



FastCAM Software Series

FastCAM[®] Reference Manual

The World Standard in Profiling Software

FastCAM[®] Reference Manual

FastCAM[®] Drawing Editor & NC Generator

FastPATH[™] Automatic NC Generator

FastPLOT[™] NC Verification

Published by

FastCAM Inc
8700 West Bryn Mawr,
Suite 800 South
Chicago IL 60631-3507 USA
Telephone: (312) 715 1535
Facsimile: (312) 715 1536
Email: fastcam@fastcamusa.com

FastCAM Pty Ltd
96 Canterbury Rd
Middle Park 3206 Victoria
Australia
Telephone: (61)3 9699 9899
Facsimile: (61)3 9699 7501
Email: fastcam@fastcam.com.au

www.fastcamusa.com

Copyright © 1998, 2001, 2002 FastCAM Inc
All rights reserved
Revision 4

Information in this documentation is subject to change without notice.

All software and documentation is proprietary.

No part of this document may be reproduced or transmitted in any form or by any means,
electronic or mechanical, for any purpose, without the express written permission of
FastCAM Inc

The enclosed CD's / diskettes are serialized and may be used only by the registered user.

It may not be resold or transferred without the consent of
FastCAM Inc

FastCAM, FastCUT, FastFRAME, FastSHAPES, FastNEST, FastTRACK, FastAIR, FastCOPY
FastPART and FastHULL are all registered trademarks of
Fagan Microprocessor Systems Pty Ltd

FastPUNCH, FastBEAM, FastPLOT, FastPATH, FastRING and FastEST are all Trademarks of
Fagan Microprocessor Systems Pty Ltd

Windows is a registered trademark of
Microsoft Corporation in the United States and other countries

All other trademarks are acknowledged.

Contents

INSTALLING FASTCAM®	14
System Requirements.....	14
Running the FastCAM® Setup Program.....	14
Installation	14
INTRODUCTION TO FASTCAM®	20
INFINITE LINE CONSTRUCTION	20
TRIMMING.....	20
POINT PLACEMENT AT INTERSECTIONS	21
NON-SEQUENTIAL PART CONSTRUCTION	21
LEAD-INS, LEAD-OUTS	21
FAIRED LINES, ELLIPSES	21
BULK DXF & IGES INTERFACE.....	21
CAD CLEAN & CAD COMPRESS	22
FastPATH AUTOMATIC PROCESS PATHING.....	22
FASTNEST.....	22
MANUAL NESTING.....	22
MACRO SUB-ROUTINES	22
INPUTS	23

ABOUT THE MANUAL.....	23
CONVENTIONS USED IN THIS MANUAL.....	24
REPRESENTING MENU ITEMS.....	24
Representing keys	24
REPRESENTING LISTS OF INSTRUCTIONS	25
REPRESENTING IMPORTANT NOTES	25
1-1 GETTING STARTED	26
1-2 YOUR SECURITY KEY	26
1-3 RUNNING THE <i>FASTCAM</i> DRAWING PROGRAM	27
1-4 DRAWING PARTS.....	28
1-4-1 SCREEN DISPLAY.....	28
1-4-2 MENU-BAR.....	28
1-5 USING THE MOUSE.....	29
1-5-1 MOUSE PRACTICE.....	30
1-6 SCREEN ORIENTATION	33
1-6-1 CARTESIAN CO-ORDINATE SYSTEM.....	34
1-6-2 ANGLES in DEGREES.....	35
1-7 ABSOLUTE Programming	35
1-8 INCREMENTAL Programming.....	42
1-9 SETTING POINTS	51
1-9-1 CONTROL POINTS	52
1-10 LINE MENU	60

1-11 ARC MENU.....	61
1-11-1 FULL CIRCLE.....	61
1-11-2 ARCS	61
1-11-3 FILLET BLENDS	62
1-12 SAVING AND RESTORING FILES.....	63
1-13 THE VIEW MENU.....	67
1-14 THE ERASE MENU.....	68
1-15 TRIMMING ENTITIES	69
1-15-1 TRIMMING LINES.....	69
1-15-2 TRIMMING ARCS	69
1-16 Producing a PROGRAM PATH.....	69
FASTCAM® QUICK START	72
1-17 NESTING THEORY AND APPLICATION	77
1-17-1 INTRODUCTION.....	77
1-17-2 FastNEST INTERACTIVE Nesting	77
1-17-3 FastNEST AUTOMATIC Nesting	77
2-1 FILES MENU.....	80
2-1-1 FILES MENU Save.....	80
2-1-2 FILES MENU Restore	81
2-1-3 FILES MENU New.....	81
2-1-4 FILES MENU DXF Restore.....	81
2-1-5 FILES MENU DXF Save	83
2-1-6 FILES MENU IGES Restore	83
2-1-7 FILES MENU IGES Save	83
2-1-8 FILES MENU HPGL Restore.....	83
2-1-9 FILES MENU DSTV Restore.....	84
2-1-10 FILES MENU StruCAD Restore.....	84
2-1-11 FILES MENU Plot Screen.....	84
2-1-12 FILES MENU Scale Plot.....	85
2-1-13 FILES MENU Part Data	85
2-1-14 FILES MENU Exit	85

2-2 LINE MENU	86
2-2-1 LINE MENU Next point/Point menu	87
2-2-2 LINE MENU Length & Angle	87
2-2-3 LINE MENU Two sides.....	88
2-2-4 LINE MENU Two points	88
2-2-5 LINE MENU Infinite Line	89
2-2-6 LINE MENU ANGLE MENU	90
2-2-6-1 ANGLE MENU Direct entry.....	90
2-2-6-2 ANGLE MENU Two sides	90
2-2-6-3 ANGLE MENU Same as	90
2-2-6-4 ANGLE MENU Relative to	90
2-2-7 LINE MENU Tan to circle	91
2-2-8 LINE MENU Tan to 2 Circles.....	91
2-2-9 LINE MENU Parallel at dist.....	91
2-2-10 LINE MENU at Rt Angles	92
2-2-11 LINE MENU Box	92
2-2-12 LINE MENU Bisect Angle	92
2-3 ARC MENU	93
2-3-1 ARC MENU Full circle	93
2-3-2 ARC MENU ARC.....	93
2-3-2-1 Start, Point, End	93
2-3-2-2 Start, Cent, End	94
2-3-2-3 Start, End, Radius	94
2-3-2-4 Cen, Rad, A1, A2.....	94
2-3-2-5 C, R, A1, Sweep.....	94
2-3-2-6 Half Circle	94
2-3-2-7 Tangent 3 Lines.....	95
2-3-3 ARC MENU Fillet-Blend.....	96
2-3-4 ARC MENU Ring	96
2-4 POINT MENU	97
2-4-1 POINT MENU Screen position	97
2-4-2 POINT MENU Absolute Co-ords	98
2-4-3 POINT MENU Incremental Co-ords	98
2-4-4 POINT MENU Length And Angle	98
2-4-5 POINT MENU Midpoint	98
2-4-6 POINT MENU Control Point	99
2-4-7 POINT MENU Intersection.....	99
2-4-8 POINT MENU On Entity	99

2-5 CONSTRUCTS MENU	100
2-5-1 CONSTRUCTS MENU Ellipse.....	100
2-5-2 CONSTRUCTS MENU Faired Line.....	100
2-5-3 CONSTRUCTS MENU CORNERS MENU.....	101
2-5-3-1 CORNERS MENU Fillet.....	101
2-5-3-2 CORNERS MENU Chamfer.....	102
2-5-3-3 CORNERS MENU Arc Chamfer.....	102
2-5-3-4 CORNERS MENU SharpCorners.....	102
2-5-3-5 CORNERS MENU Loop.....	102
2-5-3-6 CORNERS MENU Notch Corner	102
2-5-4 CONSTRUCTS MENU Offset Contour.....	103
2-5-5 CONSTRUCTS MENU Line Notch.....	103
2-5-6 CONSTRUCTS MENU Insert Gap.....	103
2-5-7 CONSTRUCTS MENU Bridging	103
2-5-8 CONSTRUCTS MENU Faired Spiral.....	104
2-5-9 CONSTRUCTS MENU Contour Points	105
2-5-10 CONSTRUCTS MENU Kerf Compensate.....	105
 2-6 ERASE MENU.....	 107
2-6-1 ERASE MENU Items 1 to 5.....	107
2-6-2 ERASE MENU Window	108
2-6-3 ERASE MENU Contour	108
2-6-4 ERASE MENU Duplicates only	108
2-6-5 ERASE MENU CAD Clean	109
2-6-6 ERASE MENU CAD Compress	109
2-6-7 ERASE MENU Entries & Exits	110
2-6-8 ERASE MENU Last.....	110
2-6-9 ERASE MENU Part.....	111
2-6-10 ERASE MENU Plate Data.....	111
2-6-11 ERASE MENU Undo.....	111
2-6-12 ERASE MENU Redo.....	111
 2-7 BLOCK MENU	 112
2-7-1 BLOCK MENU Define.....	113
2-7-1-1 BLOCK SELECT MENU All.....	113
2-7-1-1 BLOCK SELECT MENU Lines	113
2-7-1-3 BLOCK SELECT MENU Arcs.....	114
2-7-1-4 BLOCK SELECT MENU Points	114
2-7-1-5 BLOCK SELECT MENU Text.....	114
2-7-1-6 BLOCK SELECT MENU Contour	114

2-7-1-7	BLOCK SELECT MENU Parts.....	114
2-7-1-8	BLOCK SELECT MENU Window	115
2-7-1-9	BLOCK SELECT MENU Character.....	115
2-7-3	BLOCK MENU Display.....	115
2-7-4	BLOCK MENU Remove Blocking.....	115
2-7-5	BLOCK MENU Move.....	116
2-7-6	BLOCK MENU Copy	116
2-7-7	BLOCK MENU 2 Point Copy	116
2-7-8	BLOCK MENU Rotate.....	117
2-7-9	BLOCK MENU Reflect.....	117
2-7-10	BLOCK MENU Expand.....	117
2-7-11	BLOCK MENU Rect Array.....	118
2-7-12	BLOCK MENU Circ Array.....	118
2-7-13	BLOCK MENU Write Block.....	119
2-7-14	BLOCK MENU Read Block.....	119
2-7-15	BLOCK MENU Extract part.....	120
2-7-16	BLOCKS MENU Explode Parts.....	120
2-8	VIEW MENU.....	122
2-8-1	VIEW MENU Indicate Corners.....	122
2-8-2	VIEW MENU Indicate Center.....	123
2-8-3	VIEW MENU Enter Center.....	123
2-8-4	VIEW MENU Zoom In.....	123
2-8-5	VIEW MENU Zoom Out	124
2-8-6	VIEW MENU Autoscale.....	124
2-8-7	VIEW MENU Repeat.....	124
2-8-8	VIEW MENU Previous View.....	124
2-8-9	VIEW MENU Change Display.....	125
2-8-9-1	DISPLAY MENU Auto Dimension.....	126
2-8-9-2	DISPLAY MENU Axes.....	126
2-8-9-3	DISPLAY MENU Direction	126
2-8-9-4	DISPLAY MENU DXF Layers.....	126
2-8-9-5	DISPLAY MENU Toolbar.....	126
2-8-9-6	DISPLAY MENU Rapid.....	127
2-8-9-7	DISPLAY MENU Filenames	127
2-8-9-8	DISPLAY MENU Sequence Numbers.....	127
2-8-9-9	DISPLAY MENU Pierce & Stop.....	127
2-8-9-10	DISPLAY MENU Screen Label.....	127
2-8-9-11	DISPLAY MENU Plotter Label.....	128
2-8-9-12	DISPLAY MENU Plotter Pen Width	128
2-8-10	DISPLAY MENU Window	129

2-8-10-1	WINDOW MENU Store Current	129
2-8-10-2	WINDOW MENU Recall	129
2-8-10-3	WINDOW MENU Show All	129
2-8-10-4	WINDOW MENU Define	129
2-8-10-5	WINDOW MENU Clear All	130
2-8-11	VIEW MENU Scale only	130
2-9	UTILITY MENU	131
2-9-1	UTILITY MENU Change Units	131
2-9-2	UTILITY MENU Break	132
2-9-3	UTILITY MENU List Control	132
2-9-4	UTILITY MENU List Setup	132
2-9-5	UTILITY MENU Bevel Calculator	132
2-9-6	UTILITY MENU List Entities	132
2-9-7	UTILITY MENU Edit Entity	133
2-9-8	UTILITY MENU Stretch	134
2-9-9	UTILITY MENU Pts to holes	134
2-10	TEXT MARKING	135
2-10-1	TEXT MARKING Add	136
2-10-2	TEXT MARKING Edit	136
2-10-3	TEXT MARKING Move	137
2-11	CAD LAYERS	138
2-11-1	CAD LAYERS Show	140
2-11-2	CAD LAYERS Add Layers	141
2-11-3	CAD LAYERS Remove Layers	141
2-12	VERIFY	142
2-12-1	VERIFY MENU Identify	142
2-12-2	VERIFY MENU Distance	142
2-12-3	VERIFY MENU Contour	142
2-12-4	VERIFY MENU Check Dimensions	143
2-12-5	VERIFY MENU Remove last	143
2-12-6	VERIFY MENU Remove all	143
2-12-7	VERIFY MENU Show Entity	143
2-13	TRIM	143
2-13-1	Trim	144
2-13-2	Smart Trim	144

2-14 PROGRAM PATH	144
2-15 CONTROL POINTS	145
2-15-1 CONTROL POINTS Control Points	145
2-15-2 CONTROL POINTS License	145
2-15-3 CONTROL POINTS About	145
2-15-4 CONTROL POINTS NC Change	146
2-15-5 CONTROL POINTS NC Display	146
2-15-6 CONTROL POINTS Reset Files	146
2-16 LANGUAGE	146
2-17 NC PATH MENU	147
2-17-1 NC PATH MENU Next Path.....	148
2-17-2 OPERATION MENU.....	148
2-17-2-1 OPERATION MENU Process.....	149
2-17-2-2 NEXT PATH KERF MENU:.....	149
2-17-3 NC PATH MENU Delete all Paths	151
2-17-4 NC PATH MENU Delete last Path	151
2-17-5 NC PATH MENU Redefine Path.....	151
2-17-6 NC PATH MENU Output NC code	151
2-17-7 NC PATH MENU Status	152
2-17-8 NC PATH MENU Part Description	152
2-17-9 NC PATH MENU View path data	153
2-17-10 NC PATH MENU View	153
2-17-11 NC PATH MENU FastPLOT.....	153
2-17-12 NC PATH MENU FastPATH.....	154
2-17-13 NC PATH MENU Restore Drawing	154
2-17-14 NC PATH MENU Pathed DXF.....	155
2-17-15 NC PATH MENU Round All Corners	155
3-1 FastPATH Concepts	157
3-2 FastPATH One Process Quick Set-up	158
3-3 FastPath Option Menu.....	164
3-3-1 FastPATH Start FastPATH	164
3-3-2 FastPATH Batch Parts	164
3-3-2-1 BATCH PARTS Batch Parts.....	164
3-3-2-2 BATCH PARTS Select Parts	165

3-3-2-3	BATCHPARTS Select the Output Path	165
3-3-2-4	BATCHPARTS Automatic Overwrite	165
3-3-2-5	BATCHPARTS FastNEST Cutlist (Optional - Cutlist)	165
3-3-2-6	BATCHPARTS Cutlist Name (Optional - Cutlist)	165
3-3-2-7	BATCHPARTS Part Separation (Optional - Cutlist)	166
3-3-2-8	BATCHPARTS Plate Size (Optional - Cutlist)	166
3-3-2-9	BATCHPARTS Verify Part	166
3-3-3	FastPATH Edit Settings	166
3-3-4	FastPATH Save Settings	166
3-3-5	FastPATH Restore Settings	166
3-3-6	FastPATH Restore Drawing	167
3-3-7	FastPATH Output NC	167
3-3-8	FastPATH Exit	167
3-4	PROCESS	168
3-4-1	PROCESS Selected Processes	168
3-4-2	PROCESS Kerf	169
3-4-3	PROCESS Side	169
3-4-4	PROCESS Available Processes	170
3-4-5	PROCESS Arrow	170
3-4-6	PROCESS Reset	170
3-4-7	PROCESS Description	170
3-4-8	PROCESS Accept	171
3-4-9	PROCESS Cancel	171
3-5	INTERNAL ENTRY	172
3-5-1	INTERNAL ENTRY Internal Same As External	172
3-5-2	INTERNAL ENTRY Pierce in Corner	173
3-5-3	INTERNAL ENTRY Break Longest Entity	173
3-5-4	INTERNAL ENTRY Center Pierce	173
3-5-5	INTERNAL ENTRY Type	174
3-5-6	INTERNAL ENTRY Length	174
3-5-7	INTERNAL ENTRY Angle	174
3-5-8	INTERNAL ENTRY On / Off	174

3-6 EXTERNAL ENTRY.....	175
3-6-1 EXTERNAL ENTRY Position.....	175
3-6-2 EXTERNAL ENTRY Corner Pierce.....	176
3-6-3 EXTERNAL ENTRY Type.....	176
3-6-4 EXTERNAL ENTRY Length.....	176
3-6-5 EXTERNAL ENTRY Angle	177
3-6-6 EXTERNAL ENTRY On / Off.....	177
3-7 NEST SEQUENCE.....	178
3-7-1 Internal Sweep.....	178
3-7-2 Nest Start.....	179
3-7-3 Nest Sweep.....	179
3-7-4 Description	179
3-7-5 Accept.....	179
3-8 Nest Options	180
3-8-1 Hole Avoidance	180
3-8-1-1 Rectangular Rapids.....	181
3-8-1-2 Straight Rapids	181
3-8-2 Sort Marking.....	181
3-8-2 Contour Gap	181
3-8-3 Outline Shown	182
3-8-3-1 None	182
3-8-3-2 Nest Plate	182
3-8-3-3 Silhouette	183
3-9 Specials	183
3-9-1 Tag.....	184
3-9-1-1 All (fully tagged).....	184
3-9-1-2 Inside Tag.....	184
3-9-1-3 Outside Tag	185
3-9-2 Pre Pierce	185
3-9-3 Small Hole	186
3-10 Stitch Kerf	187
3-10-1 Kerf Compensation.....	187
3-10-2 Stitch Cut.....	188
3-10-2-1 Stitch Distance.....	188
3-10-2-2 Stitch Width.....	188

4-1 Fast Plot NC FILE	189
4-1-1 NC FILE SCAN	189
4-1-2 NC FILE EDIT.....	190
4-1-3 NC FILE PLOT.....	190
4-1-4 NC FILE COST.....	190
4-1-5 NC FILE LIST.....	192
4-1-6 NC FILE Kerf Compensate.....	192
4-1-7 NC FILE EXIT.....	192
 4-2 FastPLOT NC UTILITIES	 193
4-2-1 NC UTILITIES CONVERT to CAM.....	193
4-2-2 NC UTILITIES EXPAND NC.....	193
4-2-3 NC UTILITIES SORT NC.....	194
4-2-4 NC UTILITIES EXPLORER.....	195
4-2-5 NC UTILITIES FASTCAM DRAWING.....	195
4-2-6 NC UTILITIES ENABLE OPTION.....	195
4-2-7 NC UTILITIES LICENSE.....	195
4-2-8 NC UTILITIES CONVERT TO DXF	195
 4-3 FastPLOT NC PROFILE.....	 196
4-3-1 NC PROFILE CHANGE NC.....	196
4-3-2 NC PROFILE DISPLAY NC.....	196
4-3-3 NC PROFILE LICENCE.....	196
4-3-4 NC PROFILE ABOUT.....	196
4-3-4 NC PROFILE TRANSLATE NC.....	197
4-3-5 NC PROFILE BOX NEST	197
4-3-6 NC PROFILE RESET FILES	197

FASTCAM FILE STRUCTURE.....	198
Custom Marking fonts with_Fonts.cam.....	203
ADDITIONAL SOFTWARE MODULES BY FASTCAM®	257
FastCAM® Text Marker: Marking and Identification on parts	259
INTERNATIONAL OFFICES	261
FASTCAM RESELLERS	261
GLOSSARY	262
INDEX	272
World Wide Web http://www.fastcamusa.com.....	276

Software Installation Guide

Installing FastCAM®

System Requirements

To use *FastCAM®*, you need as a minimum:

- A personal computer with a Pentium II 266 MHz Processor or higher.
- A CD-ROM drive.
- 10 Megabytes of free Hard Drive space.
- 64 Megabytes of RAM.
- Microsoft Windows 95/98/ME/XP or Windows NT4/2000.
- A VGA (800 x 600) resolution graphics card or higher.

Running the FastCAM® Setup Program

The Setup program installs all the necessary files for *FastCAM®* to operate. Program directories and program Icons are also created through the Setup program.

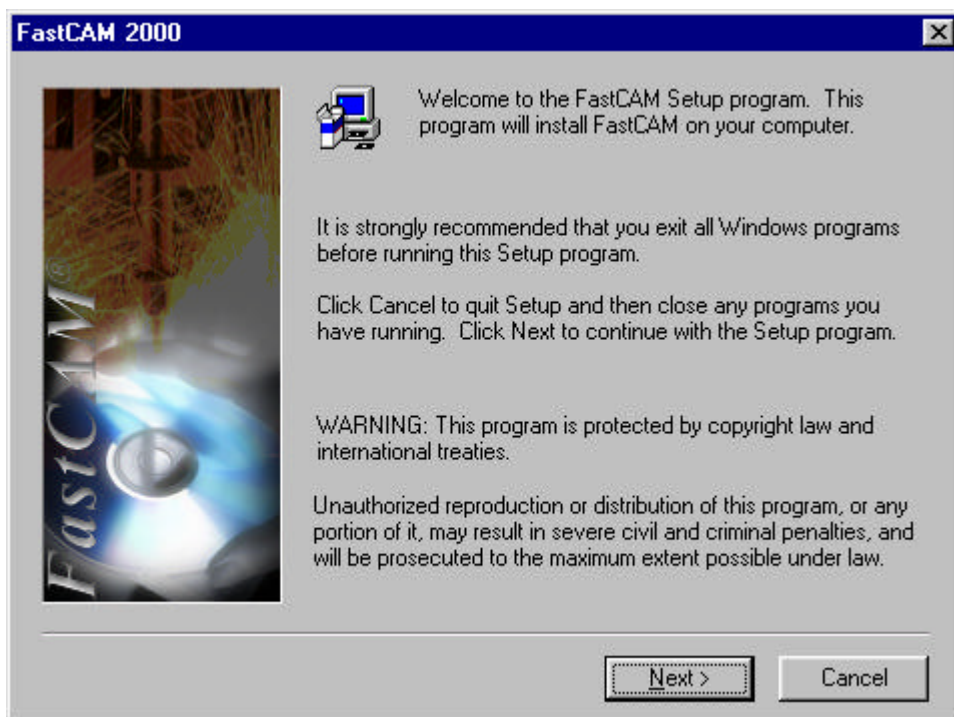
Installation

1. Run Windows 95/98 or NT.

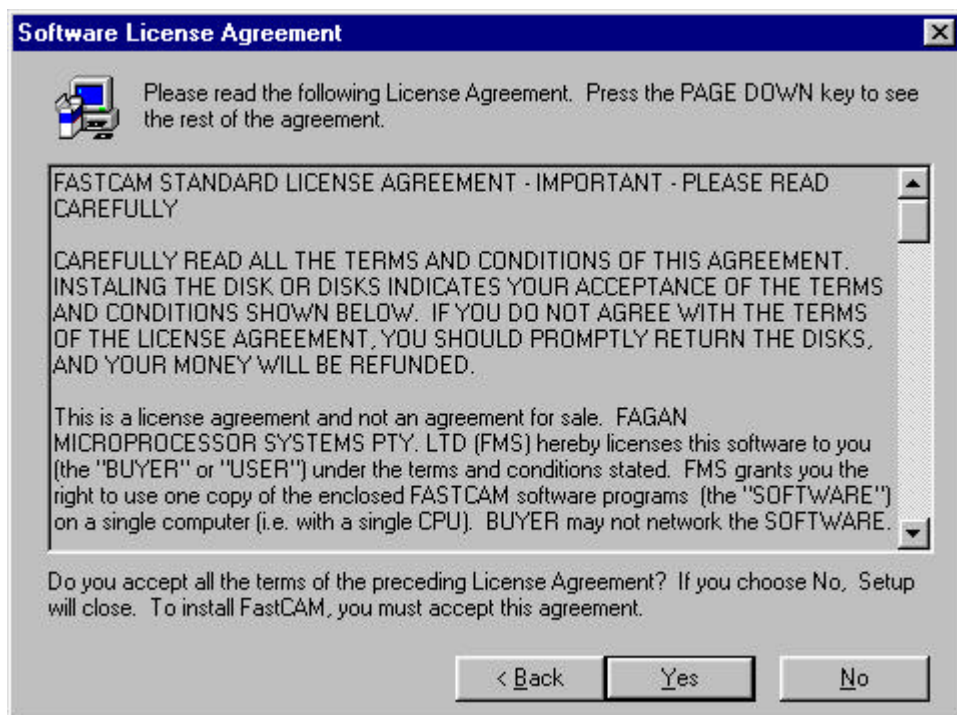
NOTE: Before trying to install FastCAM®, make sure all other Windows Applications are closed.

1. Insert your FastCAM® CD , into the appropriate drive.
2. The FastCAM CD should automatically run.
3. Follow the prompts displayed on the screen to get to the FastCAM Installation; and click on the INSTALL button.

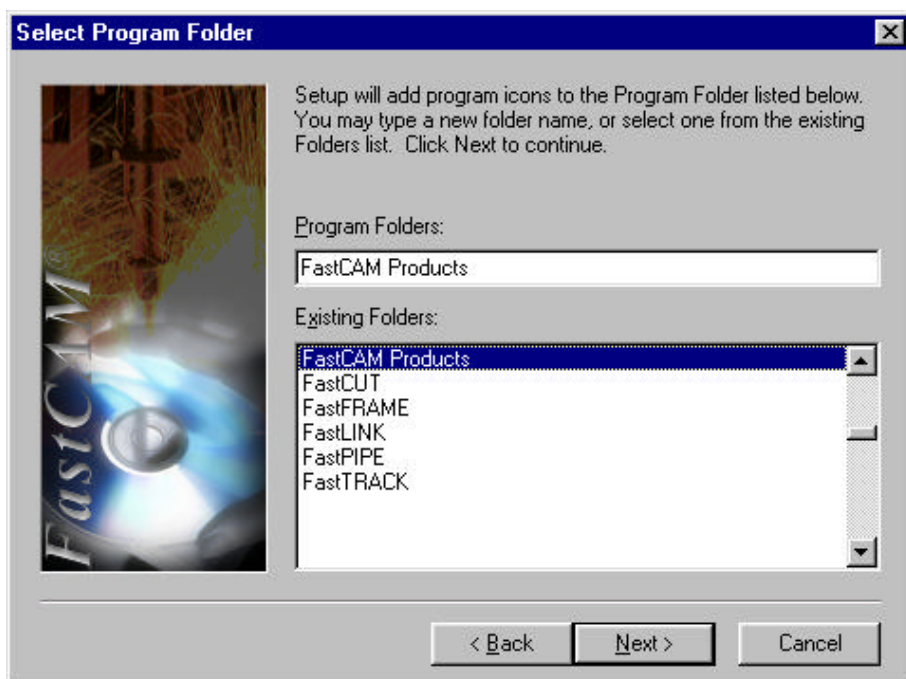
4. Read the information on the Welcome screen, and then click the NEXT button.



5. Read the License agreement fully, and click YES to accept the agreement and continue with the installation. If you do not agree with the agreement click NO, and the installation will end.



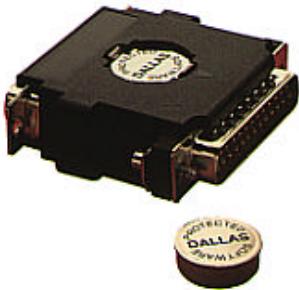
6. You will then be asked to select a Program Folder name for installation. To accept the default name, click the **NEXT** button. To install *FastCAM*® under a different folder name, type over the existing name and then click **NEXT**.



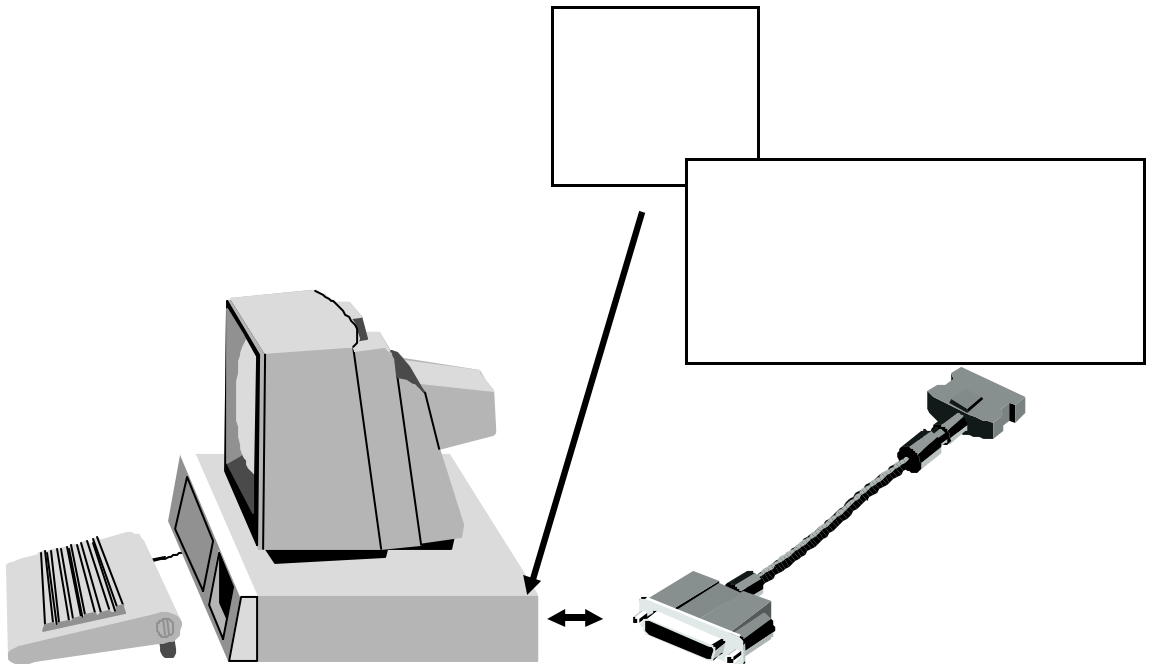
Follow the instructions on the screen

Once the installation has been completed, you can then start the applications by either:

- Accessing the programs via the **START** button, Programs, and then FastCAM 32BIT, or
- double-clicking the **ICON** on the Desktop.



NOTE: Before the FastCAM[®] programs can be run you “MUST” attach your FastLOC DONGLE to your Printer PORT. If a printer is connected to your PC, attach it to the “LPT1 Printer Port” where the printer is connected now. You then attach your Printer Cable to the FastLOC DONGLE.



Installing FastLOC Dongle

Step 1 : Unplug the printer cable from the printer port in the back of the computer.

Step 2 : Insert the FastLOC® security dongle into the printer port in the back of the computer where the printer cable just came from.

|

Step 3 : Plug the printer cable into the back of the FastLOC® security dongle. The FastLOC® security dongle is now installed. *If a printer is to be attached to the Dongle, then it needs to be turned on and On-Line.*

INTRODUCTION TO FastCAM®

FastCAM® is an advanced line drawing system that has been developed specifically for NC controlled, two dimensional cutting equipment. The program is menu & Icon driven, and easy to use.

The author has developed *FastCAM®* with the Philosophy of "dead obvious" in mind. In many of the constructions the computer will know which part of an arc, or what fillet blend is required, by how the point is selected. This is in contrast to most CAD systems that play twenty questions to determine the correct construction. Because of the "dead obvious" philosophy, the drawing system is more interactive than interrogative.

Some *FastCAM®* features that set it apart from other systems are described below.

INFINITE LINE CONSTRUCTION

Infinite lines are useful as construction lines to help locate unknown points, they can be placed at any angle through any point.

TRIMMING

Lines, and arcs, can be trimmed as required.

POINT PLACEMENT AT INTERSECTIONS

A point can be placed at the intersection of lines, arcs, circles or any combination of these entities. A point may also be placed at the projected intersection of any two entities.

NON-SEQUENTIAL PART CONSTRUCTION

The FastCAM drawing can be constructed in any order and the program path is set after part is completed.

LEAD-INS, LEAD-OUTS

Wide variety of lead-in, lead-out placements, can be constructed. A lead-in or lead-out can be placed at any point on the part.

FAIRED LINES, ELLIPSES

FastCAM® will draw truly faired lines and ellipses, through a series of known points.

BULK DXF & IGES INTERFACE

Bulk 2D-CAD files in a DXF or IGES format can be read into *FastCAM*®, modified and out-put as a DXF or IGES file or as an NC part file.

CAD CLEAN & CAD COMPRESS

The CAD Clean function is used to clean up any erroneous and unwanted entities imported into the *FastCAM*® system from a CAD file. The CAD Compress function is used to smooth out and reduce the number of entities in a CAD drawing imported into *FastCAM*®.

FastPATH AUTOMATIC PROCESS PATHING

The FastPATH automatic pathing function allows the user to automatically assign cutting paths to individual or bulk parts at the click of a mouse button. This function can also be used to path an entire nest of parts after they have been nested as CAM files.

FASTNEST

FastNEST allows rapid placement of large numbers of parts on a plate. These parts are looked at as their TRUE shape. After the parts have been placed on the plate specified, manual nesting can be used to optimize the plate usage.

MANUAL NESTING

Easy to use, manual nesting features for efficient plate utilization, complete with dynamic Drag & Drop placement of parts and Real-Time rotation.

MACRO SUB-ROUTINES

FastCAM® supports macro routine programming. This is where repetitive blocks of code can be written to a macro or sub-program and executed by a macro call. All controllers are not able to use macro's.

INPUTS

Numeric inputs for part construction can be in Metric or Imperial feet, INCH or fraction.

ABOUT THE MANUAL

This manual has been written to provide a reference guide to the *FastCAM*® one step programming system. It is a menu item by menu item description with examples in the appropriate places. It is broken down into three main sections, *FastCAM*®, FastPLOT and in a separate manual, FastNEST. *FastCAM*® is the drawing section of the program while FastPLOT is the verification program and FastNEST is the nesting section. If you have a question on how a function in *FastCAM*® or FastPLOT works, look up the menu item for an explanation.

This manual was written to accommodate the United States cutting market where INCH measurements are used primarily instead of METRIC. The examples shown throughout the manuals are based in the Imperial measurement System. Metric sizes are shown In "()" after the Inch size. *FastCAM*® will accommodate both inch and metric calculations. The type of units your *FastCAM*® software is currently configured for was determined by the installation of the software.

The primary purpose of this reference manual, is to inform the user of the functions available in the *FastCAM*® and FastPLOT programs. Please refer to the separate FastNEST manual for more information on this product.

Conventions used in this Manual

Throughout this manual information that is important to your understanding of FastCAM/FastPLOT is highlighted in certain ways. The items that are emphasised in this way include:

- menu items
- keys
- mouse buttons
- screen messages
- important notes.

Representing menu items

Each menu item is capitalized at the beginning of the description. The general form is,

2-1-5 LINE MENU INFINITE LINE

The **2** is the Chapter number,

The **1** is the Main Menu item Number,

The **5** is the Sub Menu item Number,

In this case it is *FastCAM*®

In this case Line

In this case Infinite Line

Representing keys

When you need to type a specific key, or click a particular button, the instruction is shown as:

ENTER

This means press the Enter button.

Representing lists of instructions

Lists of step by step instruction have been detailed with the use of numbered lists. To complete the task you simply follow the instructions in sequential order. For example:

1. Place the mouse arrow on the **Point** Menu option and press the **LEFT** mouse button.
1. Place the highlight bar displayed in the **Point Menu** on **Absolute co-ords** and press the **LEFT** mouse button.
1. At the prompt, press the **ENTER** button, on the box.

Representing important notes

A writing symbol is used to draw your attention to important notes in the text.

For example:



Absolute zero is still the point in the middle of the screen where the *X* and *Y* axes cross.

1-1 CHAPTER 1 - GETTING STARTED

This section will give some direction to the beginner trying to figure out what *FastCAM*® is designed to accomplish, and how the system relates to the cutting process.

The first concept that must be grasped is the over all view of the process of part cutting and where *FastCAM* fits into that process. Look at *FastCAM* as a translator between the drawing and the cutting machine. In the *FastCAM* drawing section, the programmer will draw the part on the graphic screen, set a cutting path, and generate a NC file containing Numerical Control (NC) language code. Once the NC code is generated, parts can either be taken to the cutting machine or used in the *FastPLOT* / *FastNEST* section of *FastCAM*. (*Depending on your program configuration - refer to your Software supplier*)

FastPLOT allows for the verification and editing of the parts.

FastNEST allows for the automatic/interactive nesting of all parts. If a nest is constructed, the end result is a file containing the NC code for the cutting of the nested parts.

For an overview of the *FastCAM* drawing program, please refer to the Flow chart in Appendix D.

1-2 YOUR SECURITY KEY

Your Security key (*FastLOC*) is your company's software protection device. Your *FastCAM* software programs cannot be accessed without this device being fitted. Your Security key must be plugged into a Parallel port on the rear of your PC. Most *FastCAM* systems are supplied with only one parallel port, named "LPT1"; however the *FastLOC* may be placed on any parallel port. If your *FastCAM* system hardware also included a printer, it is

necessary to place your Security key into LPT1 and attach your printer cable to the open side of the key.



If a printer is attached to your Security key, the Printer MUST be on-line to make the key operational.

1-3

RUNNING THE *FASTCAM*DRAWING PROGRAM

- 1.) Double-Click the *FastCAM* Icon on the Windows Desktop, with your mouse, **OR**



- 2.) Press the Start button, located on the Windows Taskbar; proceed to the Programs Menu, and then across to the *FastCAM* Menu, where you select the *FastCAM* program Icon.



The FastCAM program will need to be installed from the CD ROM before it can be started. Your first screen should contain a FastCAM Bitmap image.

1-4 DRAWING PARTS

1-4-1 SCREEN DISPLAY

All of your menu items and Icons will appear in the top menu bars on the Monitor. Any prompts to be answered or default values to be accepted will be displayed in the **CENTER** of the Monitor, or at the **BOTTOM** of the Monitor.

1-4-2 MENU-BAR

The first three items, LINE, ARC & POINT, are the main drawing functions and provide the basic means to draw any geometric object. The following is the *FastCAM* menu items and a brief description of each.

Files	Used to save or restore files.
Line	Used to draw lines.
Arc	Used to draw arcs.
Point	Used for point placement.
Constructs	Ellipse, Faired lines and Corners menu.
Erase	Removes Entities from the screen.
Blocks	Allows groups of entities to be moved, copied, rotated and arrayed.
View	Manipulates the graphics screen and plotters.
Utility	Utility functions.
Text Marking	Used to place Text on the graphics screen.
CAD Layers	Used to produce layers for DXF output.
Verify	Gives dimensional information about entities.
Trim	Trims a portion of an entity.
Program Path	Defines the cut process, kerf direction, cut order and lead-ins/outs.
Control Points	Shows the drawing construction points.

We will be discussing these menu items and more in CHAPTER 2 "*FastCAM*".

1-5 USING THE MOUSE

The entire *FastCAM* system can be accessed through the mouse. There is no need to use the keyboard except to enter numeric values and filenames.

Hold the mouse in the hand which is most comfortable. Grasp the side of the mouse with your thumb and ring finger. This allows your first and second finger to be used on the mouse buttons. The **RIGHT** key corresponds to the **ESCAPE** (REJECT) key on the keyboard, and the **LEFT** performs the same function as the **ENTER** (ACCEPT) key. Lay the mouse on a flat surface, preferably a mouse pad. Notice how at this stage the Mouse has an **ARROW** as a pointer (*depending on your Windows 95/NT preferences*). Click on a Menu Item and move the mouse from side to side. Notice that each Menu Item is opening to display it's contents.

To **SELECT** an item, move the highlight bar until it rests on one of the Menu Listing's Items and press the [**LEFT**] mouse button. The screen will change and the User-Input Box for the chosen selection is displayed.

To **REJECT** any item chosen, move the mouse away from any on-screen User-Input Box/Menu and press the [**RIGHT**] mouse button (*or press the **CANCEL** button, if the option is given*). The screen will change and the previous menu is displayed.

After a Menu Item has been selected and a screen input is required, the mouse will appear as a **CROSSHAIR**, (*size and color depending on your Windows 95/NT preferences*) and it is controlled by moving the mouse up, down, right and left.



LEFT mouse button is **ACCEPT** or **ENTER**, **RIGHT** mouse button is **ESCAPE** or **REJECT**.

1-5-1 MOUSE PRACTICE

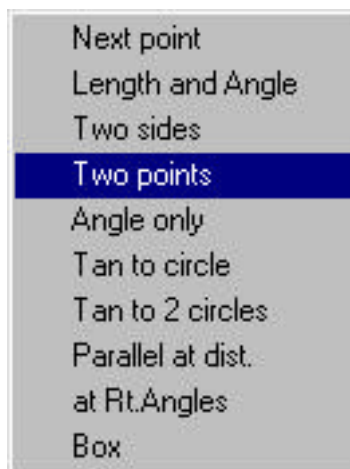
Practice using the mouse by first drawing a line between two defined points and then erase it from the screen.

- 1.) Place the mouse arrow on the **LINE** menu option and press the **LEFT** mouse button.



*You are now in the **Line Menu***

- 2.) Place the highlight bar on Line Menu item #4, **Two Points**, and press the **LEFT** mouse button.



- 3.) Click the Right mouse button and place the mouse arrow on menu item #2, **Absolute co-ords**, and press the **LEFT** mouse button.



A User Input box is now displayed in the middle of the screen. *The value of the last X, Y coordinate entered is displayed in the brackets in the center of the screen. In this case, since no values have been entered, a default **blank** value representing **<0,0>** is displayed.*

4.) Press the **ENTER** button on the input box to accept the default value of **<0, 0>**.



*The first end point of the line is indicated in the middle of the graphics screen by a **RED** point. You will notice also that the Menu heading has changed to the **Second point - point menu**. and the point menu is displayed in the center of the screen. **The first point will not be visible due to the User Input box.***

5.) The User Input box is re-displayed, enter the value of **12** for X, then hit **ENTER** on the keyboard, which will move the focus to the Y value. Enter the value of **0** for Y and press the **ENTER** button on the User Input box. A line of 12 inches will be drawn horizontally to the right of the **0,0** point on the graphics screen.

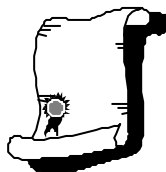
6.) Press the CANCEL button.

7.) Move off the menu and click your Right mouse button.



To view the line, simply press the Number 5 key on the KeyPad.

Congratulations!, you have just drawn your first line with **FastCAM**.



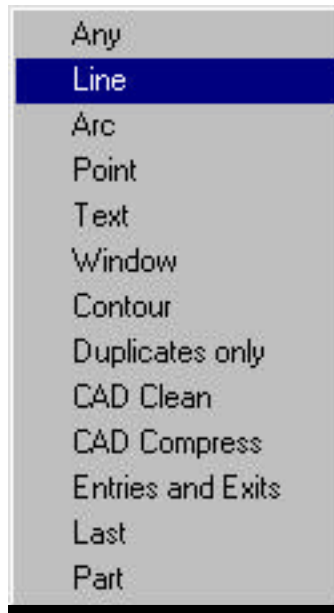
We have demonstrated the use of the mouse and the text screen, we will now call up a function requiring the use of the mouse and the graphics screen.

- 1.) Place the mouse arrow on the ERASE menu option and press the **LEFT** mouse button



*You are now in the **Erase menu**.*

- 2.) Using the mouse, move the highlight bar to select menu item #2, **Line**, and press the LEFT mouse button.



The mouse crosshair should now be on the graphics screen. Move the mouse and notice that the crosshair is following your movements. Practice moving the cursor around the screen until the movements become comfortable.

- 3.) Place the cross hairs anywhere on the line we have just drawn and press the **LEFT** mouse button.

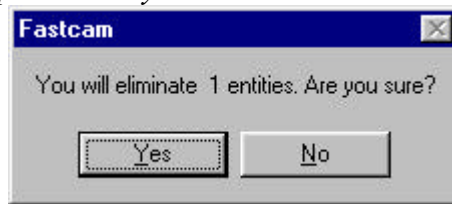


*A **RED** point will appear on the line which indicates where the line was picked.*

- 4.) Press the **RIGHT** mouse button and the line will disappear.



*A **PROMPT** now appears in the middle of the screen asking for a confirmation to permanently erase the line.*

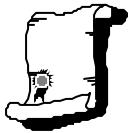


- 5.) Move the mouse onto the User Input box and click the **YES** button.

- 6.) Click your Right mouse button to exit from the Erase function.



*The line is now **PERMANENTLY** erased and your mouse cursor will return to the arrow.*



*Congratulations!, you have just Erased your first Entity in **FastCAM**.*

1-6 SCREEN ORIENTATION

Now that you have drawn your first entity, it is time to talk about the drawing screen.

1-6-1

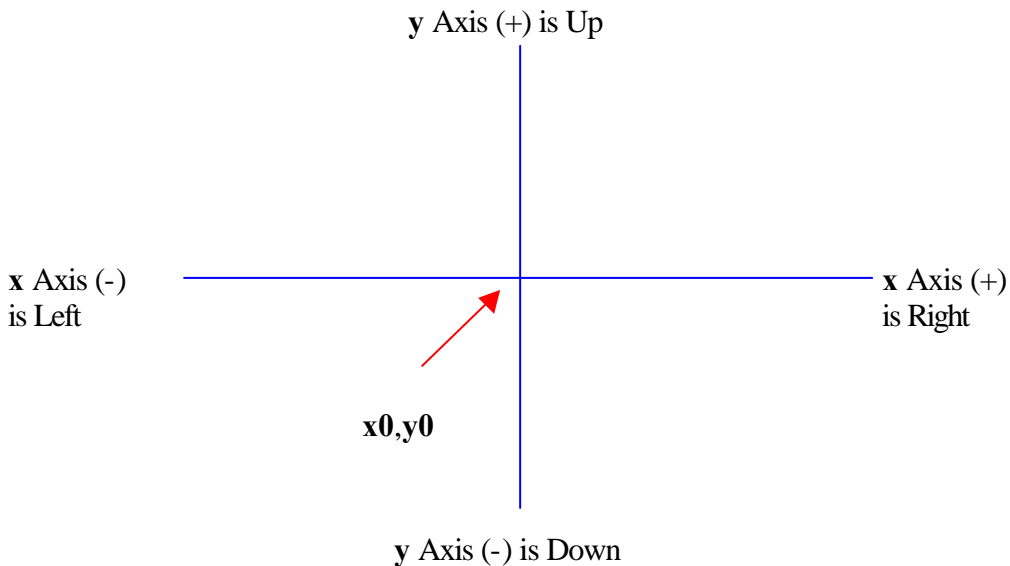
CARTESIAN CO-ORDINATE SYSTEM

Think of the drawing screen as infinite. There is a point in the middle of the screen called **ABSOLUTE ZERO**. Passing through that point are the **HORIZONTAL X Axis** and the **VERTICAL Y Axis**.

These two axes construct a two dimensional **CARTESIAN CO-ORDINATE** system.

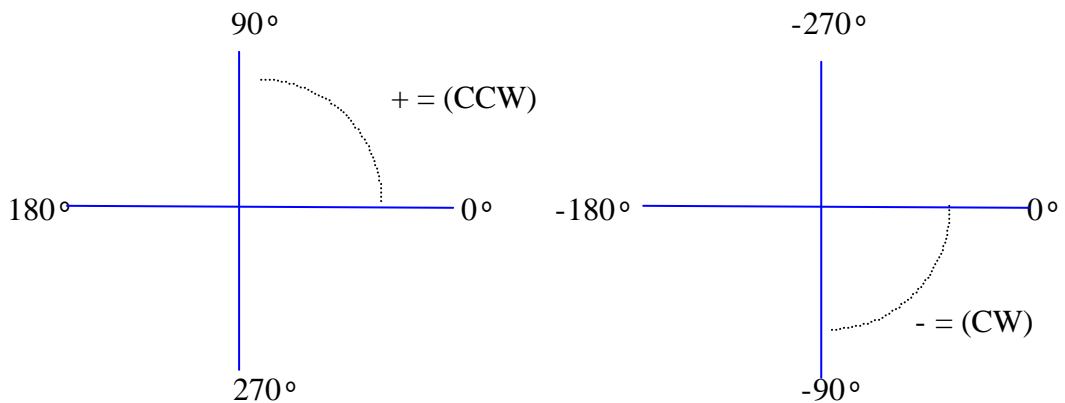
Moving **UP** on the vertical **Y** axis results in a positive **Y**, (+y) movement where as moving **DOWN** on the vertical **Y** axis results in a negative **Y**, (-y) movement.

Similarly, moving to the **RIGHT** on the horizontal **X** axis results in a positive **X**, (+x) movement and moving **LEFT** on the horizontal **X** axis results in a negative **X**, (-x) movement.



1-6-2 ANGLES in DEGREES

Angle degrees start at 0° from the positive X, (+x) or the right hand side of the screen, and increase positively in the counter clockwise direction (CCW) and negative in the clockwise direction (CW). Look at the following drawings.



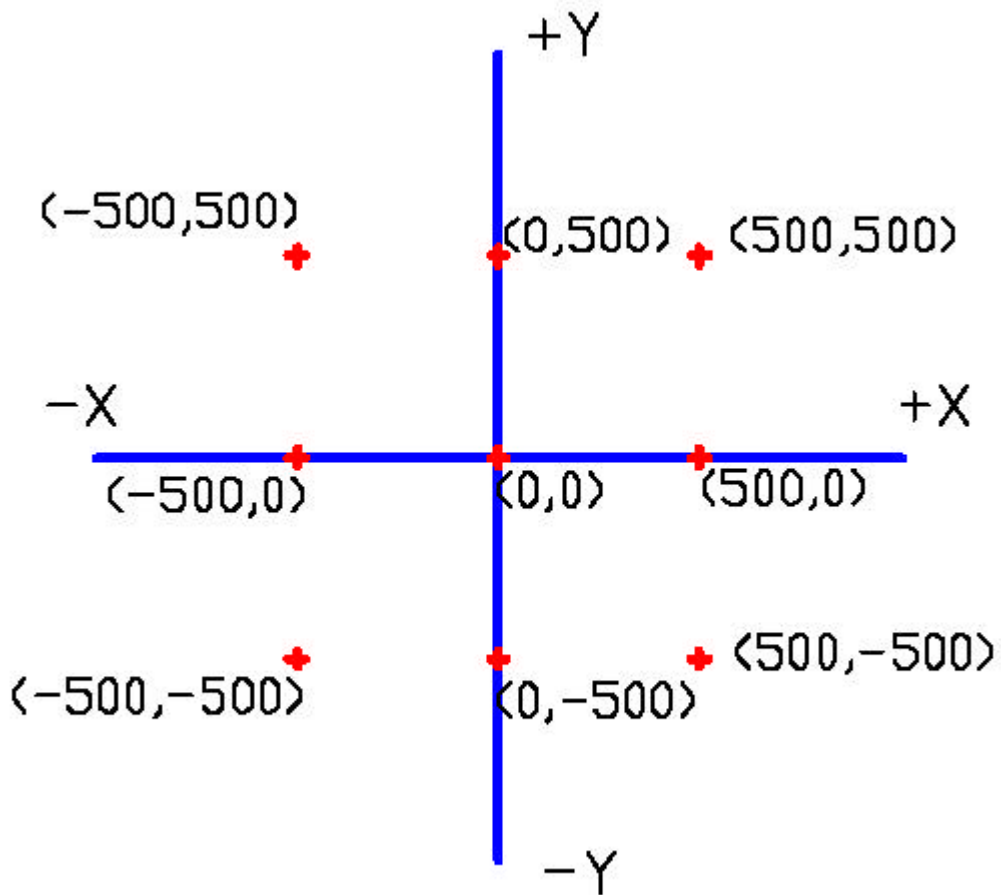
FastCAM will accept negative angle degrees , EXAMPLE: 270° is the same as -90° (negative 90 degrees)

1-7 ABSOLUTE Programming

There are two types of drawing techniques, ABSOLUTE and INCREMENTAL.

ABSOLUTE programming uses $\langle 0, 0 \rangle$ to reference all x,y co-ordinates on the drawing.

This simply means that any co-ordinate point entered is a relative movement from the point where the (x) axis line meets the (y) axis line or $(x0,y0)$. This point is referred to as $\langle 0,0 \rangle$.



Point placement using ABSOLUTE CO-ORDINATES.

EXERCISE 1:- ABSOLUTE PROGRAMMING

The aim of this exercise is to demonstrate the placement of points on the screen using absolute coordinates.

1.) Place the mouse arrow on the **Point** Menu option and press the **LEFT** mouse button.



*You are now in the **Point Menu***

2.) Note the message at the bottom of the screen, this message should tell you what point type is currently selected. Click your right mouse button and place the mouse arrow in the **Point Menu** on **Absolute co-ords** and press the **LEFT** mouse button.



*In the User Input box in the middle of the screen you will be given a default **blank** value which represents $\langle 0,0 \rangle$*

3.) At the prompt,

Press the **ENTER** button, on the box.

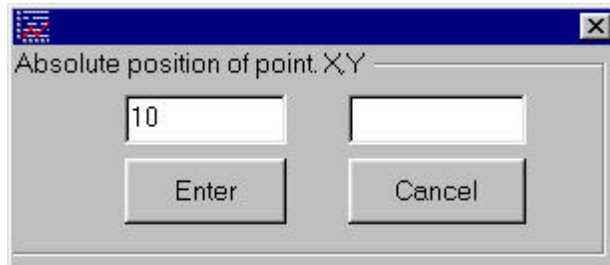


*A **YELLOW** point will be displayed in the center of the screen. NOTE: This point may not be visible due to the menu in the middle of the screen. This point is **ABSOLUTE ZERO** or $\langle 0,0 \rangle$. Now we will set points on the major axes around absolute zero.*

4.) The User input box will be re-displayed, again.

5.) At the prompt,

TYPE in **10**, and hit the ENTER key, and then type in **0**, and press the **ENTER** button.



A screenshot of a Windows-style dialog box with a blue title bar. The title bar contains a small icon on the left and a close button (X) on the right. The text inside the dialog box reads "Absolute position of point.XY". Below this text are two input fields. The first input field contains the number "10". The second input field is empty. Below the input fields are two buttons: "Enter" and "Cancel".



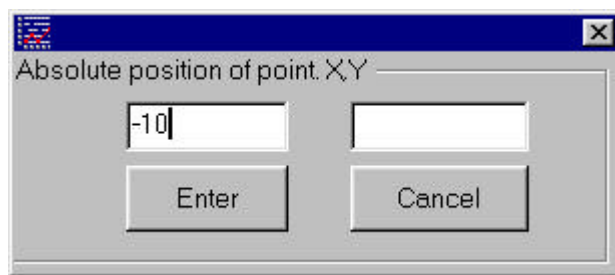
*This places a point 10" to the right of **Absolute Zero**.*

6.) On the User input box displayed in the middle of the screen,

TYPE in **-10**, and hit the ENTER key, and then type in **0**, and press the **ENTER** button.



*At the prompt in the User Input box you will be given a default value of **<10,0>** from your previous input.*



A screenshot of a Windows-style dialog box with a blue title bar. The title bar contains a small icon on the left and a close button (X) on the right. The text inside the dialog box reads "Absolute position of point.XY". Below this text are two input fields. The first input field contains the number "-10". The second input field is empty. Below the input fields are two buttons: "Enter" and "Cancel".



*This places a point 10" to the Left of **Absolute Zero**.*

7.) On the User input box displayed in the middle of the screen,

TYPE in **0**, and hit the ENTER key, and then type in **10**, and press the **ENTER** button.



At the prompt in the User Input box you will be given a default value of <-10,0> from your previous input.

A screenshot of a Windows-style dialog box titled "Absolute position of point X,Y". The dialog has a blue title bar with a close button (X) in the top right corner. Inside the dialog, there are two text input fields. The first field contains the number "0" and the second field contains the number "10". Below these fields are two buttons: "Enter" and "Cancel".



*This places a point 10" Up from **Absolute Zero**.*

8.) On the User input box displayed in the middle of the screen

TYPE in **0**, and hit the ENTER key, and then type in **-10**, and press the **LEFT** mouse button.



At the prompt in the User Input box you will be given a default value of <0,10>

A screenshot of a Windows-style dialog box titled "Absolute position of point X,Y". The dialog has a blue title bar with a close button (X) in the top right corner. Inside the dialog, there are two text input fields. The first field contains the number "0" and the second field contains the number "-10". Below these fields are two buttons: "Enter" and "Cancel".

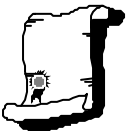
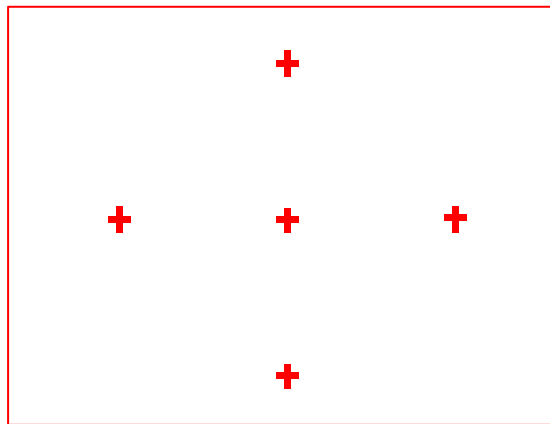


*This places a point 10" Down from **Absolute Zero**.*

9.) Now, move off the menu, and click the **RIGHT** mouse button.



Your Graphics screen should now show the pattern below. If the pattern is very small and hard to see, simply press the Number 5 key on the KeyPad.



Congratulations!, you have just placed five points on the screen using **Absolute co-ordinate positioning**.



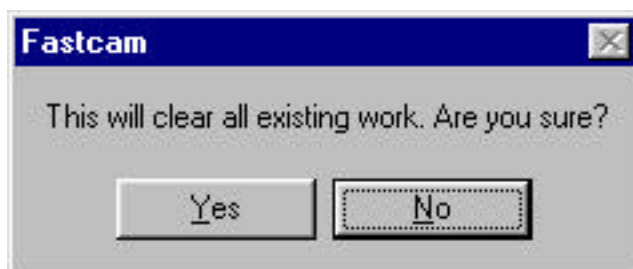
Before proceeding to the next exercise, it is necessary to clear the screen by using the **New** function found in the **Files menu**.

EXERCISE 2:- RESTART

- 1.) Place the mouse arrow on the **Files Menu** and press the **LEFT** mouse button.
- 2.) Move the highlight bar down to **New** and press the **LEFT** mouse button.



At the prompt in the middle of the screen you will be given an Input box:



- 3.) Press the YES button on the User Input box.



You have now permanently erased all of the entities on the drawing and you now have a new drawing screen. Be cautious when using the NEW function. It will permanently erase all existing work that has not been saved (see "Files" later in this chapter).

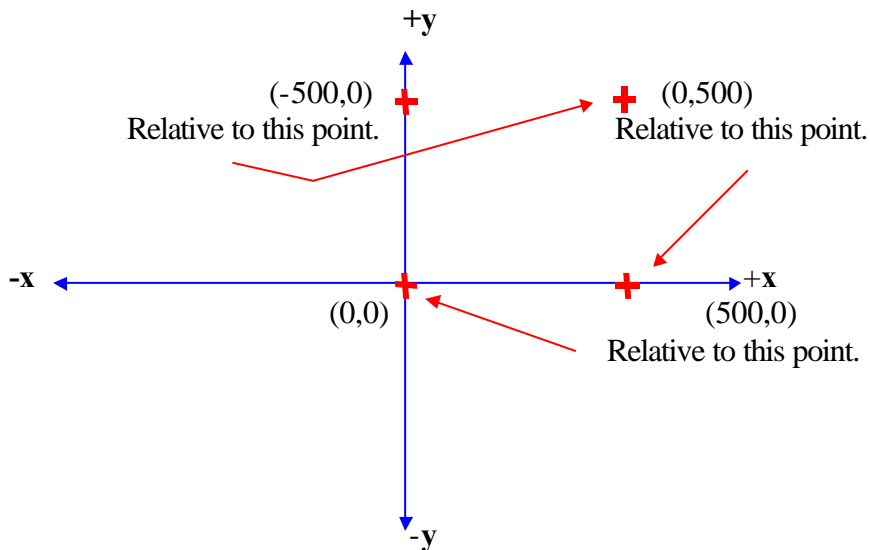
1-8 INCREMENTAL Programming

Incremental programming uses any known point on the screen to reference an (x,y) movement.

This simply means that any (x,y) co-ordinates can be entered relative to any known point on the drawing screen. If you choose a point on your screen the next point can be an (x,y) movement from the chosen point and does not have to be related to (0,0).



Absolute zero is still the point in the middle of the screen where the X and Y axes cross.



Point placement using INCREMENTAL CO-ORDINATES.

Incremental programming uses any known point on the screen for reference. When Incremental co-ordinates are selected, *FastCAM* locks to the last point entered. A prompt then asks if you wish to "Use this position for reference? YES/NO". If this is the required reference point, answer YES, if it is not then reject the point by selecting NO. When the point is rejected, the mouse cursor is displayed on the screen. Use the mouse to move the cursor to the new reference point, and select that point by pressing the LEFT mouse button. The program again asks if you wish to "Use this position for reference? YES/NO". If this is the required reference point, answer YES and then Type in the (x,y) co-ordinates of the next point you wish to place.

EXERCISE 3:- INCREMENTAL Co-ordinates

This exercise demonstrates placing points on the screen using INCREMENTAL coordinates.



*First you must set a reference point at **0,0** to work from.*

1.) Place the mouse arrow on the **Point** menu and press the **LEFT** mouse button.



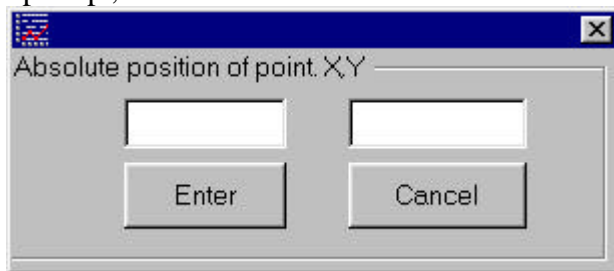
*You are now in the **Point Menu***

2.) Place the highlight bar displayed in the **Point Menu** on **Absolute co-ords** and press the **LEFT** mouse button.



At the prompt in the middle of the screen you will be given a default blank value of <0,0>

3.) At the prompt,



A dialog box with a blue title bar containing a small icon and a close button. The text 'Absolute position of point.X,Y' is displayed above two empty rectangular input fields. Below the input fields are two buttons labeled 'Enter' and 'Cancel'.

Press the **ENTER** button.



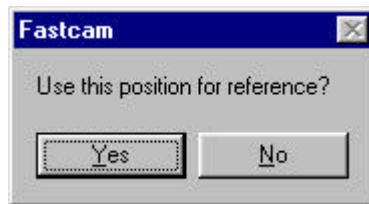
A **YELLOW** point will be displayed in the center of the screen. This point is **ABSOLUTE ZERO** or $\langle 0,0 \rangle$. Now we will set points **Relative** to this point.

- 4.) The Absolute input box will re-appear, click the CANCEL button. Move off the point menu and click the Right mouse button. Now place the mouse arrow displayed, in the **Point Menu** on **Incremental co-ord** and press the **LEFT** mouse button.



At this point a large crosshair will appear on the screen located on the last point you entered. At the prompt in the middle of the screen you will be asked if you wish to use this point as your reference point for the next point.

- 5.) At the prompt,

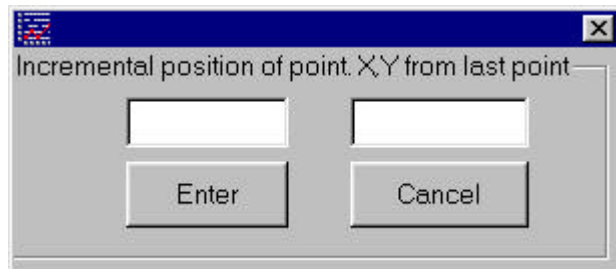


Press the **YES** button.



You are then prompted in the middle of the screen to type in the (x,y) position of the next point.

6.) At the prompt,

A screenshot of a Windows-style dialog box. The title bar is blue with a close button (X) on the right. The text inside the dialog reads "Incremental position of point. X,Y from last point". Below the text are two empty rectangular input fields. Underneath the input fields are two buttons: "Enter" on the left and "Cancel" on the right.

Type in **10**, and hit the ENTER key, and then type in **0** and press the **ENTER** button.

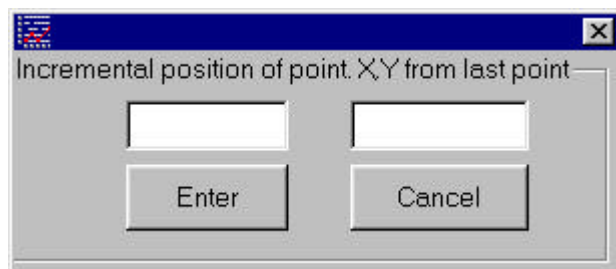


This places a point 10" to the right of the point you chose to measure from.



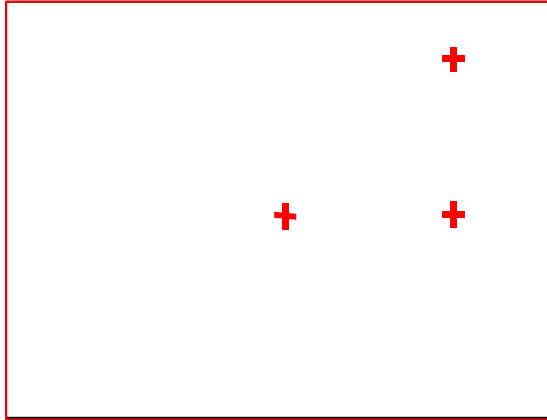
You are then prompted in the middle of the screen to type in the (x,y) position of the next point.

7.) The User input box for Incremental Co-ordinates will re-appear, at the prompt, Type in **0**, and hit the ENTER key, and then type in **10** and press the **ENTER** button.

A screenshot of a Windows-style dialog box, identical to the one in block 6. The title bar is blue with a close button (X) on the right. The text inside the dialog reads "Incremental position of point. X,Y from last point". Below the text are two empty rectangular input fields. Underneath the input fields are two buttons: "Enter" on the left and "Cancel" on the right.



This places a point 10" above the point you chose to measure from and your screen should now show the pattern below.

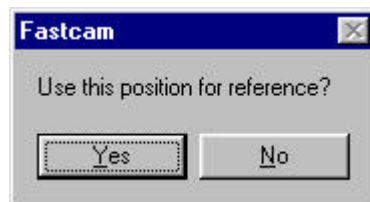


- 8.) The User input box will re-appear again, Select **CANCEL**.
Move off the point menu and click the Right mouse button.
Now place the mouse arrow displayed, in the **Point Menu** on **Incremental co-ord** and press the **LEFT** mouse button.



At this point a large crosshair pointer will appear on the screen located on the last point you entered . At the prompt in the middle of the screen you will be asked if you wish to use this point as your reference point for the next point.

- 9.) At the prompt,



Press the button labeled **NO**.

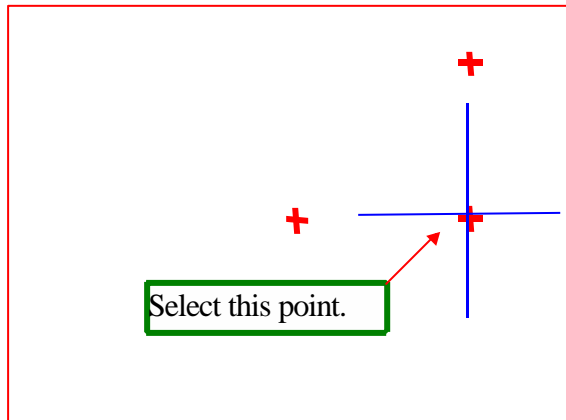


You are then prompted in the middle of the screen to Indicate the new reference point you wish to measure from. You will now have the mouse cursor on the screen.

- 10.) Move the cursor to the position shown below and press the **LEFT** mouse button to indicate this to be the point to reference the next (x,y) movement from.

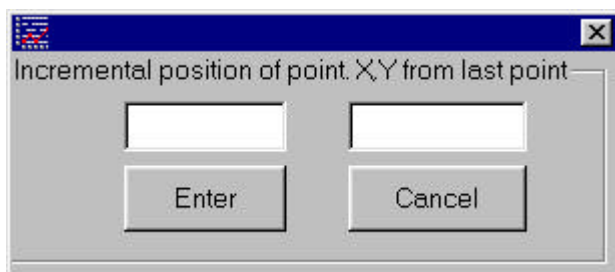
The mouse **CURSOR** does **NOT** have to be **EXACTLY** on the point you wish to choose. You need only to place the pointer within a radius of (1/8") for it to find the required point.

If you miss the required point, the program will display an information box specifying that there was "No point found". At this point simply click the left mouse button and try to select the required point again.



At this point the selected point will turn red in color (indicating that the point has been selected).

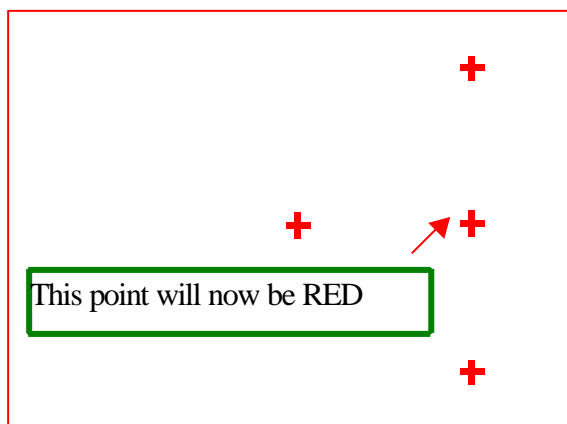
11.) At the prompt,



Type in **0**, and hit the ENTER key, and then type in **-10** and press the **ENTER** button.

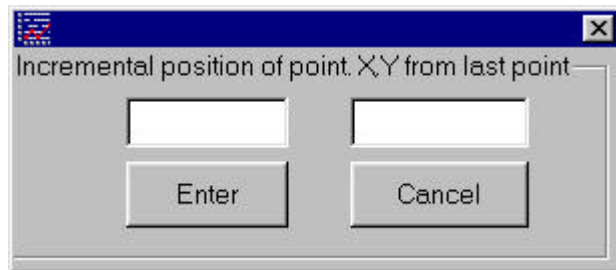


This places a point 10" below the point you chose to measure from and your Graphics screen should now show the pattern below.



*You will notice that the point we just chose as our reference point to measure from has now turned **RED**. Do not be concerned with this as we will discuss later how to (Refresh the drawing) change it back to a **YELLOW** point.*

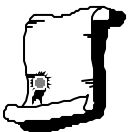
12.) At the prompt,



Type in **-10,0** and press the **ENTER** button.



This places a point 10" to the left of the point you chose to measure from and your screen should now show the pattern below.



Congratulations!, you have just placed five points on the screen using **Incremental Co-ordinate positioning**.

1-9 SETTING POINTS

So far we have used INCREMENTAL and ABSOLUTE CO-ORDS to set points on the screen. There are also several other methods to set points. Become familiar with the POINT MENU, it appears in all other types of constructions. For example, to draw a line, the end points must be known, to draw a circle the center point must be known, and to draw an arc the start, end and center points must be known. The menu will always look the same but the header is changed to fit the type of construction.

Point	
Screen position	Used to sketch points
Absolute co-ords	Places points relative to (0,0)
Incremental co-ords	Places points relative to a known point
Length and angle	Used when the length and angle are known
Line midpoint	Places a point at the center of a line
Control point	Converts a control point to a real point
Intersection	Places a point at the intersection of entities
On entity	Places a point anywhere on an entity



*Apart from Absolute and Incremental co-ordinate point placement which we have just covered, there are two other main methods of point placement which are **CONTROL POINTS** and **INTERSECTION**. For more information on Screen position, Length and angle, Line midpoint and On entity, see CHAPTER 2. "FastCAM"*

1-9-1 CONTROL POINTS

CONTROL POINTS are reference points associated with lines, arcs, and circles. When one of these entities is drawn on the screen the appropriate control points are created. A line will have two control points, one at each endpoint. A circle, five control points, one at the center, plus one at each quadrant. An arc has a minimum of three control points, one at the start, center, and end, plus a quadrant point if the arc includes 90° quadrant points. Control points are displayed by using the **Control pts,** menu item or by pressing the **Asterix** key on the KeyPad.

Control points are represented on the screen by a red cross. Control points are created when an entity is drawn on the screen and will be removed when the entity is erased. The difference between control points and real points is that real points are created by using the **Point menu** and must be removed with the **Erase menu,** while control points are created when an entity is created and removed when the entity is erased. The following exercise demonstrates the difference between the two kinds of points and how to convert a control point to a real point.

EXERCISE 4: USING CONTROL POINTS

This exercise shows how CONTROL POINTS are created, displayed on the screen, and converted to a real point.

- 1.) Draw a line on the screen by accessing the Line menu and follow the steps shown below to set the first and second endpoint's of the line.
 - a) Choose the Line menu option.
 - b) Choose the TWO points option.
 - c) Choose Absolute Co-ords, and specify 0,0.
 - d) Choose Absolute Co-ords. and specify 10,0.

- e) Hit the **CANCEL** button.
- f) Move off the point menu and click the right mouse button.



You should now have a line 10" long on your screen.

- 2.) Place the mouse arrow on the **Control pts** menu item and press select Control points from this menu with the **LEFT** mouse button.



*You will notice a **RED** cross or Control point is displayed at the **Endpoints** of your line.*

- 3.) Place the mouse arrow on the **Points** menu item and press the **LEFT** mouse button.



*You are now in the **Point** menu.*

- 4.) Place the highlight bar displayed in the **Point** menu on **Control point** and press the **LEFT** mouse button.

- 5.) Place the mouse cursor on one of the **RED** control points displayed on the end of the line and press the **LEFT** mouse button.

- 6.) Click the **RIGHT** mouse button.



*The point selected turns **YELLOW**. You have now set a real point at a control point. When the line is erased the endpoint defined as a real point will remain on the screen and the endpoint not defined will vanish.*

EXERCISE 5: INTERSECTION

Another way to set a point is by a line or arc intersection. When two entities cross on the screen, **FastCAM** recognizes their intersection as a **Control point**. A real point can be placed at this intersection by using the **Point** on a **Control point** method. If for example the two entities do not cross but a projection or extension of these entities would, and the intersecting point is needed to be found, you can place a Real point where these entities would meet.

- 1.) If you have any entities on your drawing screen, clear the screen by accessing the **File** menu and selecting **NEW**.



You should now have a clear screen.

- 2.) Draw two lines on the screen by accessing the Line menu and following the steps shown below to set the first and second endpoint's of the lines.

- a) Select the **Two Points** option.
- b) Select Absolute Co-ords for point placement.
- c) Input 8,-8 for x,y.
- d) Select Absolute Co-ords for point placement.
- e) Input 3,5 for x,y.
- f) Select Absolute Co-ords for point placement.
- g) Input -8,-8 for x,y.
- h) Select Absolute Co-ords for point placement.
- i) Input -3,5 for x,y.



You should now have two lines on your screen.

3.) Place the mouse arrow on the **Point** menu item and press the **LEFT** mouse button.

4.) Place the highlight bar displayed in the **Point** menu on **Intersection** and press the **LEFT** mouse button.



*You will now have the mouse crosshair displayed on your screen. In the middle of your screen you are prompted to select the **First** and **Second** entity you wish to find the intersection of.*

REMEMBER

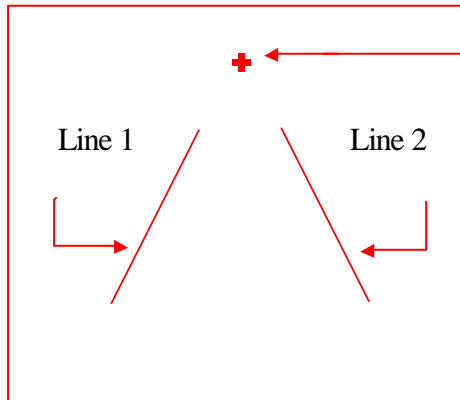
If you miss the required point, the program will **BEEP** at you and place a **RED** point on the screen. At this point simply try to select the required point again.

5.) At the prompt **Select first entity**, place your crosshair on **Line 1** as shown in the sketch below and press the **Left** button on your mouse.

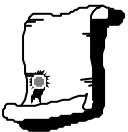


You are then prompted to Select second entity.

- 6.) At the prompt **Select second entity**, place your crosshair on **Line 2** in the sketch below and press the **Left** button on your mouse.



A yellow 'real point' will be placed here at the intersection of the two lines.



Congratulations! You have just placed a real point at the intersection of two entities. This can be done for any two entities on the drawing screen that will, at some point intersect.

EXERCISE 6: CLEANING up the DRAWING

1.) Place the mouse arrow on the **VIEW** menu item and press the **LEFT** mouse button.



*You are now in the **View** menu.*

2.) Place the highlight bar displayed in the **View** menu on **Repeat** and press the **LEFT** mouse button.



*Your screen is **REDRAWN** and all of the **RED** points you used to indicate each line has disappeared. This operation will also **CLEAN UP** any graphic display leftovers from previous operations. This can also be done by simply hitting the **ENTER** key on the Keypad.*

EXERCISE 7: ERASE

1.) Place the mouse arrow on the **Erase** menu item and press the **LEFT** mouse button.



*You are now in the **Erase** menu.*

2.) Place the highlight bar displayed in the **Erase** menu on **Line** and press the **LEFT** mouse button.



*You will now have the **mouse** crosshair displayed on your screen. At the bottom of your screen you are prompted to **Indicate** the lines to erase by pointing to them.*

3.) At the prompt,

"Indicate point"

Place your crosshair on **Line 1** and press the **Left** button on your mouse.



*A **Red** point will appear on the line where you indicated.*

4.) At the prompt

"Indicate point"

Place your crosshair on **Line 2** and press the **Left** button on your mouse

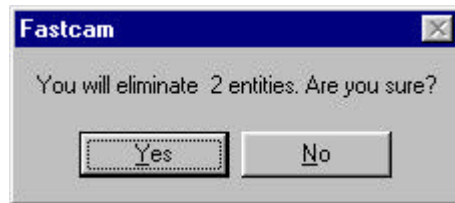


*A **Red** point will appear on the line where you indicated.*

At this point you have indicated that you wish to erase both of these lines from the drawing.

5.) Press the **RIGHT** mouse button to **Erase** the **Lines**.

6.) At the prompt,

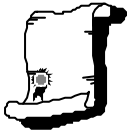


Press the **YES** button.



*At this point the two lines you indicated to be erased will be **PERMANENTLY** removed from your drawing. The **YELLOW** real point remains on the screen.*

*You will still see the **RED** points on the screen where you indicated the lines. To remove these from the screen follow the instructions in **EXERCISE 5** "Cleaning up the drawing."*



drawing.

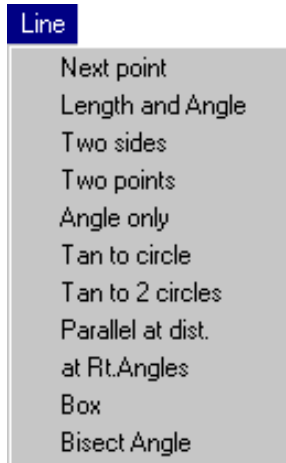
Congratulations! You have just **Erased** some entities from the

NOTES ON POINT PLACEMENT

- 1.) Absolute zero is always present on the screen and can be used as a reference point, even if there is not a real point set at 0, 0.
- 2.) Control points are construction points and show up as red crosses, real points show up as yellow crosses. Control points are removed when the entity is erased. Real points are removed by using the ERASE MENU.
- 3.) The cursor has a (1/8") window around the cross hairs to make choosing points easier. If the point is chosen outside the window the computer will beep a warning.

1-10 LINE MENU

The Line menu is second from the left on the Main menu and contains the following menu items.

**NOTES ABOUT LINE**

- 1.) Never use screen position to set the endpoint of a line unless accuracy is not important.
- 2.) If a line is drawn on the screen, using 0, 0 as a reference point, there does not need to be a point set at 0, 0 to use this point for reference.
- 3.) When using next point, the line will always be drawn from the last line endpoint no matter what point is used for a reference point on the screen.
- 4.) Infinite lines can be trimmed to the required length by using the TRIM function.

1-11 ARC MENU

The Arc menu is third from the left on the Main menu and contains the following menu items.



The **Arc menu** allows Full circles, Arcs, Fillet-blends and Rings to be drawn.

1-11-1 FULL CIRCLE

To draw a full circle, simply call up the circle function, give the radius and specify the center of the circle.

1-11-2 ARCS

Arc construction in *FastCAM* falls into the following categories.

Start,point,End
 Start,Cent,End
 Start,End,Radius
 Cen,Rad,A1,A2
 C,R,A1,Sweep
 Half circle

1. Three point arc. The start, a point on the arc, and the endpoint must be known
2. Start, center, endpoint, and direction must be known
3. Start, end, radius and direction of arc must be known
4. Center point, Radius, Angle one or start angle, angle two or end angle
5. Center point, radius, angle one of the start angle, and sweep.
 The sweep angle is the number of degrees from the start to the end of the arc. If the start angle is 45 degrees and the sweep is 180 the end angle is $45 + 180 = 225$ degrees.
6. Two points and direction of arc must be known



To draw an arc decide what is known about the arc, then follow the prompts for that selection.

*For further information on Arcs, see CHAPTER 2 "**FastCAM**".*

1-11-3

FILLET BLENDS

A Fillet-blend is an arc place between two entities to round corners and blend the intersection of the entities. **FastCAM** provides constructions for blends between lines and lines, lines and arcs, arcs and arcs, and a point and line. To use Fillet-blend select it from the ARC menu item.

Using the mouse cursor, choose the two entities to be blended. If finite lines such as a box is filleted, the part will be automatically trimmed. If infinite lines or arcs are involved in the fillet-blend, auto-trimming will not occur and the part must be trimmed with the TRIM function.

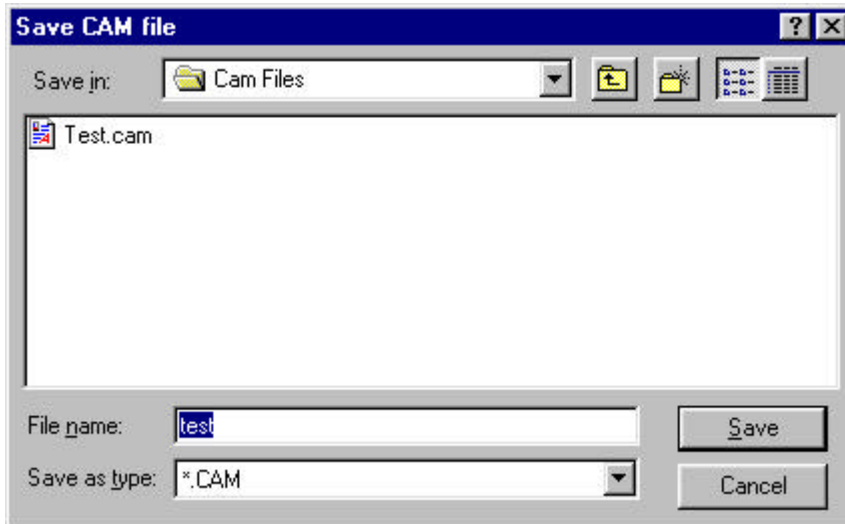


For further information on Fillet-blend , see CHAPTER 2 "FastCAM".

1-12 SAVING AND RESTORING FILES.

To use the file save function select the Files menu item and move the highlight bar to SAVE and click the LEFT mouse button.

You will be presented with the **Windows Files-Save Dialog** box.



Save is a very important function and must be used to save all geometry created. When a part is drawn on the screen, there is no permanent record of the part until a **SAVE** is performed. If the part has been drawn on the screen and not Saved, the drawing can be lost if the power is turned off or if the computer should happen to lock up. SAVES are quick and easy to do, it should become part of your programming routine.

EXERCISE 8: SAVING a DRAWING

- 1.) Draw any lines or points on the screen.
- 2.) Place the mouse arrow on to the **Files** menu item and press the **LEFT** mouse button. Move the highlight bar to **SAVE** and click the **LEFT** mouse button.



*You are now in the **PART DATA** entry screen. We will cover this screen in CHAPTER 2 "**FastCAM**". For now we will just put in a Filename.*

- 3.) Ensure you are on the line headed "**Part Name**" and type in with the **Keyboard** the Filename **TEST**



The Filename placed in this field will be used for our NC code Filename for this part.

- 4.) Press the **ENTER** button at the bottom left of the **PART DATA** screen.



*You are now in the **Windows Files-Save Dialog** box.*

- 5.) Press the **Save** button on the box.



Congratulations! You have just Saved a *FastCAM* drawing file.

EXERCISE 9: RESTORING a DRAWING

1.) Place the mouse arrow on the **Files** menu item and press the **LEFT** mouse button.

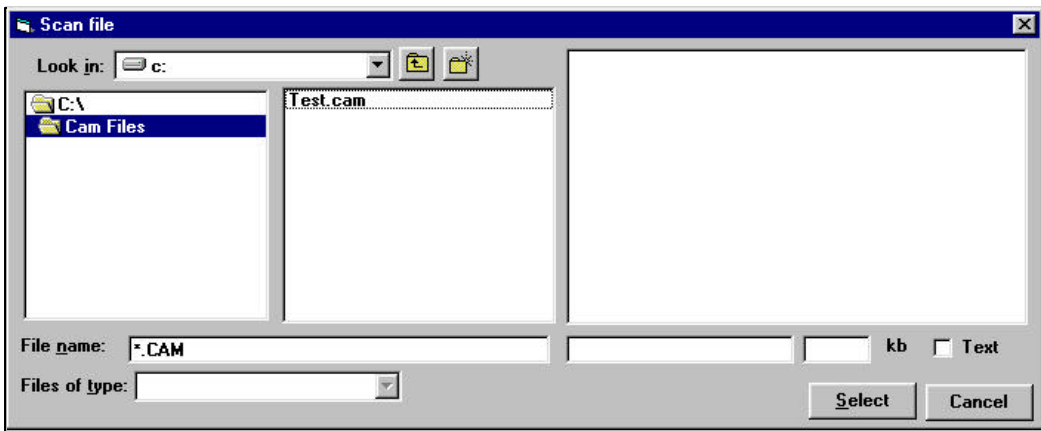


*You are now in the **FILES** menu.*

2.) Place the highlight bar displayed in the **FILES** menu on **Restore** and press the **LEFT** mouse button.



*You are now in the **Windows Open-Files Dialog box**.*



3.) Move your mouse over the part name "**TEST.CAM**" and either:

- **Double Click** the file name, or
- Press the **Select** button.



The part selected is then displayed on your screen.



Congratulations! You have just **Restored** a ***FastCAM*** drawing file.



*There is also a **DXF** and **IGES** (CAD files) interface. To save or restore a DXF or IGES file follow the same procedures as above, but use the DXF or IGES save and restore functions.*

*See CHAPTER 2. "**FastCAM**" for more information on these functions.*

1-13 THE VIEW MENU

The VIEW MENU allows the manipulation of the view on the graphics screen and allows the graphics output to be changed to the plotter or printer. By using the view menu the operator can zoom in for a close up view of a section, change the scale of the part, look at a previous view or dump a copy of the graphics screen to the printer or plotter.

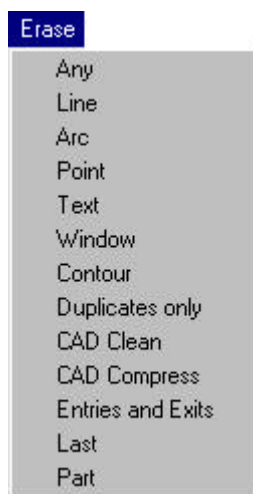


*For further information see CHAPTER 2 "**FastCAM**".*

1-14 THE ERASE MENU

The ERASE MENU is used to remove entire entities from the screen. If only part of an entity is to be removed the trim function should be used.

The Erase menu is accessed via the Menu bar and contains the following menu items.



To use the ERASE MENU place the highlight bar on the type of the entity to be erased and press the LEFT mouse button. The mouse cursor is displayed on the screen. Place the cursor on the entity to be removed and press the LEFT mouse button. A red pick point will appear. Press the RIGHT mouse button and the entity will be erased. If this is the entity you wished to remove confirm the operation by answering YES. Erase is a continuous function. You can choose multiple entities and when the RIGHT mouse button is pressed they will all be erased.

For further information see CHAPTER 2 "*FastCAM*".

1-15 TRIMMING ENTITIES

Trimming entities is a very important concept. Many times the easiest way to create a part is to draw several over lapping entities and trim out the unused parts.

1-15-1 TRIMMING LINES

To TRIM any entity, the programmer must be able to locate either a control point or a real point to use for a trim point. To TRIM a line, the trim function is selected, *FastCAM* first prompts for the item to trim, then the first and second points to be used for trimming. Indicate the points by placing the mouse cursor on the point and press the LEFT mouse button. **When an INFINITE LINE is trimmed, the section of the line outside the trim points is removed.** This is just the opposite of trimming FINITE LINES. **When a FINITE LINE is trimmed the portion of the line between the trim points is removed.**

1-15-2 TRIMMING ARCS

To trim an arc, the programmer Indicates the arc to trim by placing the mouse cursor on the arc and pressing the LEFT mouse button. Indicate the first and second trim points with the cursor. Using the cursor, you then point to the portion of the arc to be erased.

For further information see CHAPTER 2 "*FastCAM*".

1-16 Producing a PROGRAM PATH

PROGRAM PATH is used to define the type of machine process; cutting, zinc marking, punch etc., the cut order, direction and kerf. Once all paths are defined, a file containing the NC code of the part will be produced. The name of the file is limited to 256 alphanumeric characters with any three alphanumeric character extension. The three character extension is not essential unless your machine controller requires a specific filename extension.

NOTES ON PROGRAM PATHS

1.) When setting paths on circles it is necessary to indicate if the circle is to be cut on the inside or outside of the line. Inside cuts are the DROPS or interior cut outs and should be cut first.

2.) When setting a program path on an entity which is not a circle it is necessary to force the path in the required direction by moving your Cut direction Arrow with the mouse.

Simply indicate the entity to be cut first and set the direction of cut with the cut direction arrow.

3.) LEADINS and LEADOUTS can be placed on the part when requested. The choices are straight, quarter circle lead in, quarter circle lead out, half circle lead in, half circle lead out, quarter circle lead in and lead out, half circle lead in and lead out.



*If you are cutting with Kerf, you **MUST** put a Lead-in on your part.*

Lead-ins are not necessary if you are cutting with NO Kerf.

- 4.) KERF is the amount of material destroyed in the cutting process due to the width of the cutting flame.



The general standard for Kerf, is to use LEFT Kerf on all cuts, this is usually determined by the type of consumables used within your equipment.

- 5.) If a LEFT KERF is used the cut path needs to be defined COUNTERCLOCKWISE for inside cuts, and CLOCKWISE for outside cuts. If a RIGHT KERF is indicated the opposite is true, CLOCKWISE for inside cuts and COUNTERCLOCKWISE for outside cuts.
- 6.) On some controllers, the Feedrate and the amount of Kerf can be Preset in the NC program. Please check with your machine supplier for these details. In other cases only the Kerf direction is set in **FastCAM** whilst the Feedrate and Kerf value is set at the controller by the machine operator.

For further information see CHAPTER 2 "**FastCAM**".

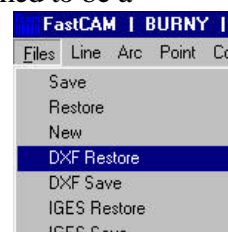
FastCAM® Quick Start

DXF to NC step-by-step guide.

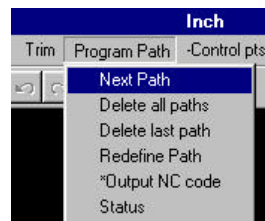
A demonstration of this guide can be downloaded from our website at:

<http://www.fastcamusa.com/demos/fcquick.exe>

This quick start guide will outline the process in which a DXF file can be loaded into FastCAM and an NC program be produced. The DXF file has been drawn separately in a CAD program as a single part. This DXF part is assumed to be a simple rectangle with a circular hole inside it.



1. Load DXF file into FastCAM.
 - a. From the main menu select **Files** *DXF Restore*.
 - b. In the DXF Options Window set *File Selection* to *Single File*. *DXF File units* will be *inch* (assuming the drawing has been drawn in inches). *CAD Clean* and *Apply Z Axis Extrusion* should be checked. The *ENTER* button can then be pressed.
 - c. Find the required DXF file then press *Select*.



2. Add cutting path to the part.
 - a. Select **Program Path** then **Next Path** from the main menu. The *Part Data* form may appear. If this form does not appear then this step can be ignored. Information can be entered into the *Part Data* form if required. *ENTER* is then pressed.

3. The Operation now needs to be defined. This is the cutting process and is set from a sub menu that will appear. A typical cutting operation in this menu is *PLASMA*. Select ***PLASMA*** from this menu. If *PLASMA* is unavailable select the top most operation.

PLASMA
OXY
SCRIBE-PLASMA
PUNCH-PLASMA
SCRIBE-OXY

4. *Kerf* must now be defined. The three options are *Left*, *None* or *Right*. The most common Kerf setting is Left. With Kerf set to Left all external (part perimeter cuts) are made larger and all internal cuts (or drops) are made smaller. External cuts must be made clockwise and internal cuts must be made counter clockwise when using ***Left Kerf***

Left
None
Right

5. A cross hair will appear and the internal circular hole should be selected. Left clicking anywhere on the circle will add an entry. The hole is an internal hole so ***Inside*** should be selected rather than *Outside* from the menu displayed. The entry can be either at the point selected or at the nearest quadrant (90 degree steps from zero.) *Nearest Quadrant* is usually a better choice. Select ***Nearest Quadrant***.

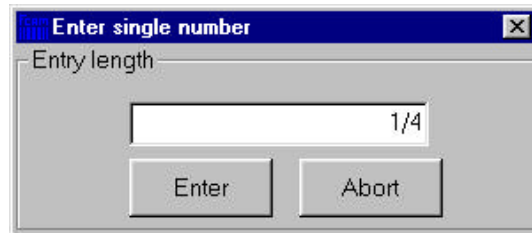
Outside
Inside

Nearest quadrant
Indicated point

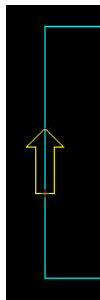
6. The entry should be set to ***Straight*** from the options displayed. There will be no exit on this cutout.

Straight
Quarter circle
Half circle
QC entryexit
HC entryexit

7. An entry length should now be set. The default entry length offered by FastCAM is a reasonable guess. Setting a specific length is a better option. For this example $\frac{1}{4}$ " (6mm) can be entered unless this length is not suitable.

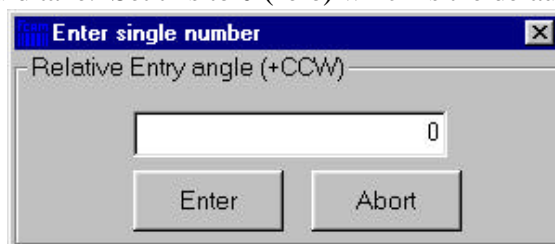


8. The next profiling operation can now be set. This will add a cutting path to the outside or external contour of the DXF part; in this example a rectangle. The path begins at the beginning of an entity. For a rectangle all entities are straight lines of which there are 4. The line should be clicked at its approximate midpoint to ensure it is selected correctly. To add a cutting path, select **PLASMA** from the menu, then **Left Kerf**. A cross hair will appear. The left hand line should be selected (at its approximate midpoint) with the cross hair. A yellow arrow will be displayed indicating a direction. This arrow should point up. If a person were to travel around this rectangle in the direction of the yellow arrow they would be moving in a clockwise direction. Move the mouse to change the direction of the yellow arrow. The left mouse button should now be pressed.

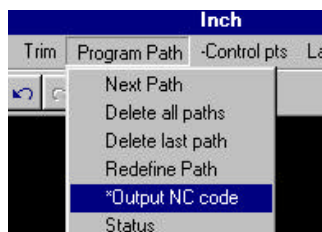


9. An **Add Entry** question will be displayed. Selecting **Yes** will allow the input of a lead-in or entry on this part. The entry type should now be selected. The best entry type for this part is straight. It will add an entry, effectively lengthening the selected line at its lower left hand corner by the amount required. As for the circular hole $\frac{1}{4}$ " (6mm) can be used. Press enter after typing this value to add the entry.

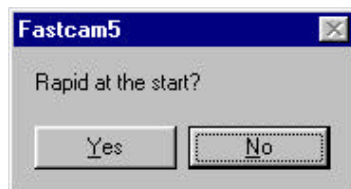
10. The **Relative Entry Angle** is now required. This is the angular direction the entry should take. Set this to **0** (zero) which is the default.



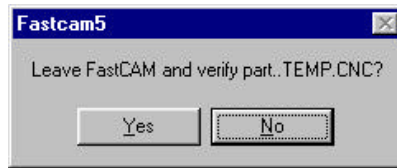
11. An **Exit** can now be set if required. The procedure for adding an exit is the same as an entry. For this example select **No**.
12. Once both cutting paths have been added the NC code can be generated. Selecting **Program Path Output NC Code** (from the main menu) does this.



13. A **Save** dialog box will appear and the directory where the NC code is to be saved to can be set. It is important to remember where the NC code is placed, as the file generated has to be subsequently sent to the NC machine. Once the directory has been selected a file name can be entered and the **Save** button pressed.
14. A beginning rapid can be added if required, by answering **Yes** to **Rapid at the start**. This option will add a first (non cutting) movement to a part so a known initial reference point is set. It may aid in cutting out single parts on the NC machine. For this example answer **NO**.



15. The Generated NC code can now be verified in FastPLOT by answering ***Yes*** to ***Leave FastCAM and Verify part.*** TEMP.CNC? This operation is recommended to ensure that the NC code generated in FastCAM is what was expected.



16. FastCAM will close and FastPLOT will open displaying the generated NC code. The program can now be saved to a floppy disk or sent to the cutting machine using a DNC link if available.

NESTING THEORY AND APPLICATION

1-17-1 INTRODUCTION

The *FastCAM* suite offers, "True shape Automatic nesting", which can be a combination of Interactive or "Manual" nesting and Automatic Nesting. Nesting functions are available in the FastNEST package that is sold separately to the FastCAM drawing program. NC code and DXF or CAM files can be nested in the professional version of FastNEST.

1-17-2 FastNEST INTERACTIVE Nesting

This method is as the name implies. The program user is interacting with the nesting process themselves. They have a hands on role within the program. It is possible to Move, Move and Rotate, Shuffle, Add, Delete and Array parts around, and on the plate. This method of nesting is the final phase of the part nesting process. The Automatic nesting procedure returns to Interactive nesting to enable user interaction and Numeric Code output for the Nest.

1-17-3 FastNEST AUTOMATIC Nesting

This method enables the user to enter a Cutlist of parts to be nested. A cutlist is a list of parts of a particular Grade and Thickness of material that must be cut. The user selects the parts to be nested, enters the required number to be cut and specifies the Plate size or shape to be used. The parts are then automatically placed on the plate from Largest to Smallest. The user may set the priority of the part placement, by placing a higher priority to certain parts. This then by-passes the Largest to smaller part placement.

The advantages of this nesting is its ability to recognize the Shape of the part to be nested.

A triangular part is seen as a triangular part. Discs or rings are seen as they are. Even parts that have holes in them are seen as opportunities for the program to place parts into the unused material. Hence this nesting method is very powerful.

Once the plate has been filled, your cutlist is again displayed. Shown are the number of each part nested and the remaining to be nested. You can accept the nest as is or optimize it yourself by moving the parts on the plate.



*For further information on nesting, see the **FastNEST** Reference Manual .*

2-1

CHAPTER 2 - FILES MENU



The Files MENU contains menu items for Saving and Restoring FastCAM geometry files. These files are automatically assigned the file extension of (.CAM). Contained in this menu are also the menu items for the importation of CAD (DXF) and (IGES) files.

Any file followed by (.CAM) extension is a FastCAM geometry file. If the file name is not followed by an extension, then the file is a FastPLOT NC Code file. If a file has already been saved and the operator requests to save the file again an overwrite prompt will appear. The ability to create Sub Directories (*Folders*) is available when saving any file in FastCAM. Refer to your Windows manual for further detail.

2-1-1

FILES MENU

Save.

The SAVE function is used to save your current drawing as a FastCAM drawing file. All drawings saved with this function are automatically assigned a file extension of (CAM).

2-1-2 FILES MENU Restore

The RESTORE function is used to restore or Open on the screen, a file that has been previously saved with the FastCAM SAVE function. The Scan File dialog box will appear and display a preview of the part. When the required CAM file is found the Enter Key or “Select” button can be pressed. The part will then be loaded into FastCAM.



2-1-3 FILES MENU New

The NEW function clears the drawing screen and starts a new part program. Be sure to save any drawings before using this function. Once new is selected and you have answered YES to the screen prompt, all of the current drawing is lost if it has not been previously saved.

2-1-4 FILES MENU DXF Restore

The DXF RESTORE function is used to Open or import a DXF file from a CAD system. Currently FastCAM is compatible with the

2002 version of AutoCAD®. Splines are accepted up to AutoCAD® version 14. Variations in the application of splines may exist in various CAD systems and versions. It is recommended that splines are removed in the native cad system before importation into FastCAM. The DXF restore function can also be used to load DWG files if the option has been enabled. Once the DXF options form has been completed and enter pressed, the files of type drop down can be changed to DWG on the DXF filename form. The DWG file will be handled in the same manner as a DXF. You have the option of choosing between Single and Bulk restoration. **(Note: BULK DXF Restore is a FastCAM® optional feature.)**

Whether you select Single or Bulk, you will need to also select the: **Unit type**, ie..Inch, Metric, or User Defined; and

Cad Clean, Yes or No Cad Clean Removes zero length entities, dimensioning and blocking. This ensures only the required drawing is loaded.

Apply Z Axis Extrusion Selected or Deselected. If selected the loaded DXF file will be folded along the Z axis, merging entities. This may be required on some parts drawn in 3D modeling software. This function is also used to load splined files. If a DXF file is splined this option must be selected.

Check for Orphans Selected or Deselected. If selected the loaded DXF file will be scanned and any short or needless entities will be deleted. This function may clean a poorly drawn DXF file removing construction lines and other short or redundant entities.

SINGLE:

Allows for the loading of a single DXF file. A single file is loaded by double clicking on it in the “Scan File” window or pressing the “Select” button. The file is then loaded into FastCAM.

BULK:

Select your files, either one or several (refer to your Windows manual, for help on selecting multiple files), and then click **SELECT**. You will then be presented with the Part Data box, which allows the user to input Material Type, Thickness and other details. An Automatic Overwrite function will then be asked, followed by the new CAM file location box. All the DXF files will be automatically saved as CAM files and the last file to be processed will be displayed on the screen.

2-1-5 FILES MENU DXF Save

The DXF SAVE function will save the current geometry as a DXF file to be read back into a CAD system.

2-1-6 FILES MENU IGES Restore

The IGES RESTORE function is used to restore or import a (IGES) file from a CAD system. The process works similarly to the DXF Restore.

2-1-7 FILES MENU IGES Save

The IGES SAVE function will save the current geometry as an (IGES) file to be read back into a CAD system.

2-1-8 FILES MENU HPGL Restore

HPGL (Hewlett Packard Graphics Language)

The HPGL RESTORE function is used to restore or import a (HPGL) file from a CAD system.

This format is most commonly found in graphics packages such as Corel Draw. The file can be very large and because arcs are

most often represented as a myriad of small lines these files can cause jerky motion of machinery when converted to NC code. HPGL files should be CAD-Compressed to reduce the line numbers and smooth the output. **NOTE: HPGL is a FastCAM® optional feature.**

2-1-9 FILES MENU DSTV Restore

DSTV is the German Standard for Steel Construction.

The DSTV RESTORE function is used to restore or import a (DSTV) file from a CAD system.

DSTV is much more powerful than the CAD DXF format and can fully describe the connection (holes) details for steel construction including the material and section. CAD packages such as Xsteel, StruCAD BoCAD and others have DSTV like output which can be utilized by FastBEAM® and FastCAM®.

NOTE: DSTV is a FastCAM® optional feature.

2-1-10 FILES MENU StruCAD Restore

StruCAD is the file type used by the 3-Dimensional CAD system of the same name.

The StruCAD RESTORE function is used to restore or import a (StruCAD -2D) file from a CAD system. **NOTE: StruCAD is a FastCAM® optional feature.**

2-1-11 FILES MENU Plot Screen

The Plot Screen function allows the user to send a graphical copy of what appears on the screen to your selected printer/plotter. The plot will appear at the same Magnification level as what appears on the screen. This may also be done by pressing the “F8” key.

2-1-12 FILES MENU Scale Plot

The Scale Plot function allows the user to Plot the Screen, as in **ITEM 2-1-11**, but with the option of selecting a desired Scale factor, eg: 1:1 or 1:5.

2-1-13 FILES MENU Part Data

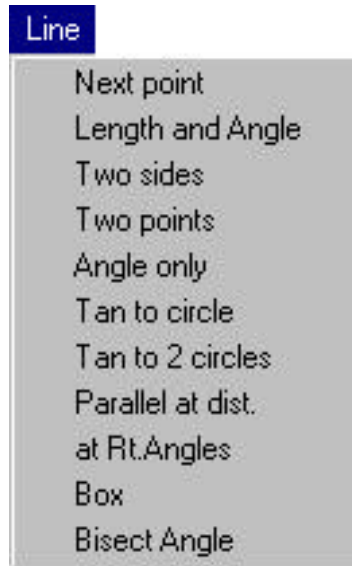
This function allows the Part Data Screen to be viewed and edited. The file must be saved after any edit to record the changes to your data. The information held in this section may appear as comments in the header section of the NC file (depending on your setup), and may be required to allocate Feed Rates and Kerf values. *(Please refer to your Software Supplier for further details.)*

2-1-14 FILES MENU Exit

This will allow you to EXIT from the program. If anything is unsaved it will ask you if you would like to save the CAM File before exiting.

2-2

LINE MENU



Line drawing with FastCAM will fall into one of two categories. These are:

1. **Lines between points.**
2. **Infinite lines.**

Lines between points are standard constructions and can be accomplished through one of the methods listed below. Infinite lines are a very important feature of FastCAM. They can be used as construction lines to reference or define areas on the screen, and then removed when the construction is complete. Infinite lines can also be drawn and trimmed to the desired length.

2-2-1

LINE MENU

Next point/Point menu

NEXT POINT is used to set points on the screen in a sequential order. An example of sequential order is to draw a box on a piece of paper without removing your pencil from the pad. Nonsequential drawing would be to draw the left side, right side, and top and bottom of a box. FastCAM allows the next point to be referenced with the standard **Point menu**. It is important to remember that the line will always be drawn from the last line endpoint, no matter how the next point is referenced. This is a common confusion to the beginner. It is also important to remember that the center of the initial drawing screen is always referenced as (0,0). If **Next point** is used with a blank screen, the line will be drawn from the center of the screen to the referenced next endpoint of the line.



*For a complete description of the **Point menu** see point description under the "POINT MENU" in this chapter.*

2-2-2

LINE MENU

Length & Angle

To use this function the length and angle of the line to be constructed must be known. When **Length & Angle** is selected, the cursor will be locked on the last point entered on the screen. If a different reference point is desired, choose **NO** to the prompt, and then check the message displayed on the screen, to see if that is the correct point type (either screen position or control point). If it is not, simply click the right mouse button and the Point menu will be displayed. Indicate the new reference point by one of the methods in the **Point Menu**.

*For more information on point placement see **Main menu / Point menu**.*

After a reference point has been determined FastCAM will prompt to enter the length needed. When the length has been entered,

choose an angle from the **Angle menu** now on the screen. *For description of angle menu see Angle menu.* The line will then be drawn as specified above.

2-2-3**LINE MENU****Two sides**

Two sides establishes the angle by prompting for an (x,y) co-ordinate relative to your current position. Draftsmen in many cases will give an angle of a component by giving the engineer two sides of a Right angle triangle to establish an angle. For example (1,1) is a 45 degree angle and (-1,-1) will establish a 135 degree angle. To enter a line using **Two sides**, the (x,y) co-ordinates of a point on the next line must be known. Entering the co-ordinates at the computer's request will draw a line to the next point. In this way, an angle can be established.

2-2-4**LINE MENU****Two points**

TWO POINTS is one of the more commonly used functions in the line menu. To draw a line using the two point method, the programmer must have enough information to position two points on the screen. The end points are found by one of the options in the standard **Points menu**.

In **Two points** the menu appears as the **First point/Point menu** and the **Second point/Point menu**. When both points have been specified, a line will be drawn between them. The points can be of the same type or they may be different, eg: First Point = Control Point, Second Point = Absolute Co-ordinates.

2-2-5

LINE MENU

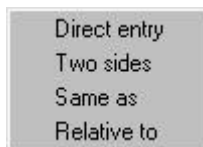
Infinite Line

INFINITE LINES can be created by using the **Angle only** option. Select **Angle only** and set a point using the **Point on line/Point menu**. This is the point an infinite line will pass through. The **Angle menu** will appear. Select **Direct entry**. Enter the degree angle of the infinite line. An **Infinite line** will be drawn on the graphics screen.

2-2-6

LINE MENU

ANGLE MENU



2-2-6-1

ANGLE MENU

Direct entry

Allows the operator to enter an angle directly from the keyboard. Negative angles are accepted.
Select **Direct entry** and type in the required angle when the angle prompt appears.

2-2-6-2

ANGLE MENU

Two sides

Allows the angle of the line to be specified by entering an (x,y) co-ordinate. For example if the (x,y) co-ordinates of 5,5 are entered, the angle of the line will be 45 degrees.

2-2-6-3

ANGLE MENU

Same as

This function will use the angle of an existing line. When the mouse cursor appears, click onto an existing line and the angle of that line will be used to construct the new line.

2-2-6-4

ANGLE MENU

Relative to

Allows a line to be constructed at an angle which is relative to an existing line. When the mouse cursor appears, click on to an existing line and specify the angle of your line relative to the line selected.

2-2-7 **LINE MENU Tan to circle**

Used to connect a **Finite** or **Infinite** line tangent to one circle. First indicate the circle or arc to be modified by picking a point close to the tangent. When choosing a point it is important to be close to the desired tangent point. If the point is too far off the tangent, a line will either be drawn to the wrong point or not drawn at all. Once the point is indicated the line will be drawn at the tangent point. The **Tangent circle / Line angle menu** will appear. At this time the line angle can be defined by angle only, using the generic angle menu, or by a given point on the screen using the **Point on line / Point menu**.

2-2-8 **LINE MENU Tan to 2 Circles**

Tangent to 2 circles allows a line to be drawn tangent, or cross tangent between two circles. To draw a tangent line between two circles, pick a point near the tangent point on the first circle by clicking the left mouse key when the cross hairs are on the desired point. Pick a point near the tangent of the second circle. A line tangent to two circles will be drawn.

To draw cross tangents, pick a tangent point on opposite sides of the circles in the same manner as above. The line will be drawn between the circles. When the tangents have been drawn you can use the trim function to remove unwanted parts of the circles.

2-2-9 **LINE MENU Parallel at dist**

Allows either an **Infinite**, **Same length** or **Extended** line to be drawn parallel to an existing line at a specified distance away.

2-2-10 LINE MENU at Rt Angles

This function allows a line to be drawn at right angles to any point on a line. First a point on the line must be defined using the POINT MENU. When the point is defined, an infinite line is drawn at right angles to the chosen entity.

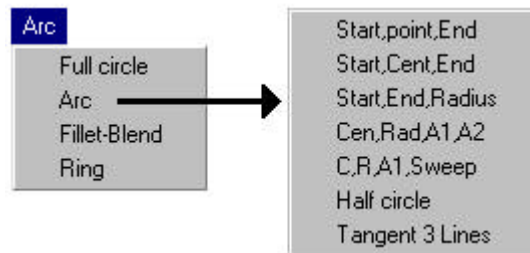
2-2-11 LINE MENU Box

Probably the most important line function. The ability to draw a box has been incorporated into FastCAM as a quick and easy to use function. A box can be used to quickly create the outline of any rectangular part to be drawn. To draw a box, use the FIRST and SECOND CORNER POINT MENUS to locate **diagonal** or **cross** corners of the box. The box will be drawn using four separate lines based on these two co-ordinates.

2-2-12 LINE MENU Bisect Angle

Bisect Angle allows the user to quickly draw an infinite line between two other lines, at half of their included angle.

2-3 ARC MENU



2-3-1 ARC MENU Full circle

Full circle allows an arc of a given radius to be drawn on the screen. FastCAM prompts the operator to enter a **Radius** of the circle. A diameter will be accepted when the value entered is followed by the letter "D" or "d". After specifying a radius, locate the center of the arc by using one of the options on the **Circle center Point menu**. (See **Point menu** for more information on point location).

2-3-2 ARC MENU ARC

The Specified arcs, Arc menu is displayed when Arc is selected from the Arc menu. Following is a description of each menu item.

2-3-2-1 Start, Point, End

To create a three point arc, a start, middle, and end point must be known. The points are selected by using the **Start, Middle** and **End Point Menus** that appear sequentially after each point is entered. Enter the point by following the prompts, and the arcs will be drawn on the screen.

2-3-2-2 Start, Cent, End

This function allows an arc to be drawn when two points on the arc (Start and End), along with the center point of the arc are known. Follow the prompts under the **Start**, **Center** and **End Point** menus, to enter the points.

2-3-2-3 Start, End, Radius

This would be the most frequently utilized specified arc. This function allows arcs to be drawn when the **Radius** along with the **Start and End** points of the arc are known. Follow the prompts to enter the points.

2-3-2-4 Cen, Rad, A1, A2

To use this function the **Center** point, **Radius**, **Start angle** and **End angle** must be known. This allows an arc to be drawn between given or known angles. Follow prompts to enter the required angles and points.

2-3-2-5 C, R, A1, Sweep

To use this function, **Center**, **Radius**, **Start angle** and the number of **Degrees** to **Sweep** the arc must be known. Sweep refers to the total number of degrees between the start and the end of the arc. For example, if the start angle is 45 degrees, and the sweep is 30 degrees in the CCW direction, the stop angle is 75 degrees. Follow the prompts to enter the required information.

2-3-2-6 Half Circle

Use this function to draw a half circle starting and finishing on two specified points. For example you may wish to draw a slot with rounded ends. After drawing the two parallel

lines you can utilize the half circle function to create the radius ends by clicking onto the end points of the lines.

2-3-2-7 Tangent 3 Lines

This function will draw an ARC, between 3 different lines. The Radius is not needed, simply select the 3 lines and the Radius is drawn.

GENERAL NOTES ON ARCS AND CIRCLES

Angles can be entered either positive or negative. Positive angles move counterclockwise and increase in a counter clockwise direction. Negative angles move clockwise and increase in a clockwise direction. When entering a size for an arc or circle the number refers to the radius. Diameters can be entered, by following the entered number with a "D" or "d". Remember that the Trim or Edit Entity functions can be used to remove unwanted parts of circles and arcs.

2-3-3 **ARC MENU** **Fillet-Blend**

Fillet blends are very often used in part geometry. FastCAM allows for easy construction of the following blends. Please see your Appendix A (*Construction Examples*) for examples.

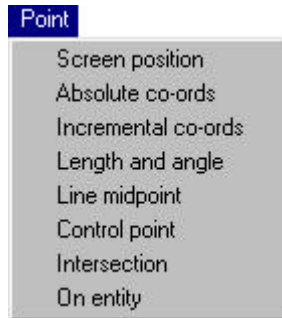
- Blends between intersecting lines.
- Blends between a point and a line.
- Blends between two circles, either inside blends or outside blends.
- Blends between a point, and a circle.

The Fillet Blend will first ask you for a Radius of the Blend, and then ask you to select the first entity and then the second entity needed to blend.

2-3-4 **ARC MENU** **Ring**

Ring simply enables the user to draw two(2) concentric circles. Specified as an ID (Inside diameter) & OD (Outside Diameter), the ring function is very useful when creating Flanges.

2-4 POINT MENU



The following is a general description of the POINT MENU. Many of the drawing functions refer to differently named POINT MENUS, but the actual menu and the method to create the points are the same. If there is any question about how to create a point any place in the system, this section should be referred to.



NOTE: When selecting any menu item that requires a point to be chosen, FastCAM will first select the **LAST** Point option chosen from the previous operation. If this point option is incorrect, simply click the right mouse button and the Point menu will appear for a different selection. When first starting FastCAM the initial point preset will be for a **SCREEN POSITION**.

2-4-1 POINT MENU Screen position

SCREEN POSITION allows a point to be placed any where on the screen. Move the cross hairs until the next point is located and click the LEFT mouse button to set the point. This method of setting points will probably be used only in sketching due to the inability to accurately locate the points.

2-4-2 POINT MENU Absolute Co-ords

Absolute refers to the coordinate system where all measurements for the part are referenced from a common or absolute zero position. Absolute zero is located at the center of the initial drawing screen when FastCAM is started. When a point is set using absolute zero an X, Y value from 0, 0 must be known. It is not necessary to initialize the point to use it for reference, but for the beginner to make part drawings more easily understood, a point should be set at absolute zero.

2-4-3 POINT MENU Incremental Co-ords

INCREMENTAL COORDINATES uses the last point entered on the screen, or any point selected, to reference the next (X, Y) point. If this is the required reference point, accept the point with the LEFT mouse button. If a new point is desired, push the RIGHT mouse button, move the cross hairs to a new point, and click the LEFT mouse button. Click the LEFT button on “YES” to confirm the position. Enter the X, Y values to reach the next point.

2-4-4 POINT MENU Length And Angle

LENGTH AND ANGLE can be used if the distance and the angle to the next point is known. The first prompt is for the length to the next point to be set. After indicating the distance the ANGLE MENU appears.

2-4-5 POINT MENU Midpoint

The MIDPOINT function locates the center point of a Finite line. Just click on a line and the mid-point will appear on the screen as a yellow cross.

2-4-6 POINT MENU Control Point

Any existing control point can be used to locate a real point. Control points are represented on the screen as red crosses. They are the reference points used to construct arc centers, arc endpoints, and line endpoints for the entities on the screen. Any control point can be converted to a real point by using the **Point Menu - Control Point** option.

2-4-7 POINT MENU Intersection

The INTERSECTION function allows a point to be set at any intersection of any two entities. To intersect two entities first select the intersection function and place the mouse cursor on one entity and press the left mouse button. Repeat for the other entity to be intersected and a point will be placed at the intersection. When creating an intersection between an entity, be certain that a point ON the arc or line to be intersected is picked and NOT an adjoining arc or line.

An intersection point may also be found even if the two entities do not cross but a projection extension of these entities would.

2-4-8 POINT MENU On Entity

The ON ENTITY function allows you to place a real point on any entity in any position. This function is NOT to be used if the point is to be placed in an exact position on your drawing.

2-5

CONSTRUCTS MENU

2-5-1

CONSTRUCTS MENU**Ellipse.**

To create a true Ellipse, enter the two major axis points. (points on the Ellipse with the greatest distance). FastCAM will prompt for the minor or smaller axis diameter. Once the axis has been entered the MINIMUM DEVIATION IN FAIRED CURVE or the tolerance of the Ellipse is set. The default is to .040" (1.1mm.) The smaller the tolerance the more accurate the Ellipse. This is accomplished by using more arcs to construct the Ellipse. The Ellipse will be drawn and the operator is asked if it is satisfactory. *(Note: if control points are displayed the arc centers used to create the Ellipse can be seen on the screen).*

2-5-2

CONSTRUCTS MENU**Faired Line.**

Curves can be created with a truly FAIRED LINE, or a line passing through the given points. This is an advantage which FastCAM has over CAD systems which use splined curves, where the lines pass close to the indicated points.

To create a FAIRED LINE: Identify the points on the screen to be used in the construction of the line. When the last point is picked, push the RIGHT mouse key or the ESC key on the keyboard. Specify if you wish to close the faired line back to the starting point with either a Yes or No response and finally specify the minimum deviation, or tolerance, allowed for the Arc Fairing process. Press the LEFT mouse key and the FAIRED LINE will be drawn. Reject if not satisfactory.

2-5-3

CONSTRUCTS MENU CORNERS MENU



The corners menu is provided to simplify commonly used corner constructions. These include fillet, chamfer, arc chamfer, sharp corner, loops and notched corners. To use any of the constructions select Corners from within the Constructs menu, and select the function. Once the function is accessed specify the distance from the corner and the mouse cursor is then displayed on the screen, simply choose the two lines which form the corner, and the corner will be modified with the excess lines trimmed.

2-5-3-1

CORNERS MENU

Fillet

FILLET: Convex curves used on corners to smooth and distribute stress. The corners fillet function can be used only on straight line intersections. If a fillet is to be placed between a line and an arc, or two arcs, the Fillet-Blend function found under ARC MENU should be used.

2-5-3-2**CORNERS MENU****Chamfer**

CHAMFER: Used to place angled corners on a box, or an appropriate angle corner on any line intersection. The chamfer angle is calculated by specifying the two distances along intersecting lines you wish to chamfer. The corner is then trimmed and the chamfered line is added.

2-5-3-3**CORNERS MENU****Arc Chamfer**

ARC CHAMFER: Used to place the center of an arc at the intersection of the lines indicated. The resulting corner will resemble a round notch with all excess lines trimmed. Useful in providing weld clearance.

2-5-3-4**CORNERS MENU****SharpCorners**

SHARP CORNERS: Extends the lines indicated past the intersection by the entered distance as well as inserting a line between the new line end points. Sharp corners are commonly used for plasma cutting turnarounds to prevent rounding of the corner by the cutting arc.

2-5-3-5**CORNERS MENU****Loop**

LOOP: Extends the two lines indicated past their intersection by the corner distance entered, then places a loop or arc with the radius of the corner distance on the ends of the lines. This construction is commonly used in plasma cutting turn around to produce square corners and also is the corner of choice for VBA (Variable Bevel Angle) plasma cutting.

2-5-3-6**CORNERS MENU****Notch Corner**

NOTCH CORNER: Used to place Right angled notched corners on a box, or an appropriate 90 Deg angled corner

on any line intersection. The notch size is specified by the input of two distances along intersecting lines you wish to notch. The corner is then trimmed and the notched corner is added.

2-5-4 CONSTRUCTS MENU Offset Contour.

The OFFSET CONTOUR function is used to offset the inside or outside contours of any part, a pre determined amount. This function is particularly useful when a part is to be cut without any kerf, or when the part is required to be oversized for machining.

2-5-5 CONSTRUCTS MENU Line Notch

The LINE NOTCH function is used to create a Convex or Concave rectangular notch on any line. You must specify the Center Point, Width and Depth of the notch to be created. After specifying the location and size you click the mouse cursor on the side of the line you wish the notch to be formed.

2-5-6 CONSTRUCTS MENU Insert Gap

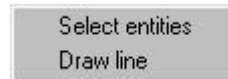
The INSERT GAP function is used to insert a gap or break in any Line or Arc. This function is useful in removing a specified length out of a Line or Arc at specific locations before Array or Block insertions. Stitch cutting can be created with this function if you first assign your cutter paths and then insert gaps of the required stitch length at the locations desired.

2-5-7 CONSTRUCTS MENU Bridging

The BRIDGE function is used to create either Simple or OverCut Bridges between parts for continuous cutting. Bridges between

parts reduce the amount of pierces required to cut a group of parts or a nest; which results in reduced cutting time and consumable costs.

After selecting the desired Bridge method, you will be asked to choose between the following types of selection processes:



The **Select Entities** method allows you to just choose two entities at a time. Basically bridging two parts together at a time.

The **Draw Line** option allows you to draw a line across multiple parts and complete the bridging between those parts in one pass.

A **Simple bridge** is created when a Positive dimension is input. By clicking onto a location on the two parts to be bridged with the mouse cursor, parallel lines are automatically inserted and the parts are trimmed between the bridge. This type of bridge is useful if you do not want your nested parts to be separated from your plate skeleton.

An **Overcut bridge** is created when a Positive dimension is input. By clicking onto a location on the two parts to be bridged with the mouse cursor, parallel lines are automatically inserted and the parts are automatically edited between the bridge to enable continuous cutting.. This type of bridge is useful in eliminating a separate pierce for each part to be cut in the nest. If an Overcut bridge is used your parts are completely separated from the plate skeleton and can be removed.

2-5-8

CONSTRUCTS MENU

Faired Spiral

Faired spiral is a useful utility designed to develop Involute Curves. It was developed specifically to allow a FastCAM customer to peel a roll of rubber with a waterjet, turning the roll

into a flat rectangular sheet. The concept of the faired spiral is that you have

1. a centre for ever increasing radius, usually 0,0
2. a fixed distance between turns being the desired thickness of the final sheet
3. a final diameter, the diameter of the roll at the start.

The first few mm may have to be trimmed to produce a rectangular sheet of constant thickness.

2-5-9

CONSTRUCTS MENU

Contour Points

Contour Points is a useful tool when required to establish construction points which are equally spaced along a contour; typically a curve. The user will be asked for a starting point (control point), from where the array of points will begin. If the contour is closed then a directional arrow will appear so you can signify the direction of the points. The spacing will then be required and finally the number of points.

This function is particularly useful in Shipbuilding, where there is a need to determine equally spaced points around the outside of a frame.

2-5-10

CONSTRUCTS MENU

Kerf Compensate

The Kerf Compensate function will adjust the loaded CAM or DXF geometry file by the amount of Kerf entered. The Kerf entered is the cutter radius. If a part is Kerf compensated with this function, then no Kerf needs to be added when outputting NC code.

2-5-11

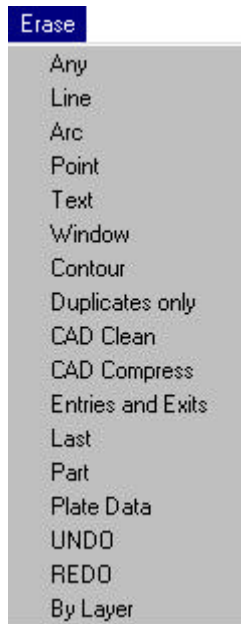
CONSTRUCTS MENU

Slot

The Slot option will draw a rectangular slot with optional radiused

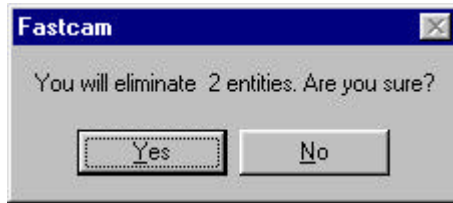
corners in the position indicated. To use the function first enter a center position then enter the overall width and height of the slot required. A corner radius is then required. This can be set to zero if a rectangular slot is required. The angle at which the slot is to be drawn on is now required. The default is zero degrees. If the radius entered is half the height of the slot, a frog mouth shape will be produced.

2-6 ERASE MENU



2-6-1 ERASE MENU Items 1 to 5

To ERASE an entity, first choose the type of entity to ERASE, from the menu items: ANY, LINE, ARC, POINT or TEXT. Pick the items, either individually or multiple by placing the mouse cursor on the entity and pressing the left mouse button. The selected entities will be highlighted. When all items have been picked, press the right mouse button, and the entities will be erased off the screen. To make the erasures permanent, select YES at the Prompt:



2-6-2 **ERASE MENU** **Window**

WINDOW allows a box to be placed around all objects to be erased. The type of entities to be erased must be specified, (ALL, LINES, ARCS, POINTS or TEXT). Only entities which fall entirely within the window will be erased. After defining the window press the RIGHT mouse button and the objects will be ERASED. If the erasure is correct, select YES; otherwise reject by selecting NO. This function will not ERASE Infinite Lines since they do not fall entirely within the window.

2-6-3 **ERASE MENU** **Contour**

The ERASE CONTOUR function is used to erase a number of connected Lines and Arcs. This function is useful for eliminating a complete part or any contour on the drawing screen. Click on to any Line or Arc in the contour and the entities are erased. You are then prompted if you are sure. If the erasure is correct, select YES, otherwise reject by selecting NO

2-6-4 **ERASE MENU** **Duplicates only**

The ERASE DUPLICATES ONLY function is used to erase any duplicate entities on the drawing. These duplicate entities may have been created by drawing a Line, Arc or Point over the top of

an existing entity. Another possible cause for duplicates is when a file is imported from a CAD program. CAD files are notorious for their duplicate entity creation.

These entities are erased to eliminate any possibility in producing double cutting of a Line or Arc in the Pathing process.

2-6-5

ERASE MENU

CAD Clean

The CAD CLEAN function is used to clean up any imported CAD files, or indeed your own FastCAM drawings. This function will remove any unwanted Blocking, Dimensioning and Zero Length lines created by the CAD system. A “Check for Orphans” YES/NO message box will appear. Answering Yes will scan the loaded drawing for “Lost” or orphan entities. These are entities that do not make up any part of the drawing . An example would be a short line outside the defining perimeter of a part, or short entities on the perimeter that are obviously left over construction lines. The check for orphans command while thorough is not a total solution; too much or too little of a drawing could be removed.

2-6-6

ERASE MENU

CAD Compress

The CAD COMPRESS function is used to Compress or reduce the number of entities within an Imported drawing from a CAD system. Cad systems are notorious for creating parts that are comprised of hundreds of small Lines or Arcs. This is particularly evident in letters created in sign making packages. Each entity on your drawing produces a separate Block or Line of Numeric Code when the NC code is generated. A large number of small movements can result in a very large NC file for a relatively simple drawing, and the cut quality will be affected.

The operation of this function begins by making a Backup copy of your file in your current directory. You are then prompted for the maximum line length to round and the maximum shape divergence you will allow from the original. You are also prompted if you

wish to compress Arcs which requires a Yes or No response. the function will then attempt to compress your drawing using the guidelines you have just set.

If no compression was possible you are returned to the original drawing and a message is displayed on the screen informing you that no compression of your file was performed. If the file was compressed you are informed of the amount of compression obtained and you are then prompted if you wish to compare the new drawing against the original. This is recommended as you may have deleted some important entities on your part that were eliminated due to the parameters you set. Depending upon the result, you can then select Yes, No or Retry from the Satisfactory menu.

2-6-7

ERASE MENU

Entries & Exits

The ERASE ENTRIES & EXITS function is used to eliminate all Automatic Entry and Exit or Lead-in and Lead-out entities on your parts. This function is most useful after you have Converted an existing NC Code program back to a CAM file and you wish to reassign a new cutter path. You are also prompted if you wish to remove the Cutting layers or Tool Paths at the same time.

An important use for this function is if you have nested all of your parts onto a plate and generated your NC code for the nest. At this time you may wish to convert your Nest Code back to a CAM file, Erase your Entries and Exits and then FastPATH your entire nest. This procedure is effective in maintaining your entries at a specific location on all of your parts.

2-6-8

ERASE MENU

Last

The ERASE LAST function will eliminate the last entity added to the screen. You can effectively erase your entire drawing one entity at a time by repeating this function although this is not the recommended method. Be sure to follow the screen prompts to confirm the erasure of the last entity.

2-6-9 ERASE MENU Part

The ERASE Part function will allow you to eliminate an entire part at once, taking into consideration, all internal cutouts contained inside a Part.

2-6-10 ERASE MENU Plate Data

Erase Plate Data will only work when a completed CAM file of a Nest is opened within FastCAM. When this is done the Plate outline will appear around the outside of the parts. To erase this outline, and any associated data select Erase Plate Data, and it will be stripped.

Most useful when you need to break a nest back into individual parts, via EXPLODE.

2-6-11 ERASE MENU Undo

If a mistake has been made, this option undoes the last operation. The last 20 steps can be undone.

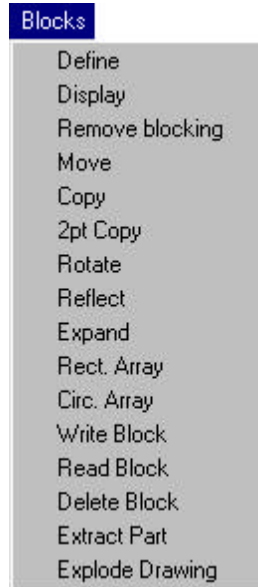
2-6-12 ERASE MENU Redo

Redoes the last undo step. The last 20 undo steps can be reversed.

2-6-13 ERASE MENU By Layer

If any entities are on a specific CAD layer, using this option will delete only those entities on the selected layer. This option can be used to quickly delete only construction lines, drawn on a separate layer, from the drawing.

2-7

BLOCK MENU**INTRODUCTION to BLOCKING**

BLOCKING is a specialized group of functions designed to allow the programmer to define a group or block of entities and **MOVE**, **COPY**, **ROTATE**, **REFLECT** (mirror) or **ARRAY** the block.

Specific uses for **BLOCKING** as mentioned above include moving, rotating or copying objects. Another common use is creating a part that has symmetry or making a sprocket or any part with a repeating pattern. First draw the pattern to repeat, define it as a block, and array the pattern the required number of repetitions.

Another valuable feature of the **BLOCKS** menu is the ability to store commonly used geometry for future use. The **READ BLOCK** and **WRITE BLOCK** functions allow for a library of constructions or parts to be stored.

The EXTRACT PART function can be utilized to separate parts to individual CAM files. For example, you may have a number of parts drawn on your screen or a complete nest of parts that you have created from converting a Nest to a CAM file. The extract part function can then be used to separate these parts into individual CAM files.

An important side note to this function is its ability to use internal text for the CAM filename. If for example you have the Part name in text enclosed within the part boundary, the extract part function will utilize this name for the CAM filename.

2-7-1 BLOCK MENU Define

2-7-1-1 BLOCK SELECT MENU All

Before any of the BLOCKING functions can be used, an object must first be defined. Defining an object tells the computer what entities to include in a specified block. The define all function permits the user to select individual entities of any type. Once all entities have been selected, the right mouse button finalizes the selection. A block origin needs to then be defined.



If an entity is defined in one block, it can not be part of another block definition. FastCAM will give a prompt telling the operator that part of the current block is defined in a previous BLOCK.

2-7-1-1 BLOCK SELECT MENU Lines

The DEFINE LINES function is used to block only Lines. A block origin is then requested.



Infinite Lines cannot be defined as part of a block.

2-7-1-3 BLOCK SELECT MENU Arcs

The DEFINE ARCS function is used to block only Arcs. A block origin is then requested.

2-7-1-4 BLOCK SELECT MENU Points

The DEFINE POINTS function is used to block only Real Points. A block origin is then requested.

2-7-1-5 BLOCK SELECT MENU Text

The DEFINE TEXT function is used to block only Text Origin Points. A block origin is then requested.

2-7-1-6 BLOCK SELECT MENU Contour

The DEFINE CONTOUR function is used differently from the above definition procedures. A Contour is defined by clicking onto any entity within the contour and pressing the Left mouse button. The function will then block all entities connected along the contour. A block origin is then requested.

2-7-1-7 BLOCK SELECT MENU Parts

The DEFINE PART function is used in the same manner as the Define Contour function. A Part is defined by clicking onto any entity within the contour and pressing the Left mouse button. The function will then block all entities connected along the contour and any geometry contained within the contour. If the Part name is contained within the

contour in Text form, the part name will be used to name the Block. A block origin is then requested.

2-7-1-8 BLOCK SELECT MENU Window

This menu item contains a sub menu of the items described above (All, Lines, Arcs, Points, Text, Character). Using the window, the selected entites will be added to the defined block. The window must fully enclose the entites required.

2-7-1-9 BLOCK SELECT MENU Character

To add your own "Font" character to the fonts.cam file this block option is used. It allows for the addition of special Text Marking characters that can be marked onto a part with the use of FastCAM's Text Marker option and a suitable marking device (Air Scribe or Plasma Marker for example.) The characters are drawn in FastCAM and labeled with a single character (A, B, C ... etc) after defined using this Character function.

2-7-3 BLOCK MENU Display

Use DISPLAY to see the BLOCKS and their numbers. Make sure all BLOCKS are present on the viewing screen by selecting AUTOSCALE in the PLOT MENU. Select DISPLAY. Any entities which have been BLOCKED will change color and be referenced with the BLOCK number.

2-7-4 BLOCK MENU Remove Blocking

Used to remove BLOCKING, either by individual BLOCK number, or by removing all BLOCKING at once. Select REMOVE BLOCKING and follow the prompts.

2-7-5

BLOCK MENU**Move**

MOVE allows the BLOCK to be moved to any place a POINT can be set, by using the POINT MENU presented in this function. Select MOVE. Indicate the BLOCK to be moved (*if more than one block is defined*) by placing the highlight bar on the number and pressing the LEFT mouse button. If no numbers are displayed, then you have only one Block defined. A BLOCK MUST BE DEFINED BEFORE IT CAN BE MOVED. When moving a BLOCK, the BLOCK origin will be moved to the POINT defined by using the POINT MENU presented in this function. You are finally asked to confirm the move. Answer YES or NO.

2-7-6

BLOCK MENU**Copy**

COPY allows replicas of the object to be drawn on the screen. Select COPY. Indicate the BLOCK to be copied. Use the POINT MENU to define a POINT where the BLOCK origin will be copied. Accept or reject the copied object. This function will repeat till cancelled allowing for the multiple copying of blocks.

2-7-7

BLOCK MENU**2 Point Copy**

TWO POINT COPY is used to COPY and ROTATE a BLOCK at the same time. To use TWO POINT COPY, the BLOCK must first be defined by using DEFINE under the BLOCK MENU and an origin must be given. When TWO POINT COPY is called up the first prompt asks to define a direction. Using the POINT MENU, set a POINT on the screen to establish the old direction. It is not important where the POINT is set, but it is important to remember where the POINT is located. Next FastCAM asks for a new origin. The new origin is the point the old origin will be moved to when the block is copied. Establish the new origin by using the POINT

MENU displayed by FastCAM. The next prompt is to establish the new direction by using the POINT MENU now on the screen. The old direction point will be oriented so it matches the new direction point. The BLOCK is copied by aligning the old origin point to the new origin, this establishes the location where the BLOCK will be copied. The old direction point is aligned to the new direction point to establish the new direction.

2-7-8 BLOCK MENU Rotate

ROTATE allows a defined block to be rotated around a POINT defined by using the POINT MENU displayed in this function. DEFINE A BLOCK. Select ROTATE. Choose the BLOCK to be ROTATED.(If more than one block exists) Define the center of rotation by using the rotation center POINT MENU. Enter the angle of rotation. The object will be rotated as specified above. If satisfactory, accept the rotation.

2-7-9 BLOCK MENU Reflect

REFLECT is used to make Mirror images, right, left, or when half of the part can be drawn and mirrored to complete the part. The part or entities to be mirrored must first of all be Blocked and a Block origin assigned.

2-7-10 BLOCK MENU Expand

EXPAND is used to either increase or decrease the size of the part by a given factor. The value is a percentage of the original size. If 2 is entered, the resulting block will be twice as large. If .5 is entered the block will be reduced by half. If a small expansion is required you may be wise to utilize the OFFSET CONTOUR function in the CONSTRUCTS MENU.



If a very large expansion is used on a BLOCK which contains arcs, it may not work correctly. As the part is expanded and the rounding errors for figuring arcs and circles increase, endpoint errors or a gap between arcs may occur.

2-7-11**BLOCK MENU****Rect Array**

RECTANGULAR ARRAY is used to copy a repeating pattern in either an X, or Y, direction. Draw an object to be repeated. Define the object as a BLOCK. Select RECT. ARRAY. Array type must be defined. The next screen asks for an X, Y distance and you can then select the point position for the array placement, by "right clicking" and selecting a point type from the POINT MENU. *Do you remember where the block origin is set? It is important.* Set a POINT, using this POINT MENU at the position you wish to have the block origin of the next part to start. This is the offset for the array. Enter the total number of repeats, including the original.

2-7-12**BLOCK MENU****Circ Array**

CIRCULAR ARRAY is used for repeating a pattern in a circular form. An example of using CIRCULAR ARRAY is to draw one tooth of a sprocket and repeat the tooth to complete the part. Draw the part to be arrayed or copied. Define it as a BLOCK. When doing a circular array it is often advantageous to put the block origin at the center of the arrayed part. When the block is rotated, it will be rotated around the origin or the center. Select CIRCULAR ARRAY. Pick the BLOCK that was just defined, to ARRAY. Choose the center of rotation. Give the total number of repetitions required. The next prompt asks for the degrees between repetitions. The default is $(360/\text{number of repetitions})$, which is the number of degrees between the repetitions for a full circle. If the repetitions need to be placed into one quadrant the number to enter at this prompt is, $(90/\text{number of repetitions})$. Accept with the LEFT mouse button and the repetitions will be drawn. If satisfactory, answer yes to the next prompt.

2-7-13**BLOCK MENU****Write Block**

WRITING A BLOCK allows a BLOCK to be saved as a .CAM file. This function is used when placing several parts on the screen to be joined together with a common cut. Define the parts for the common cut as blocks. Write each block to a file, and call all the parts on the screen using read block. Position the parts using move, rotate. Join the parts with a line, and save the entire part as a .CAM file. Select WRITE BLOCK. FastCAM prompts for a second point, so when the block is read back to the screen, there is a point for X, Y orientation. Use the POINT MENU on the text screen to define a second point. Give the block a NAME. The block has now been saved as a .CAM file.

2-7-14**BLOCK MENU****Read Block**

READ BLOCK allows a previously saved block or any other cam file to be placed on the screen. Select READ BLOCK. Choose the .CAM file from the restore box. Pick an insertion point with the INSERTION POINT: POINT MENU. The insertion point is the point where the block origin will be placed on the screen. Pick an insertion angle by placing a POINT on the screen with the INSERTION ANGLE: POINT MENU. The angle of insertion is established by corresponding the second point defined when writing the block with the insertion angle defined above. For accurate block manipulation it is important to know where the second point was defined in the original block. If the part is brought in at the wrong angle simply move, or rotate the part to the correct location.

2-7-15

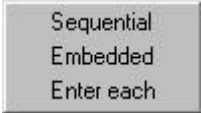
BLOCK MENU**Extract part**

EXTRACT PART allows a single part from a multiple part drawing to be removed from the drawing screen and saved as a .CAM file. Select EXTRACT PART from the menu and you are prompted to indicate the start of the outside contour of the part to be extracted. All drawing geometry enclosed within the boundaries of this contour will be extracted along with the outside contour. If the Part name is contained within the contour in Text form, the part name will be used to name the .CAM file. Type in the .CAM file name desired and press the SAVE Button. "Define Part" is then displayed. Select Yes and save the file normally. For more information please see SAVE under the FILES menu.

2-7-16

BLOCKS MENU**Explode Parts**

The EXPLODE PARTS function is used to break up a Multiple part drawing into individual part geometry files. This function can be used on any geometry drawing created with the FastCAM drawing editor or from an imported file from a CAD package. Explode operates in a similar manner to Extract, although it will perform the process automatically. A Maximum gap in contour must be set first, which specifies the distance permitted between intersections of entities. If there are gaps between intersections you can simply raise the value until it is greater than the gap and each part will be extracted as one. If gaps exist in the contour and the "gap in contour" value is not set correctly, it is possible that an erroneous part quantity may result. You will then have to choose from the following SAVE choices:



Sequential
Embedded
Enter each



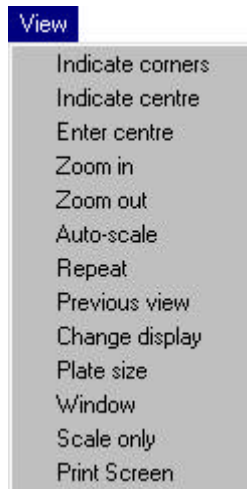
Before using the Explode function, it may be necessary to erase sections of the drawing which are not needed, in order to avoid having them exploded. For example: the Title Block of a drawing, the dimensions....etc.

Sequential: Allows a single prefix to be entered with the program automatically appending an increasing index number to it.

Embedded: Scans the part looking for text inside the part's boundary. If text is found the part is saved with that name.

Enter Each: Filename is manually entered for each part.

2-8

VIEW MENU

The VIEW MENU allows the manipulation of the view on the graphics screen. By using the view menu the programmer can zoom in for a close up view of a section of the part, change the scale of the part, or look at a previous view. The view menu appears both in FastCAM and FastPLOT. In FastPLOT the view menu also has nesting functions.

2-8-1

VIEW MENU**Indicate Corners**

INDICATE CORNERS allows a box to be drawn around a portion of the screen, it then enlarges the contents of the box to full screen view. Select the VIEW MENU from the menu options. Place the highlight bar on the INDICATE CORNERS OPTION and press the LEFT mouse button. The mouse cursor will appear on the graphics screen. Place the cursor to the left and slightly below the portion of the screen to be enlarged. Press the LEFT mouse button. This will set one corner of a box defining the area to be enlarged. Move the cursor to the right and up by using the mouse. A box will be drawn on the screen. When the box encompasses the area to be enlarged, press the LEFT mouse button. The portion of the previous screen

will be enlarged to fill the next viewing screen. Multiple enlargements of the same area are allowed by repeating the same procedure as above.

2-8-2 VIEW MENU Indicate Center

INDICATE CENTER moves the center of the next screen to the position indicated with the graphics cursor on the present screen. This function allows the programmer to pan or move the graphics screen to see entities which may be just out of view on the present screen. It is useful when a portion of a screen has been enlarged using INDICATE CORNERS, and an entity needs to be seen which is just out of the view of the enlargement. Select the VIEW MENU. Place the highlight bar on the INDICATE CENTER and press the LEFT mouse key. The cursor will appear on the graphics screen. Move the cursor to a position on the current screen, say all the way to the right center of the screen, and press the LEFT mouse key. This position will become the center of the next screen.

2-8-3 VIEW MENU Enter Center

Enter center allows an absolute X, Y coordinate to be specified as the center of the next screen. Select ENTER CENTER, specify the required absolute X, Y coordinates, press the LEFT mouse key. You are then prompted for the scale you wish to display. Press the Left Mouse Button to maintain the same scale as before. The center of the next screen will be the absolute coordinates entered above.

2-8-4 VIEW MENU Zoom In

ZOOM IN will enlarge the present screen, by a factor entered from the keyboard. The default is two times. To change the factor, enter the desired scale when the box is displayed on the graphics screen. This function will always use the current center to zoom in.

2-8-5 VIEW MENU Zoom Out

ZOOM OUT will reduce the size of the present screen by a factor entered from the keyboard. The default is two times. To change the factor, enter the desired scale when the box is displayed on the graphics screen.

2-8-6 VIEW MENU Autoscale

AUTO-SCALE, automatically scales the drawing to fit on the viewing screen. It is used on an initial drawing screen when the objects appear very small, or after an INDICATE CORNERS has been used to bring the screen back to full size. Sometimes the program will appear very small after an AUTOSCALE. This does not indicate a problem in the system, but is caused by absolute zero or extraneous points set away from the part being programmed. This will often occur with DXF transfers.

2-8-7 VIEW MENU Repeat

Redraws the existing drawing screen. This is used to remove unwanted graphics off the screen such as distance markings when verify is used, control points, or pick points left on the screen from indicating entities.

2-8-8 VIEW MENU Previous View

This function draws the view of the screen previous to the present screen. It is a great time saver when switching from a blow up view to the full size view. Only the screen immediately preceding the present screen can be viewed. If PREVIOUS VIEW is entered again, the first screen will once again be displayed.

2-8-9

VIEW MENU

Change Display

CHANGE DISPLAY pulls up a “Plot Parameters” selection box which allows changes to the graphics display as well as the ability to change the output to the plotter/printer. Select CHANGE DISPLAY from the VIEW MENU. To Activate or Deactivate a function, place the mouse cursor over the appropriate box and click the LEFT mouse button. This will place a tick in the box identifying its’ selection. Press the ENTER button for the changes to take affect.

Plot parameters

<p>General</p> <p><input type="checkbox"/> Auto Dimension</p> <p><input type="checkbox"/> Axes</p> <p><input type="checkbox"/> Direction</p> <p><input type="checkbox"/> DXF Layers</p> <p><input checked="" type="checkbox"/> Toolbar</p>	<p>NC Simulation</p> <p><input checked="" type="checkbox"/> Rapid</p> <p><input type="checkbox"/> Auxiliary Functions</p> <p><input type="checkbox"/> Filenames</p> <p><input type="checkbox"/> Sequence Numbers</p> <p><input type="checkbox"/> Pierce_Stop</p>
<p>Screen Label</p> <p><input checked="" type="checkbox"/> Path</p> <p><input checked="" type="checkbox"/> Screen Label</p> <p><input checked="" type="checkbox"/> Controller</p> <p><input type="checkbox"/> Nest Data</p>	<p>Plotter Label</p> <p><input type="radio"/> None</p> <p><input type="radio"/> Standard</p> <p><input checked="" type="radio"/> Template</p> <p>C:\Program Files\FastCAM\template.cam</p>
<p>FastNEST</p> <p><input type="checkbox"/> Progress Bars</p>	<p>Plotter Pen Width 1</p>

2-8-9-1 DISPLAY MENU Auto Dimension

The AUTO DIMENSIONING function is used in FastCAM and FastPLOT. This function is used to automatically display Cut Length Dimensions of your part drawing. If you are displaying a large part on your screen or plotter, you may only see a few dimensions displayed. If you Zoom in or use the Indicate Corners functions on more detailed parts of your drawing, extra dimensions will be displayed. This display is dependent on the current drawing Scale as to how detailed your dimensioning is.

2-8-9-2 DISPLAY MENU Axes

When activated, X, Y axes will be displayed on the screen from the absolute coordinates of 0,0.

2-8-9-3 DISPLAY MENU Direction

The DIRECTION display is only activated after a cutter path has been assigned, cut direction arrows will be placed to the side of the chosen kerf. This function is helpful in determining if your cut direction is correct. Direction arrows are automatically displayed when you leave FastCAM to verify an NC code in FastPLOT.

2-8-9-4 DISPLAY MENU DXF Layers

The DXF Layers option allows all currently allocated layers to be shown. These layers can either be added in the CAD system that produces the DXF file, or directly in FastCAM itself. *(Refer to CAD Layers later in this Chapter for more information)*

2-8-9-5 DISPLAY MENU Toolbar

This option allows the user to turn the Toolbar ON & OFF.

2-8-9-6 DISPLAY MENU Rapid

The RAPID display is only active in FastPLOT. When activated all RAPID traverses will be displayed on the graphics screen or plotter, depending on the output device specified.

2-8-9-7 DISPLAY MENU Filenames

FILENAMES is used in FastCAM and FastPLOT. When activated filenames are displayed on the graphics output device.

2-8-9-8 DISPLAY MENU Sequence Numbers

SEQUENCE NUMBERS is only used in FastPLOT Nesting. When activated the sequence numbers, or the order of parts to be cut in a nest will be displayed on the graphics output device.

2-8-9-9 DISPLAY MENU Pierce & Stop

PIERCE & STOP is only used in FastPLOT. When activated the pierce and stop of the part will be displayed as a yellow cross on the screen.

2-8-9-10 DISPLAY MENU Screen Label

LABELS are used in both FastCAM and FastPLOT. There are four individual label types possible.

- Screen Label: which displays the label in the screen,
- Path: which displays the directory path in the FastCAM TitleBar,
- Controller: which displays the name of the Controller in use (also in the TitleBar), and

- Nest Data: which displays Nest Data, relating to the current Nest.

2-8-9-11**DISPLAY MENU****Plotter Label**

As well as having a Label on screen, you can also print it out with the Drawing or Nest. The Plotter Label is either: None, Standard, or use of a Template.

To use the Template option, simply:

- select the Template option,
- Click in the Box below the Template Option,
- Select your Template.CAM file (usually located in your FastCAM directory), and click SAVE.

Once loaded the Template will remain in memory and will be used to print unless changed.

The Template.CAM file can be modified to suit each customer's requirements.

The template consists of two pathed areas with key-words representing the special textual information that is to be printed. The area bounded by path #1 is where the drawing or nest will appear. The area bounded by path #2 is where a parts list will appear when a nest is printed in FastNEST.

Simply Open the file in FastCAM and alter to your specifications. For a list of the Key-Words, see the Table located in Appendix G. For an example of the default Template see Appendix F, and for an example of a printed Nest with the Template see Appendix F.

2-8-9-12**DISPLAY MENU****Plotter Pen Width**

You can use the Pen Width option to generate Thicker lines on all Plots. Especially useful for users who have optical machines, who need to generate a plot for Optical following machines. For laser printing a pen width of 3 to 5 is ideal.

2-8-10 DISPLAY MENU Window

WINDOW allows the programmer to enlarge an area of the screen much in the same manner as INDICATE CORNERS. The important difference is that the enlarged area can be saved and referred to at a latter time. This function is advantageous when working on a part that has many detail areas that need to be referred back to frequently. The following describes the sub-menus of this function.

2-8-10-1 WINDOW MENU Store Current

The STORE CURRENT function stores the current status of the viewing window.

2-8-10-2 WINDOW MENU Recall

The RECALL function restores a previously stored screen by entering the desired window number under RECALL.

2-8-10-3 WINDOW MENU Show All

The SHOW ALL function shows all defined windows present in the viewing screen.

2-8-10-4 WINDOW MENU Define

The DEFINE function is used under the WINDOW MENU to indicate the area of the screen to enlarge. When this function is called up it allows a window to be drawn in the same way as an area to be enlarged is defined in INDICATE CORNERS. (See INDICATE CORNERS for description). Use the mouse to move the window to encompass the detail area and it will be saved for future recall.

2-8-10-5

WINDOW MENU

Clear All

The Clear All function will erase all windows that have been previously set, allowing the user to set new View Windows for further work.

2-8-11

VIEW MENU

Scale only

The SCALE ONLY function is used to change the scale of the screen or plotter. Call up the function, the current scale is given. If a new scale is desired, enter the scale and press the ENTER button. When you wish to plot a part at 1 to 1 Scale, use this option to generate your plot instead of REPEAT.

2-9 UTILITY MENU



The UTILITY menu contains special functions not generally related to normal drawing functions. The menu items on this menu enable the user to Modify differing aspects of the FastCAM program or your current drawing.

2-9-1 UTILITY MENU Change Units

The Change Units function will change the mode of FastCAM from Inch to Metric. FastCAM defaults to inch out-put when the system is turned on (*Depending on your individual setup*). Call up UTILITY from the MENU. Place the highlight bar on Change Units. Press the LEFT mouse button. The system is now in METRIC. To change back to INCH mode, simply repeat the above process.



This will only change the drawing units, if a program is generated, the units will be the same as when you started FastCAM.

2-9-2 UTILITY MENU Break

The BREAK function is used to break an entity at a fixed point. The actual geometry will not be broken as with the INSERT GAP function. This function is commonly used to insert a control point which will be used as a start point when programming the cutting path. An example of this is to break a line at line midpoint. This will cause your single entity or line to be broken into two entities or lines.

2-9-3 UTILITY MENU List Control

This function will list the operations section of the CONTROL file. This file may be viewed to ensure correct NC Cutting codes are set for generation of your NC code.

2-9-4 UTILITY MENU List Setup

Lists the computer and software configurations held in the SETUP.DAT file, relating to FastCAM/FastPLOT.

2-9-5 UTILITY MENU Bevel Calculator

Used to calculate bevel offsets. This is only active if the Variable Bevel Angle package exists in your software. For further information, refer to the Beveling documentation or your FastCAM Agent.

2-9-6 UTILITY MENU List Entities

LIST gives a numeric specification for all constructions on the screen. You can select the type of listing you desire by selecting from ALL, WINDOW or LAST. All will display a complete list of your entities to the screen or printer depending upon your selection.

Window will list only those entities contained fully within the window you define. Last will display the last entity added to your drawing screen.

The numbers under the pound sign on the left side of the screen correspond to entities as they were drawn on the screen and the absolute co-ordinates of each entity are listed.

LAYER TYPE gives the type of entity, line, arc, or circle, as well as the direction of arc and circles.

START, END, CENTER: Under each heading the X, Y coordinates of each entity are given. The values are listed in absolute coordinates with the X first and Y second.

2-9-7

UTILITY MENU

Edit Entity

The EDIT ENTITY function is one of the most **useful** functions in FastCAM. EDIT ENTITY enables you to modify any FINITE line or Arcs after they have been created. For example you may wish to change the length of a line or increase the diameter of an arc if you have made a mistake. Also the EDIT ENTITY function is an extremely useful TRIMMING tool. When using Edit Entity with a line, you are presented with 3 options:



- **POINT:** The Point option allows you to move the end point of the selected line. The end point closest to the selected point will be moved.
- **LENGTH:** The Length option will allow the length of the selected line to be altered. Once the line is selected, a prompt will appear highlighting the current length of the line. Alter this value and the line length will change.

- **ENTITY:** The Entity option allows a line to be extended to reach another line, or it's projected position. This is very useful when replacing a filleted, or notched corner

2-9-8**UTILITY MENU****Stretch**

Stretch is designed to resize a part along either the X or Y axis of the part. A window is used to enclose one section of the part, and you are then asked to select the entity you wish to stretch. Once selected, a box will be displayed showing the existing length of the entity, which you can then change to a larger or smaller value.



NOTE: When using the window to select entities, you must include at least ONE whole entity.

2-9-9**UTILITY MENU****Pts to holes**

Pts to Holes allow the user to quickly change a series of REAL points into Circles, of a specified Diameter. You simply place a window around the holes and then specify the Hole diameter.

2-10 TEXT MARKING

An optional feature in FastCAM is the TextMARKER function. This function if enabled will automatically convert text added to the part with the TEXT MARKING function to a stick font suitable for scribing. This means that with a standard controller (eg Burny, Lynx, Anca) and a suitable marking tool (Arc Writer or Air Scribe) FastCAM software can mark parts with text without the need for special options in the controller or ink jet marking heads.

The Text Command has several purposes.

1. **Screen and Printer Text:** Places the desired text on the screen at the required place usually for the identification of individual parts. The size and angle of the text can be varied to suit your requirements. Adjusting the system setup for Windows will even change the text font although wild variations may cause unsatisfactory results and is not recommended. This text will be replicated onto the printer. (The Plot)

2. **Machine Text:** Places the screen text directly onto the plate usually onto the cut parts but is often used to identify plates or remnants. Machine Text Marking is a machine function and can only be used where it is installed onto the equipment. The text font on the machine is usually fixed except for height and angle. There may be restrictions on the length of text and maximum and minimum sizes. Consult you machine manual for details. FastCAM Supports Machine Text Marking where installed onto the equipment. There are two main types.
 - **Stroke Fonts:** These are often called stick fonts due to their shape. Stroke fonts are installed in the controller ROM and are called up in the NC code which includes values for height and angle.. There are two main processes that support stroke fonts, punch marking and plasma marking. Powder marking is also available but is not that suitable to stick fonts. When using powder marking the persistence of the

marker (The time taken to turn of and on) means that the font needs to be quite large. Plasma markers and punch markers are very similar to powder marking in their operation but do not suffer the persistence problems. Powder marking and plasma marking are both thermal processes.

- **Ink Jet Markers:** These devices are very similar to Ink or Bubble Jet printers and have their fonts directly installed in the printer device. Ink Jet Markers can produce a variety of fonts but are still limited in their output. These devices have been used for many years in the packaging industry where you can see their output in any supermarket. The plate surface is critical to the output and care must be take with ink selection when cutting with Plasma.
- **FastCAM TextMARKER** This is an option for FastCAM. It allows for the automatic conversion of Text Marking information into NC code that is in a stick font format. This NC stick font is compatible with most plate marking devices. This function allows an ordinary plasma marker / air scribe (punch marker) / Laser marker to be used as an effective marking device.

2-10-1 TEXT MARKING Add

The Add Text function will first allow you to select the point on the drawing where text placement is required (*all point types can be used*). After text has been entered, you can then select the desired angle of text and height of text. Add text is a continuous function and after one line of text has been placed you are then prompted for another. To exit from here simply click the right mouse button.

2-10-2 TEXT MARKING Edit

The text editing function allows you to change the text composition, text angle, and text height. Simply select the text by choosing the corresponding insertion point (control point), and then make the changes.

2-10-3**TEXT MARKING****Move**

The text move function allows you to alter the existing position of any text. Simply select the insertion point of the text and then select the new insertion point.

2-11 CAD LAYERS

CAD LAYERS is a menu in FastCAM; relating to the use of CAD Layers for machine processes. It is not a system for the general editing of CAD layers which can be quite complex and often irrelevant to the NC process. Rather CAD LAYERS is a way of creating general process information on a drawing ready for post processing with **FastPATH**.

For layers to be recognized and used in CAD LAYERS, it is essential that they already exist in the CONTROL.DAT file (or similar) in FastCAM versions prior to FastCAM 5.9. FastCAM 5.9 can automatically map DXF layers to machine functions, see below for more information. This file will have special lines as shown in the following file excerpt for the ATLAS machine which has plasma, oxy, line marking, drilling, tapping and various standard bevel angles.

```
/DXF/
( LAYER, PROCESS BY SEQUENCE ABOVE, IMPLIED DATA AS
APPROPRIATE
CUT,1
CUT45,1, 45
CUT-45,1,-45
CUT40,1, 40
CUT-40,1,-40
CUT35,1, 35
CUT-35,1,-35
CUT30,1, 30
CUT-30,1,-30
CUTTRANSITION,1
OXY,2
POWDMARK,3
POWDCUT,3
PLASMAMARK,3
SPOTDRILL,4
DRILL,5
```

* extra number indicates second pass for TAP

DT-UNC,5,1

DT-MET-F,5,1

DT-NPT,5,1

DT-BSPT,5,1

DT-UNF,5,1

DT-MET-M,5,1

DT-MET-C,5,1

For more information on The CAD Layer function and how it could be incorporated for your machine, consult your software reseller.

2-11-1

CAD LAYERS

Show

This will toggle the colors on the screen so that each CAD layer is shown in the color of the matching process. The colors are defined in the CONTROL.DAT file (or similar). Selecting show will display a dialog box listing layers defined in the drawing and layers defined for machine processes. If the layer name and the machine process name matches this will be indicated on the left hand side of the form. If a match does not occur this will be flagged. FastCAM 5.9 and later has the ability to match specific DXF layer names to processes.

Mapping DXF Layers (FastCAM 5.9 and above)

DXF Layer name	Entitie	Match	Operation
OUTSIDEPROFILE	4	No	
INSIDEPROFILE	1	No	

TOTAL DXF LAYERS 2

DXF Layer name	Op#	Operation	Color
PLASMA1	1	PLASMA1	10
PLASMA2	2	PLASMA2	15
SCUT	1	PLASMA1	10

TOTAL PREDEFINED 3

Exit

The above diagram shows a part that has two layers. One called “OUTSIDEPROFILE” the other “INSIDEPROFILE” they are not matched to any machine processes. There are two operations or machine processes available Op#1 – Plasma1 and Op#2 – Plasma 2. When the DXF layer name is not matched the Match column will contain the word “No.” Clicking on the operation field in the left hand frame, a dropdown list box will appear, with the operation numbers able to be selected from this list. Select the required

operation to be mapped to the DXF layer. Once set, press the exit button. A question will be asked if the changes are to be saved. If they are not saved nothing will be changed. Once the layer names have been matched and saved all subsequent DXF files with that layer name will be automatically mapped.

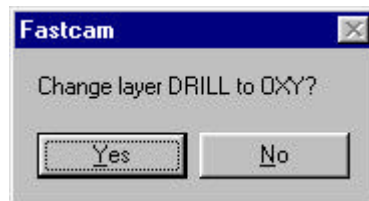
2-11-2**CAD LAYERS****Add Layers**

Allows the selection of a layer name from a pop up list in the sequence given in the CONTROL file, one entity at a time. After selection from the list, just simply click on each corresponding item.

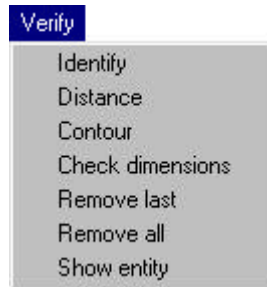
2-11-3**CAD LAYERS****Remove Layers**

Similar to Add Layers, it allows removal of layers of one type, but will remove all layers of a given type.

If you wish to change a layer name, but not to start from scratch with Remove Layers, simply choose Add Layers and select an existing entity. The system will prompt



You can then respond accordingly.

2-12 VERIFY**2-12-1 VERIFY MENU Identify**

This function lists the geometric specifications for the entity indicated. Relevant data such as length, start and end points and dimensions are displayed. Once you move your mouse after a selection, you can select another. Just click the right mouse button to end the identify process.

2-12-2 VERIFY MENU Distance

The DISTANCE function will display the distance between two (2) selected control points on the screen. If diagonal points are selected the, X and Y distance between those points are also displayed.

2-12-3 VERIFY MENU Contour

The CONTOUR function is used to check the contour length of any "connected" entities. All relevant data such as Length and Area is displayed on the screen. It will also state whether the contour is closed or open.

2-12-4 VERIFY MENU Check Dimensions

The CHECK DIMENSIONS function is used to display selected part dimensions on the graphic screen. It works between two Control Points, and also asks for a reference position for placing the dimension. Any dimensions displayed using this function can be Plotted along with your drawing.

2-12-5 VERIFY MENU Remove last

The REMOVE LAST function will remove the last check dimension added to your drawing.

2-12-6 VERIFY MENU Remove all

The REMOVE ALL function will remove all of the check dimensions which have been added to the drawing.

2-12-7 VERIFY MENU Show Entity

The SHOW ENTITY function will highlight the entity selected according to the number specified, in a bright magenta color.

2-13 TRIM

The TRIM function allows Arcs, Lines, or Circles to be shortened to the proper length. Trimming can ONLY be done between REAL POINTS (yellow +), CONTROL POINTS (red+) and any intersecting (crossing) entities.

2-13-1 Trim

Select the entity you wish to trim and then select the two trim points, When trimming full circles you will also be asked to select the portion of the circle to erase. You cannot trim off an entire line, when trimming you must leave a portion of the entity on the screen; use ERASE to eliminate an entire entity.

2-13-2 Smart Trim

Smart Trim automatically trims between two existing control points. The selected entity will be removed between two determined control points. If the entity is an infinite line (angle only) then the selected section of the entity will remain and the rest is deleted.

2-14 PROGRAM PATH

A definition of this menu is explained following the next menu item under Programming Cutting Paths

2-15 CONTROL POINTS**2-15-1 CONTROL POINTS Control Points**

The CONTROL POINTS function displays all control points on the screen. A control point is represented by a RED (+). All entities have Control Points (with the exception of infinite lines). Finite Lines have one at each endpoint, arcs have as little as three, (*or as many as five if the quadrant points are included*), one at the start, center, and end of the arc. Circles have five control points, one in the center plus one at each quadrant point. The View-Repeat item will clear the screen of all control points. See Appendix B for alternative method of displaying Control Points.

2-15-2 CONTROL POINTS License

The License option displays your user information, including the Button number of your FastLOC dongle.

2-15-3 CONTROL POINTS About

The About option displays the Fastcam.exe file information including, the date of the program, the version number, and currently loaded options.

2-15-4 CONTROL POINTS NC Change

NC Change allows the user to change between different Profile Machine Configurations, eg:

Machine A is Laser,

Machine B is Plasma

This enables you to generate files for multiple machine types without having to restart the program.

NOTE: Multiple Machine Configuration is a FastCAM option.

2-15-5 CONTROL POINTS NC Display

NC Display, display the current Machine configuration on the screen. Showing Operation On & Off Codes, and other machine specific information.

2-15-6 CONTROL POINTS Reset Files

Reset Files - resets the temporary files created by FastCAM. This can resolve problems with FastCAM becoming lost or not opening correctly due to a strange error.

2-16 LANGUAGE

This menu allows the user to change the FastCAM system into a different language. The language option is only installed on International Installations, for other language support, please contact your local reseller

Programming Cutting Paths

2-17 NC PROGRAM PATH

This menu item will take you to a different section of your FastCAM program. After selecting PROGRAM PATH, you may be prompted to "Enter Part Description?". This will only occur if you have not Saved the file and filled out the Part Data Screen. This allows for the same Part Data information to be entered as discussed under SAVE earlier in this chapter.



If you require the Kerf values and the Feedrate values automatically inserted into the part program NC code, contact your software reseller for the appropriate instructions to do this. Otherwise, all data input in the DATA Screen will feed through to the NC program in a format that will be ignored by the NC control. This option is not available to all NC controllers. Please check if this NC code modification is applicable to your equipment.

NC Path Menu

The NC PATH MENU contains functions relevant to setting the cutter path of the machine. This is done after all of the part construction is completed. The program path is started at the beginning of any selected Line or Arc. In the case of a point process such as Drilling or Pop marking, the program path is assigned to any Real Point, or circle. If it is desired to start the cutter path at some other point the line can be broken with the BREAK function located in UTILITY, and a point inserted. It is usually sufficient to use one of the existing control points for the beginning of a cutter path.

When setting the cutter path it is important to understand the concept of forcing the path around the part. To force the path right or clockwise (CW), pick the entity you wish to cut first. This may be any Line or Arc. **(NOTE: When selecting a starting position, do not select a control point, always clearly indicate the entity by itself)** A directional arrow is then displayed. Movement of your Mouse will point the arrow in the desired direction. Move your mouse so the arrow points clockwise and click the left mouse button. To force the path to the left or counter clockwise (CCW), follow the same sequence of steps but point the arrow counter-clockwise and click the left mouse button.

Pathing full (unbroken) circles will not give you a directional arrow, it will simply ask whether the cut is an Inside or Outside cut.

2-17-1

NC PATH MENU

Next Path

This function is used to manually assign cutting path information such as kerf and type of operation to be set (cutting, marking, Punch, etc). Entries and exits are added at this time. This is the first selection for pathing any item.

After choosing Next Path, you will be asked to choose an operation, and then a Kerf Type. You must then select the first contour to be pathed. Typically you would select all inside contours first, and then the outside contour of the part last. Once all paths on a part have been allocated, you can then proceed to OUPUT NC CODE, to produce the NC program.

2-17-2

OPERATION MENU



PLASMA
OXY
MARKER
DRILL
RAPID

This is only an example of an Operation menu, and its' contents will vary according to each individual installation.

2-17-2-1 OPERATION MENU Process

The OPERATION MENU is used to select the type of machine process to be used on the part. All of the cutting, marking, punching and scribe punching processes available on your cutting machine should appear in this menu. Select the type of process you wish to use after selecting Next Path. Several processes may be used on a single part. Hence this menu may be selected more than once.

2-17-2-2 NEXT PATH KERF MENU:

The KERF MENU sets the kerf of the part by highlighting LEFT, NONE, RIGHT.



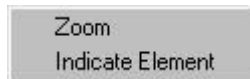


Kerf is the amount of material destroyed in the cutting process. For parts to be cut to the proper size the correct kerf setting must be used. Kerf is explained in detail in the Profiling Machine Operation Manual.



When programming cutting paths, FastCAM will automatically check that all geometry closes within the tolerances of the cutting control. Sometimes, when trimming entities and when using DXF geometry, a gap between one entity and another will be present. This may not have been seen by the programmer, particularly when the gap is very small. If such a problem exists the cutting path will stop at that gap.

A menu with two options will appear:



The ZOOM option will allow accesses to the PLOT MENU so that a zoom onto the error can be done.

If you are satisfied that the error will not cause problems, the Indicate Element option can be used to continue the cutting path. This is done in a similar way as initiating a path, the graphics cursor simply points to the next entity to be cut. The difference here is that you do not get the direction arrow as before. In this case you indicate the end of the element you wish the path to proceed to from the stoppage point. Many zooms in and out can be done without upsetting the status of the cut path. The option to Indicate Element will always remain active.



Remember, ONLY files produced by OUTPUT NC CODE can be cut.

2-17-3 NC PATH MENU Delete all Paths

The DELETE ALL PATHS function removes all process paths assigned to your part. Any lead-ins or lead-outs will remain.

2-17-4 NC PATH MENU Delete last Path

The DELETE LAST PATH function removes the last process path assigned to your part. Any lead-ins or lead-outs will remain.

2-17-5 NC PATH MENU Redefine Path

The REDEFINE PATH function allows any previously set path to be redefined to correct lead in or lead out location, or to remove the path altogether.

2-17-6 NC PATH MENU Output NC code

The OUTPUT NC CODE function is the final operation performed on a part in FastCAM. When an NC Program is generated, the geometry of the part is converted into NC language and a TEXT file is created. This is the file which will either be nested or verified and then sent to the controller to be cut. Some controllers, like the COMPUPATH require a special extension of .TXT to be placed on the file name. When generating the NC program, the file will be named consistent to the name currently in the Part Data screen. If the name in the part data needs to be changed, select

Option 7 in the NC PATH MENU, PART DESCRIPTION, and make the name change.

Steps to Produce an NC code program:

1. Program all the paths required for the parts(s)
2. Select Output NC Code from the Program Path menu.
3. Key in the file name for the program.
4. Click Save.
5. Select Rapid at start if an initial Zero position is required.



When sending individual part programs to the cutting machine, it is usual practice to set an initial zero position for the program. This allows the program to be positioned easily on the workpiece, away from the base corner for example. If this zero is not set then the zero position for the part will be its initial pierce point. The initial zero position can be any of the FastCAM point menu options. If the parts are to be nested then no "Rapid at the start" is necessary.

6. Leave FastCAM & verify part. A file BACKUP.CAM will be created when chaining from FastCAM to FastPLOT.

2-17-7

NC PATH MENU

Status

The STATUS function is used to check the status of the current programmed path. A useful check if multiple paths are being programmed using defined windows.

2-17-8

NC PATH MENU

Part Description

The PART DESCRIPTION function is used to review the part data currently recorded. This can also be used to change a file name prior to output NC code. If there is no data recorded against the part, it can be input at this time.



If changes are made to the part data at this point, a SAVE of that part must be done in order to re save the changed data. A SAVE can be done at any time, regardless of the presence of cutting paths.

2-17-9 NC PATH MENU View path data

The VIEW PATH DATA function allows a complete check of all paths programmed on a part.

2-17-10 NC PATH MENU View

The VIEW function is a quick entry into the view menu to allow access to zooming and window recalls.

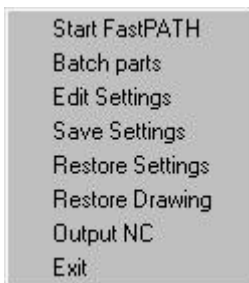
2-17-11 NC PATH MENU FastPLOT

The FASTPLOT function is direct access to the FastPLOT verification program from within FastCAM. This option eliminates the need to EXIT from the program and restart FastPLOT. If any drawings are unsaved you will be prompted to SAVE the drawing first.

2-17-12

NC PATH MENU

FastPATH



The FASTPATH MENU contains all of the functions associated with the Automatic Pathing Module. Once a path is setup and stored it can be recalled and utilized at any later stage. A full explanation of FastPATH is covered in Chapter 3. **NOTE:** **FastPATH is a FastCAM® optional feature.**

2-17-13

NC PATH MENU

Restore Drawing

The RESTORE DRAWING function is used to restore to the screen the BACKUP copy of the file you are pathing. If for any reason you are not happy with the paths you have assigned, you can use this function to quickly retrieve a copy of the file before it was pathed.



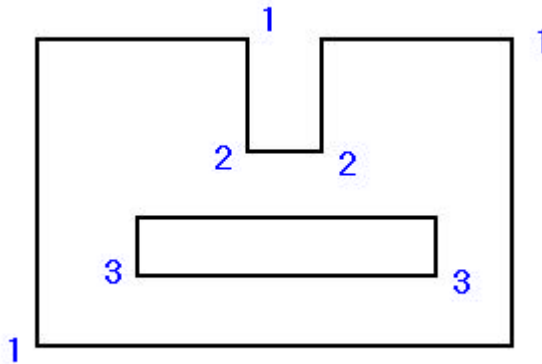
Restore Drawing is used to restore a pathed and outputted drawing in FastCAM. This can be done even after FastCAM has exited and the NC code has been displayed in FastPLOT. This function is useful when outputted paths from FastCAM have been found to be incorrect in FastPLOT.

2-17-14 NC PATH MENU**Pathed DXF**

A Pathed DXF is a special DXF file in which all entities are correctly sequenced for cutting. All internal entities would be drawn first and all linear entities are joined in correct sequence. This type of DXF file is mainly used on basic NC controllers that use a DXF file for machine control.

2-17-15 NC PATH MENU**Round All Corners**

Inside, Outside or Both cutting paths can have their corners rounded. Once the radius has been set the side that will be processed is selected. All corners are detected and rounded to the required radius. Different rounding scenarios can be flagged. The illustration below shows a typical part that requires corner rounding.



Corners marked with the number 1 are outside corners on an outside contour. Corners marked 3 are outside corners on an inside contour. Corners marked 2 are inside corners on an outside contour. Both inside, outside or either inside or outside contours can have their corners rounded. Prompts will appear and should be answered accordingly.

Chapter 3 – FastPATH™

FastPATH is a smart pathing tool for *.CAM, *.DXF or other geometry files read by FastCAM. The program is intuitive enough to path deeply nested parts, correctly determining inside and outside profiles.

This option reduces the time a programmer would have to spend pathing parts when converting individual parts to NC code.

3-1 FastPATH Concepts

The FastPATH module is included in all FastCAM professional packages. It is available as an option for FastCAM NC and FastCAM Standard.

FastPATH greatly reduces programming time in the generation NC code from individual CAD files. The program automatically allocates cutting direction, entries and exits, sequencing, cutting processes, and other NC processing options.

When generating NC code, whether for the cutting of a single part or several parts for nesting, a lot of time must be spent manually applying a cutting path to these parts. This time may be minimal if the part is simple, but can take hours for more complex parts.

To set-up FastPATH for first use, a process (oxy, plasma, laser) and an entry (lead-in) must be set. Depending on the parts to be cut, the complexity of the setup for FastPATH will change. For a nest to be pathed for a laser many settings in FastPATH may need to be set.

FastPATH can complete the pathing of partially pathed CAM files. This feature is useful when non standard or specialized pathing is required

3-2**FastPATH****One Process Quick Set-up**

There are only a few settings required to initialize FastPATH to place a cutting path around existing individual CAM files so NC code can be produced.

When FastPATH starts, the first option to be set is the required process. If a fastpath.pth file exists in your current working directory, the settings will be loaded from it.

In this example the available processes are Plasma, Oxy, and Punch. The plasma process was selected by double clicking on the PLASMA process in the available process column. Alternatively this process could have been selected by clicking on the process (PLASMA) and pressing the arrow key to place it in the Selected Processes column. Once the process is set the "kerf" and "side" values needs to be verified. They are set by double clicking the value in the appropriate column. This can be seen in Figure 3a. The "kerf" and "side" settings will change each time they are clicked. In this example kerf should be left and side should be both.

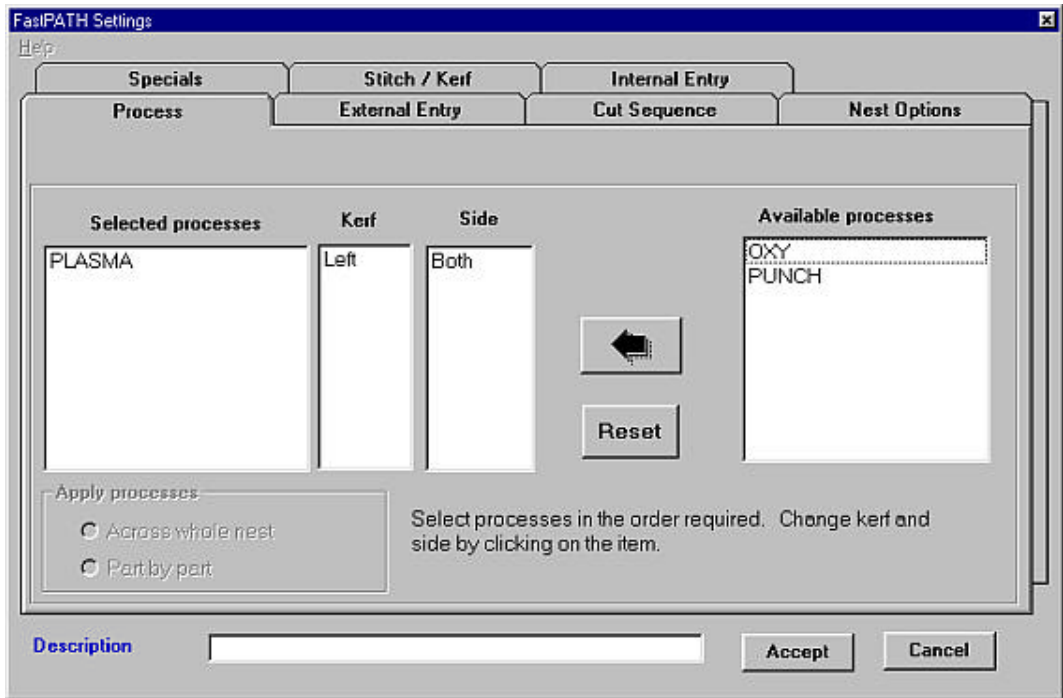


Figure 3a

The entry now needs to be configured. Select the "External Entry" tab to set the external entries. The value in the position drop down dialog box sets the entry position. In this example it is set to 'Top Left' the lead-in would then be as close as possible to the top left position. This can be seen in Figure 3b.

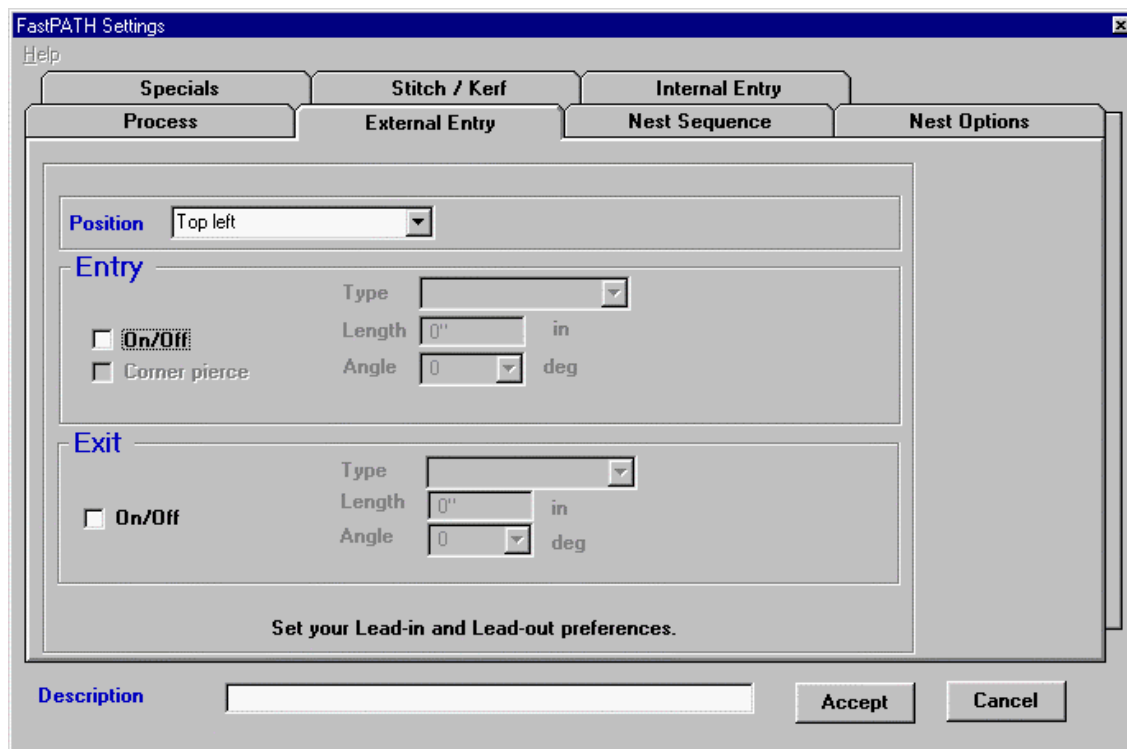


Figure 3b

The entry needs to be turned on. To do this the entry "On/Off" check box needs to be selected. When the entry is turned on the Type, length and angle of the entry can then be set. Figure 3c shows this.

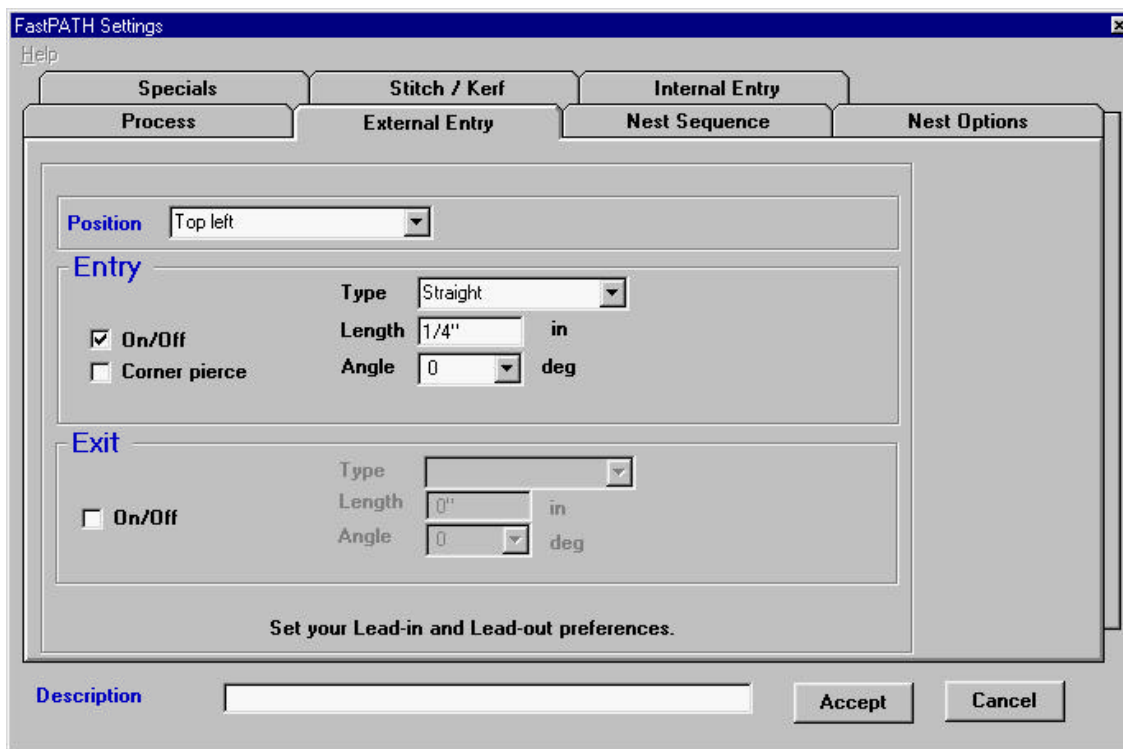


Figure 3c

The default entry style bisects and breaks the entity closest to the left of the entry position when left kerf is set. If the user prefers a corner pierce, simply check the corner pierce box for the entry. If the entry type or entry angle is unsuitable for the part, the angle and style will be adjusted to ensure proper entry. Refer to Figure 3d.

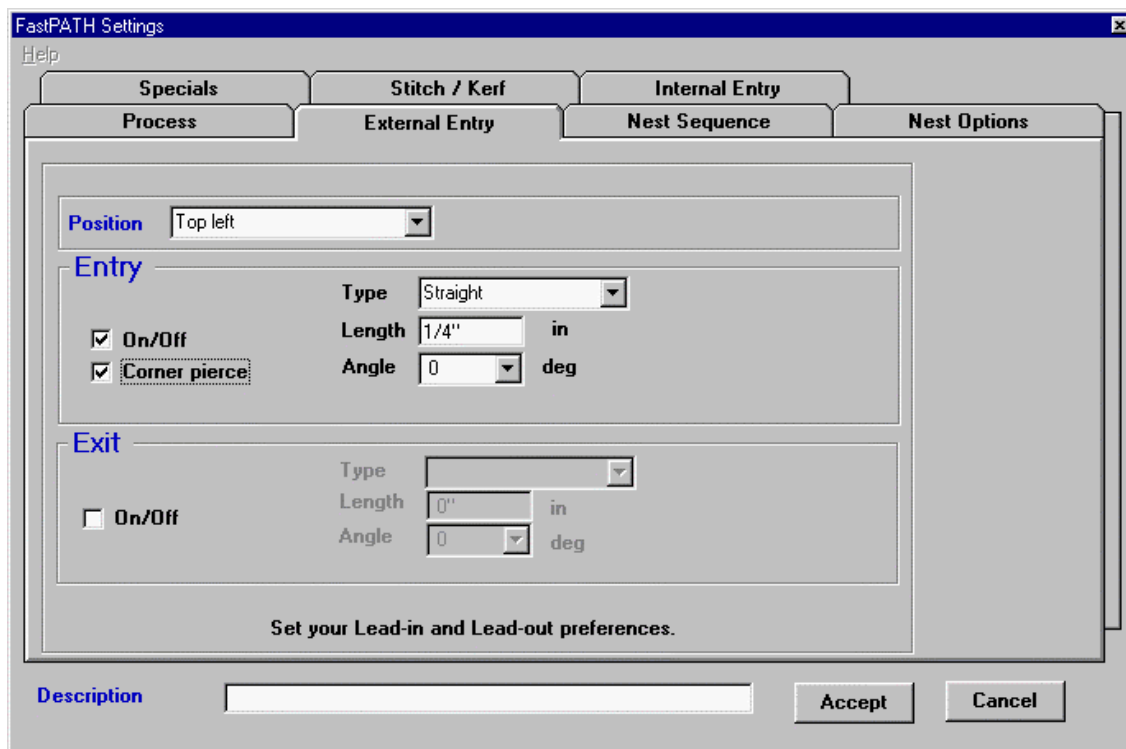


Figure 3d

If an exit is required, the on/off check box is checked. This turns on the exit parameters. These parameters configured to suit length, type, and angle.

Once the external entry is set the internal entry needs to be configured. Selecting the Internal Entry tab the user is able to quickly set the entry and exit settings to be the same as the external entries. This can be done by selecting "Internal same as external."

The center pierce value should be set to the same size as the entry lengths.

Once this is done the settings can be accepted and saved. FastPATH is now ready for use. Refer to Figure 3e for exit parameters.

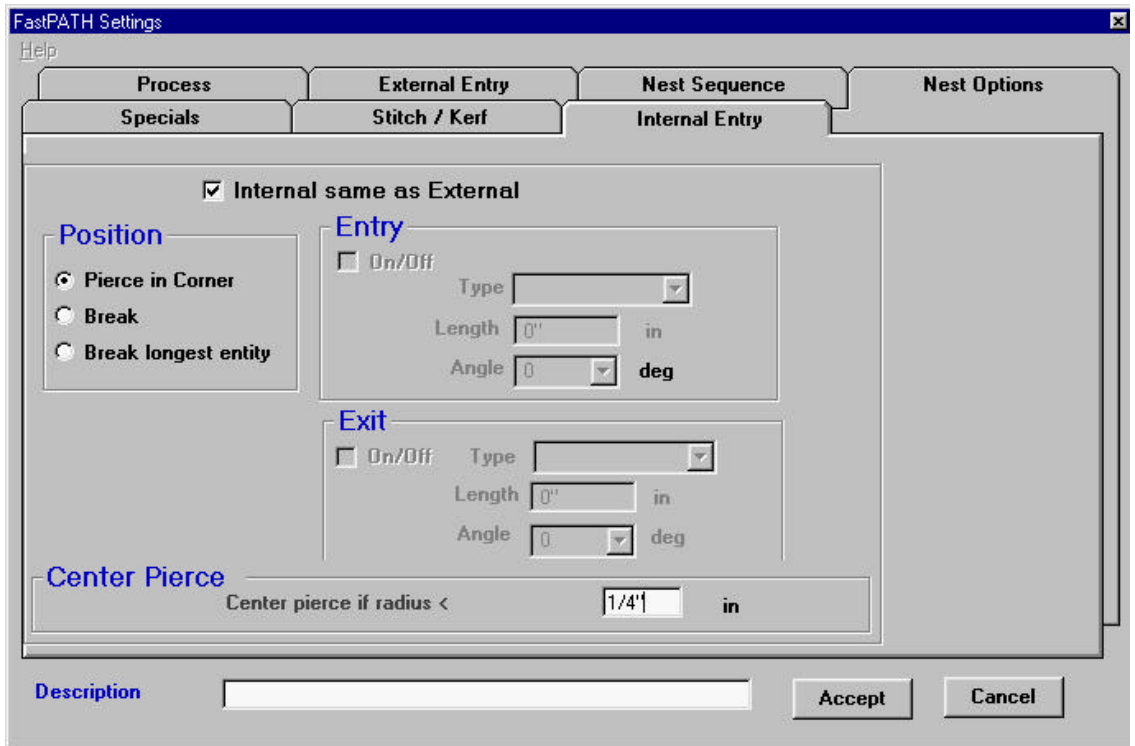


Figure 3e

The settings can now be accepted by pressing the Accept button. The FastPATH menu will then be presented and the current settings can be saved or pressing the "Start FastPATH" menu option will start FastPATH.

FastPATH Settings Reference

3-3 FastPath Option Menu

When FastPATH is first started the option menu is presented. This menu is used to load and save changes to FastPATH parameters as well as to batch process multiple parts.



3-3-1 FastPATH Start FastPATH

This function starts FastPATH, running the current settings on the loaded part.

3-3-2 FastPATH Batch Parts

Multiple CAM Files can be processed through FastPATH and a cutlist developed for FastNEST when this function is used. The Batch Parts function, when used with the explode function in FastCAM can quickly and effortlessly generate NC code from DXF files containing multiple parts.

3-3-2-1 BATCH PARTS Batch Parts

Batch Parts allows for the automatic processing of cam files, converting them to NC code. The current settings in fast path will be used to process the cam files. The steps involved in processing multiple files using the Batch Parts option are as follows.

3-3-2-2 BATCH PARTS Select Parts

The parts to be batched can be selected from the required folder using combinations of the Shift / Ctrl and mouse.

3-3-2-3 BATCHPARTS Select the Output Path

The output path needs to be selected. This is the path where the NC code files will be saved. This can be the same as the input path.

3-3-2-4 BATCHPARTS Automatic Overwrite

Existing files can be automatically overwritten or the output file's name can be changed.

3-3-2-5 BATCHPARTS FastNEST Cutlist (Optional - Cutlist)

Selecting yes to this option will produce a cutlist for FastNEST. Further questions will be asked to complete cutlist information.

3-3-2-6 BATCHPARTS Cutlist Name (Optional - Cutlist)

This field contains the file name for the cutlist. The cutlist will be saved in the current output folder selected when the "Output Path" was set.

3-3-2-7 BATCHPARTS Part Separation (Optional - Cutlist)

The value entered in this field will be the separation set for the cutlist.
This value can be changed in FastNEST if required.

3-3-2-8 BATCHPARTS Plate Size (Optional - Cutlist)

This is the required plate size for the nest. This setting can be changed later in FastNEST if required.

3-3-2-9 BATCHPARTS Verify Part

When all the parts have been converted into NC code, FastPATH gives the operator an option to verify the last part converted. If selected, FastCAM will close and the generated NC code will be displayed in FastPLOT.

3-3-3 FastPATH Edit Settings

The edit settings option loads the current FastPATH Settings, which can then be edited.

3-3-4 FastPATH Save Settings

This function saves the current FastPATH settings to a *.PTH file. The default FastPATH file is FASTPATH.PTH. FastPATH automatically retrieves the settings in this file and applies them as the default setting. Many *.PTH files can be created and saved for later loading.

3-3-5 FastPATH Restore Settings

The Restore Settings function loads a previously saved FastPATH configuration file.

3-3-6 FastPATH Restore Drawing

Restore Drawing removes all cutting paths placed on a part by FastPATH. Before FastPATH places a cutting path on a drawing, FastCAM saves the current part as a BACKUP.CAM file. This original file becomes the current file when restore drawing is selected.

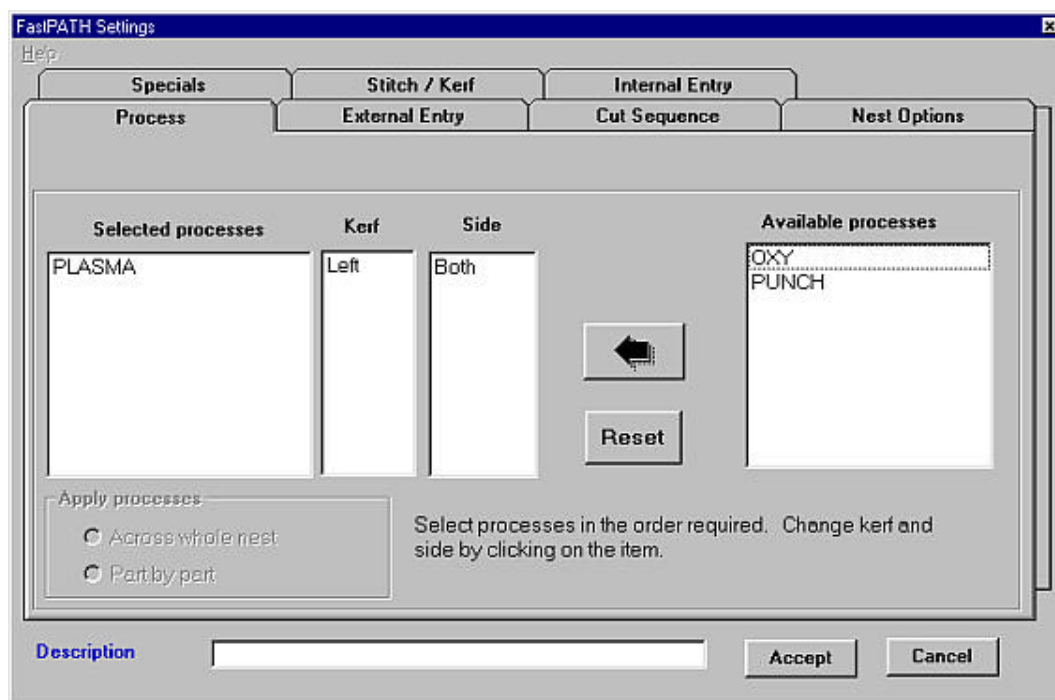
3-3-7 FastPATH Output NC

The current pathed file is converted into an NC program. If a cutting path has not been placed on a part FastPATH will process the part, placing a cutting path on it according to the current settings and will then output NC code. The file will be saved to the current directory. This can be locked to the same directory as the currently loaded / saved cam file or can be a unique output path.

3-3-8 FastPATH Exit

Selecting this option returns the user to the main FastCAM editing program.

3-4

PROCESS

3-4-1

PROCESS Selected Processes

All listed processes will be recognized and used if possible with the currently loaded profile. To select a process that is currently listed in the Available Processes column a user can either double click on the process or select it and press the large arrow to the left of the Available Processes column. If more than one process is required and it is not a specially defined process, the entity(ies) that are to be cut/marked with this process need to be on a separate layer. For example, if a part is to be cut with a PLASMA process and marked using a MARKING process, the entities defining each path must be on a different layer. Refer to 2-11 Cad Layers

for more information.

The layer for each process, excluding the final process, needs to be defined. All processes that are selected will be mapped to a defined layer, if the process and corresponding layer are configured.

The reset button is used to remove all currently Selected Processes and place them back in the Available Column, for reselection.

The Apply Processes function is not used in FastPATH when it is launched in FastCAM. This function is a FastNEST only function and is covered in the FastNEST manual.

3-4-2 PROCESS Kerf

By double clicking in this area the kerf can be set for the specified process. The kerf can be set to LEFT, RIGHT, or NONE depending on operator preferences. The cutting direction is adjusted to suit the kerf.

3-4-3 PROCESS Side

When the value in this column is double clicked, the side on which the process will be cut is selected. The available options are INSIDE, OUTSIDE, or BOTH. Using this function it is possible to automatically path a part that is cut with an OXY process on all inside contours and with PLASMA on the outside. To do this, two processes would need to be listed. The OXY process would be first and set to INSIDE; the PLASMA process would be second and set to OUTSIDE.

For standard profile cutting with the same process, the SIDE setting should be set to BOTH.

3-4-4 PROCESS Available Processes

All cutting/marking/drilling/punching processes currently defined in FastCAM’s NC profile are listed here. If the process appears in this right hand column, it is available but will not be used. To push a process over to the Selected Processes column, it can be either double clicked on or selected by left clicking and ‘pushed’ to the Selected Process column by using the large arrow to the left of the Available Processes column. The reset button is used to remove all currently Selected Processes and place them back in the Available Column.

3-4-5 PROCESS Arrow

The Arrow is used to push an available process across to the selected process column. An Available Process is selected by left clicking on it with the mouse. An available process can also be moved to the Selected Process column by double clicking on it.

3-4-6 PROCESS Reset

The Reset button is used to clear all Selected Processes and place them back in the Available Process Column.

3-4-7 PROCESS Description

The Description field is used to enter a detailed description of the FastPATH settings that have been set. This allows a user to ensure retrieved settings are as expected without the need to view each individual setting.

3-4-8 PROCESS Accept

The Accept button loads the currently set parameters into FastPATH. Once the parameters are set, the user is able to save these settings for later use.

3-4-9 PROCESS Cancel

The Cancel button cancels the current FastPATH settings, returning the user to the main FastPATH file menu.

3-5 INTERNAL ENTRY

The image shows the 'FastPATH Settings' dialog box with the 'Internal Entry' tab selected. The dialog has a title bar with 'FastPATH Settings' and a close button. Below the title bar is a 'Help' button. The main area is divided into several tabs: 'Process', 'External Entry', 'Nest Sequence', 'Nest Options', 'Specials', 'Stitch / Kerf', and 'Internal Entry'. The 'Internal Entry' tab is active, showing options for internal entry settings. At the top, there is a checkbox labeled 'Internal same as External'. Below this, there are three main sections: 'Position', 'Entry', and 'Exit'. The 'Position' section has three radio buttons: 'Pierce in Corner', 'Break' (which is selected), and 'Break longest entity'. The 'Entry' section has a checkbox 'On/Off' (unchecked), a 'Type' dropdown menu set to 'Straight', a 'Length' input field set to '1/4"', and an 'Angle' dropdown menu set to '0'. The 'Exit' section has a checkbox 'On/Off' (unchecked), a 'Type' dropdown menu set to 'Straight', a 'Length' input field set to '1/4"', and an 'Angle' dropdown menu set to '0'. At the bottom of the 'Internal Entry' section, there is a 'Center Pierce' section with a label 'Center pierce if radius <' followed by an input field set to '1/4"' and the unit 'in'. At the very bottom of the dialog, there is a 'Description' label followed by a text input field, and two buttons: 'Accept' and 'Cancel'.

FastPATH Settings

Help

Process External Entry Nest Sequence Nest Options

Specials Stitch / Kerf Internal Entry

☐ Internal same as External

Position

☐ Pierce in Corner

☒ Break

☐ Break longest entity

Entry

☐ On/Off

Type: Straight

Length: 1/4" in

Angle: 0 deg

Exit

☐ On/Off

Type: Straight

Length: 1/4" in

Angle: 0 deg

Center Pierce

Center pierce if radius < 1/4" in

Description

Accept Cancel

3-5-1 INTERNAL ENTRY Internal Same As External

This setting is enabled by default in FastPATH. When it is set, all internal paths will be processed the same way as external paths. All entry options are disabled.

3-5-2 INTERNAL ENTRY Pierce in Corner

When set, all entries will start at a corner as close as practicable to the start position specified for external entries. If a well defined corner does not exist the entry will be close to where it is required.

3-5-3 INTERNAL ENTRY Break Longest Entity

When selected the longest entity is determined from an internal contour. The entity selected is then broken at its midpoint and an entry and exit attached. This option is ideal when pathing intricate lattice work or similarly detailed components. The longest entity is usually the farthest away from the contour, thus allowing an adequate entry and exit if necessary.

3-5-4 INTERNAL ENTRY Center Pierce

The Center Pierce function allows for small holes to be processed correctly. When a lead-in length is specified that length will be used regardless of the profile required. If a hole is to be cut that is smaller than the lead-in length, the lead-in needs to be shortened. The Center Pierce value sets the smallest size hole that will be processed with the default lead-in. For example if a 1/4" (6.35mm) lead-in is used the smallest hole that can be comfortably cut, assuming a center pierce is 1/2" (12.7mm) diameter hole. This hole has a radius of 1/4" so the radius should be set to the same value as the lead-in length. Depending on results required, this value can be altered.

If the Center Pierce radius is set to zero, the setting has no effect on the currently set FastPATH settings and will be ignored.

3-5-5 INTERNAL ENTRY Type

The entry Type can be either straight, quarter circle, half circle or none. All these entry types are the same as the entry types available in FastCAM, when a part is manually pathed.

3-5-6 INTERNAL ENTRY Length

The Length of the entry is specified in this field. If a circular entry is specified in the Type field then the arc radius is specified here. Both fractional and decimal inches can be specified in this field, as well as millimeters depending on the units set in FastCAM.

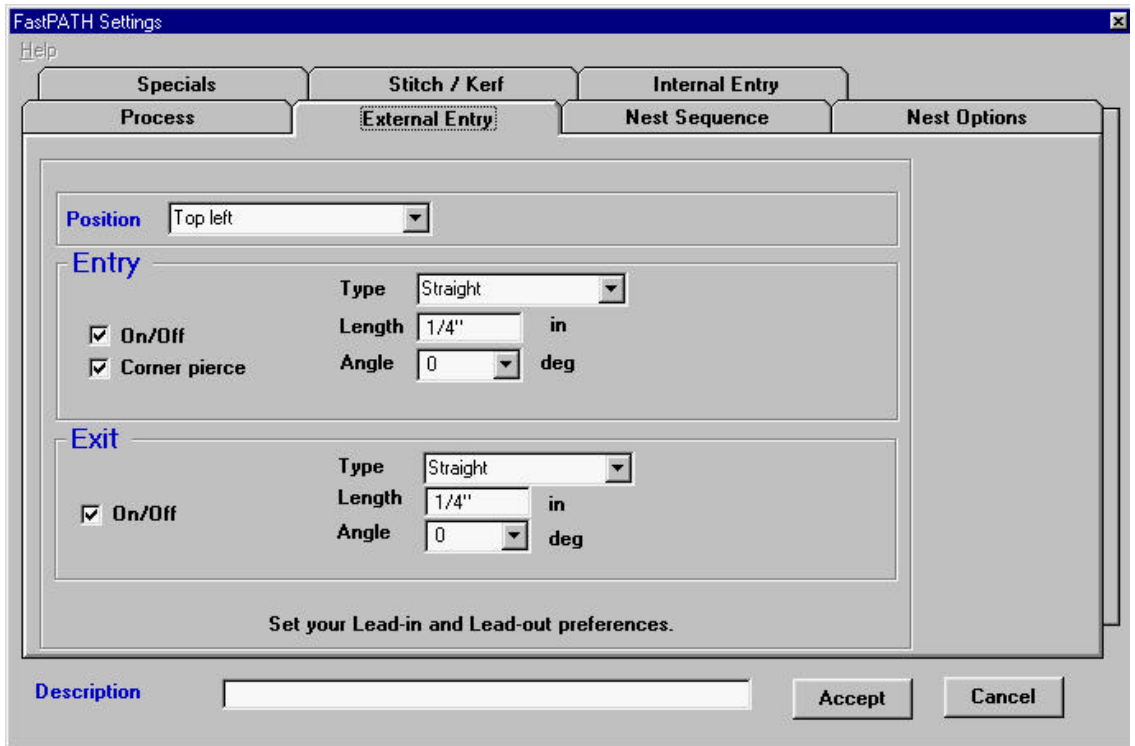
3-5-7 INTERNAL ENTRY Angle

This field sets the entry Angle. It can be either 0, 45, or 90 degrees. The entry angle is measured relative to the direction of cut. For circular entities with straight entries the appropriate entry angle will be substituted for this angle if it is inappropriate. If the entry angle is inappropriate FastPATH will substitute a more suitable angle.

3-5-8 INTERNAL ENTRY On / Off

When the box is checked an internal entry and/or exit is turned on. If unchecked the entry and/or exit is turned off.

3-6 EXTERNAL ENTRY



The image shows the 'FastPATH Settings' dialog box with the 'External Entry' tab selected. The dialog has a title bar with 'FastPATH Settings' and a 'Help' button. Below the title bar are four tabs: 'Specials', 'Stitch / Kerf', 'Internal Entry', and 'External Entry' (which is active). Under the 'External Entry' tab, there are four sub-tabs: 'Process', 'External Entry', 'Nest Sequence', and 'Nest Options'. The 'External Entry' sub-tab is active. It contains the following settings:

- Position:** A dropdown menu set to 'Top left'.
- Entry:**
 - ☒ On/Off
 - ☒ Corner pierce
 - Type:** A dropdown menu set to 'Straight'.
 - Length:** A text box with '1/4"' followed by a unit dropdown set to 'in'.
 - Angle:** A text box with '0' followed by a unit dropdown set to 'deg'.
- Exit:**
 - ☒ On/Off
 - Type:** A dropdown menu set to 'Straight'.
 - Length:** A text box with '1/4"' followed by a unit dropdown set to 'in'.
 - Angle:** A text box with '0' followed by a unit dropdown set to 'deg'.

At the bottom of the dialog, there is a text box labeled 'Description' and two buttons: 'Accept' and 'Cancel'. A note at the bottom of the settings area reads 'Set your Lead-in and Lead-out preferences.'

3-6-1 EXTERNAL ENTRY Position

The Position box sets the Entry Position. This is the place where the programmer would like entries on the part. If a part does not have a defined corner, for example, it is notched in the position that is required; the best available place is selected. This may not be the programmer's

desired entry position. To overcome this, always ensure a definite point exists where the entry is required. When corner pierce is not selected, entries will always be placed in the middle of an entity to the left or right (depending on kerf) of the selected start position. The start position will always be as close as practicable to the position specified, considering the entry type. The positions available are Top Left, Bottom Left, Top Right, Bottom Right, Top, and Right.

3-6-2 EXTERNAL ENTRY Corner Pierce

With a tick next to Corner Pierce the entry (or lead-in) will start at the corner of an entity. If it is not ticked, the entity closest to the start position selected by the "Position" field will be broken in the middle and an entry attached in this position.

3-6-3 EXTERNAL ENTRY Type

The entry type can be either straight, quarter circle, half circle or none. All these entry types are the same as the entry types available in FastCAM, when a part is manually pathed.

3-6-4 EXTERNAL ENTRY Length

The length of the entry is specified in this field. If a circular entry is specified in the Type field then the length here is the arc radius. Both fractional and decimal inches can be specified in this field, as well as millimeters depending on the units set in FastCAM.

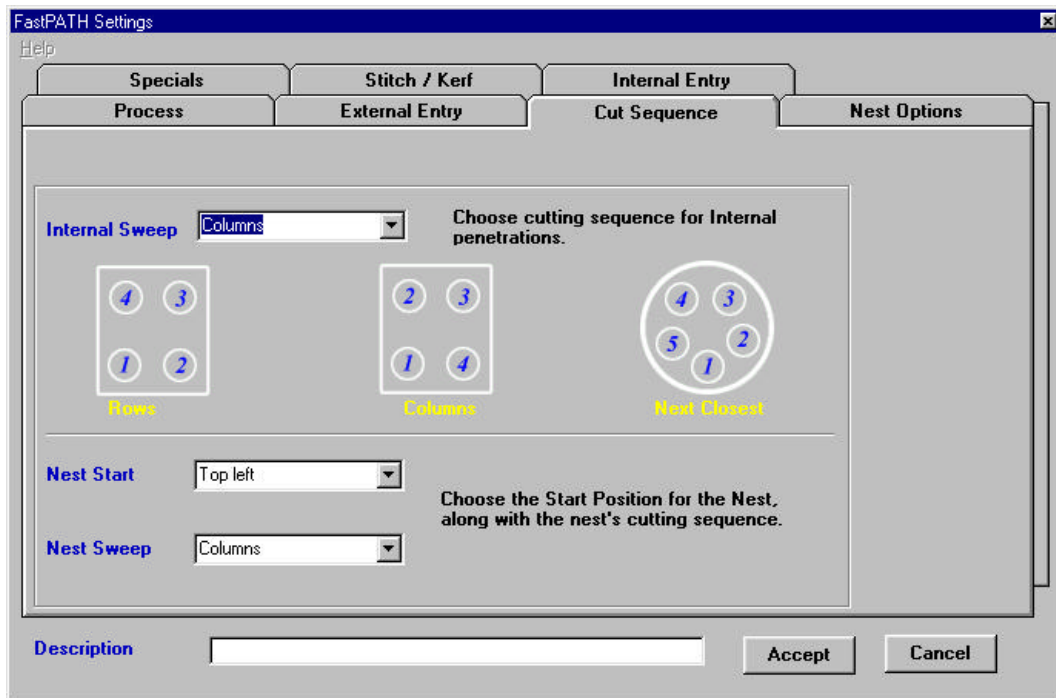
3-6-5 EXTERNAL ENTRY Angle

This field sets the entry angle. It can be either 0, 45, or 90 degrees. The entry angle is measured relative to the direction of cut. For circular entities with straight entries the appropriate entry angle will be substituted for this angle if it is inappropriate.

3-6-6 EXTERNAL ENTRY On / Off

When this box is checked an entry or exit is turned on. If unchecked the entry or exit is turned off.

3-7

NEST SEQUENCE

3-7-1

Internal Sweep

This setting specifies how FastPATH will path internal penetrations or drops/cutouts inside parts. The internal sweep can be either Horizontal, Vertical, or Next Closest. Each setting is better suited to a specific situation. By referring to the diagrams a user will be able to ascertain the best approach. The first cut penetration will always be towards the bottom left hand side of the part.

3-7-2 Nest Start

The Nest Start position specifies where the first rapid or torch home position is on a plate. This setting is used when an entire nest with a plate boundary is pathed. The starting position can be in any corner of the plate top left, bottom left, top right, bottom right. The setting here overrides the default setting that is loaded with FastCAM

3-7-3 Nest Sweep

The Nest Sweep describes how the nested parts are going to be cut from the plate. The nest sweep can be either to the Right or Up the plate. The sweep will then snake through the parts in the direction specified.

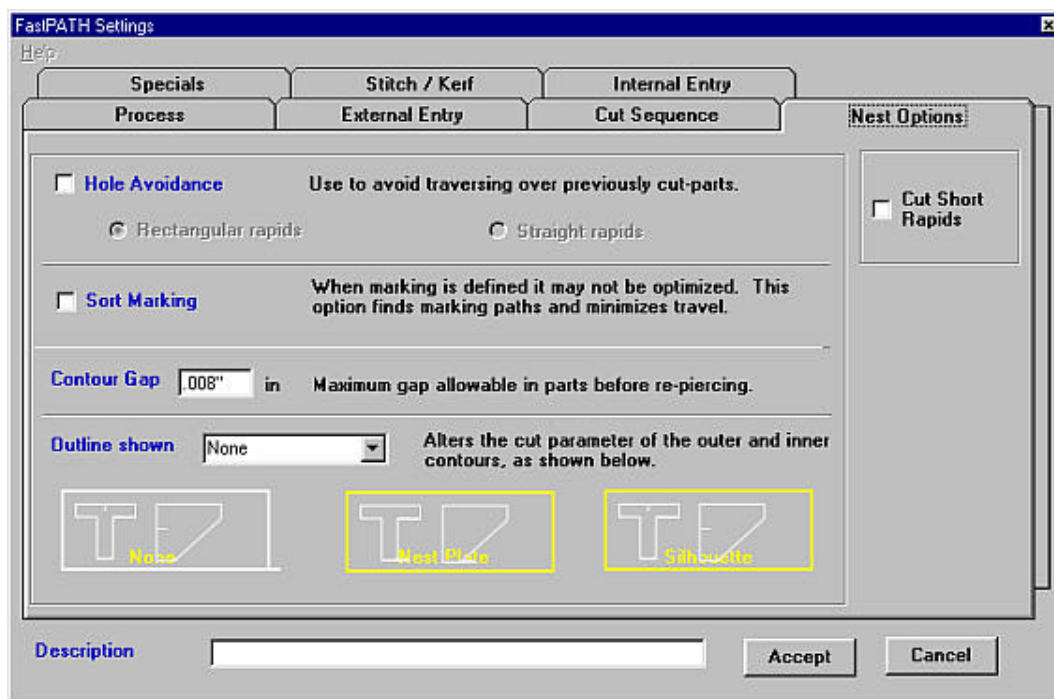
3-7-4 Description

The Description field is used to enter a detailed description of the FastPATH settings that have been set. This allows a user to ensure retrieved settings correct without the need to view each individual setting.

3-7-5 Accept

The Accept button loads the currently set parameters into FastPATH. Once the parameters are set, a User is able to save these settings for later use.

3-8 Nest Options



3-8-1 Hole Avoidance

Turning on Hole Avoidance allows FastPATH to produce a nest that could be run unattended. Each part is cut, ensuring the rapid between parts does not traverse over previously cut parts. If the cutting head will pass over a previously cut part a “head lift” command is issued. This type of output is suited to high-speed laser cutting but can be used for most processing applications if the machine supports it.

Two types of Hole Avoidance programming are available: Rectangular Rapids or Straight Rapids.

3-8-1-1 Rectangular Rapids

When Hole Avoidance is set to Rectangular Rapids the torch will stay down as long as practicable. The torch will traverse between cut parts, lifting only when necessary.

3-8-1-2 Straight Rapids

When Straight Rapids are selected, the torch will rise and traverse directly to the next part that is required. The torch will therefore pass over previously cut parts. With the head retracted this should not cause a problem.

3-8-2 Sort Marking

When parts are marked as well as cut it is optimal to mark the parts first then cut the profiles. This processing sequence is especially true when using a powder marker and cutting with plasma underwater. When Sort Marking is checked all marking will be sorted so it is performed first, with the cutting of parts to follow.

3-8-2 Contour Gap

When DXF parts are loaded into FastPATH and processed, there is a possibility that the contour is not closed. FastPATH overcomes this problem with the ability to set a Contour Gap. The Contour Gap is the distance below which a contour is considered to be closed. For example, there is a break in a part's contour of 0.005 inches (0.13mm) and another break in the contour of 0.01. (0.25mm). If the Contour Gap is set to 0.001 inch (0.03mm), the break of 0.005 inches (0.13mm) is jumped over while

the break of 0.01 inches (0.03mm) creates a 0.01 inch (0.03) break in the contour. Only the single contour break is recognized.

3-8-3 Cut Short Rapid

This function will cut rather than rapid between the exits and entries of parts that are close together. The rapids will only be cut if

1. rapid movement is <50mm
2. rapid movement is less than 10% of plate size
3. rapid movement does not cross another part boundary.

When paired and used with FastNEST'S Common cutting of pairs, plate utilization and speed of processing is increased.

3-8-4 Outline Shown

Depending on the part to be cut the Outline Shown specifies how a part will be processed by FastPATH. When the output from FastPATH is not correct it is usually this setting that is not appropriate for the part or parts. For example no output would be generated for a path with a single outside contour when FastPATH is set to Nest Plate.

3-8-3-1 None

If the part to be FastPATHed is just that, a single part, then outline shown should be set to None. Most single files will be processed with this setting. This setting is also used when part files are bulk converted in FastPATH using the Batch Parts function.

3-8-3-2 Nest Plate

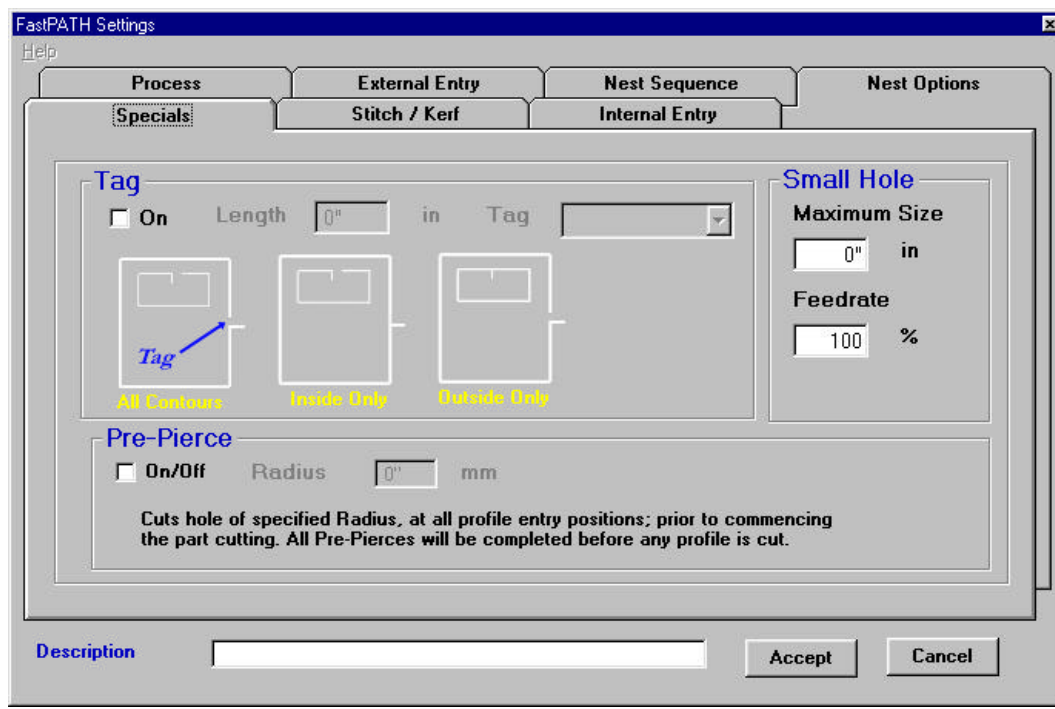
If a complete nest is to be pathed, with the plate outline drawn around the parts, then Nest Plate should be selected. If a complete nest is to be pathed

without a plate outline around the parts, then none should be selected, or a plate outline drawn. When FastPATH is called from within FastNEST, the Outline Shown setting will always be set to Nest Plate. It cannot be changed.

3-8-3-3 Silhouette

Silhouette is used to path a part (or nest) that will be cut like a stencil or silhouette. All profiles inside the external boundary are treated as internal holes, rather than external paths. If you currently have FastCAM's FontGEN program the text output can be bridged in FastCAM, then a plate outline drawn around it and the text will be cut as a stencil.

3-9 Specials



3-9-1 Tag

The Tag option allows nested parts to remain attached to the skeleton. The tagged parts can be either:

3-9-1-1 All (fully tagged)

When this option is set both parts and drops (internal cut outs) stay attached to the main skeleton by the tag width specified.

3-9-1-2 Inside Tag

Drops (internal cut outs) remain attached to parts. The part is separated from the skeleton

3-9-1-3 Outside Tag

Drops (internal cut outs) are cut from the part. The part remains attached to the skeleton

The contour to be tagged is selected from the dropdown list. The specified tag length is applied to both inside and outside tags.

3-9-2 Pre Pierce

The Pre Pierce function is used to process a nest by first pre piercing all entries and then returning to those entries, using them as an edge start. This has the benefit of a better quality cut. An old or worn torch can be used to cut the pierce hole. The tip can then be replaced with a newer or better quality tip and the profile is then cut. Doing this could also provide better utilization of a tip, as a tip that can no longer cut nicely could still be used for piercing.

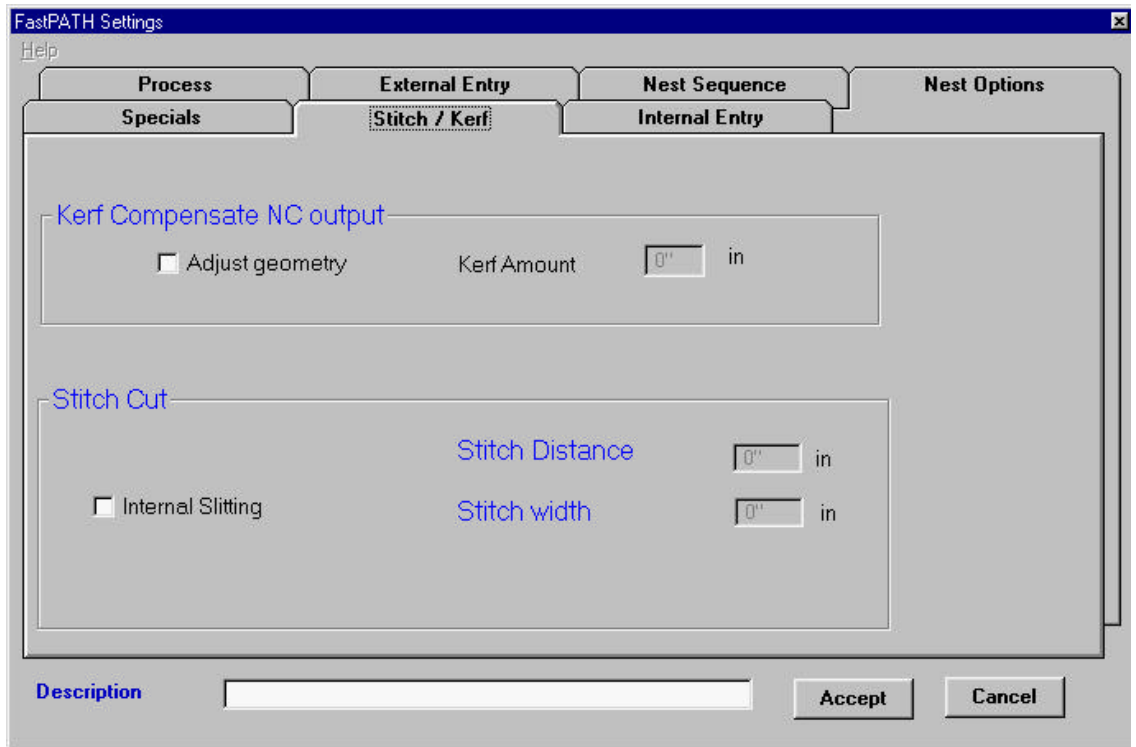
The hole diameter, is the hole diameter cut by the torch for each pierce. When the torch returns to cut the profiles after first piercing, the cutting path will start at the edge of the hole. The edge start will thus require less heat input and also produce an easier start.

3-9-3 Small Hole

The Small Hole option allows the variation in cutting speed when cutting "holes" less than a specified size. The feedrate used is a percentage of the initial feedrate for the part. For example if the initial cutting feedrate was set to 120 in/min (3050mm/min) then a Small Hole Feedrate of 80% would change the feedrate to 96 in/min (2440 mm/min)

An important factor to note is that the hole does not need to be circular. A rectangular hole that fits inside the hole size specified will have its cutting speed reduced. The hole does not need to be closed. If a notch was cut in a plate and the notch was smaller than the hole specified then it too, would be subject to the reduced feedrate.

3-10 Stitch Kerf



3-10-1 Kerf Compensation

This is an optional feature to FastPATH. This function becomes available when the Kerf option is purchased for FastCAM. When initialized, the Kerf Compensation will modify a part's geometry based on the kerf width specified. All internal paths will be made smaller and all external paths will be made larger. This ensures all parts when cut will be cut to size.

When the NC code is produced for the finished part or nest, kerf will still be enabled in the code. It is paramount that no extra kerf be added to the parts on the controller. This problem can be overcome by adding kerf in

FastPLOT, where the kerfed output will be final.

IMPORTANT

The Kerf specified in all FastCAM's kerf compensation routines is the cutter radius. Many kerf tables specify cutter diameter. This value needs to be halved if it is to be used in FastPATH. All programmers need to ensure the value entered is correct.

3-10-2 Stitch Cut

Turning on Internal Slitting automatically Stitch Cuts an internal path according to the set parameters. The internal path cannot be closed nor can it touch an outside boundary. This function can be used for automatically stitch cutting nested lobster backs or penetrations in cones for example.

3-10-2-1 Stitch Distance

The Stitch Distance is the amount of cut or the distance the torch will travel while cutting the stitch.

3-10-2-2 Stitch Width

The Stitch Width is the separation distance between stitches or how far the torch will travel without cutting.

For a circle to be stitch cut it needs to be broken as well as a separation between both ends. This can also be achieved by drawing an arc which swings through a near complete circle, for example 359 degrees.

Chapter 4 - FASTPLOT™

The FastPLOT program provides the capability to Verify, Edit and Sort part programs.

NOTE: FastPLOT is a FastCAM optional feature

4-1 Fast Plot NC FILE

4-1-1 NC FILE SCAN

Scan allows for the rapid viewing of the geometry in NC code. Scan will display all the files in the current directory, which you can choose to open in FastPLOT for further verification or editing. *(If you need to change to a different directory, please refer to your Windows 95/NT manual).*

To select a part to be verified, simply click on the part and then click the SELECT button *(you can also simply double-click the part name)*. The part will then be displayed on the screen, showing all paths and Start & Stop points.

Just like the FastCAM File Restore box, after one file is selected the mouse is represented with an hourglass, by moving or clicking the mouse, the SCAN open box will be shown again for another selection. This method is simply giving you a quick view method to find the correct file.

Once the files have been verified, and you have finished, you simply hit the CANCEL button. This will then bring up a prompt, asking you if you would like to edit the selected file.

If you select NO, you will then be placed back at the FastPLOT Main Menu.

By selecting YES, the EDITING feature of FastPLOT will be started.

4-1-2 NC FILE EDIT

The EDIT function allows the NC Code of the program to be verified and manually edited if necessary.

Like the SCAN function, the Open box is presented, which enables the selection of the file to be edited. Upon file selection, the NC FILE EDITOR screen will be displayed, and the code for the program will be listed down the left hand side of the screen.

4-1-3 NC FILE PLOT

The PLOT function is used to PLOT out an image of either a part program or a completed Nest program.

The windows open box is displayed, from which a part to plot is selected, following this the windows printer box will be displayed. After selecting the required printer settings, click the OK button and the View menu will be displayed in the center of the screen. This view menu allows you to select the Change Display option, from which you can change display properties for the print out. The Plotter Pen Width allows the darkening of lines that are printed. A value of 5 produces a reasonable result. The Plotter Label can be either None (no label), or Screen (Simple part information), or Template (More complex part information). None is selected by default.

4-1-4 NC FILE COST

The Cost function allows costing to be performed on individual NC parts or nested programs. Some user information is required, such

as: Cutting speed, Rapid speed, Cost per pierce, Handling Time, Part Thickness, Machine Rate, and Material Cost.

From the windows open box the program to be costed is selected. Once a file is selected the Costing screen will then be displayed. Input the required values, and the cost of the part will be displayed. This screen can also be printed. See the following graphic for example.

NC Profile Costing Print Exit

Filename test Date 03-22-1998 Time 22:45:13

Blocks 49	Part Thickness <input type="text" value="25"/> mm	Rect. Width 966.82 mm
Characters 405	Material <input type="text" value="MILD STEEL 250"/>	Rect. Height 933.34 mm
Machine LYNX	Density <input type="text" value="8000"/> kg/m3	Rect. Weight 180.474 kg
Control CONTROL.D.	Cut Area .9 m2	Utilization 99.33 %
Torches 1	Cut Weight 179.274 kg	Scrap Weight 1.201 kg

Process	Distance (metres)	Feedrate (mm/min)	Pierces / Starts	\$/Pierce	Secs/Pierce	Time (Minutes)
RAPID	1.33	6000	0			.22
PLASMA	4.12	3750	5	.25	5	1.51

+ Handling Minutes Pierce Costs \$ 1.25

Total Time 6.74 Minutes @ \$/Hour = Machine Time \$ 13.47

Weight .18 Tonnes @ \$/Tonne = Material Cost \$ 144.38

Total \$ 159.1

4-1-5 NC FILE LIST

List directs the NC file to the printer. The code will be printed in column format. The "listing" includes the following data.

- **Line Number.** The actual line number in the NC code
- **X,Y Absolute position.** The position of each block of code in absolute, from the first position in the NC code. This is extremely useful for determining torch position when re-starting jobs after power failure or at other times when the zero of the job is lost
- **The code itself.** Useful in debugging programs or locating a position for restarting programs.
- **Type of program,** listing appears as Inc or Abs. Meaning the NC code is in Absolute or incremental form.
- **Units,** mm or inch. Refers to the units of the coding.

4-1-6 NC FILE Kerf Compensate

This function will automatically kerf compensate an NC file. Once the file has been selected the amount of kerf should be entered. The kerf offset can be inspected and the kerf compensated file saved. This is an option for FastPLOT and suited for Burny Controllers.

4-1-7 NC FILE EXIT

This will allow you to EXIT from the program.

4-2 NC UTILITIES

4-2-1 NC UTILITIES CONVERT to CAM

The Convert to CAM function, allows you to Convert/Translate any NC program into a CAM file. All cutting paths, lead-in/outs, drill processes will all be converted back to the original geometry which was used to create the NC program. Convert can be used in many cases, but the most common is if the original geometry file (*CAM file*) has been lost, and changes need to be made to the part. Simply convert the NC program back and make the changes; there is no need to redraw the complete part.

Once convert to CAM is selected, you will be presented with the windows open box, from which you can choose the file to be converted. FastPLOT will process the file, and save it as a CAM file in the same directory from which it came. You will then be asked to Print out a listing, after your choice has been given, you have the option to go into FastCAM or to stay in FastPLOT for further verification.

4-2-2 NC UTILITIES EXPAND NC

This feature allows the part or parts in NC code format to be expanded by a value. This value can be a percentage, factor or a distance.

- **Percentage:** This allows a part to be expanded by a value expressed as a percentage. i.e. a percentage of 200% (a value of 2) would make the part twice as big.
- **Factor:** This allows a part to be expanded by a value expressed as a factor. i.e. a factor of two would make the part twice as big.

- **Distance:** This allows a part to be expanded by a value expressed as an offset from the original component. This can be used to compensate for kerf where the kerf has not been allowed in the original NC code.



Caution should be used in expanding parts as the expansion will apply equally to the inside as well as the outside. Expansion is often used to compensate for a large thermal input (e.g. cutting heavy plate), which would otherwise shrink the component beyond the accuracy required. Negative values can also be used to shrink a part if required.

4-2-3 NC UTILITIES

SORT NC

Sort NC allows processes to be sorted for NC code containing multiple processes. The most common would be for marking within a nest. This allows all marking to be completed before cutting commences. Other uses are to process nests where the process is not available. i.e. marking is not installed on the machine. In that case the marking can be sorted to the back or front of the code to make editing easier.

Sorting can be set up automatically for FastCAM® users with the FastPATH option installed. Processes available on profiling equipment can include the following.

- Marking, Powder and Plasma Marking
- Multiple cutting process i.e. Plasma heavy and light plus Oxygen
- Text marking, Vector and Ink-Jet
- Drilling and tapping
- Beveling. Including, Plasma single and multi pass, Oxy single and multi torch
- Spot Drilling

4-2-4 NC UTILITIES**EXPLORER**

Short Cut option to start Windows Explorer.



This function currently only operates under Windows 95

4-2-5 NC UTILITIES**FASTCAM DRAWING**

The FASTCAM drawing function allows direct access to the FastCAM Drawing and Pathing program from within FastPLOT. This option eliminates the need to EXIT from the program and restart FastCAM.

4-2-6 NC UTILITIES**ENABLE OPTION**

Additional FastCAM components can be added using this function. Some of these components include, Automatic Nesting, FastPATH, Kerf Compensation, Stick Font Marking, other CAD drawing file formats. Please contact your FastCAM sales representative for further information.

4-2-7 NC UTILITIES**LICENSE**

Displays information about the Authors of the software and the licensing for the program.

4-2-8 NC UTILITIES**CONVERT TO DXF**

Automatically converts an NC file to a DXF. This allows the conversion and subsequent opening of NC files in programs such as AutoCAD.

4-3 NC PROFILE

4-3-1 NC PROFILE CHANGE NC

This function allows the user to change NC Control types. If you have purchased multiple post processors it will be enabled. It is disabled by default.

4-3-2 NC PROFILE DISPLAY NC

This option displays the current control file in a spreadsheet format, purely for visual checking and confirmation of control type.

4-3-3 NC PROFILE LICENCE

Displays information about the Authors of the software and the licensing for the program. It also shows the Program's License Number that is used to enable program options.

4-3-4 NC PROFILE ABOUT

Provides information about the version and date of the program as well as any loaded options.

4-3-4 NC PROFILE**TRANSLATE NC**

This is an optional module for FastPLOT. The FastTRAN module allows the automatic conversion of NC code from one controller type to another. This is quite useful when a new machine is purchased, for example a Burny 2.5 plus, and old programs (from a Linde) need to be cut. This option also includes mapping of differing machining functions. An example is Oxy code to Plasma or Drilling to Spot Marking. Please contact your FastCAM reseller for more information.

4-3-5 NC PROFILE**BOX NEST**

This function is designed for controllers that support subroutines. It can also be used effectively without them. By selecting an NC file, then the plate size, and part separation, the plate is automatically filled with the selected part. If required, subroutines are used to reduce the size of the generated NC code.

4-3-6 NC PROFILE**RESET FILES**

Resets the temporary files created by FastCAM. This can resolve the problem of FastCAM becoming lost or not opening correctly due to a strange error.

APPENDIX A

FastCAM[®] File Structure

FastCAM's one step programming system is located in the following directory:-

C:\Program Files\Fastcam

The programs FASTCAM5.EXE FASTNEST5.EXE and FASTPL5.EXE control the three main sections of the FastCAM suite.

FASTCAM5.EXE is the line drawing, parts construction, and NC code generation program.

FASTPL5.EXE allows the programmer to visually verify; cut direction, rapids, punch marks, & order of cut.

FASTNEST5.EXE is the interactive and automatic true-shape nesting program.

All the main programs produce several types of file extensions. Each extension represents a different type of file used within or produced by the main program.

The following is a list of the extensions and how they relate to FASTCAM5.EXE, FASTNEST5.EXE or FASTPL5.EXE.

FILE EXTENSIONS:

1. FASTCM5.EXE FILE EXTENSIONS:

***.CAM**

FastCAM geometry files (***CAM Files***) are produced when a **File Save** is done in FastCAM. These files are used in FastCAM and FastPLOT. **They cannot be taken to the controller to be cut.**

***.PTH**

FastPATH data files produced when FastPATH is used and the settings are saved.

***.TMP**

FastCAM temporary file, containing information regarding File Paths and FastPATH file names.

BACKUP.CAM

FastCAM backup files. These files are created when NC code is generated. It is a CAM file of the last generated NC part.

***.* FILES WITHOUT AN EXTENSION**

No extension files are produced when a part program file is generated under **OUTPUT NC CODE** in **PROGRAM PATH**. These files contain NC language and can be used in the FastPLOT program for verification, nesting, or can be taken directly to the controller and cut.



NOTE: *The cutting of these files depends on your Control file setup, Please ask your software supplier for technical assistance.*

Also note that the “no extension” files may be given an extension by the user, e.g.: ***.PRT**, or ***.TXT**. The extension has no effect on the NC program.

2. FASTPLOT.EXE FILE EXTENSIONS:***.TMP**

FastPLOT temporary file, containing information regarding File Paths and NEST data.

***.EXP**

Used when a file is expanded in NC Utilities-Expand NC. Distinguishes the expanded part file from the original file. Can only be performed on an NC file.

***.SRT**

Used when a nest is sorted in NC Utilities-Sort NC. Distinguishes the sorted file from the original file. Can only be performed on an NC file.

***.* FILE WITHOUT AN EXTENSION**

Produced in FastPLOT by ending a nest. This file contains NC code and can be used in FastPLOT for verification or taken to the controller to be cut.

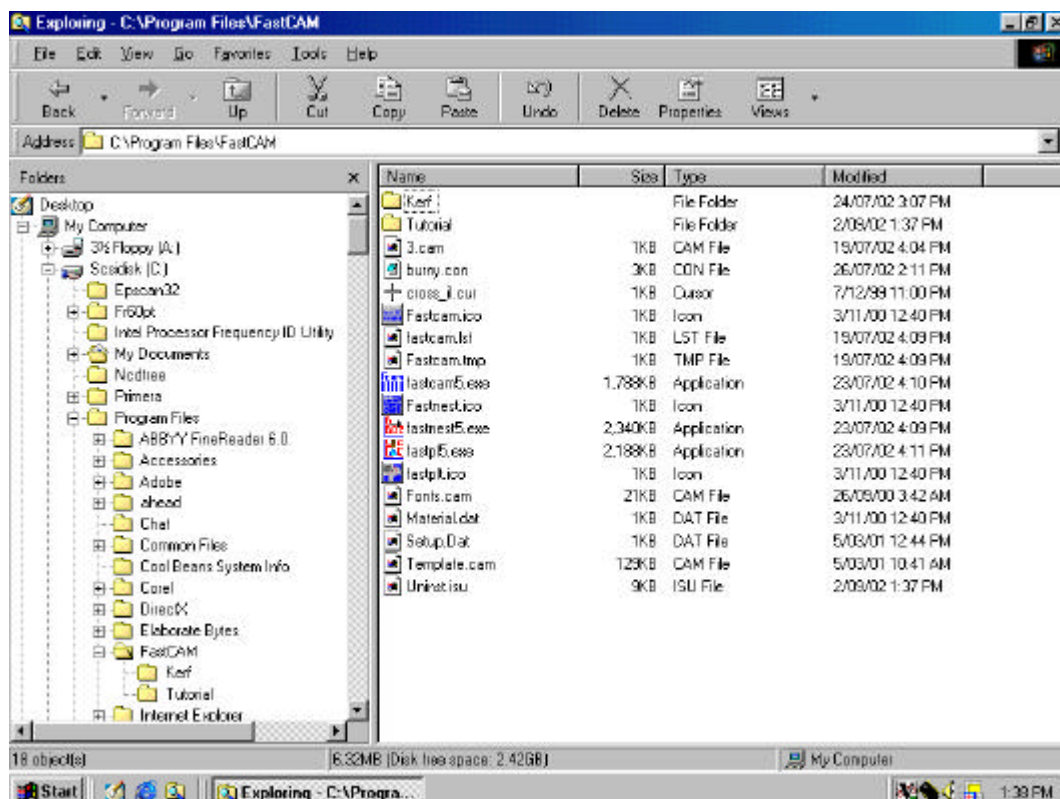


NOTE: The “no extension” files may be given an extension by the user, e.g.: ***.PRT**, or ***.NC**. Some controllers require a specific extension, so please consult your Controller manual for reference.

3. FastNEST FILE EXTENSIONS.

FastNEST file extensions can be found in the FastNEST manual.

FASTCAM DIRECTORY



Above is a basic list of the files contained in the FastCAM directory, just after installation.



NOTE: Other files may appear in this directory after the program has been used, e.g.: FastCAM.TMP.....etc.

DESCRIPTION:

Burny.con:

This file provides the G and M codes or controller codes, which are placed into the NC program. Contact your software reseller prior to making any changes to this file.

NOTE: This file may be under a different name: ANCA2000.CON, LYNX.CON, CONTROL.CON....etc.

Fastcam.bmp

The opening FastCAM picture file, if present

Fastcam.ico:

This is the FastCAM icon (picture) file.

Fastcamc.txt:

This is a language file for chinese, the name may change depending on your installed languages (if present).

Fastcamo.txt:

This is the generic ENGLISH language file.

Fastcm5.exe:

The main FastCAM drawing program.

Fastnest5.exe:

The Automatic/Interactive Nesting program (the inclusion of this file depends on the modules that have been purchased)

Fastnest.ico:

This is the FastNEST icon (picture) file.

Fastpl5.exe:

The main plotting, and verification program.

Fastplt.ico:

This is the FastPLOT icon (picture) file.

Feeds.dat:

This is the Feedrate table that can control the Feedrate of the profile machine. This is only installed on selected machines, contact your software reseller for further information.

Material.dat:

This is a list of materials, which can be assigned to a part or program from within FastCAM/FastPLOT.

Readme First.wri:

This is a WRITE document which explains details on the connection of your FastLOC Dongle.

Setup.dat:

Setup.dat configures the computer so FastCAM will perform correctly. The file contains certain information, which can be changed, see SETUP.DAT later in this chapter.

Template.cam:

This is the English Template file used when printing in FastCAM/FastPLOT or FastNEST.

Templatec.cam:

This is the Chinese Template file used when printing in FastCAM/FastPLOT or FastNEST.

Cross_il.cur:

This is the cursor file for Windows NT and 2000.

Fonts.cam

This is the PlateMARKER font file. The fonts defined in this file will be used to mark plate.



NOTE: *If some of these files listed are deleted or destroyed your software may not function properly, if at all.*

CONTROLLERS & “CONTROL.DAT”

FastCAM supports the following controllers: ESSI, ANCA 200, MYNUC, LASER LAB, AVCA2000, PLATEMATE, AUTOPATH ESAB, COMPUPATH CREONICS, BURNY, PEDDIMAT, GE 1050, MAZAK, UCNC, FABRICATOR, ALLEN BRADLEY, FAGOR, TANAKA, CINCINATTI LASER, IWATANI, JHE, HYBRID, LYNX, LASER LAB 32bit, PDF 32, ALLEN BRADLEY 2, VISION 1000, ESAB, FARLEY ATLAS, FICHEP, DYNAPATH, KOMATSU, TANAKA, FANUC, SIEMENS, MESSER AUTOPILOT, FANUC ROUTER, JOLUMA, FAVOR LASER, FARLEY FABRICATOR, SILVERS, BOLICHANG BEVEL and VISION 2000 controllers. Each controller uses either the standard Word Address or ESSI programming language.



NOTE: *Many other controllers, are supported, please contact your software supplier for further information.*

Within FastCAM the file CONTROL.DAT sets up the specific tool codes for each controller. When NC code is generated, FastCAM looks to this file for the proper codes. For a programmer, there is little need to change this file, after initial setup.

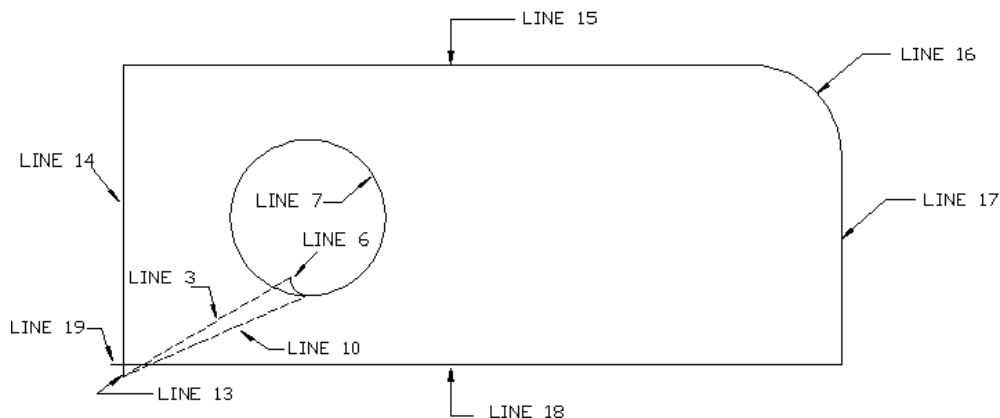
WORD ADDRESS LANGUAGE DESCRIPTION

Word Address language is the standard NC programming language in the United States. It consists of a series of G codes, M codes, X,Y, and I,J movements. G codes are used to define direction of kerf, and the type of entity to be drawn. These are fairly standard between controllers, while the M codes or miscellaneous codes are controller specific.

M codes are used to turn on and off tool processes and provide other specific controller functions. The X,Y values represent movement along the horizontal axis (X), or the vertical axis (Y).

A positive X will be a move to the right, while a negative X is a move to the left. Similarly, a positive Y is a move up and a negative Y is a move down. The I and J are used to locate the center of an arc or circle.

Look at the following print out of the part program for the drawing below.



FastCAM Output

DESCRIPTION

1. G70	Inch units
2. G91	Incremental
3. G0X0.44Y5.70	Rapid Traverse
4. G41	Kerf Left
5. M20	Plasma On
6. G3X1Y-1I1	CCW arc
7. J5	CCW Circle
8. M21	Plasma Off
9. G40	Kerf Off
10. G0X-10.13Y-4.52	Traverse move
11. G41	Kerf On
12. M20	Plasma On
13. G1Y.5	Line
14. Y17.82	Line
15. X29.25	Line
16. G2X3Y-3J-3	Clockwise Arc
17. G1Y-14.82	Line
18. X-32.25	Line
19. X-.50	Line

20. M21	Plasma Off
21. G40	Kerf Off
22. M02	End of Program

The first two G codes, G70 and G91, tell the controller that the program will be in English and Incremental units.

All Word Address programs produced by FastCAM will start with this coding. After these two preparatory G codes the program starts the cycle which is repeated.

The Kerf is turned on left with a G41.

The Tool is turned on with a, “tool-on” M code. Note that each controller has unique M codes for each process supported; (Plasma, Oxy, Punch, Marker, & Zinc marker).

Following the “tool on” codes is a series of G moves to perform the cutting functions.

G0	Rapid Traverse
G1	Line
G2	Clockwise Arc
G3	Counter Clockwise Arc

Each G code creating a move, is followed by an “X,Y” value defining the endpoint of the move.

When a rapid traverse or line is defined the **X,Y** values are all that is needed. When an arc or circle is defined, in addition to the **X,Y** there must also be an **I,J** value. As with a linear movement the **X,Y** defines the endpoint of the arc or circle, while the **I,J** defines the location of the center in relation to the start of the arc or circle. For example, if a clockwise arc with a radius of two inches is drawn, the block of code is: G2X4Y0I2J0.

Notice that the **I** movement is the distance along the X axis from the start of the arc to the center, and the **J** movement is the distance along the Y axis from the start of the arc to the center of the arc. A circle will have an

X,Y value of zero, because the start point and the endpoint of the circle are in the same place.

There are a couple of other points worth mentioning concerning *FastCAM* out-put and Word Address programming language. *FastCAM* produces a stripped down code with all unnecessary information removed.

By producing this form of code it is possible to load larger programs into the limited memory of the controller. The information being removed are modal codes and zero values. Modal codes are all of the G codes which remain in effect until another type G code is required.

For example, in the following program, a series of lines are drawn to form a box, only the first G1 is required, and any value of zero is removed.

<i>FastCAM</i> Output	Long Hand	Description
G70	G70	Inch
G91	G91	Incremental
G41	G41	Kerf left
M20	M20	Plasma on
G1Y4	G1X0Y4	Line
X4	G1X4Y0	Line
Y-4	G1X0Y-4	Line
X-4	G1X-4Y0	Line
M21	M21	Plasma off
G40	G40	Kerf off
M02	M02	End of program

CONTROL.DAT - LINATROL

/OPERATIONS/

*OPER	CONT	KERF	COL	LTYPE/ CHAR	PLOTTER	ON,OFF,START,STOP
-------	------	------	-----	----------------	---------	-------------------

PLASMA,	2,	-1,	15,	1,	1,	M7,M8
MARKER,	1,	0,	13,	3,	1,	M9,M10,M11,M12

/KERF RIGHT/

G42

/KERF LEFT/

G41

/KERF OFF/

G40

* GENERAL UTILITY STATEMENTS

/OPENING STATEMENTS/

/COMMENT ON/

/COMMENT OFF/

/RAPID ON/

/RAPID OFF/

/CLOSING STATEMENTS/

M02

/HALT/

/START OF NEST/

/END OF NEST/

M02

This is a brief description of the **G** and **M** codes commonly used in the LINATROL controller. For a detailed description please see your Linatrol programmers manual.

LINATROL M CODES:

M00 PROGRAM STOP:

M00 is a program interrupt normally used to allow the machine operator to check a part's dimensions or to remove parts from the plate. To continue execution of the program, the operator must press start push button.

M02 END OF PROGRAM:

All programs created with *FastCAM* will end with an M02. M02 turns off all active codes and raises all heads to the full up position.

M7 PLASMA ON:

Finds initial height if height sensor is active. If auto height sensing is not active the operator should lower the torch to the proper height and press the proceed button.

M8 PLASMA OFF:

Turns off plasma and raises torch.

M9 ZINC MARKER PROCESS ON:

Activates zinc marker.

M10 ZINC MARKER PROCESS OFF:

Turns off the zinc marker.

LINATROL G CODES:

For more information about Autopath **G** and **M** codes refer to the Linatrol programming manual.

G70 ENGLISH UNITS

G71 METRIC UNITS

G90 ABSOLUTE PROGRAMMING MODE

Absolute programming is when all points are referenced from an absolute 0,0.

G91 INCREMENTAL PROGRAMMING MODE

Incremental programming is when one point location is referenced from the previous point on the part, not necessarily absolute 0.

G00 or G0 RAPID TRAVERSE

Rapid traverses are the moves required between cuts to position the tool for the next cut. Rapid traverses are executed at full machine speed.

G01 or G1 LINEAR MOVE, OR A LINE

G1 indicates a straight line move and will be followed by an X,Y movement.

For example: G1X5Y1 This code will move the machine five inches in the X and one inch in the Y.

FastCAM output drops any coordinates which are 0, so it is possible to see a line of code with only an X or a Y.(Except for Allen Bradley and some Union Carbide controllers).

G02 or G2 CLOCKWISE ARC OR CIRCLE

G2 defines a Clockwise Arc or Circle. In addition to the X,Y values arcs and circles have an I and J value used to define the center of the arc or circle.

G03 or G3 COUNTER CLOCKWISE ARC OR CIRCLE

G3 defines a Counter Clockwise Arc or Circle. In addition to the X,Y values arcs and circles have an I and J value used to define the center of the arc or circle.

G40 DISABLE KERF**G41 LEFT KERF**

Kerf is the amount of material destroyed in the cutting process. In *FastCAM* only the direction of kerf is usually set and not a value. A kerf value can be inserted, which can be overridden by the machine operator at the controller. Kerf left must be used for plasma cuts unless a special right swirl ring is being used. A good practice is to always use left kerf to make problem solving easier. With left kerf the cut direction is counter clockwise for inside cuts and clockwise for outside cuts.

G42 RIGHT KERF

When using right Kerf, cut direction is clockwise for inside cuts and counter clockwise for outside cuts.

DEFAULTS & "SETUP.DAT"

SETUP.DAT

1. VERSION,VER 3.0 * MARCH 1997*
2. CUSTOMER, FMS
3. DRIVE,C
4. TORCHES,6
5. BLOCKNUMBERS,2
6. COMMENTS,0
7. CONTROL,18,CONTROL.CON
8. NEST,BOTTOM
9. PLACES,3
10. UNITS,1,1,0 'inch input, inch output, incremental

SETUP.DAT is used to setup the **PC** to run *FastCAM* properly. Some of the information can be changed by the user. If you are not sure what can be changed call **FastCAM** for technical support. If certain parts of this file are altered or destroyed your *FastCAM* system will not work properly, or at all. Please be careful when changing this file.

DESCRIPTION:

1. **VERSION**- Version of *FastCAM* currently in use.
2. **CUSTOMER**- Name of user licensed to operate *FastCAM*.
3. **DRIVE**- Hard drive on which *FastCAM* program resides. If a large, segmented Hard drive is being used then the segment on which *FastCAM* is operating must appear here.
4. **TORCHES**- Maximum number of Torches to be used in multiple torch nesting.

5. **BLOCKNUMBERS**,2- Inserts blocknumbers in both part programs and nest files.
6. **COMMENTS**,0- Turns off comment lines in output programs. Must be present for Burny control, running in EIA code.
7. **CONTROL**- Control validation. Should not be changed, will affect validity of parts constructed.
8. **NEST**- Specifies starting point for a nest. Can be set at top right, top left or bottom left, bottom right.
9. **PLACES**- Sets the number of decimal places to be output in an NC program. Three decimal places means that the software will round to the nearest 1/1000.
10. **UNITS**-1,1,0- Sets the system default to be Inch Input, Inch Output, Incremental. For Metric input, Metric Output, Absolute the setting would be 0,0,1 The Input and Output can be changed to metric and/or absolute output or any combination.

APPENDIX B

COMMON CONSTRUCTION EXAMPLES

This section of the manual has been added to give the user a visual example of how the construction functions in the software should appear on your screen.

This section includes examples of:

- Blending of Arcs and Circles
- Corner Constructions
- Fillet Blends
- Intersecting Lines and Circles
- Control Points
- Parallel Lines
- Lines Tangent to Circles
- Arc Construction
- Point Definitions (Absolute & Incremental)

APPENDIX C

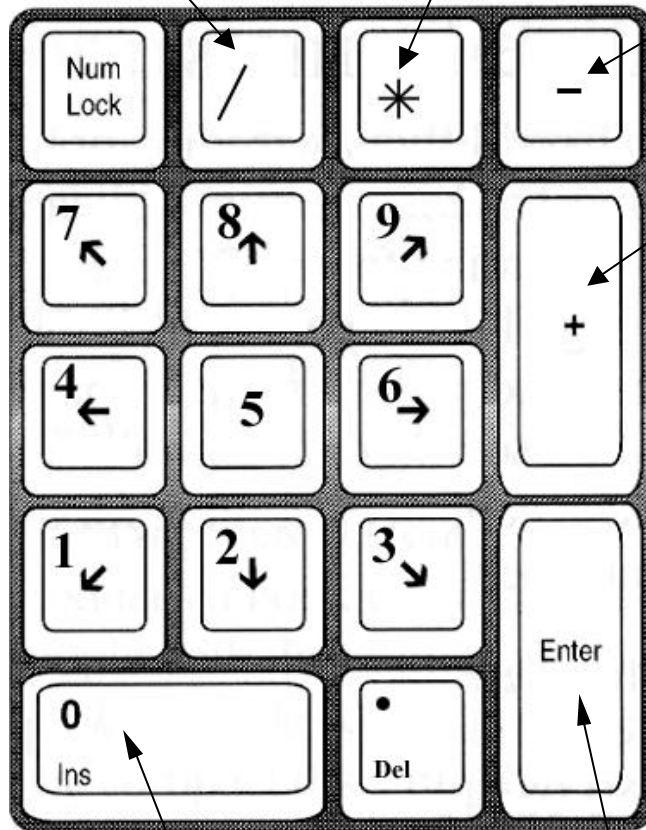
Reference Sheet for Enhanced Keypad Functions

Show CAD Layers

Control Points

Zoom Out
(Smaller)

Zoom In
(Larger)



Pan & Zoom, Speed Toggle

View - Repeat
(Refresh the screen)

APPENDIX D

THE TOOLBAR

The following is a list of definitions for the Toolbar used in FastCAM.



FILES-NEW



FILES-OPEN



FILES-SAVE



PLOT-FILE



LINE-BOX



LINE-TWO (2) POINTS



FULL CIRCLE



REAL POINTS



NOTCH CORNER



CHAMFER CORNER



FILLET CORNER



VIEW-AUTOSCALE



VIEW-INDICATE CORNERS



ADD TEXT



STRETCH



TRIM



ERASE LAST



PREVIOUS VIEW



SHOW CONTROL POINTS



UNDO

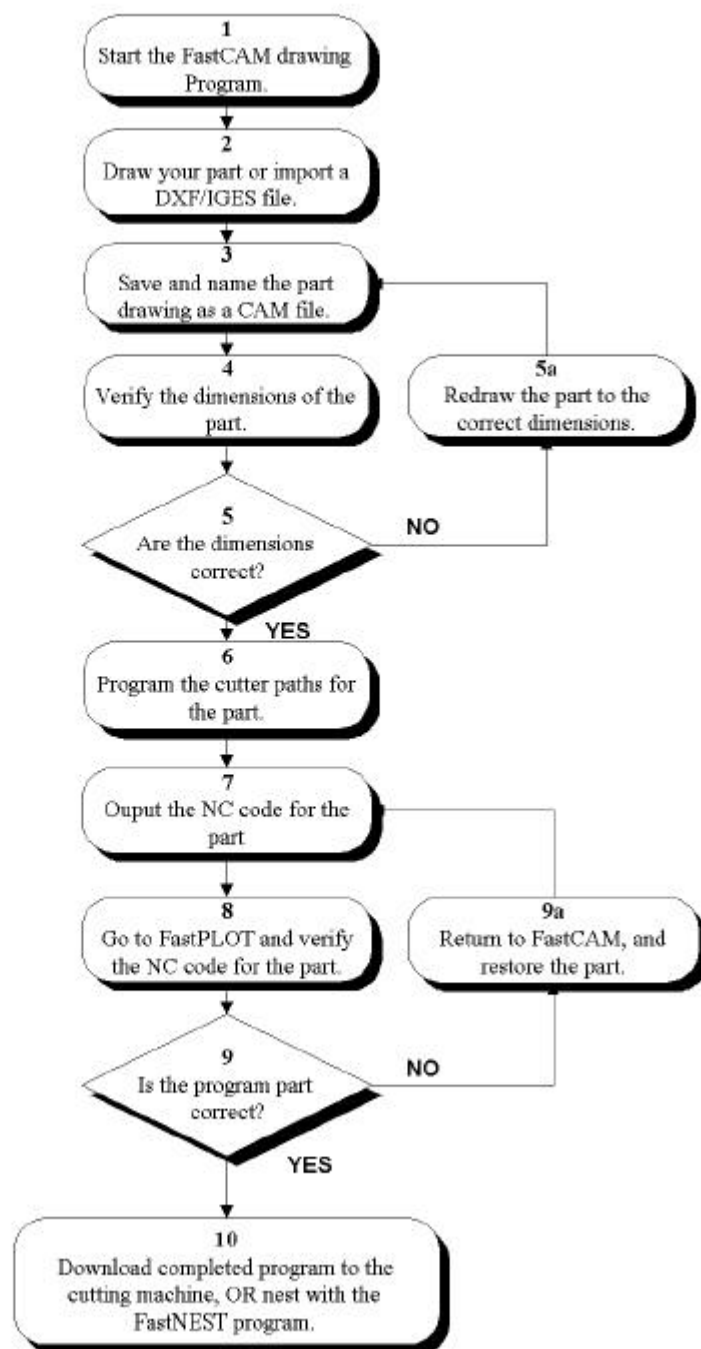


REDO



OUTPUT NC CODE

Overview of *FastCAM*



The following is a list of the Key words available for use within the Template

Key	Description	Inch	Metric
@area	part or nett area	49 sq.ft.	3.6 m2
@center	plot centre as x and y		103.431 304.1
@comments	comments		This is a comment
@control	NC control number		0 ESSI
@controlfile	name of control data file		ESSI2.DAT
@customer	customer		Atlas Steel
@cut	total cut length in mm or feet	11.6 ft.	3.45 m
@date	date		11/11/98 13:45
@density	density	4000 lb/ft3	1894 kg/m3
@drawing	drawing number or description		Drawing # 53
@drills	number of drills used in changer		2
@drilled	number of drilled holes		35
@fastcam	program name		fastcm99, fastpl99,fastnest
@feedrate	feedrate	60 in/min	1032 mm / minute
@heatnumber	heat number		H344
@job	job number		JJF3403
@kerf	kerf	.25 in	3.2 mm
@location	plate location description		BACK YARD
@mark	total mark length in mm or feet	4.7 ft	1.5 m
@material	material		STEEL GR350
@mill	mill number		HT341

Key	Description	Inch	Metric
@name	part name or nest name or NC name		TEST1.CAM, TEST1, NEST1
@nameprocess	Displays the most used cutting process		
@path	path of nc output		c:\test\customer1\
@pierces	number of pierces		15
@platearea	plate area	36.5 ft2	3.5m2
@platedesc	plate description		SLIGHTLY BUCKLED
@platename	plate name or number		MS10035
@plateweight	Plate weight for nest	2015lb	953 kg
@process	processing time		135.3 minutes
@pname	programmer name		M.J.F
@rapid	total rapid length in mm or feet		5.9 m
@remnant	remnant weight	1252.31 b	544.3 kg
@revision	Drawing revision number		Rev 1.31
@scale	scale		3.5 : 1
@scrapweight	Scrap weight	356.5lb	134 kg
@size	Size of part or plate		3.5 m x 1.5 m
@spacing	Torch spacing	4.3ft	1.2 m
@text	number of text operations		34
@thickness	thickness	8.34 in	20.5 mm
@torches	number of torches		1
@utilization	plate utilization or rect area for part		
@weight	nett weight of part or nest	9.35lb	3.5 kg

Additional Software Modules by FastCAM®

All products will run on Windows 98/2000/NT/ME or XP

FastNEST® Interactive Nesting Module:

Simple drag & drop part nesting with, dynamic part rotation, interactive arraying and high speed block nesting, including plate remnant nesting.

FastNEST®:

Fully automatic true shape nesting module for bulk processing. Highly optimizable nesting system with inbuilt FastPATH® automatic pathing system.

FastPATH™: Automatic & Intelligent Pathing Module.

Including DXF/IGES interface for CAD with automatic batch processing for FastNEST®. FastPATH® costs CAD and DXF nests and parts instantly. Generates NC code immediately. Eliminates hours of manual pathing. Two click positive and negative Bridging. Dramatic improvement in machine performance, saving up to half the cutting time. Dramatic savings in plasma consumables. Requires FastCAM.

FastSHAPES® Plate Development Programs

True Thick plate developments outputs NC code for any machine / controller combination. Outputs include DXF 2D & 3D, manual lofting layouts, FastCAM CAM format & NC Code including plate marking, Screen layout shows forming angle for pressing and shaded 3D model. Suite of software solutions to a number of common engineering and drafting tasks over a broad range of applications. The user provides an engineering specification of the task, a FastSHAPE® program provides the detail - patterns and /or solid geometry - as required. Over 18 individual FastSHAPE® programs in all.

FastTRACK® Remnant Tracking System

Plate and remnant plate tracking system. Accurate, simple and fast tracking of all remnant by exact shape, grade and thickness. Find Plate - rapid search of remnant stocks for a specific size, material, grade and thickness - find that plate fast! Stand alone product or FastCAM® compatible for automatic updating from nesting system.

FastFRAME[®]: Space Frame Fabrication

FastFRAME[®] addresses the long standing problem of developing pipe intersections for simple and complex pipe structures. System reduces months of work for a drawing office to less than one day for complete development of the pipe templates.

FastBEAM[®]: NC for Beamline Drills

Fast graphic programming system for Beamline equipment supporting Peddinghaus, Wagner & Ficep. Accepts output from most 3D structural CAD systems and DSTV generally. Beamlines used for the drilling, punching and cutting of rolled sections, such as Beams, Channels, Angles and Flat Bars.

FastCOPY[®]: Conversion of Templates to NC Code

Direct NC digitizing and optical template conversion system. With FastCOPY[®] it takes a programmer no more than 5 to 10 minutes to digitize a complex and large profile very accurately, including all holes, fillets, bends and the like. Parts can be digitized to any chosen scale. FastCOPY[®] is unique, worldwide in its operation, simplicity and accuracy.

FastCUT[®]: Optimization and Estimation for rectangular and linear

Simple to use! Designed to find the optimum layout of parts to be cut from any flat sheet material or lengths from rods and bars. Provides Clients with quick and accurate quotes. It even takes remnant into account. Anyone who has a need to shear flat SHEET or BAR material is a potential user of FastCUT[®].

FastCUT[®] Labels

Labeling program for FastCUT data files. Identification of parts produced using FastCUT optimization.

FastCUT[®] Cut to Length (CTL): Shear Cut to length lines

Software program that enables you to instantly calculate ideal blank sizes from coil based materials. Ideally used by large manufacturer or Steel Service Center who use large quantities of coil for processing. Results can be optimized for ideal single length blanks or mixed lengths.

FastLINK™ :DNC Communication

Upload and download NC files from Machine or PC depending upon controller type. Background or foreground program operation. Also multiple machine support available.

FastTRAN :Universal CNC File Translator

Universal translator for NC Code files to NC Code. Support for over 32 Controllers and multiple process. Need to convert Linde files into Burny code then FastTRAN is a must.

FastCAM® Profile Quoter: Quick Profile Estimation

Quote faster and quote smarter with this powerful tool. Quickly generate a quote for your customer using the library of shapes included in the program or use your own unique DXF/CAM file. Instantly calculates all data necessary for your processing estimate.

FastCAM® FontGEN: Turn your Window Font into NC Code

Now you can convert the millions of Windows Fonts into NC cutting files. Cut the text as is or explode for nesting. Type it in Word and cut it with FastCAM®. No CAD knowledge required.

FastCAM® Outline:

Ideal for converting non-dimensioned artwork into cutting files. Ornamental iron work, signs, machinery logos etc. Reads graphic files in various formats including BMP, TIFF, JPG. Converts data into DXF format suitable for processing via NC Machinery.

FastCAM® Kerf : Perfect for all Burny Controls

Simply run your NC programs with FastCAM® Kerf software on your PC and watch them fly the moment they download to the controller. Speed up production. No Kerf calculations to slow your production – do in seconds what can take hours on the control.

FastCAM® Text Marker: Marking and Identification on parts

Unique text marking utility for burning machines. FastCAM's Text Marker function converts CAD text directly into stroke font movements for marking letters and numbers directly onto profiled parts. Requires FastCAM® plus appropriate marking device or equipment fitted to the machine.

Tradesman in a Box: (TIB) Layout and Fabrication Software for Shop Floor

Software for Boilermaking and Sheetmetal. Ideal for hand cutting or optical pattern production. Loft patterns directly from printout. A complete shop layout system in a box – drawing and development.

- ✓ **Additional workstation licenses are available for all products.**
- ✓ **Educational versions available.**
- ✓ **SMA – Twelve Month Service Agreements can be purchased which include free* upgrades**

Many of the above full retail products are available for 30 day free trial and can be downloaded from our web site:

www.fastcamusa.com

International Offices

Customers can obtain additional information on the FastCAM® product range by contacting your Local Reseller or from any of our International Offices. They offer high quality and expert service.

<http://www.fastcamusa.com>

North America / Europe:

FASTCAM Inc:

8700 West Bryn Mawr,

Suite 800 South

Chicago 60631-3507 USA

Tel: (312) 715 1535

Fax: (312) 715 1536

Email: fastcam@fastcamusa.com

Australia:

FASTCAM Pty Ltd

96 Canterbury Rd,

Middle Park 3206

Victoria, Australia

Tel: (61) 3 9699 9899

Fax: (61) 3 9699 7501

Email: fastcam@fastcam.com.au

China:

FASTCAM China

Level 21, HSBC Tower,

101 Yin Cheng East Rd

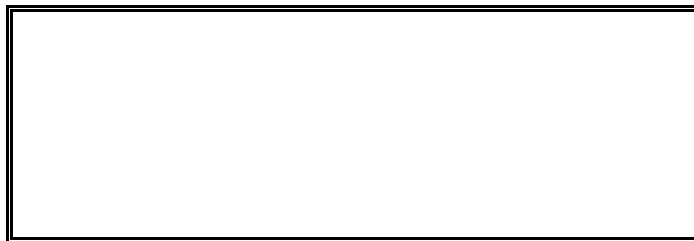
Pudong, Shanghai 200120 China

Tel: (8621) 2890 3127

Fax: (8621) 2890 3131

Email: fastcam@fastcam.com.cn

Or contact your local Reseller



Glossary

ABSOLUTE PROGRAMMING - The method of programming parts in which all points on the part are referenced by an X,Y move from a 0,0 position.

ANGLE- The relation of two straight lines originating from a single point.

ARC - A continuous portion of a circle. One of the primary units of construction found in the *FastCAM* drawing program. Arcs have three control points, start, center radius, and stop (or end).

ARRAY - A group of parts arranged into rows and columns. Arrays are found in *FastCAM* under BLOCKING and in FastPLOT under NESTING.

AUTOSCALE - Re-scales the graphics screen to a size which includes all points and entities in the current view. Found in *FastCAM* and FastPLOT under the menu header PLOT.

BLOCKS - Found in *FastCAM* under the main menu header BLOCKS. Used for defining a group of entities into one block of information so they can be moved, copied, or used in any of the BLOCK functions.

CAD - Computer Aided Drafting. Most drafting these days is done on computer using one of the many drafting packages, with the resulting prints being generated on a plotter. CAD systems can interface with *FastCAM* if they produce a DXF file or IGES file. The drawing section of *FastCAM* can be thought of as a CAD system with the out-put being the coded geometry of the part.

CAM - Computer Aided Manufacturing. The output of *FastCAM* is the Numeric Code (NC data) used to drive the computer located at the shape cutting machine.

CARTESIAN COORDINATE SYSTEM - Refers to an X,Y coordinate system with the X axis lying on the horizontal and the Y axis running in the vertical plane. Using this X,Y coordinate system any point on a flat two dimensional part can be referenced.

CNC - Computer Numeric Control used to control flame cutting machines, milling machines, lathes and other computerized metal working equipment.

CONTROLLER - The computer located on the shape cutter and used to process the part program. Common controllers are the Linatrol, Autopath, Burny, Compupath, Linde, and Allen Bradley.

CONTROL POINT - Found in the *FastCAM*. drawing program. Represented on the graphics screen by a red plus sign. Control points are created when entities are constructed, and deleted when the entity is erased or trimmed. Lines have two control points, one on each end, arcs have a minimum of three, start, stop, center, and circles have five, four quadrants and the center. Control points can be used as construction or reference points.

COPY - To duplicate or make another image of the same geometry, file or object. When a copy is produced the original is preserved.

CPU - Central Processing Unit is the heart of the computer. The CPU consists of the hard drive for information storage, a floppy drive for information transfer, processors and connectors to send information to peripheral devices.

CROSSHAIR - Represented on the graphics screen usually as a small white crosshair. The crosshair is moved by using the mouse.

CUTTING DIRECTION - It is important to use the correct cutting direction when programming a part using kerf. If the cut direction is wrong the part will be too small by the size of the kerf.

CUTTING PROCESS - When used in the *FastCAM* manual it refers to the means by which the material is being cut. The most common cutting processes are oxy/fuel, plasma, routing.

DIAMETER - Refers to the distance across a circle through the center. Diameters can be entered into the *FastCAM* system by following the arc dimension input with a (d). This will indicate that the input is a diameter instead of a radius.

DIGITIZE - Coding information about the part into the computer by use of a digitizing puck and board. A point is located on the board and the PC records the coordinates. *FastCAM* does not support a digitizer. FMS has a digitizing system called FastCOPY.

DNC Link - Cable link between the PC and the shape cutter controller. The cable is normally fiber optics or wire cable.

DIRECTORY - Found in the WINDOWS operating system and contains stored part files.
Directories will normally be set up for different jobs, customers, plate thickness etc.

DOS PROMPT - DOS prompts appear at the top left corner of the text screen when the PC is in the DOS mode. They look like: C:\ for the C drive and A:\ for the A or floppy drive.

DOWNLOAD - The process of transferring information from the PC to the shape cutter controller. This can be accomplished either through a DNC link or by using a mini file.

DWELL - Defined in most controllers as a continuation of the current active process for the duration specified in the dwell code. A common use for the dwell is during the center punch cycle, allowing the controller to process the tool on code and provide enough firing time to produce the required punch mark. Many controllers use the form of G04FX where the X stands for the length of dwell in seconds. For specific information concerning your controller see your controller's programming manual.

DXF FILES - Files produced by AutoCAD and other CAD systems that are used to transfer geometric information into *FastCAM*. The DXF interface allows the geometry of the parts to be created and drawn on the CAD system, and the information to be transferred to *FastCAM* geometry for verification, modification, production of Word Address code, and nesting.

WORD ADDRESS PROGRAMMING LANGUAGE - The standard two dimensional shape cutter programming language. It consists of G codes, M codes,

kerf and tool on codes, and X,Y movements.

ENTITY - When used in the *FastCAM* manual it refers to the most basic drawing units. These are points, lines and arcs.

ERASE - Used to remove a complete entity from the screen. If a portion of an entity is to be removed, then the trim function should be used. Erase is found in *FastCAM* under the main menu item ERASE.

ESSI PROGRAMMING LANGUAGE - The standard two-dimensional shape cutter programming language in Europe. This manual was written for North America and does not cover ESSI programming.

ELLIPSE - An oval. Found in *FastCAM* under the main header CURVES. To use ellipse the major or long axis and the minor or short axis dimensions must be known.

FAIRED LINE - A line made of arcs passing through a series of points. Used to create a smooth surface to reduce drag from air or water.

FASTNEST - FastNEST is the block nesting program found in FastPLOT.exe. It is designed to rapidly place a large number of parts on the plate. The parts are nested by placing a block around the perimeter of the part, and nesting the blocks.

FASTPL97.EXE - The nesting and verification section of the *FastCAM* drawing system. Found in the FastCAM directory.

FEED RATE - The cutting speed of the shape cutter. This is normally added at the time the part is cut to provide for flexibility when cutting the same part out of different materials, but can also be placed into the part program using the text editor. See your controller's programming manual for the proper codes.

FILENAME - Limited to 255 alphanumeric characters. Used in FastCAM under FILE SAVE, PROGRAM PATH, BLOCKING, and in FastPLOT under NEST and FASTNEST.

FILES - Contain either a source program like FASTCM97.EXE or a data file for

program support or finished parts. As a *FastCAM* programmer you will be using files to store finished parts, parts in construction, and nests.

FILLET BLEND - Found in *FastCAM* and normally used to smooth corners and smooth connections between entities. Fillet blends can be used between lines and lines, lines and arcs, points and lines, points and arcs, arcs and arcs, arcs and points, etc.

FINITE LINE - Construction functions found in *FastCAM* under LINE menu. Finite lines are lines that have two definable end points. This is in comparison to infinite lines which have no end points.

FLOPPY DISK - Storage medium used to transfer information between the PC and the shape cutter controller. They are also used for long term storage of information. Floppy disks come in high density (1.4 mbyte) and low density (720 K). It is important to match the density of the disk purchased to the hardware being used.

FUNCTION KEY - Function keys are the top row of the key board, and are numbered from one to twelve. The only time function keys are used in the *FastCAM* drawing program is in the *FastPLOT* program with the text editor.

G CODES - Found in the Word Address coded output. G codes describe general functions of the program. These include kerf direction, line description, and other operation functions.

INCREMENTAL PROGRAMMING - The method of programming parts in which the next point placed on the screen is referenced from the last or previous point placed on the screen.

INFINITE LINE - A line with no endpoints, continuing to infinity in both directions.

KERF - The amount of material destroyed in the cutting process. As the metal is cut a predictable amount of material is destroyed which is called the kerf. *FastCAM*, by default sets only the direction of kerf and leaves the numeric value to be entered at the controller. This allows for flexibility, for the operator at the

NC controlled burn table. Kerf codes are as follows: Kerf left - G41, Kerf right - G42, Kerf off - G40. Alternatively the kerf can be added to a part in FastCAM or with FastCAM's optional Kerf program.

LEAD IN - Used to move the pierce point away from the part outline. When a pierce is done, the resulting hole is larger than the cut line. If the pierce was placed on the edge of the cut line, the resulting part will be flawed. Lead ins can be found in *FastCAM* under PROGRAM PATH.

LEAD OUT - Lead outs are used to cut past the end of the part so the part will fall loose. Lead outs may not be applicable on all parts. Lead outs can be found in *FastCAM* under PROGRAM PATH.

LINE - Line construction is found in *FastCAM* under the main menu header LINE.

M CODES - Word Address language codes used to activate controller specific functions. Some of these functions include tool on, tool off, and other various codes.

MACRO PROGRAM - Macro programs are subroutines used to store repetitive code, and can be used at a function call from the main program.

MDI PROGRAMMING - Manual Data Input. Creation of a part program without the use of a computerized programming system. All calculations and data input is done manually.

MENU - A list of commands which lead to sub-menus used in part construction.

MOUSE - *FastCAM* uses a mouse to control the programs

MOUSE CURSOR - Represented on the graphics screen usually as a small white arrow. The mouse cursor is moved by using the mouse. It is generally used to access menu items and to click on buttons.

NEST - When multiple parts are placed onto a plate and arranged to minimize waste, by using the nesting functions provided by FastPLOT.

NUMERIC CODE (NC CODE) - WORD ADDRESS or ESSI code used to drive the shape cutter controller.

OUT-PUT DEVICE - Refers to either the graphics screen which is the default for *FastCAM* or the plotter/printer.

OXY FUEL - The flame cutting process in which oxygen and another fuel is used for the cutting process. The other fuels include acetyline, propane, natural gas, etc.

PARALLEL PORT - Primarily a printer port, a parallel port comes standard on almost all PC's. A data outlet used to communicate between the PC and the printer or plotter. A parallel interface transmits data one byte at a time, with all eight bits of the byte transmitted to the receiving device (printer or plotter) at the same time, moving through eight separate wires, in parallel.

PIERCE - The point at which the cut is started. The pierce is usually done at the start of a lead-in to make a cleaner start corner.

PICK POINT - Found in *FastCAM* drawing program and is represented on the graphics screen as a red plus sign. Pick points show the position an entity is selected. Do not confuse pick points and control points.

PLASMA - The cutting process where inert gases and an electric arc are used to cut metal.

PLOT - When the out-put device is changed to the plotter/printer, a drawing of the part to be cut is produced. This is referred to as the plot.

PLOTTER - The out-put device used to produce a hard copy of the part to be cut. The plotter can be accessed either from *FastCAM* or *FastPLOT*. When a part is plotted from *FastCAM* the part will be drawn in the order the lines were placed on the screen. When a part is plotted from *FastPLOT* the part is drawn in the order the paths were set, or the order of cut.

POST PROCESSOR - Used in *FastCAM* to convert the raw geometry file to an WORD ADDRESS or ESSI format for a particular controller. The post processor

also checks to confirm that all codes in the program are compatible with the particular controller for which it was written. Each brand of controller will have its own post processor.

PRINTER - Another out-put device used to produce a hard copy of the part to be cut

PROGRAM - The numeric code produced by *FastCAM* used to drive the shape cutter controller.

PROGRAM PATH - Found in *FastCAM* and is used to set the direction of kerf, order and direction of cut line, placement of lead ins and lead outs, and defines the cutting process.

PROMPT - Prompts in *FastCAM* appear in the middle and at the bottom of the screen.

QUADRANT - An arc of 90 degrees and one quarter of a circle. In *FastCAM* each quadrant of a circle will be indicated by a red control point, and can be used for referencing other points. An arc will contain a quadrant point if the arc passes the point. When trimming parts, do not confuse the quadrant points with the intended trim points.

RADIUS - One half a diameter. The measurement of a circle from the center to the circumference. All *FastCAM* arc and circle inputs are in radius measurements unless the numeric input is followed by a (d) indicating a diameter input.

RAPID TRAVERSE - The shape cutting machine's movement from the end of one cut to the pierce of the next cut. When a rapid traverse move is executed on most controllers, the cutting torches raise to provide clearance. The torches also move in a straight line, at full speed, to the next pierce point.

REAL POINT - Found in the *FastCAM* drawing program. It is represented on the graphics screen by a yellow plus sign. Real points must be created using the point menu and deleted by using the erase menu.

REFLECT - A blocking function used to create a mirrored portion of an object,

or to make a left and right part. Found in *FastCAM* under the main menu header **BLOCK**.

SCREEN POSITION - Found in *FastCAM* POINT MENU. Screen position is a method of placing a point on the screen without regard to the numeric accuracy of the point.

SERIAL PORT- Formally called Asynchronous Communications Adapter. The serial interface is a full two-way communication path between the PC and the device. The serial interface is based on a standard RS-232, which is a two-way transmitting of asynchronous serial data. The path is two-way so that data may be transmitted back and forth. Asynchronous means that the transmission of data is not based on a predefined timing pattern. The serial interface is designed for a wide variety of uses. It is very flexible, however it also has a confusingly wide range of parameters and variations that can be adjusted to suit many needs. If you have a DNC link on your system you will be using the RS-232 cable connected to the serial port on the back of your PC.

SHAPE CUTTER - Cutting machine used to burn or cut parts in a two dimensional plane.

SWEEP ANGLE - Refers to the number of degrees in the angle formed from the start of an arc to the end of an arc. To calculate sweep angle subtract the smaller angle from the larger angle.

TANGENT - The point at which a line blends to an arc or circle. Tangents are found in *FastCAM* under the main menu header **LINES**.

TRIM - Found in *FastCAM* under **TRIM** and is used to remove part of an entity. If the entire entity is to be removed, use **ERASE**.

TWO DIMENSION - *FastCAM* is a two dimensional programming system. It uses an X,Y coordinate system to reference. This is in contrast to three or more axis systems used to program mills and laths.

UPLOAD - To transfer information, part programs, or files from the controller on the shape cutter to the computer.

ZINC MARKING - Plate marking system in which melted zinc is used for permanent marking.

Index

*
 *. * 193, 194
 *.CAM 192
 *.EXP 194
 *.NC 194
 *.PRT 193
 *.PTH 193
 *.SRT 194
 *.TMP 193
 *.TXT 193

A

Absolute CO-ords 98
 ABSOLUTE Programming....35, 36,
 204
 Absolute zero42, 59
 ANGLE MENU90
 ANGLES35
 ARC61, 93
 ARCS61
 ARRAY111
 at Rt Angles92
 Auto Dimension.....125
 Autoscale123
 AUTOSCALE.....223
 Axes125

B

BLOCKING.....111
 Box.....92
 BOX.....222

Break 131
 BREAK 143
 Bridging 103

C

CAD Clean.....108
 CAD Compress108
 CAD LAYERS135
 CARTESIAN CO-ORDINATE.....34
 Chamfer102
 Change Display.....124
 Check Dimensions139
 Circ Array117
 CIRCLE61
 CONSTRUCTS101
 Contour107, 138
 Control131
 CONTROL207
 Control Point.....99
 CONTROL POINTS52, 139, 141,
 223
 Control.con196
 CONTROL.DAT136, 198, 202
 CONVERT187
 COPY.....111
 CORNERS101
 COST184
 CROSSHAIR.....29

D

Direct entry90
 Direction125

Distance	138
DSTV Restore	84
Duplicates only.....	107
DXF Layers.....	125
DXF Restore.....	81
DXF Save	83

E

EDIT	184
Edit Entity	132
Ellipse.....	100
Entities.....	131
Entries & Exits	109
ERASE.....	57, 68
Exit	85
Expand	116
EXPAND NC	187
Explode.....	119
Extract part	119
EXTRACT PART	112

F

Faired Line	100
Fastcm99.exe	196
FASTCM99.EXE	192
FastCOPY.....	252
FastCUT	252
FastLINK.....	253
FastLOC	26
FastNEST.....	77, 251
FastPATH.....	150, 251
Fastpl99.exe	196
FastPLOT.....	149
FASTPLOT	183
FASTPLOT.EXE	192
FastSHAPES	251
FastTRACK	251
FastTRAN	253

Feedrate	71, 143
Feeds.dat.....	197
FILES	80
Fillet	101
FILLET BLENDS	62
Fillet-Blend.....	96
Full circle.....	93
FULL CIRCLE.....	222

G

G code	200
G codes	198
G CODES	204
gap	146
Gap	103

H

HPGL Restore	83
--------------------	----

I

Identify	138
IGES Save	83
INCREMENTAL.....	35, 42
INCREMENTAL Co-ordinates.....	44
42	
Incremental CO-ords.....	98
INCREMENTAL PROGRAMMING	204
Indicate Corners	121
INDICATE CORNERS.....	223
INFINITE LINE.....	69
Infinite lines.....	60
Interactive Nesting	251
INTERACTIVE Nesting	77
Intersection.....	99
INTERSECTION	54

K

Kerf	143, 199, 200
KERF	71, 145
KERF LEFT.....	202
KERF OFF.....	202
KERF RIGHT.....	202
KeyPad.....	52

L

LEADINS	70
LEADOUTS	70
Length & Angle	87
LICENSE.....	189
LINATROL	202, 203, 204
LINE	60, 86
LIST.....	186
Loop	102
LYNX.CON.....	196

M

M codes.....	198
M CODES	203
Marker.....	200
MARKER	202, 203
Material.dat:.....	197
Midpoint	98
MOUSE	29
MOVE.....	111

N

NC CODE	144
NC program.....	71
NC PROGRAM PATH.....	143
NEST	207
NESTING	77
New	81
NEW	222

Next Path	144
Next point	87
Notch.....	103
NOTCH.....	222

O

Offset Contour	103
On Entity.....	99
OPEN	222
Output NC code	147
Overcut bridge	104
Oxy	200

P

Parallel at dist	91
Part Data	85
Pierce & Stop	126
Plasma	199
PLASMA	202, 203
Plot.....	84
PLOT	184, 222
Plotter Label.....	127
POINT MENU	97
POINTS	51
PROGRAM PATH	69, 140

R

Rapid	126, 200
Read Block.....	118
REAL POINTS	139, 222
Rect Array.....	117
REFLECT	111
Relative to	90
Repeat	123
RESTART.....	41
Restore	66, 81
RESTORE.....	183
ROTATE.....	111

S

Same as.....	90
Save	63, 80
SAVE	222
Scale only	129
SCAN	183
SCREEN DISPLAY	28
SCREEN ORIENTATION	33
SECURITY KEY	26
Setup	131
Setup.dat	197
SETUP.DAT	206
Simple bridge	104
SORT NC	188
STRETCH	223
StruCAD Restore.....	84

T

Tan to 2 Circles	91
TEXT.....	223
TEXT MARKING.....	133
Toolbar	125

TOOLBAR.....	222
TORCHES	206
TRIM	139, 222
TRIMMING	69
Two points	88
Two sides.....	88, 90

U

Units	130
UNITS	207
UTILITY	143

V

VERIFY	138
VIEW	67

W

WORD ADDRESS	198
Write Block	118

Z

Zoom In.....	122. <i>See</i> Indicate Corners
Zoom Out	123

For product support contact your Reseller or Offices below
(Refer to the inside of the Manual for contact numbers)



The World Standard in Profiling Software

World Wide Web <http://www.fastcamusa.com>

FastCAM
700 West Bryn Mawr,
Suite 800 South
Chicago 60631-3507 USA
Tel: (312) 715 1535
Fax: (312) 715 1536
Email: fastcam@fastcamusa.com

FastCAM
96 Canterbury Rd
Middle Park, Victoria 3206
Australia
Tel: (61) 3 9699 9899
Fax: (61) 3 9699 7501
Email: fastcam@fastcam.com.au

FastCAM
Level 21, Shanghai Senmao, Int Bldg
101 Yin Cheng East Rd
Pudong Shanghai 200120 China
Tel: (8621) 2890 3127
Fax: (8621) 2890 3131
Email: fastcam@fastcam.com.cn
