

Table of Contents

INTRODUCTION.....	6
ABOUT EZ-DNC / FILTERMAX HELP.....	6
ABOUT PCAM SOLUTIONS INC.	6
INSTALLATION.....	6
SYSTEM REQUIREMENTS.....	6
INSTALLING THE SOFTWARE.....	7
CONNECTING THE KEY RING.....	7
GETTING STARTED	7
OVERVIEW.....	7
CONFIGURING CONTROL PARAMETERS.....	7
CONVERTING NC FILES.....	8
PREVIEWING NC FILES.....	8
SAVING NC FILES.....	8
SAVING YOUR SETTINGS.....	9
SETTING OPTIMIZATION PARAMETERS.....	9
MENU COMMANDS	9
FILE MENU COMMANDS.....	9
EDIT MENU COMMANDS.....	10
FORMAT MENU COMMANDS.....	11
OPTIMIZE MENU COMMANDS.....	11
PREVIEW MENU COMMANDS.....	12
DNC MENU COMMANDS.....	12
AUTOMATION MENU COMMANDS.....	12
WINDOW MENU COMMANDS.....	13
TECH NOTES	13
OVERVIEW TECHNICAL NOTES.....	13
BRIDGEPORT CONTROLS.....	14
DIGITIZED XYZ FILES.....	14
G90 AND G91 COMMANDS.....	14
LANGUAGE ACCEPTANCE.....	14
OFFSET COMMANDS.....	14
OPTIMIZATION FAILURE.....	14
PLANE RESTRICTIONS.....	14
APPENDIXES	15
ASCII CHARACTER CODES DECIMAL SYMBOL KEYSTROKE.....	15
SERIAL PORT DIAGRAMS.....	17
CONTACT SUPPORT	18
CONTENTS.....	19
NEW IN RELEASE 6.....	19
THE MENU COMMANDS.....	19
OVERVIEW OF THE MENU COMMANDS.....	19
RS232 SETTINGS.....	20
DNC SETTINGS.....	20
DNC RESTART SETTINGS.....	21
DNC RESTART FROM CURSOR	22
WAIT FOR XON	23
SHOW IN EDITOR.....	23

SEND TEXT	23
RECEIVE TEXT	24
BAUD RATE	24
DATA BITS.....	24
STOP BITS	24
PARITY.....	24
FLOW CONTROL.....	25
NONE	25
XON/XOFF	25
RTS	25
XON/RTS	25
COMMUNICATIONS PORT	25
PROGRAM START (SEND).....	25
PROGRAM END (SEND).....	26
BLOCK END	26
SKIP LINE PREFIX (SEND).....	26
SEND BEFORE RECEIVE	26
PROGRAM START (RECEIVE).....	26
SKIP LINE PREFIX (RECEIVE)	26
SEND SPECIAL.....	26
UNDO.....	27
CUT	27
COPY.....	27
PASTE.....	27
CLEAR	27
SELECT ALL	27
SEARCH.....	27
SEARCH AND REPLACE	28
CONTINUE SEARCH	28
COMPARE TO FILE.....	28
GOTO COLOR.....	29
RENUMBER SETTINGS.....	29
RENUMBER.....	29
NEW	29
OPEN	30
CLOSE	30
SEEK.....	30
SEEK : THE FILE SELECTION WINDOW.....	31
FILELIST	31
HEADER TEXTBOX.....	31
RESET BUTTON	31
OPEN BUTTON.....	32
INSERT	32
DELETE.....	32
SAVE	32
SAVE AS.....	32
SAVE AS DXF.....	32
LOAD WORDS.....	33
LOAD SETTINGS.....	33
SAVE SETTINGS.....	33
PRINTER SETUP	33
PRINT	33
LAST FILES LIST	33
EXIT	33
DIALOG BOX	34
TOOLBAR.....	34

FILE FILTER.....	34
FIRST LINE.....	34
SECOND LINE.....	34
ANY LINE.....	34
LIST OF DRIVES.....	34
LIST OF DIRECTORIES.....	34
FIND.....	34
LIST OF FILES.....	35
EXIT BUTTON.....	35
FONT.....	35
MARGINS.....	35
SINGLE SPACE.....	36
GRAY BACKGROUND.....	36
AUTOMATIC LINEBREAK.....	36
SHOW CR/LF.....	36
UPPERCASE.....	37
UNITS (INCH/METRIC).....	37
WORD LIST.....	37
USE RECYCLE BIN.....	37
GETTING STARTED.....	37
OVERVIEW INSTALLATION INFORMATION.....	38
SETTINGS... ..	38
CONVERT.....	39
CONVERT TO DIRECTORY.....	39
TYPE.....	39
CHORDAL TOLERANCE.....	39
TANGENCY TOLERANCE.....	39
DECIMAL DIGITS.....	40
COMMENT PREFIX.....	40
SINGLE QUADRANT.....	40
CORNER OPTIMIZATION.....	40
CORNER ANGLE.....	40
CORNER MAX. / MIN. RADIUS.....	40
CORNER MAX. / MIN. FEED.....	41
CORNER DECELERATION.....	41
CORNER TOLERANCE.....	41
REGISTER SETTINGS.....	41
PREVIEW / X-AXIS.....	42
PREVIEW / Y-AXIS.....	42
PREVIEW / Z-AXIS.....	42
PREVIEW / X-INCREMENT.....	42
PREVIEW / Y-INCREMENT.....	42
PREVIEW / Z-INCREMENT.....	43
PREVIEW / X,Y,Z-ARC.....	43
PREVIEW / R-ARC.....	43
PREVIEW / FEED.....	43
PREVIEW / DRILL DEPTH.....	43
PREVIEW / RETRACT PLN.....	43
PREVIEW / CUT-IN.....	43
PREVIEW / FEED-DRILL.....	44
PREVIEW / DWELL.....	44
PREVIEW / RPM.....	44
PREVIEW / PITCH.....	44
G-CODE SETTINGS... ..	44
PREVIEW / DRILLING.....	45
PREVIEW / SPOTFACING.....	45

PREVIEW / CHIPBREAK	45
PREVIEW / DEEPHOLE.....	45
PREVIEW / REAMING.....	45
PREVIEW / BORING (ORTG).....	46
PREVIEW / BORING (STOP).....	46
PREVIEW / TAPPING.....	46
PREVIEW / REV.TAPPING.....	46
PREVIEW / TURN-THREAD.....	47
PREVIEW / CONST -SURF.....	47
PREVIEW / CONST -SPIND.....	47
PREVIEW / UNITS/REV.	47
PREVIEW / UNITS/MIN.....	47
PREVIEW / MAX.RPM.....	47
PREVIEW / SKIP LINE 1-3.....	47
OTHER SETTINGS.....	47
PREVIEW / CONTROL TYPE.....	48
PREVIEW / PROGRAM ADDRESS	48
PREVIEW / MACRO CALL.....	48
PREVIEW / MACRO ADDRESS.....	49
PREVIEW / MACRO START	49
PREVIEW / MACRO END.....	49
PREVIEW / MACRO REPEAT	49
PREVIEW / TOOL ADDRESS	49
PREVIEW / TOOL CHANGE.....	49
PREVIEW / CHANGE TIME.....	50
PREVIEW / RAPID.....	50
PREVIEW / ARC CENTER DEFINITION	50
PREVIEW / ORIENTATION.....	50
PREVIEW / STOP	50
PREVIEW / REVERSING.....	50
PREVIEW / SPECIAL FUNC.....	50
PREVIEW / FUNCTIME.....	51
PREVIEW / RETRACTION.....	51
PREVIEW / START DRILL NEXT BLOCK.....	51
PREVIEW / IGNORE Z.....	51
PREVIEW / USE PITCH.....	51
PREVIEW / REVERSE ARCS IN G17/18/19	51
PREVIEW.....	51
RUN SINGLE STEP.....	52
STOP	52
RUN CONTINUOUS.....	52
REWIND AND CLEAR.....	52
VIEW SETTINGS.....	53
ACTIVATE TEXT EDITOR.....	53
TOOL.....	53
TM (TIME).....	53
DISPLAY AREA.....	53
PREVIEW SHORTCUTS.....	53
PREVIEW / WINDOW CENTERS	54
PREVIEW / STEP SIZE	54
PREVIEW / A UTO REDRAW.....	54
PREVIEW / PRINT	54
PREVIEW / SHOWPATH.....	55
PREVIEW / SHOWPOINTS.....	55
TOOLPATH PREVIEW.....	56
OVERVIEW OF TOOLPATH PREVIEW	56

ZOOMING IN	57
CHANGING THE WINDOW CENTER.....	57
ZOOMING IN BY TWO.....	57
ZOOMING OUT BY TWO.....	57
DEFAULT DISPLAY AREA SETTINGS.....	57
TOOLBAR BUTTONS.....	57
OVERVIEW OF TOOLBAR BUTTONS.....	58
CASCADE	58
HORIZONTAL.....	59
VERTICAL.....	59
DISPLAY WINDOWS / ACTUAL FILENAMES.....	60
NEW WINDOW.....	60
ARRANGE ICONS.....	60
MACRO EDITOR.....	60
MACRO DIRECTORY.....	61
UPDATE MENU.....	62
MACRO LIST	62

Introduction

About EZ-DNC / FilterMAX Help

This help system covers two software products named as EZ-DNC and FilterMAX. Both programs share the same menu and command structure with the exception of the additional Optimize menu of FilterMAX which offers G-Code optimization by fitting arc/nurbs to linear moves.

EZ-DNC

EZ-DNC provides a single, complete environment for creating, editing, verifying and DNC transfer of CNC programs. The editor supports most advanced word processor features, special features include block renumbering, seeking by file contents and file comparison. You can communicate with up to 64 CNCs simultaneously using the robust DNC feature. The backplotting window lets you plot the current g-code file one move at a time or all at once. A very precise machining time estimator is located in the backplotting window. NC toolpaths can also be converted to DXF format using the Convert to DXF feature.

FilterMAX

In addition to the complete functionality of EZ-DNC, FilterMAX converts point-to-point G-code toolpaths to tangent arc moves or NURBS, resulting in files that are up to 98% shorter. Instead of cutting thousands of linear segments, the machine will cut tangent arcs, giving smoother surface finishes. Small linear moves can slow down a CNC Control's processing time significantly, arc moves generated by FilterMAX eliminate this delay allowing the machine to run much faster. Feedrates can also be optimized for sharp corner deceleration.

About PCAM Solutions Inc.

We are the developer of EZ-CAM family of CAD/CAM software systems originally created by Bridgeport Machines in 1980. Since then over 10,000 users worldwide have benefited from EZ-CAM's powerful, robust and user-friendly features. Today, EZ-CAM under PCAM's management continues to lead the way with innovations in CNC programming. PCAM's team includes both software and machining specialists. This mix of expertise, combined with a close collaboration with a variety of machine shops, continues creating the optimal tools for making your shop more efficient.

Software Development, Worldwide Sales & Marketing

Contact : PCAM Solutions Inc., USA, Sevket Tarhan
Email : sevket@ezcam.com
Phone : 212-249 5832
Web : www.ezcam.com

Installation

System Requirements

This software is designed to be compatible with Microsoft Windows 32Bit versions like Windows 95 / 98 or NT4.

Operating System	: Microsoft 32 Bit operating system (Windows 95 / 98 , NT4.0, Windows 2000)
Processor	: Pentium class
RAM	: min. 32 MB
Graphic Adapter	: VGA / SVGA support
Diskspace	: about 12MB
Others	: parallel printer port for connecting Key-Ring

Installing the Software

One of the Microsoft Windows 32Bit operating systems (Windows 95 / 98 , NT4 / 2000) must be installed prior to software installation. The installation procedure will automatically create a directory, copy all the program files, and create a program group and icon.

1. Insert the program CD in the CD-drive.
2. Select the RUN command from the START Menu of Windows Desktop.
3. Type X:\SETUP where X is the letter of your CD drive.
4. Follow the instructions of the setup routine.

Important !

It is **not** necessary to have the Key-Ring installed prior or during installation. Only when starting the software, the presence of the Key-Ring is checked.

Connecting the Key Ring

Packaged with the software is a protection device known as a "key". This "key" plugs into the parallel printer port at the back of the computer. The key is programmed by the software manufacturer to protect the software purchased with your package.

Getting Started

Overview

This NC program optimizer utility is designed to address the needs of the advanced NC programmer who regularly generates complex 3D tool paths. The primary purpose of the software is to reduce the overall program length of 3D surfacing G-Code programs by, wherever possible, replacing chordal linear moves with arc moves using the standard coordinate plane selections (i.e. G17, G18 and G19). In doing so, a typical surfacing program can be reduced in length by as much as 90 percent while improving surface finish and smoothness as a result of the arc fitting. In addition to the arc fitting optimization, this software also includes an integrated NC text editor, a tool path verification mode, and CNC communications utilities, allowing a program to be edited, verified and downloaded to the CNC control via an RS232 connection.

The optimization process will yield the best results when used on surfacing programs that are cut parallel to either the X-Y, X-Z or Y-Z planes. When creating an NC program on your CAM system, try to use a machining strategy that will ensure this type of cutting. The optimized program will consist primarily of arc moves (G2/G3) in either the X-Y (G17), X-Z (G18), Y-Z (G19) planes with the remaining moves being linear (G1). This software will also fit arcs in programs that use other types of machining strategies, however the file size reduction will be much less.

When optimizing an NC program that already contains arc moves, the software requires that the appropriate plane selection code (G17, G18, or G19) precede each section that contains the arc moves. Additionally, any program that contains incremental and absolute commands (G90 and G91) must have them on separate program lines that do not contain any X-Y-Z moves.

Configuring Control Parameters

This software is designed to be compatible with Fanuc controls and requires no modifications to begin optimizing and viewing files of this format. NC files that are formatted for other machine controls may also be optimized or verified, although certain information must be defined first. To configure the parameters for controls other than Fanuc, you need to know the following information:

Program word letter addresses.

Arc center format.

G-Codes.

M-Codes.

Macro definitions.

Once the above information has been obtained, the appropriate information must be entered into the [Register Settings](#), [G-Code Settings](#) and [Other Settings](#) dialog boxes. These dialog boxes are located in the Preview menu.

Converting NC Files

Note

Do not use the Open command (located in the File menu) unless you wish to edit the original file before conversion. Loading the file before converting will take unnecessary time.

To convert NC files

1. Click on Settings under the Optimize menu. Set the chordal and tangency tolerances to the values you require. Tolerances may also be set within the program by inserting \$CT.001 (for .001 tolerance) and \$TT1 (for 1 degree tangency) in a comment line. Use this feature to change the tolerance for roughing and finishing passes within the same program. If you require a radius designation of R to be output in the converted file, check the Radius Output box, otherwise I, J, K center values will be output. Click OK.
2. Optimized files are renumbered by the settings under Renumber. If you do not want line numbers in the new file, then enter 0 for both Start Number and Step Size. The default increment is 1. If your program is metric then change the Inch/Metric setting under Format.
3. Click on Convert under the Optimize menu and specify the NC file to be converted. Click OK. The message "Converting Lines to Arcs" will appear. When completed, the original number of entities is shown with the new number, along with the percentage by which the file has been reduced. Click OK. The converted file will now appear in the editor. You may use Preview to view the toolpath.

Previewing NC Files

The Preview command is used to graphically simulate the machine tool's execution of the NC program. An active window must be displayed before the Preview command can be used. Since the Convert command automatically loads the file into an active window, the Preview command can be used immediately after converting a file.

To begin previewing a file, choose the Preview command from the Preview menu. A new window will appear placed on top of the existing window. The Preview window contains six buttons along the top of the window. Four of these buttons function similar to a VCR, allowing a file to be played in single steps, played continuously, stopped, or rewound to the beginning of the file. There is also a button used to adjust the viewing parameters such as the window center coordinates, the window size, or the 3D view orientation. This button appears as an X-Y axis.

The simulation is started by choosing the button that has two arrows pointed toward the right of the screen (Run Continuous). Assuming that the view settings are adjusted correctly, the program tool path will display in the Preview window. The verification may be stopped at any time by selecting the button that contains a single square (Stop). To replay the simulation choose the button that contains two arrows pointed toward the left of the screen (Rewind and Clear) to rewind the file. To verify the program in a single step mode, choose the button that contains a single arrow pointed toward the right of the screen (Run Single Step).

See also

[Preview...](#)

Saving NC Files

Once a file has been converted and previewed, the last step is to save the optimized file. To do this, choose the Save As command (located in the File menu). A dialog box will appear allowing a new name to be specified. If desired, the original file may be overwritten by choosing the Save command.

Saving Your Settings

Once the Preview and Optimize settings have been made, these parameter settings should be saved, allowing them to be reloaded at a later time. It is recommended that a settings file for each control be created. To save the settings, choose the **Save Settings** command from the File menu.

Setting Optimization Parameters

When optimizing a program, there are two settings that will affect the amount of file reduction and the smoothness of the machined surface. These two settings are **Chordal Tolerance** and **Tangency Tolerance**. If radius arc (R) format is required instead of arc center (I,J,K), choose the Radius Output check box. The following recommends tolerance settings for both roughing and finishing operations.

Roughing Passes

The larger **chordal tolerance** you can give, the more the file can be reduced. If you are leaving .020 of material, then a .010 tolerance will give you a good file reduction and still leave at least .010 of material for the finish pass to remove. **Tangency tolerance** should be set to 10 degrees to ensure maximum reduction.

Finishing Passes

To get a great finish and reduce hand polishing time, set the "chordal" or "surface" tolerance in your CAM system to ".0003". For this software, **chordal tolerance** should be set to ".0005". You may use smaller tolerances, but be sure that the tolerance for this software is larger than that of your CAM system. As long as your CAM output is controlled by a chordal tolerance, the output of this software will be closer to the original surface than the straight-line toolpath it takes in. In the above example, the maximum deviation will be .0005, *not* .0008. Larger finish tolerances have been used successfully, but finish may begin to be affected in some cases. **Tangency tolerance** should be set to 2 degrees to start. A lower tangency tolerance may slightly improve the part finish, but may also inhibit file reduction. If you can use a larger tolerance, the file reduction will be better. It is recommended that you raise or lower the tolerance one degree at a time and note the change in file reduction if finish is very important.

Menu Commands

File Menu Commands

The File menu is comprised of a variety of commands which are primarily used to load and save files. Most of the commands within the File menu utilize standard Microsoft Windows dialog boxes. Refer to the Microsoft Windows users manual for more information concerning the loading and saving of files using dialog boxes. A list of the four most recently used files is displayed in this menu. Click on the desired file to open. This feature is particularly useful when you need to access files quickly.

For more information click on the graphic below

File	
<u>N</u> ew	Ctrl+N
<u>O</u> pen...	Ctrl+O
<u>C</u> lose	
<u>S</u> ave	
Save <u>A</u> s...	Ctrl+S
Save As DXE...	
<u>S</u> eek...	
<u>I</u> nsert...	Ctrl+I
<u>D</u> elete...	
Load <u>W</u> ords...	
<u>L</u> oad Settings...	
Save <u>S</u> ettings...	
<u>P</u> rinter Setup...	
<u>P</u> rint	
<u>1</u> pocket.FMX	
<u>2</u> pocket.txt	
<u>3</u> MILL2.TXT	
<u>4</u> MILL.TXT	
<u>E</u> xit	Ctrl+E

Edit Menu Commands

This software contains a full screen editor and a clipboard (a temporary memory buffer) allowing program text to be copied and reinserted into the active window. The Edit menu contains most of the commands that are used when editing an NC file. Many of the edit commands require that a text selection be made before any commands can be chosen. A mouse or the cursor keys may be used to select text and to place the insertion point.

To select text using a mouse, place the cursor next to the characters to be edited, click the mouse once and continue to hold the mouse button down while moving the mouse to the left or right. When the desired characters are highlighted, release the mouse button to complete the selection. A single word that is separated by spaces, tabs or carriage returns can be selected by double clicking on the desired word. To select text using the keyboard, use the cursor keys to move the insertion point next to the characters to be selected, then while holding the SHIFT key down, move the cursor keys to the left or right to complete the selection.

For more information click on the graphic below

Edit	
<u>U</u> ndo	Ctrl+Z
C <u>u</u> t	Ctrl+X
<u>C</u> opy	Ctrl+C
<u>P</u> aste	Ctrl+V
<u>C</u> lear	Ctrl+D
S <u>e</u> lect <u>A</u> ll	Ctrl+A
<u>S</u> earch...	F2
S <u>e</u> arch & <u>R</u> eplace...	
<u>C</u> ontinue Search	F3
Compare to <u>F</u> ile...	
<u>G</u> oto Color	F4
R <u>e</u> number Settings...	
R <u>e</u> number	F5

Format Menu Commands

The Format menu contains commands that control the format and display of the NC text file. Some menu function as toggles where they are either turned on or off as indicated by the check mark displayed next to the menu.

For more information click on the graphic below

Format	
<u>F</u> ont...	
<u>M</u> argins...	
<input checked="" type="checkbox"/> Single space	
1.5 spacing	
Double space	
<u>G</u> ray Background	
<u>A</u> utomatic Linebreak	
<u>S</u> how CR/LF	
<input checked="" type="checkbox"/> <u>U</u> ppercase	
Units - <u>M</u> etric	
<u>W</u> ordList	Ctrl+W
<input checked="" type="checkbox"/> <u>U</u> se Recycle Bin	

Optimize Menu Commands

The Optimize menu contains the parameters and commands used to control and initiate the optimization process. Before optimizing a file, make sure that all of the settings in the Preview menu accurately reflect the control format of the file that is to be converted.

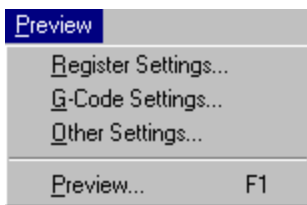
For more information click on the graphic below



Preview Menu Commands

The Preview menu contains the settings used to define the CNC control format and the commands needed to graphically execute a part program file, simulating the machining process. Before the preview mode can be used to verify a program, the control parameters must be set through the Register Settings, G-Code Settings, and Other Settings dialog boxes. Once these parameters have been established, they should be saved using the Save Settings command located in the File menu. Typically, each control type requires its own unique settings, therefore it is best to save a parameter file for each.

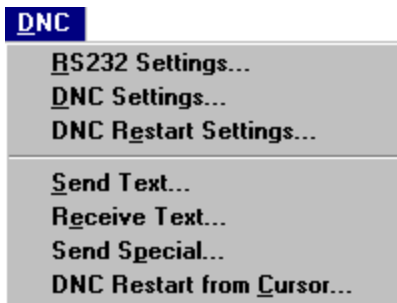
For more information click on the graphic below



DNC Menu Commands

The DNC menu is used to transmit NC files to and from a CNC control. The parameter settings allow the software to be configured to support a variety of controls.

For more information click on the graphic below



Automation Menu Commands

The "Automation Menu" is the most important new feature of FilterMAX / EZ-DNC release 6 . By implementing the Sax Basic Script Engine, it provides the user with a Visual BASIC compatible macro language and a completely interactive design and development environment. In addition to the standard BASIC language support, the script language gives also full access to all functions and properties of the EZ-DNC text windows as well as other Windows applications like EZ-CAM which support OLE.

For more information click on the graphic below



For more information on the macro Editor and the language please see the Sax Basic help.

Window Menu Commands

The Window menu provides you with three standard options for viewing the contents of your files. As you open files, the Window menu records the name of each file. A check mark placed by the name of the file indicates that the file is in active window. Only one window can be the active window. To make another window the active window, click the title bar or select the file from Window menu.

For more information click on the graphic below



Tech Notes

Overview Technical Notes

About the Software

[Language Acceptance](#)

[Plane Restrictions](#)

[Offset Commands](#)

[G90 and G91 Commands](#)

[Digitized XYZ Files](#)

[Bridgeport Controls](#)

[Serial Port Diagrams](#)

Troubleshooting

[Optimization Failure](#)

Reference Information

[ASCII Character Codes](#)

Bridgeport Controls

If you have a control which requires absolute arc center coordinates or reversed G2/G3 directions in the G18, G19 planes (i.e. Bridgeport Controls), you must check the appropriate box in the Other Settings dialog box located in the Preview menu.

See also

[Other Settings...](#)

Digitized XYZ Files

Digitized XYZ files may be converted by adding G1 to the beginning of the file. Tangency tolerance may need to be as high as 30 degrees to get a significant reduction for laser digitizing.

G90 and G91 Commands

G90/G91 commands can be made anywhere in the file with or without XYZ input. If G90/G91 is specified in the middle of a XZ or YZ planar pass, optimization output is performed according to the G90/G91 status at the beginning of the pass and then the specified mode is turned on.

Language Acceptance

This software accepts G-Code programs and Heidenhain conversational only.

Offset Commands

Any work coordinate offset commands which are sensitive to the current work plane must be preceded by the appropriate G17, 18, or 19 command. Otherwise, the software will assume the offset to occur with respect to the last plane which was converted to arcs. In most cases, you can just add a G17 to the offset command line.

Optimization Failure

If the software fails to optimize, check to be sure that correct machining methods have been employed in your CAM system. Your G-Code should lie on the X-Y, Y-Z, or X-Z planes.

Plane Restrictions

The software will only output arc moves in the X-Z, Y-Z, or X-Y planes.

Appendixes

ASCII Character Codes Decimal Symbol Keystroke

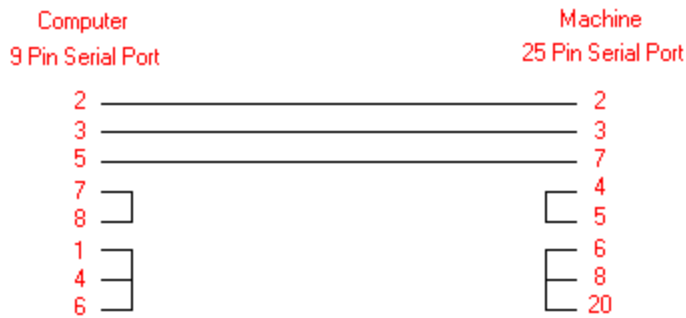
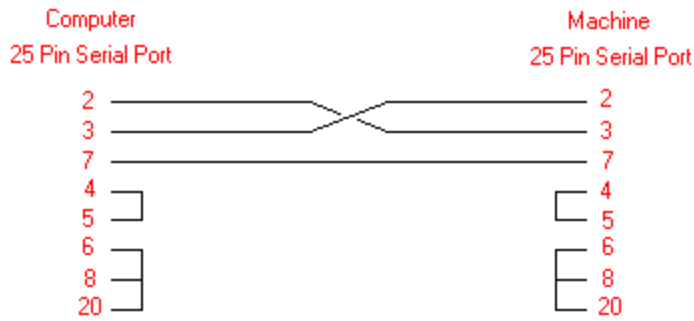
0	NUL	CTRL @
1	SOH	CTRL A
2	STX	CTRL B
3	ETX	CTRL C
4	EOT	CTRL D
5	ENQ	CTRL E
6	ACK	CTRL F
7	BEL	CTRL G
8	BS	CTRL H
9	HT	CTRL I
10	LF	CTRL J
11	VT	CTRL K
12	FF	CTRL L
13	CR	CTRL M
14	SO	CTRL N
15	SI	CTRL O
16	DLE	CTRL P
17	DC1	CTRL Q
18	DC2	CTRL R
19	DC3	CTRL S
20	DC4	CTRL T
21	NAK	CTRL U
22	SYN	CTRL V
23	ETB	CTRL W
24	CAN	CTRL X
25	EM	CTRL Y
26	SS	CTRL Z
27	ESC	CTRL [
28	FS	CTRL %
29	GS	CTRL]
30	RS	CTRL ^
31	US	CTRL _
32	SP	SP
33	!	!
34	"	"
35	#	#
36	\$	\$
37	%	%
38	&	&
39	'	'
40	((
41))
42	*	*
43	+	+
44	,	,
45	-	-
46	.	.

47	/	/
48	0	0
49	1	1
50	2	2
51	3	3
52	4	4
53	5	5
54	6	6
55	7	7
56	8	8
57	9	9
58	:	:
59	;	;
60	<	<
61	=	=
62	>	>
63	?	?
64	@	@
65	A	A
66	B	B
67	C	C
68	D	D
69	E	E
70	F	F
71	G	G
72	H	H
73	I	I
74	J	J
75	K	K
76	L	L
77	M	M
78	N	N
79	O	O
80	P	P
81	Q	Q
82	R	R
83	S	S
84	T	T
85	U	U
86	V	V
87	W	W
88	X	X
89	Y	Y
90	Z	Z
91	[[
92	\	\
93]]
94	^	^
95	`	`
96	·	·
97	a	a
98	b	b
99	c	c
100	d	d
101	e	e

102	f	f
103	g	g
104	h	h
105	i	i
106	j	j
107	k	k
108	l	l
109	m	m
110	n	n
111	o	o
112	p	p
113	q	q
114	r	r
115	s	s
116	t	t
117	u	u
118	v	v
119	w	w
120	x	x
121	y	y
122	z	z
123	{	{
124		
125	}	}
126	~	~
127	DEL	DEL

Serial Port Diagrams

The diagrams below show typical software handshake cables for connection to either a 25 pin or a 9 pin computer serial port.



Contact Support

Technical Support - USA

Contact: EZ-CAM Inc., Steve Genther
 Email : sales@ezcamsoft.com
 Phone : 201-420 7360
 Web : www.ezcamusa.com

Technical Support – Germany

Contact: EZ-CAM Software Gmbh, Malcolm Dearing
 Email : ezcamsoft@aol.com
 Phone : +49 2161-694630

Technical Support - Japan & Far East

Contact: FCS International, K. Inoue
 Email : fcsi@fcsi.co.jp
 Phone : +81 462-731433

Technical Support – England

Contact: Sodi-Tech E.D.M. Ltd, Stuart Burbage
 Email : stuart@sodi-techedm.co.uk
 Phone : +44 2476-511677

Contents

Help topics are divided into categories represented by the icons below. To select a category with the mouse, point to its icon then click. With the keyboard, press Tab to highlight the category you want then press Enter. To return to this screen, select the Contents button at the top of the Help window. If you are using the software for the first time, select the "Optimize" icon. It contains a series of procedures which you must complete in order to use the software.



New in Release 6

The new release 6 of EZ -DNC / FilterMAX comes with some significant improvements over previous versions. What follows is a list of the new features and functions with some additional information.

List of important new features:

1. Full 32Bit support. All controls, functions, routines, dialogs and components have been converted to 32Bit. Support of long filenames.
2. Integration of new Sax Basic Script language. This functionality is accessible through the new automation menu and replaces the old proprietary macro language. The Sax Basic Language provides the core language definition and is Visual Basic for Applications(TM) compatible.
3. New "Restart from Cursor" DNC function allows to start sending nc files from any position.
4. Enhanced optimization functions in FilterMAX
5. Usage of Recycle Bin for file delete functions
6. Revised Helpfile

The Menu Commands

Overview

Select a Topic

[File Menu Commands](#)

[Edit Menu Commands](#)

[Format Menu Commands](#)

[Optimize Menu Commands](#)

[Preview Menu Commands](#)

[DNC Menu Commands](#)

[Automation Menu Commands](#)

[Window Menu Commands](#)

Overview of the Menu Commands

A command is an instruction that tells the software to do something. You can quickly choose the most common commands by using the mouse to click buttons on [toolbars](#). For example, to print the contents of the active window,

you click the Print button on the toolbar.

All commands are grouped on menus. Some commands carry out an action immediately; others display a dialog box so that you can select options.

To choose a command from a toolbar

On the **toolbar**, click a button.

To choose a command from a menu

1. Point to a menu name and click the left mouse button.
2. Point to a command name and click the left mouse button.

A command name followed by an ellipsis (...) indicates that the command has additional options you can use to refine the function of the command.

Using the keyboard To make the menu bar active, press the ALT key and then press the underlined letter in the menu name to display a menu. To choose a command, press the underlined letter in the command name. For example, to choose Open from the File menu, press ALT, F, O.

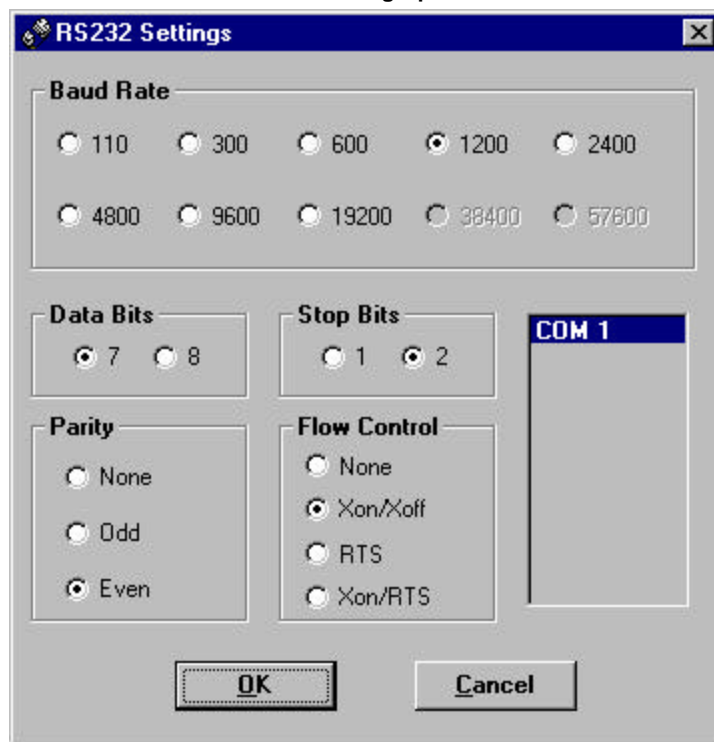
Tip

If you select a menu by mistake, you can close the menu without choosing a command by clicking outside the menu or pressing the ESC key.

RS232 Settings...

When this command is selected, a dialog box will appear allowing the configuration of the communications parameters. To determine the parameter settings, refer to the control manual or manufacturer. When the Save Settings command is selected from the File menu, the RS232 settings will be saved in the specified parameter file.

For more information click on the graphic below



DNC Settings...

The DNC Settings command is used to configure optional communications parameters that may be required by the CNC control. These parameters can be specified by typing an actual literal character, or for nonprintable characters,

by specifying the ASCII decimal equivalent with left and right arrow brackets around it. See [ASCII character Codes](#) for more information on ASCII decimal equivalents.

For more information click on the graphic below

DNC Settings

Send

Program Start: <13>

Program End: <4><13><10>

Block End: <13><10>

Skip Line Prefix: [*

Wait for XON Show in Editor

Receive

Send before receive:

Program Start: 0

Skip Line Prefix: [*

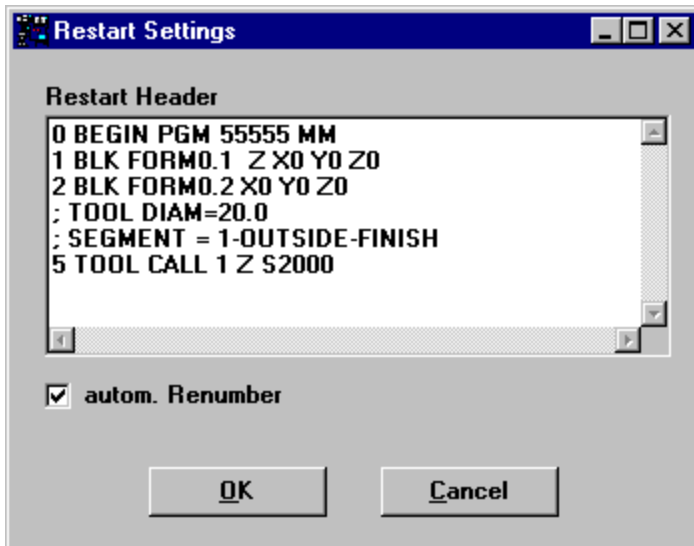
Block End: <13><10>

OK Cancel

DNC Restart Settings

The DNC Restart function is used to resume sending a program from any text position. The restart position is defined by the current cursor position. This function is very useful when restarting transmission of a large program from a specific position (for example after tool damage or tool change).

For most CNC controls, the restart process is the same as a new file transmission. Depending on the control type, it expects a program header with information like tool number, tool offset, feedrate, speed, etc. The "Restart Header" textbox is used to define the program header to provide all that information to the control. Normally it looks like a start section of a standard nc file. Take care to set the correct parameters and be aware of possible formatting restrictions of the control. All parameters regarding the communication are set in the [DNC Settings](#) dialog. The restart settings can be saved together with all other communication parameters by using the [Save Settings](#) command in the file menu.



When the **Automatic renumbering** function is activated, the software checks all lines for numbering and automatically recalculates them before sending them to the control. The settings for renumbering can be changed by using the **Renumber Settings** dialog in the Edit menu.

There are some more rules that apply to the header text itself:

- no empty lines allowed, first empty line in the text window marks the end of header
- maximum of 20 lines are allowed
- total size of header is limited to 242 characters

Important !

Software does not check the user input header for wrong settings with respect to tool number or other machining parameters.

Make sure to set the correct machining parameters to avoid tool crashes or workpiece damage !!

DNC Restart from Cursor

This dialog is used to restart a program from any position.

1. Define a Restart Header by using **DNC Restart Settings** dialog.
2. Place the cursor in the line where you want to start sending data.
3. Select DNC Restart from cursor command from the DNC menu (last minute changes to the header can also be done here).
4. Depending on the Flow Control settings get the machine ready first and then press START button of the software or vice versa.



Wait for Xon

This parameter defines any character(s) that must be received by the software before start sending a file to the CNC control. This setting is only used when Handshake setting is Xon/Xoff.

Communication process when using “Wait for Xon” function:

1. Open the file that is to be send to the machine control
2. Load the communication settings for that particular control (if not already done)
3. Select “Send Text” command from the **DNC menu** . A window displaying the message “Wait for Xon” will appear.
4. Start the receiving function on the machines control. The machine will send the Xon character (ASCII <17>) and the software will automatically start sending the data.

Show in Editor

When this option is activated, the software scrolls through the text in the active text window when it is being sent to the control. This takes time and computer resources, as a result the amount of data sent may be reduced dramatically, especially for high baudrates. It is recommended not to turn this option on when sending big files using high baudrates as in the case of large 3D programs.

Send Text...

The Send Text command is used to initiate a transfer of the data contained in the active window to the CNC control. When this option is selected, a dialog box will be displayed indicating the status of the data transfer. When Xon/Xoff Handshake is activated it will also display the message “Wait for Xon”. To cancel the operation, click the Stop button.



Receive Text...

The Receive Text command is used to initiate a data transfer from the CNC control to the active window. When this option is selected, a dialog box will be displayed indicating the status of the data transfer. At the begin it shows the message "Waiting for text". When the first characters are received, the text changes to "Receiving Text". As long as the process is ongoing, the mouse cursor will be flickering. Only after pressing the Stop button, the received data will be displayed in the actual text window.



Baud Rate

This setting controls the data transfer rate from one device to another. The baud rate can be set from 110 to 19200. To change the baud rate or many of the other options in this dialog box, simply place the cursor inside the circle next to the desired setting and click the mouse.

Data Bits

The Data Bits option can be set to either 7 or 8. This setting defines the character length that is required by the CNC control, and should be set accordingly.

Stop Bits

The stop bits option can be set to either 1 or 2. In most cases, this option should be set to 1.

Parity

This option can be set to either None, Odd, or Even. If the control requires 8 data bits then this setting would typically be set to None. If the control requires 7 data bits then this option should be set to Even.

Flow Control

This setting is used to specify the type of protocol (handshake) used to coordinate the flow of data from the computer to the CNC control and vice versa. The way in which the control is designed to communicate and the type of cable being used, will most likely dictate which flow control option should be selected.

None

This option should only be used for simple connections made at extremely low baud rate settings. With this setting , the data is transferred without any checks to ensure that one device can receive the file as quickly as the other is sending.

Xon/Xoff

In most cases the Flow Control should be set to Xon/Xoff. This type of transfer protocol is called software handshaking. With this option, when the buffer of the receiving device becomes full , an invisible control character (DC3 reader stop) is sent to the sending device to temporarily suspend the transmission. When the buffer of the receiving device becomes free, a different control character (DC1 reader start) is sent to the sending device and the data transfer is resumed. This type of transfer requires that only two data wires (#2 and #3) be used for transferring the file as well as managing the flow control.

RTS

The RTS protocol is used for what is called a hardware handshake interface. This means that the flow control management between the sending device and the receiving device is accomplished by using additional cable wires. These cable wires act as a switch, by changing their voltages when the receiving devices buffer becomes full, and then also, when it can continue receiving again. Since there are several variations on hardware handshake cable configurations, it is recommended that the cable configuration be obtained from the control manual or manufacturer.

Xon/RTS

The Xon/RTS option is a combination of methods. With this option the flow control is managed by using both control characters and the additional cabling connections and therefore also requires a hardware handshake cable connection.

Communications Port

This setting is used to select the desired serial port where the communications cable should be connected to. The list only shows ports that are available for communication (automatically checked by the software). Number of ports is limited 32.

Tip !

In case of trouble with the automatic port detection the number of the ports in the listbox can be set manually. This can be achieved by editing the **PortMax** setting in the desired parameter file (*.PAR). If this value is set to a number greater 1000 (expl. 1004 => port 1-4), the automatic detection is not used. But remember, limit is still 32 ports.

Program Start (send)

This option is used to define any additional characters that should be sent when beginning a program transfer to the

CNC control. Some controls require a special character to be sent to signal that a transfer is being initiated.

Program End (send)

This option is used to define any additional characters that need to be sent at the end of the program file, when sending to the CNC control. Some controls require that a special character be received before the transmission will be terminated and the file saved.

Block End

This option is used to define the end of block character(s) required by the CNC control when sending files. Also when receiving files from the machine, the software uses this setting to arrange the line that are received. This parameter is normally set to <13><10> (carriage return and line feed). For more information on ASCII decimal equivalents refer to the [ASCII Character Codes](#).

Skip Line Prefix (send)

The Skip Line Prefix option is used to define the start character of any program lines that should be ignored when the file is transferring to the CNC control. This is typically used to define the start character of a program comment, allowing any comments to be stripped from the program during the transfer to the control.

Send Before Receive

This parameter defines any characters that must be sent to the CNC control to signal the control to begin sending a file to the computer. Some controls will not begin sending a file until they receive a special character. This parameter is typically set to <17> (DC1 reader start) however any ASCII character may be specified.

Program Start (receive)

This option is used to tell the software when to begin expecting program information being sent from the control to the computer. This parameter is typically set to either % or <13><10> for carriage return and line feed.

Skip Line Prefix (receive)

This option is used to define the start character of any line that should be ignored when transferring a program from the control to the computer. This is typically used to strip any program comments from the file being received.

Send Special...

When Send Special is selected system starts waiting to receive Program Address + Filename + EOB. When it detects this sequence it extracts the filename and adds File Extension to it (standard *.TXT). After searching for the file in the current directory system starts waiting for XON from the control. After receiving XON this file is sent directly from the harddisk. When transfer is complete system starts waiting for the next filename. Program Address can be customized by using [Other Settings](#) dialog in the Preview menu. File Extension of the last saved or open file is automatically remembered.

Undo

The Undo command reverses actions in the order they were performed. This command will undo the last ten actions made in the active window.



The Undo button

Cut

The Cut command is used to delete selected characters from the active window and copy them to the clipboard. Each time this command is used, the contents of the clipboard is replaced with the new selection.



The Cut button

Copy

The Copy command is used to copy selected text to the clipboard so that it may be reinserted into any active window. Each time text is copied, the contents of the clipboard is replaced with the new selection. This option is useful for duplicating repetitive sequences elsewhere within the active window.



The Copy button

Paste

The Paste command is used to insert the contents of the clipboard into any active window at the current cursor position. This command does not change the contents of the clipboard.



The Paste button

Clear

The Clear command is used to delete the current selection of the active window without copying it to the clipboard. This command provides a way to delete a selection while also protecting the contents of the clipboard.

Select all

The Select All command is used to select the entire contents of the active window. This command is useful when making global program edits such as changing the font type or size.

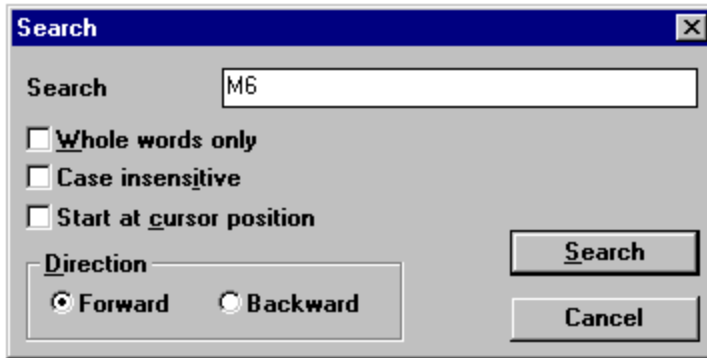
Search...

The Search command is used to locate a specific sequence of characters. This command is useful for locating items such as spindle speeds, feed rates, tool changes, program stops, etc.. When this command is selected, a dialog box is displayed. It allows you to enter the characters to be searched for and provides choices in how the search is to be conducted. A search can be made either forwards or backwards depending on the setting of the Direction button. A search will start at the beginning of the file unless the Start at cursor Position check box is chosen. Also, you can specify whether to search for the Whole word only as opposed to any occurrence, and can choose to Ignore Case

sensitivity by selecting these corresponding check boxes. The search dialog box is displayed below.

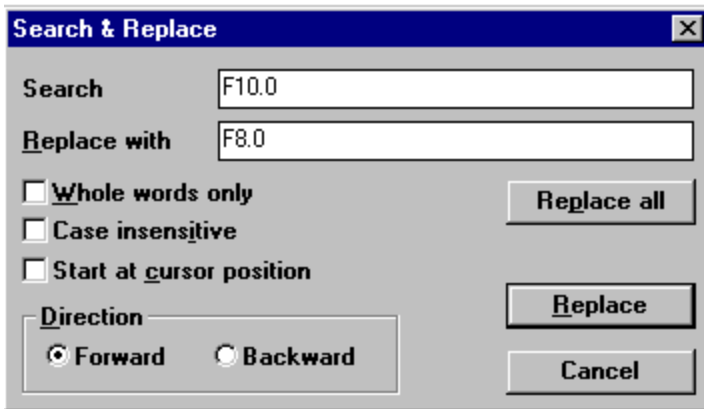


The Search button



Search and Replace...

The Search & Replace command behaves much like the Search command. Unlike the Search command, the Search & Replace command allows you to enter both the characters to be searched for and the characters to replace them with. This feature is very useful for performing global edits to a program, such as replacing all occurrences of one feed rate with a new one. When this command is selected, the Search and Replace dialog box will be displayed.



Continue Search

This command will continue the search operation beginning at the current position using the previously specified search criteria.



The Continue Search button

Compare to File...

The Compare to File command is used to compare the contents of one file with another. This command activates a dialog box which prompts you to select a file for comparison. Once you have selected a file, the system automatically compares the two files. The results of the comparison are indicated by red and blue text.

For example, if you were working in an active window named "File A" and wanted to compare it to a disk file named "File B" you would first select "File B" from the dialog box. The resultant file comparison would be displayed in the

active window as follows:

All red text indicates that the information is contained in "File A" but not in "File B".

All blue text indicates that the information is contained in "File B" but not in "File A".

Goto Color

The Goto Color command is used to search for the next occurrence of a color change within the active window. This command is especially useful for file comparison.

ReNUMBER Settings...

The ReNUMBER Settings command is used to resequence numbers in the active window. When selected, a dialog box will appear prompting for the Start Number, Step Size, Max Number, Min Digits, and Skip Characters. See below.



See also

[ReNUMBER](#)

ReNUMBER

The ReNUMBER command is used to renumber the contents of the active window according to the criteria entered in the ReNUMBER Settings dialog box.



The ReNUMBER button

See also

[ReNUMBER Settings...](#)

New

The New command creates a new window. The Window menu will automatically update the list of available windows. A check mark next to the name of the file indicates the active window. By default, each new window is named "untitled". New windows cannot be created in succession. In order to create a second "new" window, the active window must contain some type of information. When closing any window, the system will prompt you to save any changes.



The New button

Open...

The Open command is used to load existing files. Once a file has been loaded, it may be either edited, renumbered, or graphically verified.



The Open button

Close

The Close command prompts you to save any changes to the active window and then closes the window.

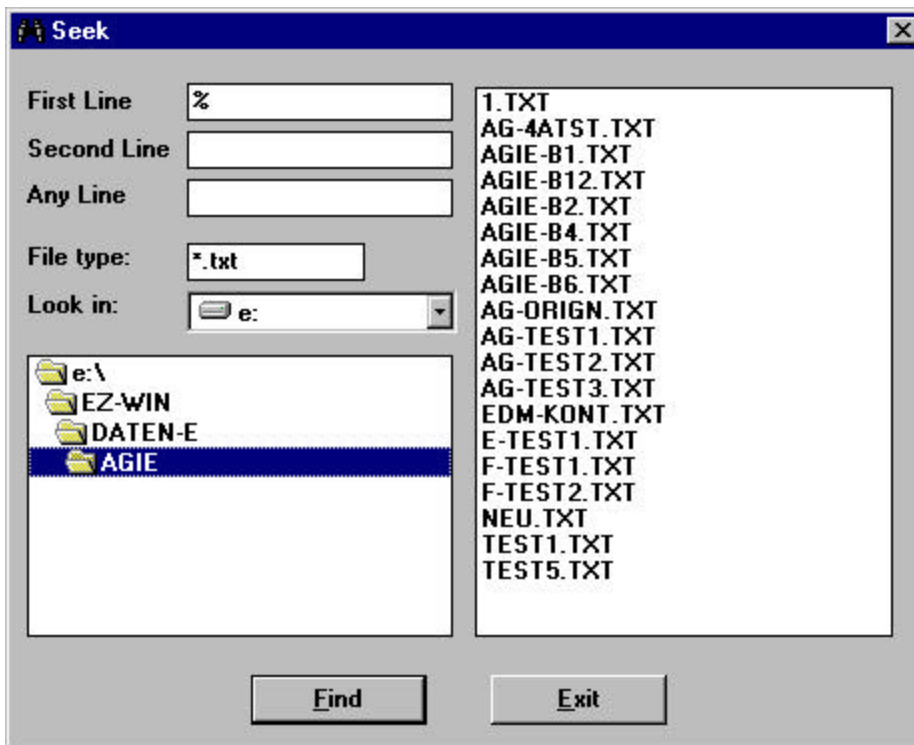
Seek...

The Seek command is used to locate and open a text file based on a user-specified set of search criteria. When this command is selected, a dialog box will display prompting for the Seek parameters and the directory from where to begin the search. Once the search is completed, a dialog box will list all of the files that were found based on the given search parameters. The user may then either open a file by selecting it from the list and choosing Open, or continue with a new search by choosing Reset. The Seek dialog box is used to specify the seek parameters.



The Seek button

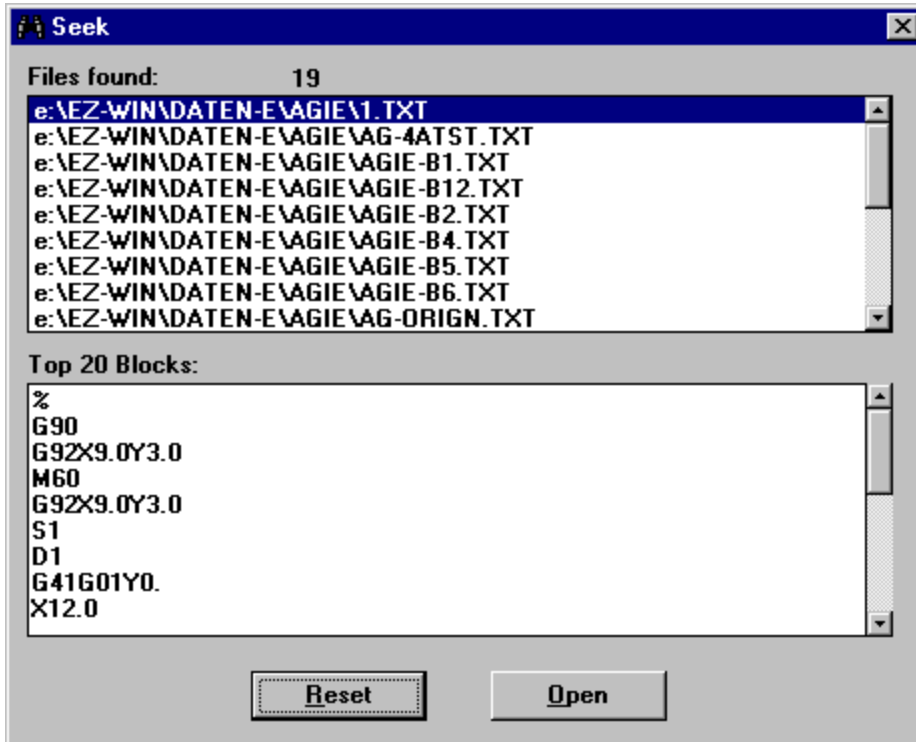
For more information click on the graphic below



Seek : The File Selection Window

This window is displayed when selecting FIND button on the SEEK dialog. According to the previous defined search strings and the filetype that was used, all files that have been found are displayed in the upper filelist. The textbox below displays the first 20 lines of the selected file. By using the OPEN button the currently selected file can be opened.

For more information click on the graphic below



Filelist

This listbox shows all files that have been found using the previously defined search strings.

Header Textbox

This textbox shows the first 20 lines of the actual selected file in the file listbox.

Reset Button

Use this button to go back to the first seek window to define new search strings or change file type setting.

Open Button

Use this button to open the currently selected file into a new textwindow.

Insert...

The Insert command is used to place the contents of an existing file into a specific location of the active window. When insert is selected, a standard file dialog box is displayed allowing the selection of the file to be inserted. The selected file is then placed at the current cursor position of the active window. This feature is useful for importing standard sequences of events such as tool changes, macros, etc.

Delete...

The Delete command provides a means for deleting unwanted files without the need to use Explorer. When selected, a dialog box will display allowing the user to locate and select the file to be deleted.

See also

[Use Recycle Bin](#)

Save

The Save command is used to save the contents of the active window. If the active window has not been previously saved, then a dialog box will display allowing a name to be specified. Once the file has been named, each time this command is selected the file will be saved using the current file name that is displayed on the title bar of the window.



The Save button

See also

[Save as...](#)

Save as...

Like the Save command, the Save As command is used to save the contents of the active window. Each time that Save As is selected, a file dialog box is displayed allowing a new file name to be entered. When this option is used, the current file name will be changed to the new name that is specified.

See also

[Save](#)

Save as DXF...

This option is used to create a DXF (Direct Exchange Format) geometry file for the active window. This format represents each linear and circular move of an NC program file. When this menu is selected a dialog box will appear allowing the DXF file name to be specified. This option is very useful for generating arc and line cross-section geometry of an optimized program, allowing it to be read back into a CAM system for further surface development or design documentation.

Load Words...

The Load Words command is used to load a list of commonly used words into the system. The Word List that you create can be used interchangeably with your NC text file. When the Word List is loaded, you must select the Word List command from the Format menu to turn it on.

See also

[Word List](#)

Load Settings...

The Load Settings command is used to load the CNC parameter settings used by the Optimize, Renumber, and Preview commands. The parameter file contains all of the formatting details necessary to interpret an NC program of a specific control.



The Load Settings button

Save Settings...

The Save Settings command is used to save the current CNC parameter settings used by the Optimize, Renumber, and Preview commands. A different parameter file can be saved for each type of CNC control, allowing unique settings to be retrieved for each prior to performing a program optimization or verification.

Printer Setup

This command opens the standard windows printer setup dialog.

Print

The Print command is used to print the contents of the active window. The file will be printed using the default printer which can be changed by using **Printer Setup** command.



The Print button

Last Files List

This area is used to display the names last four files that have been worked with.

Exit

The Exit command prompts you to save any unsaved changes to your files and then ends the application.

Dialog Box

A window that displays the available options when you choose a command.

Toolbar

A bar with buttons that perform some of the most common tasks such as opening, saving, and optimizing files.

File Filter

Conducts a search based on the file extension. For example, if you wanted to search for macro files, you would enter `*.mac`

First Line

Conducts a search based on information contained in the first line of the file. Files which contain the specified text in the first line will be displayed.

Second Line

Conducts a search based on information contained in the second line of the file. Files which contain the specified text in the second line will be displayed.

Any Line

Conducts a search based on information contained in any line of the file. Files which contain the specified text will be displayed.

List of Drives

Select the drive to be searched.

List of Directories

Select the directory to be searched. Subdirectories are automatically included. This option is useful for narrowing the scope of the search.

Find

Begins the search.

List of Files

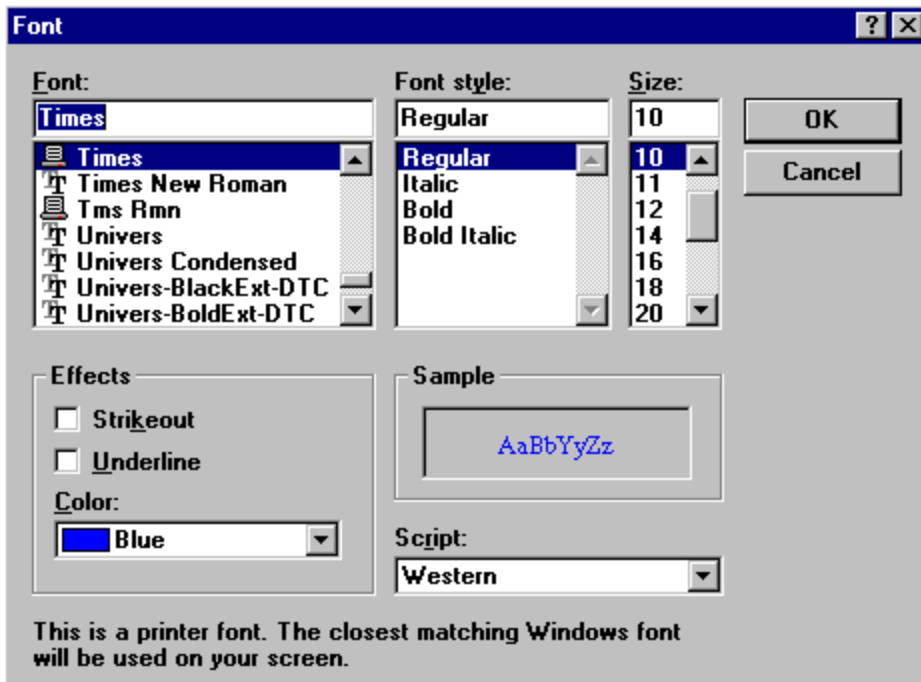
Displays all files in the active directory that match the **File Type** setting.

Exit Button

Cancels the search.

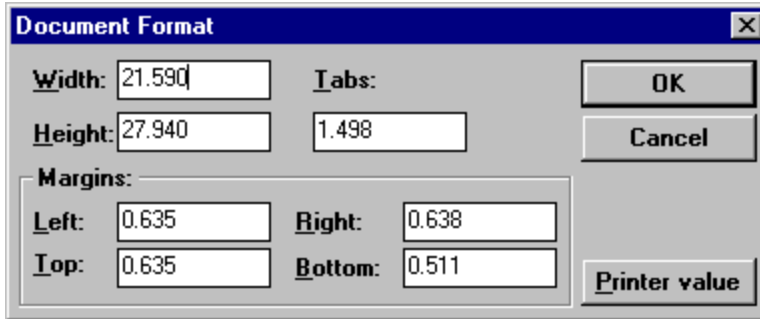
Font...

The Font command allows you to change the font currently being used by the software. You can specify the style (such as bold and, italic), font size, color, and font effects.



Margins...

The Margins command is used to define the page settings. When this command is chosen, a dialog box will display prompting for the paper width and height, tab spacing, left and right margins, and top and bottom margins. There is also a button named Printer value which is used to automatically establish the page settings based on the current printer.



Single space

This command causes every new line to be separated from the previous line by **one**, **one and a half** or **two** blank lines.

Gray Background

This command is used to change the text editor background color from its default of **white**, to **gray**.

Automatic Linebreak

When this option is turned on, any program line that exceeds the working limits of the paper (paper width minus left and right margins) will automatically be broken and the remainder of the line will be displayed on the next line.

Important:

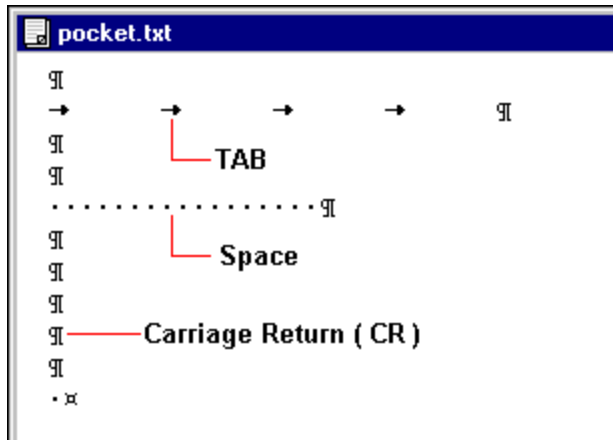
This option also affects receiving of data via the serial port. If the received text contains no block end characters and this option is turned off, all characters that are received are lost after the first line reaches maximum width.

See also

[DNC Settings](#) for information where to set block end characters

Show CR/LF

When this option is turned on, all characters such as Tab, Carriage Return and Line Feed are displayed in the textwindow.



Uppercase

When this option is turned on, every character that is typed will be forced to appear as upper case regardless of whether the shift key is used or not.

Units (Inch/Metric)

This command toggles the dimension units from Inch to Metric and vice versa. This command must be set to the correct units prior to optimizing or graphically verifying a file.

Word List

This command is used to display a list of predefined words. When activated, the Word List window is displayed on the right-hand side of the screen. The vertical alignment of this window allows you to work simultaneously with your existing file and the Word List. If the contents of the Word List is too large to be displayed at once, it will be displayed in a scrollable window. Any line of text that appears in the Word List can be directly inserted into the active window. All you have to do is click on a line of text in the Word List window. The selected text will be copied into the active window at the current cursor position. Make sure that your cursor is in the correct position before selecting any text from the Word List. If you make a mistake, you can click on the Undo button. You may also select multiple lines of text from the Word List window. If the information should reside on separate lines, you will have to create the line breaks manually by using the vertical bar character "|". The Word List can be toggled on and off by selecting this command.

Use Recycle Bin

This option affects the functionality of the Delete command in the file menu. When activated, the deleted files are moved to the windows recycle bin from where they can be restored. Otherwise the files are removed from disk.

Getting Started

[Overview](#)

First Things First

- [Configuring Control Parameters](#)
- [Setting Optimization Parameters](#)
- [Saving Your Settings](#)

Working With Files

- [Converting NC Files](#)
- [Previewing NC Files](#)
- [Saving NC Files](#)

Overview Installation Information

The Release 6 of EZ -DNC and FilterMAX is designed to be compatible with Microsoft Windows 32Bit operating systems like Windows 95 / 98 or NT4. Please see the following sections for more information on system requirements and installation.

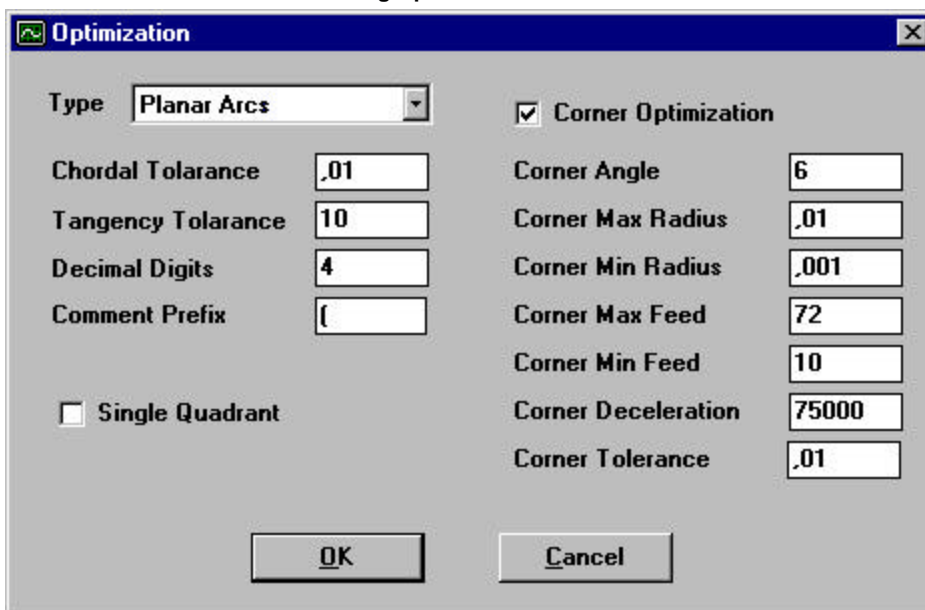
Instructions

- [System Requirements](#)
- [Installing the Software](#)
- [Connecting the Key Ring](#)

Settings...

The Settings command is used to establish the parameters that are used to control the arc and nurbs fitting optimization. When this command is selected, a dialog box is displayed allowing the entry of the arc and nurbs fitting tolerances and other formatting options. Once the parameters have been set, they can be saved using the Save Settings command located in the File menu.

For more information click on the graphic below



Convert...

When the Convert command is selected, a file selection dialog will appear allowing the selection of the file to be optimized. Once the desired file has been selected, the optimization process begins and a dialog box showing the status of the conversion is displayed. When the optimization is complete a dialog box will display showing how many blocks were in original file and how many blocks the new program contains. It also displays the percentage of program size reduction. The optimized file is loaded into a separate text window. The filename of this window is preset using the original filename with the new extension „FMX“ . The new file can then be verified, saved to disk, or sent directly to a machine control.

The time required to optimize a file depends on several factors including the size of the file and the tolerances that are specified. In general, several minutes will be required to complete the process.

Convert to Directory

When using this option, the optimized files are stored directly on the harddisk instead of loading them into separate MDI text windows as with the „Convert“ option. The system uses same directory and original filename with the extension „FMX“ for the new file. Multiple files can be converted at once by using “*” and a valid extension as the name of the input file.

Type

This setting gives you a list of 5 options to define the type of output commands for an optimized file.

Planar Arcs

Only moves on XY, XZ and YZ plane are output as arc moves corresponding to the selected/active **machine control** setting. All others will be linear moves.

Bridgeport Bezier

All optimized moves are output in the special „Bezier“ format (G7) available on actual Bridgeport 3D controls (EZ-TRAK).

Fanuc Nurbs

All optimized moves are output in the special „Nurbs“ format (G6.2) available as an option for fanuc 3D controls.

Fanuc 3D Arcs

All optimized moves are output in the special „Nurbs“ format (G2.4) available as an option for fanuc 3D controls.

Only Corners

Only corner and feedrate optimization without arc fitting.

Chordal Tolerance

The Chordal Tolerance controls how closely the arc moves will be fitted to the points of the original linear moves.

Tangency Tolerance

The Tangency Tolerance controls how closely each arc is placed to being tangent to the adjacent move. Setting the tolerance to a smaller value will ensure that the newly fitted arcs are placed more smoothly from one to another.

Decimal Digits

The Decimal Digits setting controls the number of places after the decimal point that the arc coordinates will be output in the NC program.

Comment Prefix

This parameter defines the start character for a program comment. The purpose of this option is to provide a means to specify a new chordal tolerance and tangency tolerance for different operations within a program. This feature is useful for controlling tolerances based on the type of operation. If the program does not contain any comments with tolerance settings, then the entire program will be optimized using the Chordal Tolerance and Tangency Tolerance specified in the Optimization dialog box.

Single Quadrant

Check this option if your control does not support arcs crossing multiple quadrants.

Corner Optimization

When this option is activated, the software calculates the angle between consecutive moves. If the angle is bigger than “Corner Angle” then fillet radius and feedrate slowdown modifications are applied.

First operation:

The software tries to insert a fillet with “Max Corner Radius”. If this fillet can not be inserted then other fillet radii between “Max Corner Radius” and “Min Corner Radius” are tried. If a successful radius is found then the distance between the inserted fillet circle and corner point is calculated. If this distance is less than “Corner Tolerance” then fillet is output into the G-Code file.

Second operation:

Feedrate modification is applied depending on the fillet radius from the first operation. If no fillet could be inserted then “Min Corner Feed” is selected. Otherwise a feedrate between “Max Corner Feed” and “Min Corner Feed” is selected depending on the fillet radius.

Example:

Max Corner Radius : *0.01 mm*
Min Corner Radius : *0.001 mm*
Max Corner Feed : *200 mm/min*
Min Corner Feed : *50 mm/min*

Result with corner radius: <i>0.01mm</i>	⇒ Feedrate: <i>200mm/min</i>
Result with corner radius: <i>0.005mm</i>	⇒ Feedrate: <i>125mm/min</i>
Result with Successful corner radius: <i>0.001mm</i>	⇒ Feedrate: <i>50mm/min</i>

Corner Angle

There is no filleting in case the calculated corner angle between two consecutive elements is smaller than this setting.

Corner Max. / Min. Radius

When inserting a fillet, the software tries to use “Max Corner Radius”. If this fillet can not be inserted then other fillet radii between “Corner Max. Radius” and “Corner Min. Radius” are tried (with approx. 20 steps in size between). If a successful radius is found then the distance between the inserted fillet circle and corner point is calculated. If this

distance is less than “Corner Tolerance” then fillet is output into the G-Code file corresponding to the **Type** setting.

Corner Max. / Min. Feed

When optimizing, feedrate modification is applied depending on the fillet radius from the first optimizing operation. If no fillet could be inserted then “Min Corner Feed” is selected. Otherwise a feedrate between “Max Corner Feed” and “Min Corner Feed” is selected depending on the fillet radius.

Example:

Max Corner Radius : 0.01 mm
Min Corner Radius : 0.001 mm

Max Corner Feed : 200 mm/min
Min Corner Feed : 50 mm/min

Result with corner radius: 0.01mm ⇒ Feedrate: 200mm/min
Result with corner radius: 0.005mm ⇒ Feedrate: 125mm/min
Result with Successful corner radius: 0.001mm ⇒ Feedrate: 50mm/min

Corner Deceleration

For this parameter the “Deceleration” constant of the machine tool control should be input. This constant is in **(mm/sec) / sec** and specifies the amount of feedrate (mm/sec) which can be changed per second. For more information about the appropriate value for your machine see the controls handbooks, parameter files or ask your local machine tool dealer.

Example:

Corner Deceleration = 1,000,000 (mm/min)/min

Feedrate in the original G-Code File = 1000mm/min

Selected Corner Feedrate = 125mm/min

Calculated distance needed to slow down from F1000 to F125 :

$$\Rightarrow (1000 \times 1000 - 125 \times 125) / (2 \times 1,000,000) = \underline{0.49 \text{ mm}}$$

In this example software would output F125 command 0.49mm before the corner point (or corner radius). If deceleration value is set to 0 then no feedrate optimization is applied and F125 command will be output with the fillet move.

Corner Tolerance

There is no filleting if the distance between the inserted fillet circle and corner point is bigger then this parameter.

Register Settings...

The Register Settings command is used to describe the specific word formatting of an NC file that is to be optimized using the Optimize command or verified using the Preview command. The letter address for each program word (i.e. axis coordinates, spindle speed, feed rate, etc.) can be specified. Also, a factor value can be applied to each word allowing the software to interpret a word as being a different value then was read from the NC program. By default, the software assumes that a file that is programmed in Inch will have four places after the decimal point, and that the decimal point will be included with each word's value. For some controls it may be necessary to use these factors to achieve the desired graphic verification. For instance, if a control does not use decimal point programming but uses trailing zeros format instead, then a Factor value of .0001 could be used to tell the software that the decimal point position is four digits from the right.

For more information click on the graphic below

	Address	Factor		Address	Factor
X-Axis	X	1.0	R-Arc	R	1.0
Y-Axis	Y	1.0	Feed	F	1.0
Z-Axis	Z	1.0	Drill Depth	Z	1.0
X-Incremen	U	1.0	Retract Pln	R	1.0
Y-Incremen	V	1.0	Cut-in	Q	1.0
Z-Incremen	W	1.0	Feed-Drill	F	1.0
X-Arc	I	1.0	Dwell	P	1.0
Y-Arc	J	1.0	RPM	S	1.0
Z-Arc	K	1.0	Pitch	J	1.0

See also
[Configuring Control Parameters](#)

Preview / X-Axis

Address of horizontal milling axis (X). For lathe operations the X-Axis is automatically displayed as vertical axis and in case of diameter input the factor has to be set to 0.5.

Preview / Y-Axis

Address of vertical axis for milling operations (Y).

Preview / Z-Axis

Address of 3rd (depth) axis for milling operations. For lathe the Z-Axis is automatically displayed as horizontal axis.

Preview / X-Increment

Address for incremental movements of the horizontal (milling) or vertical (lathe) axis (default "U"). Some controls (for exam. Fanuc lathe controls) use different axis addresses for incremental moves instead of standard G90/G91 settings.

Preview / Y-Increment

Address for incremental movements of the vertical (milling) axis (default "V").

Preview / Z-Increment

Address for incremental movements of the 3rd (depth for milling) or horizontal (lathe) axis (default “W”). Some controls (for examp. Fanuc lathe controls) use different axis addresses for incremental moves instead of standard G90/G91 settings.

Preview / X,Y,Z-Arc

Circle center address.

X-Arc ⇒ Address of horizontal axis circle center (default “I”)

Y-Arc ⇒ Address of vertical axis circle center (default “J”)

Z-Arc ⇒ Address of 3rd (depth) axis circle center (default “K”)

Preview / R-Arc

Address for arc radius when using radius input for arc definition. As a standard , arcs < 180° are defined using positive radius values. For arcs > 180° the radius value is defined by a negative value. For some controls (expl. Bosch CC100) this may just be the opposite way. In that case use a negative factor (-1.0).

Preview / Feed

Feedrate address for standard milling and lathe operations (default “F”)

Preview / Drill Depth

Drill depth when using canned cycles (default “Z”).

Factor settings for interpretation of input value:

Factor = 1.0	=> absolute
Factor = -1.0	=> incremental
Factor = 0	=> based on G90/91 setting

Preview / Retract PIn

Retract plane when using canned cycles (default “R”).

Factor settings for interpretation of input value:

Factor = 1.0	=> absolute
Factor = -1.0	=> incremental
Factor = 0	=> based on G90/91 setting

Preview / Cut-In

Incremental depth for Chipbreak or Deephole canned cycles (default “Q”)

Preview / Feed-Drill

Feedrate when using canned cycles (default "F")

Preview / Dwell

Dwell time address. Sometimes dwell time is defined by using 1/1000 seconds. In that case use "0.001" as factor setting.

Preview / RPM

Address for spindle speed in revolutions per minute

Preview / Pitch

Address for pitch setting when using pitch (inch / rev.) instead of standard feedrate (inch / minute) setting in canned tapping cycle. Default setting "S".

G-Code Settings...

This command is used to describe the specific G-Codes of the NC file that will be executed in Preview. When selected, this menu will display a dialog box containing parameters for defining Z-Axis canned cycles and other commonly used G-Codes. The Skip Line option is used to define the control's block skip character so that Preview will ignore any program lines that are preceded with the this character. Any G-Code settings that are not applicable can be ignored when defining these parameters.

For more information click on the graphic below

Parameter	Value	Parameter	Value
Drilling	G81	Turn-Thread	G32
Spot Facing	G82	Const Surf	G96
Chipbreak	G73	Const Spind	G97
Deephole	G83	Units/rev	G99
Reaming	G85	Units/min	G98
Boring (Ortg)	G76	Max. RPM	G50
Boring (Stop)	G86	Skip Line-1	G18
Tapping	G84	Skip Line-2	
Rev Tapping	G74	Skip Line-3	

OK Cancel

Note

Although many of the NC-Codes are user configurable, some of them are not. Following is a list of all of the codes that are recognized by the software. These codes cannot be redefined by the user.

Interpolation Codes G0,G1, G2, G3, G12, G13
Plane Selection G17, G18, G19
Absolute / Incremental G90, G91
Program End M2, M30
Return to Initial Point / Return to R Point (Canned Cycles) G98, G99

See also

[Configuring Control Parameters](#)

Preview / Drilling

Drilling canned cycle command (default **G81**).

Following parameters are used for time calculation:

Retract Pln , **Drill-Depth** , **Feed-Drill** (See [Register Settings](#) dialog)

Rapid (units/min) (See [Other Settings](#) dialog)

Preview / Spotfacing

Spotfacing canned cycle command (default **G82**). Drill cycle with dwell time when reaching depth.

Following parameters are used for time calculation:

Retract Pln , **Drill-Depth** , **Feed-Drill** , **Dwell** (See [Register Settings](#) dialog)

Rapid (units/min) (See [Other Settings](#) dialog)

Preview / Chipbreak

Chipbreak canned cycle command (default **G73**). Drill cycle with a short rapid retract move after every cut-in step for chip breaking.

Following parameters are used for time calculation:

Retract Pln , **Drill-Depth** , **Feed-Drill** , **Cut-in** (See [Register Settings](#) dialog)

Rapid (units/min) (See [Other Settings](#) dialog)

Preview / Deephole

Deephole canned cycle command (default **G83**). Drill cycle with a complete rapid retract move after every cut-in step for chip removal.

Following parameters are used for time calculation:

Retract Pln , **Drill-Depth** , **Feed-Drill** , **Cut-in** (See [Register Settings](#) dialog)

Rapid (units/min) (See [Other Settings](#) dialog)

Preview / Reaming

Reaming canned cycle command (default **G85**). Drill cycle with depth and retract moves by using feedrate setting.

Following parameters are used for time calculation:
[Retract Pln](#) , [Drill-Depth](#) , [Feed-Drill](#)(See [Register Settings](#) dialog)

Preview / Boring (Ortg.)

Boring canned cycle command (default **G76**). Drill cycle with spindle orientation when reaching depth.

Following parameters are used for time calculation:
[Retract Pln](#) , [Drill-Depth](#) , [Feed-Drill](#)(See [Register Settings](#) dialog)
[Rapid \(units/min\)](#) , [Orientation\(sec\)](#)(See [Other Settings](#) dialog)

Note

At the bottom of each hole, the software adds time for spindle orientation. As this is depending on the spindle speed, an average time (**Orientation (sec)**) can be set.

Preview / Boring (Stop)

Boring canned cycle command (default **G76**). Drill cycle with spindle stop when reaching depth.

Following parameters are used for time calculation:
[Retract Pln](#) , [Drill-Depth](#) , [Feed-Drill](#)(See [Register Settings](#) dialog)
[Rapid \(units/min\)](#) , [Stop](#) (See [Other Settings](#) dialog)

Note

At the bottom of each hole, the software adds time for spindle stop. As this is depending on the spindle speed, an average time (**Stop**) can be set.

Preview / Tapping

Tapping canned cycle command (default **G84**).

Following parameters are used for time calculation:
[Retract Pln](#) , [Drill-Depth](#) , [Feed-Drill](#)(See [Register Settings](#) dialog)
[Rapid \(units/min\)](#) , [Reversing](#) (See [Other Settings](#) dialog)

Note

At the bottom of each hole, the software adds time for spindle reversing. As the time for reversing is depending on the spindle speed, an average time (**Reversing**) can be set.

Preview / Rev.Tapping

Reverse Tapping canned cycle command (default **G74**).

Following parameters are used for time calculation:
[Retract Pln](#) , [Drill-Depth](#) , [Feed-Drill](#)(See [Register Settings](#) dialog)
[Rapid \(units/min\)](#) , [Reversing](#) (See [Other Settings](#) dialog)

Note

At the bottom of each hole, the software adds time for spindle reversing. As the time for reversing is depending on the spindle speed, an average time (**Reversing**) can be set.

Preview / Turn-Thread

Thread command (default **G32**) for lathe simulation (only one pass, retraction by using rapid feedrate)

Preview / Const-Surf

Command (default **G96**) when using constant feedrate setting (inch/min or mm/min) for lathe simulation. The spindle speed varies with the workpiece diameter. The spindle speed is therefore very often limited by special commands for safety of workpiece.

Following parameters are used for time calculation:

Max.RPM (See [Register Settings](#) dialog)

Preview / Const-Spind

Command (default **G97**) when using constant spindle speed setting (rpm/min) for lathe simulation

Following parameters are used for time calculation:

Max.RPM (See [Register Settings](#) dialog)

Preview / Units/Rev.

Lathe command (default **G99**) for calculating feedrate moves by using mm/rev setting.

Following parameters are used for time calculation:

Units(rev) (See [Register Settings](#) dialog)

Preview / Units/min

Lathe command (default **G98**) for calculating feedrate moves by using mm/min setting.

Following parameters are used for time calculation:

Units(min) (See [Register Settings](#) dialog)

Preview / Max.RPM

Lathe command (default **G50**) for limiting spindle speed.

Preview / Skip Line 1-3

Code lines containing one of the characters or text sequences defined here are skipped during simulation.

Other Settings...

The Other Settings command is used to describe specific details of the CNC machine and control for which the NC file was written. These settings are used by Optimize and Preview when interpreting an NC file. There are parameter settings for defining the letter addresses and M-Codes for macros, tool numbers, tool changes, and special functions such as pallet changes, rotary table indexing, etc.. There are also settings that can be used to define the rapid rate and the times required for various operations, allowing for more accurate Preview time calculations.

For more information click on the graphic below

Mill-Fanuc	Orientation(sec)	2,0
Program Address	Stop(sec)	0,5
Macro-Call	Reversing(sec)	0,5
Macro-Adresse	Special Func	M70
Macro-Start	Func Time(sec)	5,0
Macro-End	Retraction	0,6
Macro-Repeat	<input checked="" type="checkbox"/> Start Drill next block	
Wkzg.Address	<input type="checkbox"/> Ignore Z	
Tool Change	<input type="checkbox"/> Use Pitch	
Change Time(sec)	<input type="checkbox"/> Reverse Arcs in G17	
Rapid(units/min)	<input type="checkbox"/> Reverse Arcs in G18	
Incremental I, J, K	<input type="checkbox"/> Reverse Arcs in G19	

See also

[Configuring Control Parameters](#)

Preview / Control Type

This option is used to specify the type of machine for which an NC program was written. When this item is selected a list of three machine types will display, Mill-FANUC, Turn-FANUC, and Mill-HEIDENHAIN. For most conventional G-Code Mills and Lathes the Fanuc options should be used. The Heidenhain option is only intended to be used for milling programs that are written in Heidenhain conversational format.

Preview / Program Address

Definition of program address (default 0). For example Fanuc: O2000 ⇒ Program number 2000. This setting is used by the [Send Special](#) command to detect the Address/Name of the program that is requested by sending a dummy text from the control to the PC.

Preview / Macro Call

Definition of macro call address (default M98 for Fanuc).

Example:

M98 P2000 ⇒ Call of subprogram number 2000.

Preview / Macro Address

Definition of macro call address (default **M98**) .

Example :

M98 **P**2000 ⇒ Call of subprogram number 2000.

Preview / Macro Start

Definition of macro start section address (default **O**) .

Example :

```
...  
( Main program )  
...  
G91 G28 Z0 M05  
M30  
O2000  
...  
( sub program )  
...  
M99
```

Preview / Macro End

Definition of macro end section address (default **M99**) .

Example :

```
...  
( Main program )  
...  
G91 G28 Z0 M05  
M30  
O2000  
...  
( sub program )  
...  
M99
```

Preview / Macro Repeat

Definition of macro repeat command (default **L**) .

Example :

M98 **P**2000 **L**10 ⇒ Call of subprogram number 2000 and execute it 10 times.

Preview / Tool Address

Tool number address (default **T**) .

Preview / Tool Change

Tool change command (default **M06**) .

Preview / Change Time

Tool change time (default 8 sec) .

Preview / Rapid

Rapid feedrate in units / sec.

Preview / Arc Center Definition

Definition of arc center programming mode.

Possible settings:

Incremental I,J,K : Definition of circle center always in incremental mode.
Distances (values) are measured from startpoint of circle to midpoint.

Absolute I,J,K : Definition of circle center always in absolute mode.

Incremental + absolute : Definition of circle center always depending on the actual mode setting (G90/G91).

Note:

Settings are only used for Preview when “machine type” is **not** set to “Mill-Heidenhain”.

Preview / Orientation

This setting is used to specify the time required to stop and orient the spindle when using a boring cycle that supports this function. When Preview encounters the G-Code that is specified in the Boring (ortg) cycle, the orientation time will be added to the calculation time for each hole that is being bored. If your machine does not support spindle orientation, this value can be set to zero so that the time calculation will not be affected.

Preview / Stop

This settings is used to specify the time required to stop the spindle when using a boring cycle that stops the spindle before retracting. When Preview encounters the G-Code that is specified in the Boring (stop) cycle, the stop time will be added to the calculation time for each hole that is being bored.

Preview / Reversing

This parameter is used to specify the time required to reverse the spindle when a tapping cycle is used. Each time that Preview encounters the G-Code that is specified in the Tapping cycle, the reverse time will be added to the time calculation for each hole that is being tapped.

Preview / Special Func

This setting allows to define a command that may occur in the nc-code and for which a time should be calculated. For example some kind of special M-function for bar feeder operations. The time that is calculated when this command is detected, is defined under the **Func Time** setting in the “Other Settings” dialog.

Preview / FuncTime

Definition of the time (sec) that is calculated when the **Special Func** command is detected.

Preview / Retraction

This setting is used to specify the retraction distance used by the chip break drilling cycles. Each time that Preview encounters the G-Code that is specified in the Chip Break cycle, the retract distance will be figured into the time calculation for each hole.

Preview / Start Drill next block

This option is used to define when the Z-Axis canned cycles are initiated. Most machines will activate the canned cycle at the current location as soon as the G-Code is read. Some controls however, will not execute the canned cycle until the following block, in this case the Start Drill Next Block option should be selected.

Preview / Ignore Z

If this setting is activated, the system skips all "Z" axis command / parameters in lines where drilling operation are performed.

Preview / Use Pitch

If activated, the system uses pitch (inch or mm / round) instead of Drill-Feed commands for time calculation in tapping cycles.

Preview / Reverse Arcs in G17/18/19

This setting reverses simulation of standard arc direction (G02/G03) in G17/18/19 plane.

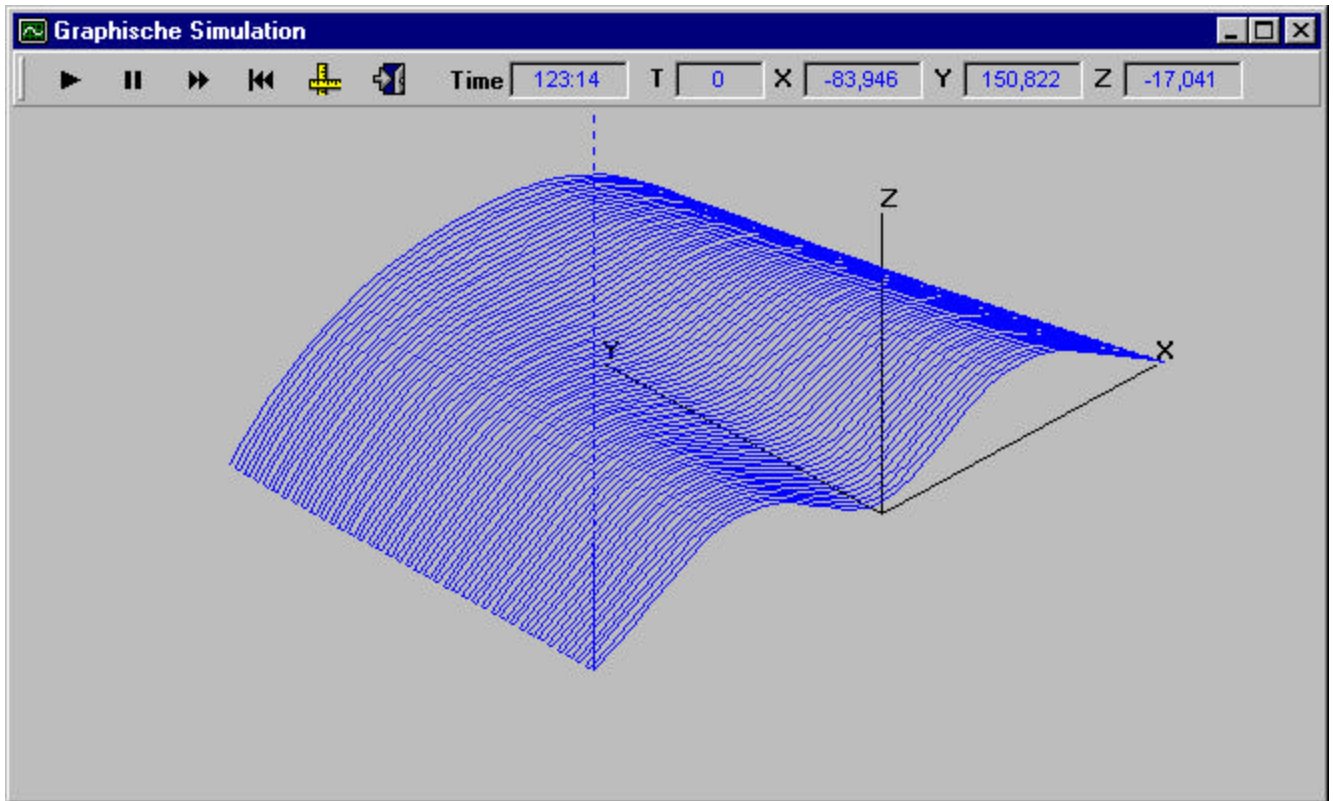
Preview...

The Preview command is used to begin the verification process. When this command is selected, a new window will be displayed. There are six buttons that are used to control the verification process.



The Preview button

For more information click on the graphic below



Run Single Step

This button is used to verify a program one block at a time. Each time the button is selected, the next program line is read.

Stop

The Stop button is used to pause the tool path verification. The verification can be restarted from the current position by choosing either the Run Single Block or Run Continuous button.

Run Continuous

When this button is selected, the program will begin verifying from the current position, executing each program block until the end of the file is reached or until the Stop button is selected.

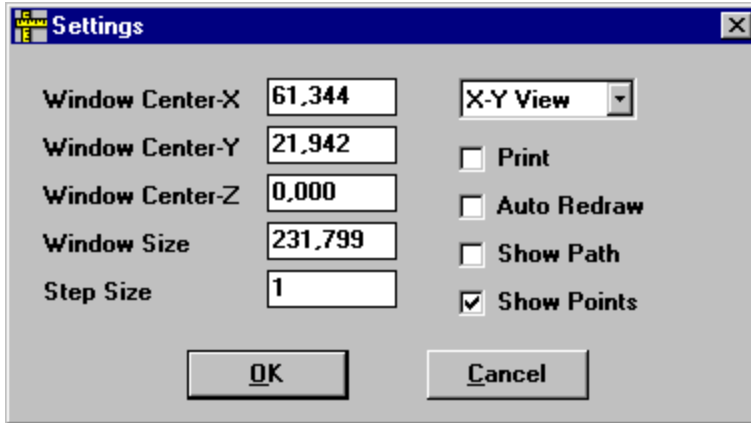
Rewind and Clear

This button is used to rewind the NC file back to its beginning while clearing the graphics display of the previous verification.

View Settings

When this button is selected, a dialog box will display, allowing the entry of the desired Window Center coordinates, and the Window Size. There is also a view selection list that allows any standard view orientation to be selected. Four check boxes are available to select options to Print, AutoRedraw, Show Path, and Show Points.

For more information click on the graphic below



Activate Text Editor

When this button is selected the text editor window is activated and the Preview window is closed.

Tool

Displays the current tool being used during the machining process.

Tm (Time)

Displays the estimated machining time.

Display Area

The display area is used to preview the toolpath of your part. The display area is dynamic. You may zoom in or out while previewing.

See also

Controlling toolpath verification by using [Preview Shortcuts](#)

Preview Shortcuts

There are some shortcuts that provide functions to ZOOM, PAN or MOVE the view of the toolpath inside the preview window.

[Ctrl + Mouse Click](#)

The simulation coordinate center is moved to the actual cursor location. Rewind and start simulation to verify toolpath with the new center position.

Alt + Mouse click

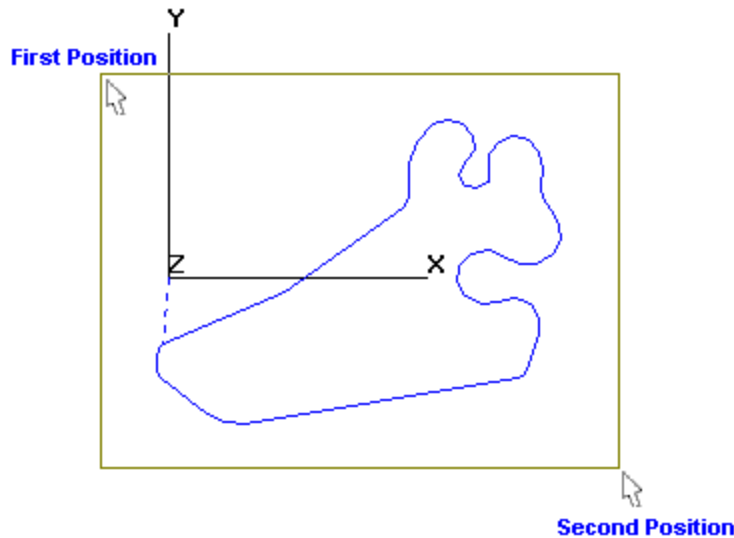
The simulation coordinate center is moved to the actual cursor location and the size of the displayed area is multiplied by factor 2. Rewind and start simulation to verify toolpath with the new center position.

Shift + Mouse click

The simulation coordinate center is moved to the actual cursor location and the window is zoomed by factor 2. Rewind and start simulation to verify toolpath with the new center position.

Selecting Free Zoom Window

For free zooming just click at one corner of the area you want to zoom. Keep the left mouse button pressed and drag the mouse over the preview window. You will see a frame moving with the cursor. When you release the mouse button the actual preview window is set automatically to the size and center of the previously selected zoom window. Rewind and start simulation to verify toolpath within the new window.



Preview / Window Centers

These settings define the absolute coordinate center and the size of the preview window for the XYZ axes in [Inch](#) or [MM](#) depending on the [Units](#) setting in the format menu.

Preview / Step Size

The Step Size option is used to control the number of program lines that will be executed each time that the Run Single Step option is used.

Preview / Auto Redraw

If this option is toggled on, you can move the preview window across the windows desktop and the previously verified toolpath is still visible on the new location because the windows operating system always keeps a bitmap copy in the memory to restore the previous view. This of course takes a lot of system resources and significantly slows down verification speed. Therefore this option is toggled to off by default.

Preview / Print

If this option is toggled on, a picture of the verified toolpath is automatically send to the printer after verification if

finished

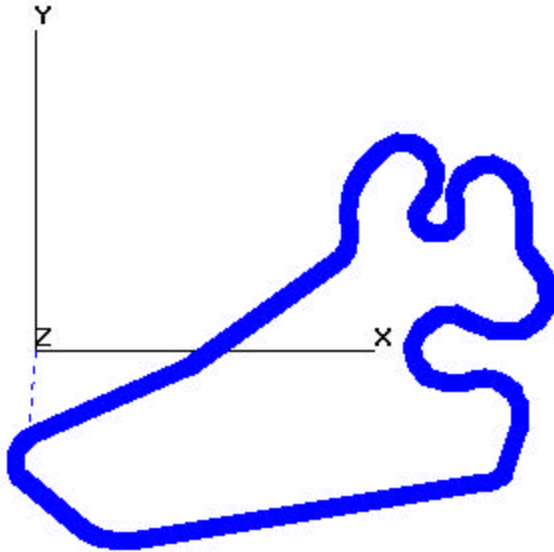
Preview / Showpath

This option allows simulation of the tool diameter if XY view in the preview window is activated. The tool diameter is defined by a special comment directly in the nc program code.

(*DIA 0.05) Comment Tooldiameter = 0.05 inch

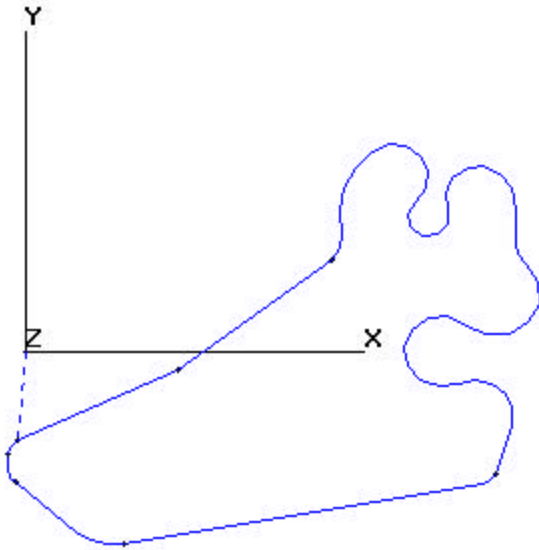
Code example:

```
%  
(MILL8-27-93)  
G91G28Z0M5  
T1M6 (*DIA0.05)      Toolchange command and special comment defining simulation diameter  
G80G90G0G17G40G98
```



Preview / Showpoints

When this option is activated, the software shows small circles on the start and endpoints of every verified element (lines, circles).



Toolpath Preview

Overview

Using Preview

[Preview...](#)

[Zooming In](#)

[Changing the Window Center](#)

[Zooming In by Two](#)

[Zooming Out by Two](#)

[Default Display Area Settings](#)

Overview of Toolpath Preview

The Preview command is used to graphically simulate the machine tool's execution of the NC program. An active window must be displayed before the Preview command can be used. Since the Convert command automatically loads the file into an active window, the Preview command can be used immediately after converting a file.

To begin previewing a file, choose the Preview command from the Preview menu. A new window will appear placed on top of the existing window. The Preview window contains six buttons along the top of the window. Four of these buttons function similar to a VCR, allowing a file to be played in single steps, played continuously, stopped, or rewound to the beginning of the file. There is also a button used to adjust the viewing parameters such as the window center coordinates, the window size, or the 3D view orientation. This button appears as an X-Y axis.

The simulation is started by choosing the button that has two arrows pointed toward the right of the screen (Run Continuous). Assuming that the view settings are adjusted correctly, the program tool path will display in the Preview window. The verification may be stopped at any time by selecting the button that contains a single square (Stop). To replay the simulation choose the button that contains two arrows pointed toward the left of the screen (Rewind and Clear) to rewind the file. To verify the program in a single step mode, choose the button that contains a single arrow pointed toward the right of the screen (Run Single Step).

Zooming In

To enlarge the display of the NC file verification, use the mouse to create a zoom box, by placing the cursor at a corner of the desired zoom window, and click the mouse, continuing to hold the mouse button down while moving the cursor to the diagonally opposite corner of the desired window. When the zoom box is sized as desired, release the mouse button and the screen will be zoomed in. Restart the simulation to verify the toolpath in the new window.

Changing the Window Center

To change the window center, place the cursor at the desired center position and click the mouse while holding down the CTRL key.

Zooming In by Two

To zoom in by a factor of two, place the cursor at the desired center position and click the mouse while holding down the SHIFT key.

Zooming Out by Two

To zoom out by a factor of two, place the cursor at the desired center position and click the mouse while holding down the ALT key.

Default Display Area Settings

When optimizing files, the optimization process automatically sets the window center and size parameters so that the part can be viewed in its entirety. Should you dynamically zoom in and out while previewing the part, you may want to keep in mind the following default settings:

US(Inch)

Window Center -X 0

Window Center -Y 0

Window Center -Z 0

Window Size 20

Metric

Window Center -X 0

Window Center -Y 0

Window Center -Z 0

Window Size 500

Toolbar Buttons

Overview
















Click on the buttons for more information.



Creates a new file.



Opens an existing file.

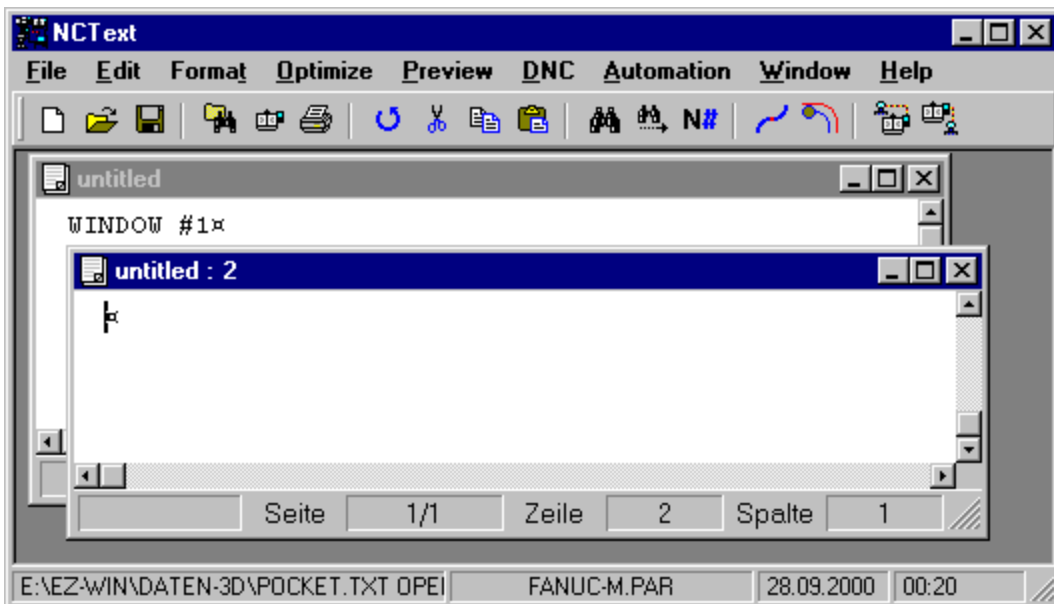
- | | |
|---|--|
|  Searches for a file in any directory. |  Saves the file. |
|  Loads machine settings |  Prints the contents of the file. |
|  Reverses changes (Undo command). |  Cuts the selection to the clipboard. |
|  Copies the selection to the clipboard. |  Pastes contents of the clipboard. |
|  Searches for specified text. |  Repeats the last search. |
|  Renumbers the file. |  Optimizes the file. |
|  Activates the toolpath preview. |  Sends data to the machine control. |
|  Receives data from the machine control. | |

Overview of Toolbar Buttons

As previously mentioned, one of the uses of this software is for NC file editing. To make the process of editing a file more convenient, a toolbar containing frequently used commands in the form of buttons, are placed across the top of the edit window just below the pull down menus. The toolbar buttons can be accessed at anytime without the need to choose a command from a menu.

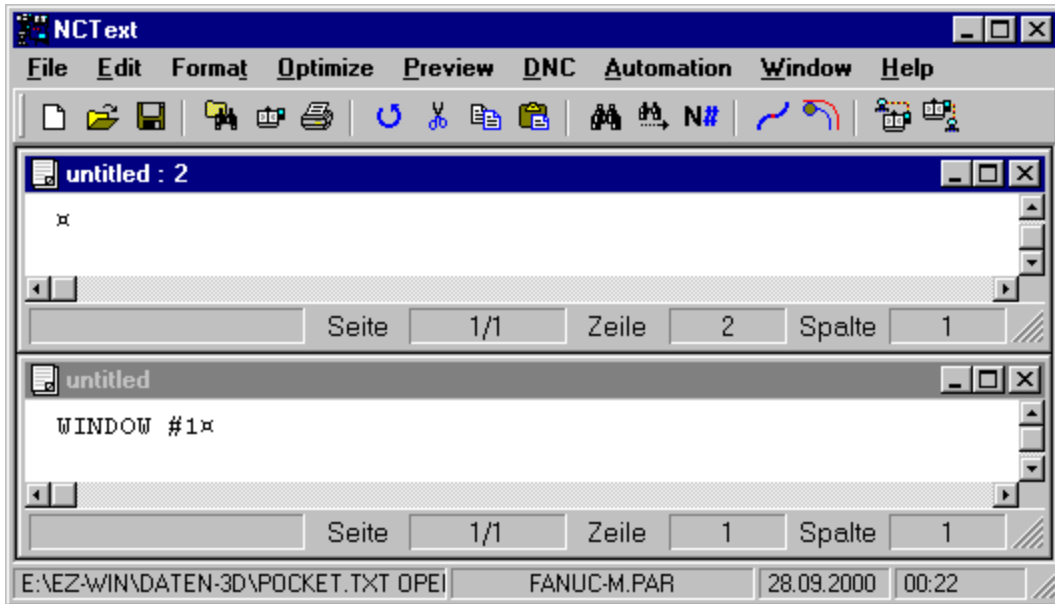
Cascade

Arranges the windows so they overlap, starting in the upper-left corner of the window. The title bar of each window remains visible, making it easy for you to select any window.



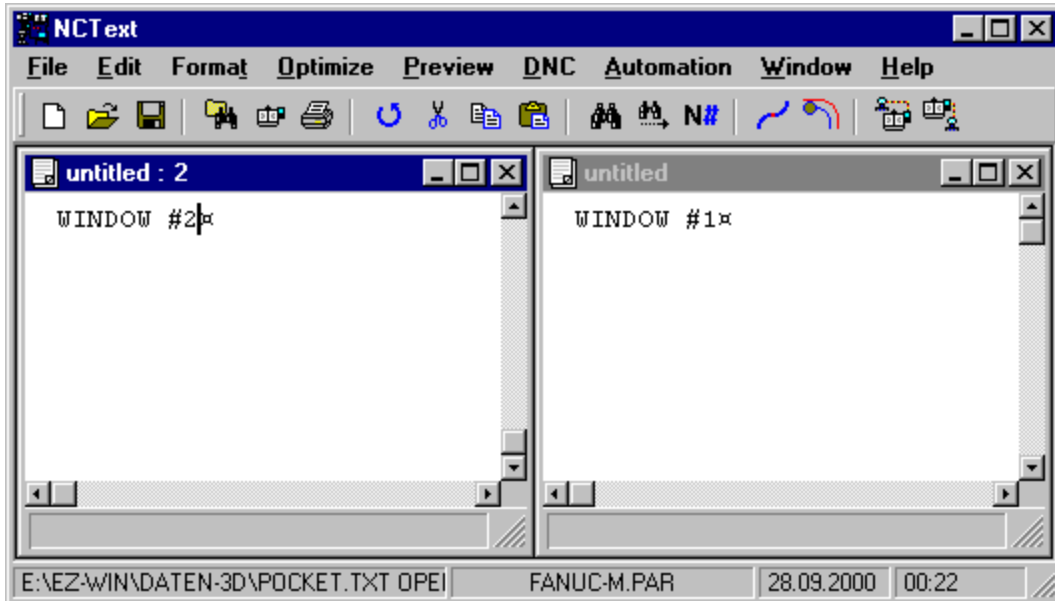
Horizontal

Arranges the windows horizontally, making it easy for you to compare the contents of individual files. The top window will be the active window. Only one window can be active at a time. To make another window active, click the title bar.



Vertical

Arranges the windows vertically, making it easy for you to compare the contents of individual files. The left window will be the active window. Only one window can be active at a time. To make another window active, click the title bar.



Display windows / Actual Filenames

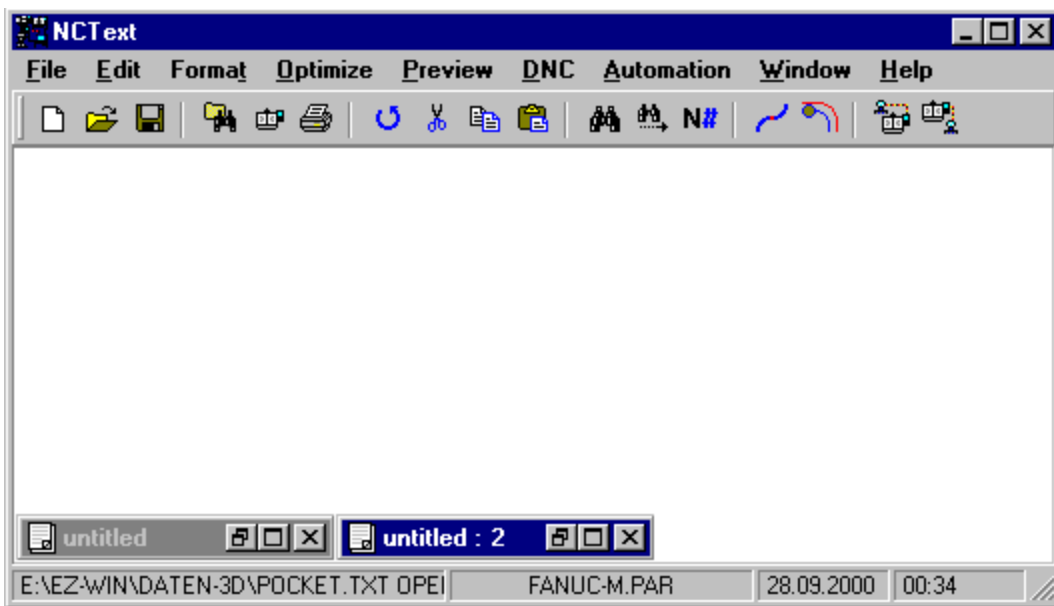
This area is used to switch between the open MDI textwindows. Normally the filenames of the actual loaded files are displayed. If no filename has been assigned yet, "untitled" will be displayed.

New Window

This option opens a new textwindow. It is the same functionality as the [New](#) command in the file menu.

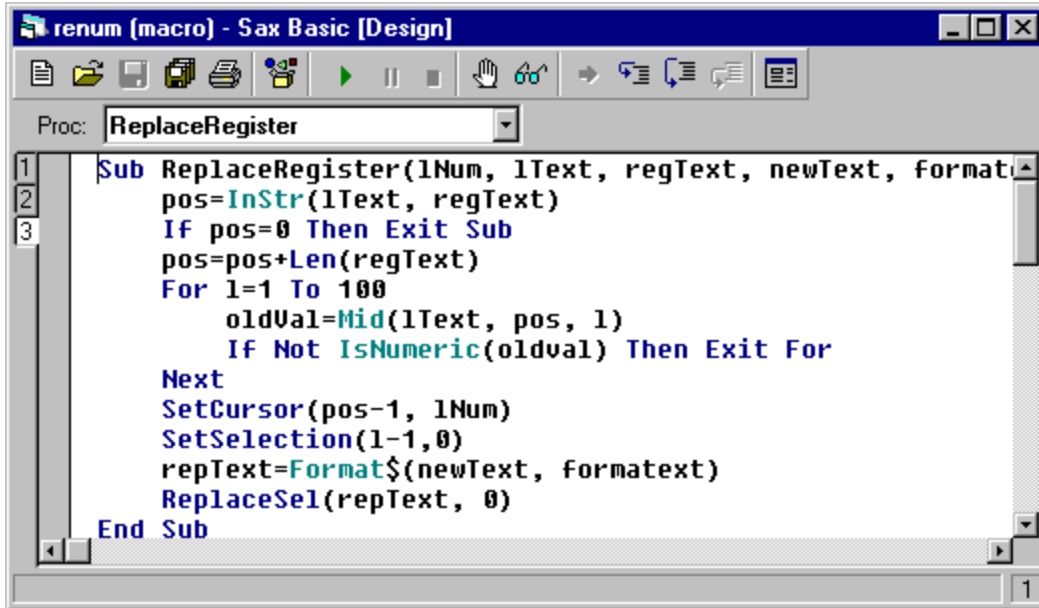
Arrange icons

This option is used to arrange the icons of all minimized textwindow in the lower leftmost area of the main window.



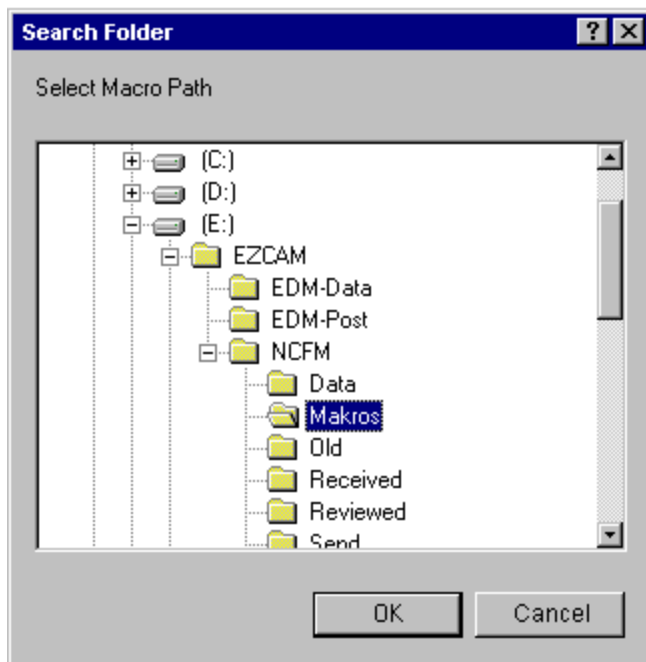
Macro Editor

This command opens the Sax Basic Editor. This is an interactive design environment for developing, testing and executing Sax Basic scripts. For more information about the editor and basic script development please see Sax basic Help.



Macro Directory

All basic script files should normally be located in one "macro" directory. This command is used to open a standard windows dialog to select and define the standard macro directory. After a directory is selected, it is scanned for *.bas files. All files that are found are displayed with their filenames inside the macro list in the automation menu from where they can be started. The selected macro directory is automatically stored in the software's initialization files to be remembered for the next sessions.



Update menu

This command is used to scan the actual macro directory for *.bas files. All files that are found are displayed with their filenames inside the macro list in the automation menu from where they can be started. Each time a new macro has been developed and stored in the actual macro directory, this command has to be used to show the name of the new macro in the macro list.

Macro List

This menu area is used to list and start macros (*.bas files) that are available in the actual macro directory. The list is updated when the software is started, a new macro directory is selected, or when the [Update menu](#) command is used.

Index

A

About EZ-DNC/Filtermax	6
About PCAM	6
Arrange icons	60
ASCII Character Codes	15
Automatic Linebreak	36
Automation Menu Commands	12

B

Bridgeport Controls	14
----------------------------------	----

C

Cascade	58
Changing the Window Center	57
Clear.....	27
Close	30
Compare to File...	28
Configuring Control Parameters	7
Connecting the Key Ring	7
Contact Support	18
Contents	19
Continue Search.....	28
Convert...	39
Converting NC Files	8
Copy	27
Corner angle.....	40
Cut.....	27

D

Default Display Area Settings	57
Delete...	32
Digitized XYZ Files.....	14
Display windows / Actual Filenames	60
DNC Menu Commands	12
DNC Restart from Cursor	22
DNC Restart Settings	21
DNC Settings...	20

E

Edit Menu Commands	10
Exit	33

F

File Menu Commands	9
Font...	35
Format Menu Commands	11

<i>G</i>	
G90 and G91 Commands	14
G-Code Settings	44
Getting Started	37
Goto Color.....	29
Gray Background	36
<i>H</i>	
Horizontal	59
<i>I</i>	
Insert	32
Installation Information	38
Installing the Software	7
<i>L</i>	
Language Acceptance	14
Load Settings	33
Load Words	33
<i>M</i>	
Macro Directory	61
Macro Editor	60
Macro List	62
Margins	35
<i>N</i>	
New.....	29, 30
New Window	60
NewTopic 1	19
<i>O</i>	
Offset Commands.....	14
Open	30
Optimization Failure	14
Optimize Menu Commands	11
Other Settings	47
Overview	7
Overview of the Menu Commands	19
Overview of Toolbar Buttons	58
Overview of Toolpath Preview	56
<i>P</i>	
Paste.....	27
Plane Restrictions	14
Preview / Arc Center Definition	50
Preview / Auto Redraw	54
Preview / Boring (Ortg.)	46
Preview / Boring (Stop)	46
Preview / Change Time	50
Preview / Chipbreak	45
Preview / Const-Spind	47
Preview / Const-Surf	47

Preview / Control Type	48
Preview / Cut- In	43
Preview / Deephole	45
Preview / Drill Depth	43
Preview / Drilling	45
Preview / Dwell.....	44
Preview / Feed.....	43
Preview / Feed-Drill.....	44
Preview / FuncTime	51
Preview / Ignore Z.....	51
Preview / Macro Adress.....	49
Preview / Macro Call	48
Preview / Macro End.....	49
Preview / Macro Repeat.....	49
Preview / Macro Start	49
Preview / Max.RPM.....	47
Preview / Orientation.....	50
Preview / Pitch	44
Preview / Programm Adress.....	48
Preview / Rapid	50
Preview / R-Arc	43
Preview / Reaming.....	45
Preview / Retract Pln	43
Preview / Retraction.....	51
Preview / Rev.Tapping.....	46
Preview / Reverse Arcs in G17/18/19	51
Preview / Reversing	50
Preview / RPM.....	44
Preview / Showpath.....	55
Preview / Showpoints	55
Preview / Skip Line 1-3	47
Preview / Special Func.....	50
Preview / Spotfacing.....	45
Preview / Start Drill next block.....	51
Preview / Step Size	54
Preview / Stop	50
Preview / Tapping.....	46
Preview / Tool Adress.....	49
Preview / Tool Change	49
Preview / Turn-Thread.....	47
Preview / Units/min	47
Preview / Units/Rev.	47
Preview / Use Pitch.....	51
Preview / Window Centers.....	54
Preview / X	43
Y	
Z-Arc.....	43
Preview / X-Axis	42
Preview / X- Inkremen.....	42
Preview / Y-Axis	42
Preview / Y- Inkremen.....	42
Preview / Z-Axis	42
Preview / Z- Inkremen.....	43

Preview Menu Commands	12
Preview Shortcuts	53
Preview	51
Preview/Print	54
Previewing NC Files	8
Print	33
Printer Setup	33
R	
Receive Text	24
Register Settings	41
Renumber	29
Renumber Settings	29
RS232 Settings	20
S	
Save	32
Save as DXF	32
Save as	32
Save Settings	33
Saving NC Files	8
Saving Your Settings	9
Search and Replace	28
Search	27
Seek	31
Seek	30
Select all	27
Send Special	26
Send Text	23
Serial Port Diagrams	17
Setting Optimization Parameters	9
Settings	38
Show CR/LF	36
Show in Editor	23
Single space	36
System Requirements	6
T	
Technical Notes	13
The Menu Commands	19
Toolbar Buttons	57
Toolpath Preview	56
U	
Undo	27
Units (Inch/Metric)	37
Update menu	62
Uppercase	37
Use Recycle Bin	37
V	
Vertical	59

W

Wait for Xon..... 23
Window Menu Commands..... 13
Word List..... 37

Z

Zooming In..... 57
Zooming In by Two 57
Zooming Out by Two 57