Computer Aided Design and Manufacturing Software

Table of Contents

1.0	Getti Introd	ing Started luction	3
2.0	Men	u Descriptions	
	2.1	File Menu	7
	2.2	Switch Menu	
	2.3	Edit Menu	
	2.4	Draw Menu	15
	2.5	View Menu	17
	2.6	Options Menu	
	2.7	CAM Menu	
	2.8	Help Menu	
3.0	Conf	figuration and Setup	
	Milling	<u>g Example</u>	
	3.1	Drawing Title Block	
	3.2	G-code Control Setup	
	3.3	Setting Tool Definitions	
	3.4	Saving Parameters to a Setup File	

Routing Example

3.5	Drawing Title Block	31
3.6	G-code Control Setup	31
3.7	Setting Tool Definitions	32
3.8	Saving Parameters to a Setup File	32

4.0 Tutorials

4.1	Milling Example Tutorial	33
4.2	Routing Example Tutorial	39

5.0 Appendix

5.1	Keyboard Scan Codes	
J. I		

Computer Aided Design and Manufacturing Software

Getting Started With MultiCAM Mill

Thank you for purchasing the MultiCAM Mill Computer Aided Manufacturing software. This manual contains a command reference section and a tutorial section. Please read both thoroughly before using the software. Later you may refer to the reference section as needed.

Minimum System Requirements

An IBM PC XT/AT or 100% compatible MS-DOS or PC-DOS version 3.1 or later. An EGA or VGA monitor and display adaptor. A Microsoft compatible mouse.

Installing MultiCAM Mill on a Hard Disk

With the computer turned on and at the C: > prompt, do the following:

- 1. Put the MultiCAM Mill diskette in drive A:
- 2. Type:

A:

3. At the A:\> prompt, type: INSTALL

Getting Started With MultiCAM Mill

Installing MultiCAM Mill on a Floppy Disk.

While a hard drive is recommended; it is possible to operate MultiCAM Mill from a floppy disk. You will need a high density disk drive; a 1.2Meg 5.25" or a 1.44Meg 3.5" will do.

To prepare a bootable disk perform the following:

1. Format a blank 1.2 Meg or 1.44 Meg disk using the /S option as follows:

```
Format a: /s
```

- 2. Copy the COMMAND.COM and the MOUSE.COM files from your dos diskette to this new disk.
- 3. Copy all files from the supplied disk(s) to the new disk.
- Insert the new disk and reboot by pressing the <CTRL><ALT><DELETE> keys simultaneously. You Should see the design screen illustrated in *Figure 1.1.1*. Study the items shown on the screen.

Installing The Security Key

Install the security key (see Figure 1.1.1) on a parallel port on the computer. It can be attached to either LPT1, LPT2, or LPT3. Tighten the screws using a small screwdriver.

Starting MultiCAM Mill

**Important: Multicam Mill will not run properly in a DOS shell under Windows. Exit Windows before running. ** At the DOS prompt (C:\>), type the following:

CD\MCAM *<ENTER>*

MOUSE <ENTER>

MCAM <ENTER>

This will start the MultiCAM Mill software.



Figure 1.1.1 Security Key

Computer Aided Design and Manufacturing Software

Pull-Down Menu Command Reference

Figure 1.1.2

MultiCAM Mill Design Screen

Visual Grid Lines

(0,0) Origin Reference Marker and Tool Start Position

Start Point Marker

Observe the top line of the screen in *Figure 1.1.2.* Each word is a heading for a pull-down menu. The pull-down menu will appear when the heading is selected by the mouse or keyboard.



Tool Paths in design

To access a pull-down menu, use the mouse to position the arrow on top of a pull-down menu heading and click the left mouse button. Alternatively, you can use the keyboard to access a pull-down menu by pressing <ALT> and the first letter of a heading.

Once the menu is in view, you can select any of its options by positioning the arrow on the option and clicking the mouse button.

Computer Aided Design and Manufacturing Software

File Menu

File Switch	Edit Dra
New Load Revert to s Import DXF Save Save As	Ctrl+L saved File Ctrl+S Ctrl+A
Delete file	e
Print Pictu Save Pictuu Diew Pictuu DOS S <mark>h</mark> ell	ure F10 re F8 re
Exit	

Figure 2.1

Selecting the File Menu produces the list of options shown in *Figure 2.1.*. Some of the selections have *Hot Keys* associated with them. Hot keys allow access to a command quickly through the keyboard and they are indicated to the right of the selection. For example, the Hot key used to load a drawing is <CTRL>-L which means holding down the Ctrl key and pressing L simultaneously.

The letters in reverse or other color than the rest of the option text are called *QuickSelect* keys. These allow access to a command only while the pull-down menu is displayed.

File-New

Clears the design in memory and establishes the defaults for a new drawing.

File-Load

Displays the dialog window shown in *Figure 2.1.1.* which displays all designs in the current directory. Just *double click* on the file name to load it.

File Switch Edit Draw File Name: F.CDM	View Options CAM Drawing:(Untitled) Drawing File Selector	Help	
G :\MCAH\	iles	Dirs/Drives	
naze.cdn star naze2.cdn nk.cdn pyranid.cdn pyranid2.cdn snooth4.cdn	r5.cdn	 [-A-] [-B-] [-C-] [-G-]	
	Ok Cancel		
Entities: 0 Tool Paths: 0	Zoon=X 1 X -0.	3861 ¥ 1.9275 Z	-0.1580

Figure 2.1.1 Load File Dialog Window

File-Revert to Saved

Reloads the last saved version of the current drawing.

ile Switch Edit Draw
New Load Ctrl+L Revert to saved Import DXF File Save Ctrl+S Save Ss Ctrl+A
Delete file
Print Picture F10 Save Picture F8 View Picture DOS S <mark>h</mark> ell
Exit

Figure 2.1

File-Import DXF File

Reads in a DXF "drawing exchange file" created with AutoCAD[®] or other drafting program and coverts it to MultiCAMTM format in memory. A dialog window will appear on the screen (see *Figure 2.1.2*) allowing you to select the desired DXF file using the mouse or keyboard. Refer to the *Edit-Move Profile* command to align the profile if necessary. Importing a DXF file allows you to use drawings originally created with other CAD software without having to draw it from scratch. Lines or arcs that are not related to the profile of the part design such as autodimensioning lines may need to be deleted after importing the drawing.

File Sei	itch Edit Draw U File Hane:	iew Options rawing:«Untitle DXF File Sele	CAM Help ed>		
	BINKCARN FILL	es		Pirs/Prives	
	No files!				
	C	Ok C	ancel		

Figure 2.1.2 Import DXF File Window

File-Save

Quickly saves the current design to disk. If the design is a new one this command allows you to enter the name of the design. The Extension .CDM is automatically assigned.



File	Switch	Edit	Draw
Ne La Re Im Sa	w ad evert to s port DXF eve ave	Ctrl aved File Ctrl Ctrl	l+L l+S l+A
De	lete file		
Pr Sa Vi DO	int Pictu ve Pic <mark>t</mark> ur ew Pictur S S <mark>h</mark> ell	re I e e	710 F8
Ex	it		

Figure 2.1

File-Save as...

Allows saving a design to a new file name or disk/directory. Displays the dialog in *Figure 2.1.3* to select the disk/ directory. The Extension .CDM is automatically assigned.



Figure 2.1.3 Save As Dialog Window

File-Delete

Deletes an existing part program from the disk. The **CAD File Selector** dialog window appears and allows the user to choose a part program in the current directory. Use the **Dir/Drives** box to change to other drives and directories. Choose the OK box when the correct filename appears in the File name box. *See Figure 2.1.1*

File-Save Picture

Saves the current graphics screen to a CAMPICnn.PCX file where nn is an integer from 01 to 99 and is incremented by 1 each time this option is used. The file is saved to the current directory.

File-Print Picture

Prints the current graphics screen on the printer specified in the PRINTER OPTIONS dialog window.

File Menu

ile Switch Edit Dra	ω
New Load Ctrl+L Revert to saved Import DXF File Save Ctrl+S Save 1 s Ctrl+A	
Delete file	
Print Picture F10 Save Pic <mark>t</mark> ure F8 Diew Picture DOS S <mark>h</mark> ell	
Exit	

Figure 2.1

File-View Picture

Displays a previously saved picture on the screen. The PCX File Selector dialog window allows the user to choose the picture to be viewed from the current directory. Use the **Dir/ Drives** box to change to other drives and directories. Choose the OK button when the correct filename appears in the File name box. See **Figure 2.1.4.**

File Switch Edit Draw View Options CAM Drawing:(Untitled) PCX File Selector File Name: .PCX	Help
G:\NCAN\ Files	Dirs/Drives
campic01.pcx campic07.pcx campic02.pcx campic00.pcx campic03.pcx campic09.pcx campic04.pcx campic10.pcx campic05.pcx campic11.pcx campic06.pcx campic12.pcx	 [-A-] [-B-] [-C-] [-C-]
0k Cancel	
Entities: U Iool Paths: U Zoow=X 1 X -U.	.3264 Y 0.8771 Z -0.1580

Figure 2.1.4 PCX File Selector Window

File-Dos Shell

Temporarily exits to DOS and allowing you to execute most DOS commands. Type EXIT at the DOS prompt to return to MultiCAM Mill.

File-Exit

Exits MultiCAM Mill software and returns control to DOS.

Computer Aided Design and Manufacturing Software

Switch Menu

Switch	Edit	Draw	٧i
to M	K Edito	or	<u>ה</u> ב
to M	illMast	ter Pro	

Figure 2.2

The Switch Menu shown in *Figure 2.2* provides quick access to Manufacturing and CNC file editing software.

Switch-To MK Editor

Exits the MultiCAM Mill program and initiates the MK editor. This is a full text editor used to create and edit part programs. (Refer to MK Editor Manual for more information).

Switch-To MillMaster Pro ™

Exits the MultiCAM Mill program and initiates the MillMaster Pro program. This is the full function G-code Machining and simulation software. (Refer to the MillMaster Pro manual for more information).

Edit Menu

dit	Draw	View	Oj
Und Del Cop Pas	o ete y te	Esc Del	
Fli Rev Joi Mov Ere Cha Tri Ent	p Arc erse [n Endpo e End-] ak segn nge Rac e Prof m Entit er Pos: ity Int	F Dints J Points V Ment B Hius ile M ty T ition E fo I	

Figure 2.3

The Edit Menu shown in *Figure 2.3* provides access to functions that alter or modify a design. All of theses selections have *Hot Keys* associated with them. Hot keys allow access to a command quickly through the keyboard and they are indicated to the right of the selection. For example, the Hot key used to undo a line just drawn is the Escape or <Esc> key.

The letters in reverse or other color than the rest of the option text are called *QuickSelect* keys. These allow access to a command while the pull-down menu is displayed.

Edit-Undo

Erases the last entity drawn or reverses the last function performed. If the last operation was an undo then selecting undo again will return the drawing to the status it was in before the first undo.

Edit-Delete

Erases a drawing entity from the screen. Select this menu option then position the mouse pointer near the entity and click.

Edit-Copy

Copies a profile into the Paste Buffer.

- 1. Select this command then position the *START MARKER* to the beginning of the segment and click the right mouse button or press the *</NSERT>* key on the keyboard.
- 2. Point to the segment desired and click the right mouse button.

Edit-Paste

Copies the *Paste Buffer* to the current *START MARKER*. The Start Marker automatically advances to the next copy position for quick repeat paste commands.

Edit-Flip Arc

Quickly changes between concave and convex arcs. Select this command then position the arrow next to the arc you want to change and click the left mouse button.

MultiCAM Mill

Edit Menu

Edit Draw	View	0
Undo	Esc	
Delete Copy	Del	
Paste		
Flip Arc	_ F	
Reverse	Dir D	
Move End-	Point V	
Break seg	ment B	
Change <u>R</u> a	adius File M	
Trim Enti	ity T	
Enter Pos	sition E	
Enercy II		

Figure 2.3

Edit-Reverse Arc Dir

Quickly changes between clockwise and counterclockwise arcs. Select this command then position the arrow next to the arc you want to change and click the left mouse button.

Edit-Join End Points

Connects two end points of lines or arcs with a straight line. Select this command then position the arrow pointer between the end points you wish to join and click the left mouse button.

Edit-Move End-Point

Allows the user to relocate one or more end-points in the profile to another location thereby stretching, adjusting the taper angle or eliminating a segment.

Edit-Break Segment

Splits one segment in two at the location you specify maintaining the same radius or taper angle for both new segments.

Edit-Change Radius

Allows the user to specify the exact radius for an arc. Select this command then position the arrow next to the arc you want to change and click the left mouse button.

Edit-Move Profile

Allows repositioning the complete design relative to the origin. Position the *START MARKER* at the desired destination point then position the arrow on a point on the design profile you would like to be positioned at the *START MARKER* then click the left mouse button.

Edit-Trim Entity

Allows the user to trim away overlapping entities to make a clean, continuous intersection.

- 1. Choose the section of the entity you want to trim off.
- 2. Select the entity that you want to trim to by positioning the arrow over that entity and pressing the left mouse button.

Edit Menu

Edit	Draw	View	0)
Und	lo	Esc	
Del	.ete	Del	
Cop Pas	y te		
Fli	p Arc	F	
Rev	erse i	Dir D	
Joi	n Endpo	oints J	
Mov	e End-l	Point V	
Bre	ak segr	ment B	
Cha	nge <mark>R</mark> ad	lius	
ijov Tri Ent	e Prof: m Entition	ile M ty T ition F	
Ent	ity In:	fo I	

Figure 2.3

Edit-Enter Position

Allows the user to position the cursor at an exact position by entering the coordinates on the keyboard. Select this command and type the **X** and **Y** coordinates in the dialog window of the location. Press the *<ENTER>* key **NOT** the mouse button. The Arrow will be positioned precisely at the desired coordinates. You should use the keyboard <enter> key to finish a segment you are drawing or *<INSERT>* key to reposition the start marker.

Edit-Entity Info

Displays the numeric information of a segment.

Document Revision B5

Figure 2.4.1

Placements

MicroKinetics Corporation

Computer Aided Design and Manufacturing Software

MultiCAM Mill

Draw Menu

Draw	View	Opti
√ Lin Rec Arc Cin	ne stangle s scle	L R A C
Se I Uns Uns	lect select select A	11



The Draw Menu is used to select the drawing mode using a line, rectangle, arc, or circle. Some of theses selections have *Hot Keys* associated with them. Hot keys allow access to a command quickly through the keyboard and they are indicated to the right of the selection. For example, the Hot key used to select line mode is the <L> key.

Draw-Line

Selects the straight line pen mode for drawing vertical, horizontal, and tapered lines at any angle. Note the selection *check mark* on the left of the word, Line, in *Figure 2.4*. This mark indicates that the line mode is currently selected. To draw a line, just reposition the *START MARKER*, if necessary, position the arrow (end marker) at the end of the line, and click the left mouse button.

Draw-Rectangle

Selects the Rectangle Drawing mode. To draw a rectangle, simply place the *START MARKER* at one corner of the rectangle and position the arrow (end marker) at the diagonal corner and click the left mouse button.

Draw-Arc

Selects the Arc Drawing mode. For drawing concave and convex arcs, all you need to do is indicate the start point, the end point and a point on the arc. See *Figure 2.4.1.*

- 1. Position the start point (P1) and press the right mouse button to relocate the *START MARKER*.
- 2. Position the arrow to the end point (P2) and click the left mouse button.
- 3. Position the arrow to a point (P3) that the arc passes through and click.

Selects the Draw Circle mode. To draw a circle, simply place the *START MARKER* at one point (P1) on the circle and position the arrow and click the left mouse button at each of two other points on the circle (P2 and P3). See *Figure 2.4.1.*

Circle and Arc Point Draw-Circle



Computer Aided Design and Manufacturing Software

Draw Menu

Draw	View	Opti
√ Li Re Ar Ci	ne ctangle c rcle	L R A C
Se Un Un	lect select select	A11

Figure 2.4

Draw-Select

This allows the user to select entities on the screen. Drawing entities must be selected prior to using the **CAM-Verify/ Accept Pocket** and **CAM-Verify/Accept Island** functions.

Draw-Unselect

This allows the user to unselect an entity that is currently selected.

Draw-Unselect All

This unselects ALL selected entities.

2

MultiCAM Mill

Computer Aided Design and Manufacturing Software

View Menu

View Opti	ons	CAM	Help
Redraw		Ctr	1+R
Genter Zoom In Zoom Jut Full Vie	: W	Ctr Ctr Ctr Ctr	+1+C +1+1 +1+0 +1+F
√ Show Par Show ⊡nd Show too √ ⊡ool Rad	t Tem Mark 1 Pati lius o	plate ers hs utline	•
Toggle (olor/	B₩	F9



The *View Menu* allows you to change the way you see the design. See *Figure 2.5.*

View-Redraw

Regenerates the graphics at the same zoom level.

View-Center

Centers the display around the START MARKER.

View-Zoom In

Magnifies the area around the START MARKER.

View-Zoom Out

Condenses the area around the START MARKER.

View-Full view

Zooms in or out as necessary and centers the part on the screen.

View-Show Part Template

Enables/disables the part template under the design.

View-Show End Markers

Enables/disables the display of segment end markers, this visually indicates the beginning and end points of all segments.

View-Show Tool Paths

Enables/disables the display of the tool path entities computed by MultiCAM Mill.

View-Tool Radius Outline

Enables/disables the display of tool radius lines which provide a realistic view of the tool path.

View-Toggle Color/BW

Toggles between Color and Black & White. For best results, change to B&W just before a screen dump.

Options Menu

Dptions CAM Help
Drawing Title Block G-code <mark>C</mark> ontrol Setup
Grid Selections Printer Options
Save As Default Revert to Saved Defaults Load Setup Save Setup Save Setup
Drawing Area <mark>C</mark> olors Menu & <mark>N</mark> indow Colors



The Options menu is used for configuring the system, setting up Metric or Inch programming, specifying printer type and port selection and setting up the G-code Post-processor. The Options menu also allows changing the colors for all the individual functional areas of the system. See *Figure 2.6.*

Options-Drawing Title Block

Allows you to enter the Drawing Title, Designer Name, Revision, Date, Material used, Part Length, Width, and Height, and to specify mm or Inch units of measure. See *Figure 2.6.1.*

Sui	tch Edit Draw View Options CAM Help Drawing: <untitled></untitled>	
	Drawing Title Block	1
	Drawing Title: Sample	
	Designer Hame: John Doe	
	Units Part Length: 2.0000 Width: 2.0000	
	* Inch (X) (Y) nn Part Height: 0.5000 Accept	

Figure 2.6.1 Drawing Title Block Window

Options-G-code Control Setup

Allows the user to enter the tool start and change position, specify the appropriate work hold method, and the path the CNC file will be directed to when compiled by MultiCAM Mill.

See *Figure 2.6.2.*

Computer Aided Design and Manufacturing Software



Options CAM Help
Drawing Title Block G-code <mark>C</mark> ontrol Setup
Frid Selections Printer Options
Save As Default Revert to Saved Defaults Load Setup Save Setup Save Setup
Drawing Area G olors Menu & <u>N</u> indow Colors

Figure 2.6



Figure 2.6.2 G-code Setup Dialog Box

Options-Grid Selections

Allows the user to select a Visual Grid size, a Cursor Grid size and to individually enable or disable each. Also, pressing <G> toggles both grid selections. See *Figure 2.6.3.*



Figure 2.6.3 Visual and Cursor Grid Setup

Options Menu

Jptions CHM Help
Drawing Title Block G-code C <mark>ontrol Setup</mark>
Grid Selections Printer Options
Save As Default Revert to Saved Defaults Load Setup Save Setup Save Setup
Drawing Area Solors Menu & Iindow Colors



Options-Printer Options

Allows selection of LPT1, LPT2, or LPT3 for the printer port and the type of printer (Epson/IBM) or LaserJet. See *Figure 2.6.4*.



Figure 2.6.4 CNC Printer Options

Options-Save As Default

Stores all the current settings to the default configuration file. All the options as set will be stored on disk and will be in effect upon first executing MultiCAM Mill or upon starting a new drawing by selecting **File-New**.

Options-Revert to Saved Defaults

Clears current settings and restores the defaults from disk.

Options-Load Setup

Allows the user to load a setup file. A setup file has a .MSU filename extension and configures MultiCAM Mill with the setup parameters in the file. This allows the user to quickly change between different setups.

See *Figure 2.6.5.*



Options CAM Help
Drawing Title Block G-code <mark>C</mark> ontrol Setup
Frid Selections Printer Options
Save As Default Revert to Saved Defaults Load Setup Save Setup Save Setup
Drawing Area G olors Menu & <u>N</u> indow Colors

File Switc	h Edit Draw U D ile Name: •.HSU	iew Options rawing∹Untitl Setup File Sel	CAN H ed>	slp		
G	r\NCRH\ Fil nill.nsu router.nsu	es		Dirs/ [-A-] [-B-] [-C-] [-G-]	Drives	
		0k C	ancel			
Entities: O	Tool Paths: 0	Zoon-X 1	X 1.21	169 Y	1.1676	z -0.1000

Figure 2.6.5 Setup File Select Window

Options-Save Setup

Saves the current setup parameters to the current setup file.

Options-Save As

Allows the user to save the current setup parameters into a setup file. See *Figure 2.6.6.*

File S#	Itch Edit Byau View Options CAN Help Drawing:(Untitled> Save Setup As File Name: THLENSU D:\MCAN\	
	[- ∩ -] [-⊮-] [-C-] [-0-]	
·····	Ok Cancel	
Entities:	0 Tool Paths: 0 Zoow=X 1 X 1.1112 Y 0.9888 Z -	N: -9 , 1000

Figure 2.6.6 Save Setup As Window

Figure 2.6

Options Menu

Drawing Title Block G-code <mark>C</mark> ontrol Setup
Grid Selections Printer Options
Save As Default Revert to Saved Defaults Load Setup Save Setup Save Setup
Drawing Area Colors Menu & Nindow Colors

Figure 2.6

Options-Drawing Area Colors

Allows selection of colors for the design area. This includes the part fill color, the background color, the draw pen color, the roughing cuts color etc.. See *Figure 2.6.7*.

Switch Ed	lit Draw View Options CAN Help Drawing: <untitled></untitled>	
	Background Color	
	Part Fill Color	
	Draw Pen Color	
	Entity Accept Color	
	Visual Grid Color	
	Origin Marker Color	
	Accept	

Figure 2.6.7 Drawing Color Setup Window

Options-Menu & Window Colors

Allows selection of Menu and Text colors for the pull-down menu's and the Dialog windows. See *Figure 2.6.8.*

File	Switch Edit	Braw View Options CAH Hel; Drawing:(Untitled) Hema Colors Setup	
		Yeau bar mornal Menu bar item selected Menu bar alt-key active Menu option enabled Menu option disabled Menu option Hot key	
		Henu bar normal	
		Recept Cancel	
Entiti	ies:0 Tool F	aths: D Zoon-X 1 X 1.258	5 Y 0.6834 Z 0.1588

Figure 2.6.8 Menu Colors Setup Window

Computer Aided Design and Manufacturing Software

CAM Menu

CAM Help	
Select Tool Ctrl+ Pocketing Specs Ctrl+ Verify/Accept Pocket Verify/Accept Island	r P
Rough Cut Pocket Ctrl+ Place Tinish Path Ctrl+ Add Tachine Cycle	K H
Indo Last Place Edit Path∕cycle Clear All Paths	
Make C <mark>N</mark> C File Ctrl+ Browse CNC File	N

Figure 2.7

The CAM menu contains functions to select the current tool, to enter pocketing specifications, to rough and finish cut pockets, and to create CNC program files. See *Figure 2.7*.

CAM-Select Tool

Allows the user to enter the diameter of each of six tools and to select the current tool to be used. See *Figure 2.7.1.*

File Switch	ch Edit Draw View Options CAM Help Drawing: <untitled></untitled>	
	Iurret Tools 1: Twist Drill 2: Spot Face Tool 3: Mill Cuttor 4: Ball-End Cutter 5: I-Slot Cutter 6: Dove-Tail Cutter	
	Edit # 3: Will Cutter Nool Path Color	
Entities: 0	Tool Paths: 0 Zoon=X 1 X 1.9958 Y 2.	0542 Z -0.1

Figure 2.7.1 Tool Select Dialog Window

CAM-Pocketing Specs

Allows the user to set the pocketing and profiling parameters used by the CAM-Rough Cut Pocket function. Enter the start and final pocket depths, specify the amount of material to be left for the finishing cut, enter the spacing between successive roughing cuts, and the maximum cut depth of the roughing cuts. Enter the feed rates the post-processor will use, and specify cutting direction. See **Figure 2.7.2.**

Select Tool	Ctrl+T
Pocketing	pecs Ctrl+P
Verify/Acce	pt Pocket
Verify/Acce	pt <mark>I</mark> sland
Rough Cut F	Ocket Ctrl+K
Place Finis	h Path Ctrl+H
Add Machine	Cycle
Undo Last P	'lace
Edit Path∕o	sycle
Clear All P	'aths
Make C <mark>N</mark> C Fi	le Ctrl+N
Browse CNC	File

Figure 2.7

Switch	Edit Draw View Options CAM Help	
	Drawing: <untitled></untitled>	
	Pocketing & Profiling Parameters	
	Final Depth: .2 Start Depth: O	
	Rough Clearance X&Y: .005 Z: 0	
	Rough Cut Spacing: .05	
•••••		
••••	Cut Depth Per Pass: .2	
•••••		·····
		·····
	Feed Rates: (Rough) 5 (Finish) 4	
	(Normal Feed Unly)	
	<pre></pre>	
	* (Bidirectional)	
	Fool Datho: 0 Zoow-Y 1 Y 1 E200 V 1	0722 7 -0 20

Figure 2.7.2 Pocketing Specs Dialog Window

CAM-Verify/Accept Pocket

This function uses the entities currently selected with the Draw-Select function to define the border of a pocket. This function checks the part profile for any errors during this operation.

CAM-Verify/Accept Island

This function uses the entities currently selected with the Draw-Select function to define the border of a island. This function checks the part profile for any errors during this operation.

CAM-Rough Cut Pocket

This function uses the previously accepted pocket and island borders and creates a tool path for the roughing cuts of the pocket.



0	AM Help	
	Select Tool Ctrl+T Pocketing Specs Ctrl+P Verify/Accept Pocket Verify/Accept Island	
	Rough Cut Pocket Ctrl+K Place Tinish Path Ctrl+H Add Tachine Cycle	
	Undo Last Place Edit Path/cycle Clear All Paths	
	Make CNC File Ctrl+N Browse CNC File	

Figure 2.7

CAM-Place Finish Path

Allows the user to place a finish cut path along any section of the part profile. Simply click the left mouse button near the start of the path then again in the direction of the path and finally on what side the tool is to cut on. If there is an intersection or an ambiguous entity path, then use the Draw-Select function to define the section of the part profile to be considered.

CAM-Add Machine Cycle

Allows the user to easily use a canned drilling or boring cycle in the current design. See *Figure 2.7.3*.

- Position the start point and press the right mouse button to relocate the *STARTMARKER* or choose <ALT>E-E and enter the exact position needed. Press the <INSERT> key to move the *START MARKER* to that position.
- 2. Choose <ALT>C-M and the Cycle Select and Definition Dialog Box with appear on the screen.
- 3. Choose the appropriate cycle and enter proper specifications. Choose [ACCEPT].

A cycle icon " I will appear at the *START MARKER* position indicating a canned cycle.

Cycle Select	t & Definition	1
Canned cycles 1: E81=D-111 Hole 2: G82=Spot Face	Final Depth: 0.5000	
3: G83=Deep Hole 4: G85=Boring Hole 5: G87=Deep Hole /Chip	Incr Depth: 0.1000	
 6: G89=Boring H/Dwell	Dwell (Sec): 1	
 Edit # 1: G81=Drill Hole	Feed Rate: 8	1
L	Accept Cancel	

Figure 2.7.3 Cycle Selection Dialog Window

CAM Menu

Select Tool	Ctrl+T
Pocketing Specs. Verify/Accept Po Verify/Accept Is	Ctrl+P cket land
Rough Cut Pocket Place Finish Pat Add Machine Cycl	Ctrl+K h Ctrl+H e
Undo Last Place Edit Path/cycle. Clear All Paths	
Make C <mark>N</mark> C File Browse CNC File.	Ctrl+N

Figure 2.7

CAM-Undo Last Place

Removes the last roughing or finishing cut path from the screen.

CAM-Edit Path/Cycle

Allows the user to edit a tool path or cycle.

See *Figure 2.7.4.*

File Switch	Edit Draw View Options CAN Help Drawing: <untitled></untitled>	
	View/Edit Path Segments	
	Tool Path #: Type: 1	
	Start Point> X: 1.0675 ¥: 0.7175	
	End Point> X: 1.5325 ¥: 0.7175	
	Center Point> X: 0.0000 ¥: 0.0000	
	Z Depth: -0.100 FR: 8 Tool: 103	
	Next Prev Ins Del Exit	•
Entities: 8 1	ool Paths: 8 Zoon=X 1 X 1.5669 Y	1.0037 Z -0.1000

Figure 2.7.4 Edit Path/Cycle Dialog Window

CAM-Clear All Paths

Removes all the paths currently on the screen.

CAM-Make CNC File

Generates the G-code file for use with the Mill Master Pro CNC Machining and Simulation software.



Cam	Help	
	<mark>Select Tool</mark> Pocketing <mark>S</mark> pecs Verify/Accept Pock Verify/Accept <mark>I</mark> sla	Ctrl+T Ctrl+P et nd
	Rough Cut Pocket Place Rinish Path Add Nachine Cycle.	Ctrl+K Ctrl+H
	Undo Last Place Edit Path/cycle Clear All Paths	
	Make C <mark>N</mark> C File Browse CNC File	Ctr1+N

Figure 2.7

CAM-Browse CNC File

Allows you to view a CNC file on screen. This provides a quick look at the code generated without having to exit the program. See *Figure 2.7.4.*

File Swit	tch Edit Drau File Name: *.Ch G:\MCAM\	View Op Drawing:<1 CNC Filo	tions CAM Untitled> e Selector =	He	lp				
	I	Files			Dirs.	/Drives	╷║		
	acctest.cnc arcs.cnc char.cnc g85test.cnc m inmsub1.cnc m inmtest.cnc m	entest.cnc (letter0.cnc (ltrdemo.cnc) haze.cnc) hemsub1.cnc s hemtest.cnc s	mill6.cnc outtest.cnc pyramid.cnc pyramid2.cnc smooth4.cnc star5.cnc		 [-A-] [-B-] [-C-] [-G-]				
		Ok	Cancel						
Entities: () Tool Paths: () Zoom	=X 1 X	1.702	26 Y	0.5716	Ζ-	-0.1	500

Figure 2.7.4 CNC File Browse Window

Help Menu



Figure 2.8

The Help Menu contains selections that allows the user to view the help dialog box, identify the revision of the software, and view the current amount of memory being used. See *Figure 2.8*.

Help-View Help

This will display a window of instructions on how to use the mouse and the keyboard to use the functions of this software. The window has a vertical scroll bar on the right. Click on the upper arrow to go down and the Upper arrow to go up. Click anywhere in the middle to close the dialog window. See *Figure 2.8.1.*

Help-Software Revision

This will display the revision of the software that is currently being used.

Help-Memory Usage

This will display the number of lines in the current part program, the amount of available part program memory, and the total amount of conventional memory available.

File Swit	tch Edit Dra	e View Opti Drawing:«Un	ons CAM titled>	Help		
SELECTIM		ncan.)	1p			
U P t a	sing the Mouse asition the mouse itle and CLICK asition the mouse and CLICK the le	se pointer to t the left mouse se pointer to t ft mouse buttom	he desired button. Th he desired	l menu hen 1 option		
U: Pv in	sing the Keyboa all down menus a n combination w	rd are selected by ith the 1st let	using the ter of a r	s≺ALT> ke senu title		
Entities: (9 Tool Paths:	9 Zoon=X	1 X	1.9467 Y	2.0542	↓ 2 -0.1500

Figure 2.8.1 Help Window

Configuration & Setup (Milling Example)



Figure 3.1 Drawing Title Block

G-code Con Start Pox> X	trol Parameters
Icol Change> X∣ -1	¥ 0 2 05
Output CNC file to:	
Hork-Hold Nethod (Nagnetic Chuck) * (Hold-Down Clamps)	X Autonatic Connents Accept

Figure 3.2 G-code Control Setup This section will show you how to configure MultiCAM Mill. Follow the procedures carefully and you will quickly learn how to design your own parts. Enter the values as indicated below.

3.1 Drawing Title Block

- Choose Options-Drawing Title Block with your mouse or <ALT>-O D from the keyboard. Enter EXAMPLE 1 for the [Drawing Title] and your name in the [Designer Name] box. See *Figure 3.1.*
- Leave the [Revision] as A and the [Date] is automatically entered from the system clock. Enter Wax in the [Material] box.
- 3. Enter 2.0000 for the [Part Width] and [Part Depth] and .4000 for the [Part Height].
- 4. Click on [ACCEPT].

3.2 G-code Control Setup

 Choose Options-G-code Control Setup with the mouse or <ALT>-O C from the keyboard. Enter the X, Y and Z starting position and tool change position values. See *Figure 3.2*. The starting position will be used in the G92 statement when the part program is created. The tool will be located at the tool change position every time a new cutting tool is selected.

Enter X = 0, Y=0, and Z = 0.05 for the starting position and X = -1, Y = 0, and Z = 0.05 for the tool change position.

- 2. Leave the [Output CNC File To] box empty. This will send the generated CNC file to the current directory.
- 3. Choose the work-hold method. Select [Hold-Down Clamps] for the tutorial.
- 4. Select [Automatic Comments].
- 5. Choose [Accept].

Configuration & Setup (Milling Example)



Figure 3.3 Tool Definition Window

3.3 Setting Tool Definitions

This section will show you how to determine your tool geometry and enter it in MultiCAM Mill. Note: The default tool geometry values for Tools 1-6 are set for typical values. These should be adjusted to describe your particular tools.

Choose Tool #3 for the tutorial (0.125" Dia. Endmill).

- 1. Determine the diameter of your tools.
- 2. Choose **CAM-Select Tool** with the mouse or <ALT>-C T from the keyboard. See *Figure 3.3.*
- 3. Enter the diameter of the tool that is selected. When selecting a tool via the keyboard it is necessary to press <Enter> after using the up/down arrow keys to move the highlighter.
- 4. Highlight the next tool and repeated Step 3. Do this until all of your tool diameters have been entered.
- 5. Choose [Accept].

3.4 Saving Parameters to a Setup File

- 1. Choose **Options-Save Setup As** with the mouse or <ALT>-O A from the keyboard. Enter CUSTOMWX in the File Name box. The .MSU extension is added automatically.
- 2. Choose [OK].
- 3. Choose **Options-Save As Default** with the mouse or <ALT>-O D from the keyboard. MultiCAM Mill will automatically be configured with these parameters at startup.

Configuration & Setup (Routing Example)



Figure 3.4 Drawing Title Block



Figure 3.5 G-code Control Setup

This section will show you how to configure MultiCAM Mill. Follow the procedures carefully and you will quickly learn how to design your own parts. Enter the values as indicated below.

3.5 Drawing Title Block

- Choose Options-Drawing Title Block with your mouse or <ALT>-O D from the keyboard. Enter EXAMPLE 2 for the [Drawing Title] and your name in the [Designer Name] box. See *Figure 3.4.*
- Leave the [Revision] as A and the [Date] is automatically entered from the system clock. Enter Wood in the [Material] box.
- 3. Enter 12.0000 for the [Part Width] and [Part Depth] and 0.7500 for the [Part Height].
- 4. Click on [ACCEPT].

3.6 G-code Control Setup

 Choose Options-G-code Control Setup with the mouse or <ALT>-O C from the keyboard. Enter the X, Y and Z starting position and tool change position values. See *Figure 3.5*. The starting position will be used in the G92 statement when the part program is created. The tool will be located at the tool change position every time a new cutting tool is selected.

Enter X = 0, Y=0, and Z = 0.05 for starting positions and X = -1, Y = 0, and Z = 0.05 for tool change positions.

- 2. Leave the [Output CNC File To] box empty. This will send the generated CNC file to the default path.
- 3. Choose the work-hold method. Select [Hold-Down Clamps] for the tutorial.
- 4. Select [Automatic Comments].
- 5. Choose [Accept].



Figure 3.6 Tool Definition Window

3.7 Setting Tool Definitions

This section will show you how to determine your tool geometry and enter it in MultiCAM Mill. Note: The default tool geometry values for Tools 1-6 are set for typical values. These should be adjusted to describe your particular tools.

Choose Tool #4 for the tutorial (0.375" Dia. Endmill).

- 1. Determine the diameter of your tools.
- Choose CAM-Select Tool with the mouse or <ALT>-C T from the keyboard. See *Figure 3.6.*
- 3. Enter the diameter of the tool that is selected. When selecting a tool via the keyboard it is necessary to press <Enter> after using the up/down arrow keys to move the highlighter.
- 4. Highlight the next tool and repeated Step 3. Do this until all of your tool diameters have been entered.
- 5. Choose [Accept].

3.8 Saving Parameters to a Setup File

- Choose Options-Save Setup As with the mouse or <ALT>-O A from the keyboard. Enter CUSTOMWD in the File Name box. The .MSU extension is added automatically.
- 2. Choose [OK].
- Choose Options-Save As Default with the mouse or <ALT>-O D from the keyboard. MultiCAM Mill will automatically be configured with these parameters at startup.

Tutorial (Milling Example)

4.1 Milling Example Tutorial

In this tutorial you will design and edit the part in *Figure 4.1.1.* Before going ahead, you should first complete *Section 3-Configuration and Setup (Milling Example).* This tutorial assumes that MultiCAM Mill has been properly installed (see *Section 1- Getting Started)* and is currently running on your computer.



Figure 4.1.1 Tutorial Example

 Choose Options-Load Setup with the mouse or <ALT>-O L from the keyboard. Select MILL.MSU and choose [OK]. This will reconfigure MultiCAM Mill with the parameters entered in the previous section. The screen should now look like *Figure 4.1.2.*

Tutorial (Milling Example)



Figure 4.1.2 Startup Screen

- To draw the two rectangles, press <R> on the keyboard. This selects the RECTANGLE drawing mode.
- 3. Locate the *START POINT MARKER* to X0.2 and Y1.8 by positioning the mouse arrow and then pushing the right mouse button (<RMB>).
- 4. Move the arrow to X1.8 and Y0.2 and click the (<LMB>). You will see a square on the screen.
- 5. Draw a small rectangle by positioning the *START POINT MARKER* at X1.2 and Y1.5 and set the diagonal corner at X1.5 and Y1.1. You should now have two rectangles on the screen.
- To draw the triangle island, choose <L> on the keyboard. This selects the LINE drawing mode.
- Position the START POINT MARKER at X1.35 and Y0.85. Move the arrow to X1.6 and Y0.4 and click the <LMB>. Move the arrow to X1.1 and Y0.4 and click the <LMB>. Move the arrow back to X1.35 and Y0.85 and click the <LMB>. There should now be a triangle on the screen.

Tutorial (Milling Example)

- 8. To draw the last island, draw two more lines, one from X0.5, Y1.3 to X0.5, Y0.6 and the second one from X0.8, Y1.3 to X0.8, Y0.6.
- Change to ARC mode by pressing <A> on the keyboard. You will now do the two arcs at the ends of the last island.
- 10. Position the START MARKER at X0.5, Y1.3. Move the mouse arrow to X0.8, Y1.3 and press the <LMB>. You will see a second small cross on the screen, these are the two end points of the arc. Now move the mouse arrow to X0.65, Y1.4 and press the <LMB>. You now have an arc on the screen, but does it have the correct radius? You can find out by pressing <I> on the keyboard, select the arc with the mouse, and select [OK]. Change the radius of the arc to 0.156" by choosing Edit-**Change Radius** and then selecting the arc. Enter 0.156 and choose [ACCEPT]. The arc should now have the correct radius. Position the START MARKER at X0.5, Y1.3 and choose **Edit-Copy** with mouse and select the arc. Position the START MARKER at X0.5, Y0.6 and choose Edit-Paste with the mouse. A copy of the arc is now at the opposite end of the island. To flip the second arc, press <F> and select the arc with the mouse. The drawing is now complete. See Figure 4.1.3.



Figure 4.1.3 Completed Part Profile

Computer Aided Design and Manufacturing Software

Tutorial (Milling Example)

- 11. Save the current file by choosing <ALT>-F A and enter EXAMPLE1 in the *File Name* box and select [OK]. The .CDM extension is automatically added to the filename.
- 12. Choose Draw-Select with the mouse. This allows you to select drawing entities so you can use the **CAM-Verify/ Accept Pocket** and **CAM-Verify/Accept Island** functions.
- 13. Select all segments of the square that borders the three islands and choose **CAM-Verify/Accept Pocket**. You have now defined the border as the pocket perimeter.
- 14. Now select both arcs and the two lines that join them by positioning the mouse arrow near them and pressing the <LMB>. After all four entities are selected, choose CAM-Verify/Accept Island with the mouse. You have now defined this entity group as an island. Do the same for the small rectangle and then the triangle.
- 15. Select **CAM-Rough Cut Pocket** with the mouse or choose <ALT>-C R from the keyboard. MultiCAM Mill is now processing the part profile and generating a tool path. When complete, the screen should look like *Figure 4.1.4.*



Figure 4.1.4 Completed Roughing Cut Tool Path

4

Tutorial (Milling Example)



Sequence of Finish Path Points For Pockets

Figure 4.1.5



Sequence of Finish Path Points For Islands



- 16. Place the finishing cuts by selecting **CAM-Place Finish Path** with the mouse or <ALT>-C F from the keyboard.
- 17. Position the mouse arrow near the lower left corner of the large square and press the <LMB>. This is the starting point (P1) of the cutting move. Move the mouse arrow above that point and press the <LMB> again. This point (P2) determines the direction of the cutting move. Position the mouse arrow to the right of the vertical line and press the <LMB>. This point (P3) indicates that th tool compensation is to the right of the selected segment, therefore, the inside of the square will be cut. The finishing cut path for the border is now on the screen. *Figure 4.1.5.*
- 18. Press <F3> to repeat the last function. Positon the mouse cursor near the lower left corner of the triangle and press the <LMB> (P1). Move the mouse arrow just to the right and press the <LMB> again (P2). Move the mouse arrow below the triangle and press the <LMB> again (P3). A finish cut path is now around the triangle. *Figure 4.1.6.*
- 19. Repeat Step 18 for last two islands. The screen should now look like *Figure 4.1.7.*



Figure 4.1.7 Completed Tool Path

Computer Aided Design and Manufacturing Software

Tutorial (Milling Example)

20. Press <CTRL>-N to the make the CNC file.

In addition to the commands used in the above tutorial, there are others that you may wish to experiment with to become a proficient user of MultiCAM Mill. These include:

- <ESC> to undo the last drawing command.
- to *delete* a segment.
- <F> to *flip* an arc.
- <D> to reverse the arc *direction*.
- <J> to *linearly* join two end points.
- <V> to *move* an end point.
- to *break* a segment in two.
- <M> to *move* the entire profile.
- <T> to *trim* overlapping drawing entites.
- <E> to enter an *exact* position.

Computer Aided Design and Manufacturing Software

Tutorial (Routing Example)

4.2 Routing Example Tutorial

In this tutorial you will design and edit the part in *Figure 4.2.1.* Before going ahead, you should first complete *Section 3-Configuration and Setup (Routing Example).* This tutorial assumes that MultiCAM Mill has been properly installed *(see Section 1- Getting Started)* and is currently running on your computer.



Figure 4.2.1 Tutorial Example

 Choose Options-Load Setup with the mouse or <ALT>-O L from the keyboard. Select ROUTER.MSU and choose [OK]. This will reconfigure MultiCAM Mill with the parameters entered in the previous section. The screen should now look like *Figure 4.2.2.*

Tutorial (Routing Example)



Figure 4.2.2 Startup Screen

- To draw the outside border, press <R> on the keyboard. This selects the RECTANGLE drawing mode.
- 3. Locate the *START POINT MARKER* to X1.0 and Y1.0 by positioning the mouse arrow and then pushing the right mouse button (<RMB>).
- 4. Move the arrow to X11.0 and Y11.0 and click the (<LMB>). You will see a square on the screen.
- 5. Draw a small rectangle by positioning the *START POINT MARKER* at X2.0 and Y8.5 and set the diagonal corner at X4.0 and Y3.0. You should now have two rectangles on the screen.
- Erase the top line of the small rectangle by pressing and selecting the line with the mouse.
- Copy one of the vertical lines to start forming the other two islands. Position the *START POINT MARKER* at X-1.0 and Y3.0. Press <C> on the keyboard and select the left-most vertical line of the small rectangle. Move the *START POINT MARKER* to X2.0 and Y3.0 and press <P> on the keyboard. Push <P> again.



Tutorial (Routing Example)

- To do the remaining lines, position the *START POINT MARKER* at X1.0 and Y3.0. Press <C> on the keyboard and select the right-most vertical line of the small rectangle. Move the *START POINT MARKER* to X4.0 and Y3.0 and press <P> on the keyboard. Push <P> again. Move the *START POINT MARKER* to X5.0 and Y3.0. Move the mouse arrow to X7.0 and Y3.0. Draw another line from X8.0 and Y3.0 to X10.0 and Y3.0.
- 9. Change to ARC mode by pressing <A> on the keyboard.
- 10. Position the START MARKER at X2.0, Y8.5. Move the mouse arrow to X4.0, Y8.5 and press the <LMB>. You will see a second small cross on the screen, these are the two end points of the arc. Now move the mouse arrow to X3.0, Y8.5 and press the <LMB>. You now have an arc on the screen, but does it have the correct radius? You can find out by pressing <I> on the keyboard, select the arc with the mouse, and select [OK]. Change the radius of the arc to 1.125" by choosing Edit-Change Radius and then selecting the arc. Enter 1.125 and choose [ACCEPT]. The arc should now have the correct radius. Position the START MARKER at X-1.0, Y8.5, press <C> on the keyboard and select the arc. Position the START MARKER at X2.0, Y8.5 and press <P> on the keyboard. Press <P> again. The drawing is now complete. See Figure 4.2.3.



Figure 4.2.3 Completed Part Profile

Computer Aided Design and Manufacturing Software

Tutorial (Routing Example)

- 11. Save the current file by choosing <ALT>-F A and enter EXAMPLE2 in the *File Name* box and select [OK]. The .CDM extension is automatically added to the filename.
- 12. Choose Draw-Select with the mouse. This allows you to select drawing entities so you can use the **CAM-Verify/ Accept Pocket** and **CAM-Verify/Accept Island** functions.
- 13. Select all segments of the square that borders the three islands and choose **CAM-Verify/Accept Pocket**. You have now defined the border as the pocket perimeter.
- 14. Now select the left-most arc and the three lines that join them by positioning the mouse arrow near them and pressing the <LMB>. After all four entities are selected, choose **CAM-Verify/Accept Island** with the mouse. You have now defined this entity group as an island. Do the same for the other two islands.
- 15. Select **CAM-Rough Cut Pocket** with the mouse or choose <ALT>-C R from the keyboard. MultiCAM Mill is now processing the part profile and generating a tool path. When complete, the screen should look like *Figure 4.2.4.*



Figure 4.2.4 Completed Roughing Cut Tool Path

Tutorial (Routing Example)



Sequence of Finish Path Points For Pockets





- Place the finishing cut paths by selecting CAM-Place Finish Path with the mouse or <ALT>-C F from the keyboard.
- 17. Position the mouse arrow near the lower left corner of the large square and press the <LMB>. This is the starting point (P1) of the cutting move. Move the mouse arrow above that point and press the <LMB> again. This point (P2) determines the direction of the cutting move. Position the mouse arrow to the right of the vertical line and press the <LMB>. This point (P3) indicates that the tool compensation is to the right of the selected segment, therefore, the inside of the square will be cut. The finishing cut path for the border is now on the screen. *Figure 4.2.5.*
- 18. Press <F3> to repeat the last function. Position the mouse cursor near the lower left corner of the left-most island and press the <LMB> (P1). Move the mouse arrow just to the right and press the <LMB> again (P2). Move the mouse arrow below the island and press the <LMB> again (P3). A finish cut path is now around the island. *Figure 4.2.6.*
- 19. Repeat Step 17 for last two islands. The screen should now look like *Figure 4.2.7.*



Figure 4.2.7 Completed Tool Path

Computer Aided Design and Manufacturing Software

Tutorial (Routing Example)

20. Press <CTRL>-N to the make the CNC file.

The differences between the milling example and the routing example are as follows:

- 1. The material size and type is 2" x 2" x 0.40" wax in the milling example and 12" x 12" x 0.75" wood in the routing example.
- 2. The visual and cursor grids are 0.2" and 0.025" respectively in the milling example and from 1.0" and 0.05" respectively in the routing example.
- 3. The selected tool is #3 (0.125" Std. Endmill) in the milling example and #4 (0.375" Ball Endmill) in the routing example.



Appendix A



The numbers on the top of the keys are the scan (or "make") codes. These codes are used for programming the active jog key for any axis and direction.