawing".

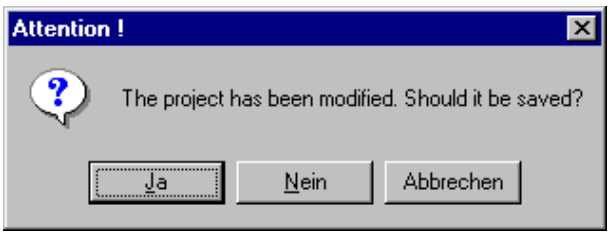
NC-program files
In the window "existing NC programs" those formats are displayed with which existing NC-programs for this project have been saved.

File identification of a project:
As example we use the project name "TEST". The individual files have the following names:

Project file TEST.DFM
(DFM= description file mill)

CAD drawing file TEST.DW1
(DW = drawing)

NC program file TEST.M03
(M = machine, 03 = code of the NC control)

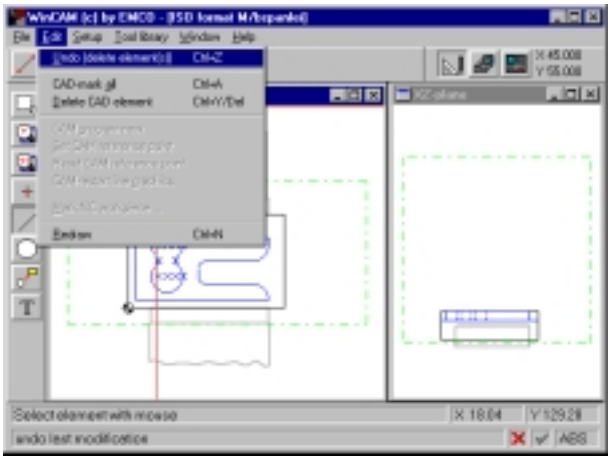


Menü "File, quit"

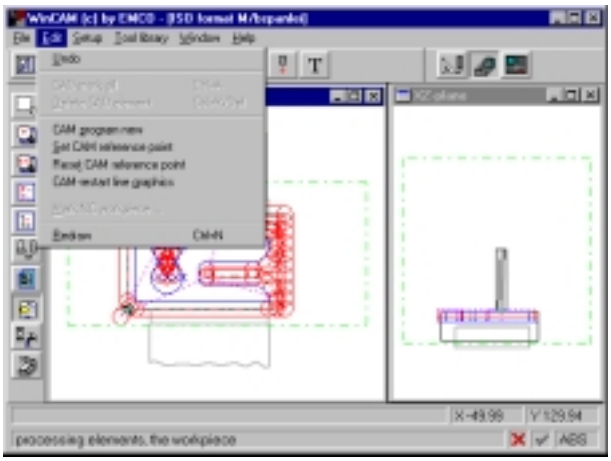
Quit

After the safety inquiry (Should it be saved?) the window WinCAM is closed and the program is terminated.

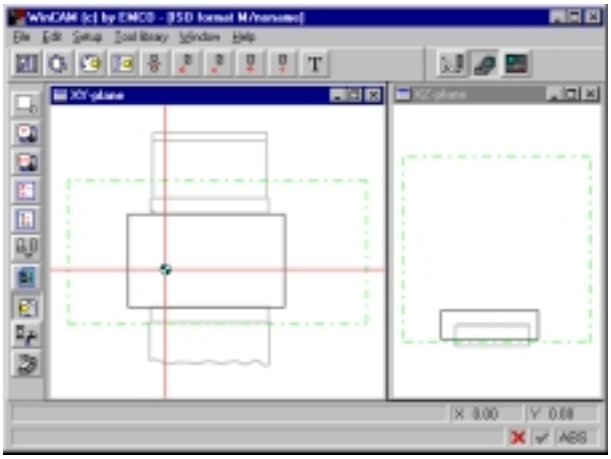
Further possibilities to terminate the program are to close the WinCAM window by means of the double click or with ALT+F4 or to terminate the task. Please look up your Windows manual.



Menu "Edit in the CAD mode"



Menu "Edit in the CAM mode"



Set CAM reference point

Menu "Edit"

Mind that in CAD mode the CAM commands are displayed screened and, in inverted order, in CAM mode the CAD commands are screened. Only in the NC-mode the command "Mark NC" workpiece is not screened. Screened commands cannot be selected. Therefore change the WinCAM mode.

Undo

Beside Undo the last CAD-operation is displayed. After the selection of the function the last CAD-operation is cancelled and the original state is reestablished. If the function "Undo" is displayed only grey WinCAM has not got anything in the memory which could be cancelled. 10 CAD-operations can be cancelled one by one.

CAD-Mark all

All drawn elements are marked.

Delete CAD element

The various delete functions are described precisely in chapter B. Please consult this chapter.

CAM program new

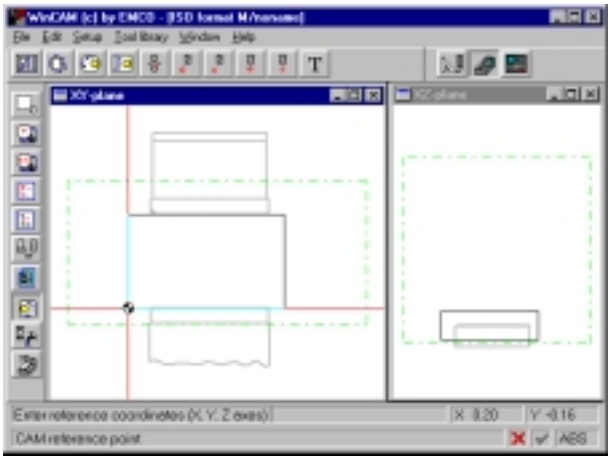
After confirmation of the safety inquiry the existing CAM program is deleted (CAM program = NC-program for the set target control as well as for the entry in the CAM-editor) and all settings are reset by means of this function.

Set CAM reference point

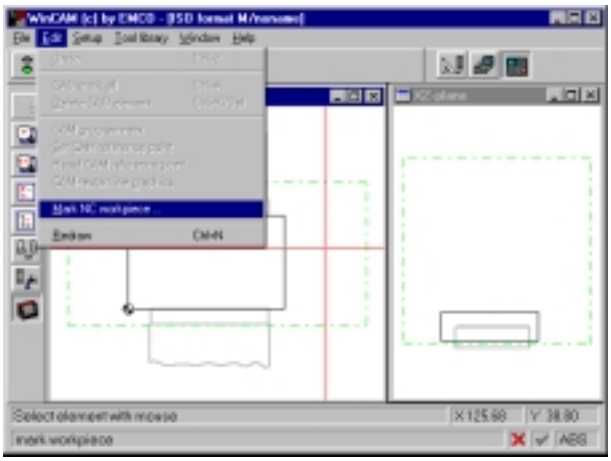
WinCAM sets the CAM reference point automatically at the left front edge of the unmachined part (XY-level) or at the left top edge of the unmachined part (XZ-level).
By this function only the CAM coordinate zero point with effect on the CAM commands can be shifted. It is independent from the CAD reference point.

The position can be entered via:

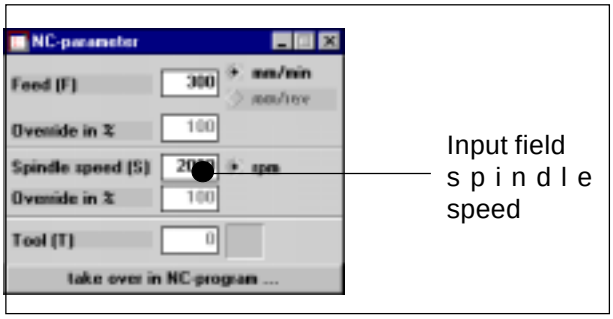
1. present cursor position and mouse click
2. with the catch mode menu and mouse click
3. input of the shift coordinates in X,Y,Z



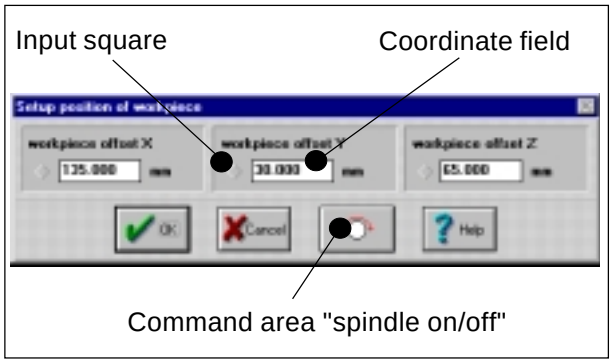
Reset CAM-reference point



Menu "Edit in the NC mode"



WinCAM window "NC parameter"



*WinCAM window "*setup position of workpiece*"*

Reset CAM reference point

WinCAM resets the CAM reference point again on the left front corner of the unmachined part (XY-level) and/or at the left top corner of the unmachined part (XZ-level).

Start CAM line graphics

The CAM simulation is started and, thus, the graphics is updated.

Redraw

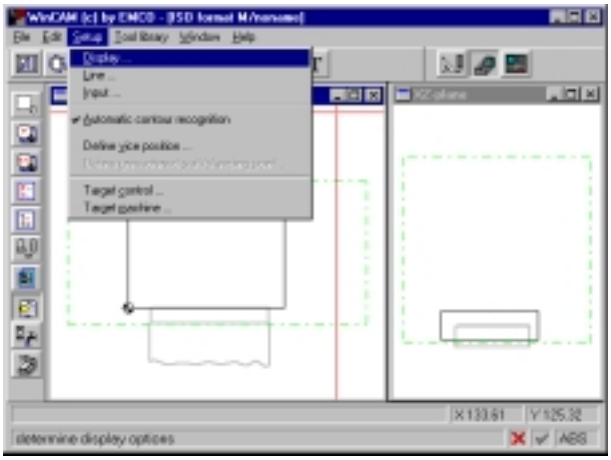
After entering the function the CAD screen is newly built up.
After delete and change functions it may occur that lines are displayed only incompletely on the screen. If this is the case use the function "redraw" or the zoom commands (full picture, zoom drawing elements) to get a new representation of the screen.

Mark NC workpiece

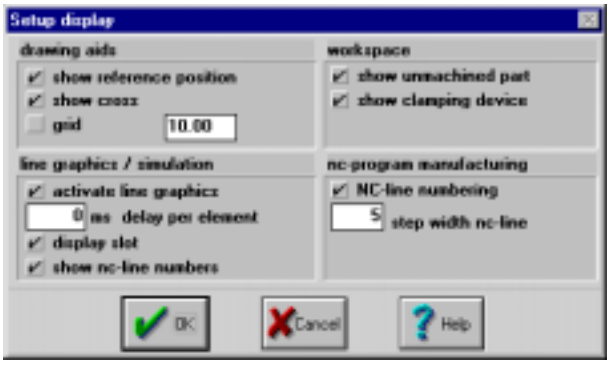
This function can only be selected in the NC-mode and serves for referencing the CAM reference point with the actual unmachined part in the machine.
Please mind that "mark NC workpiece" is an additional function and in general this referencing is fixed automatically by the specifications "define vice position" or "define unmachined part/clamping point" .
Please proceed as follows:

- Select the tool (using the command symbol "change tool") which you use for marking (The tool must have already been measured).
- Set an adequate spindle speed in the WinCAM window "NC parameter" (open the window by means of the command symbol "NC parameter on/off"). The window can be opened without problems.
- Select the command "mark NC workpiece". The WinCAM window "setup position of workpiece" is opened.
- Switch on the spindle by means of the command area.
- Approach by means of the NC movement keys (figures 4,6 X - 1,9Y - 2,8Z) successively the left front top corner point of the unmachined part in the directions X,Y,Z and acknowledge each time the displayed coordinate with the input square.
- Switch off the spindle again by another click on the command area.

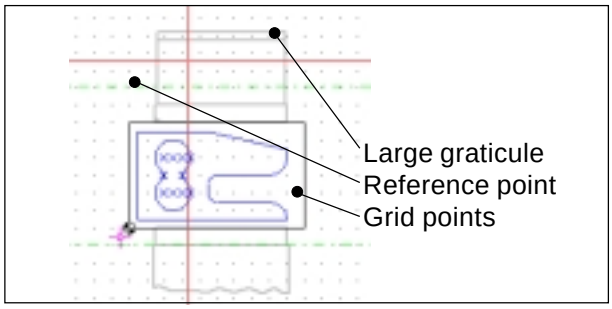
If you know the coordinate values you can also enter them directly into the coordinate field.



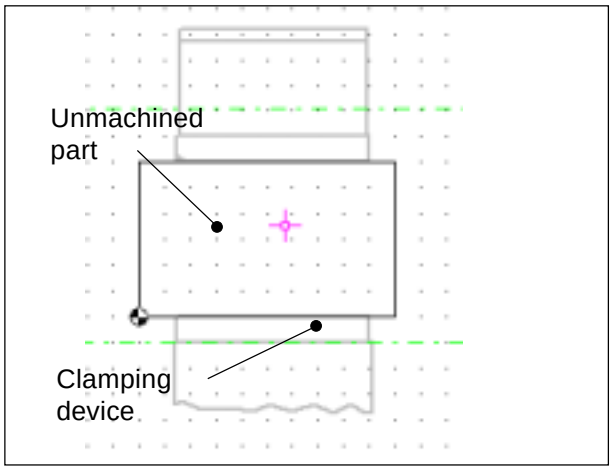
Menu "Setup"



WinCAM window "setup, display"



Explanations of the displays



Explanations of the displays

Menu "Setup"

Mind that according to the WinCAM mode some commands can be displayed only in screened form. Screened commands cannot be selected. Therefore change the WinCAM mode (CAD, CAM, NC).

Display

After the input of the command the WinCAM window "setup display" appears.

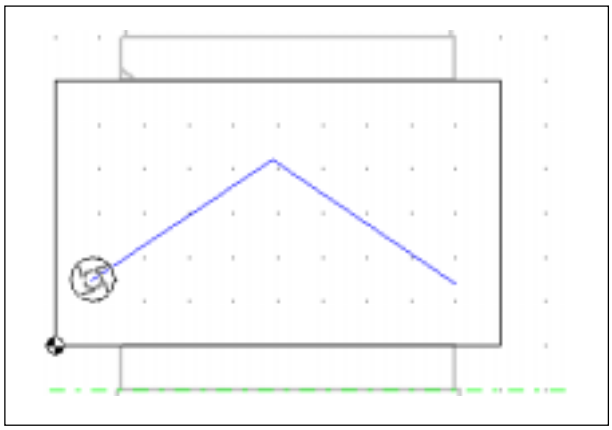
Show reference position
The coordinate zero point is displayed only when there is a hook in the input square. Click into the input square to change the status.

Show cross
Shifting the mouse cursor in the CAD window from a small graticule to a large window-filling graticule (with hook).

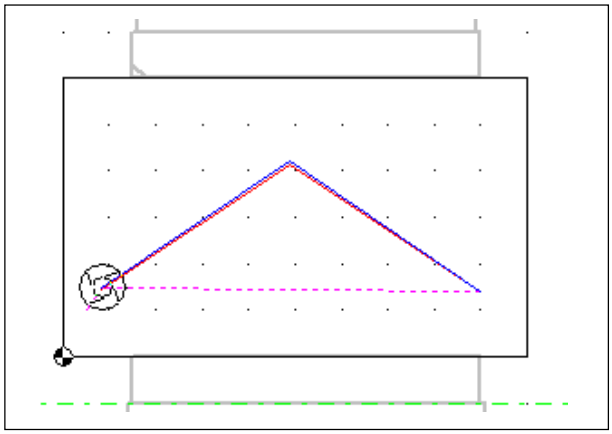
Grid
Grid points are overlayed as orientation or drawing help. The grid begins in the reference point. The grid points have the distance indicated in the opposite input field. You can enter the desired grid width into the input field.

Show unmachined part
The unmachined part is overlayed in the windows of the XY and XZ-levels.

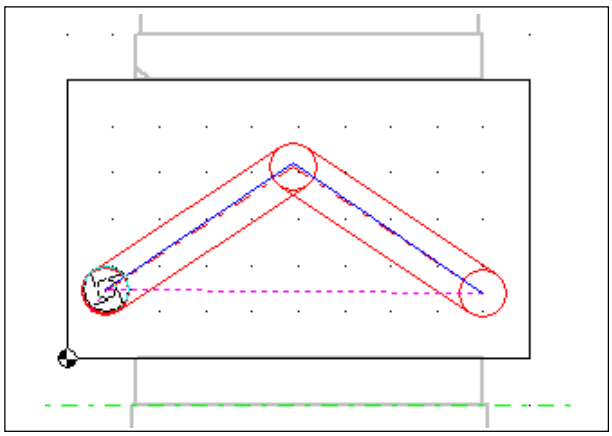
Show clamping device
The clamping device is overlayed in the window of the XY-level.



Display - envelope and line grahics off



Display - envelope off and line graphics on



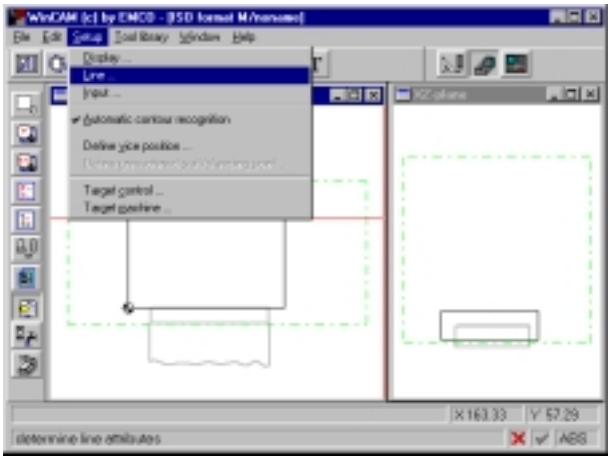
Display - envelope and line graphics on

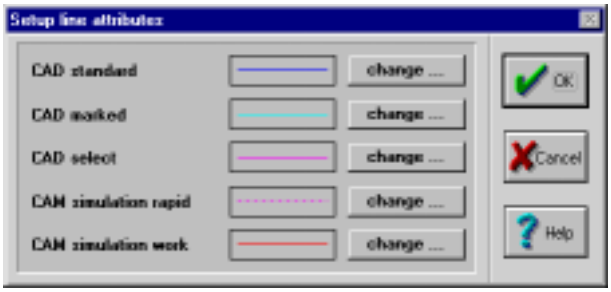
Activate line graphics
This function switches on the display of the traversing paths (tool centre) in CAM mode.

Display slot
If this function is active the whole area reached by the tool is displayed. The envelope is the travelling circumferential range of the rotating tool.

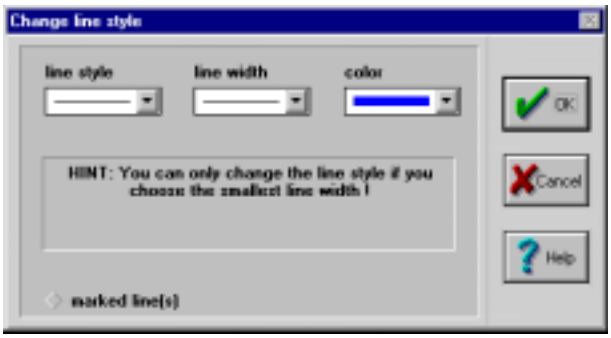
Delay per element
If no time is indicated here, the CAM simulation is performed so fast that following the individual traverse movements is difficult. Set here an attendance time and this time is slipped in after each traverse movement (NC-block)
With periods of 200 up to 300 ms the individual movements can be followed well. However, with larger NC programs the simulation takes respectively longer.
By means of the "ESC" key the rest of the NC-program is simulated without attendance time.

Esc

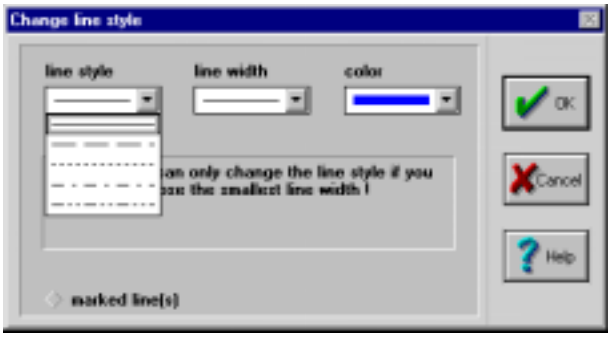




Menu "setup, line"



Menu "Setup, line, change"



Menu "Setup, line, CAD standard, change"

Line

Here the WinCAM window "setup line attributes" appears. Here you can set the line style, the line width and the color of the lines displayed in the CAD or CAM window.
The following line groups can be changed:

CAD standard
Lines for representation of the drawing (workpiece contour).

CAD marked
Lines which have been marked by clicking on them (one or more).

CAD select
The lines indicated for selection by WinCAM (with commands which offer various solution possibilites - e.g. tangents).

CAM simulation rapid
The NC-path lines travelled in rapid motion displayed by WinCAM.

CAM simulation work
The NC-path lines travelled with feed displayed by WinCAM.

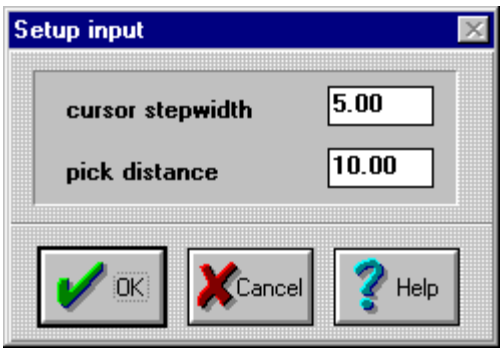
Click "change" at the desired line group. The WinCAM window "change line style" appears. Click on one of the arrow symbols and a selection window is opened.

Line style
Select one of the indicated line types (full line, long broken line, short broken line, dot-dash line, dot-dash-dash line). Only the full line can be displayed in all line widths. For the other line types the smallest line width has to be used.

Line width
Select one of the 5 indicated lines varying in width.

Color
Select one of the 8 displayed line colors.Take into consideration that the color range depends on the basic setting of Windows.

Marked Line(s)
Square not marked: From now the newly set line attributes are valid for all lines.
Square marked with cross: The set line attributes are valid only for the lines marked previously (e.g. for subsequent change of the line colors).



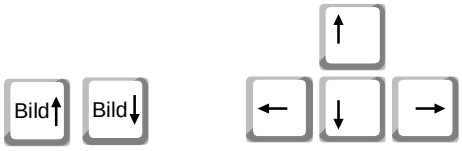
WinCAM window "Setup, input"

Input

After the selection of the command the WinCAM window "Setup input" appears.

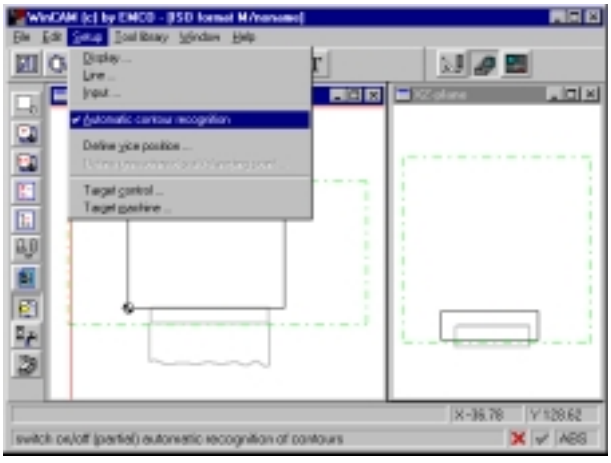
Cursor step width
The mouse cursor can also be moved by means of the arrow keys (unless the input of a coordinate is expected).

The step width per arrow key can be indicated in the input field.



You can change the step width (in fixed jumps) also using the keys Page Down and Page Up. This is only possible if you are in the mode where you can move the cursor by means of the arrow keys. Observe the status display.

Pick distance
The pick distance is the area round the cursor cross which is searched (picked) by WinCAM during the selection of elements.
Enter the pick distance into the input square.



Automatic contour recognition on

Automatic contour recognition

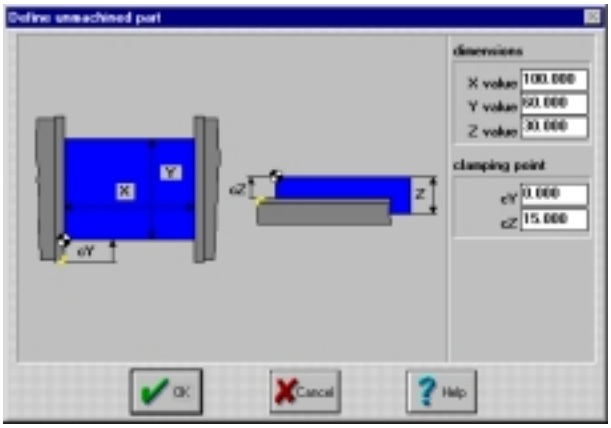
When selecting machining cycles the indication of a contour to be machined is expected.

WinCAM is able to recognize connected drawing geometries and thus to recognize a contour. This facilitates the contour indication. In this case you have to switch on the function (with hook).

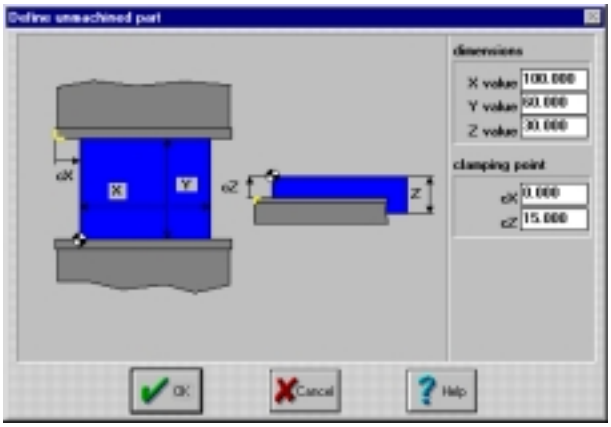
With very complex geometries it may be easier to define the contour by selecting individual geometries (e.g. many ramifications or if an omission of defined contour sections is desired. In this case switch off the function (without hook).

WinCAM window "setup position of clamping device"

Hint:
In chapter E NC-examples the definition of the machine vice position is described in detail by means of an example.



Vice position X



Vice position Y

Define vice position

It serves for measuring the vice position in the CNC machine. It is measured to the left front upper fixed vice jaw edge. The offset dimensions refer to the machine zero point.

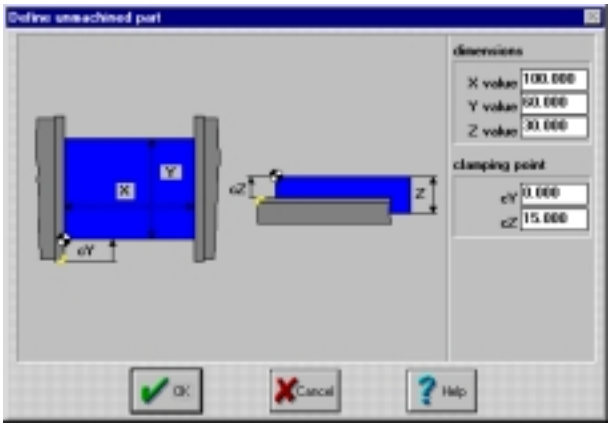
Please proceed as follows:

- Select the tool (by means of the command symbol "tool change") which you use for measuring. It does not matter if you use an edge tracer or an optical device. WinCAM must only be informed about the tool (diameter, length). If necessary, define the tool with the tool library.
- Set an appropriate spindle speed in the WinCAM window "NC parameter" (open the window with the command symbol "NC parameter on/off"). The window can be opened straight away. Switch on the spindle. This item is omitted with optical measuring.
- Approach the vice reference point by means of the NC movement keys (figures 4.6 X - 1.9 Y - 2.8 Z) subsequently from the directions X,Y,Z and acknowledge the indicated coordinate each time by means of the input square. The determined coordinate is transferred into the input field.

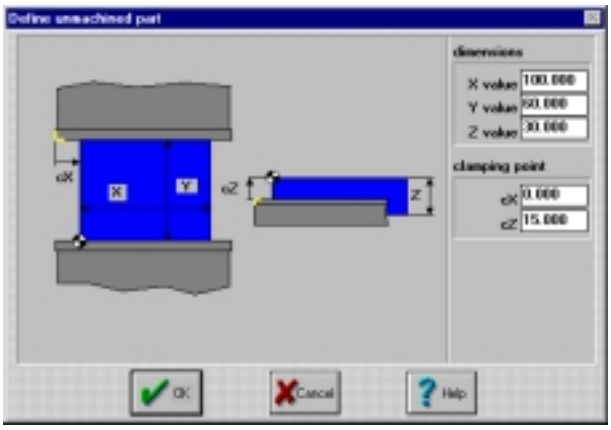
If the coordinate values are known you can enter them directly into the coordinate field. With WinCAM working places without NC-machine here a value can be entered which shows the clamping device in-centre in the working area.

Orientation of clamping device
According to the connected NC-machine the vice can be mounted in X- or Y-direction. Adjust the screen representation here. Click on the corresponding input field.

Note:
The vice position has to be defined only in case of initial operation of the NC-machine and with a modification of the vice.
The vice position has also an influence on the representation in the simulation. If you can see the unmachined part and/or the vice only partially or not at all you have to check (or change) the offset data.



WinCAM window "define unmachined part" with vice position in X-direction



WinCAM window "define unmachined part" with vice position in Y-direction

Define unmachined part / clamping point

After calling-up the function the WinCAM window "define unmachined part" is displayed. The kind of representation of the vice depends on the definition of the vice position (mounted in X- or in Y-direction) in the WinCAM window "*setup position of clamping device"*.

By means of this function the dimension of the unmachined part and its position in the vice are defined. Thus, also the coordinate zero point for the CAD and the CAM-mode of WinCAM is automatically defined.

The vice position in the working area of the machine is set in the WinCAM window "*setup position of clamping device"*.

Enter the following into the window:

Dimensions

X-value	length of the unmach. part in X-direct.
Y-value	width of the unmach. part in Y-direct.
Z-value	heigth of the unmach. part in Z-direct.

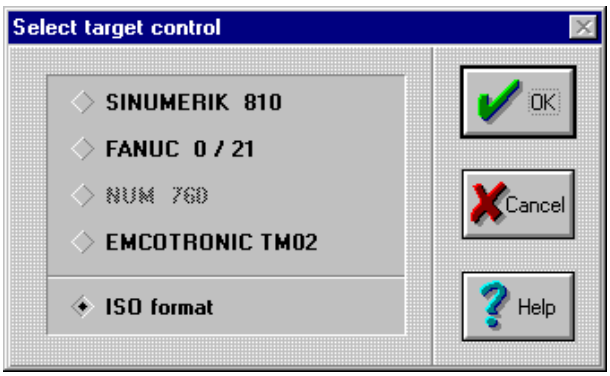
Clamping point

Offset of the coordinate zero point of the unmachined part in relation to the vice coordinate point.

cX	offset in X-direction
cY	offset in Y-direction
cZ	offset in Z-direction

Note
In chapter E NC-examples the definition of the unmachined part/clamping point is exactly described by means of an example.

Note
According to the vice position either the cX or the cY-value is indicated. Then the value is automatically 0 and thus cannot be entered.



WinCAM window "select target control"

Hint:
The target controls (e.g. Sinumerik, Fanuc) are not supported by the standard version. They are supported optional.

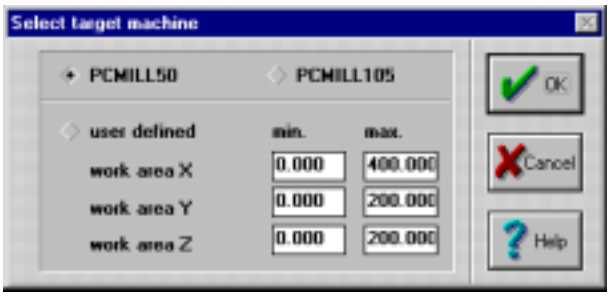
Target control

After the input of the function the WinCAM window "*select target control*" is displayed. Here select a control by clicking on the input square.

This input has an influence on the NC-program (in the editor) generated by WinCAM. If a control (e.g. Sinumerik or Fanuc) is selected its cycles are alternately assisted whereas in case of the selection of DIN all cycles are risolved in single movement steps.

For a project NC-programs for various target controls can be generated. However, mind that only the NC-program selected here is processed by WinCAM (in the editor). If you need, e.g., the NC-program for all possibilities, processing in CAM mode has to be carried out separately for each target control.

The NC-programs are managed for each control independently from each other. During saving and loading (menu file) the NC-programs of all target controls are saved and loaded.



*WinCAM window "*select target machine*"*

Target machine

After entering the function the WinCAM window "select target machine" is displayed.

Here the working area of the NC-machine managed in WinCAM is defined. Other settings (such as e.g. the vice position) are based on them.

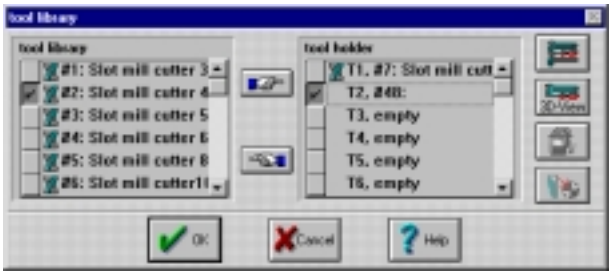
PC Mill 50:
If you have connected a "PC Mill 50" to your PC click in this input square.

User defined:
If you have got another machine type click on the input square "user defined" and enter the working area of the 3 coordinates X,Y,Z .

Both input fields for each coordinate define the working area - from (1st input field) to (2nd input field) - in relation to the machine zero point.

Note:
With a self defined target machine an EMCO-NC-machine (PC-Mill 50) cannot be controlled in NC-mode.

Menu "Tool library"

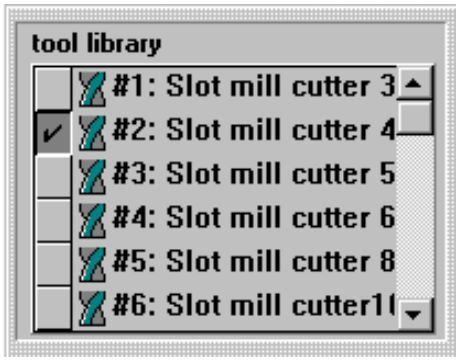


*WinCAM window "*tool library*"*

After the selection of the menu line the Win CAM window "tool library" is overlayed. Here the tool system of the NC machine can be handled:

- Defining a new tool and adding it to the tool library.
- Erasing an existing tool.
- Removing a tool from the tool library and inserting it into the toolholder.
- Removing a tool from the toolholder and returning it to the library.
- Measuring a tool at the machine.
- Define a tool for the 3D-simulation.

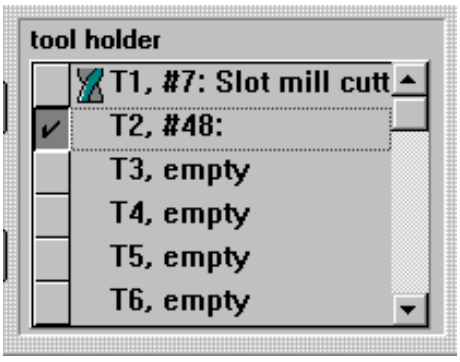
Tool library



Field tool library

50 tools can be stored in the library. Each tool has a tool number between 1 and 50.
In the field tool library a symbol of the tool type, the tool number (e.g. #1), as well as the tool name are indicated. At the left of the field a tool can be selected (hook).
If a tool is in the toolholder it is indicated as "assembled" and thus cannot be handled.
The name of the tool will also be displayed in the bottom line of the screen.

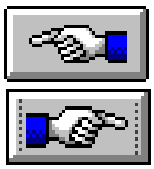
Tool holder



Field tool holder

In the toolholder of WinCAM tools can be assembled just like with the NC-machine in the tool turret.
In the field a symbol of the tool type, the tool post number, the tool number (e.g. #2), as well as the tool name are indicated. At the left of the field a tool can be selected (hook).
The note "T?,empty" means that there is no tool clamped in this tool post.
The name of the tool will also be displayed in the bottom line of the screen.

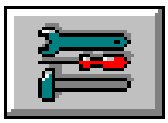
Switch areas "Transfer"



By selecting the respective hand:

- the selected tool of the tool library is transferred to the selected toolholder post.
- the selected tool of the toolholder post is returned to the tool library.

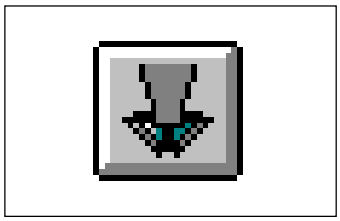
Switch area "Tool data"



After the selection the WinCAM window "tool data" is opened.
Mind that this function is only available if you have selected previously a tool in the tool library.
Now you can define the tool with the selected number in the window "tool data".

Tool data

WinCAM window "tool data"



Angle cutter

Note:
A change of a tool can only be effected from the library. If the tool is in the holder it has to be returned before to the library.

This window permits you to specify the tool for the tool number selected before.

Tool type
Click on an adquate icon. This is then displayed in the tool library and in the tool holder and serves for control.

Tool angle
In general here you enter 0 degree for driller and milling cutter. For angle cutters enter the respective angle.

Tool length
Here you enter the length of the tool in Z-direction. This is a decisive factor for the simulation in the XZ-level.

Cutting radius
Enter the radius of the cutting edge of the driller or milling cutter (not the diameter). This value is necessary for the simulation in XY-level and the calculation of the radius compensation .

Tool name
The name entered here appears in the tool library, the toolholder and in the editor.

Comment
The text entered here serves only for further comments on the tool.

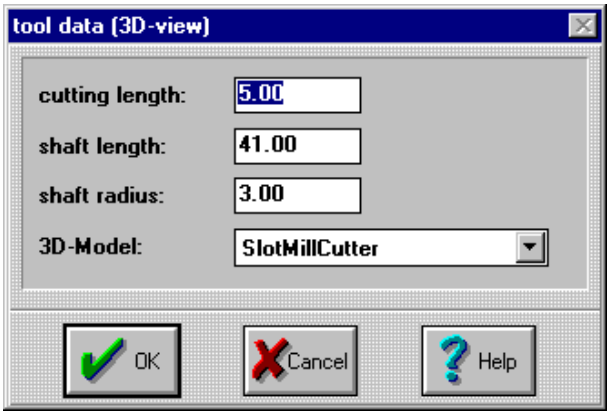
Switch area "Delete"

After entering the function the tool selected before is erased from the library. A safety inquiry is integrated.

Note:
This function is only available if you have previously selected a tool in the tool library.
Erasing a tool is only possible from the tool library. If the tool is in the holder you have to return it previously to the library.

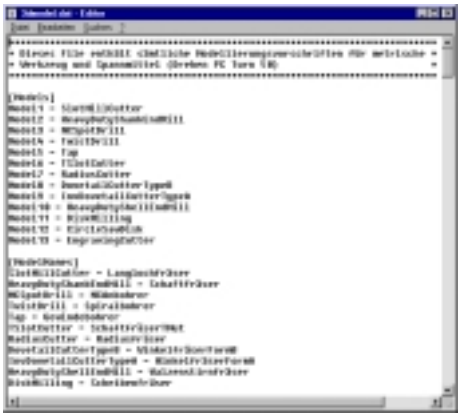
WinCAM safety inquiry

Control area "3D-View"



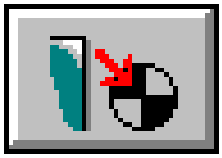
WinCAM window "Tool data (3D-View)"

After the input of the function the WinCAM window "Tool data (3D-View)" appears. It shows the actual 3D-model for the selected tool. For all WinCAM standard tools the respective 3D-models are already defined. However, it is possible to define special forms of a tool (listing 3D-model) and/or completely separate tools.



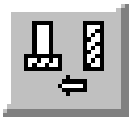
File "3dmodel.dat"

All data for the 3D-simulation of the tools are contained in the file "3dmodel.dat" in the directory ".\wincam\work\mill50". This file can be modified with the Windows editor.
The definition of the tools is very complex.
For creating your own tools you will find a description in the chapter "Creading Tool Models".

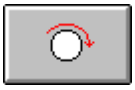


Note:
In chapter E NC-examples the measurement of a tool is exactly described by means of an example.

WinCAM window "tool measurement"



Switch area "Change tool"
By means of the switch area the WinCAM window "tool change" is opened. With it you switch from the reference point to the tool to be measured. This is not a function which serves to switch on to a further second tool to be measured.



Switch area "Spindle"
If the tool measurement is carried out with running spindle the spindle can be switched on and/or off with this switch function.

WinCAM window "Reference tool milling"

Switch area "Tool measurement"

After selecting the switch area the WinCAM window "tool measurement" appears.

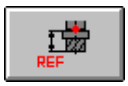
This function is only available in the NC-mode and if a tool has been selected in the toolholder.
The tool measurement is necessary for the tool in the NC-machine so that WinCAM can compensate the new position of the cutting edge of the tool after a tool change.

Tool measurement

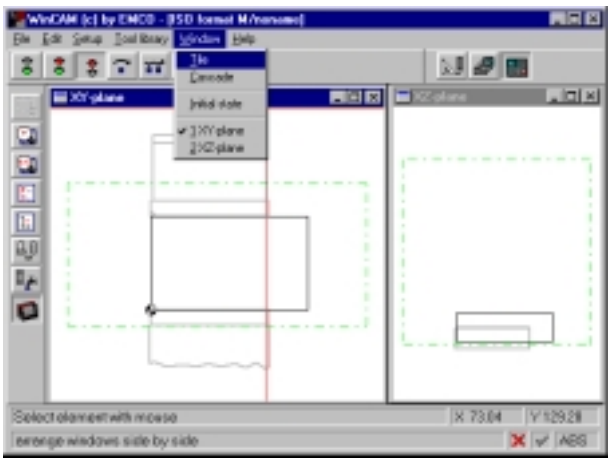
The following steps are necessary for measuring a tool:

- Assemble toolholder by calling up the tool library.
- Click on the assembled tool in the holder so that it is active (with hook).
- Click on the switch area "tool measurement".
- Referencing the spindle area (without clamped tool) on a reference part (in Z-direction) as reference position.
- Take this reference position over into the input square (reference point Z).
- Reference the tool tip on the reference part as tool offset.
- Take this tool offset over into the input square (tool offset Z).
- Escape from the window with O.K. Only then the new tool data are also saved.

Now the tool is measured and the tool coordinates are saved by WinCAM. For the measurement of another tool call up the function "tool measurement" a second time (the reference point can be taken over) after the selection of the respective tool in the toolholder.



Control area "reference tool"
By means of the control area "reference tool" the WinCAM window "reference tool milling" is opened. Here enter the X, Y or Z-offset values of the reference tool. The offset values are the dimensions of the reference tool by which the reference point is offset with regard to the tool coordinate zero point during the tool measurement. Thus, you save the manual calculation of the tool coordinates to be measured.



Menu "window"

Menu "Window"

With this menu the WinCAM windows of the XY-plane and of the XZ-plane are changed.

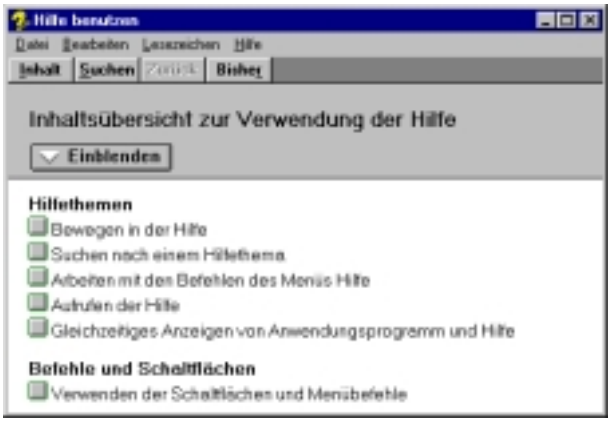
Tile
Both windows of the XY- and of the XZ-plane are arranged side by side.

Cascade
Both windows of the XY- and of the XZ-plane are arranged overlapped.

Initial state
Both windows of the XY- and of the XZ-plane are arranged just as during start of WinCAM.

XY-plane
The window of the XY-plane is activated. If the window has been set down as symbol then it is opened.
The active window is marked by a hook.

XZ-plane
The window of the XZ-plane is activated. If the window has been set down as symbol then it is opened.
The active window is opened by a hook.



Menu "help, using help"

Menu "Help"

The help menu in WinCAM is set up in accordance with the Windows help functions.

Using help

If you are not familiar with the operation and/or the possibilities of the Windows help you will get all the information by means of the menu function "using help".

Menu "help, short instructions"

Short instructions

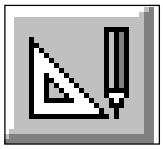
Here the WinCAMhelp window is opened. You can go through the help pages or select a catchword in the index. The best thing to do is to try out the help function yourself.

Menu "help, about WinCAM"

About WinCAM

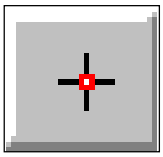
After the input of the function the WinCAM information window is displayed.

Chapter G CAD commands

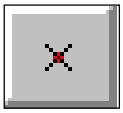


CAD mode
By clicking on the shift symbol "CAD" the CAD command symbols are activated. The CAD mode is active until it is deselected again with CAM or NC.
After the start of WinCAM the CAD mode is automatically activated.

The 3 direct commands have already been explained earlier.



Point menu



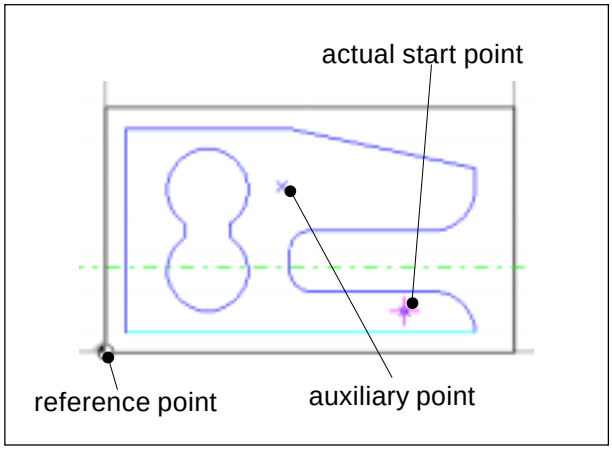
Set auxiliary point

After the selection of the symbol the position of the next auxiliary point is inquired.

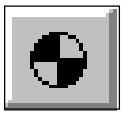
The position can be entered by:

1. the actual cursor position with left mouse key
2. the absolute coordinates X,Y or X,Z (from the reference point).
3. the incremental coordinates X,Y or X,Z (from the actual start point).

Auxiliary points are used as construction help during the creation of a drawing. Auxiliary points are recognized by the pick mode menu (right mouse key) "search point". After setting the auxiliary point the command is automatically deselected. An auxiliary point can be deleted after it has been selected by means of the normal delete functions.



CAD auxiliary point



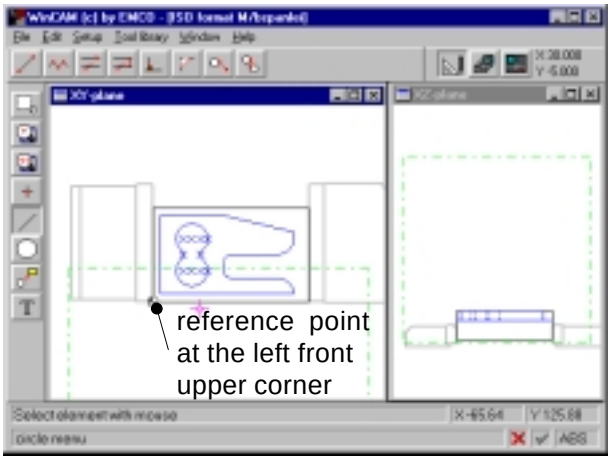
Set reference point

The CAD reference point is automatically set on the left front upper corner of the unmachined part.

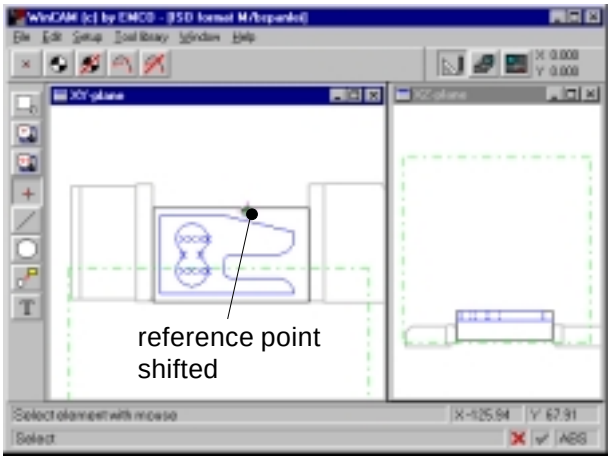
With this function the reference point and thus the coordinate system can be shifted from its present position.

After the selection of the symbol WinCAM expects a coordinate input in the status line which is to be terminated with ENTER.
If the absolute value input is set (ABS) the shift dimension is based on the old CAD reference point.
If the incremental value input is set (INC) the shift dimension is based on the actual geometry start point (marker position).

The reference point is to be set separately for each window (XY- and XZ-plane).



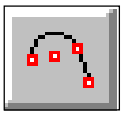
CAD reference point - standard setting



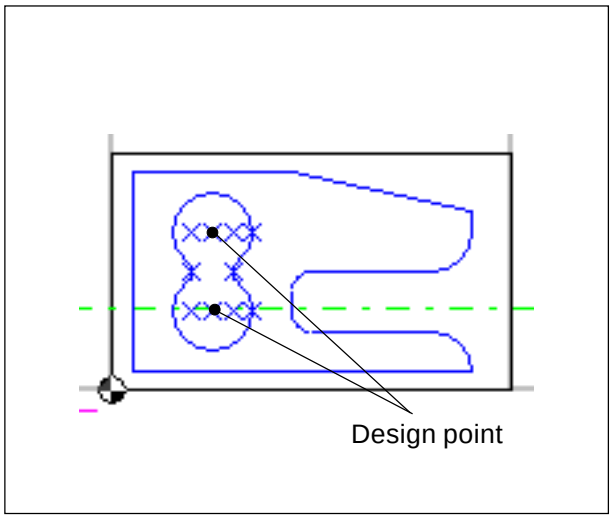
CAD - reference point shifted

Reset reference point

After selecting this command the reference point is reset on the left front upper corner of the unmachined part.



Display design points of marked elements

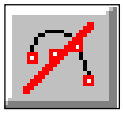


Design points

During the design of elements certain coordinate points are set which define the elements.

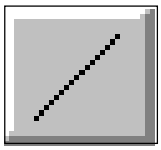
By this function only those design points are set which belong to marked elements.

These design points (exept circle and arc centre points) are also activated by the search function "search point". Here it does not matter if the points have been made visible or not.

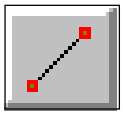


Suppress design point(s)

By means of this function the display of the design points is faded out again. However, they are not deleted and can thus be faded in again with the above function.



Line menu



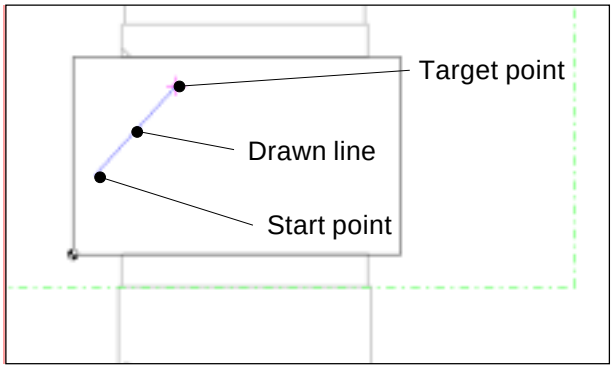
Draw line

After selecting the symbol the start point of the line has to be entered. This can be carried out by:

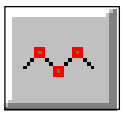
1. present cursor position and mouse click
2. with the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

Then you have to indicate the target point of the line.

Start and target point of each line are saved as design points.
The command is not selfholding and thus must be reactivated for each line.
If you have to draw various lines in succession you better use the command "poly line".



Draw line

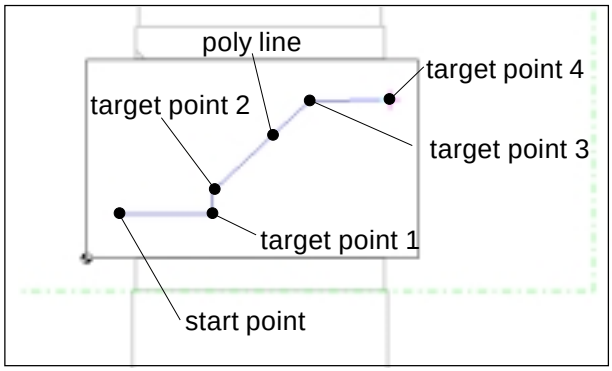


Poly line

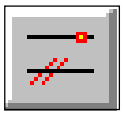
After the input of the start point the first target point has to be entered. The first line of the poly line (polygon) is drawn at once. Now the command expects the input of the next target point and so on.

Each entered point is saved as design point.

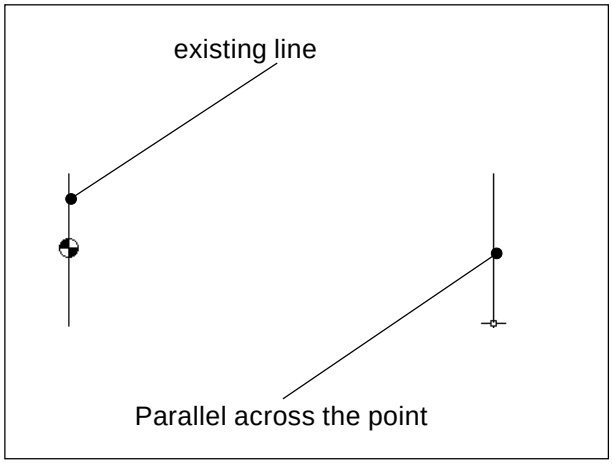
The command is selfholding and must be interrupted (command symbol, "Esc"-key or click input field interrupt).



Poly line



Parallel with point indication

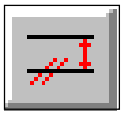


Parallel with point indication

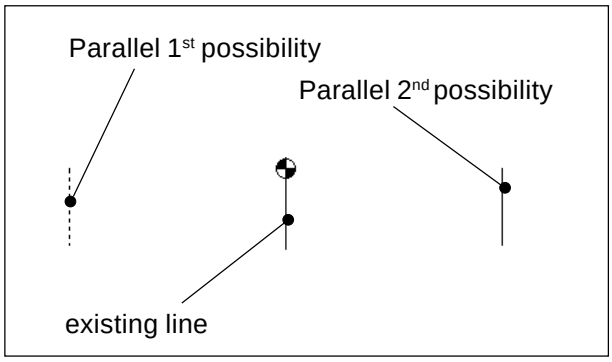
After the selection of the symbol the element to be copied in parallel has to be selected. Subsequently you have to specifiy a point across which the parallel shall run.

Both parallel shifted end points of the lines are saved as design points.

The command is not selfholding and must be reactivated for each parallel.



Parallel with distance

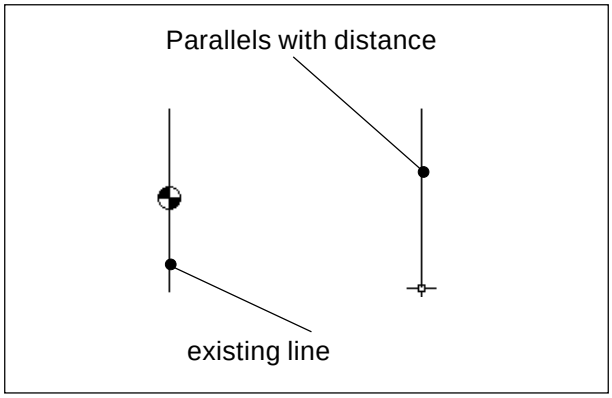


Possible parallels

After selecting the symbol the element which is to be copied in parallel is to be selected. In the subsequent the distance is to be entered in which the parallel shall run.

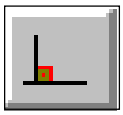
Since here there are two possible parallel lines the desired one is to be selected with "space" and to be confirmed with the key "ENTER". The actual parallel is drawn in full line, the inactual one in broken line.

Both parallel shifted end points of the lines are saved as design points.

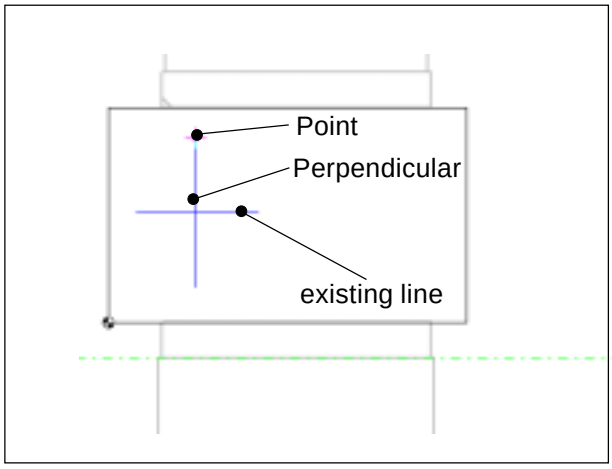


Parallel with distance

The command is not selfholding and must be reactivated for each parallel.



Perpendicular

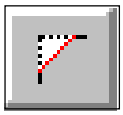


Perpendicular

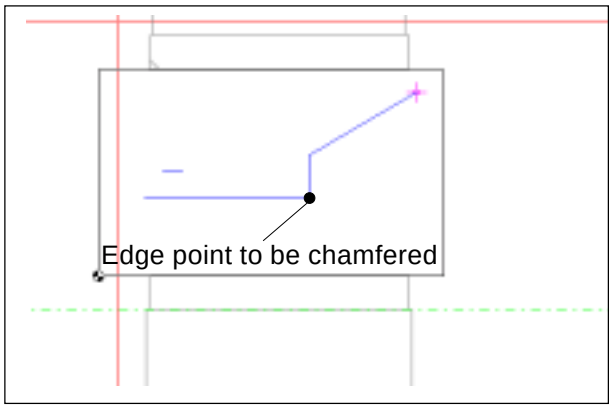
After the selection of the symbol the line to which the new line is to be constructed in perpendicular has to be selected. Subsequently a point has to be indicated up to which the perpendicular shall lead.

The resulting intersection point of the perpendicular to the existing line is saved as design point.

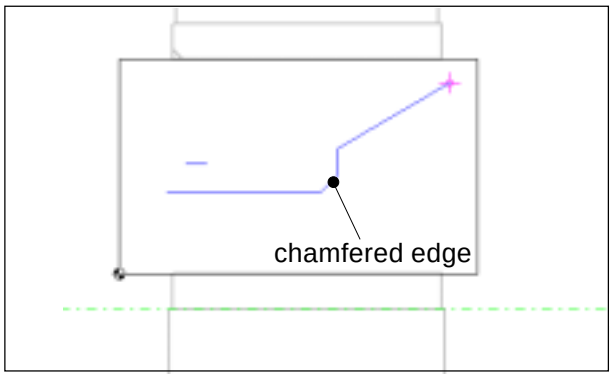
The command is not selfholding and must be saved for each perpendicular.



Insert chamfer



Edge point to be chamfered



Chamfered edge

After the selection of the function the edge to be chamfered has to be defined:

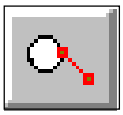
1. by clicking near the edge point
2. by a determination of the coordinates of the edge point

Both elements which form the edge are displayed as selected.

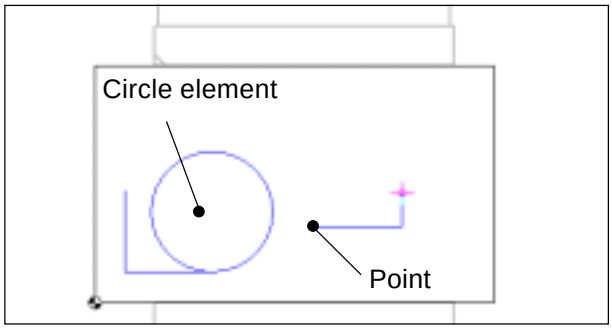
Subsequently the chamfer length has to be entered and confirmed with "ENTER".

The two new design points resulting from the chamfer edge points are saved.

The command is not selfholding and must be reactivated for each chamfer.

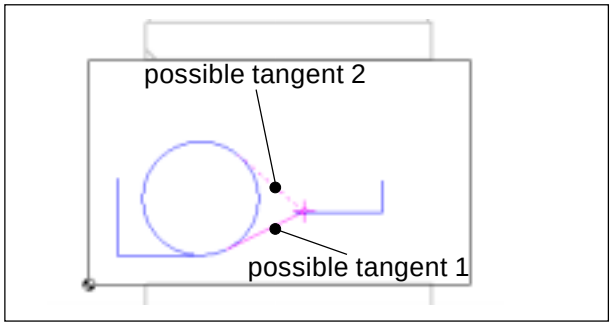


Tangent point - circle



Tangent point - circle

After entering the function you have to select the circle element and then to define the point across which the tangent shall lead.

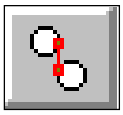


Possible tangents

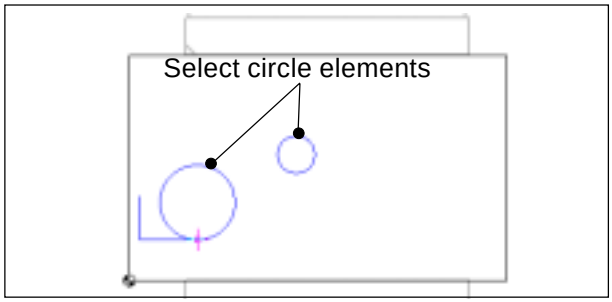
Now the two possibilites for selection are shown. Select with "space" and confirm with "ENTER".

Both end points of the tangent are saved as design points.

The command is not selfholding and must be reactivated after each tangent.

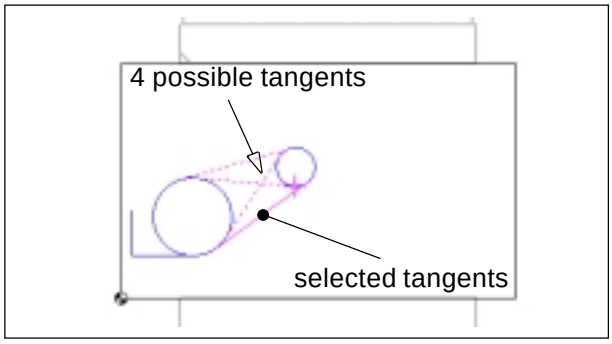


Tangent circle - circle



Tangent circle - circle

After entering the function both circle elements between which the tangents are to be drawn have to be selected. The selection "more elements" has to be carried out by means of the "SHIFT" key.

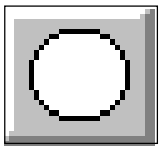


Possible tangents

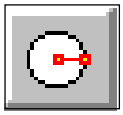
Thereupon 4 possible tangents are displayed. Now a tangent can be selected (full line) by means of "space". The selection has to be confirmed with "ENTER". The other tangents in broken lines are deleted.

Both end points of the tangent are saved as design points.

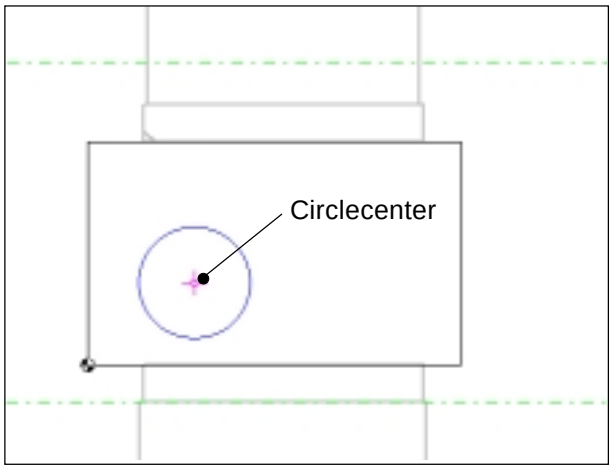
The command is not selfholding and must be reactivated after each tangent.



Circle menu



Circle with center and radius



Circle with center and radius

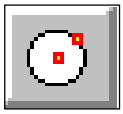
After selecting the function the center has to be defined. This can be carried out by:

1. actual cursor position and mouse click
2. with the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

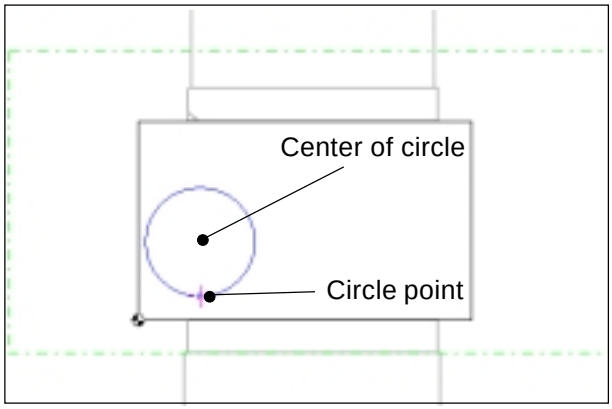
Then the radius of the desired circle has to be entered and confirmed with "ENTER".

Center and a circle point are saved as construction points.

The command is not selfholding and must be reactivated for each circle.



Circle with circle point and center



Circle with circle point and center

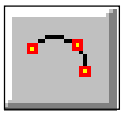
After entering the function the center has to be entered. Subsequently a point at the circumference of the desired circle.

The points can be entered by:

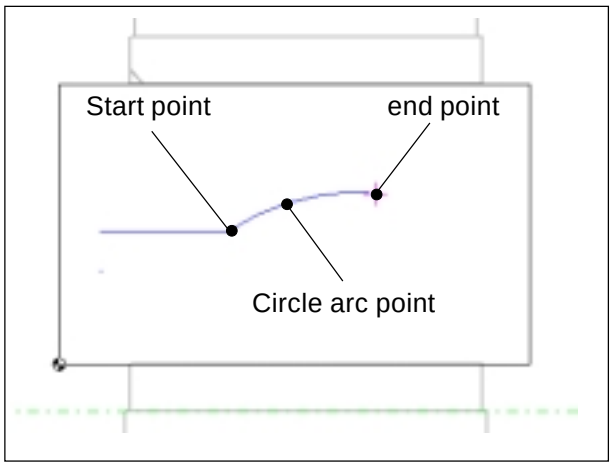
1. actual cursor position and mouse click
2. with the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

The center of the circle and the circumferential point are saved as design points.

The command is not selfholding and must be reactivated for each circle.



Circular arc with start, end and circle point



Circular arc with start, end and circle point

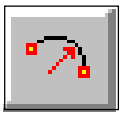
After selecting the function the start point of the circle arc has to be defined, then the end point and finally a point on the circular arc.

The points can be entered by:

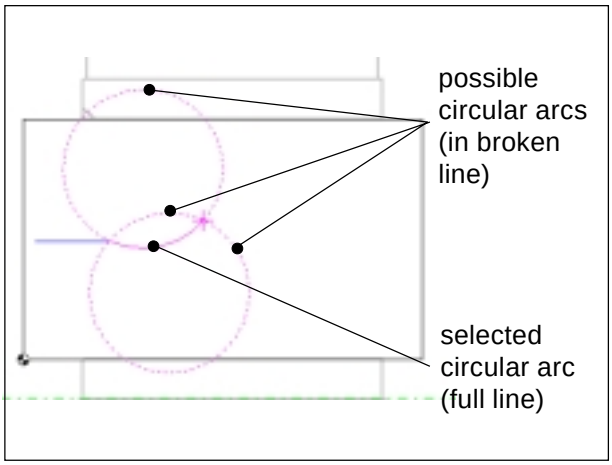
1. actual cursor position and mouse click
2. the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

The circular arc is drawn. Start, end and centre point are saved as design points.

The command is not selfholding and must be reactivated for each arc.



Circular arc with start, end and radius



Possible circular arcs

After selecting the function the start point of the circular arc has to be defined, subsequently the end point and finally the circle radius in mm.

The points can be entered by:

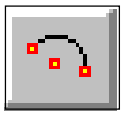
1. actual cursor position and mouse click
2. the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

This definition results in 2 possible cricles with 2 possible circular arcs. Select the desired circular arc with the "SPACE" key and confirm with "ENTER".

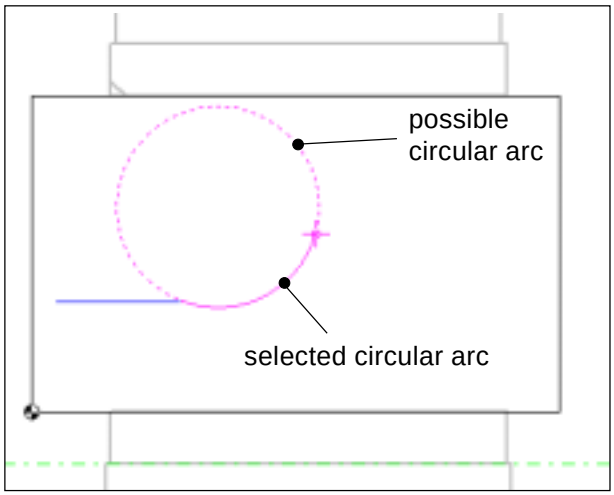
The selected (full line) circular arc is drawn.

Start, end and center point are saved as design points.

The command is not selfholding and must be reactivated for each arc.



Circular arc with start, end point and center



Circular arc with start, end point and center

After selecting the function the start point of the circular arc has to be defined, then the end point and finally the center.

The points can be entered by:

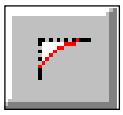
1. actual cursor position and mouse click
2. with the pick mode menu and mouse click
3. a coordinate input (INC or ABS)

This definition results in 2 possible circular arcs. Select the desired circular arc with the "SPACE" key and confirm with "ENTER".

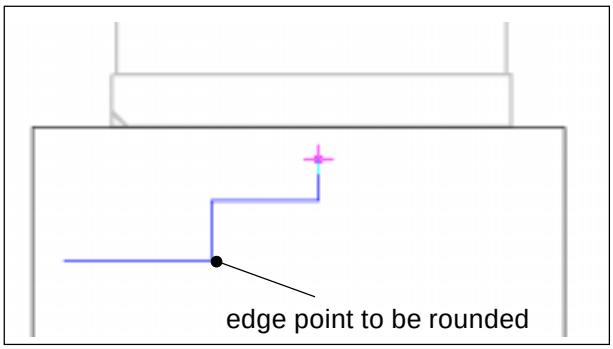
The selected (full line) circular arc is drawn.

Start, end point and center are saved as design points.

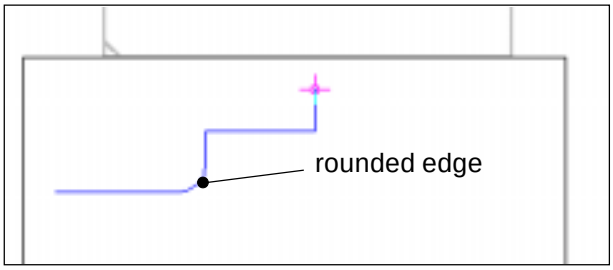
The command is not selfholding and must be reactivated for each arc.



Insert radius



Edge point to be rounded



Radius inserted

After selecting the function the edge to be rounded has to be defined:

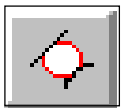
1. by clicking near the edge point
2. by a determination of the coordinates of the edge point

Both elements forming the edge are displayed as selected.

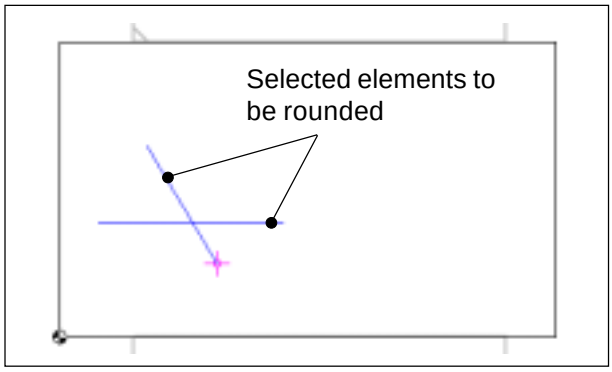
Subsequently the chamfer radius has to be entered and confirmed with "ENTER". The rounding is drawn.

The circular end points and the center are saved as design points.

The command is not selfholding and must be reactivated for each rounding.



Rounding drawing elements

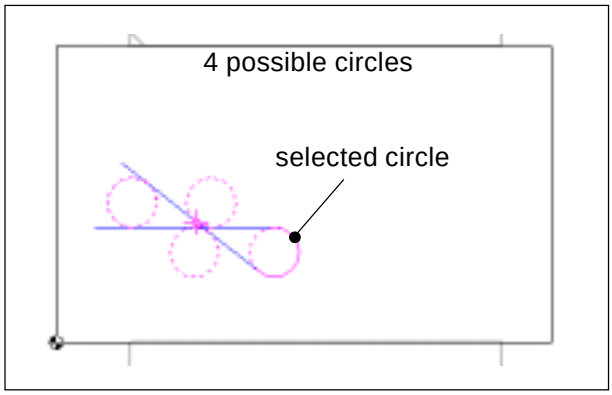


Selection of the elements to be rounded

Example of a rounding of line and line

After the input of the function both elements between which a rounding is to be inserted have to be selected ("SHIFT" key and mouse click). Then the radius of the rounding has to be entered and confirmed with "ENTER".

Now all possible rounding circles are displayed. Click on the desired circle by means of the mouse cursor. In the following all other circles are deleted.

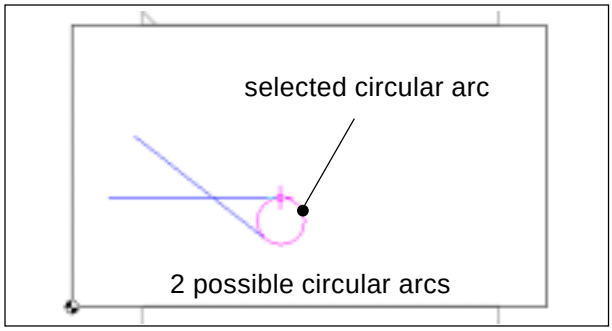


Selection of the possible circles

Select the desired circular arc by means of the "SPACE" key and confirm with "ENTER". The rounding is drawn.

The intersection points of the lines with the circular arc and the center are saved as design points.

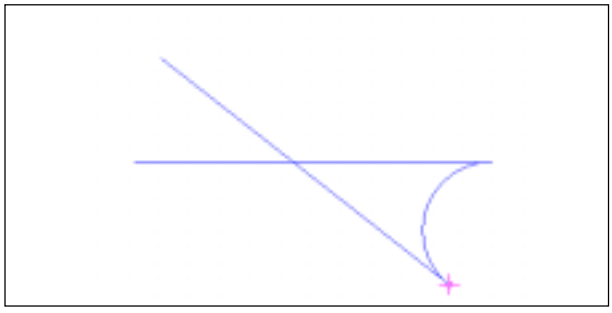
The command is not selfholding and must be reactivated for each rounding.



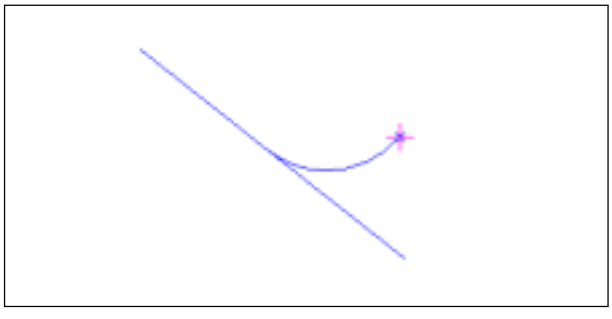
Selection of the possible circular arcs

Example of a rounding of line and point

Also the rounding of line and point is possible. The procedure is the same as described on the opposite but here you can select only 2 possible circular arcs.

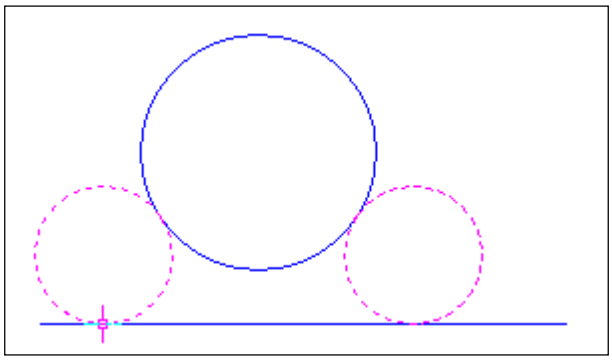


Rounded line and line

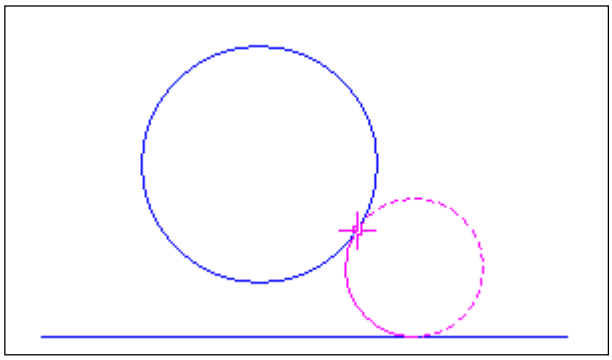


Rounded line and point

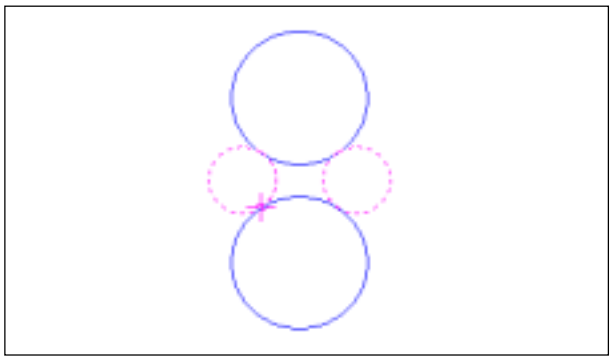
Example of a rounding of line and circle



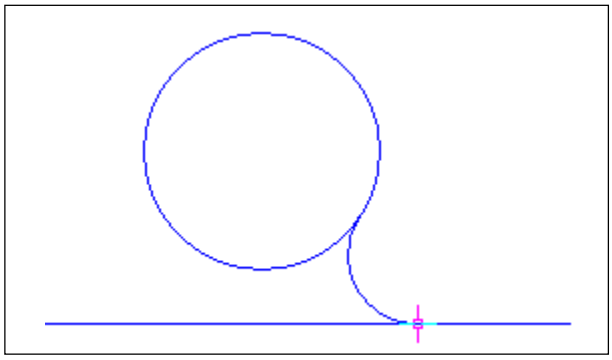
Selection of elements to be rounded



Possible circles

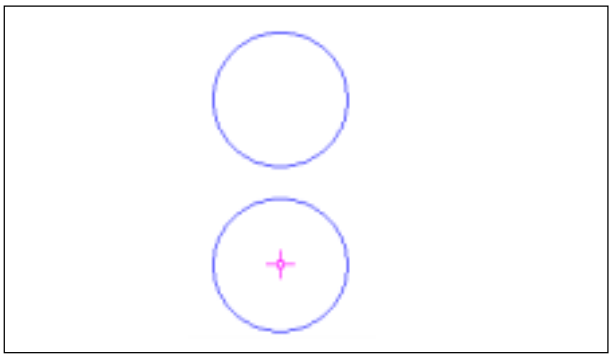


Possible circular arcs

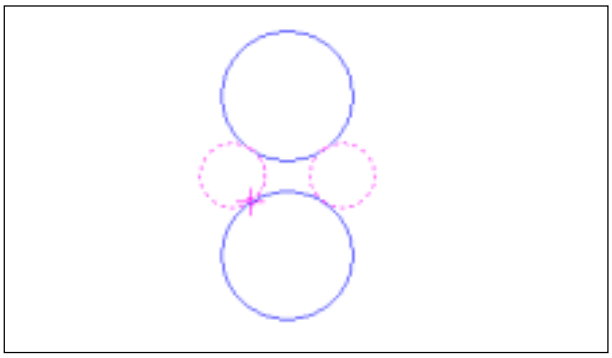


Rounded line and circle

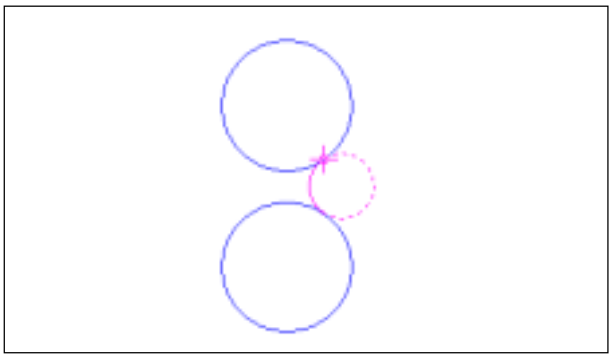
Example of a rounding of circle and circle



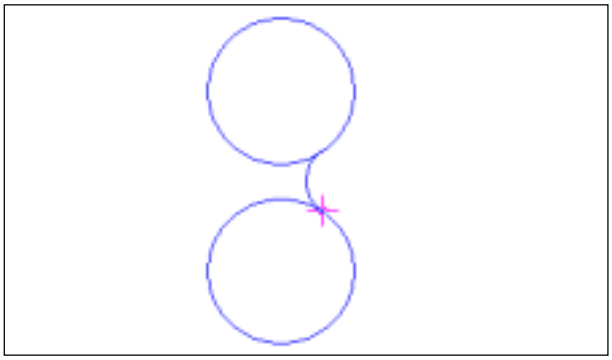
Selection of the elements to be rounded



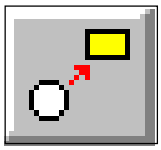
Possible circles



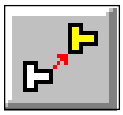
Possible circular arcs



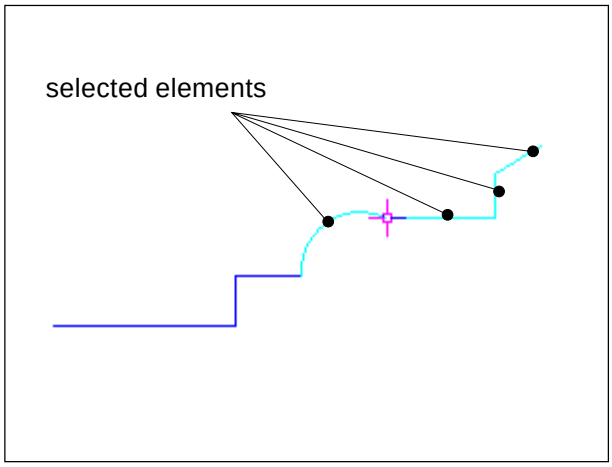
Rounded circle and circle



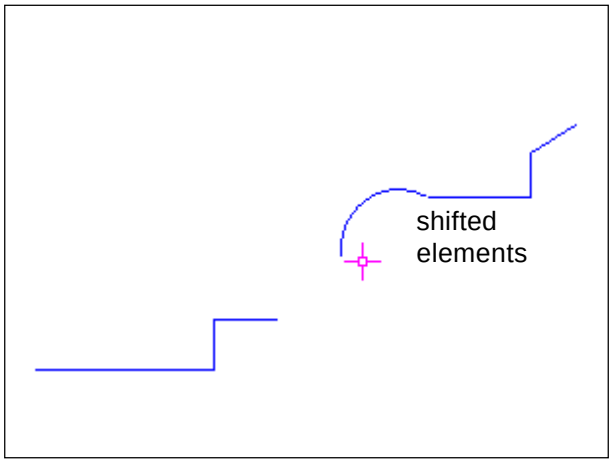
Modifying menu



Shift drawing elements incrementally



Selection of elements



Shifted elements

After entering the function the elements to be shifted have to be defined.

1. Selection of an element by mouse click
2. Selection of more elements via key "SHIFT" and mouse click.
3. Selection by means of command symbol "select drawing elements".

Selected elements change their color.

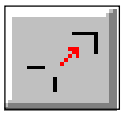
The selection must be terminated with the hook ("confirm with contour determination").

Now WinCAM expects an input of the incremental shift in X and Y and/or X and Z. After the confirmation with "ENTER" the elements are shifted.

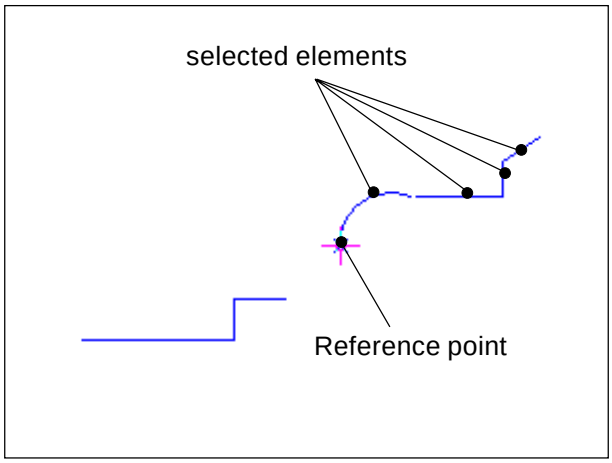
The incremental shift refers to the position of the selected elements.

The design position points are also shifted.

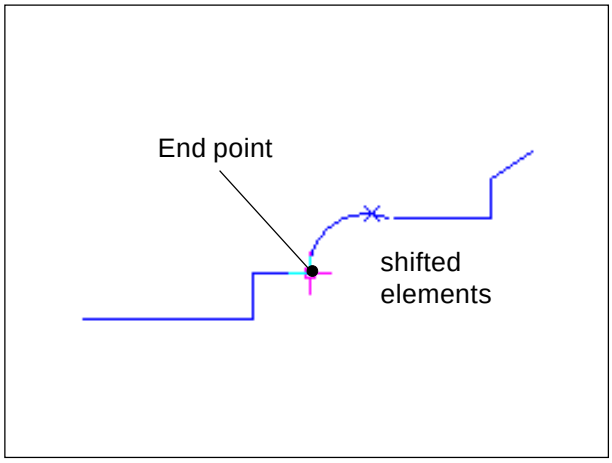
The command is not selfholding and must be reactivated for each shift.



Shift drawing elements absolutely by means of reference point



Selection of elements



Shifted elements

After entering the function the elements to be shifted have to be defined.

1. Selection of an element by means of mouse click
2. Selection of more elements via key "SHIFT" and mouse click.
3. Selection of the command symbol "select drawing elements".

Selected elements change their color.

The selection must be terminated with the hook ("confirm with contour determination").

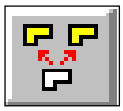
Now WinCAM expects the definition of a reference point and an end point to which the offset is to be related.

The definition of reference point and end point can be influenced by the shift key "ABS" and "INC". With "INC" the reference point definition refers to the actual point at present.

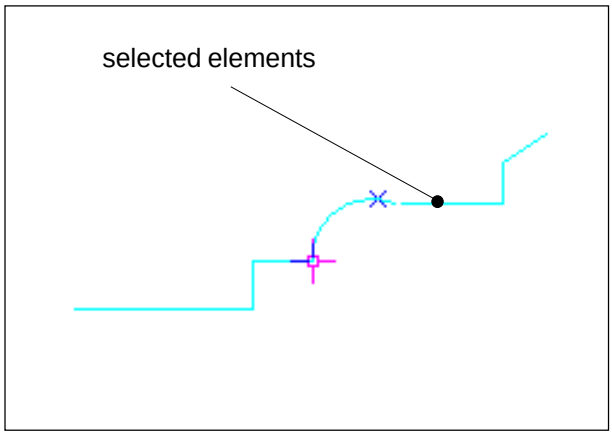
The design points are also shifted.

The command is not selfholding and must be reactivated for each shift.

With each command e.g. contour drawings can be connected.



Copy drawing elements incrementally



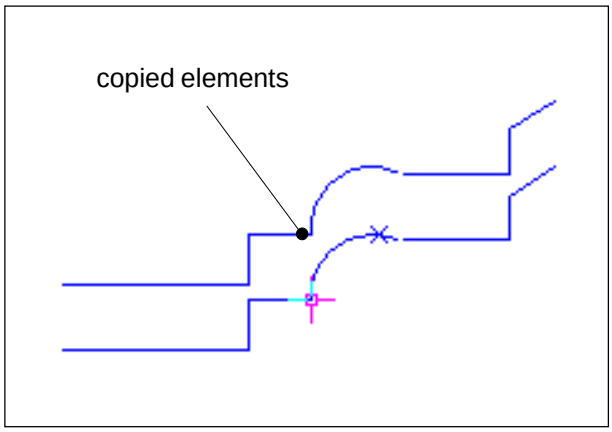
Selection of elements

After the input of the function the elements to be copied have to be defined.

1. Selection of an element by mouse click.
2. Selection of various elements via key "SHIFT" and mouse click.
3. Selection by means of command symbol "select drawing elements".

Selected elements change their colour.

The selection must be terminated with the hook ("confirm with contour determination").



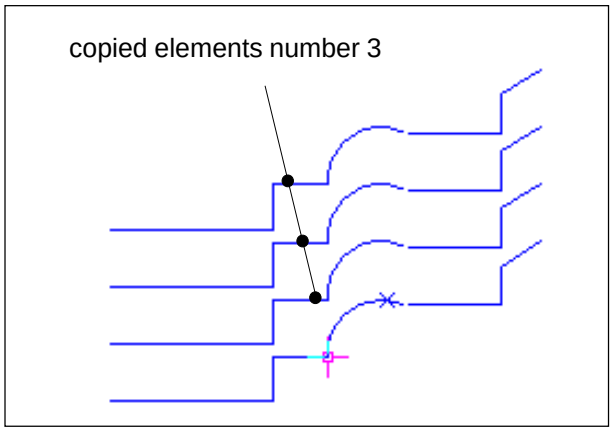
Copied elements (number 1)

Now WinCAM expects an input of the incremental shift of the copied elements and the number of copies.

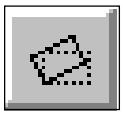
The incremental shift refers to the position of the selected elements. In case of various copies always to the position of the previous copy concerned.

The design points are also copied.

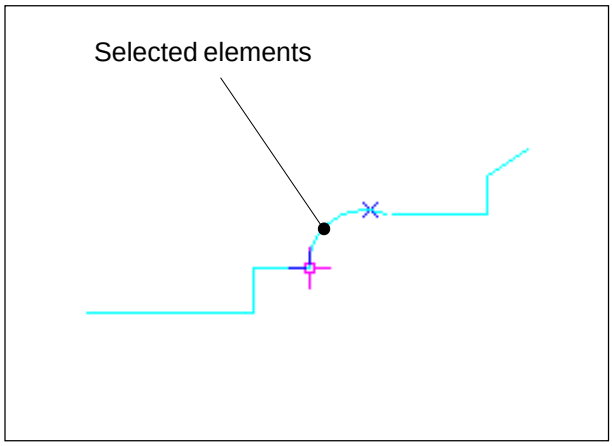
The command is not selfholding and must be reactivated for each copy procedure.



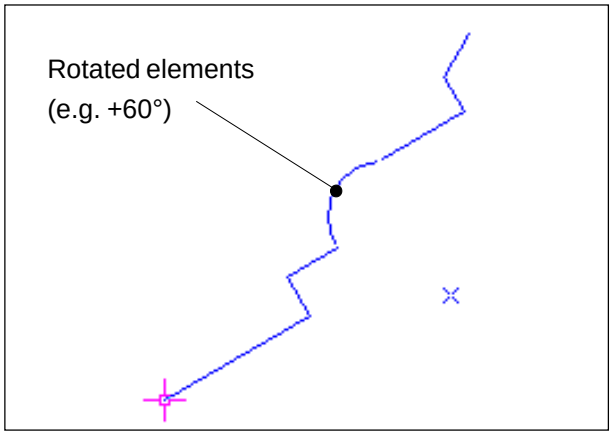
Copied elements (number 3)



Rotate drawing elements



Selection of elements



Rotate drawing elements

After the input of the function the elements to be rotated have to be defined.

1. Selection of an element by mouse click.
2. Selection of various elements via key "SHIFT" and mouse click.
3. Selection by means of command symbol "select drawing elements".

Selected elements change their colour.

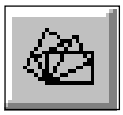
The selection must be terminated with the hook ("confirm with contour determination").

Now WinCAM expects the coordinates of the rotation point and the rotation angle.

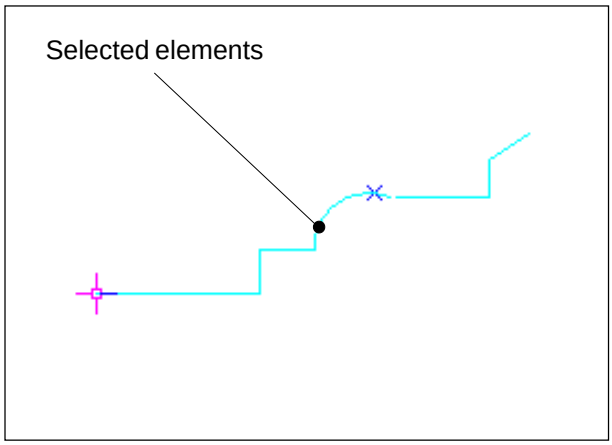
The rotation point is the point around which the elements are rotated. The rotation angle can be entered with positive or negative sign.

The design points are also rotated.

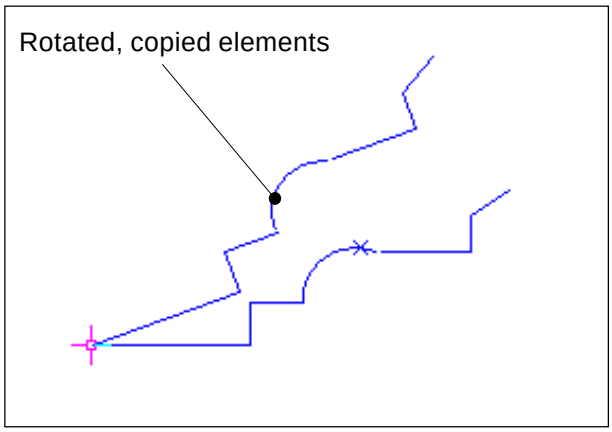
The command is not selfholding and must be reactivated for each rotation procedure.



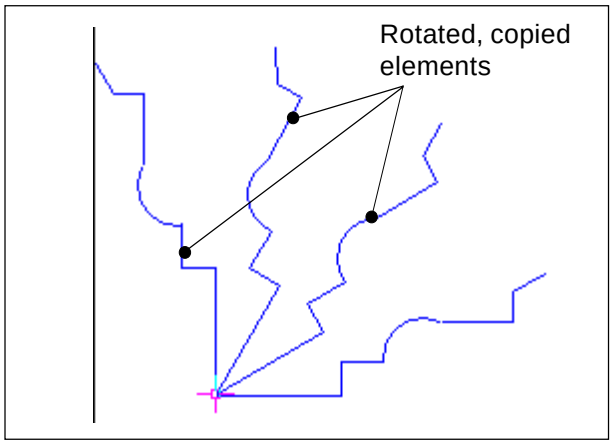
Rotate and copy drawing elements



Selection of elements



Rotated, copied elements (e.g. 30°, number 1)



Rotated, copied elem. (e.g. each 30°, number 3)

After the input of the function the elements to be rotated have to be defined.

1. Selection of an element by mouse click.
2. Selection of various elements via key "SHIFT" and mouse click.
3. Selection by means of command symbol "select drawing elements".

Selected elements change their colour.

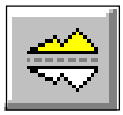
The selection must be terminated with the hook ("confirm with contour determination").

Now WinCAM expects the coordinates of the rotation point, the rotation angle and the number of copies.

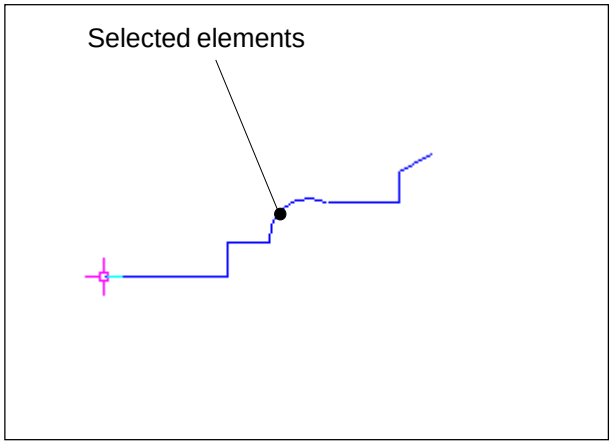
The rotation point is the point around which the elements are rotated. The rotation angle can be entered with positive or negative sign. In case of various copies the angle always refers to the position of the previous copy concerned.

The design points are also rotated.

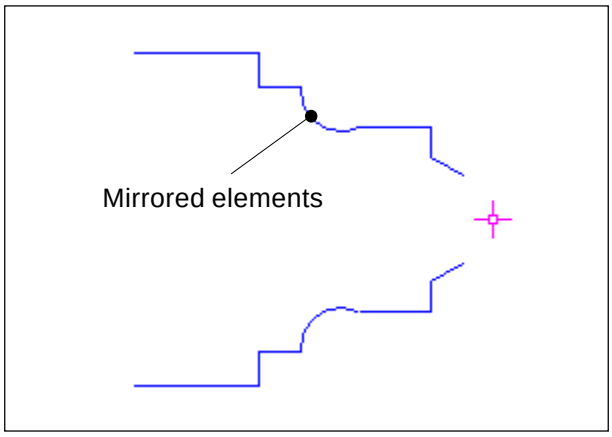
The command is not selfholding and must be reactivated for each rotation procedure.



Mirror drawing elements



Selection of elements



Mirrored elements

After the input of the function the elements to be mirrored have to be defined.

1. Selection of an element by mouse click.
2. Selection of various elements via key "SHIFT" and mouse click.
3. Selection by means of command symbol "select drawing elements".

Selected elements change their colour.

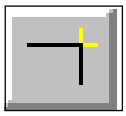
The selection must be terminated with the hook ("confirm with contour determination").

Now WinCAM expects the coordinates of the mirror axis.

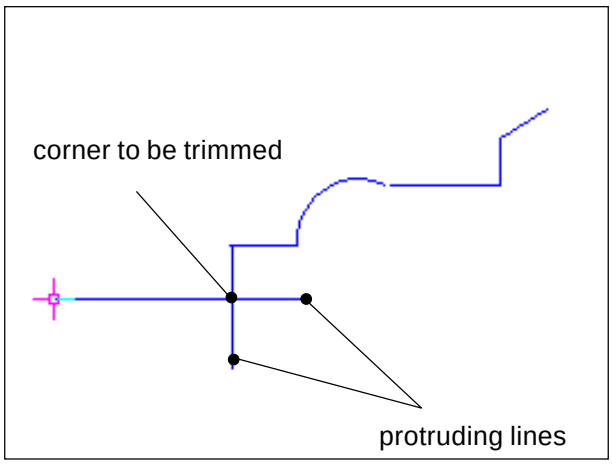
Mirroring is defined by means of the input of the start and end point of the mirror axis. By means of this input you can determine any mirror axes.

The design points are also mirrored.

The command is not selfholding and must be reactivated for each mirroring procedure.



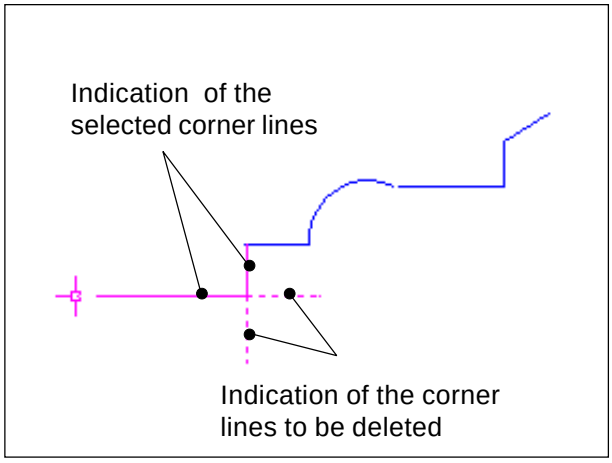
Trim corner



Selection of the corner to be trimmed

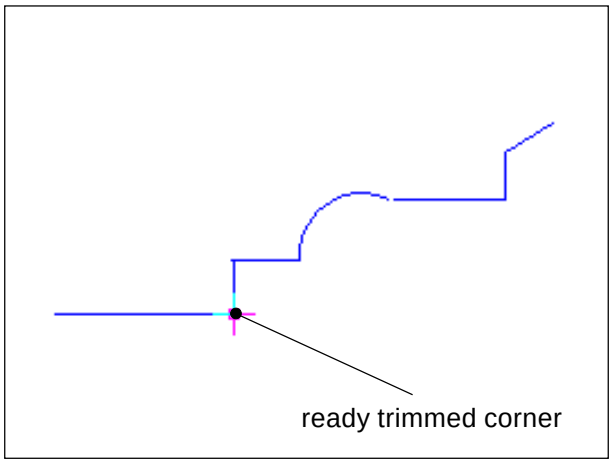
After selecting the function WinCAM expects the input of the corner to be trimmed:

1. by clicking on the corner by means of the mouse
2. by input of the coordinates of the corner point.



Selection of the lines to be trimmed

WinCAM shows all possible trims. The elements to be trimmed are drawn in broken lines. Select the desired result by means of the "SPACE" key and confirm with "ENTER".

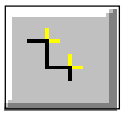


Trimmed corner

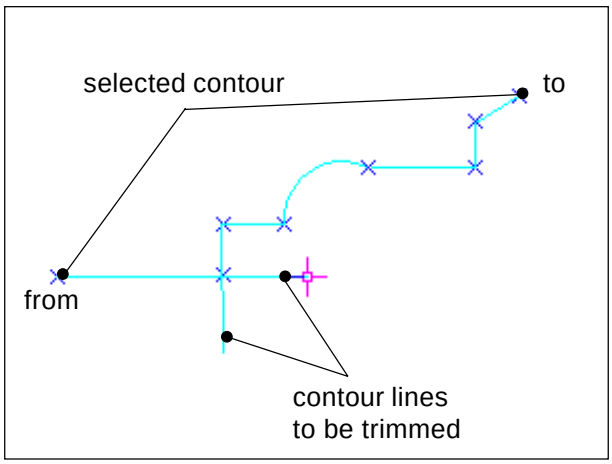
Please mind that it is only possible to trim continuous lines in the corner. If a line is divided in the corner you can delete the redundant line by means of the delete function.

In the trimmed corner an intersection point results which is saved as design point.

The command is not selfholding and must be reactivated for each trim.



Trim contour



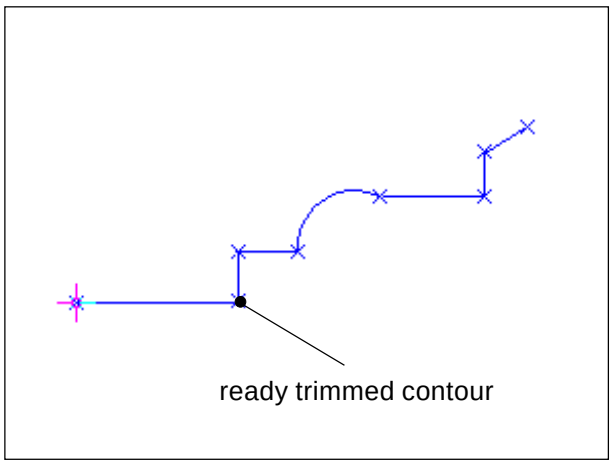
Selection of the contours

After the selection of the function WinCAM expects the input of the contour to be trimmed. Select the individual elements of the contour in the right order so that they form continuous lines (straight line/ circle) (i.e. there must always be an interface from one to the next element).

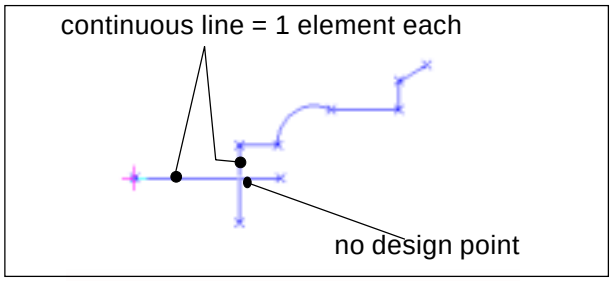
Select more elements by pressing the "SHIFT" key and mouse click. Terminate the selection with the actual green hook "confirm with contour determination".

One design point results respectively in the new corners which is saved.

The command is not selfholding and must be reactivated for each contour.



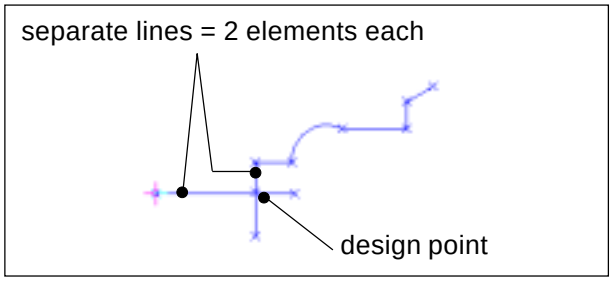
Trimmed contour



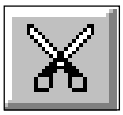
Continuous lines

Please mind that it is only possible to trim continuous lines. If a line is divided in the corner the redundant line can be deleted by means of the delete function.

It might also happen that several overlapping lines falsify the result. Always construct a trim contour (clear connected elements) in the CAD part. You will save a great deal of corrections of the drawing since these errors are noticed in the CAM part of the program.

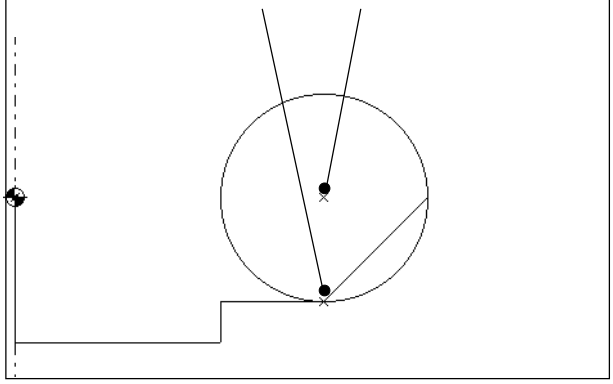


Separate lines



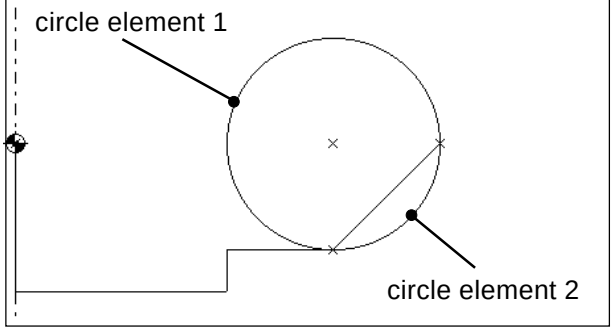
Cut elements

The selected circle has only 2 design points (centre and cercle). It consists of only one element.



Selection of the element to be cut

After the division the circle consists of 2 elements and 2 circumferential points. Each partial circle can be selected and deleted individually.



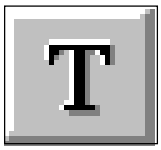
Divided element

After the selection of the function the element to be cut has to be selected. Then the cutting point (intersecting point with another element) has to be defined by means of the mouse (mouse click near the desired cutting point).

If you work on contours switch on the design points for the respective elements. In case of an unclear crossing of lines deleting a line (and subsequent "undo") can clarify the matter.

Since it is only possible to cut at intersecting points a new design point is not saved.

The command is not selfholding and must be reactivated for each division.



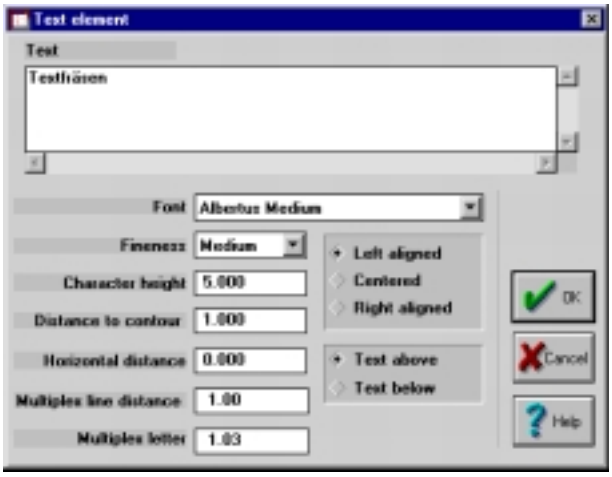
Text milling

In the following lines the procedure of text milling is described exactly.
For text milling a quick-run spindle is recommendable to achieve better engraving results.

Text milling examples

The text to be processed always refers to a contour or a contour element (see accompanying examples).

After the contour end the text is automatically continued tangentially.



Text element

CAD

- Select contour element key and draw contour.
- Confirm key T .
- Mark contour element(s) by mouse click.
- Confirm with key ✓ .
- Text field "text element" is opened.
- Text element input:

  - Text: Texts with various lines can be created. The line make-up is carried out in the input box by means of the keys STRG + ENTER

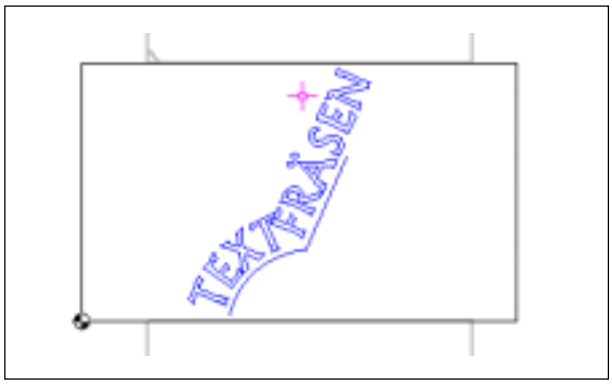
  - Font: All TT-fonts can be used except for fonts with symbolic character settings. New fonts can be installed in the Windows system control.

  - Fineness: To achieve better results, with larger fonts you have to work with higher fineness.

  - Character height: height of the capitals in mm.

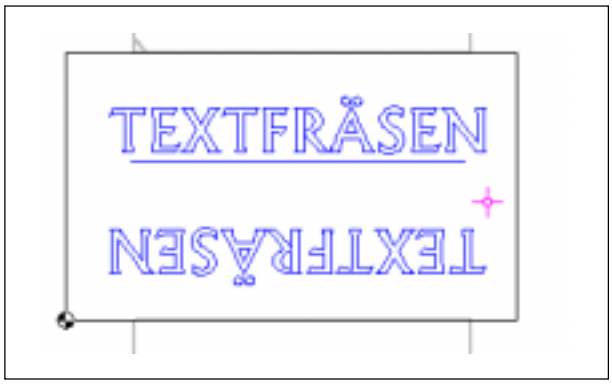
Note:
Due to the conversion procedure a time delay may occur during the text selection when initially choosing a font.

Note:
During drawing export (dxf) the text is represented in a number of lines and thus, can no longer be conceived and processed as text.



Overlapping of letters

Position of letters circle above/below



Text mirroring

-Distance to contour: distance of the lettering to the contour.

- Horizontal shifting: horizontal shifting relating to the contour.

- Line distance: multiple line distance.

- Character distance: Overlapping of letters can be avoided by increasing the character distance.

- Character alignment: alignment independent of horizontal shift

-Position of letters: with line above/below, with circle external/internal

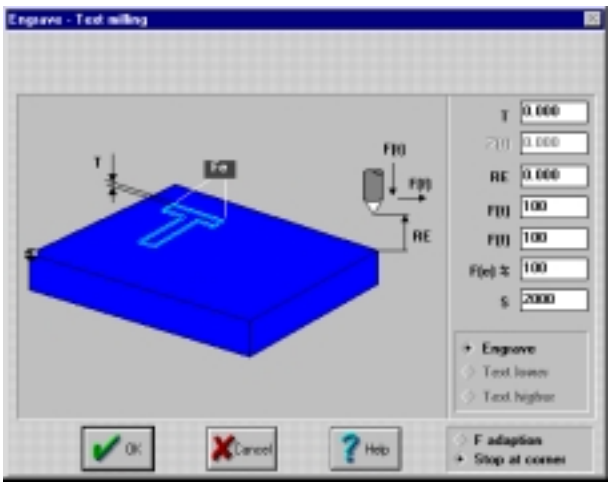
enter and confirm with  .

Mirroring texts

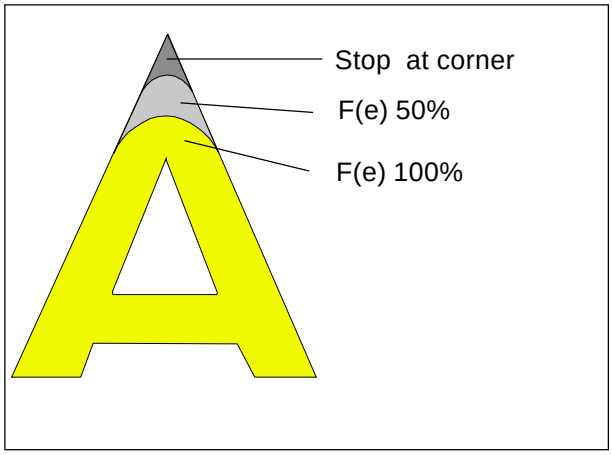
Mirroring a text is carried out in the CAD menu.

- Actuate key  and key  , mark text and confirm with  .
- Indicate position valutes (X,Z) and confirm with ENTER.
- The text is mirrored to be read correctly.

Note:
During text modification in the "text element", mirrored and turned etc. texts are corrected automatically.



Engrave - text milling



Corner feed description

CAM

- Enter all processing and tool parameters.
- Press key T .
- Mark text and confirm with ✓ .
- Text field "Engrave - text milling" is opened.
- Engrave - text milling

  - Grooving depth T: enter negative value.

  - Retrieve level RE: lifting dimension of the tool during transition to next contour.

  - Delivery advance F(t): advance in depth (delivery).

  - Engrave feed F(f): feed in surface.

  - Variable corner feed reduction F(e):
    Percentage on the automatically calculated adaption value. Selecting the key adaption serves for the reduction of the milling feed in corners to achieve better engraving results. The feed is only reduced in the corners, the programmed value is maintained on the straight lines (saving time).

  - Speed s

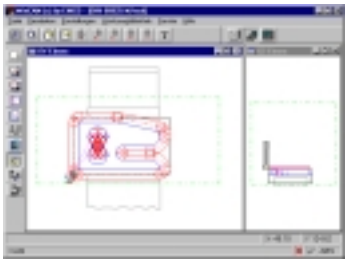
  - Adaption/ stop at corner
    Adaption: The adaption value is a value calculated automatically to reduce the milling feed in corners.

    Stop at corner: Tool feed stops. The tool may continue running and thus, a contour violation may occur.

Chapter H CAM Bases

WinCAM data transmission

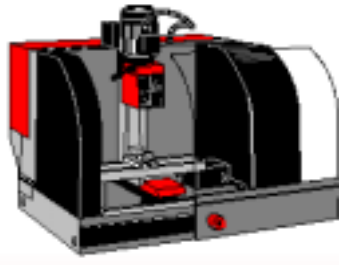
CAD drawing



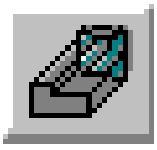
CAM machining

3D-simulation

NC program



NC machining



Basic knowledge

In CAM mode you generate interactively an NC program which is necessary to work on a workpiece contour with a CNC machine.

Data transmission
Win CAM is a complete product from CAD via CAM to NC. Therefore, in a working session it is possible to change between the modes CAD, CAM and NC at any time. Sometimes this is also necessary (e.g. to define provisionally a limitation line for a cycle outside of the workpiece).

Editor
The NC program generated with WinCAM is immediately deposited in the editor and can be studied there. Also a modification of the NC program in the editor is possible at any time. After leaving the editor the CAM screen is automatically updated.
The editor is the instrument to cancel machinings in CAM mode.

Working-off the NC program
The generated NC program can be worked off subsequently in NC mode on a connected machine.

Note:
In chapter D of this manual you can find an example to copy.

Shift CAD - CAM

By clicking the command symbol CAM you can shift at any time into CAM mode.

In CAM mode other command symbols are shown which are described more precisely in the chapter CAM commands. Also observe the changes due to the CAM mode described in the chapter menu lines.

CAM Initialization

Before you are able to supply an NC program with WinCAM previously you must supply WinCAM with information. Therefore first you have to draw the workpiece contour in CAD mode. In CAM mode you also have to enter some data for the basic setting (e.g. CAM reference point, tool definition). Only then you can call up the machining commands without error message.

1. A drawn contour must exist. For this reason draw it first in CAD mode or load a drawing.

2. Some initializations can be selected according to your requirements in the menu "setup":
   - Display - Activate line graphics
     - Display slot
     - Delay per element
   - Line - CAM simulation rapid
     - CAM simulation work
   - Automatic contour recognition
   - Target control
   - Target machine

3. The adequate tools must exist in the tool library and be loaded in the toolholder.

4. Define the tool with which you want to machine by means of the command symbol "tool change". Also the respective speed and feed can be entered in the displayed NC-parameter window.

5. Switch on the main spindle of the machine by means of the command symbols "periphery" and "main spindle on/right" (or on/left).

6. Traverse the selected tool in rapid motion to the start point of machining by means of the command symbol "movement in rapid motion in plane" and "movement in rapid motion in feed direction".

7. Now you can call up the first machining cycle.

Notes:

- If an existing CAM-program is to be deleted prior to starting the new one, select the menu item "CAM program new" in the menu line "edit".
- You can set a new CAM reference point at any time by means of the menu item "set CAM reference point" in the menu line "edit".
- Speed and feed can also be entered in the NC-parameter window which is switched with the command symbol "NC parameter on/off".

CAM machining

This consists basically of:

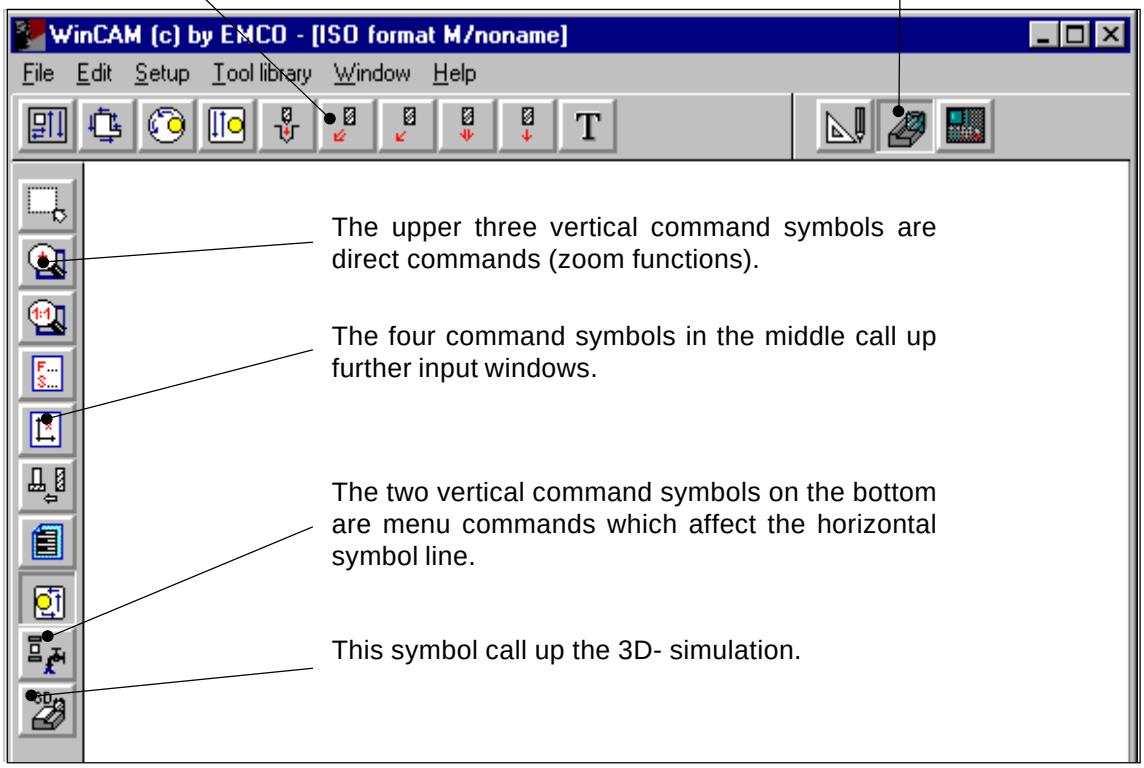
1. Selection of the machining cycle or the traverse movement.
2. Definition of the contour to be machined.

3. Automatic proceeding by WinCAM.

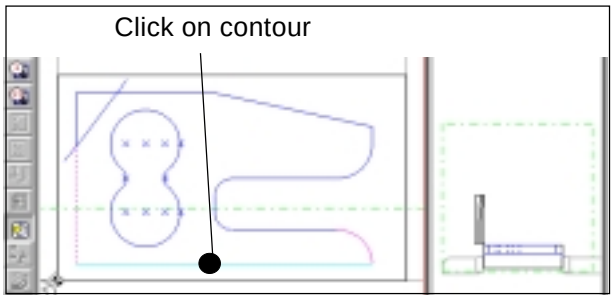
Command symbols

In the horizontal symbol line those command symbols appear which can be called up by means of the two vertical menu commands.

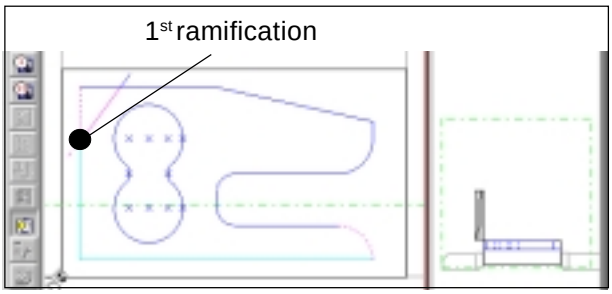
By means of the 3 shift symbols you can shift from CAD into CAM and/or into the NC-mode. Due to the shift all other command symbols are shifted. Here you switch on CAM.



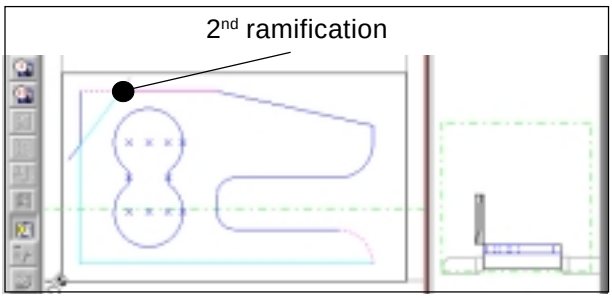
Command symbols



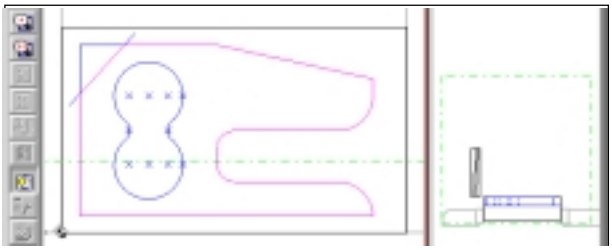
Define contour - selection of the contour



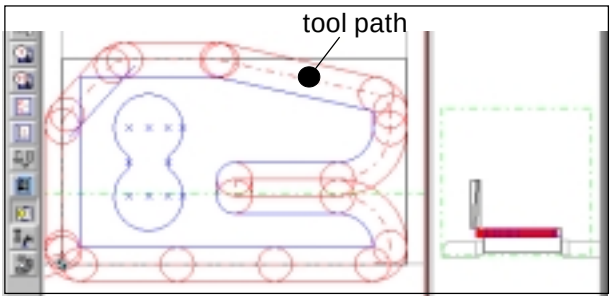
Define contour - display 1st crossing



Define contour - display 2nd crossing



Define contour - contour determined



Define contour - contour after machining

Defining the contour

After the selection of a machining command WinCAM emits the message "search contour" in the status line. Now click on or near an contour element to be machined.

Automatic contour recognition

This is switched on with the menu "setup - automatic contour recognition" (status with hook).

- If it is switched off the contour has to be defined by selecting all elements and terminated with the green hook (confirm with contour determination).
- With contour recognition switched on (automatic) WinCAM tries to find the whole contour. Here are two possibilities which are explained below. In this respect look at the opposite fig. example.

1. The contour is clear

There are no ramifications in the contour.

- WinCAM indicates the contour as selected (different colour) and expects the
- contour confirmation with the actual green hook.

2. The contour is not clear

Due to ramifications of the contour there are several possible machining paths.

- In this case WinCAM indicates the possible paths in broken lines. By means of the
- "space" key switch between the possibilities. The actual variant at the time is drawn in full line.
- The selected contour variant is confirmed with "Enter".
- In case of various ramifications several of such selection procedures will be necessary. When the contour is ready determined confirm with the
- green hook (contour confirmation)

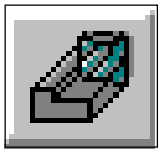
Skip contour elements

E. g. : An internal edge is too narrow for the roughing tool and will be skipped for roughing.

- Switch off "setup - automatic contour recognition".
- Every element must be confirmed with ENTER. With ramifications you can select as described above.
- When the query is on the element to be skipped, press the right mouse button and select "Search point".
- Click (left mouse button) on the next contour point which should be machined.
- This point is a ramification and a query follows.

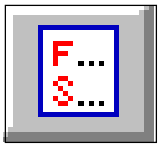
Note:
Contour ramifications are only possible if a common construction point connects two elements which cross or touch each other. Two continuous lines crossing each other do not effect a contour ramification.
The hook of the contour confirmation becomes green only when the contour has been sufficiently defined for the selected cycle.

Chapter I CAM commands



CAM mode
The CAM command symbols are activated by clicking on the shift symbol "CAM". The CAM mode remains active until it is deactivated again by means of CAD or NC.

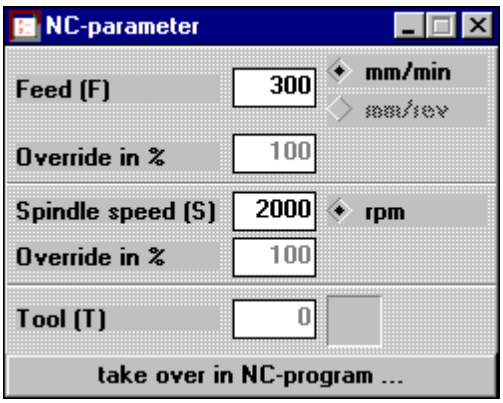
The 3 direct commands have already been explained earlier.



NC parameter on/off

By selecting the function the WinCAM window "NC-parameter" is opened.
Lead the mouse cursor from one input field into the other (also possible with the "Tab" key) and enter successively:

Feed (F) in mm/min
Spindle speed (S) in rpm



WinCAM window "NC-parameters"

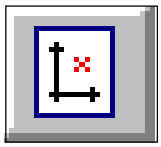
If a tool (T) has already been selected it is displayed in screened mode. In addition to the indication of the actual tool number also the pictograph of this tool is displayed.

The entered values are saved in the NC-program with a click on the line "take over in NC-program". If this line is displayed in screened mode the line is without function.

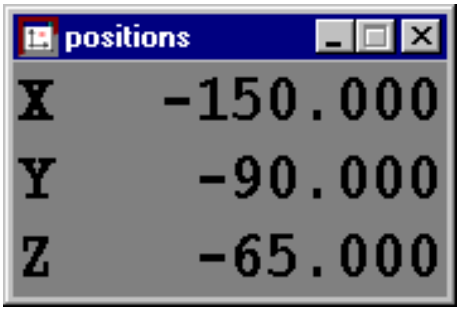
Whenever you like you can offset the window to an undisturbing screen position.

The window is closed again by another click on the command symbol "NC-parameter on/off".

The displays override in % are of no significance in CAM mode. They indicate the changed value in NC-mode. Please consult chapter K.



Position display on/off



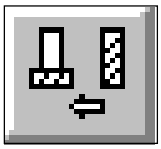
WinCAM window "positions"

By selecting the function the WinCAM window "positions" is opened.

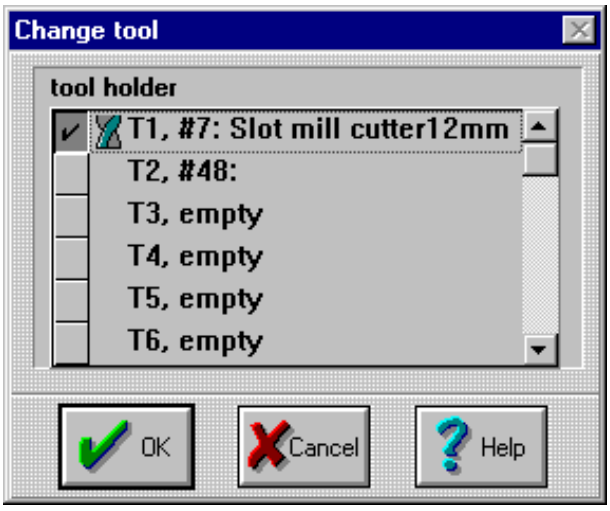
In this window the actual CAM coordinates X,Y,Z (= tool position) are displayed in relation to the active CAM-reference point.

You can move the window to an undisturbing screen position and also change its size at any time.

By another click on the command symbol "Position display on/off" the window is closed again.



Change tool



Change tool

By selecting the function the WinCAM window "change tool" is opened.

The presently actual tool is marked with a hook.

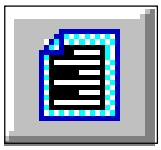
Here you can select one of the tools being in the toolholder.
Click into the field near the respective tool by means of the mouse and confirm with O.K.

The selected tool is entered into the NC program and the window is automatically closed.

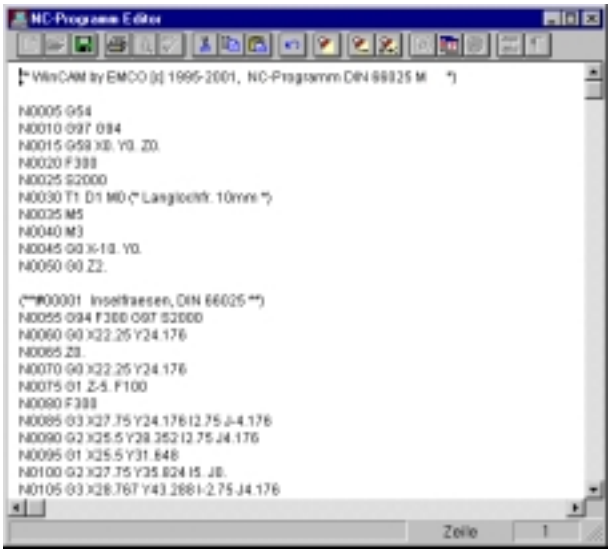
You can move the window to an undisturbing screen position at any time. Together with this window also the WinCAM window "NC-parameter" is opened for information and closed again.

By means of the field "Help" you have the help functions displayed.

By a click with the mouse cursor on the field "Cancel" the window is closed again without entry into the NC-program.



Editor (change program)



NC program editor

By selecting the function the WinCAM window "NC-program-editor" is opened.

Here you can look into the NC program generated up to now and/or change it.

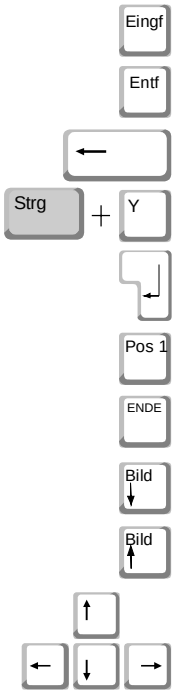
All changes in the editor have an immediate effect on CAM machining since there is only one data source. When the editor is left after a change the CAM screen is immediately updated.

By another click with the mouse cursor on the command symbol "editor" the window is closed again.

Operating the editor

Since in general the NC program is automatically generated by WinCAM the main task of the editor is the control and/or change of programs. The operation is therefore limited to the basic commands of text processing.

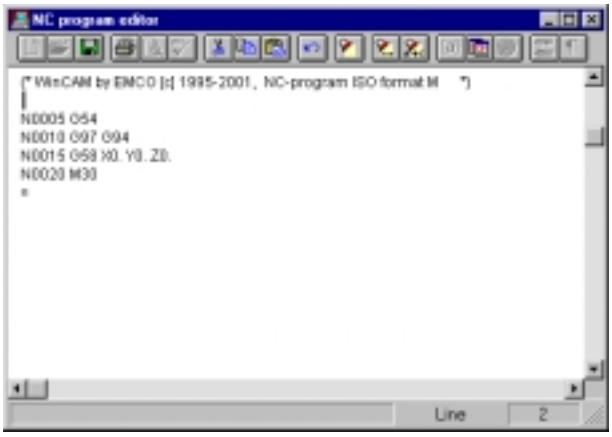
The following commands are possible:



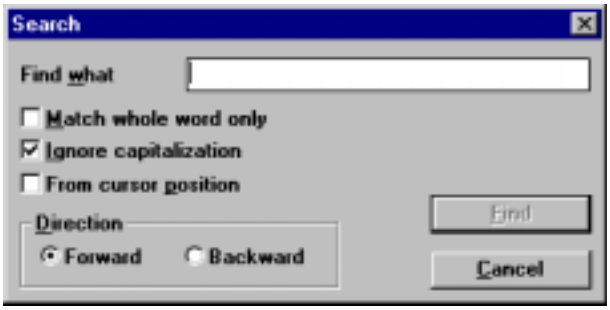
- Changing mode from inserting characters to overwriting by means of the key "Insert".
- Deleting characters via the cursor using the key "Delete".
- Deleting a character to the left of the cursor using the key "Backspace".
- Deleting a line by means of the key combination "Ctrl + Y".
- Inserting a line via the key "Enter". This is only possible in insert mode.
- Skip of the cursor to the beginning of the line by means of the key "Home".
- Skip of the cursor to the end of the line by means of the key "End".
- Turning to the next page in the editor by means of the key "Page Down".
- Turning to the preceding page in the editor using the key "Page Up".

- Traverse the cursor by means of the arrow keys.

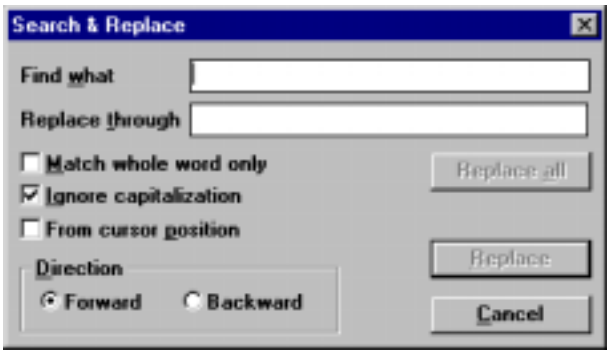
Mind the continuous function when you press the key for a prolonged period.



NC Program Editor



Input Window Search



Input Window Replace

Editor

The editor includes all functions of a simple Windows editor.
The functions are activated by clicking on the symbols.
Not active symbols are displayed in grey.
By pointing on the symbol with the mouse cursor a text window shows the meaning of the symbol.

Save

The alterations in the NC programs will be stored. Especially with longer working in the editor, periodic saving is recommended.

Print

The NC program will be printed at the standard printer (Windows setting).

Cut

The marked area will be copied to the clipboard and deleted in the NC program.

Copy

The marked area will be copied to the clipboard and stays in the NC program.

Paste

The contents of the clipboard will be copied into the program at the cursor position.

Cancel

The last operation will be cancelled.

Search

You can search for commands, characters, numbers etc..

Search again

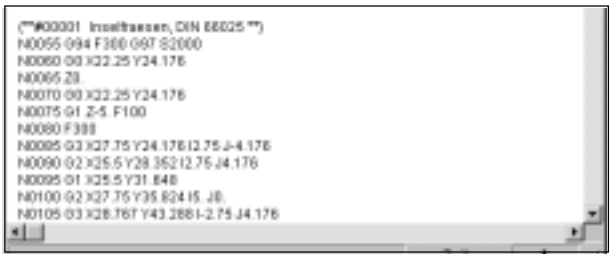
Search again for the previous searched topic.

Replace

You can search for commands, characters, numbers etc. and replace this by other signs.

Date

The actual system date will be inserted as comment in the NC program at the actual cursor position.



Editor - comments

Entries in the editor

The entries in the editor are NC programs according to DIN 66025. Here the knowledge on the syntax of such programs is assumed. Please consult your manual of the CNC machine.

Machining cylces

Whether machining cycles were resolved into individual G-commands by WinCAM or not depends on the specification in the menu "setup - target control" and on the cycle setting (cycle/DIN 66025).

Comments

Individual machining blocks are divided via comments by WinCAM. You, too, can insert such comments. Every text between the brackets () is not worked off.

Automatic entries

The following WinCAM actions write directly into the editor:

- Entries in the window "NC-parameter"
- Entries into the window "change tool"
- All executed machining commands
- All confirmed periphery commands

Significance of the editor

The NC-program in the editor is the result of the work with WinCAM. It defines the exact order of machining the workpiece and thus also the machining steps of the tools displayed in the CAM window.

The CAD part of WinCAM is not only intended for creating a workpiece drawing but for drawing contours which are machined in the CAM part. The result of it is the NC-program in the editor which is subsequently worked off on the CNC machine.

Therefore it is possible to draw auxiliary contours for the machining in CAD part, to work them off subsequently in the CAM part and finally to delete the contour again without losing the entries in the editor. Thus, the workpiece drawing always remains clear.

Changes in the editor have no effect on the CAD drawing. Since after each change the CAM screen is updated, changes in the editor have an effect on that one, though.

Target control
The NC-program of the editor is managed, saved and loaded for the control defined in the WinCAM window "select target control" (menu "setup").

Saving and loading the editor

The whole project is saved and/or loaded and thus also the NC program in the editor by means of the functions "save", "save as" und "open" in the menu "file".

A project can contain up to 3 NC-programs. For each target control set an own NC-program can be generated which is automatically deposited with an own extension of the project name during storing (also see the chapter menu lines).

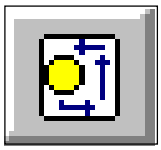
Deleting the whole editor

By means of the function "new" in the menu "file", as well as "CAM program new" in the menu "edit" after the safety inquiry the CAM screen and in addition to it also the editor, except for its basic settings, are deleted.

Direct NC Input

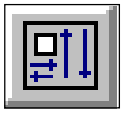
In the CAM mode NC commands can be entered directly into the command line, the editor need not to be opened.
Block numbering will be continued in the editor.
NC commands are identified with the sign "#".

Example:
G0 X10 Y10 Z2 (Approaching in 3 axes)



Machining

General
The display of the traversing paths is carried out according to the definition in the menu "Setup - line - setup line attributes" (e.g. full line for work feed, broken line for rapid motion).
In the WinCAM windows of the machining cycles there are some data already known to the control due to the contour definition (e.g. radius of the circle pocket). If you want to transfer these values you can skip these input fields.



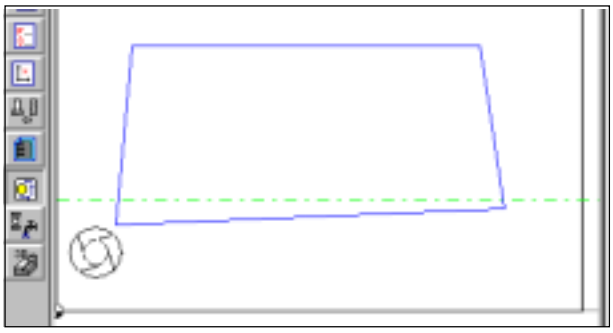
Pocket milling with islands

By selecting the function a machining cycle is started which serves for milling an area within a closed contour. In this contour there may also be an island contour which is not machined then. A furter interlinking of contours is not admissible.

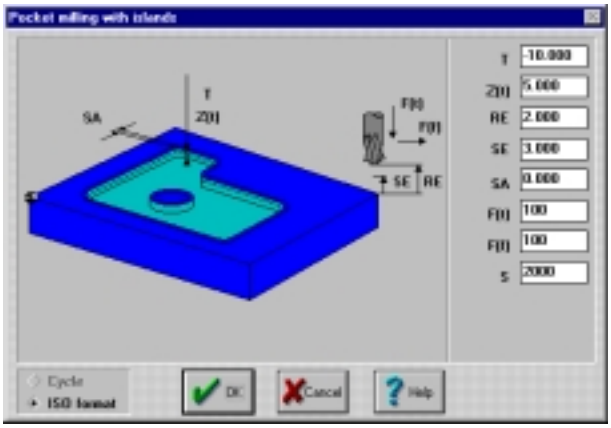
Just like with all other kinds of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

The command is divided into 3 steps:

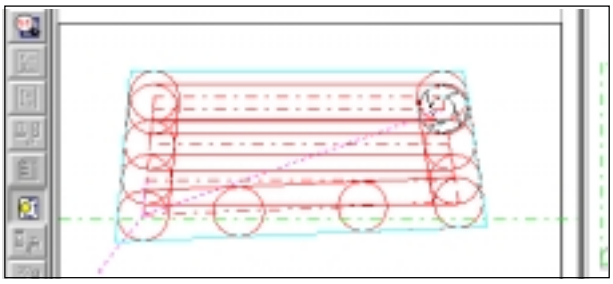
1. Define the closed contour which is to be machined. The contour definition has to be terminated by pressing the contour confirmation (green hook) twice.
2. Input of the cycle parameter in the appearing picture "pocket milling with islands".
3. The automatic machining of the area at the CAM screen and the corresponding input of the NC-program in the editor.



Define closed contour



*WinCAM window "*pocket milling with islands*"*



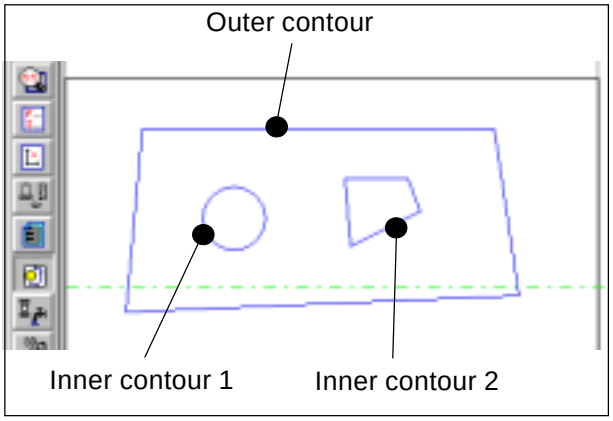
Machined area

WinCAM window "pocket milling with islands"

Enter the cycle parameters into this window and confirm the inputs with a click on the field "O.K.".
By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

T	Depth = absolute depth of the milled area in relation to the CAM reference point
Z(t)	Advance depth = advanced depth per milling procedure in relation to the start plane.
RE	Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point
SE	Start plane = absolute height of the cycle start plane in relation to the CAM reference point
SA	Smoothing overmeasure = the cycle keeps away from the contour by this measure
F(t)	Z feed (feed) = feed with which is advanced in Z direction
F(f)	XY feed = feed with which is advanced in XY-plane.
S	Spindle speed = speed of the main spindle during the cycle



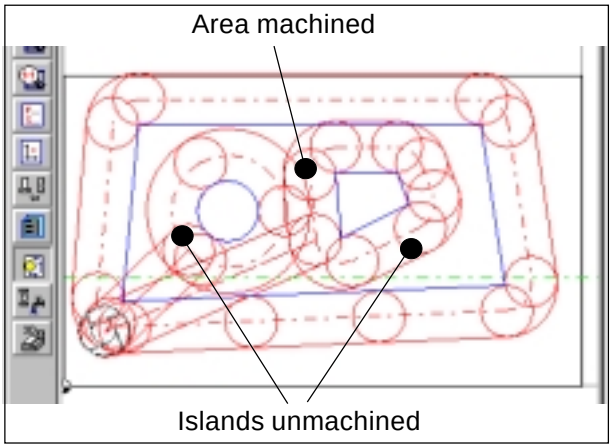
Defining island contours

Input cycle or ISO format
The status display cycle or DIN (according to the target control) indicates if the cycle is resolved into individual path commands or not.

Machining of island contours
By means of the command "pocket milling with islands" also closed contours within an area to be machined can be defined.

For contour selection proceed as follows:

1. First select the outer contour and confirm with the green hook.
2. Select all inner contours one after another and confirm each with the green hook.
3. Terminate the contour selection by pressing again the green hook.

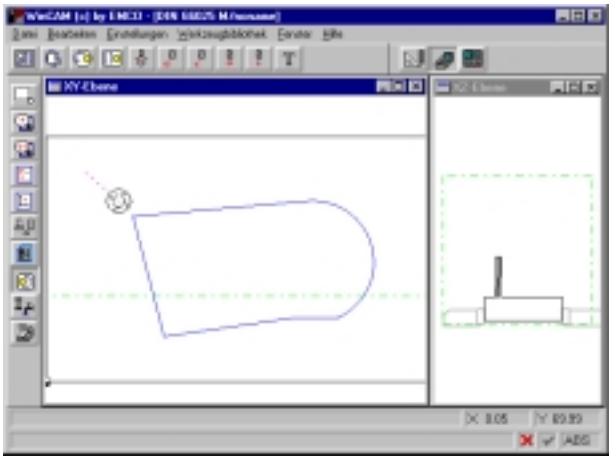


Machining with islands

Skip contour elements
E. g. : An internal edge is too narrow for the roughing tool and will be skipped for roughing.

- Switch off "setup - automatic contour recognition".
- Every element must be confirmed with ENTER. With ramifications you can select as described before.
- When the query is on the element to be skipped, press the right mouse button and select "Search point".
- Click (left mouse button) on the next contour point which should be machined.
- This point is a ramification and a query follows.

Follow contour



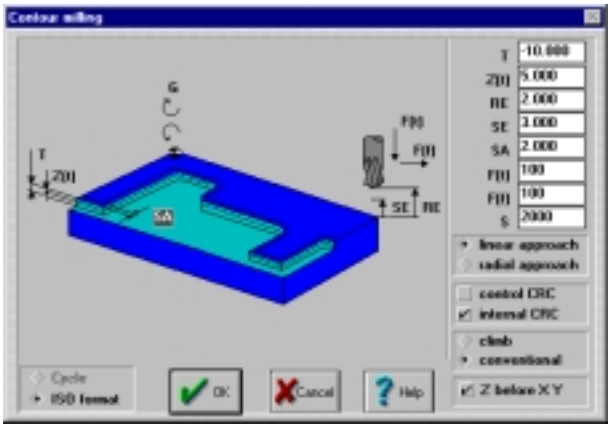
Define contour

By selecting the function a machining cycle is started which serves for milling along a contour.

Just like with every other kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

The command is divided into 3 steps:

1. Define the contour to be machined.
2. Input of the cycle parameters in the appearing picture "contour milling".
3. The automatic machining of the contour at the CAM screen and the corresponding input of the NC program in the editor.



WinCAM window "contour milling"

WinCAM window "contour milling"

Enter the cycle parameters into this window and confirm the inputs with a click on the field "O.K.". By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

T Depth = absolute depth of the milled contour in relation to the CAM reference point

Z(t) Advance depth = advanced depth per milling cycle in relation to the start plane

RE Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point

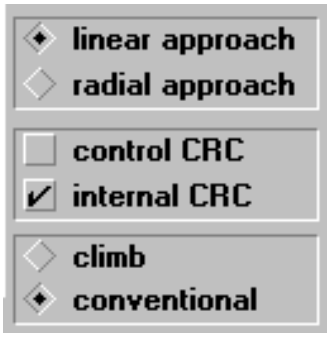
SE Start plane = absolute height of the cycle start plane in relation to the CAM reference point

SA Smoothing overmeasure = the cycle keeps away from this contour by this measure

F(t) Z feed (feed) = feed with which is advanced in Z direction.

F(f) XY feed = feed with which is advanced in XY plane.

S Spindle speed = speed of the main spindle during the cycle

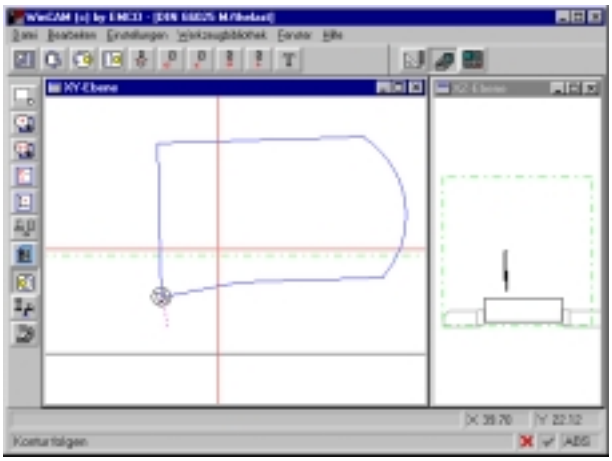


Input fields "follow contour"

Input of approach direction
Set the approach direction by a mouse click into the respective input square. The selected direction is marked by a point.

Linear approach:
The workpiece is directly approached in linear form in rapid motion and after machining of the contour again moved away in linear form.

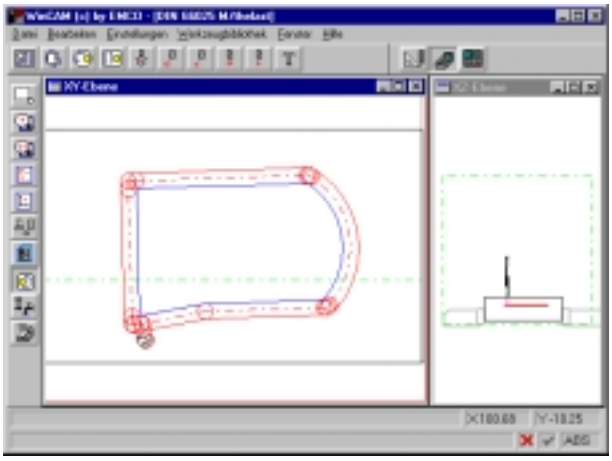
Radial approach:
The workpiece is approached by the tool in a tangential circular arc (in relation to the workpiece contour) and after machining of the contour moved away again in a tangential arc.



Machined contour - without CRC

Input cutter radius compensation (CRC)
Select the CRC by a mouse click into the respective input square. The selected CRC is marked by a hook. If no hook is visible the CRC is switched off.

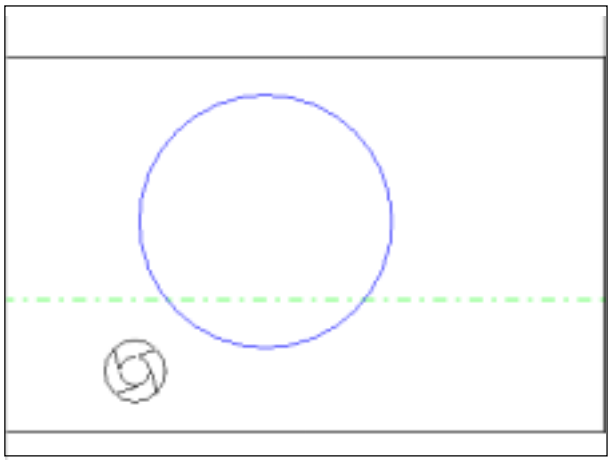
1. No CRC:
   The cycle follows with the tool center point exactly on the selected contour.
2. Control CRC:
   WinCAM generates the NC-program of the contour without CRC, however it sets the CRC codes (G41, G42, G40) in accordance with the target control. Then the CRC is calculated by the target control itself.
3. Internal CRC:
   Win CAM calculates itself the CRC. The cycle follows with the tool center point in the radius distance of the contour.



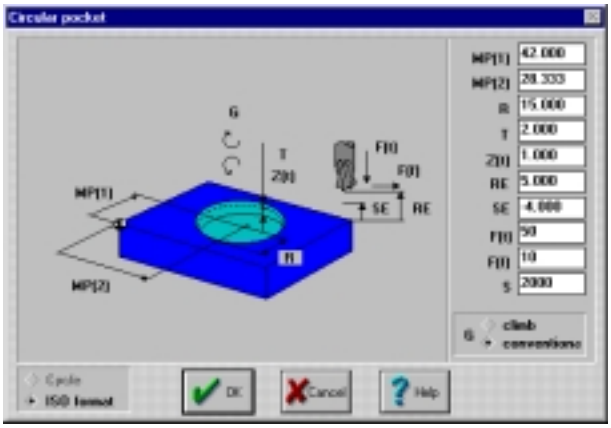
Machined contour - with CRC

Input of the milling direction
According to your decision to mill in climb or conventional way click into the respective input square.

Input cycle or ISO format
The status display cycle or DIN indicates (according to the target control) whether the cycle is resolved in individual path commands or not.

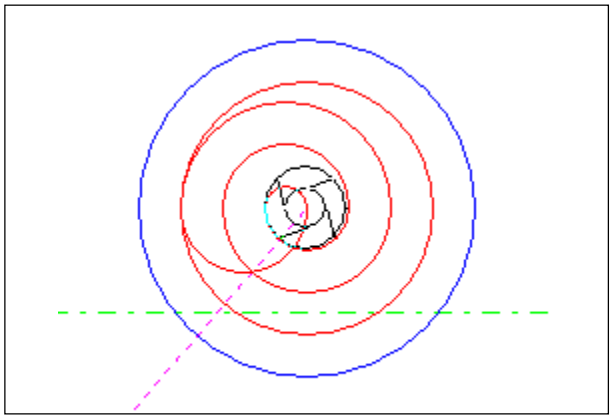


Define circle



WinCAM window "circular pocket"

Input cycle or ISO format
The status display cycle or DIN (according to target control) indicates whether the cycle is resolved in individual path commands or not.



Machined circular pocket

Circular pocket

By selecting the function a machining cycle is started which serves for milling a circular area.

Just like with every other kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

The command is divided into 3 steps:

1. Define the circle to be machined (circular element or center)
2. Input of the cycle parameters into the appearing picture "circular pocket".
3. The automatic machining of the circular area at the CAM screen and the corresponding input of the NC program in the editor.

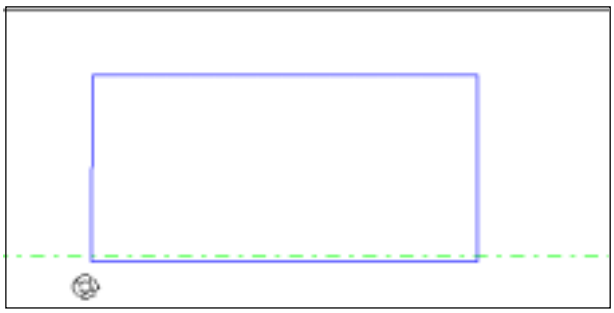
WinCAM window "circular pocket"

Enter the cycle parameters into this windows and confirm the inputs with a click on the field "O.K.".
By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

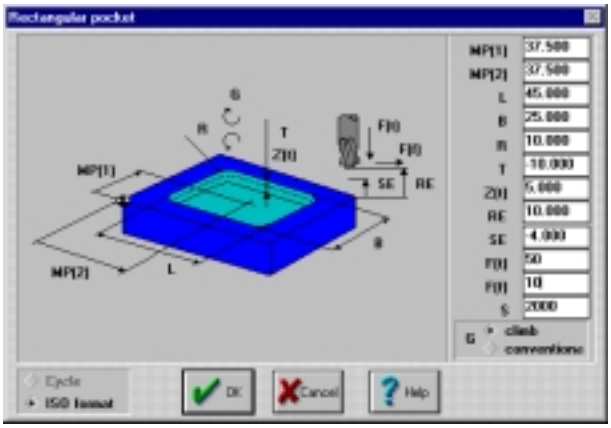
The inputs in detail:

MP(1) Center axis 1 = absolute coordinate of the center in Y-axis.
MP(2) Center axis 2 = absolute coordinate of the center in X-axis.
R Radius = radius of the circular pocket.
T Depth = absolute depth of the milled circular pocket in relation to the CAM reference point.
Z(t) Advance depth = advanced depth per milling cycle in relation to the start plane.
RE Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point
SE Start plane = absolute height of the cycle start plane in relation of the CAM reference point
F(t) Z feed = feed with which is advanced in Z direction.
F(f) XY feed = feed with which is advanced in XY plane.
S Spindle speed = speed of the main spindle during the cycle.

Input of the milling direction:
According to your decision to mill in climb or conventional way click into the respective input square.



Define rectangle

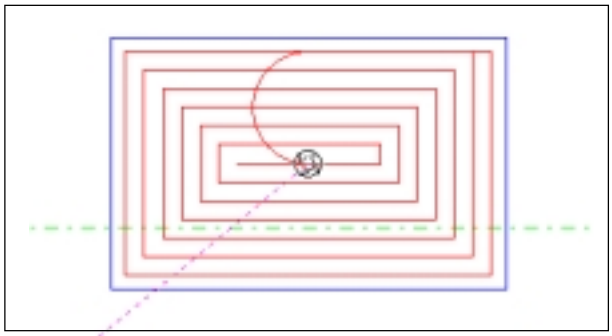


WinCAM window "rectangular pocket"

Input of the milling direction:
According to your decision to mill in climb or conventional way click into the respective input square.

Input cycle or ISO format
The status display cycle or DIN indicates (according to target control) if the cycle is resolved in individual path commands or not.

Transversal rectangular pocket
If DIN programming is used the pocket can also lie inclined in the plane.



Machined rectangular pocket

Rectangular pocket

By selecting the function a machining cycle is started which serves for milling a rectangular area. Just like with every other kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

The command is divided into 3 steps:

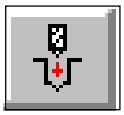
1. Define the rectangle to be machined.
2. Input of the cycle parameter in the appearing picture "rectangular pocket".
3. The automatic machining of the rectangular area at the CAM screen and the corresponding input of the NC program in the editor.

WinCAM window "rectangular pocket"

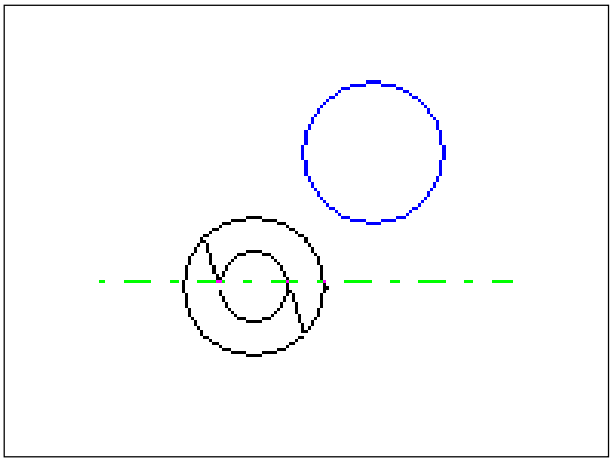
Enter the cycle parameters into this window and confirm the inputs with a click on the field "OK".
By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

MP(1) Center axis 1 = absolute coordiante of the rectangle center in Y-axis.
MP(2) Center axis 2 = absolute coordinate of the rectangle center in X-axis.
L Pocket length = incremental length of the rectangle.
B Pocket width = incremental width of the rectangle.
R Radius = radius of the four roundings of the rectangle
T Depth = absolute depth of the milled area in relation to the CAM reference point.
Z(t) Advance depth = advanced depth per milling cycle in relation to the start plane.
RE Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point
SE Start plane = absolute height of the cycle start plane in relation to the CAM reference point
F(t) Z feed = feed with which is advanced in Z-direction.
F(f) XY feed = feed with which is advanced in XY plane.
S Spindle speed = Speed of the main spindle during the cycle.



Drilling



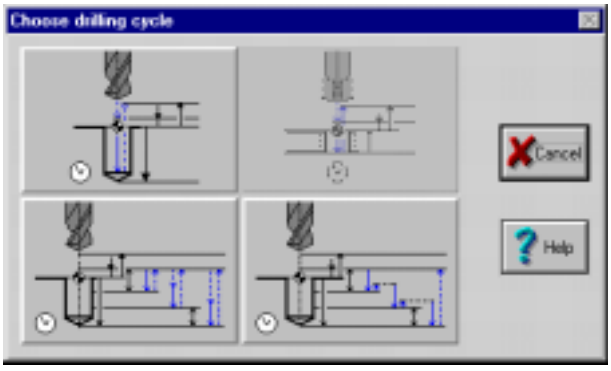
Define circle geometry

By selecting the function one of four machining cyles is started which serves for milling a bore.

Just like with every other kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

The command is divided into 4 steps:

1. Defining the drilling hole(s) (select center of circle geometries)
2. Selection of one of the four drilling cycles in the WinCAM window "choose drilling cycle".
3. Input of the cycle parameters in the appearing picture of the respective drilling cycle.
4. The automatic machining of the bore at the CAM screen and the corresponding input of the NC program in the editor.



*WinCAM window "*choose drilling cycle*"*

WinCAM window "choose drilling cycle"
In this window you find 4 drilling cycles which can be called up by selection by means of the mouse cursor. Then the corresponding WinCAM window of the selected cycle is displayed.

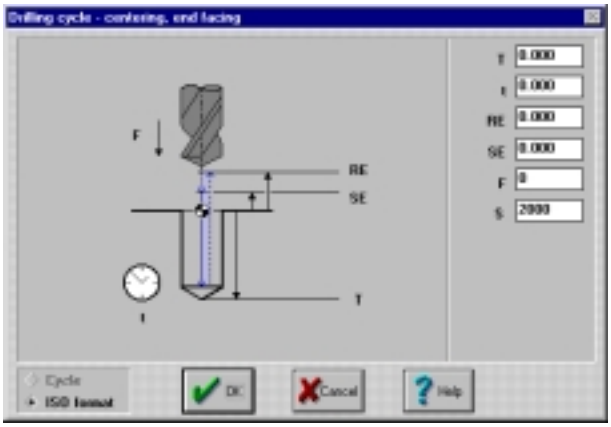
The following drilling cycles can be selected:

1. Drilling cycle removal
   For easy drilling with feed and return with rapid motion.

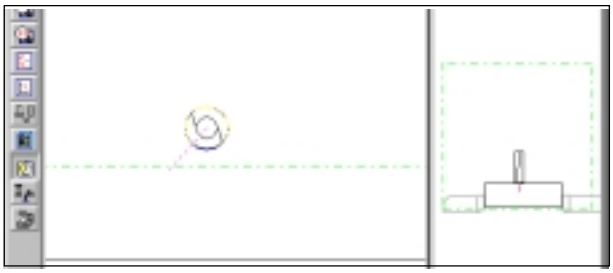
2. Deep hole drilling (removal)
   For machining deeper bores in steps with feed and immediate removal in rapid motion.

3. Deep hole drilling (chip breaking)
   For machining deeper bores and/or material with bad cutting properties in steps with feed and short returns in rapid motion for chip breaking.

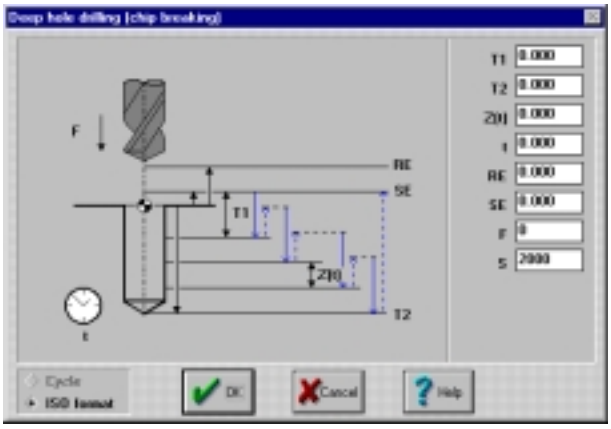
4. Drilling - thread cutting
   Thread-cutting is possible and selectable only with a connected NC-machine with encoder. With NC-machines without encoder this cycle is displayed in screened mode and thus without function (e.g. PC Mill 50).



*WinCAM window "*Drilling cycle removal*"*



Bore ready machined



WinCAM window "Deep hole drill. (chip breaking)"

SE Start plane = absolute height for the start of machining with feed in relation to the CAM reference point

F Advance = feed with which is advanced in Z-direction

S Spindle speed = speed of the min spindle during the cycle

Input cycle or ISO format

The status display cycle or DIN indicates (according to target control) whether the cycle is resolved in individual path commands or not.

WinCAM window "Drilling cycle removal"

Enter the cycle parameters into this window and confirm the inputs with a click on the field "OK". By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

T Depth = absolute depth of the bore in relation to the CAM reference point

t Dwell time = time of rest remained on the bore ground before return is carried out

RE Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point

SE Start plane = absolute height for the start of machining with feed in relation to the CAM reference point

F Advance = feed with which is advanced in Z-direction

S Spindle speed = spindle of the main spindle during the cycle

Input cycle or ISO format

The status display cycle or DIN indicates (according to the target control) whether the cycle is resolved in path commands or not.

WinCAM window "Deep hole drilling (chip breaking)"

Enter into this window the cycle parameters and confirm the inputs with a click on the field "OK". By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

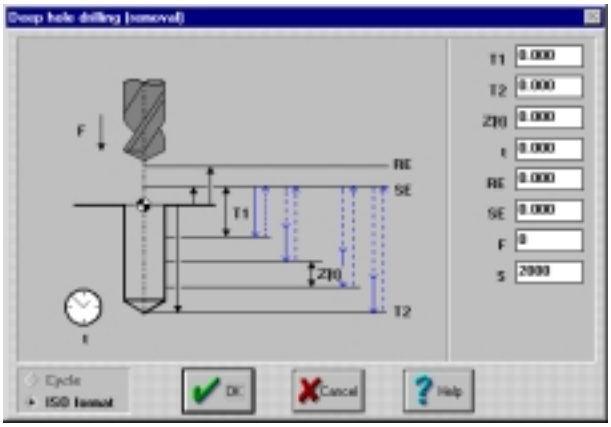
T1 First advance = incremental measure for the first bore step in relation to the start plane

T2 Overall depth = absolute measure for the overall depth of the bore in relation to the CAM reference point

Z(t) Advance depth = incremental measure for the advance between the chip breaking steps.

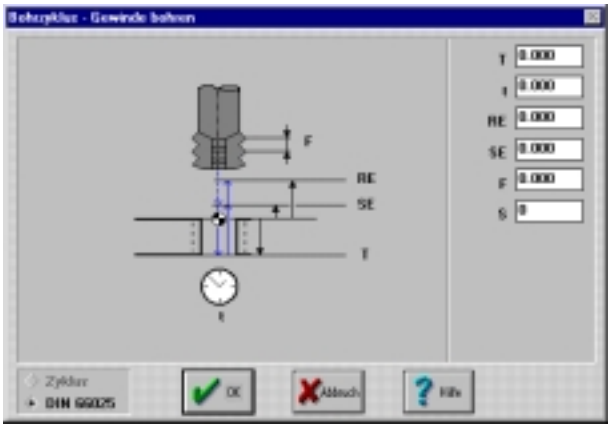
t Dwell time = time or rest on the bore ground before the return is carried out.

RE Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point



WinCAM window - "Deep hole drilling (removal)"

Input cycle or ISO format
The status display cycle or DIN (according to the target control) indicates whether the cycle is resolved in individual path commands or not.



*WinCAM window "*Drilling - thread cutting*"*

Input cycle or ISO format
The status display cycle or DIN (according to the target control) indicates whether the cycle is resolved in individual path commands or not.

Hints:
Thread-cutting is possible and selectable only with a connected NC-machine with encoder. With NC-machines without encoder this cycle is displayed in screened mode and thus without function (e.g. PC Mill 50).

WinCAM window "Deep hole drilling (removal)"

Enter the cycle parameters into this window and confirm the inputs with the field "OK".
By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

The inputs in detail:

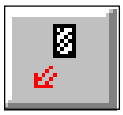
T1	First advance = incremental measure for the first drilling step in relation to the start plane
T2	Overall depth = absolute measure for the overall depth of the bore in relation to the CAM reference point
Z(t)	Advance depth = incremental measure for the advance between the removal steps
t	Dwell time = time of rest on the bore ground before return is carried out.
RE	Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM-reference point
SE	Start plane = absolute height for the start of machining with feed in relation to the CAM reference point
F	Advance = feed with which is advanced in Z-direction
S	Spindle speed = speed of the main spindle during the cycle.

WinCAM window "Drilling - thread cutting"

Enter the cycle parameters into this window and confirm the inputs with a click on the field "OK".
By clicking on "Cancel" the window is closed and the command interrupted.
The field "Help" calls up the online help.

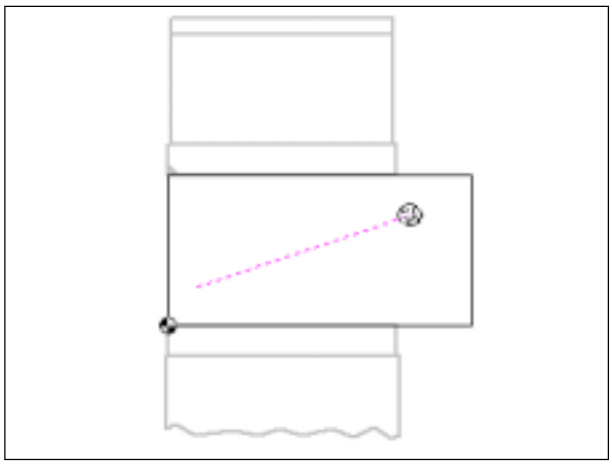
The inputs in detail:

T	Depth = absolute depth of the thread bore in relation to the CAM reference point
t	Dwell time = time of rest on the bore ground before return is carried out
RE	Return plane = absolute height for the traverse movements in rapid motion in relation to the CAM reference point
SE	Start plane = absolute height for the start of machining with feed in relation to the CAM reference point
F	Thread pitch
S	Spindle speed = speed of the main spindle during the cycle.



Movement in rapid motion in plane

By selecting the function the tool can be traversed in XY or XZ-plane in rapid motion.



Movement in rapid motion in plane

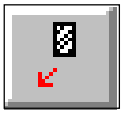
Just like with every kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

After clicking on the command symbol with the mouse WinCAM expects the definition of the end point. This can only be carried out by:

1. actual cursor position and mouse click
2. the pick mode menu and mouse click
3. a definition of the coordinates (INC or ABS)

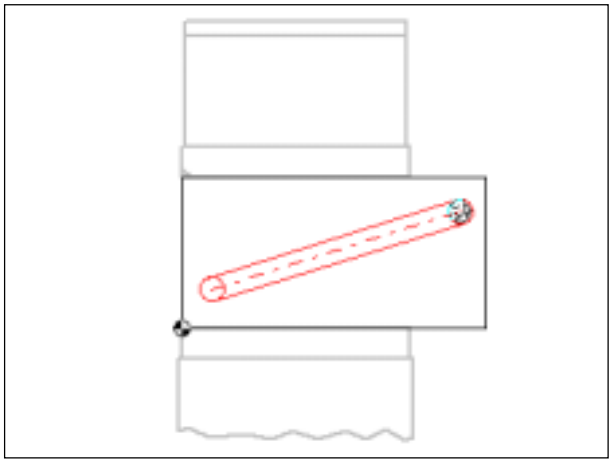
Mind that the desired window is active. The input of the traverse movement into the NC program is carried out automatically.

The display of the traverse path is carried out according to definition in the menu "Setup - line - setup line attributes" (e.g. broken line).



Movement in operating feed in plane

By selecting the function the tool can be traversed in XY or XZ-plane in the actual operating feed.



Movement in operating feed in plane

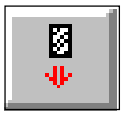
Just like with every kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

After clicking on the command symbol with the mouse WinCAM expects the definition of the end point. This can only be carried out by:

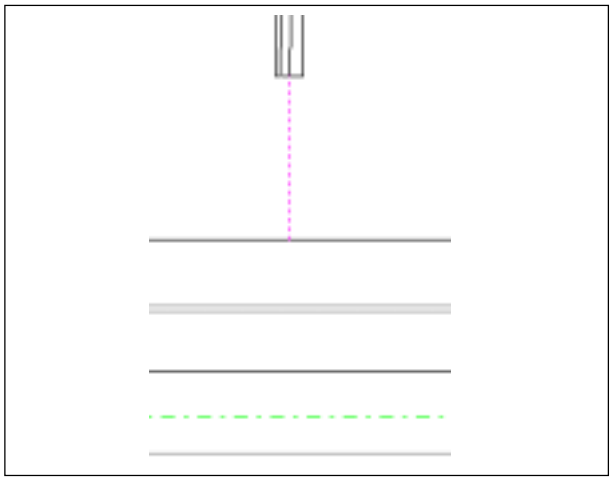
1. actual cursor position and mouse click
2. the pick mode menu and mouse click
3. a definition of the coordinates (INC or ABS)

Mind that the desired window is active. The input of the traverse movement into the NC program is carried out automatically.

The display of the traverse path is carried out according to definition in the menu "Setup - line - setup line attributes" (e.g. full line).



Movement in rapid motion in feed direction



Movement in rapid motion in feed direction

By selecting the function the tool can be traversed in Z-direction in rapid motion.

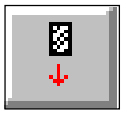
Just like with every kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

After clicking on the command symbol with the mouse WinCAM expects the definition of the end point. This can only be carried out by:

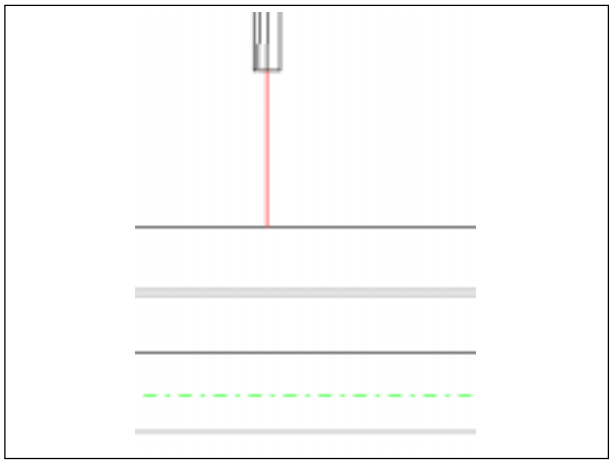
- a definition of the coordinates (INC or ABS)

The input of the traverse movement into the NC program is carried out automatically.

The display of the traverse path is carried out according to definition in the menu "Setup - line - setup line attributes" (e.g. broken line).



Movement in operating feed in feed direction



Movement in operating feed in feed direction

By selecting the function the tool can be traversed in Z-direction in the actual operating feed.

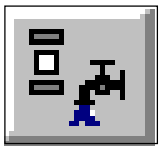
Just like with every kind of machining WinCAM must know the initialization of the machine (see chapter CAM bases).

After clicking on the command symbol with the mouse WinCAM expects the definition of the end point. This can only be carried out by:

- a definition of the coordinates (INC or ABS)

The input of the traverse movement into the NC program is carried out automatically.

The display of the traverse path is carried out according to definition in the menu "Setup - line - setup line attributes" (e.g. full line).



Periphery

The task of the periphery function is to carry out the respective input into the NC program and thus in the editor.

Status display
You can recognize the actual status of the periphery according to the previous NC program by the position of the command symbol. With NC-machines without automatization accessory the respective symbols are displayed in screened mode.
In the following some examples in this concern:

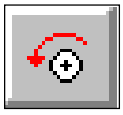
Main spindle off

Main spindle in counterclockwise direction

Main spindle in clockwise direction

Clamping device opened
Chip blow-out off
Automatic door opened

Clamping device closed
Chip blow-out off
Automatic door closed

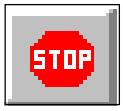


Main spindle on/left

After entering the function the respective input

- M04 for main spindle on in anticlockwise direction -

is carried out in the editor.

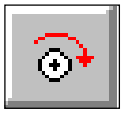


Main spindle stop

After entering the function the respective input

- M05 for main spindle STOP -

is carried out in the editor.

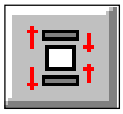


Main spindle on/right

After entering the function the respective input

- M03 for main spindle on in clockwise direction -

is carried out in the editor.

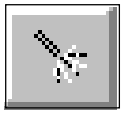


Clamping device close/open

After entering the function the respective input (e.g.)

- M26 for open clamping device - or
- M25 for close clamping device -

is carried out in the editor.
Mind that the clamping device can only be switched with open door.

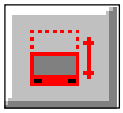


Chip blow-out on

After entering the function the respective input (e.g.)

- M71 for chip blow-out on-

is carried out in the editor.
The blow-out time is set to 3 seconds.

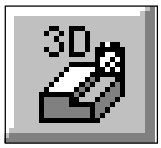


Automatic door open/close

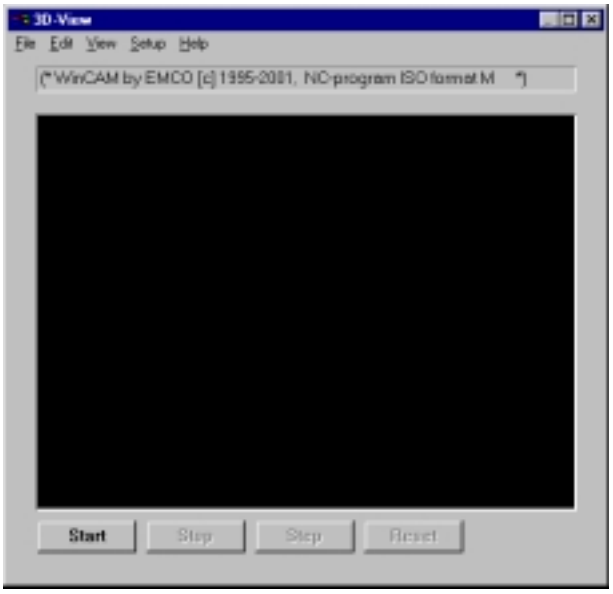
After entering the function the respective input (e.g.)

- M60 for open automatic door - or
- M61 for close automatic door -

is carried out in the editor.



3D-simulation



WinCAM window "3D-View"

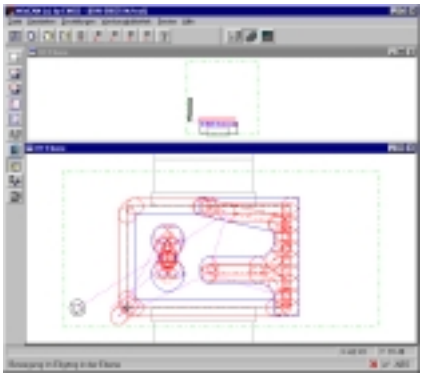
Via the symbol "3D-View" an additional program is called up by means of which the NC-program created in CAM mode can be simulated. The WinCAM window "3D-View" is shown. This window can be changed in size and form by means of the usual Windows functions.

The 3D-simulation has access to all WinCAM data available. Therefore, all information necessary for the simulation (unmachined part dimensions, position of the unmachined part in the clamping device, tool geometries, NC-program incl. cycles, etc.) are provided automatically. For that reason you are able to concentrate completely on the representation and operation of the simulation.

Note:
You can simulate only what you have previously created in CAM mode as NC-program or what has been loaded as NC-program.
Functions not available at the moment are displayed in a grid.

The way to ready simulation:

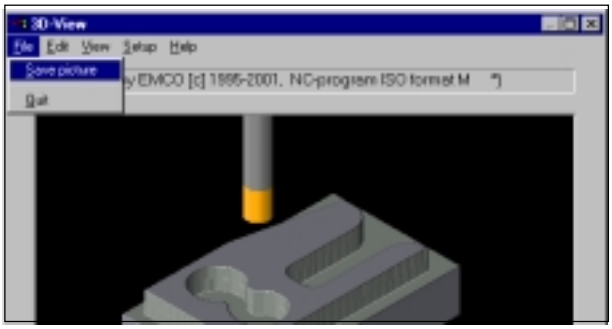
- Process NC-program in CAM mode
- Call up the simulation by means of the symbol "3D-View"
- If necessary, define view and selections
- Click control area "Start"



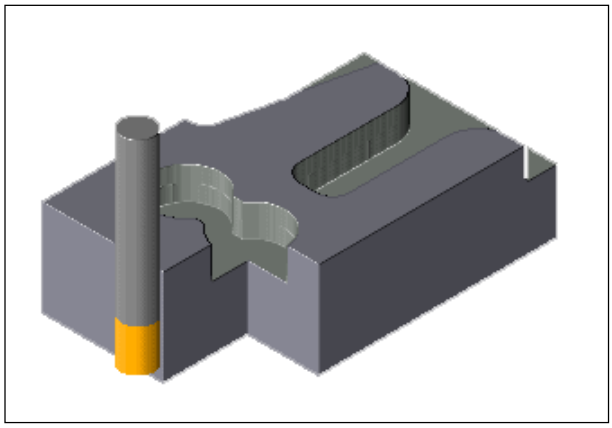
Workpiece in CAM mode

Workpiece without display of section

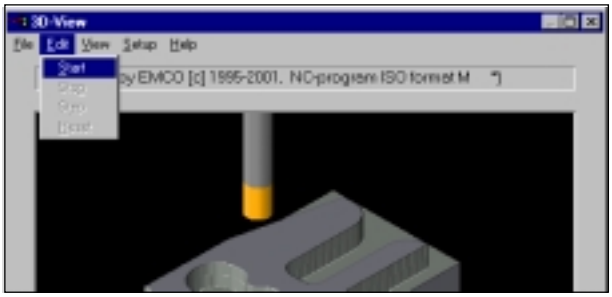
Workpiece with display of section



Menu "File"



Saved "Bitmap" picture



Menu "Edit"

Operation of simulation

One menu line and 4 control areas are available for operation. Below the menu line there is a status line in which the actual NC block is shown during the simulation.

Menu "File"

Save picture
This menu item calls up the Windows window "Save picture". The displayed simulation picture is saved in the indicated path, with the indicated file name as bitmap file (with the extension *.BMP). Then this file can be further processed and printed out with nearly all picture processing programs.

Quit
The simulation window is closed by a click on this menu item. You are back again in the WinCAM program.

Menu "Edit"

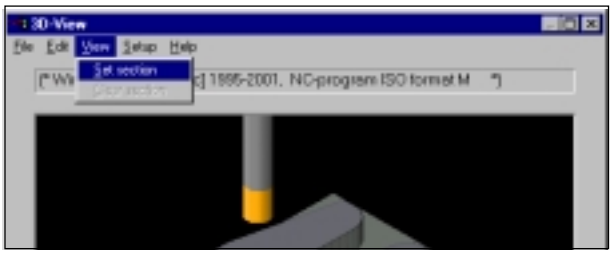
Start
Starts the NC program, simulation starts.

Stop
Stops the NC program. The simulation is in intermediate hold. A further processing is possible at any time by means of Start.

Step
Editing a single step of the NC program is carried out. Then the simulation goes into the intermediate hold.

Reset
The simulation is cancelled. By means of Start the NC program is restarted from the beginning.

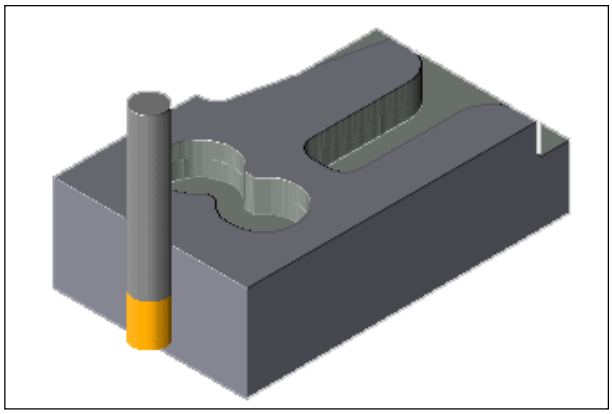
Note:
The 4 control areas in the lower part of the window have the same function. They simplify the operation.



Menu "View"

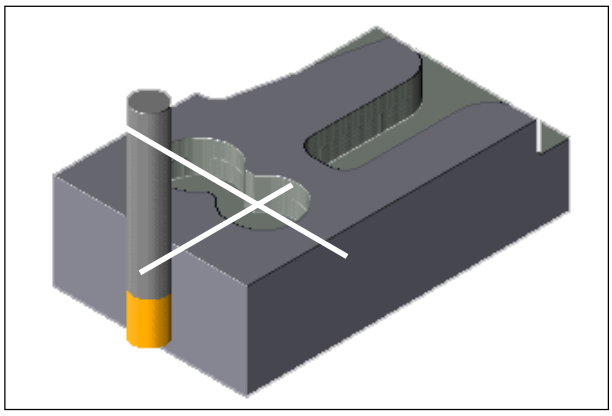
Menu "View"

Set section
By means of this menu item you define the position of the cutting surfaces. Draw the cross cursor by means of the mouse to the desired position and click the mouse key. The display is reestablished according to the section.

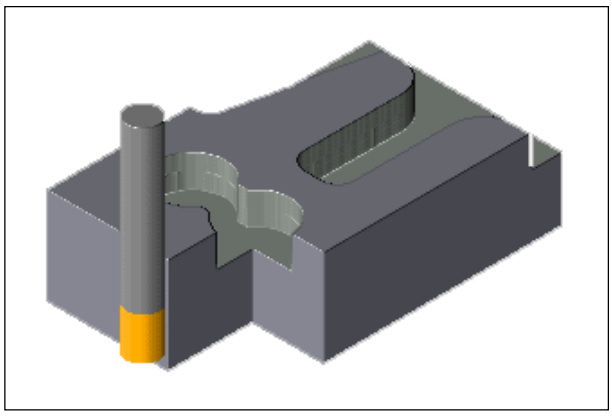


Workpiece uncut

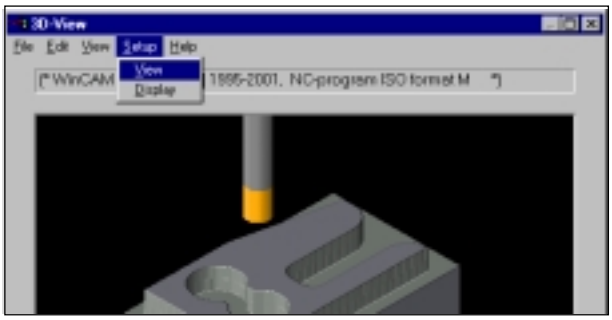
Clear section
If a section has been defined previously, you can cancel it with the function "clear section". The workpiece is displayed uncut again.



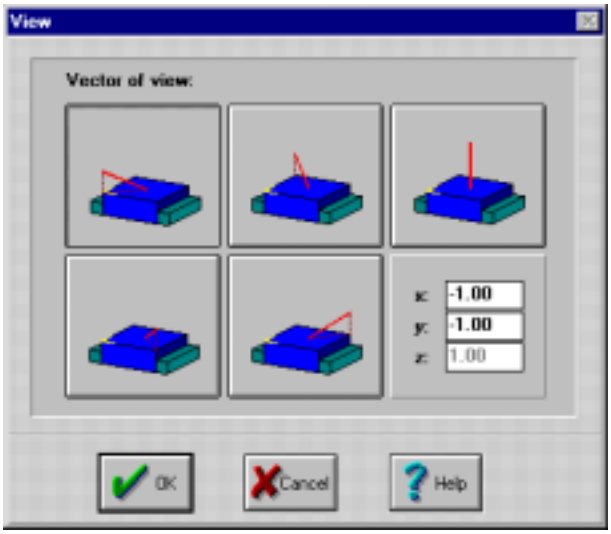
Define course of section with cursor cross



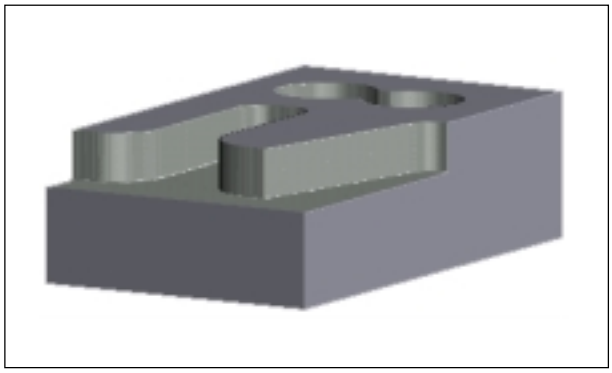
Workpiece cut



Menu "Setup"



WinCAM window "View"

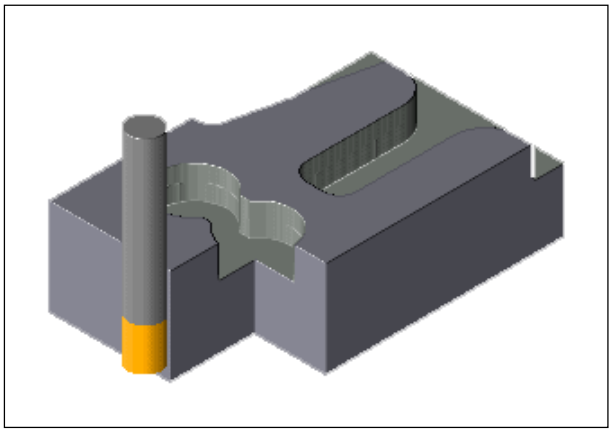


Example user defined view direction

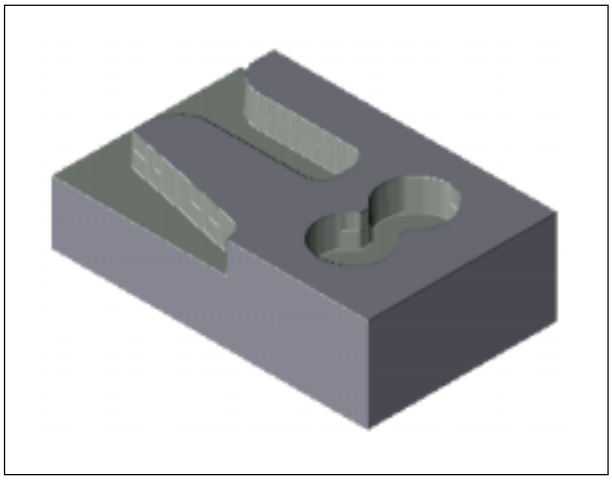
Menu "Setup"

View

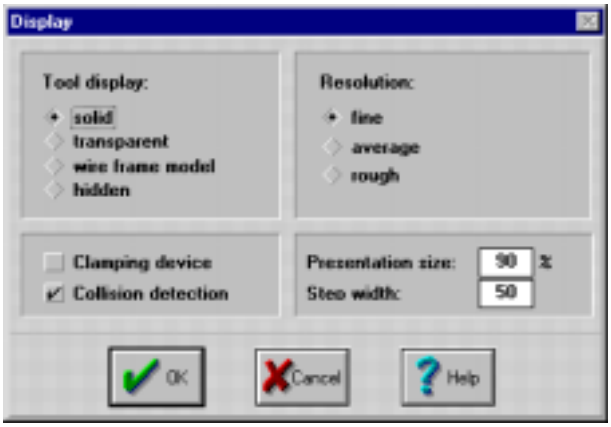
The WinCAM window "View" is opened. Here indicate the view direction of the simulation by mouse click onto the respective control area.
By means of the first five control areas standard directions are preset. You can also determine the direction yourself by the input of numerical values (relation x to y vector). To do this lead the mouse cursor into the input area "x" or "y" and click. Now you can enter the values and confirm with ENTER or OK. The values are saved and are taken into account during the start of the simulation.



Example standard direction



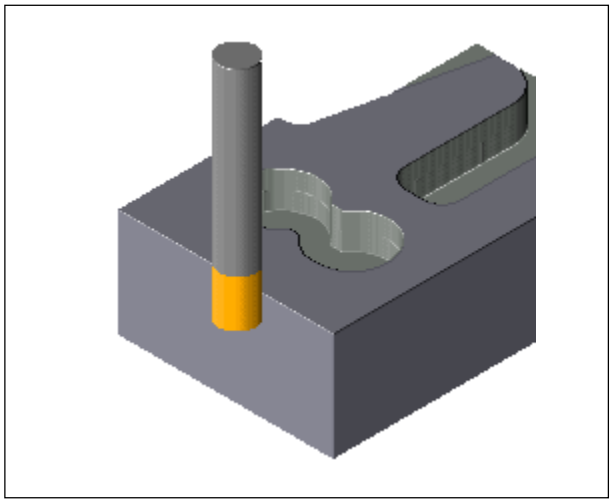
Example changed standard direction



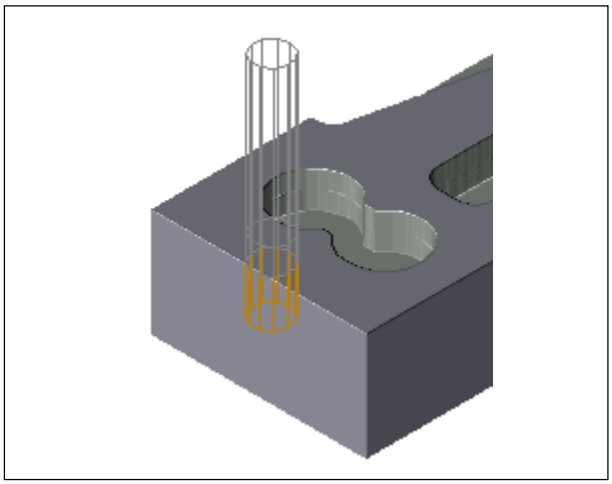
WinCAM window "Display"

Display
The WinCAM window "Display" is opened.

Tool display:
Solid: The tool is displayed shaded. It overlaps the tool.
Transparent: The tool is displayed shaded but transparently. The workpiece shines through the tool.
Wire frame model: The tool is only displayed as wire model.
Hidden: The tool is not visible.
Notes:
The lower cutting part of the tool differs in colour from the shaft of the tool.
During the the collision detection it is controlled, among other items, whether the cutting part of the tool touches the workpiece.



Tool in "Solid" presentation



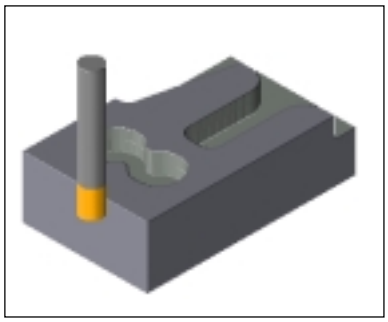
Tool in "wire frame model" presentation

Step width: Here enter the step width in the range of 1 to 99. The simulation divides every machining step as often as indicated. Small step widths cause a fluent simulation without jerks. Large step widths cause a quick simulation.

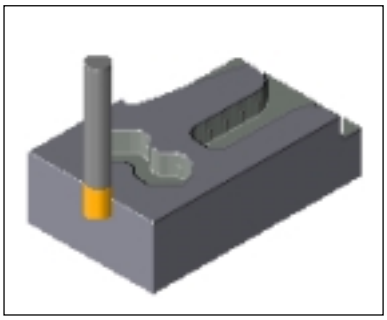
Clamping device: The vice is displayed.
Note:
With the clamping device display switched off, clamping device collisions are not controlled.

Note:
The finer the resolution, the larger the presentation and the more details the display has, the slower the simulation is. For this reason set a rougher and smaller display in case of low computer capacity.

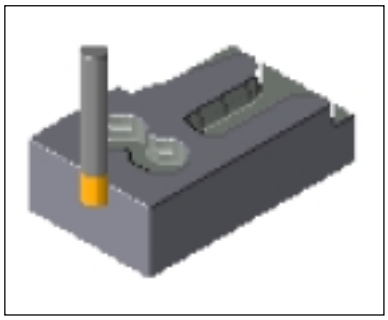
Resolution:
Fine: The simulation is displayed in the highest resolution.
Average: The simulation is displayed in average resolution.
Rough: The simulation is displayed in rough resolution.
Note:
With rough resolution of the simulation indication errors of collisions may occur due to rounding errors. In this case change to "fine" resolution.



Simulation in resolution "fine"



Simulation in resolution "average"



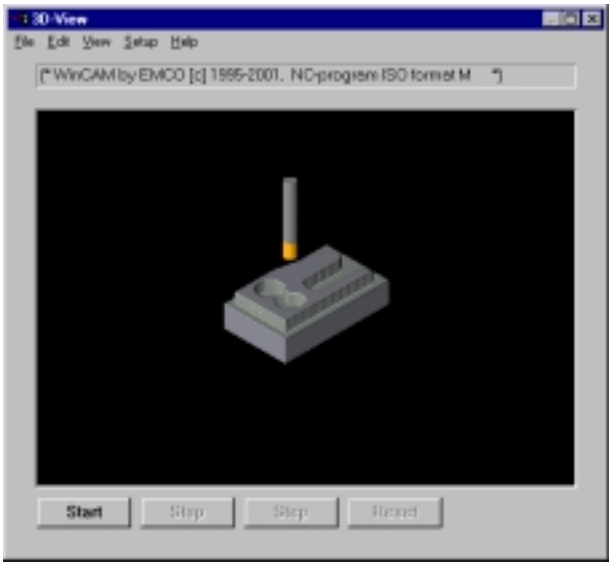
Simulation in resolution "rough"

Collision detection: WinCAM calculates an indadmissible collision of the tool, cancels the simulation and emits a respective error message.
The following situations are controlled with the collision detection:

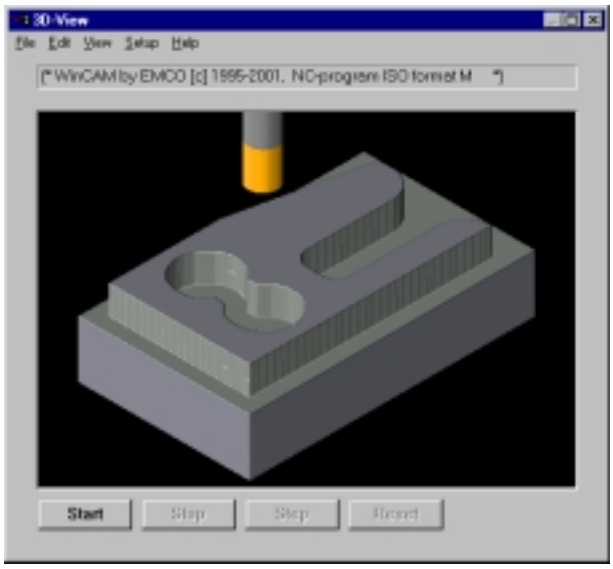
- Contacts of tool and workpiece during rapid motion movements.
- Contacts of tool and clamping device (does not occur, unless the clamping device is shown)
- Contacts of non-cutting tool parts with the workpiece or the clamping device.

Notes:
With collision detection switched off no machining is indicated in case of rapid motion collisions.
With rough resolution of the simulation indication errors of collisions may occur due to rounding errors. In this case change to "fine" resolution.

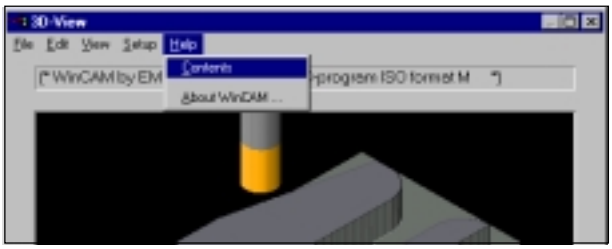
Presentation size: Here select between 50 % and 100 %. With 100 % the tool is presented in the largest possible size.



Presentation size 50 %



Presentation size 100 %



Menu "Help"

Menu "Help"

Also the simulation program of WinCAM is included in the WinCAM online help.

Contents
This menu item jumps into the WinCAM help. There search the desired item with the usual windows functions.

About WinCAM
The menu item calls up the WinCAM window "About EMCO WinCAM".

Creation of Tool Models

You can create your own tools for 3D simulation.
The model language for creating the tool models is described in the following.

The tool definition consists of two parts:

Tool parameter (in the tool library)
Tool model (in the tool model file)

In the tool library the parameter (length, diameter, ...) and the used tool model will be defined.

The tool model defines the basic form of the tool.
Similar tools with different lengthes can base on the same tool model.

For creating tools you have to edit the files TOOLxyy.DAT and 3DMODELx.DAT, meaning:
TOOLxyy.DAT= tool library
3DMODELx.DAT = tool model file
x: M: metric, I: inch and
yy: DT: German, EN: English, FR: French, SP:Spanish.

e.g.: TOOLMDT: tool library, metric, german version

Note:
For altering the tool files use the Windows-Editor or DOS-Edit.

Structure of the Tool Library

e.g.: File TOOLMEN.DAT:

1. tool

```
[Tool1M]
IndexHolder=0
ToolAngle=0.000000 ..... *
CutRadius=1.500000 ..... *
CutLength=5.0 ..... *
CutAngle=0.000000 ..... *
ShaftRadius=3.0
ShaftLength=41.0
ToolLength=49.000000
ToolKind=30
ToolName=Langlochfr. 3mm ..... *
Comment=HSS, DIN 327 Form B ... *
CutKind=0 ..... *
ToolHolderType=VERTICAL
ToolModel=Langlochfräser
```

Parameter of the tool, these will be taken over to the model.
The names of these variables are free, but must not be a command name (see behind).

The tool parameter marked with a * have fixed names and will be displayed in the tool selection menu of Win 3D-View.
These names must not be altered.

Tool name, max. 20 characters
Comment, max. 20 characters
*Cutter tip position**

Reference to model instruction

2. tool

```
[Tool2M]
IndexHolder=0
ToolAngle=0.000000
.....
```

* *The cutter tip position always has to be entered for the tool clamped over the turning centre.*
When the tool is clamped below the turning centre, the system will consider this automatically.

Structure of the Model File

e.g.: File 3DMODELM.DAT:

List of all model instructions

Name of the model instructions in the correct sequence

```
[Models]
Model1 = SlotMiller
Model2 = ShellEndMill
Model3 = TwistDrill
Model4 = NCDrill
Model5 = Tap
```

Name of the model instruction

model commands (command lines)

End of the model instruction

```
[SlotMiller]
Length1 = Subtraction (ToolLength, ShaftLength)
Shaft2 = Cylinder (ShaftRadius, ShaftLength)
Shaft2 = 3DTranslation (Shaft2, 0, 0, Length1)
Length1 = Subtraction (Length1, CutLength)
Cutter = Cylinder (CutRadius, CutLength)
Shaft1 = Cylinder (CutRadius, Length1)
Shaft1 = 3DTranslation (Shaft1, 0, 0, CutLength)
Shaft = 3DUnion (Shaft1, Shaft2)
MillTool (MILL_REMOVECOLOR, Cutter, MILL_CUTTERC...)
TOOL_MODEL_END
```

Name of the model instruction

```
[ShellEndMill]
Length1 = Subtraction (ToolLength, ShaftLength)
Shaft2 = Cylinder (ShaftRadius, ShaftLength)
.....
```

The file 3DMODELx.DAT contains some other sections (e.g.: [ModelNames]); do not alter them.

When you want to add a model instruction, first you must add the model istruction name at the end of the section [Models] with a continous number.

Example:
In the example above you have to add the new tool (chamfer miller) as following:

```
[Models]
Model1 = SlotMiller
Model2 = ShellEndMill
Model3 = TwistDrill
Model4 = NCDrill
Model5 = Tap
Model6 = ChamferMiller
```

The new model instruction must be added as last model instruction in the file 3DMODELx.DAT.

Model Commands

The shape of the tool is defined by model commands in the model instruction.

The created 2D or 3D objects can be set together in any complex way. Any number of new variabes can be generated. The name of a variable must not be a commands name!

Every line which contains a semicolon (;) will be interpreted as comment line. The length of a comment or a command in the model instruction is limited to 255 characters (1 line).

All entered details for a tool in the tool library (file TOOLXYY.DAT) as tool angle, cutter angle, cutter length,... can be used with the same name for the tool model.

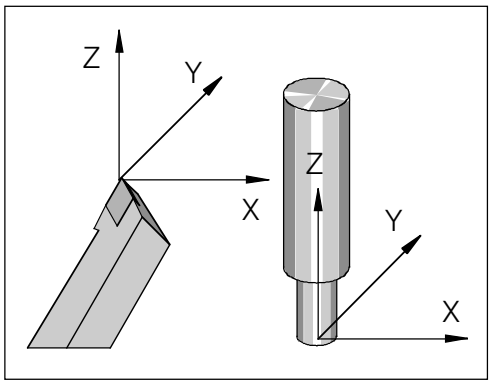
NOTE: For the metric tool libraries all numbers will be interpreted as millimeter, for the inch tool libraries as inch!

The tool modelling occurs in a coordinate system. The zero point of this coordinate system is the tool tip of the turning tool or the tip or face centre of the milling tool.

Structure of a command line:

Name = command (parameter 1, parameter 2, ...) or
Name = number constant

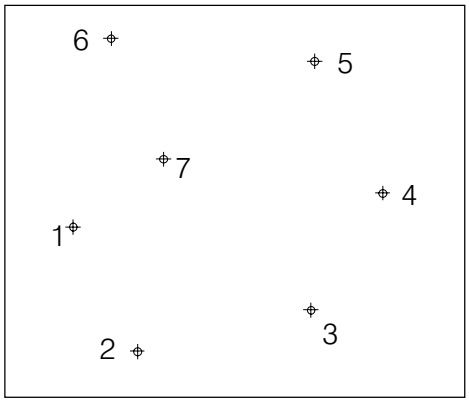
A command can call up only parameter, but not other commands, that means, nesting of commands is not possible



Coordinate origins for tool models

2D Modelling

2D modelling is necessary only for turning tools.
The 2D contours for the cutting part and the tool shaft (holder) will be used only internally for the collision detection and must be entered always over the turning centre!



DefinePoints defines a set of points

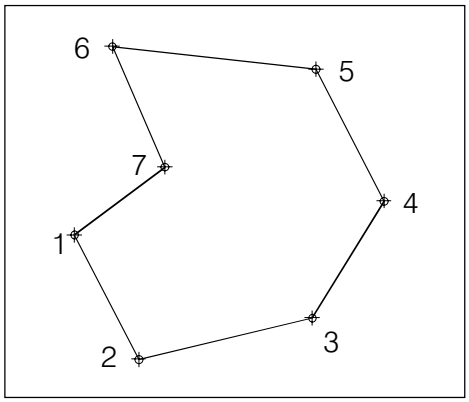
Points Set

To program a line draft, the edge points of this draft have to be determinated with the command *DefinePoints* before.
In this example the points set has the name "Points1":

Points1 = *DefinePoints* (x1, y1, r1, x2, y2, r2, ...)

Definition of a set of points in the 2-dimensional space, the number of parameter must be a multiple of three.
If r1, r2, r3,..≠0, these values determine an transition radius in the edge:
r > 0...arc counterclockwise
r < 0...arc clockwise
The maximum seectorial angle of an arc is 180 °!

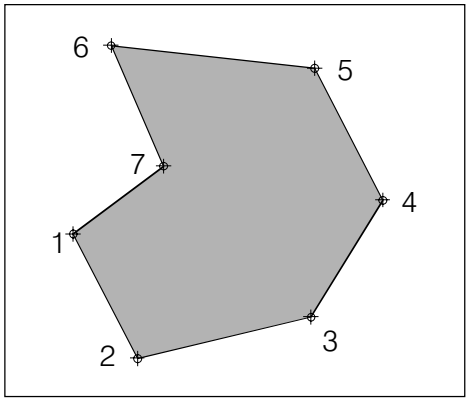
Line Drafts and Planes



SimplePolygon connects a set of points with a line

The command *SimplePolygon* combines points to a closed line draft.
These line draft will not be displayed at the screen, it is used for collision detection.
In this example the line draft has the name "Contour1" and connects the set of points called "Points1":

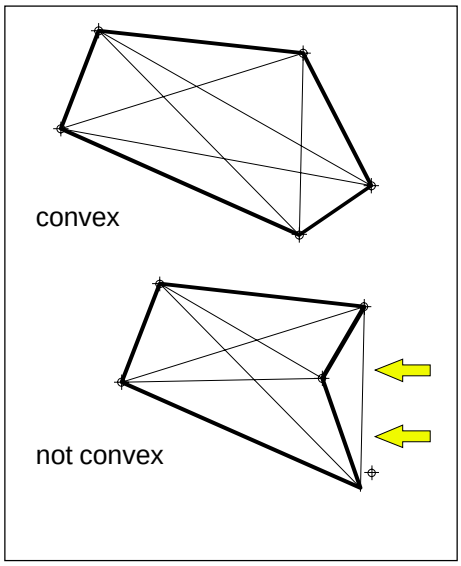
Contour1 = *SimplePolygon* (Points1)
Definition polygon draft without intersection points



SimplePolygonGraphic connects a set of points with a closed line draft and fills the plane

The command *SimplePolygonGraphic* combines points to a closed line draft. "ContourColour" is the colour of the lines, "FillColour" is the colour of the plane (see "Colour Constants").
These plane will be displayed at the screen with simulation.
In this example the plane has the name "Contour2" and connects the set of points "Points1":

Contour2 = *SimplePolygonGraphic* (Points1, ContourColour, FillColour)
Definition of a 2D graphic object (plane)



With convex line drafts all connection lines between the single points are inside the contour.

The command *ConvexPolygon* combines points to a closed line draft. *ConvexPolygon* is similar to *SimplePolygon*, but the line draft must be convex.

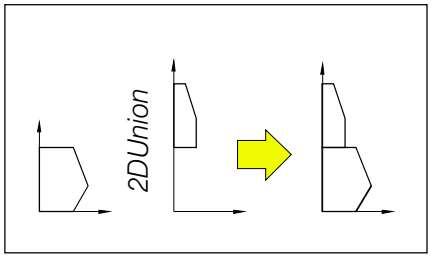
A line draft is convex when all connection lines between the single points are inside the contour.

Only from convex line drafts a solid object can be created with3Dtool modelling.

Contour = *ConvexPolygon* (Points)
Definition of a convex polygon draft

The command *ConvexPolygonGraphic* combines points to a closed line draft. *ConvexPolygonGraphic* is similar to *SimplePolygonGraphic*, but the line draft must be convex.

Contour = *ConvexPolygonGraphic* (Points, ContourColour, FillColour)
Definition of a convex 2D graphic object (plane)

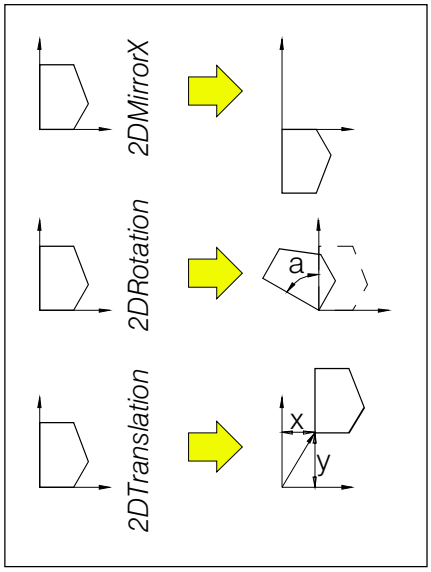


Combining 2D objects

Combining 2D Objects

The command *2DUnion* combines two 2D objects to one.
The objects to be combined are called "Object1" und "Objec2" und in this example, the combined object is called "Joint1":

Joint1 = *2DUnion* (Object1, Object2)
Combining 2D objects.



2D Transformations

2D Transformations

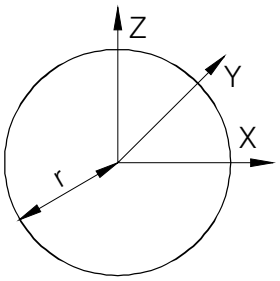
2D-Object2= *2DMirrorX* (2D-Object1)
Mirror around X axis

2D-Object2= *2DRotation* (2D-Object1, a)
Rotation counterclockwise for the angle a (in degrees).

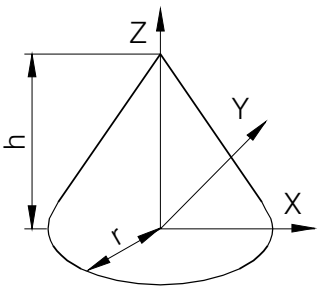
2D-Object2= *2DTranslation* (2D-Object1, x, y)
Shifting for the vector (x, y)

3D Modelling

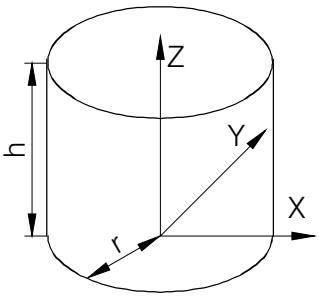
Basic Solid Bodies



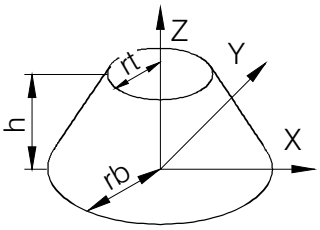
Sphere:
3D-Object = *Sphere* (r)
Sphere with centre in (0, 0, 0) and radius r.
NOTE: r > 0!



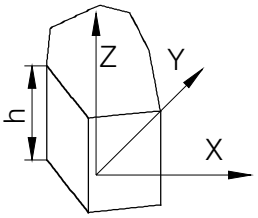
Cylinder:
3D-Object = *Cylinder* (r, h)
Cylinder with centere of the base in (0,0,0), radius r and height h.
NOTE: r and h > 0!



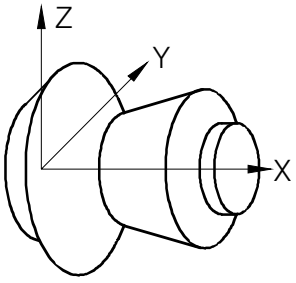
Cone:
3D-Object = *Cone* (r, h)
Cone with centre of the base in (0, 0, 0), radius of the base r and height h.
NOTE: r and h > 0!



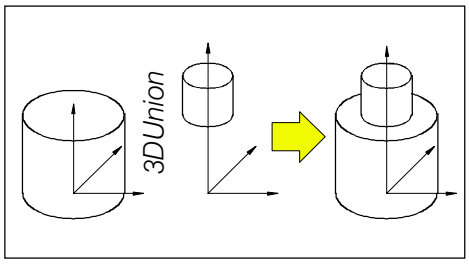
Trunc Cone:
3D-Object = *TruncCone* (rb, rt, h)
Trunc cone with centre of the base in (0,0,0), Radius of the base rb, radius of the trunc plane rt and height h.
NOTE: rb, rt and h > 0!



Prism:
3D-Object = *ConvexPrism* (Punkte, h)
Prism with convex base plane, which is defined by a convex polygon draft and height h. The set of points for the polygon draft is 2D modelled.
NOTE: h > 0!

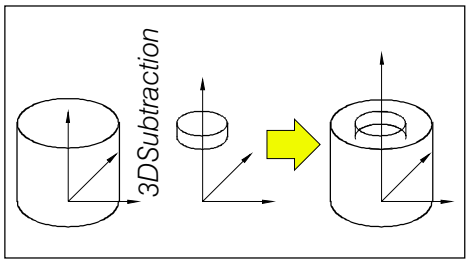


Rotational Solid Body:
3D-Object = *RotationSymetrical* (Punkte)
Creates a rotational symmetric 3D object by rotating a taken-over polygon draft (it need not to be convex!) around the X axis.
The set of points for the polygon draft is 2D modelled.
NOTE: The entire Y values of the points of the polygon draft must be > 0!

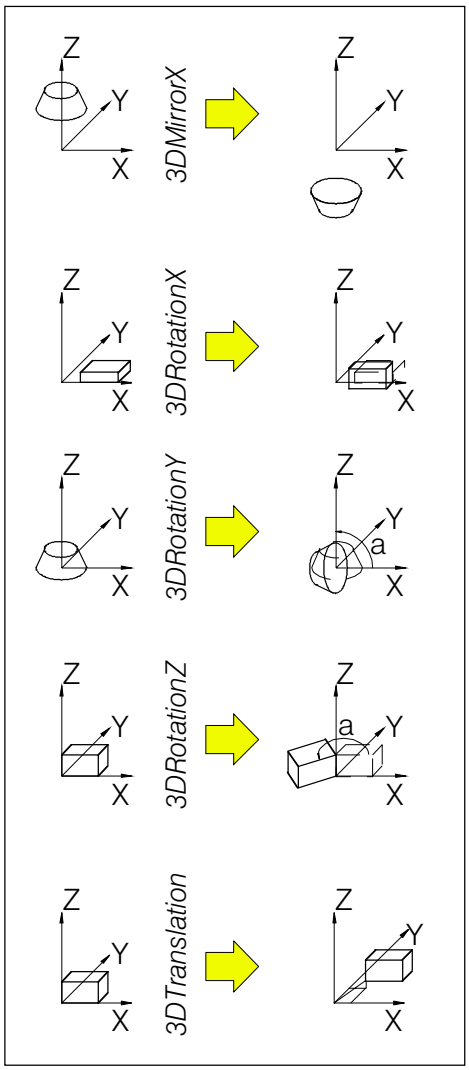


Combining 3D objects

3DObject3 = *3DUnion* (3DObject1, 3DObject2)
Combining 3D objects which can intersect in any way.



3DObject3 = *3DSubtraction* (3DObject1, 3DObject2)
Creating a 3D object by subtraction of Object2 from Object1 (cutting out). The 3D objects can intersect in any way.



3D Transformations

3DObject2 = *3DMirrorX* (3DObject1)
Mirror around the X axis

3DObject2 = *3DRotationX* (3DObject1, a)
Rotation conterclockwise around the X axis for the angle a (in degrees, 360° is a complete revolution)

3DObject2 = *3DRotationY* (3DObject1, a)
Rotation conterclockwise around the Y axis for the angle a (in degrees, 360° is a complete revolution)

3DObject2 = *3DRotationZ* (3DObject1, a)
Rotation conterclockwise around the Z axis for the angle a (in degrees, 360° is a complete revolution)

3DObject2 = *3DTranslation* (3DObject1, x, y, z)
Shifting for the vector (x, y, z)

Mathematic Functions

To calculate missing values vor tool modelling you can use the following mathematic functions.
The argument of a operation can be a variable or a constant.

Variable = *Addition* (Var1, Var2)
Addition of two values

Variable = *Subtraction* (Var1, Var2)
Subtraction of two values

Variable = *Multiplication* (Var1, Var2)
Multiplication of two values

Variable = *Division* (Var1, Var2)
Division of two vaues

Variable = *Sinus* (Var1)
Sinus of the argument (in degree)

Variable = *Cosinus* (Var1)
Cosinus of the argument (in degree)

Variable = *ArcusSinus* (Var1)
Arcus sinus of the argument
(-1,0 < Var1 < 1,0)

Variable = *ArcusCosinus* (Var1)
Arcus cosinus of the argument
(-1,0 < Var1 < 1,0)

Variable = *ArcusTangens* (Var1)
Arcus tangens of the argument

Variable = *SquareRoot* (Var1)
Square root of the argument
(Var1 > 0)

Colour Constants

You can use the colour constants which are defined in the file 3DVIEW.INI (in the machine data directory) for all modelling instructions which need colours.
The names of these colour constants must not be altered.
The RGB values of these colour constants can be altered in any way.

The entries in the section ColorDefinitions are used for colour screens, the entries in the section MonochromeDefinitions are used for monochrome screens.

The entries in the file 3DVIEW.INI are described in the chapter "Settings in the File 3DVIEW.INI".

Tool Definition

The tools for the simulations are defined with the parameters for the commands *MillTool* and *TurnTool*.
The names of these parameter are free setable. Win 3D-View reads the values of these parameters from the model file (3DMODELx.DAT) and displays the tools depending on these values.

The command *MillTool* defines a milling tool.
MillTool (CutColor, 3DCutter, CutterColor, 3DShaft, 3DColor)

Meaning of the parameter:

CutColor	Cutting colour
3DCutter	3D object of the cutting part
CutterColor	Colour of the cutting part for 3D
3DShaft	3D object of the tool holder (tool shaft)
ShaftColor	Colour of the tool holder (tool shaft) for 3D

The command *TurnTool* defines a turning tool.
TurnTool (CutColor, 2DCutter, 2DShaft, 2DTool, 3DCutter, CutterColor, 3DShaft, 3DColor)

For turning tools the 2D and 3D modelling must be defined separately.

Meaning of the parameter:

CutColor	Cutting colour
2DCutter	2D polygon draft of the cutting part *These entry is used for the collision detection and must be programmed always over the turning centre.*
2DShaft	2D polygon draft of the tool holder (tool shaft) *These entry is used for the collision detection and must be programmed always over the turning centre.*
2DTool	2D graphic object of the tool *These entry is used for the screen display and can be programmed over or below the turning centre.*
3DCutter	3D object of the cutting part
CutterColor	Colour of the cutting part for 3D
3DShaft	3D object of the tool holder (tool shaft)
ShaftColor	Colour of the tool holder (tool shaft) for 3D

For modelling 2DCutter or 2DShaft use the commands *ConvexPolygon* or *SimplePolygon*!

For the 2 dimensional display the parameter 2DTool is used, the 2D graphic object described with 2DTool can be programmed over or below the turning centre!
For modelling 2DTool use the commands *ConvexPolygonGraphic* or *SimplePolygonGraphic*!

Tools must be modelled with a tool shaft.

Examples for Tool Modelling

On the following pages are examples for a typical milling and for a typical turning tool.

When you want to create your own tool models, in the most cases you will copy the modelling of a similar tool and alter the entries.

Example: Milling

File TOOLMEN.DAT:

```
.....
[Tool1M]
IndexHolder=0
ToolAngel=0.000000
CutRadius=3.000000
CutLength=5.0
ShaftRadius=6.0
ShaftLength=35.0
ToolLength=49.000000
ToolKind=30
ToolName=Slot Miller 6mm
Comment=HSS, DIN 327 Form B
CutKind=0
ToolHolderType=VERTICAL
ToolModel=SlotMiller
.....
```

*Tool holder number**
*Tool angle**
*Cutter radius**
*Cutter length**
Shaft radius
Shaft length
Tool length
Tool type
*Name of the tool**
*Comment**
*Cutter position**
Tool holder type
Modelling instruction for the tool

*The points marked with * will be displayed in the tool selection menu of Win 3D-View.*

The values for variables entered in TOOLMDT.DAT will be taken over into modelling

File 3DMODELM.DAT:

```
.....
[SlotMiller]
```

The variable Length1 (=length between tool tip and shaft beginning) is the difference from tool length and shaft length.
The values for ToolLength and Shaftlength will be read from the file TOOLMDT.DAT

Length1 = *Subtraction* (ToolLength, ShaftLength)
Length1 = 49.0 - 35.0 = 14.0

The 3D object Shaft2 is a cyclinder with the radius ShaftRadius and the height Shaftlength

Shaft2 = *Cylinder* (ShaftRadius, ShaftLength)
Cylinder with r = 5.0, h = 35.0

The 3D-Object Shaft2 will be shifted in Z for Length1 and named Shaft2 again

Shaft2 = *3DTranslation* (Shaft2, 0, 0, Length1)
Shifting for 14.0 mm in Z (upwards)

The variable Length2 (=length between upper cutter edge and shaft beginning) will be calculated

Length2 = *Subtraction* (Length1, CutLength)
Length1 = 14.0 - 5.0 = 9.0

The 3D object for the cutting part will be defined

Cutter = *Cylinder* (CutRadius, CutLength)
Cylinder with r = 3.0, h = 5.0

The 3D object for the lower shaft part will be defined

Shaft1 = *Cylinder* (CutRadius, Length1)
Cylinder with r = 3.0, h = 9.0

The 3D object Shaft1 will be shifted in Z for the cutter length.

Shaft1 = *3DTranslation* (Shaft1, 0, 0, CutLength)
Shifting for 5.0 mm in Z (upwards)

The tool shaft consists of the parts Shaft1 and Shaft2

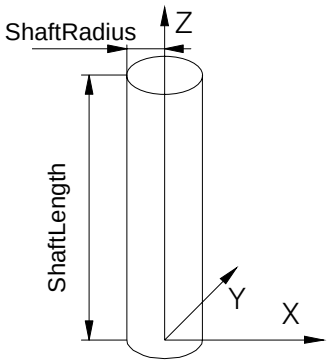
Shaft = *3DUnion* (Shaft1, Shaft2)
Combining the objects Shaft1 and Shaft2 to Shaft

Definition of the miller with the colour constants from 3DVIEW.INI, the cutting object Cutter and the shaft object Shaft

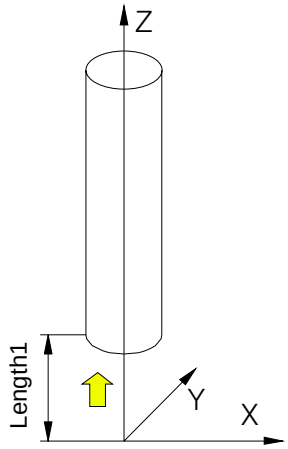
MillTool (MILL_REMOVECOLOR, Cutter, MILL_CUTTERCOLOR, Shaft, MILL_SHAFTCOLOR)

End of modelling

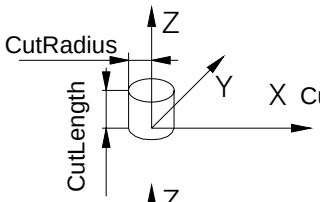
TOOL_MODEL_END



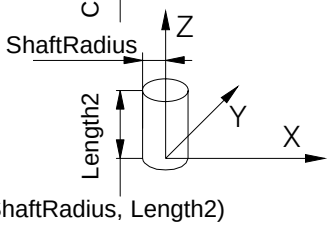
Shaft2 = Cylinder (ShaftRadius, ShaftLength)



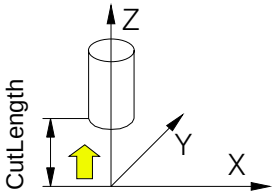
Shaft2 = 3DTranslation(Shaft2, 0, 0, Length1)



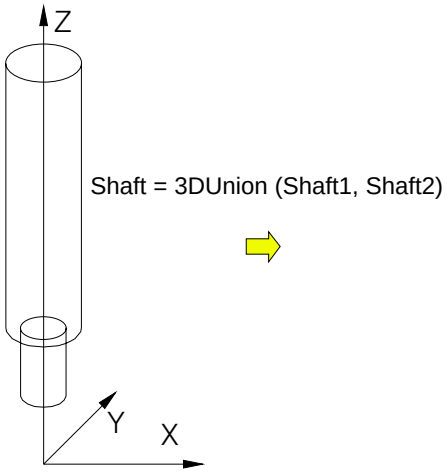
Cutter = Cylinder (CutRadius, CutLength)



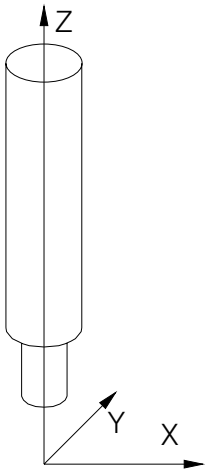
Shaft1 = Cylinder (ShaftRadius, Length2)

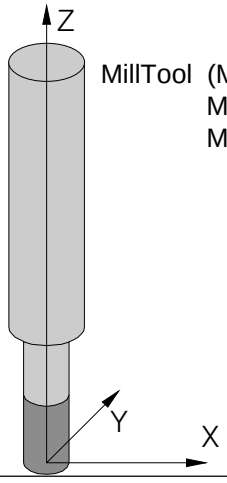


Shaft1 = 3DTranslation(Shaft2, 0, 0, CutLength)



Shaft = 3DUnion (Shaft1, Shaft2)





MillTool (MILL_REMOVECOLOR, Cutter, MILL_CUTTERCOLOR, Shaft, MILL_SHAFTCOLOR)

= MILL_SHAFTCOLOR

= MILL_CUTTERCOLOR

Example: Turning (tool over turning centre)

File TOOLMEN.DAT:

```
.....
[Tool3T]
IndexHolder=0
ToolAngel=120.500000
CutAngel=27.500000
CutRadius=0.400000
CutLength=7.750000
ToolKind=2
CutKind=4
ToolName=Right Copy Tool
Comment=
ToolHolderType=VERTICAL
ToolModel=RightCopyTool
.....
```

*Tool holder number**
*Tool angle**
*Cutter Angle**
*Cutter radius**
*Cutter length**
Tool type
*Cutter position**
*Tool name**
*Comment**
Tool holder type
Modelling instruction for the tool

*The points marked with * will be displayed in the tool selection menu of Win 3D-View.*

The cutter tip position always has to be entered for the tool clamped over the turning centre.
When the tool is clamped below the turning centre, the system will consider this automatically.

Simplified example without cutter radii:

File 3DMODELM.DAT:

```
.....
[RightCopyTool]
```

Calculation of the contour points for the cutter bit:

The calculations of the different parameters are described in the drawings on the page beside.

Angle1	=	*Addition* (CutAngel, 90)
Angle1	=	*Subtraction* (ToolAngel, Angle1)
Angle2	=	*Addition* (ToolAngel, CutAngel)
Angle2	=	*Subtraction* (180, Angle2)
Sinus1	=	*Sinus* (Angle1)
Cosinus1	=	*Cosinus* (Angle1)
Sinus2	=	*Sinus* (Angle2)
Cosinus2	=	*Cosinus* (Angle2)
x1	=	*Multiplication* (CutLength, Sinus1)
x1	=	*Subtraction* (0, x1)
y1	=	*Multiplication* (CutLength, Cosinus1)
x3	=	*Multiplication* (CutLength, Cosinus2)
x3	=	*Subtraction* (0, x3)
y3	=	*Multiplication* (CutLength, Sinus2)
x2	=	*Addition* (x1, x3)
y2	=	*Addition* (y1, y3)

Defining of the set of points for the cutter bit:

The points calculated above are defined as set of points with the name CutPoints

CutPoints = *DefinePoints* (0, 0, 0, x1, y1, 0, x2, y2, 0, x3, y3, 0)

You can see in the drawing: W1 = TA - CA - 90°

In the modelling a calculation can be proceeded only in single steps, so two steps with two commands are necessary:
Angle1 = *Addition* (CutAngle, 90)
Angle1 = *Subtraction* (ToolAngle, Angle1)

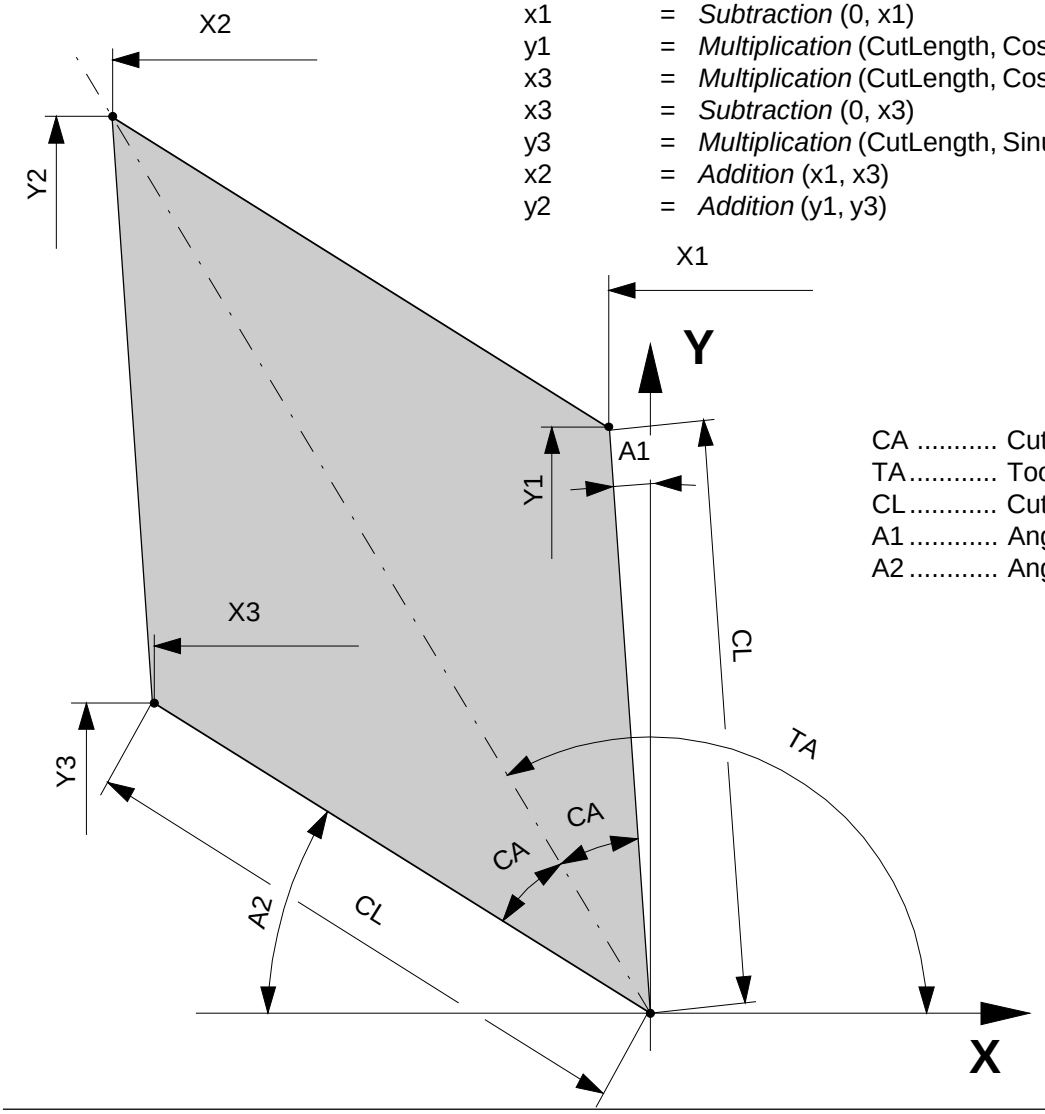
The same for Angle2: W2 = 180 - (TA + CA)
Angle2 = *Addition* (ToolAngle, CutterAngle)
Angle2 = *Subtraction* (180, Angle2)

The calculations for the X and Y values of the points are:

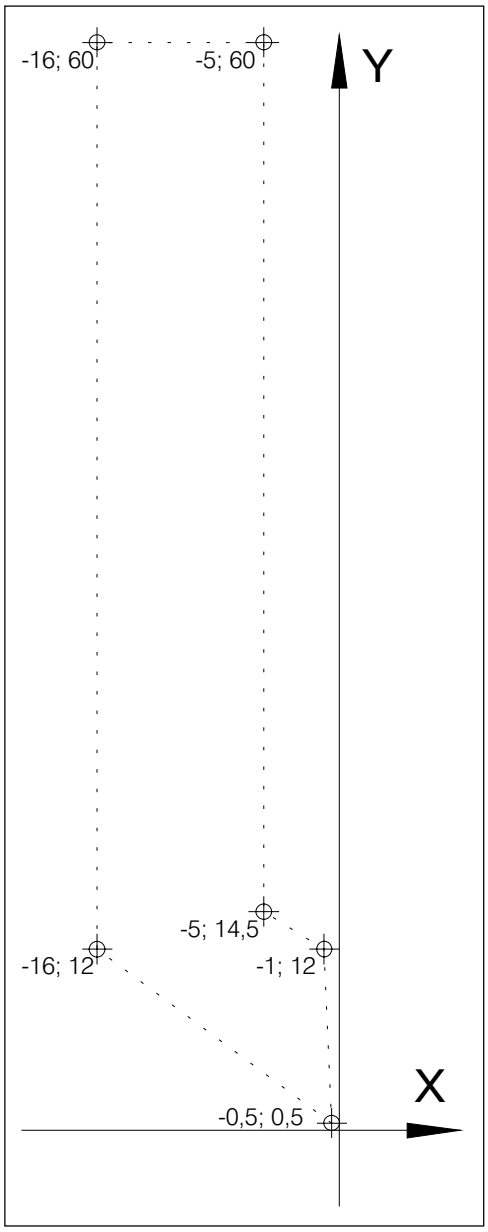
X1 = - CL x sin (Angle1) Y1 = CL x cos (Angle1)
X3 = - CL x cos (Angle2) Y3 = CL x sin (Angle2)
X2 = X1 + X3 Y2 = Y1 + Y3

The X values have a negative sign, because the cutter is on the left side of the axis (in the negative area).

The following commands will result in the modelling language:
Sinus1 = *Sinus* (Angle1)
Cosinus1 = *Cosinus* (Angle1)
Sinus2 = *Sinus* (Angle2)
Cosinus2 = *Cosinus* (Angle2)
x1 = *Multiplication* (CutLength, Sinus1)
x1 = *Subtraction* (0, x1)
y1 = *Multiplication* (CutLength, Cosinus1)
x3 = *Multiplication* (CutLength, Cosinus2)
x3 = *Subtraction* (0, x3)
y3 = *Multiplication* (CutLength, Sinus2)
x2 = *Addition* (x1, x3)
y2 = *Addition* (y1, y3)



CA ..... CutAngle
TA ..... ToolAngle
CL ..... CutLength
A1 ..... Angle1
A2 ..... Angle2



Points and polygon draft for the shaft

Defining the set of points for the tool holder (fixed values are used here, no values will be taken over from the file TOOLMDT.DAT.)

PntsShaft = *DefinePoints* (-0.5, 0.5, 0, -16, 12, 0, -16, 60, 0, -5, 60, 0, -5, 14.5, 0, -1, 12, 0)

The polygon draft for the cutter bit will be created with the command ConvexPolygon.

Cutter2D = *ConvexPolygon* (CutPoints)

The polygon draft for the tool shaft will be created with the command SimplePolygon.

Shaft2D = *SimplePolygon* (PntsShaft)

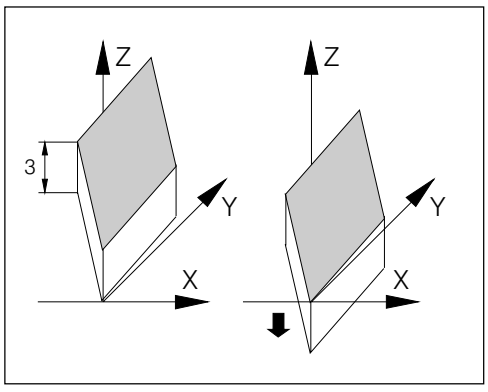
For the screen display a 2D graphic object must be defined. The tool with the name 2DTool consists of the two parts CutGraphic and Shaftgraphic.

CutGraphic= *ConvexPolygonGraphic* (PointsCutter, TURN_2DCUTTERCOLOR, TURN_2DCUTTERCOLOR)

ShaftGraphic= *SimplePolygonGraphic* (PointsShaft, TURN_2DSHAFTCOLOR, TURN_2DSHAFTCOLOR)

2DTool is the combination of the two 2D graphic objects

2DTool = *2DUnion* (ShaftGraphic, CutterGraphic)



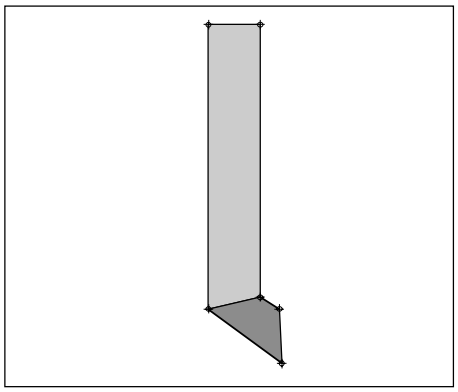
Modelling and shifting the 3D cutter bit

3D-MODELLING
The 3D object Cutter3D (cutter bit) is a prismatic extension of the cutter contour. The height of the prism is 3 mm.

Cutter3D = *ConvexPrism* (PointsCutter, 3)

For that the tool tip is in the coordinate zero, the prism must be shift down for its height.

Cutter3D = *3DTranslation* (Cutter3D, 0, 0, -3)



Shaft divided in convex polygons

Only convex polygons can be extended to 3D, so the shaft must be splitted in convex polygons (PntsShaft1 and PntsShaft2).

PntsShaft1 = *DefinePoints* (-0.5, 0.5, 0, -1, 12, 0, -5, 14.5, 0, -16, 12, 0)
PntsShaft2 = *DefinePoints* (-16, 12, 0, -16, 60, 0, -5, 60, 0, -5, 14.5, 0)

The 3D object Shaft3D is a prismatic extension of the shaft contour (same way as with the cutter bit)

Shaft3D1 = *ConvexPrism* (PointsShaft1, 12)
Shaft3D2 = *ConvexPrism* (PointsShaft2, 12)
Shaft3D = *3DUnion* (Shaft3D1, Shaft3D2)
Shaft3D = *3DTranslation* (Shaft3D, 0, 0, -12)

Tool and shaft will be rotated for 60° (in the same position in which it is also on the machine - the tool points from backside to oblique down)

Rotation of the tool around the turning axis

Cutter3D = *3DRotationX* (Cutter3D, 60)
Shaft3D = *3DRotationX* (Shaft3D, 60)

Calling the model function for the tool

TurnTool (TURN_REMOVECOLOR, Cutter2D, Shaft2D, 2DTool, Cutter3D, TURN_3DCUTTERCOLOR, Shaft3D, TURN_3DSHAFTCOLOR)

End of the tool modelling

TOOL_MODEL_END

Recapitulation of the command lines:

```
[RightCopyTool]
Angle1      =  Addition (CutAngel, 90)
Angle1      =  Subtraction (ToolAngel, Angle1)
Angle2      =  Addition (ToolAngel, CutAngel)
Angle2      =  Subtraction (180, Angle2)
Sinus1      =  Sinus (Angle1)
Cosinus1    =  Cosinus (Angle1)
Sinus2      =  Sinus (Angle2)
Cosinus2    =  Cosinus (Angle2)
x1          =  Multiplication (CutLength, Sinus1)
x1          =  Subtraction (0, x1)
y1          =  Multiplication (CutLength, Cosinus1)
x3          =  Multiplication (CutLength, Cosinus2)
x3          =  Subtraction (0, x3)
y3          =  Multiplication (CutLength, Sinus2)
x2          =  Addition (x1, x3)
y2          =  Addition (y1, y3)
CutPoints   =  DefinePoints (0, 0, 0, x1, y1, 0, x2, y2, 0, x3, y3, 0)
PntsShaft   =  DefinePoints (-0.5, 0.5, 0, -16, 12, 0, -16, 60, 0, -5, 60, 0, -5, 14.5, 0, -1, 12, 0)
Cutter2D    =  ConvexPolygon (CutPoints)
Shaft2D     =  SimplePolygon (PntsShaft)
CutGraphic  =  ConvexPolygonGraphic (CutPoints, TURN_2DCUTTERCOLOR, TURN_2DCUTTERCOLOR)
ShaftGraphic=  SimplePolygonGraphic (PntsShaft, TURN_2DSHAFTCOLOR, TURN_2DSHAFTCOLOR)
2DTool      =  2DUnion (ShaftGraphic, CutterGraphic)
Cutter3D    =  ConvexPrism (PointsCutter, 3)
Cutter3D    =  3DTranslation (Cutter3D, 0, 0, -3)
PntsShaft1  =  DefinePoints (-0.5, 0.5, 0, -1, 12, 0, -5, 14.5, 0, -16, 12, 0)
PntsShaft2  =  DefinePoints (-16, 12, 0, -16, 60, 0, -5, 60, 0, -5, 14.5, 0)
Shaft3D1    =  ConvexPrism (PointsShaft1, 12)
Shaft3D2    =  ConvexPrism (PointsShaft2, 12)
Shaft3D     =  3DUnion (Shaft3D1, Shaft3D2)
Shaft3D     =  3DTranslation (Shaft3D, 0, 0, -12)
Cutter3D    =  3DRotationX (Cutter3D, 60)
Shaft3D     =  3DRotationX (Shaft3D, 60)
TurnTool (TURN_REMOVECOLOR, Cutter2D, Shaft2D, 2DTool, Cutter3D, TURN_3DCUTTERCOLOR, Shaft3D, TURN_3DSHAFTCOLOR)
TOOL_MODEL_END
```

Settings in the File 3DVIEW.INI

In the file 3DVIEW.INI the basic settings are stored.
These settings can be altered with an editor.
Most of these settings can be altered easier with the WinCAM surface.

Note:
For altering the file 3DVIEW.INI you should have experience in working on WINDOWS or call an expert, otherwise the function of the software could suffer.

3DVIEW.INI	3DVIEW.INI consists of several sections. The following sections are valid for milling and turning:
[Window] windowWidth=490 windowHeight=475	Determining the size of the simulation window. Width and height will be entered in pixels. .
[Simulation]	General simulation data
zoomFactor=90	Zoom factor in %, with 100% the whole simulation window will be used for the simulation.
resolution=2	Resolution coarse - fine - very fine
viewVectorX=-1.000000 viewVectorY=-1.000000 viewVectorZ=1.000000	Determining of the viewing direction.
toolDisplayMode=0	Tool display wire model - volume model
stepWidth=10	Simulation step width in mm
fixtureOn=0	Clamping device display
lightSource0X=-0.707107 lightSource0Y=0.707107 lightSource0Z=1.000000 lightSource1X=-1.000000 lightSource1Y=-1.000000 lightSource1Z=0.000000	Determining of the position of the two light sources, will be used for the shaded view.
collisionDetection=1	collision detektion on (1) / off (0)
logfile=0	When logfile is active(1) all commands will be recorded which are sent from WinCAM to 3DView. Used for error detection.

The sections on the next page are different for turning and milling.

Settings for Turning

[PartingOff]
MetricMaxWidth = 3
InchMax Width = 0.12

Parted off workpiece parts (after cutting off), which are smaller or equal to the entered value (in mm or inch) in Z-direction, will not be displayed at the screen any more.

[MCodesTailstock]
Forward = 20
Back = 21

Definition of the M codes to move the tailstock.

[ColorDefinitions]					Definition of the screen colours for colour screen
TURN_BACKGROUND	= 0.0,	0.0,	0.0,	0	background colour
TURN_2DCUTTERCOLOR	= 0.875,	0.625,	0.0,	0	colour of the cutting part 2D display
TURN_2DTAPCUTTERCOLOR	= 1.0,	0.0,	1.0,	0	colour of the tapping tool 2D display
TURN_2DSHAFTCOLOR	= 0.4,	0.4,	0.4,	0	colour of the tool shaft (holder) 2D display
TURN_3DCUTTERCOLOR	= 0.375,	0.25,	0.0,	2	colour of the cutting part 3D display
TURN_3DTAPCUTTERCOLOR	= 1.0,	0.0,	1.0,	2	colour of the tapping tool 3D display
TURN_3DSHAFTCOLOR	= 0.75,	0.75,	0.75,	2	colour of the tool shaft (holder) 3D display
TURN_REMOVECOLOR	= 0.5,	0.5,	0.55,	2	Colour of the workpiece after chip removing
TURN_TAPREMOVECOLOR	= 1.0,	0.0,	1.0,	2	Colour of the workpiece after threading chip removing
TURN_CLAMPINGDEVICECOLOR	= 0.5,	0.5,	0.5,	2	Clamping device and tailstock colour
TURN_TAILSTOCKSECTIONCOLOR	= 0.5,	0.5,	0.5,	0	Tailstock section colour
TURN_WORKPIECECOLOR	= 0.5,	0.5,	0.55,	2	Workpiece colour
TURN_SECTIONCOLOR	= 0.5,	0.5,	0.55,	2	Section colour of the workpiece
[MonochromeDefinitions]					Definition of the screen colours for monochrome screen
TURN_BACKGROUND	= 0.0,	0.0,	0.0,	0	background colour
....	...				...

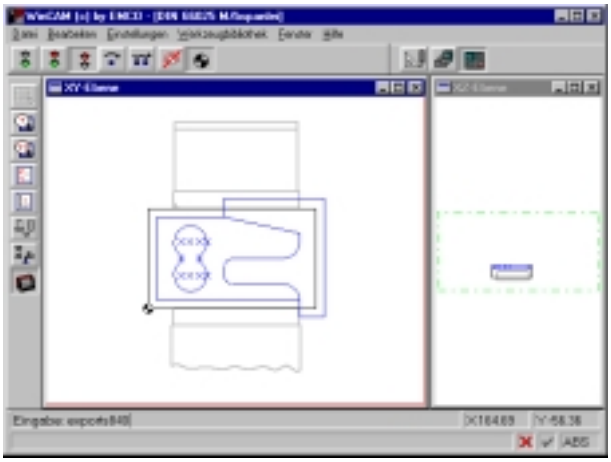
Settings for Milling

MILL_BACKGROUND	= 0.0,	0.0,	0.0,	0	background colour
MILL_CUTTERCOLOR	= 0.75,	0.5,	0.0,	2	colour of the cutting part 2D display
MILL_TAPCUTTERCOLOR	= 1.0,	0.0,	0.0,	2	colour of the tapping tool 2D display
MILL_SHAFTCOLOR	= 0.75,	0.75,	0.75,	2	colour of the tool shaft (holder) 2D display
MILL_REMOVECOLOR	= 0.5,	0.5,	0.5	2	Colour of the workpiece after chip removing
MILL_TAPREMOVECOLOR	= 1.0,	0.0,	0.0,	2	Colour of the workpiece after threading chip removing
MILL_CLAMPINGDEVICECOLOR	= 0.5,	0.5,	0.5,	2	Clamping device colour
MILL_WORKPIECECOLOR	= 0.5,	0.5,	0.55,	2	Workpiece colour
MILL_SECTIONCOLOR	= 0.51,	0.5,	0.55	2	Section colour of the workpiece
[MonochromeDefinitions]					Definition of the screen colours for monochrome screen
MILL_BACKGROUND	= 0.0,	0.0,	0.0,	0	background colour
....	...				...

Setting the colours (RGB model):

A colour is defined by four parameters:
Colour = RED GREEN BLUE SHADING

For RED, GREEN and BLUE numbers between 0 and 1 can be entered.
The input 0, 0, 0 is black, 1, 1, 1 is white.
For SHADING you can enter:
0 no shading
1 normal shading
2 enhanced shading.



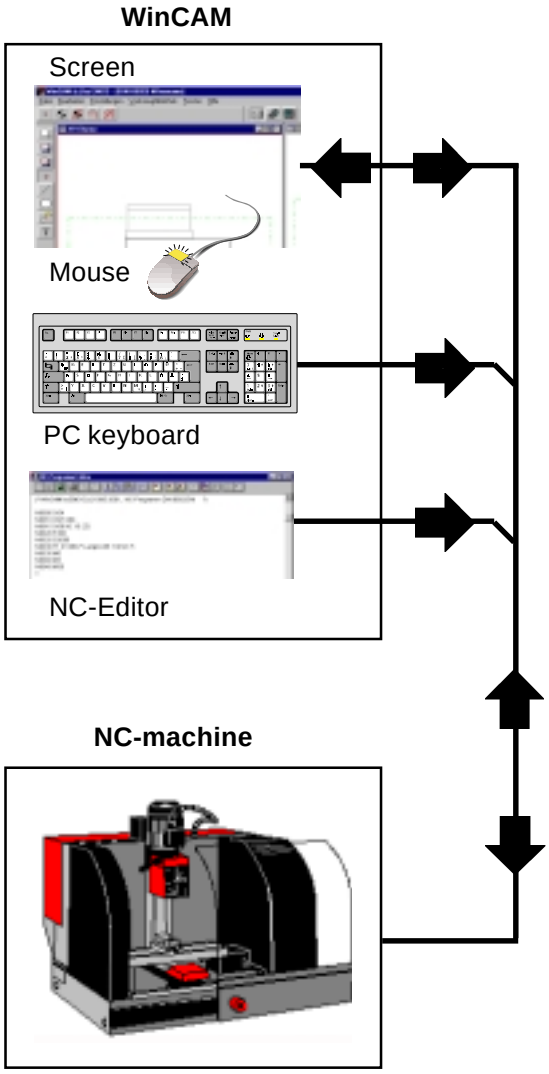
Data export with "exports840"

Program export with"exports840"

In the CAM mode it is possible to convert a program from DIN format to Sinumerik 810/840 format. By entering the keyword „exports840“ the conversion will be started and a untied project file with the suffix *.s8m will be created and stored in the working directory.

Chapter J NC bases

NC data flow



Note:
In chapter E of this software description examples for the operation of the machine are shown.

Basic knowledge

In NC mode you can control an EMCO PC Mill 50 which is connected to the PC via an interface card. In the following this NC-machine is therefore described as sample machine.
The following options of machine control via NC mode of WinCAM are possible:

Machining
You can start the NC program generated in CAM mode on the connected NC-machine and influence it during machining via command symbols and various PC keys.

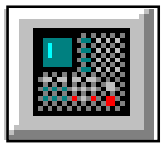
Manual operation
With the command symbols and/or various PC keys the machine can be manually traversed in all axes and also functions (door open/close, spindle on/off, etc.) can be switched.

Status messages
WinCAM inquires the actual status of the NC-machine and emits them on the screen (e.g. door open/close).

Set-up operation
Special functions enable the measurement of the clamping device and the tools.

Error messages
WinCAM and the NC-machine complement each other in the inquiry of occurring error situations. WinDAM emits the respective error messages on the screen.

Literature hint
WinCAM is designed for the use of different NC-machine types and also control types. The manual you have in your hands is a software description of WinCAM and presupposes the knowledge of the command block and the operation of NC machines as well as skills in chip removal production. If necessary, you can get further information in the literature of the NC-machine.



Shift NC Mode

By clicking the command symbol NC you can shift at any time into the NC mode.

In NC mode other command symbols are displayed which are further described in the chapter NC commands. Also observe the changes due to the NC mode described in the chapter menu lines.

With WinCAM operating posts configured with NC machine an error message is displayed during shifting in case of unconnected NC machine. Ignore this message if you want to continue work without machine. Close this error window and set it as symbol on an undisturbing position at the screen.

NC Initialization

Before you are able to work off an NC program with WinCAM the NC-machine has to be adequately prepared:

In the following a check list for initialization:

1. The NC machine must be switched on (power supply, compressed air).

2. The NC machine must be referenced.

3. The clamping device must be defined (see function "define vice position").

4. The position of the unmachined part must be defined (see function "define unmachined part/ clamping point" or function "mark NC work piece").

5. The tools used must be in the toolholder (see function "tool library") and have to be measured (see function "tool measurement").

6. The settings in the function "target machine" must correspond with the used NC-machine.

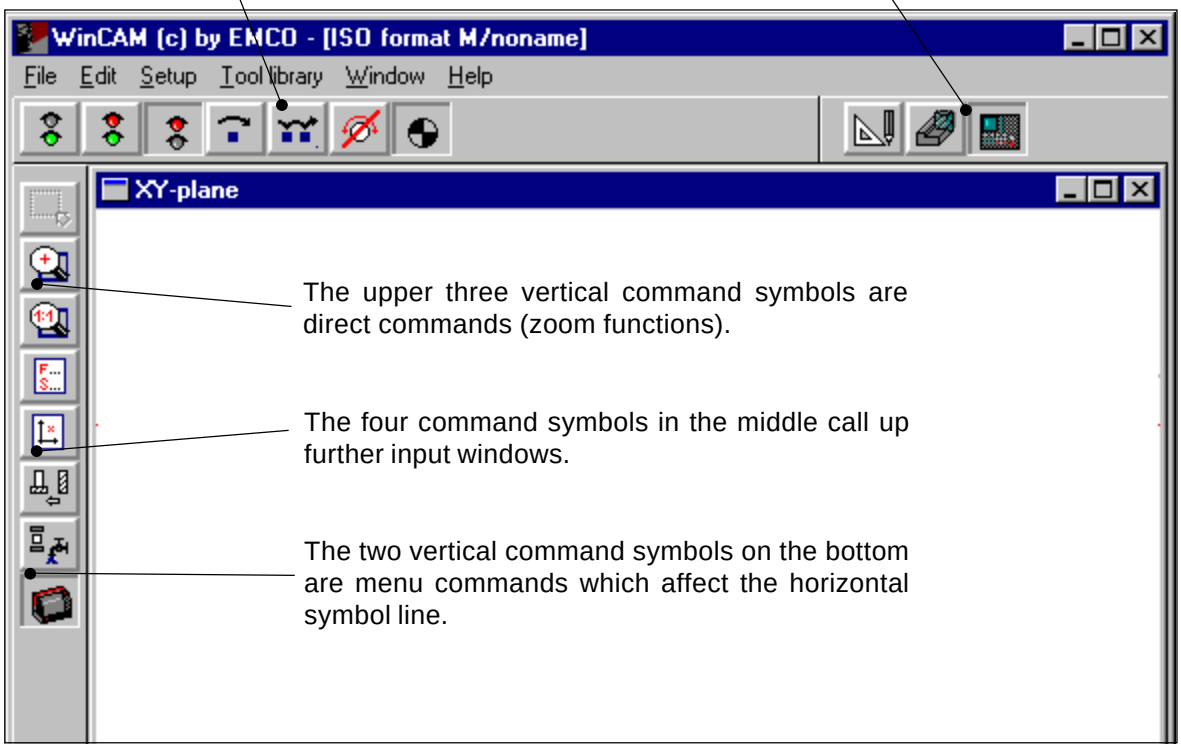
7. The NC program suitable for the target control must exist in the editor (load or generate NC program).

8. Now the machining procedure can be started.

Command symbols

In the horizontal symbol line those command symbols are displayed which can be called up by means of the two vertical menu commands.

With the 3 shift symbols you can shift from CAD into CAM and/or into the NC mode. Due to the shift all other command symbols are shifted. Here you switch on NC.



Command symbols

WinCAM window "NC-alarm"

Note:
If no NC machine is connected to the PC you can ignore the error messages displayed in NC mode. Place the WinCAM window "NC alarm" as symbol in a corner.

NC error messages

If errors have occurred at the NC-machine the WinCAM window "NC-alarm" is opened.
The NC errors can have the following causes:

- generated by the NC machine (e.g. EMERGENCY-OFF)
- generated by the NC program (e.g. wrong G-command)

The error messages are clear, thus listing them has been neglected for this reason.

Indication of the alarm lamp

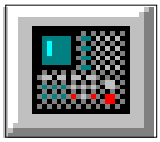
- Alarms - lamp rotates
- Messages - lamp stands still

Command areas

Reset acknowledges an NC alarm. If the alarm cause has been eliminated in the meantime the alarm window is closed. If this is not the case the window remains open.

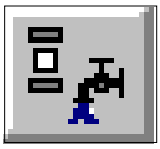
Help opens the WinCAM help for the NC alarm.

Chapter K NC Online-commands



NC mode
By clicking on the shift symbol "NC" the NC command symbols are activated. The NC mode remains active until it is deselected again with CAD or CAM.

The 2 direct commands (zoom functions) have already been explained earlier.



Periphery

The task of the periphery functions is to switch the respective part of the NC machine. The range of the indicated periphery functions depends on the installation.

Status display
By the position of the command symbol you can recognize the actual status of the periphery at the NC machine. Here some examples in this respect.

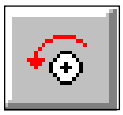
Main spindle off

Main spindle in counterclockwise direction

Main spindle in clockwise direction

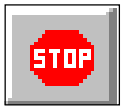
Clamping device opened
Chip blow-out off
Automatic door opened

Clamping device closed
Chip blow-out off
Automatic door closed



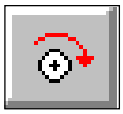
Main spindle on/left

After entering the function the main spindle is switched on in counterclockwise direction. The symbol margin reverses for status display. The speed is set via the WinCAM window "NC-parameter".



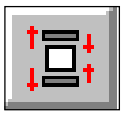
Main spindle stop

After entering the function the main spindle is switched off. The symbol margin reverses for display status display.



Main spindle on/right

Afte entering the function the main spindle is switched on in clockwise direction. The symbol margin reverses for status display. The speed is set via the WinCAM window "NC parameters".

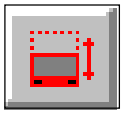


Clamping device close/open

After entering the function the clamping device is opened or closed. The symbol margin reverses for status display every time.
Mind that the clamping device can only be switched with open door.

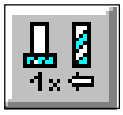
Chip blow-out on

After entering the function the chip blow-out is switched on for 3 seconds.



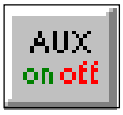
Automatic door open/close

After entering the function the automatic door is closed or opened. The symbol margin reverses for status display every time.



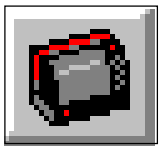
Single tool change

After actuating this function, the tool drum will be swivelled for one position.



AUX ON/OFF

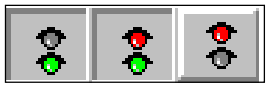
After input of the function the auxiliary drives will be switched on or off. The margin of the symbol inverts for indication.



Program execution

Status display
By means of the position of the command symbol you can recognize the actual status of the NC-machine status. See some examples in this respect:

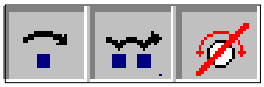
NC program runs, NC machine machines



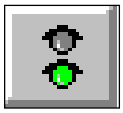
NC program runs, NC machine in intermediate stop

NC program and NC machine switched off

Skip block off
Single block operation off
No test run without main spindle

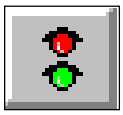


Skip block on
Single block operation on
Test run without main spindle



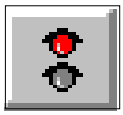
Start program (Cycle Start)

By clicking on the command symbol NC program run is started. This is the same function as "Cycle Stat" at the NC machine.



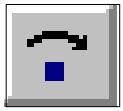
Stop program (NC-Stop)

By clicking on the command symbol the NC program run is stopped but not interrupted. A continuation of the program is possible at any time with "Start program". This is the same function as "Feedhold" and/or "Intermediate stop" at the NC machine.
In intermediate stop you can carry out manually various changes (e.g. with the periphery commands). However, mind that the original state has to be reestablished before continuation of start (e.g. M03/04).



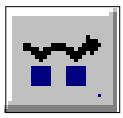
Stop program (Reset)

By clicking on the command symbol the NC program run is interrupted. A continuation of the program is not possible any more, only a new start. This is the same function as "Reset" at the NC machine.



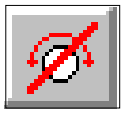
Skip block (Skip)

By clicking on the command symbol the function "skip block" is activated.
Depending on the set target control blocks can be marked as skip blocks in the editor (see manual of the NC control). These blocks are skipped during the execution of the NC program.



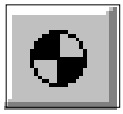
Single block operation (Single)

By clicking on the command symbol "start program" only a single NC-program block is worked off at a time. This is the same function as "Single" at the NC machine. The status "single block operation" is indicated by the reversed symbol margin.



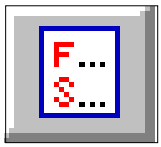
Test run without main spindle(Dryrun)

By clicking on the command symbol the status test run is set. During the start of the NC program by means of the function "start program" the main spindle is not switched on and the slides are moved with fixed feed speed. Therefore carry out the test run only without workpiece. This is the same function as "Dryrun" at the NC machine. The status "test run without main spindle" is indicated by the reversed symbol margin.



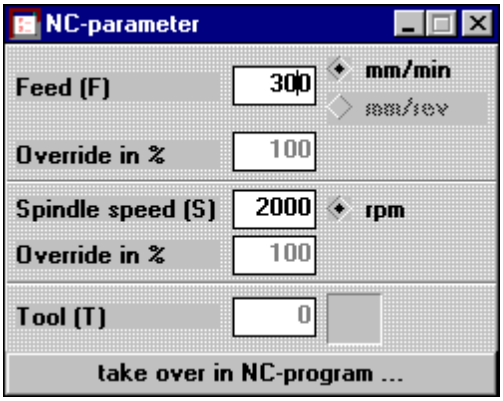
Approach reference point

By clicking the command symbol the reference point of the machine is approached. The symbol is also status indication. With approached reference point the reversed symbol margin is shown.



NC parameters on/off

By selecting the function the WinCAM window "NC-parameters" is opened.
Lead the mouse cursor into the input field "Spindle speed (S)" and enter the speed. This speed is taken over during the next input of the functions main spindle on/right or on/left. The displayed speed is always the entered one or the one set last by the NC program. This display remains unchanged also by an eventually changed "Override".

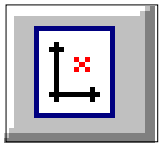


WinCAM window "NC-parameters"

Screened displays
The feed (F) cannot be entered in this mode. During the use of the PC traverse keys it is set fixed with 600 mm/min, it can, however, be changed by means of the PC-override keys. In this case the actual feed value is displayed.

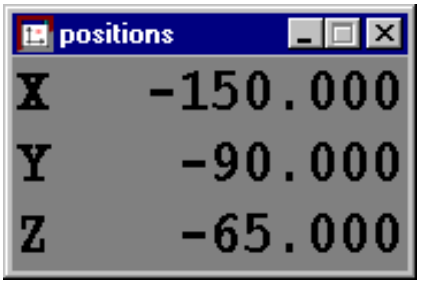
If a tool (T) has already been selected it is displayed in screened mode. In addition to the actual tool number the pictograph is shown.
The line "take over in NC-programm" is without function.

The window is closed again by another click on the command symbol "NC-parameter on/off".



Positions display on/off

By selecting the function the WinCAM window "positions" is opened.

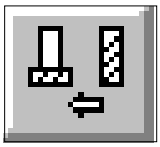


WinCAM window "positions"

In this window the actual NC coordinates X,Y,Z (position of the tool cutting edge in relation to the workpiece zero point) are displayed.

You are able to move the window at any time to an undisturbing screen position as well as to change the window size.

The window is closed again by another click on the command symbol "Positions display on/off".

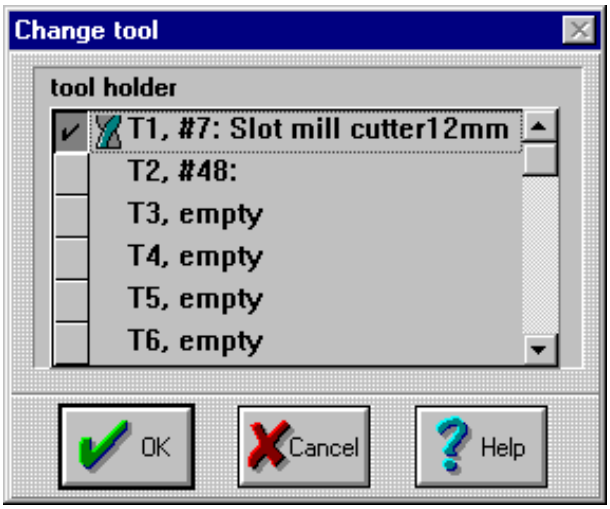


Tool change

By selecting the function the WinCAM window "change tool" is opened.

The presently actual tool is marked with a hook.

Here you can select one of the tools in the toolholder.
Click with the mouse in the field beside the respective tool and confirm with O.K.



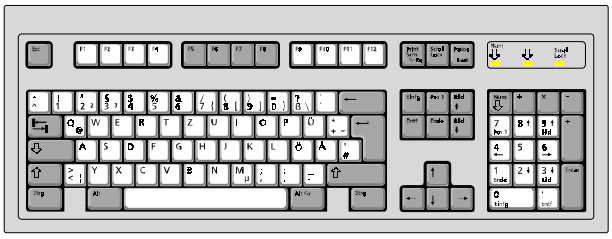
Change tool

If the connected NC machine has an automatic tool changing device the selected tool is changed. If the NC machine has no tool changing device (such as e.g. the PC Mill 50) a main spindle hold (M0) is emitted and the tool has to be changed manually. In both cases the actual tool data are transferred to the control.

You can move the window at any time to an undisturbing screen position.

By means of the field "Help" you can have the help function displayed.

By a click on the field "Cancel" by means of the mouse cursor the window is closed without tool change.



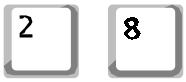
NC-keyboard assignment

In NC mode of WinCAM it is possible to control the NC machine by means of the PC keyboard. By pressing the keys or key combinations mentioned below mentioned of the numerical block the following control is effected:

Traversing the slides
Feed fixed with 600 mm/min with 100% override.
X-axis
key (4) traverse in -X-direction
key (6) traverse in +X-direction

Y-axis
key (1) traverse in -Y-direction
key (9) traverse in +Y-direction

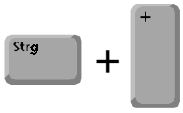
Z-axis
key(2) traverse in -Z-direction
key (8) traverse in -Z-direction



Setting speed and feed
Changes the feed starting from programmed or fixed value.
Key (+) - increase feed override

Key (-) - reduce feed override

Changes the speed of the main spindle starting from programmed or entered value.



Key (Ctrl)+(+) - increase speed override

Key (Ctrl)+(-) - reduce speed override

Program keys

Key "Enter" - start NC program (Cycle Start)

Key "Delete" - stop program (NC-Stop)

Key "Insert" - stop program (Reset) and error reset

Capter L Software installation

System Requirements

For running WinNC the following minimum configuration is required:

- PC Celeron or Pentium (II or III)
  400 MHz IBM compatible
- 64 MB RAM (for a 3DView- installation is a minimum of 32MB necessary)
- For the graphic- card is a minimum of 2 MB necessary (depth of shade minimum 16Bit)
- 10 MB free hard disk space for every installed control type
- WINDOWS 95/98 or ggf. Windows NT

Software Installation

- Start Windows 95 / Windows 98
- Machine installations under WinNT are not possible
- Insert installation disk 1 in drive A.
- Click on START (bottom left), select RUN.
- Enter: A:\SETUP and click on OK.
- The installation program will be started
- The installation is menu driven. Study thoroughly the several points.

Notes for Network Installation

Notes:
Network installation is possible only with the multiple licence version of the software.
The network installation must be done by the networks administrator (Supervisor) only.

WinNC is installed once on the server.
The users have their own (private) directories for workpiece programs and setting data.
The directories for workpiece programs and setting data must be set for every single user.

First install WinNC on the server and then on all working stations.
In the installation program select "network installation" and indicate whether installing on the server or on a working station.

1. Way

Indication of a local directory for the local data of every working station.
(e.g.: C:\WINNC)

2. Way

This is the most flexible way. The local data of every working station will also be stored at the server.

- Set up directory for user:
  If this is not yet done (as usual in most networks), for every user a directory must be set up in which only the user has acces.
  e.g.: SYS\USERS\USER1
  SYS\USERS\USER2 (Novell)

- In the system loginscript (for Novell) a free drive letter must be mapped on the directory.
  e.g.: letter H is free:
  map ROOT H:=SYS\USERS\%LOGIN_NAME
  (for Novell nets)

- On that directory in which WinNC is installed at the network server, every user must have execution rights.

- WinNC can be installed at the server now. As directory for local net working station the corresponding user-specific drive letter must be entered
  (e.g.: H:\WINNC).

- With the now following installation on the working stations the directory must be entered in which WinNC is installed at the server.

Settings of the Interface Board

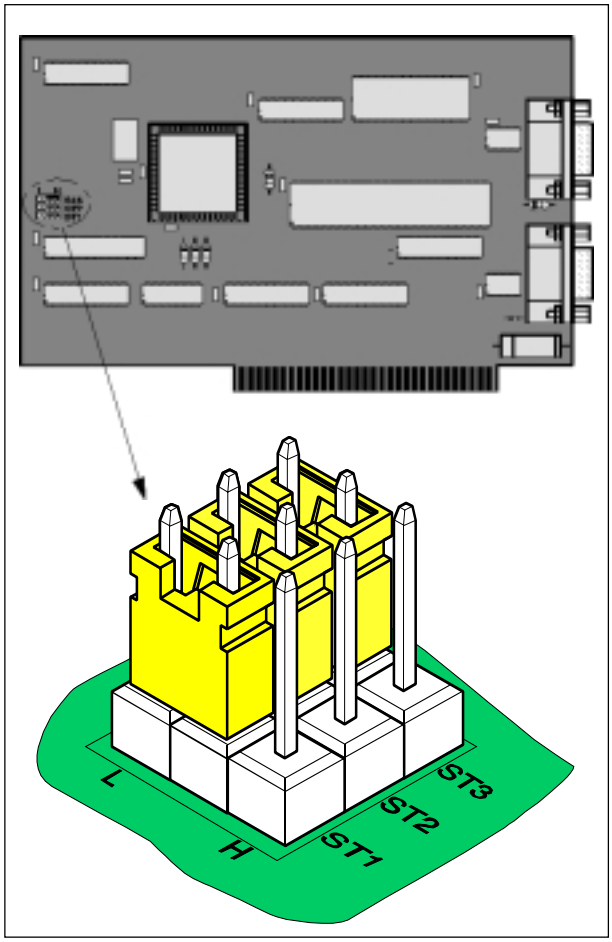
RS 485 Board

(PC TURN 50, PC MILL 50,
PC TURN 120, PC MILL 100)

With installation of the software a certain memory area (**memory area CC000 - D0000**) is assigned to the interface card.

If this area is already occupied, e.g. by another card or an Expanded Memory Manager, an alarm appears.

After this alarm act as following:



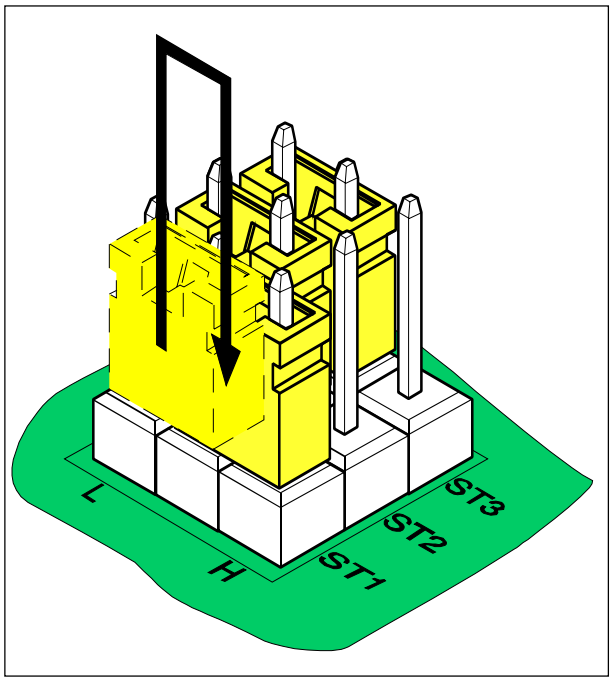
Jumper position on the interface board

Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.
The following memory areas are available:

RS 485						
Nr.	Jumper ST1	Jumper ST2	Jumper ST3	Hexadecimal Memory Area		
1*	L	L	L	CC000	to	CC7FF
2	H	L	L	CC800	to	CCFFF
3	L	H	L	CD000	to	CD7FF
4	H	H	L	CD800	to	CDFFF
5	L	L	H	CE000	to	CE7FF
6	H	L	H	CE800	to	CEFFF
7	L	H	H	CF000	to	CF7FF
8	H	H	H	CF800	to	CFFFF

*) Basic position



Alter jumper positions

Sequence:

Danger

Mount and dismount the interface card only while the PC is disconnected to the net.
Pull power cable!

- Remove the interface board from the PC.
- Place the jumpers ST1 - ST3 in the required position (positions 1 - 8 see table on previous page).
- Mount the interface board in the PC.
- Connet the PC to line and switch on.
- Retry installation of the software.
  When the alarm occurs again, try the next jumper position for installation.

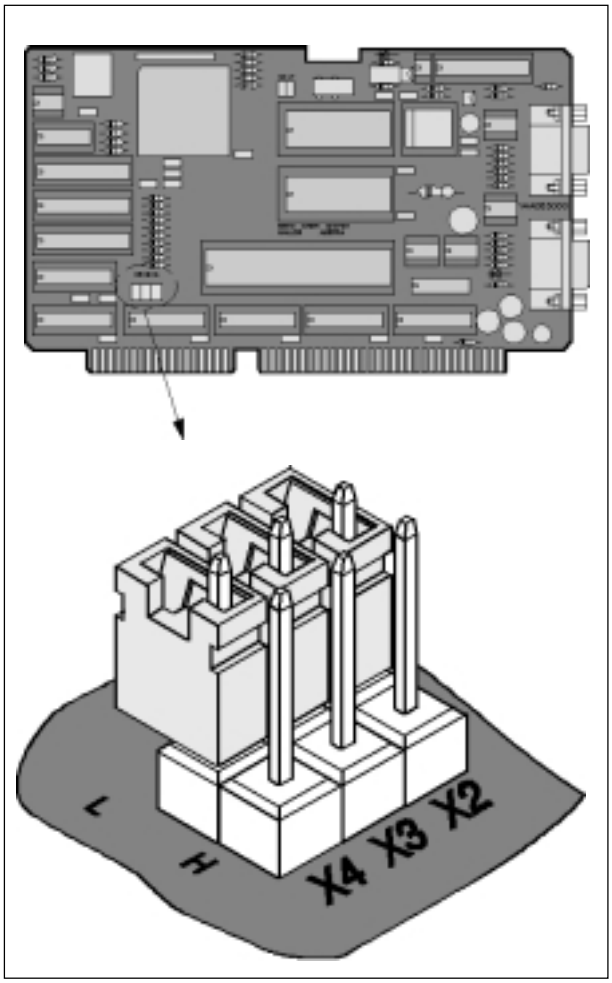
PCCOM (RS 422) Board

(PC TURN 55, PC MILL 55
PC TURN 125, PC MILL 125
PC TURN 155, PC MILL 155)

With installation of the software a certain memory area (**memory area CC000 - CFFFF**) is assigned to the interface card.

If this area is already occupied, e.g. by another card or an Expanded Memory Manager, an alarm appears.

After this alarm act as following:



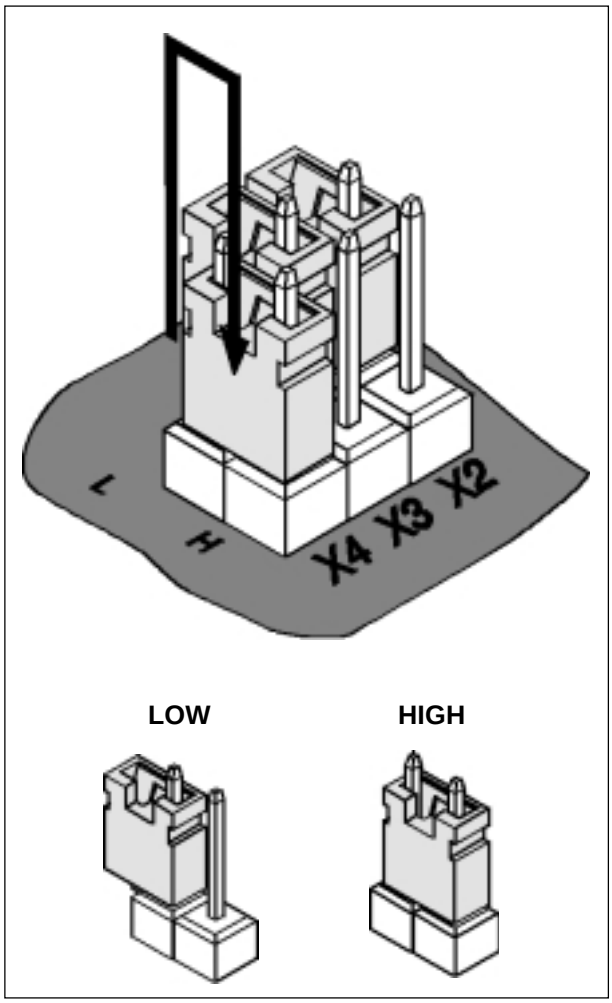
Jumper position on the interface board

Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.
The following memory areas are available:

RS 422						
Nr.	Jumper			Hexadecimal Memory Area		
	X2	X3	X4			
1*	L	L	L	CC000	to	CC7FF
2[1)]	L	L	H	D8000	to	D87FF
3	L	H	L	CF800	to	CFFFF
4[1)]	L	H	H	E0000	to	E07FF
5	H	L	L	CE000	to	CE7FF
6[1)]	H	L	H	DF800	to	DFFFF
7[1)]	H	H	L	D0000	to	D07FF
8[1)]	H	H	H	E8000	to	E87FF

*) Basic position
1) From PCCOM- Board- version 1



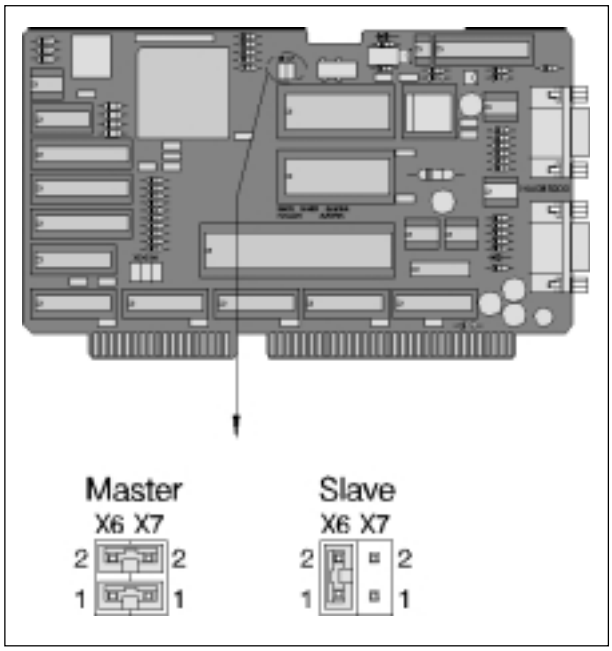
Alter jumper positions

Sequence:

Danger

Mount and dismount the interface card only while the PC is disconnected to the net.
Pull power cable!

- Remove the interface board from the PC.
- Put the jumpers X2 - X4 in the requierd position (positions 1 - 8 see table on previous page).
  Pins not connected: L
  Pins connected: H
- Mount the interface board in the PC.
- Connet the PC to line and switch on.
- Retry installation of the software.
  When the alarm occurs again, try the next jumper position for installation.



PCCOM setting Master - Slave

PCCOM Master-Slave Setting

Several PCCOM boards can be installed in the PC, e.g. to control more than four axes on a machine.

In this case one of the boards must be set as Master, all other boards must be set as Slaves.

When only one board is installed it also must be set as master.

At delivery all cards are set as Master.

Setting occurs with the jumpers X6 and X7.

Network card

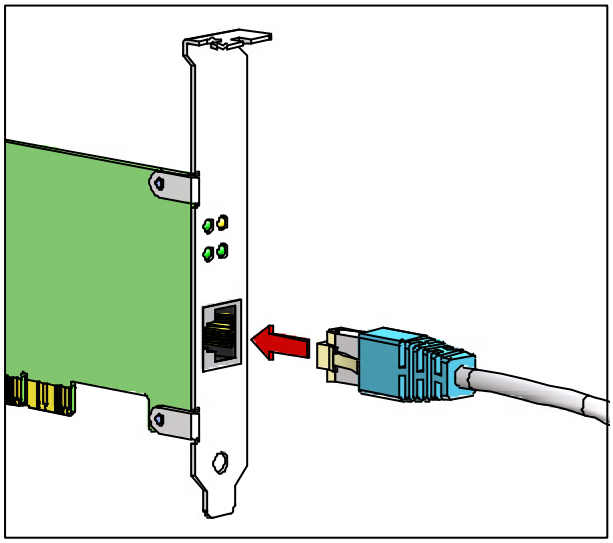
for:
Concept Turn 55
Concept Mill 55
Concept Turn 105
Concept Mill 105

Note:
With a machine installation one network card must be reserved exclusively for the control of the machine.

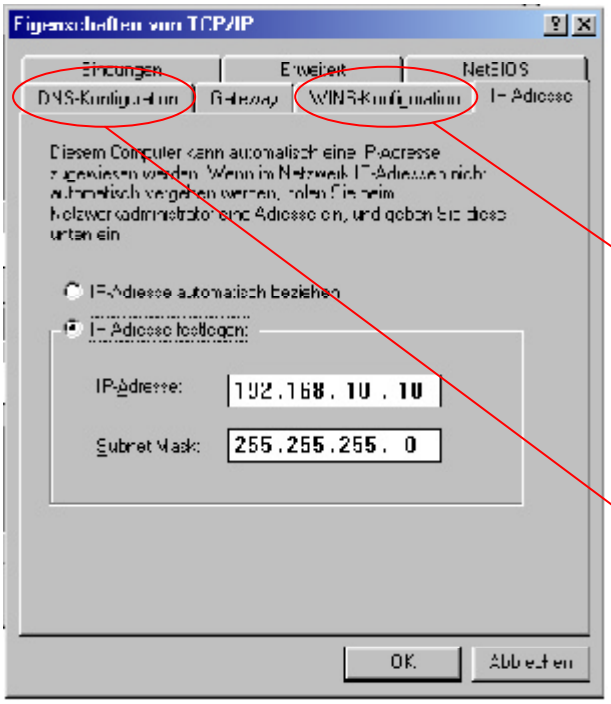
Network card type: TCP/iP enabled network card

Setting of the network card for the local connection to the machine:

IP- Adress: 192.168.10.10
Subnetmask 255.255.255.0

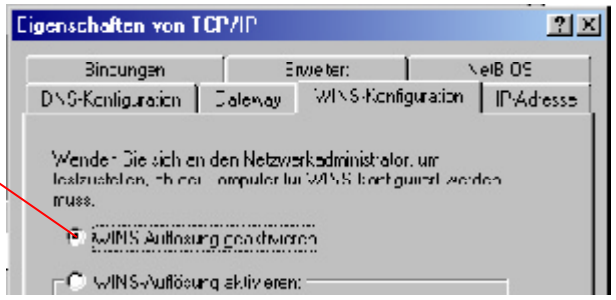


Connection of the machine to the PC



Properties of TCP/IP

Furthermore, in the registers "DNS configuration" and "WINS configuration" are to be disabled.



Disable WINS configuration

Disable WINS configuration