Table of Contents

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.

4. 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

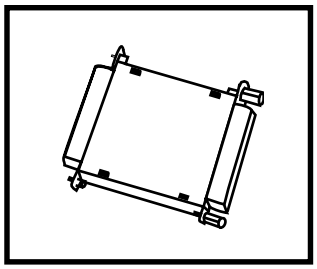


Figure 1.1.1 Security Key

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>*

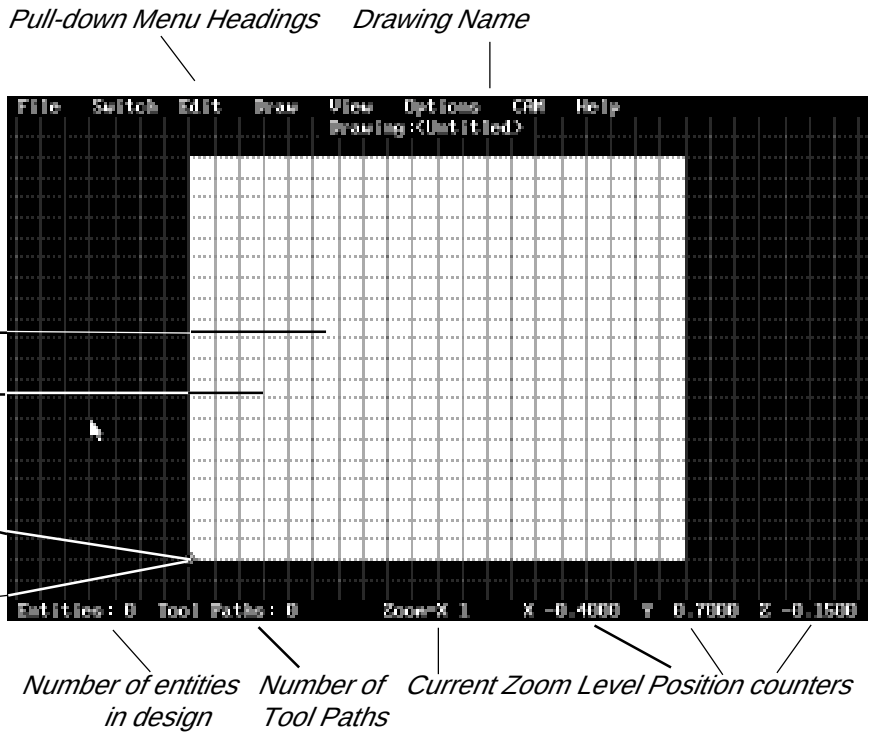
This will start the MultiCAM Mill software.

Pull-Down Menu Command Reference

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.

Figure 1.1.2

MultiCAM Mill Design Screen



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.

File Menu

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.

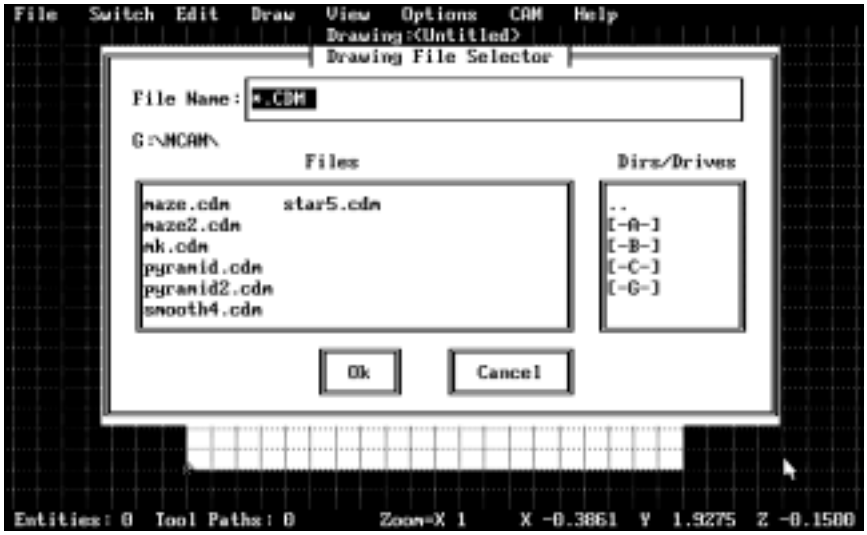


Figure 2.1.1 Load File Dialog Window

File-Revert to Saved

Reloads the last saved version of the current drawing.

File Menu

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 MultiCAM™ 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 auto-dimensioning lines may need to be deleted after importing the drawing.

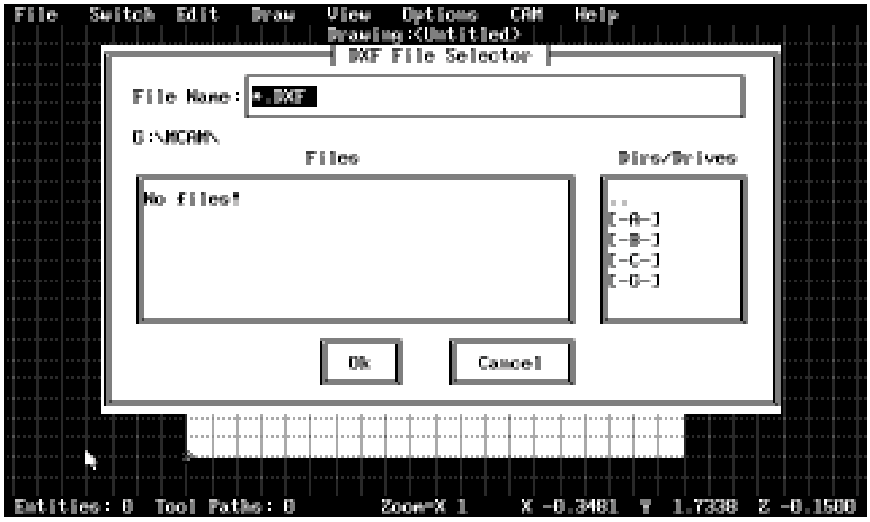


Figure 2.1.2 Import DXF File Window

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 Menu

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.

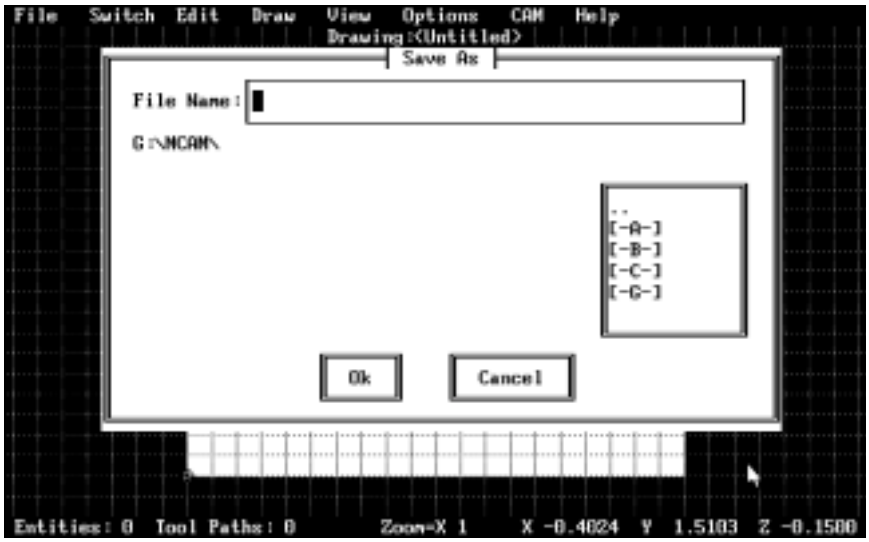


Figure 2.1.3 Save As Dialog Window

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

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.***

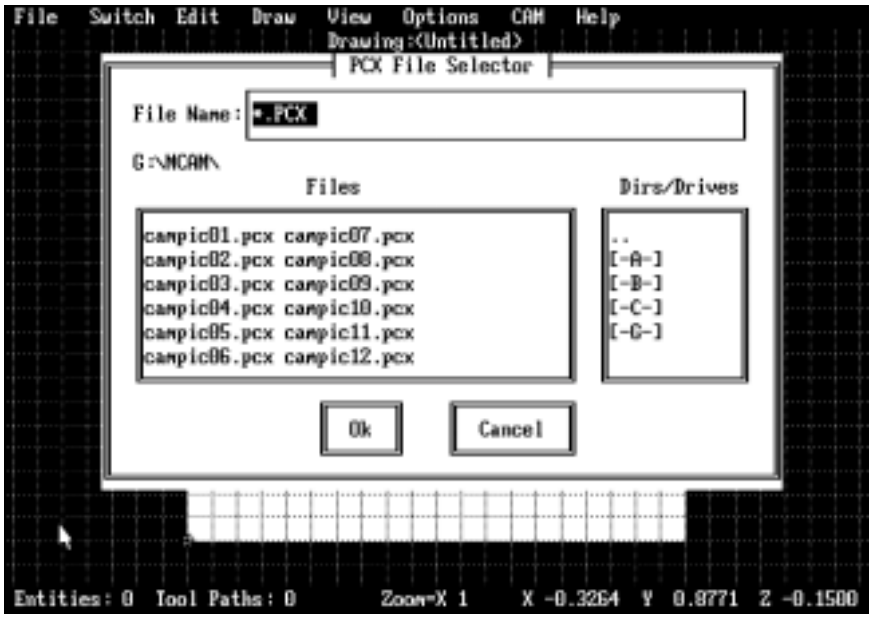


Figure 2.1.4 PCX File Selector Window

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.

Switch Menu

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

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 *<INSERT>* 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.

Edit Menu

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

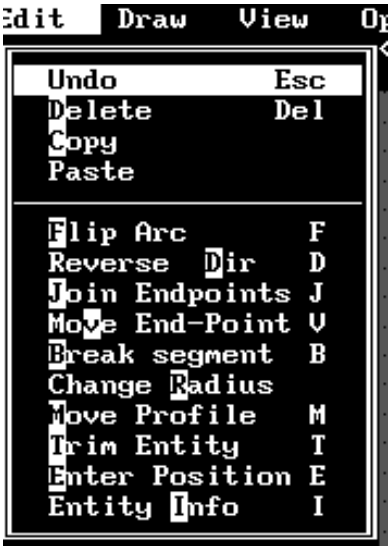


Figure 2.3

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.

Draw Menu

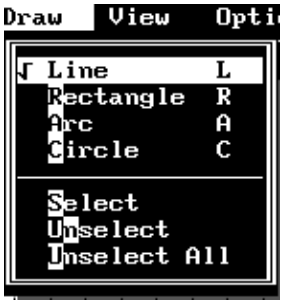


Figure 2.4

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.

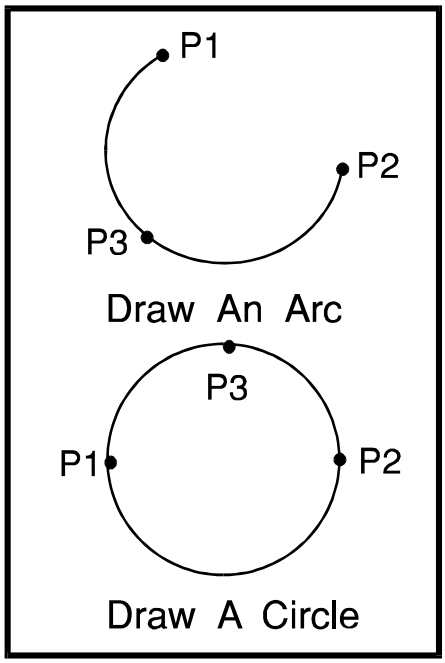


Figure 2.4.1
Circle and Arc Point Placements

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.

Draw-Circle
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.***

Draw Menu

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.

View Menu

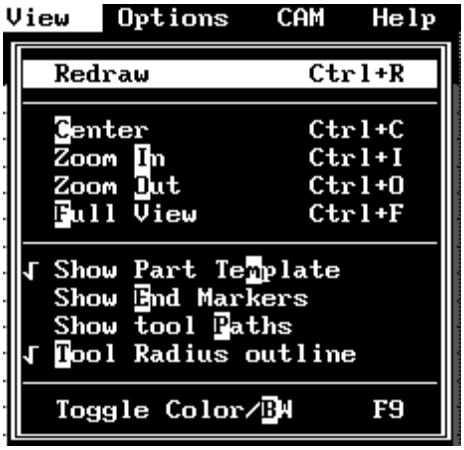


Figure 2.5

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

Figure 2.6

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.***

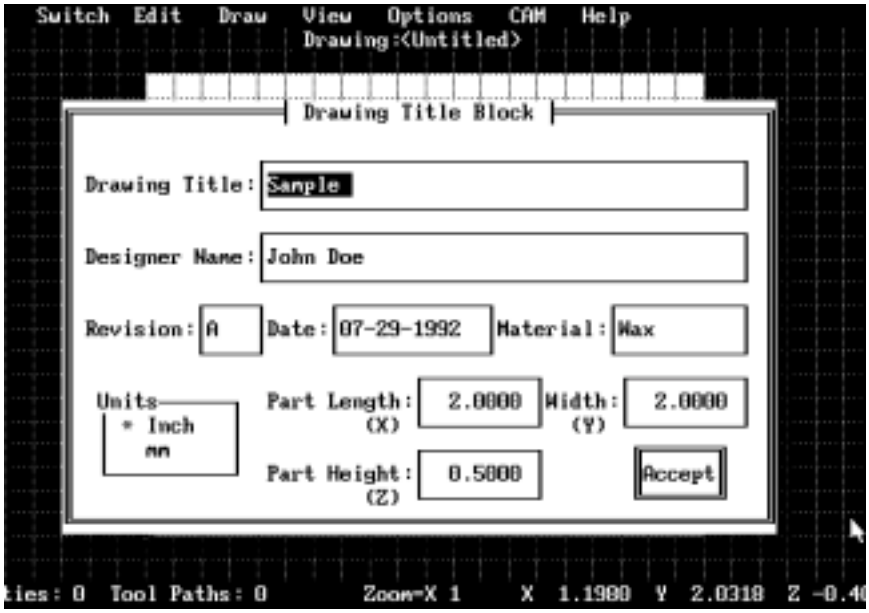


Figure 2.6.1 Drawing Title Block Window

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.***

Options Menu

Figure 2.6

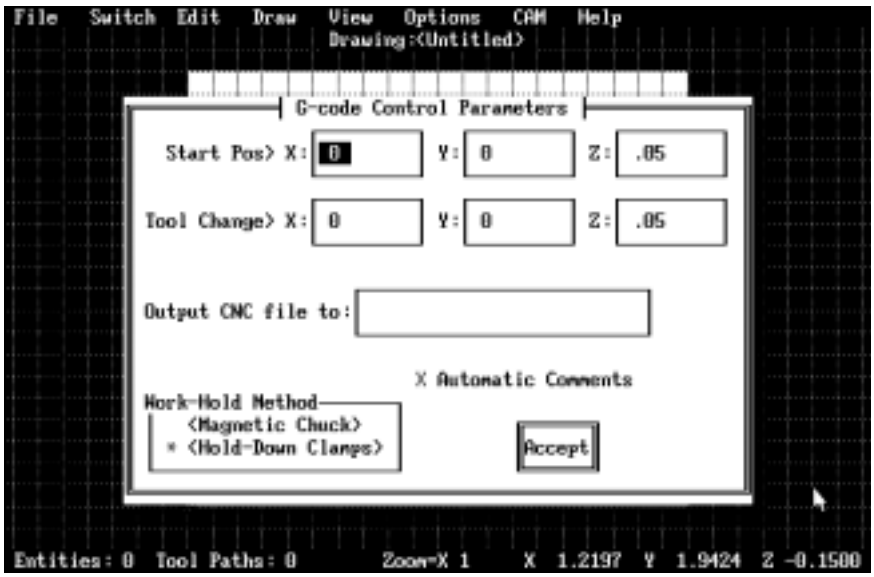


Figure 2.6.2 G-code Setup Dialog Box

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.***

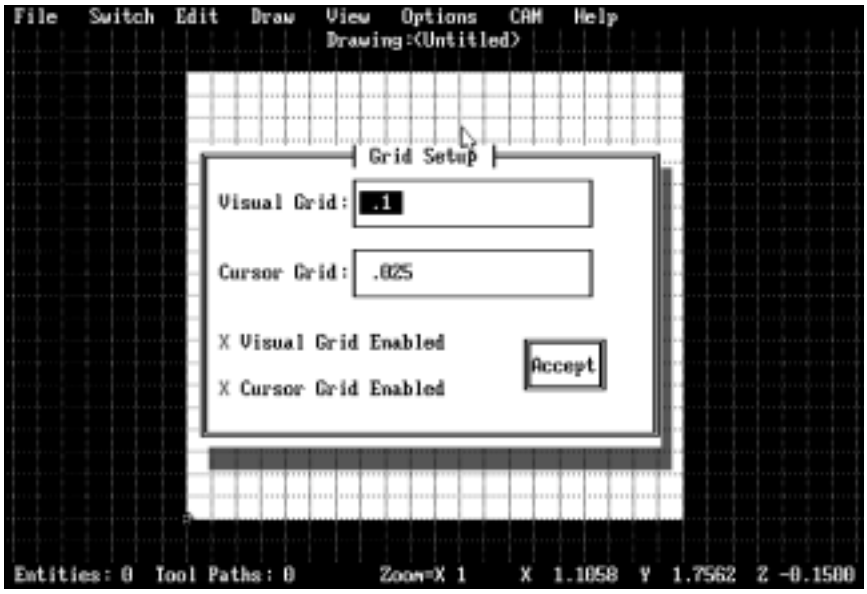


Figure 2.6.3 Visual and Cursor Grid Setup

Options Menu

Figure 2.6

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***.

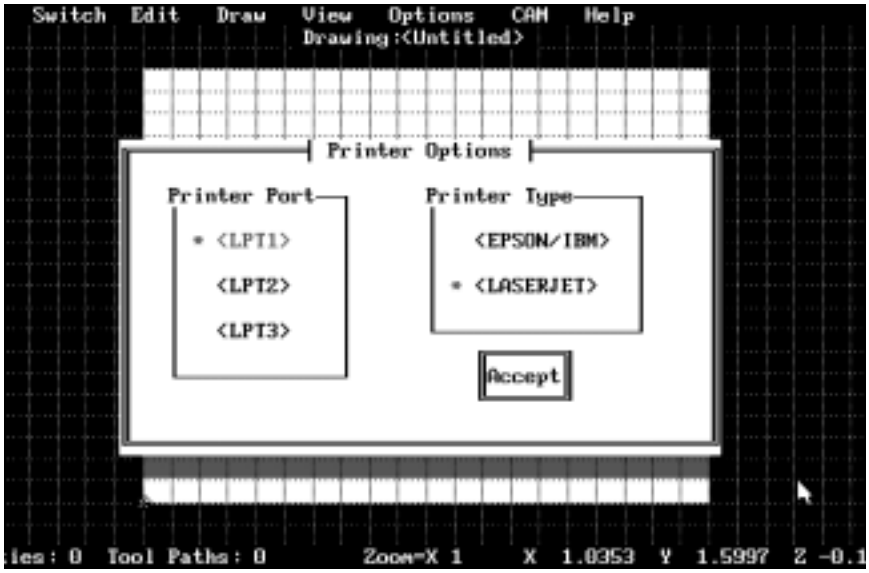


Figure 2.6.4 CNC Printer Options

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 Menu

Figure 2.6

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.***

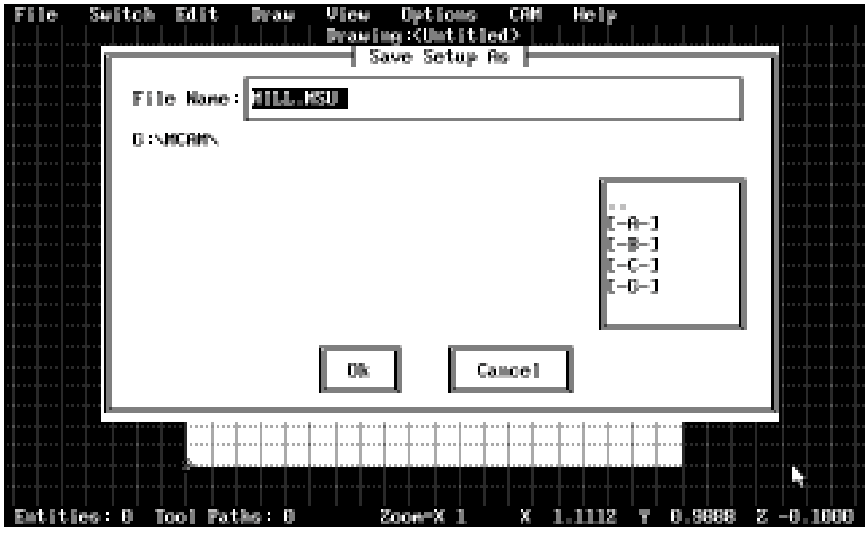


Figure 2.6.6 Save Setup As Window

Options Menu

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***.

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.***

Figure 2.6.8 Menu Colors Setup Window

CAM Menu

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.***

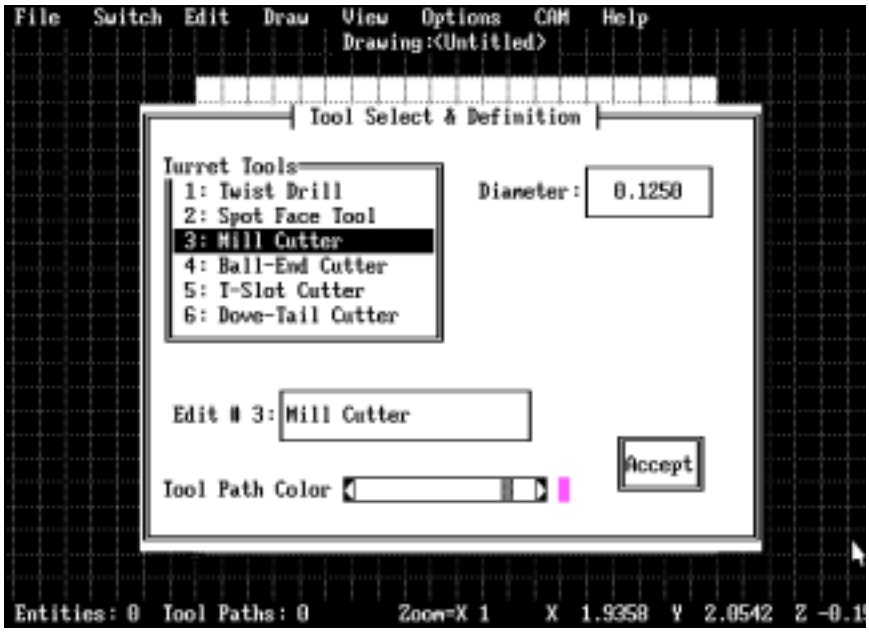


Figure 2.7.1 Tool Select Dialog Window

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.***

CAM Menu

Figure 2.7

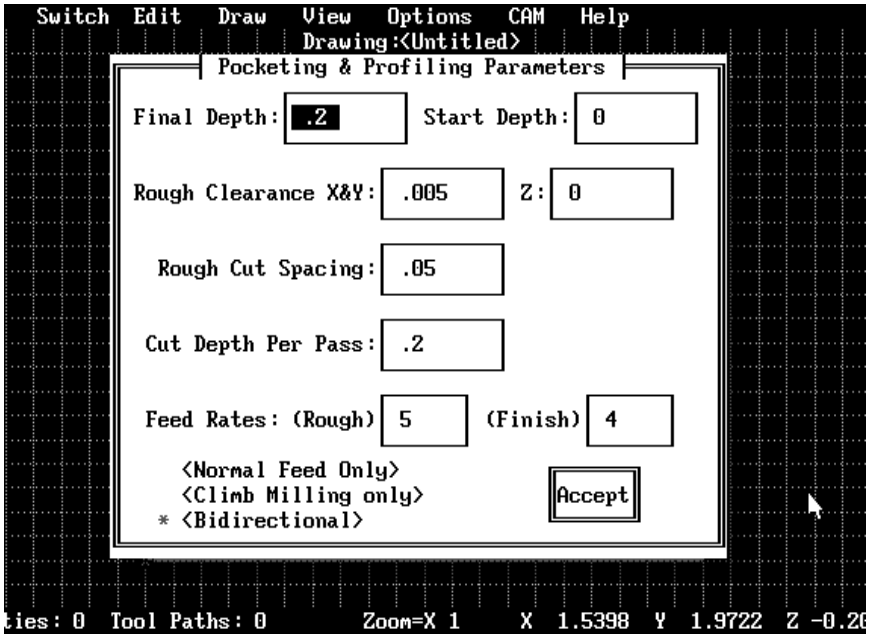


Figure 2.7.2 Pocketing Specs Dialog Window

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.

CAM Menu

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***.

1. Position the start point and press the right mouse button to relocate the *START MARKER* 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 " ◎ " will appear at the *START MARKER* position indicating a canned cycle.

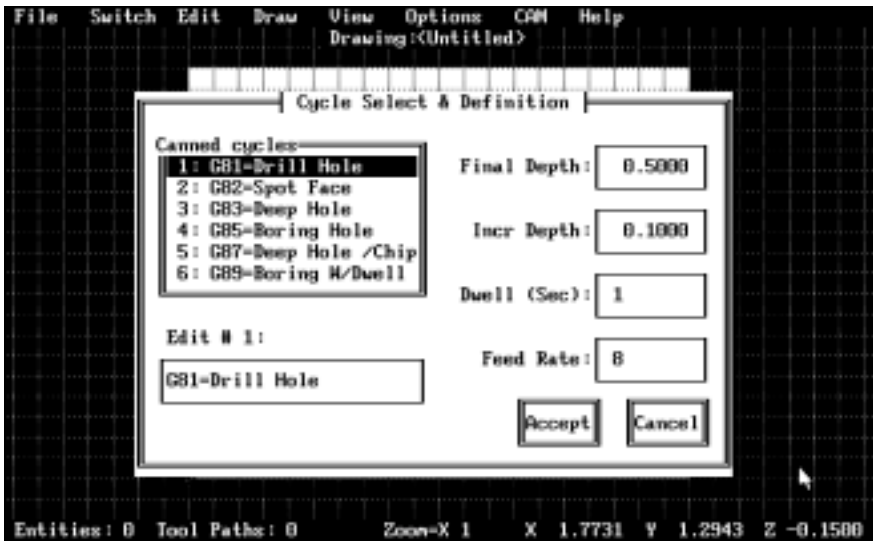


Figure 2.7.3 Cycle Selection Dialog Window

CAM Menu

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.***

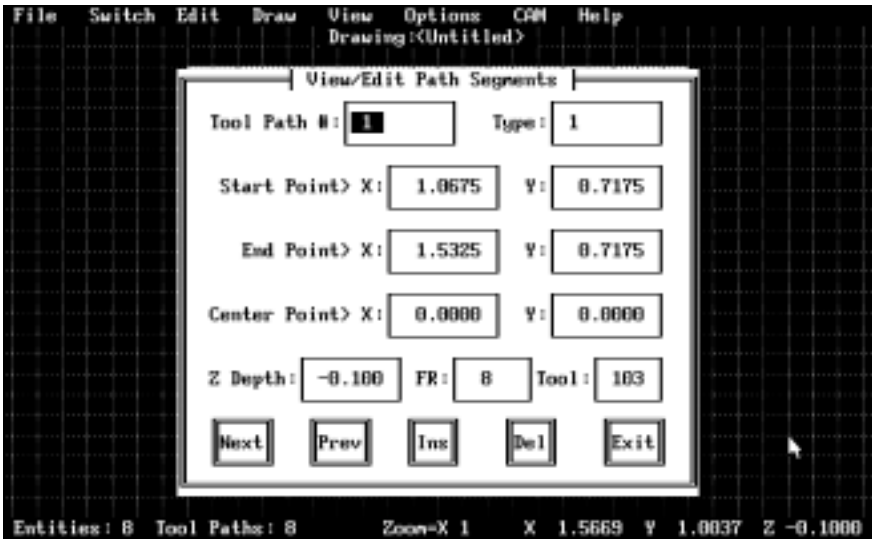


Figure 2.7.4 Edit Path/Cycle Dialog Window

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 Menu

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.***

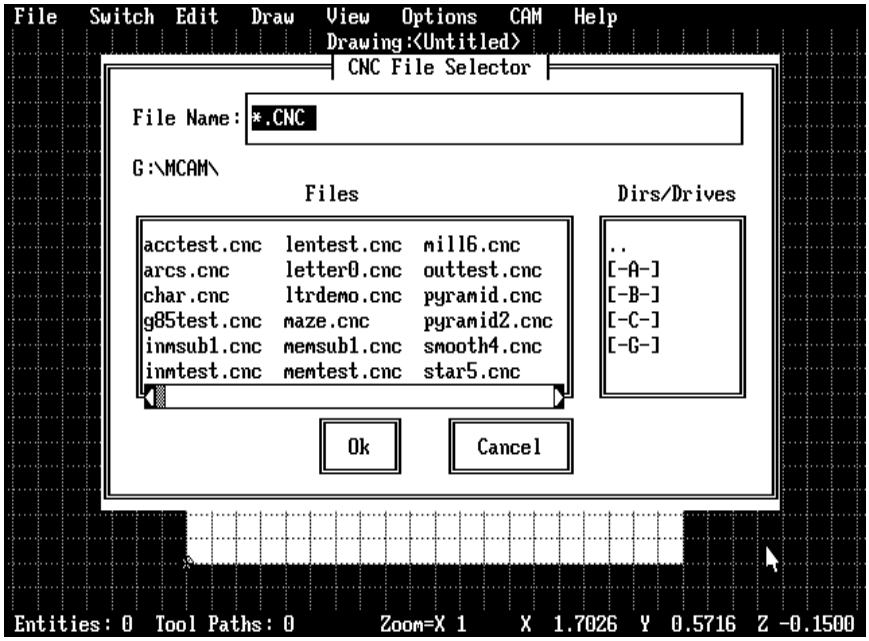


Figure 2.7.4 CNC File Browse Window

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.

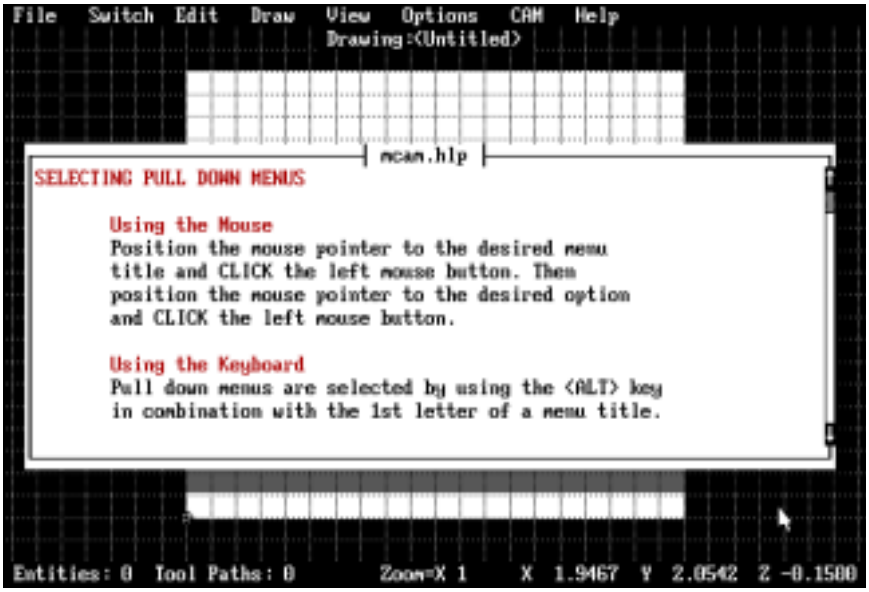


Figure 2.8.1 Help Window

Configuration & Setup

(Milling Example)

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

1. 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.***

2. 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].

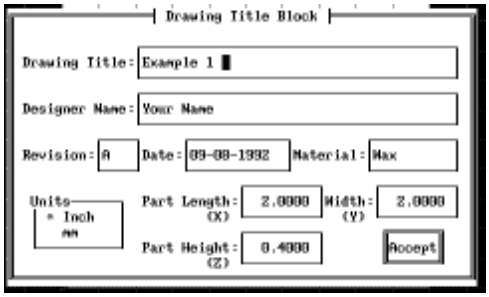


Figure 3.1
Drawing Title Block

3.2 G-code Control Setup

1. 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].

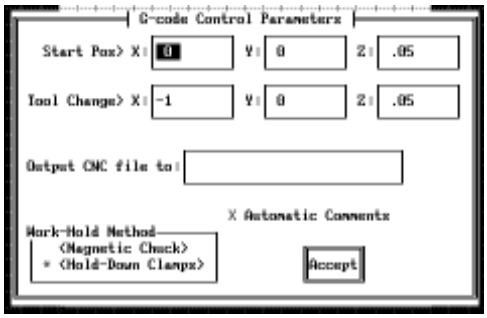


Figure 3.2
G-code Control Setup

Configuration & Setup
(Milling Example)

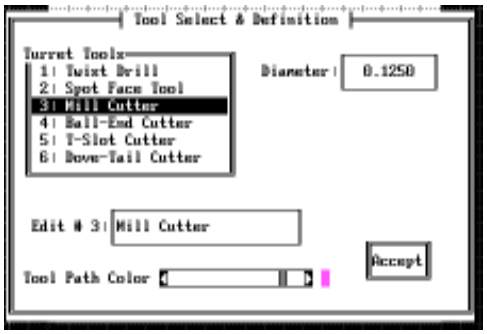


Figure 3.3
Tool Definition Window

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)

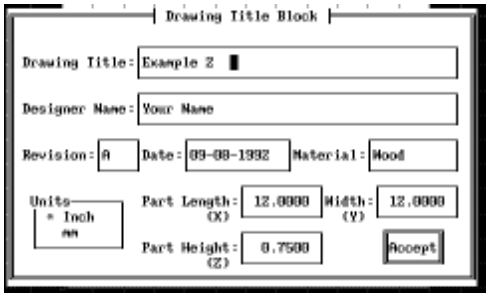


Figure 3.4
Drawing Title Block

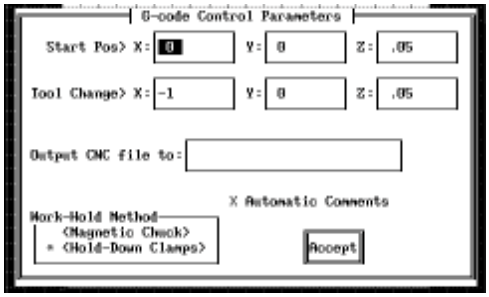


Figure 3.5
G-code Control Setup

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

1. 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.***

2. 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

1. 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].

Configuration & Setup

(Routing Example)

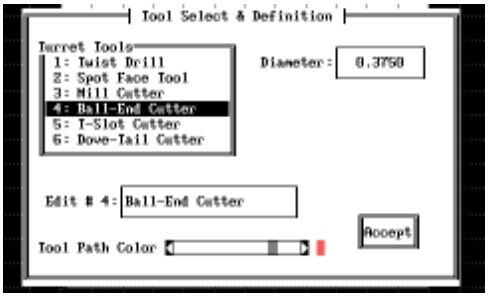


Figure 3.6
Tool Definition Window

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.

2. 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

1. 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].

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.

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.

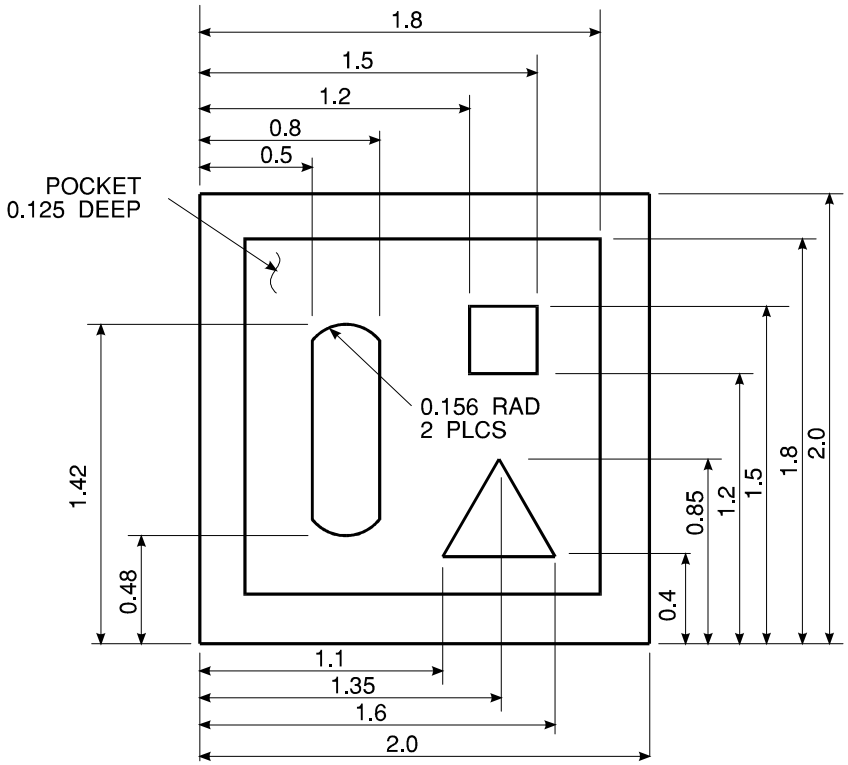


Figure 4.1.1 Tutorial Example

1. 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)

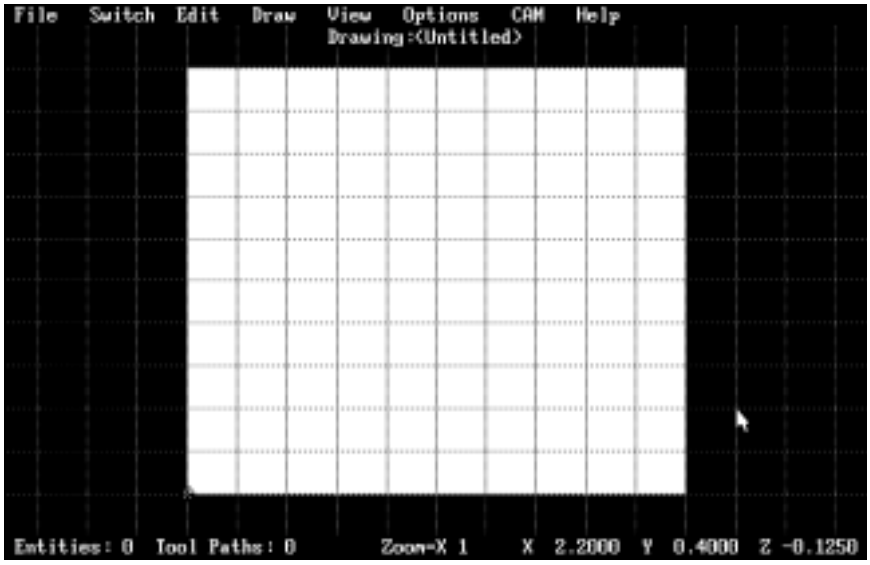


Figure 4.1.2 Startup Screen

2. 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.

6. To draw the triangle island, choose <L> on the keyboard. This selects the LINE drawing mode.

7. 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.

9. 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.***

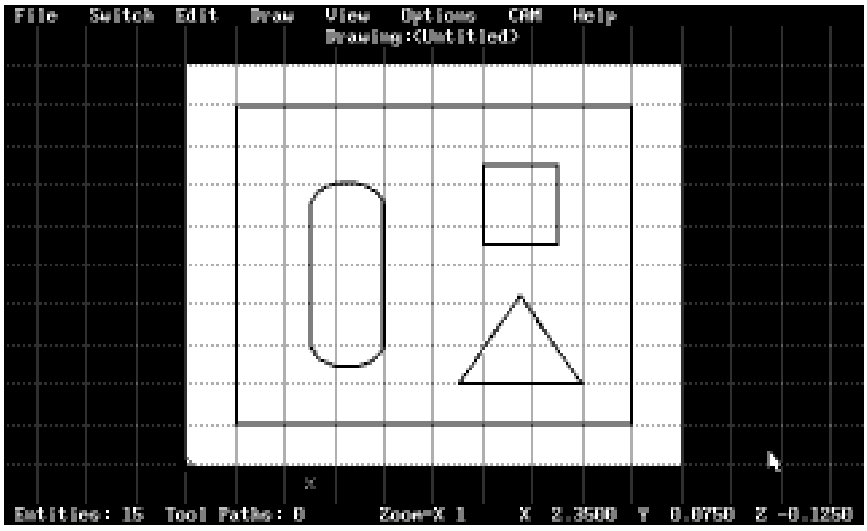


Figure 4.1.3 Completed Part Profile

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.***

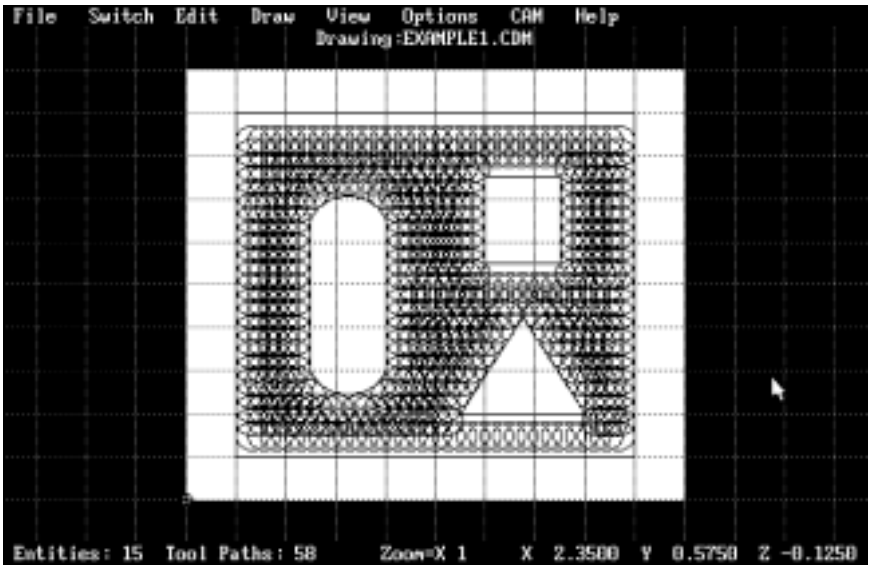
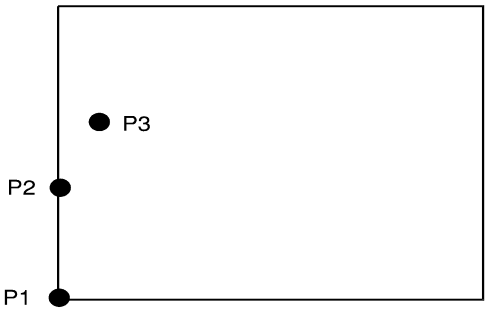


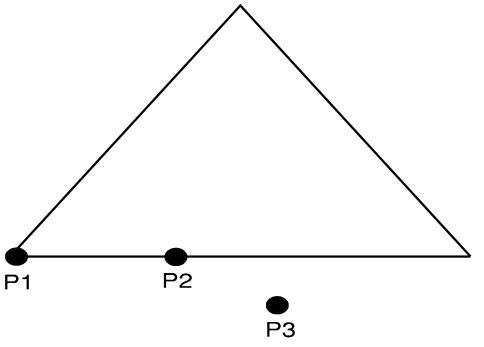
Figure 4.1.4 Completed Roughing Cut Tool Path

Tutorial
(Milling Example)



Sequence of Finish Path Points For Pockets

Figure 4.1.5



Sequence of Finish Path Points For Islands

Figure 4.1.6

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.***

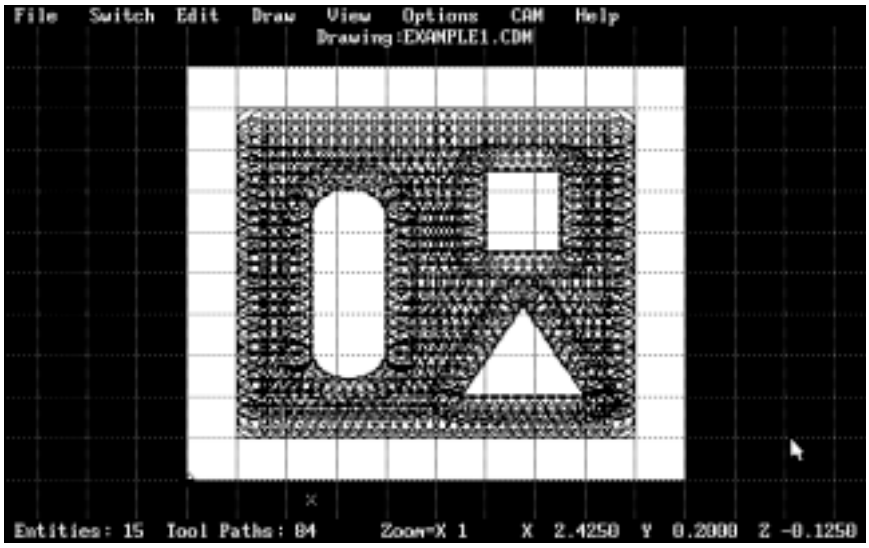


Figure 4.1.7 Completed Tool Path

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.
- <DEL> 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.
- <B> to *break* a segment in two.
- <M> to *move* the entire profile.
- <T> to *trim* overlapping drawing entites.
- <E> to enter an *exact* position.

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.

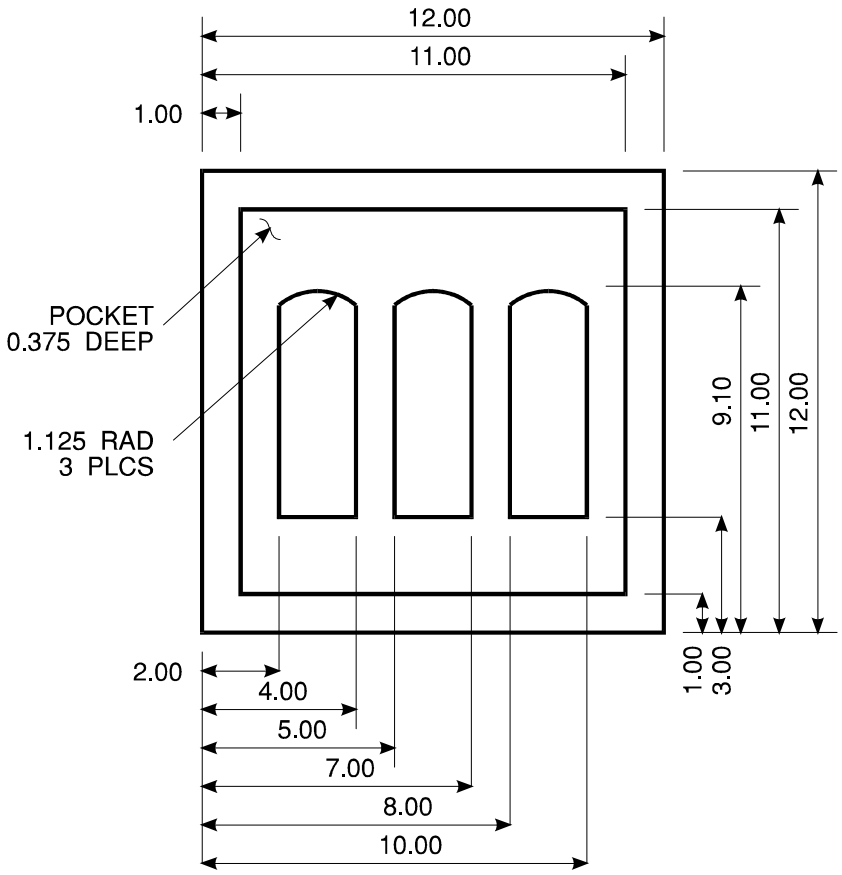


Figure 4.2.1 Tutorial Example

1. 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)

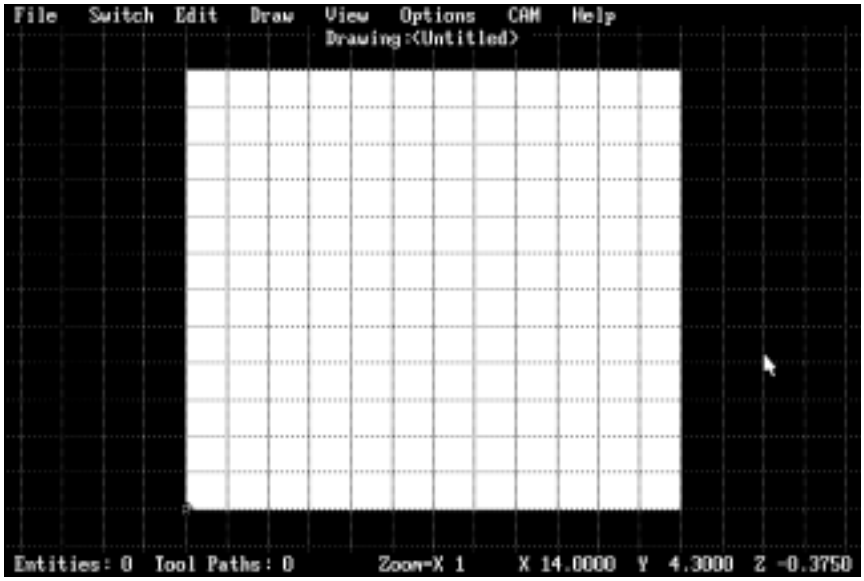


Figure 4.2.2 Startup Screen

2. 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.

6. Erase the top line of the small rectangle by pressing <DEL> and selecting the line with the mouse.

7. 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)

8. 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.***

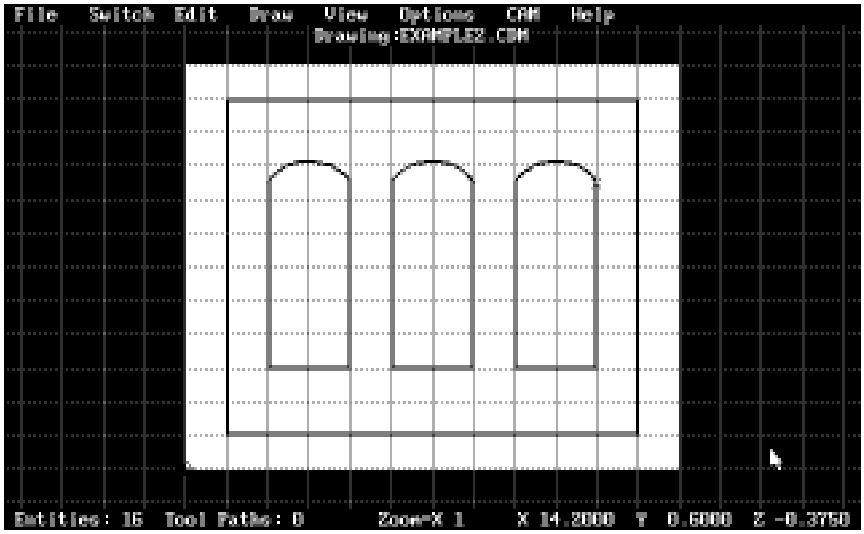


Figure 4.2.3 Completed Part Profile

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.***

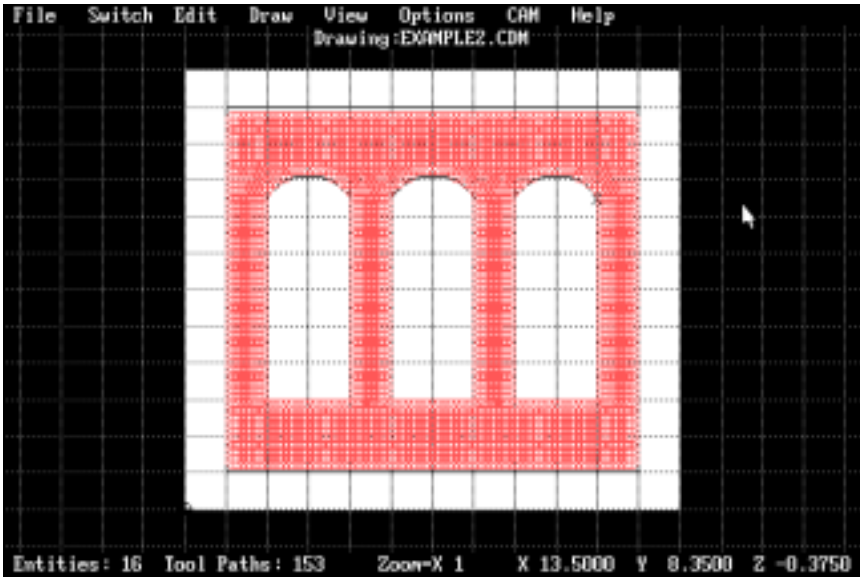
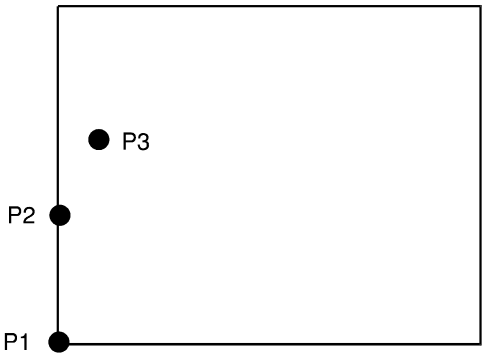


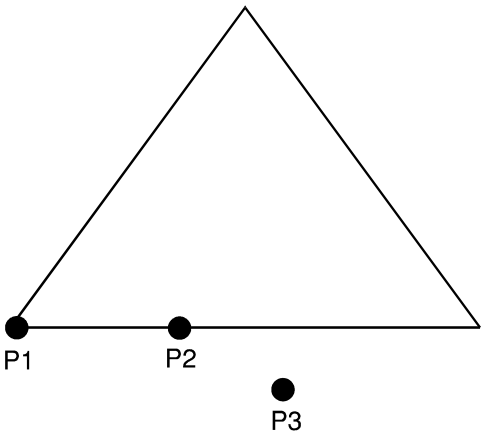
Figure 4.2.4 Completed Roughing Cut Tool Path

Tutorial
(Routing Example)



Sequence of Finish Path Points For Pockets

Figure 4.2.5



Sequence of Finish Path Points For Islands

Figure 4.2.6

16. 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.***

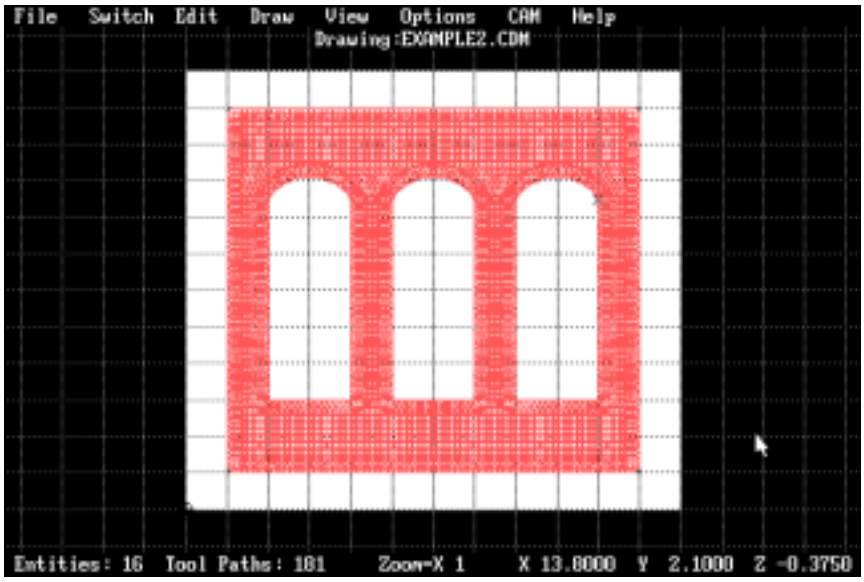


Figure 4.2.7 Completed Tool Path

Tutorial
(Routing Example)

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

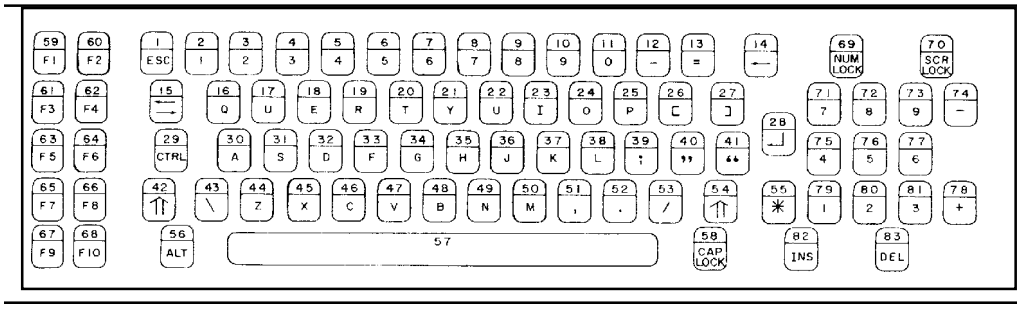
The differences between the milling example and the routing example are as follows:

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.