Chapter 9

CAM Settings, Pocketing, and NC Format

The Tool Help function

One of the problems you face when you want to contour or pocket a shape is setting an appropriate tool size. MillWrite provides a function to help you determine which size of tool would be appropriate for the particular shape your considering. From the **Main Menu** (click it with the mouse or press Esc), pick the **CAM Functions**, and then pick **Help me determine appropriate tool sizes**. Or just press Alt S.

The arrow that was created in Chapter 3 will be used as an example of how this function works. Figure 9-1 shows the screen after the tool help function has started. The buttons at the top of the screen provide the following options:

- *By **Dragging** or By **Diameter***
  If you pick the **By Dragging** option, then as you move the mouse the diameter of the tool will change. This lets you check a wide variety of tool diameters very quickly. If you pick the **By Diameter** option, you'll check only a specific tool diameter.

- ***Continuous** or In **Steps***
  These options are available only if you pick the **By Dragging** option. As you drag the mouse the tool diameter will change, and you have the option of letting the diameter change in specific **steps** or **continuously**. In this example a step size of 1/64th of an inch has been specified.

After you've set the options you want for the tool help function, put the mouse onto the object you want to check. The mouse icon will change to show the words ***Drag Tool Size***, as seen in Figure 9-1. Then click the *left* mouse button and move the mouse. Because the ***By Dragging*** option has been chosen, as you move the mouse the offsets and circles will change in increments of the step size of 1/64th of an inch.

Figure 9-2 shows the result of dragging the tool diameter to .125. MillWrite has drawn a tool path for a tool of that size and put circles at the ends of all lines and arcs of that path. The circles show the diameter of the tool. The tool size is displayed at both the top and bottom of the screen.

How to interpret the results

If you're trying to set the size of the roughing tool, then the size of these circles can be used to judge whether the roughing tool is too large or too small. In such a case you simply look at the circles to see if they fit properly. But if you're trying to set the finishing tool, then you want to make sure the tool will fit into the corners. In that case you look at the circles to see how much material is remaining at the corners. In Figure 9-2, the tool size is 0.125 in. This would be appropriate for a roughing tool but it leaves a lot of material uncut at the head of the arrow, so it would not be appropriate for a finishing tool.

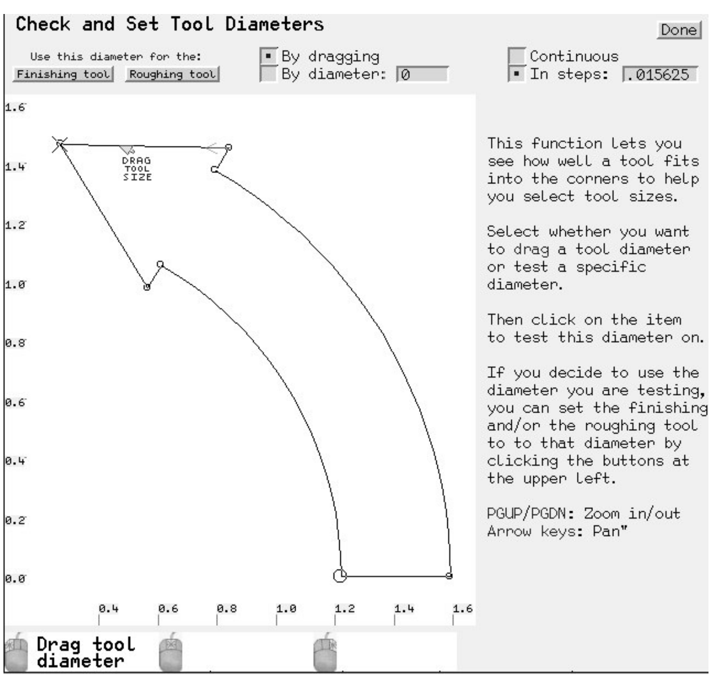

Figure 9-1

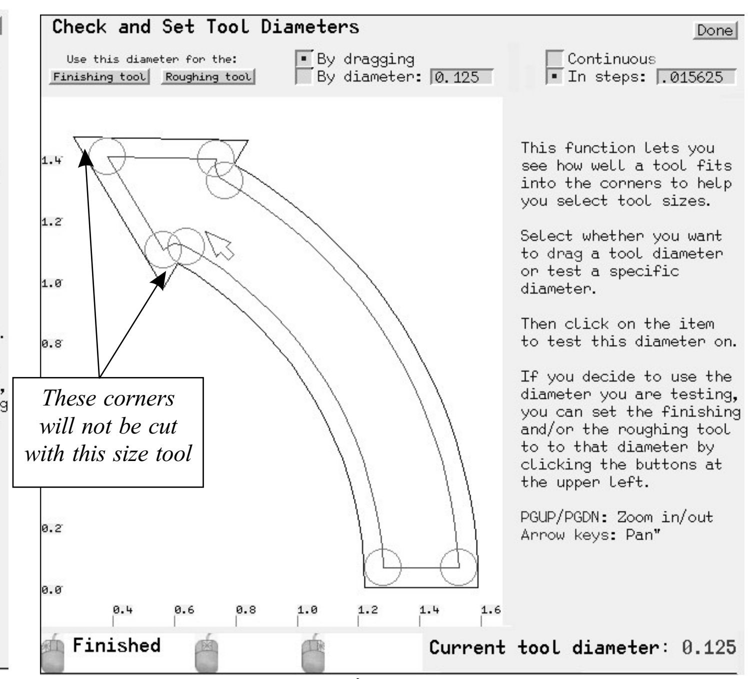

Figure 9-2

The option to drag a tool diameter allows you to quickly check a variety of tool sizes. You may find an appropriate diameter for both a roughing tool and for a finishing tool. You could remember the appropriate tool sizes and then set the item to use those tool sizes. Or you could click one of the two buttons in the upper left corner of the screen. If you click the ***Finishing Tool*** button, MillWrite will set the diameter of finishing tool for that item to whatever the current tool diameter is. And if you click the roughing tool button, MillWrite will set the roughing tool for that item to the current tool diameter. But for now let's just remember the tool size of 0.125 in is appropriate for a roughing tool so that you can see how to manually set the tool sizes.

How do you pocket an item?

The arrow, as created in Chapter 3, was set to **engrave**. Let's assume now that you want to **pocket** the interior of the arrow. Touch any line or arc of the arrow with the mouse and the parameters for that line or arc will appear on the right side of the screen. Move the mouse to the parameters area (or as the message at the lower right corner reminds you, press the letter **E** for **Edit**). Then click on the **Usage** field and then pick **Spiral Pocket**.

Pocketing requires a closed boundary

In Chapter 3 the individual lines and arcs that make up the arrow were joined together into one polyline. If the arrow still consisted of individual lines and arcs, MillWrite would have joined all of them together into a single, closed polyline when you picked the spiral pocket option. The reason is that spiral pocketing requires a closed boundary.

However, if MillWrite couldn't figure out how to make a closed boundary, MillWrite would have displayed an error message that the item you hi-lited doesn't form a closed boundary, and the usage would not be set to spiral pocket. In such a case, you would have to fix the drawing, or join the items together into a polyline yourself.

Set the tool parameters

When the arrow was set to engrave, MillWrite provided only one tool field. After picking the spiral pocket option, MillWrite changes the parameters to allow both a roughing tool and a finishing tool. You have three options in regards to tools when pocketing:

- ***Only*** *a* ***finishing*** *tool*
- ***Only*** *a* ***roughing*** *tool*
- ***Both*** *a roughing and finishing tool*

Using ONLY a Finishing Tool

Click the ***Finishing Tool*** data field and the finishing tool parameters appear (Figure 9-3). Underneath the OK button is the ***Automatic Tool Changer Number*** for this tool. Even if you don't have an automatic tool changer, you need to put a number in this field or MillWrite will assume you don't want this tool.

Below the automatic tool changer number is the ***cut per pass*** field. This is a percentage of the tool diameter. This specifies what percentage of the tool should be cutting in each pass. For example, if you have a 1 in. diameter tool and you want only a quarter of an inch of material to be cut in each pass, you would specify a cut per pass of 25%.

Underneath the feed and speed fields is the ***final cut thickness***. If you leave this blank or enter 0, that means you do not want the final cut to be any different from any of the other pocketing cuts. But if you want the final cut to take a sliver of material in order to get a smooth finish, then enter a small value here, such as .01 inches.

Next is the ***lead-in on final cut*** field, which is either yes or no. If yes, MillWrite will bring the tool into the final pass in an arc rather than a straight line. The purpose of this is to avoid tool marks.

Next are the fields that define the type of tool this is. Then comes the ***depth per cut*** and the ***final depth***. If you enter a value in the depth per cut field, you're telling MillWrite that this tool can cut a maximum depth of this amount. Therefore, if the final depth is greater than the depth per cut, MillWrite will make more than one pass of the tool.

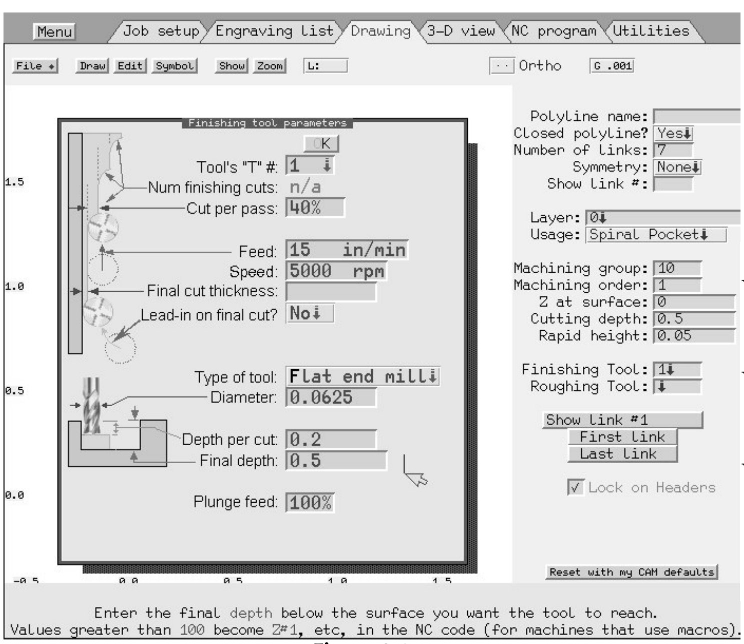

Figure 9-3

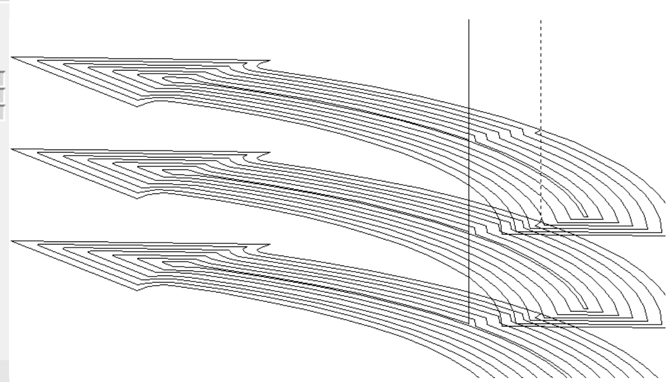
Figure 9-4

If you want only one pass of the tool, either enter a 0 for the depth per cut or make the depth per cut at least as deep as the final depth.

In Figure 9-3 the depth per cut is 0.2 in. and the final depth is 0.5 in. Therefore MillWrite will make three passes of the tool. Each pass will be one-third of the total depth, which is 0.1667 inch per pass.

The last field is the ***plunge feed*** field. This is a percentage of the ***XY feed*** rate (the XY feed rate is near the top of this window). If plunge feed is set for 100%, that means the tool will plunge into the material with 100% of the feed rate that it cuts in the XY direction. If the feed rate is 15 in. per minute, but if you want the plunge rate to be only 5 in. per minute, you would set the plunge feed to 33%.

When you're done setting the tool parameters click the OK button at the top. Since there is no cancel button, click the **right** mouse button or press Esc to cancel.

Finishing tools do not spiral down

Now that the finishing tool has been set, let's take a look at the result. Click the 3D view page and use the left mouse button to rotate the tool path until you can see all three levels, as in Figure 9-4. The vertical line is the tool dropping down into the part. It cuts the first level, then drops straight down to cut the second level, and drops down to cut the last level.

This shows one of the limitations of the finishing tool. Specifically, the finishing tool only drops straight down into the part, it doesn't spiral down. If you want the tool to spiral down into the part rather than drop straight down you would not use a **finishing** tool. Instead you would have to use a roughing tool to pocket this arrow because the roughing tool has an option to spiral down into the part. So let's go back to the drawing page and change this arrow to use only a roughing tool.

Using ONLY a Roughing Tool

To let MillWrite know that you do not want to use the finishing tool any longer, enter 0 in that field, or press the Delete key when the cursor is on it. Then click the **Roughing Tool** field to bring up the roughing tool parameters, as seen in Figure 9-5. Many of these fields are the same as they are with the finishing tool. But there are some different fields here. For example, under the **Final Cut Thickness** is a field where you can specify the thickness of material the tool should leave for the finishing tool. Since we will not use a finishing tool and this example, leave this field blank or enter a 0 so that the roughing tool removes all the material.

The very last field is the **Plunge Angle**. Here's where you specify the angle at which the tool will spiral down. A value of 10% will lower the tool gradually. Because the tool spiral down gradually, you do not necessarily need to reduce the plunge feed rate, which is why in this example the plunge feed is set to 100%. After setting these values, click the OK button, and then click the 3D view page to see the result. This time you'll see a corkscrew path, which is the tool spiraling down into the part, as seen in Figure 9-6.

Using BOTH a Finishing and Roughing Tool

The third option with tools is to use both a roughing and a finishing tool. MillWrite assumes the finishing tool is smaller in diameter than the roughing tool. MillWrite will use the finishing tool to clean out the corners that the roughing tool cannot get to.

Go back to the drawing page and pick a finishing tool. The process is exactly as previously described. However, the finishing tool will only be used to clean out the corners that the roughing tool cannot get to, so pick a smaller diameter for the finishing tool.

The finishing tool and the roughing tool need different tool numbers or MillWrite will not insert a tool change in the NC program. In Figure 9-5 the roughing tool has a tool number of 1, so either give the finishing tool some other number, or give the roughing tool a different number.

In Figure 9-5 the roughing tool is not leaving any material for the finishing tool. To get a better finish, enter some small value for this field such as 0.01.

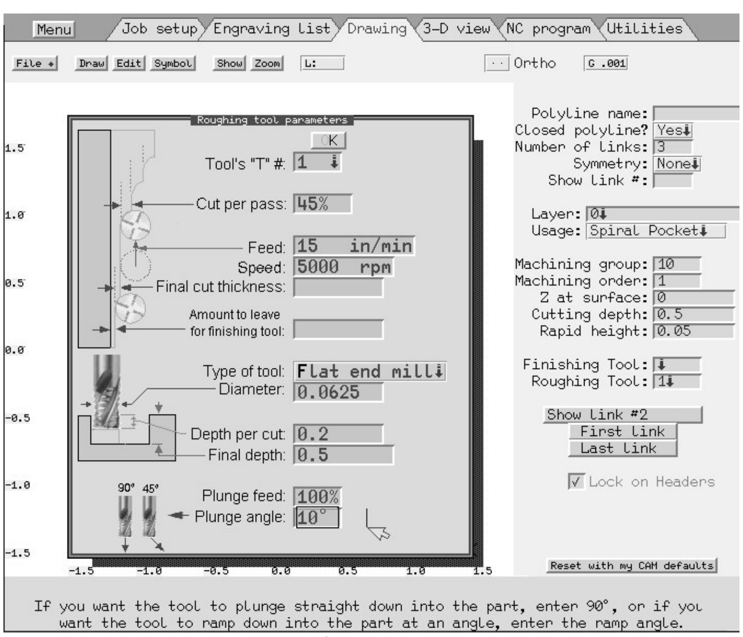

Figure 9-5

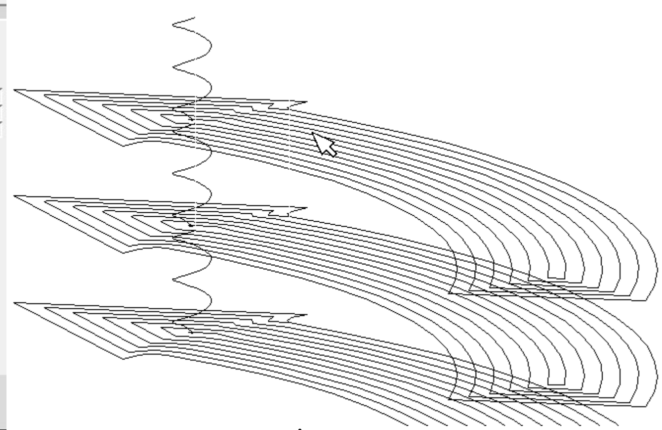

Figure 9-6

Now view the results. If you click the ***Top*** button, you'll get of view of the tool path similar to that in Figure 9-7. On your computer screen the roughing tool and finishing tool will have different colors to make them easy to identify, but in this manual they both look the same, so Figure 9-7 identifies which is the roughing tool and which is the finishing tool. The small sections of tool path in the corners are the finishing tool cleaning out the areas that the roughing tool couldn't get to.

When you return to the drawing page the tool path will still be visible. If you click the **Show** button at the top of the drawing page you can change how the tool path is being displayed.

The last two options in the **Show** menu determine how the tool path is displayed at the drawing page. You can click one or both of those fields. To understand what the options do, first click the **Tool Width In Pockets** option. This creates the display seen in Figure 9-9. Instead of drawing a thin tool path, MillWrite draws a solid tool path that represents the width of the tool. The purpose for this is to make it easy to see which material has not been cut. If all the material has been cut, the tool path will overlap each other and create a solid colored object. The white areas are the areas that have not been cut.

If you bring the **Show** menu back up and this time check the **Tool Diameters In Pockets** option, you'll get the display similar to Figure 9-10. Now MillWrite is showing circles that are the diameter of the tool. The purpose for this is to let you see the relative sizes of the roughing and finishing tools.

Bring up the **Show** menu once more and this time turn off the **Tool Width In Pockets** option. Now you get a display similar to that in Figure 9-11.

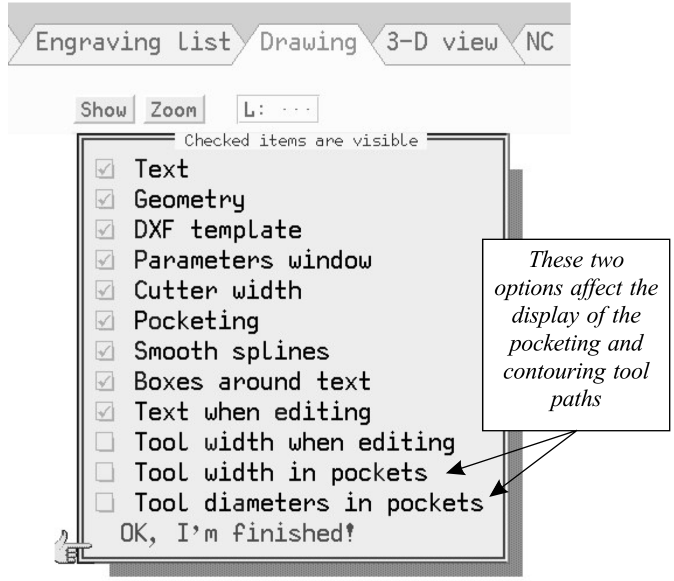

Figure 9-8

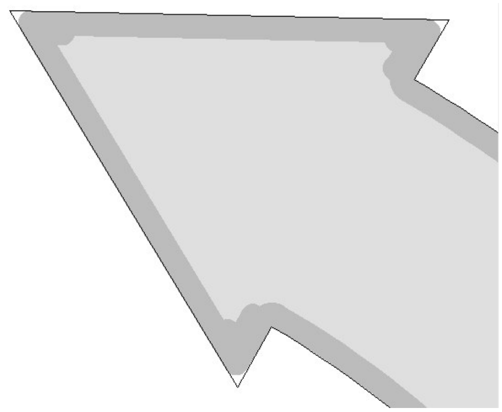

Figure 9-9

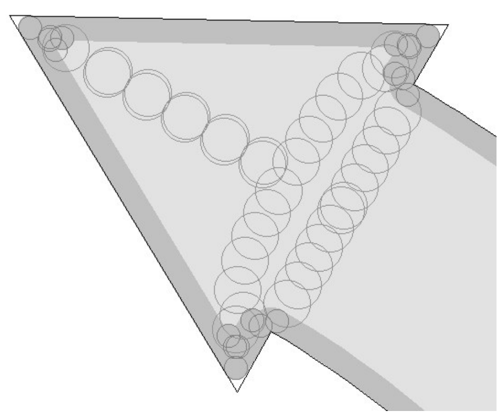

Figure 9-10

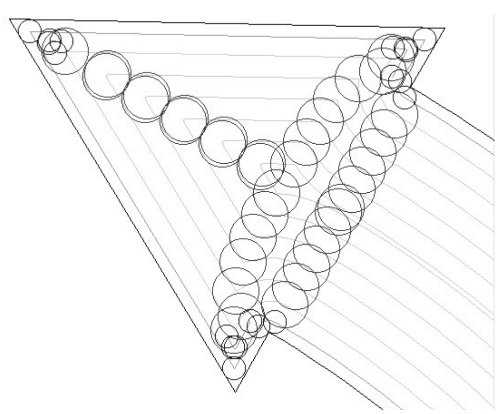

Figure 9-11

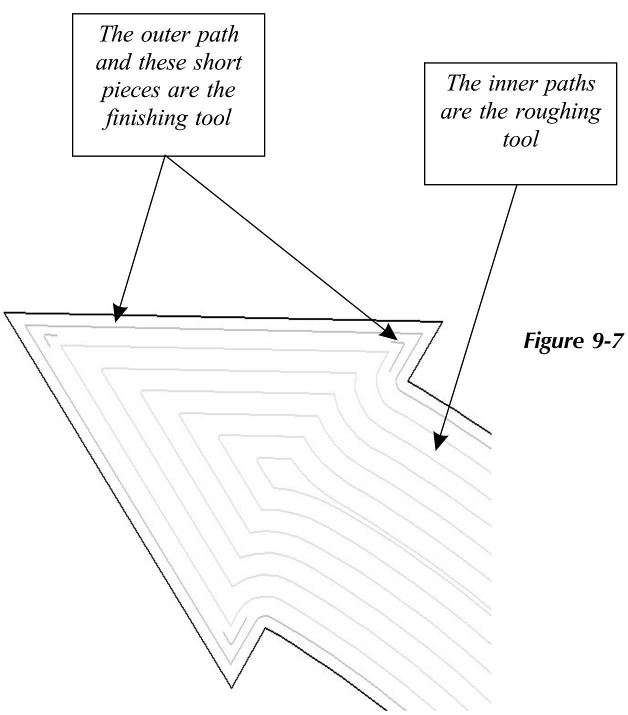

Figure 9-7

Machining Groups

Every item that you set to either engrave, contour, or pocket belongs to a ***Machining Group***. A job can have 32,000 machining groups in it. The groups are numbered from 1 to 32,000.

When MillWrite creates an NC program, it processes all the items in machining group 1, then processes all the items in machining group 2, and so on, up to machining group 32,000.

If you don't specify the machining group an item belongs to, MillWrite will assign it to machining group 10. Since all your items will be in one machining group, they will all be processed in one batch. This is usually acceptable when you are engraving, so for most engraving jobs you can ignore the machining group issue completely. However, when you are pocketing or contouring you may want certain machining events to occur in a certain order, in which case you have to specify machining groups yourself.

Another situation in which you have to specify the machining groups is when you have islands and boundaries on top of each other and MillWrite makes a mistake about which island belongs to which boundary.

You can assign an item to a machining group whenever you see its parameters along the right side of the screen at the drawing page. Figure 9-12 shows the parameters for a polyline. You could click the machining group field and enter a value for it. If you have a lot of items to set however, this is a slow process.

A faster way is to ***select*** all the items that you want in a certain group. The selection function has a button that lets you set all selected items to a particular machining group.

It doesn't matter what number you use for a machining group. The numbers are similar to the line numbers in NC programs; ie, they only show the order the events occur. Also, as with line numbers in NC programs, you should skip a few numbers so that you can later insert another operation between the existing groups. If you don't have any gap in your machining groups, you can pick the menu item to add a gap, but it is easier to leave gaps so you can reduce the time you spend insertng gaps. You can find this function in the **Main Menu** under the **Machining Order/Group** option.

Note about islands: If you set the machining group for an island, you must give it the same group number as the boundary that it is an island for.

If you leave the machining group for an island *blank* or if you set it to *zero*, MillWrite will figure out which boundary it is an island to. This will normally work fine, except in cases where you have complex jobs that have boundaries on top of boundaries, in which case the island may fit into more than one boundary. If you have problems with islands, set the machining group of the island to the same group as its boundary. That will guarantee that MillWrite knows which island belongs to which boundary.

Machining Order

A machining group may have thousands of lines, arcs, etc, to be engraved or contoured or pocketed. The order that those items are machined is important because it affects the time spent by the machine on rapid moves.

You can set the order yourself for any item, or you can let MillWrite set the order. A machining order of 1 cuts first, 2 cuts second, etc, and 32,000 is last. You can set an item to a machining order that is already used; the other items will move aside for it. For example, if you set a circle to a machining order of 4, and if there is already some item with a machining order of 4, it will become a machining order of 5, and the others after it will move down in the machining order sequence also.

If a drawing has lots of items, it is faster to let MillWrite set the machining order. In this version of MillWrite there are two choices for letting MillWrite set the machining order for you:

- *Reduce rapid moves.*
  MillWrite looks at the starting positions for each item in a machining group and sets the machining order to reduce the distance of the rapid moves between items.
- *Reduce rapid moves within each layer*
  This is the same as a first option except that MillWrite processes the items on the layer by layer basis. For example if you have items on layers No. 2 and 3, MillWrite will reorder all the items on layer 2, and after that has been completed MillWrite will reorder all the items on layer 3. This would only be of use to you if you plan to machine the items on layer by layer basis.

You can find these functions in the **Main Menu** under the **Machining Order/Group** option.

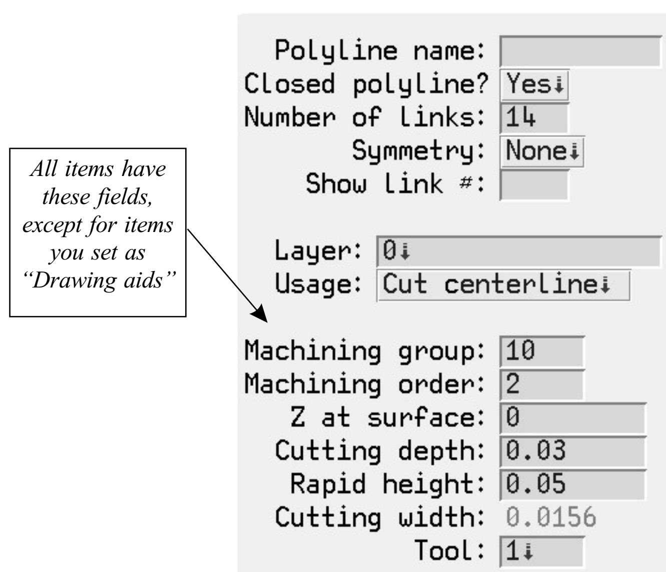

Figure 9-12

Set the NC format

At the **Job Setup** page click the **NC Format** field, as seen in Figure 9-13. A list of NC formats will appear (Figure 9-14). These are samples that you can edit or delete. To delete one, just put the cursor on it and press the Delete key. To edit one, put the cursor on it and move the mouse to the right, or press the right arrow key (ie, →) on the keyboard.

Notice in Figure 9-14 that the mouse has an arrow underneath it that points to the right, and there is another arrow pointing to the right at the edge of the list. These arrows are to remind you that moving the cursor bar to the right will bring up another screen.

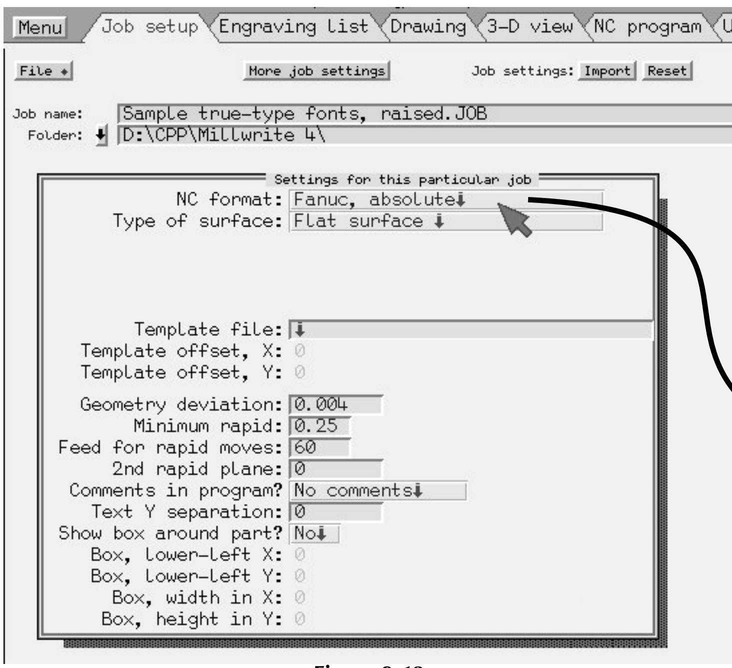

Figure 9-13

Referring to Figure 9-15, some of the fields are:

Name for the spec

The name for the specification is for your use only so that you can remember which machine you set this specification for.

NC Mode and Arcs Centers

If you find your machine it is cutting large circles when it should be cutting small arcs, you set these fields incorrectly. Most machines always use incremental arc centers, but some machines have incremental arc centers only when the X, Y, and Z values are also incremental.

Starting Code

To change how the NC programs start you change the **Starting Code**. Click on the **Starting Code** and a window will appear. Whenever you type in this window will appear at the start of your NC programs. This is where you cancel tool offsets, turn on coolant, or set the program number.

If you specified absolute X, Y, and Z values, add a G90 to the starting code. If you specified incremental values, added G91 instead.

An example of what you might put into the starting code is shown in Figure 9-16. That block of of code wll start the NC program with a **%**, then put in a program number, then specify the tool number.

Because you do not know what the tool number will be when you create these NC formats, use a place holder (a pound sign **#**). When MillWrite creates an NC program it will replace the pound signs with values. The place holders work for the letters T, S, F, and Z. The replacements work like this:

- T#: the # becomes the tool number.
- S#: the # becomes the spindle speed in rpm.
- F#: the # becomes the feed rate.
- Z#: the # will be replaced with the Z height for rapid moves. Therefore, a G0Z# will become a rapid move to a point just above the surface of the part.

Ending Code

Whatever you type in the **Ending Code** field will be placed at the end of the NC program. This is where you put an M30 or an M2. If your machine does not automatically pick up the tool at the end of a program, you could pick the tool up by putting a **G0Z#** in the Ending Code.

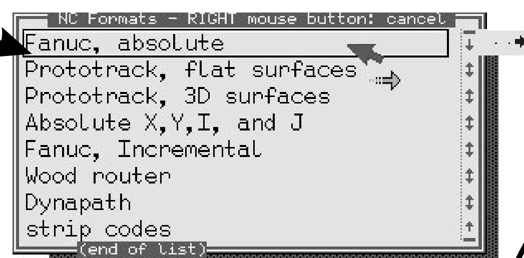

Figure 9-14

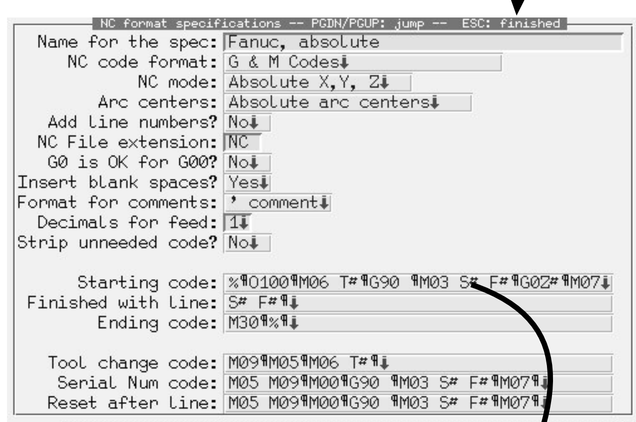

Figure 9-15

```
%
O100
M06 T#
G90
M03 S# F#
G0Z#
M07
```

Figure 9-16