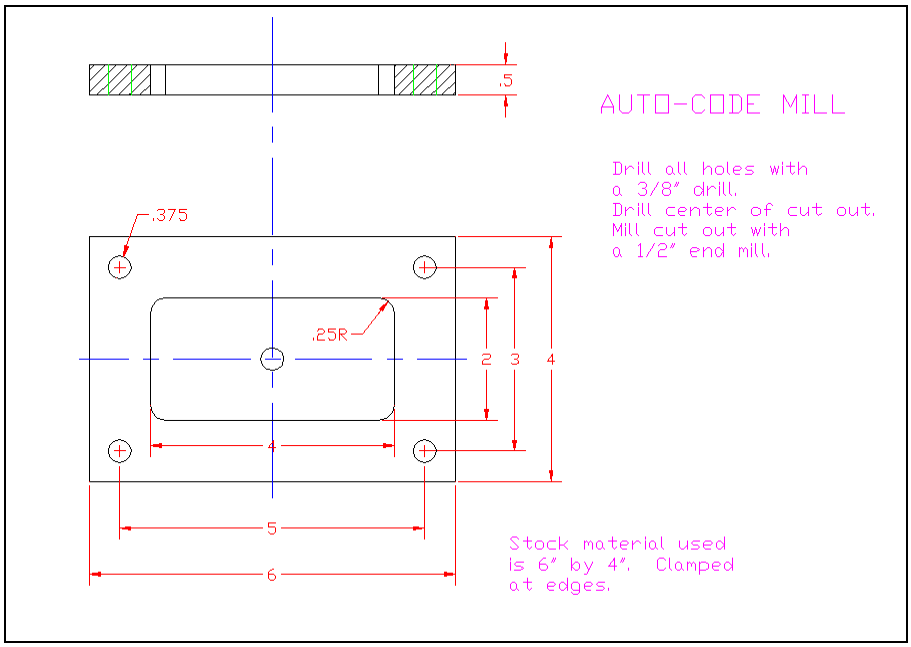
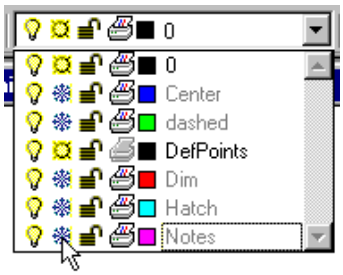
AUTO-CODE Mill Tutorial 1

Using the 3D Mill interface, this tutorial will step you through how to run AUTO-CODE in AutoCAD. This tutorial introduces tool changes, drilling, and simple pocket cutouts.

Load the drawing MILL1.DWG located in the NCG42\CLASS\ directory.



This drawing is ready to use in AUTO-CODE. The part is positioned so that the lower left corner of the rectangle is at 1,1. The contours are drawn as polylines and the drill holes are drawn as circles. All the layers for dimensions and notes can be turned off to reduce the clutter or they may remain on while AUTO-CODE is running. For this tutorial we will be turning them off.



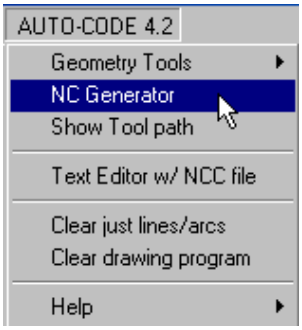
In AutoCAD, select the layer control list and freeze the layers that are not used for programming the part.

To freeze the layers, change the sun icon into a snowflake.

The only layers you will need for this tutorial are 0 and DefPoints. After turning off the extra layers, you should only see the top and side views of the part. We are going to focus on the top view only.

Starting with a 4 by 6 by ½ inch stock material, the part will be clamped along the top and bottom edges. It is not necessary to show the clamps in AUTO-CODE. It is necessary that you know where they are located relative to the drawing.

For this basic part, we are going to drill the five holes indicated by circles and then cut out the rectangular area in the center using a half-inch end mill. The drill holes will be 3/8" (0.375") in diameter. With the drawing open, the next step is to launch AUTO-CODE.



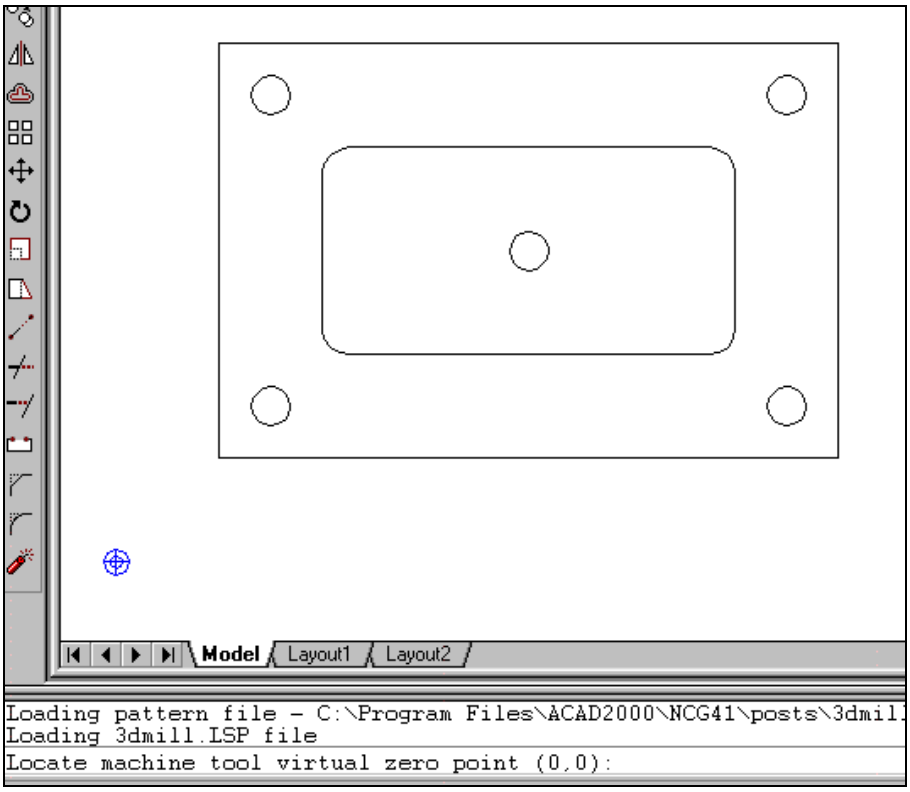
Start AUTO-CODE by selecting the **NC Generator** option from the AUTO-CODE 4.2 pull down menu.

If you do not see the AUTO-CODE 4.2 pull down menu item then AUTO-CODE is not installed on your computer. See the installation troubleshooting section for more help in getting AUTO-CODE 4.2 properly installed on your computer.

AUTO-CODE will load the primary program modules and will then check to see how many post processor options you have in your directories. If there is only one option, it will be selected automatically. If more than one option exists, a list of available options is shown in a standard Windows File Selection dialog box.

Select the **3DMILL** interface and **Open** it.

AUTO-CODE will then ask you to locate the machine zero point.

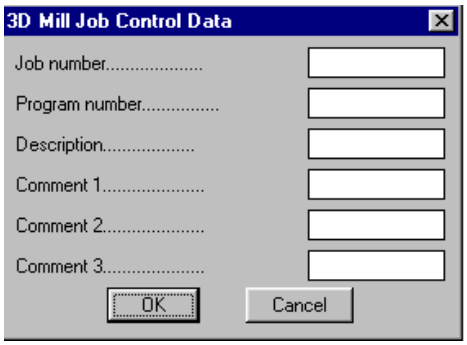


Press the **Enter** key to accept the zero point default at 0,0.

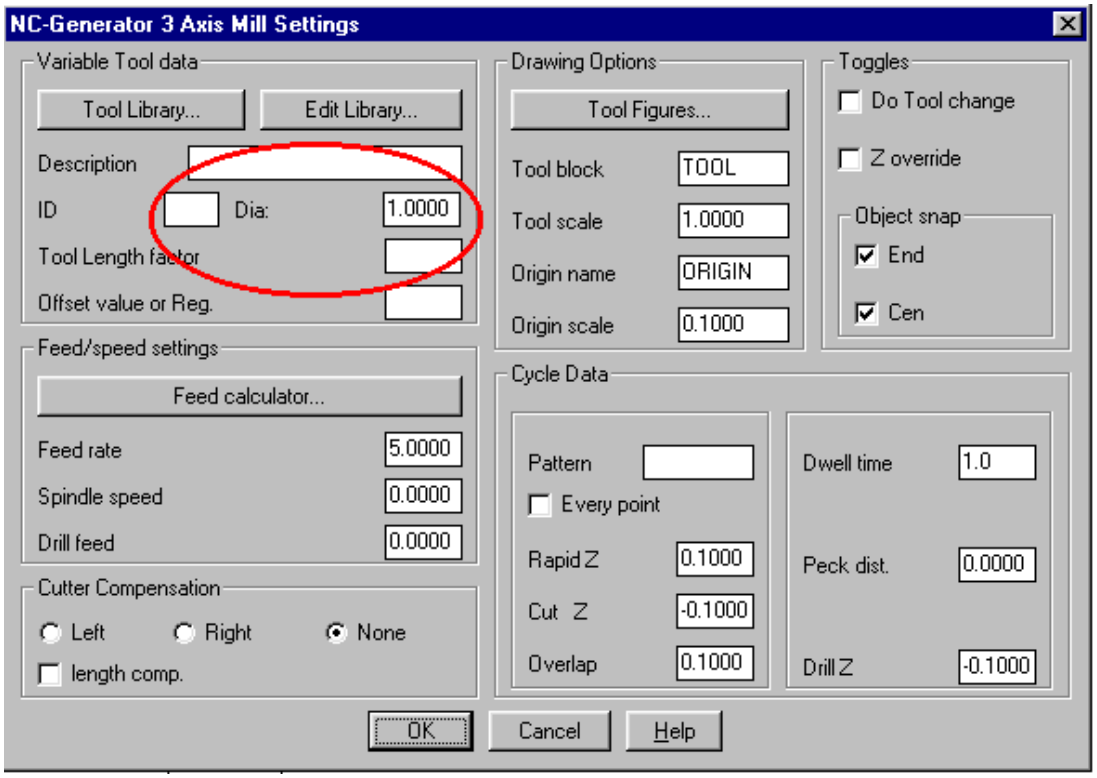
AUTO-CODE's zero point can be shifted by simply moving the Zero point icon as seen in the previous figure. This object is a normal block insert containing attributes informing AUTO-CODE about the drawing and programming environment.

Custom machine interfaces in AUTO-CODE can contain a number of dialog boxes and custom entry points. The 3D mill interface has a basic dialog box appearing at the start that asks about job information.

Note: Those interested in customizing AUTO-CODE are directed to look at the 3DMILL.LSP source file as well as the online customization help accessed through the AUTO-CODE 4.2 Help menu option.



You can fill in this dialog box supplying the program number and description of your choice. When finished, press the **OK** button to move on to the next dialog box.

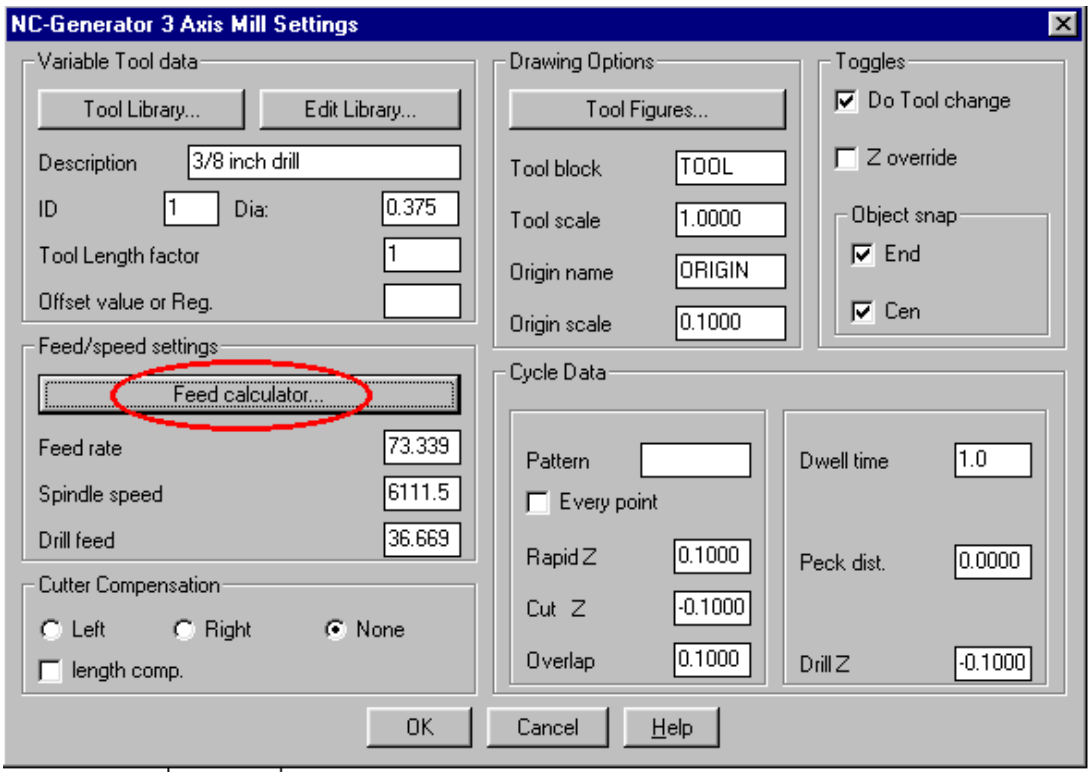


The dialog box shown is for the tool settings. We will define a 3/8-inch drill to use by filling in the data fields from the following table. For this tutorial we will just enter the information for this tool only.

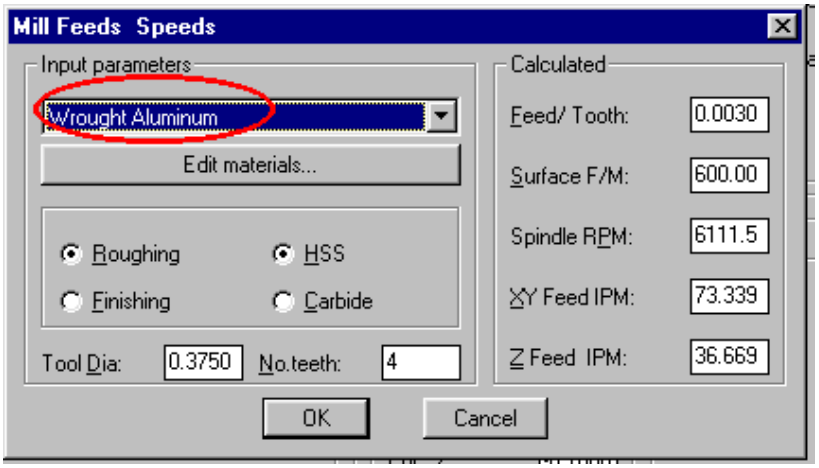
Description:	3/8 inch drill
ID:	1
Dia:	0.375
Tool Length Factor	1

Hint: Use the Tab key to move from one field to the next after typing in the value. It is much faster than typing in the value, then moving the mouse to the next input field.

Press the **Feed Calculator** Button to activate the feed rate calculator.



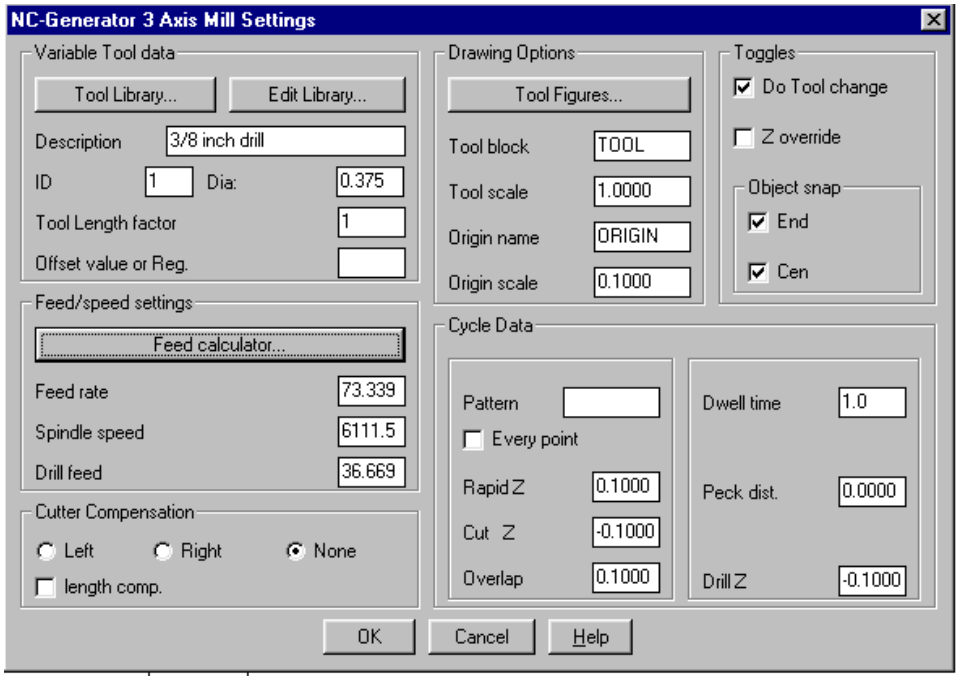
The feed rate calculator has cutting feed information stored on disk that can be pulled up as needed. You can edit the materials list and supply your own names and values by pressing the Edit Materials button.



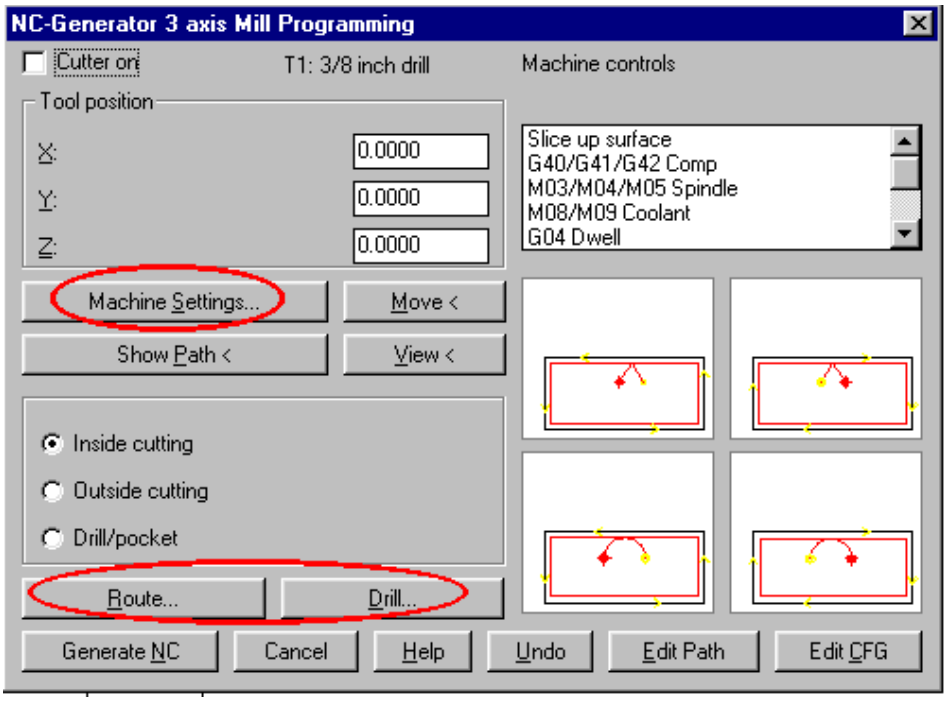
For our tutorial we are working with **Aluminum** and will use the default values as calculated. Press the **OK** button to return to the tool settings dialog box.

The data from the feed rate calculator is placed in the feed and speed settings for the tool.

Note: The values from the feed rate calculator are merely suggestions and should not be used until after you have adjusted them to match your own shop practices. You may need to round the values off as well. In this tutorial we will leave the values as they were calculated.



The remaining parameters are acceptable so the next step is to press the **OK** button to proceed to the AUTO-CODE Main dialog box for 3-axis millwork.



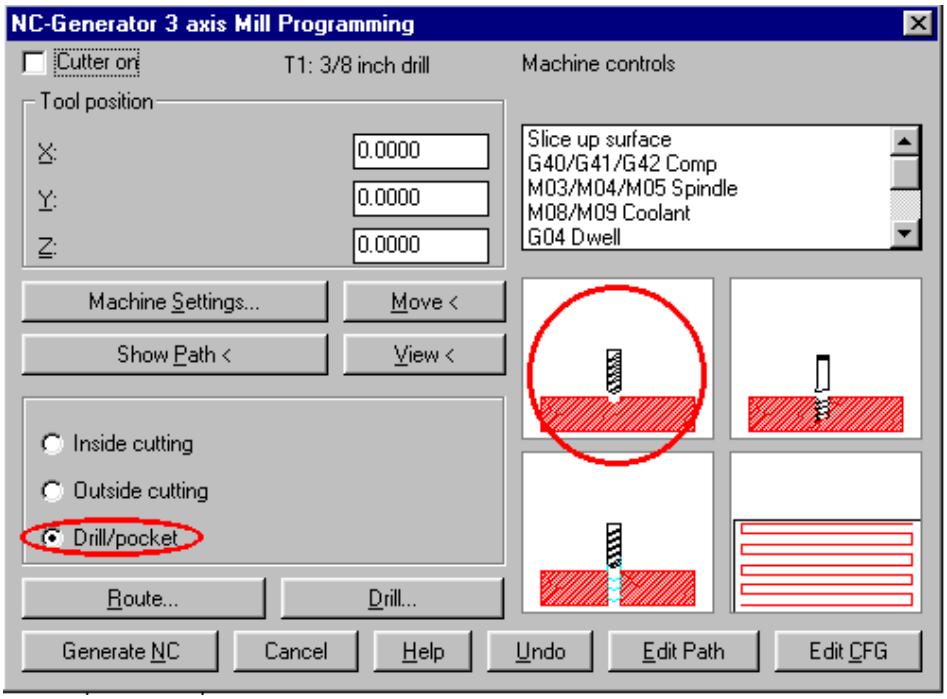
From this dialog box you can change tools, pick a cutting operation to perform, run special machine controls, and do manually defined movements. The current tool selection is displayed up at the top of the dialog box along with the cutter status.

For most programming operations you will not change the values in the upper left of the dialog box. They are provided as reference information. The normal sequence of operations is to select a tool with the **Machine Settings** command button and then perform an operation with the tool such as **Route** or **Drill**.

The icons to the right of the dialog box are quick entries into the route and drill operations of AUTO-CODE. They set up parameters for leaders, drill operations, and for the commands so that you do not have to set them up individually.

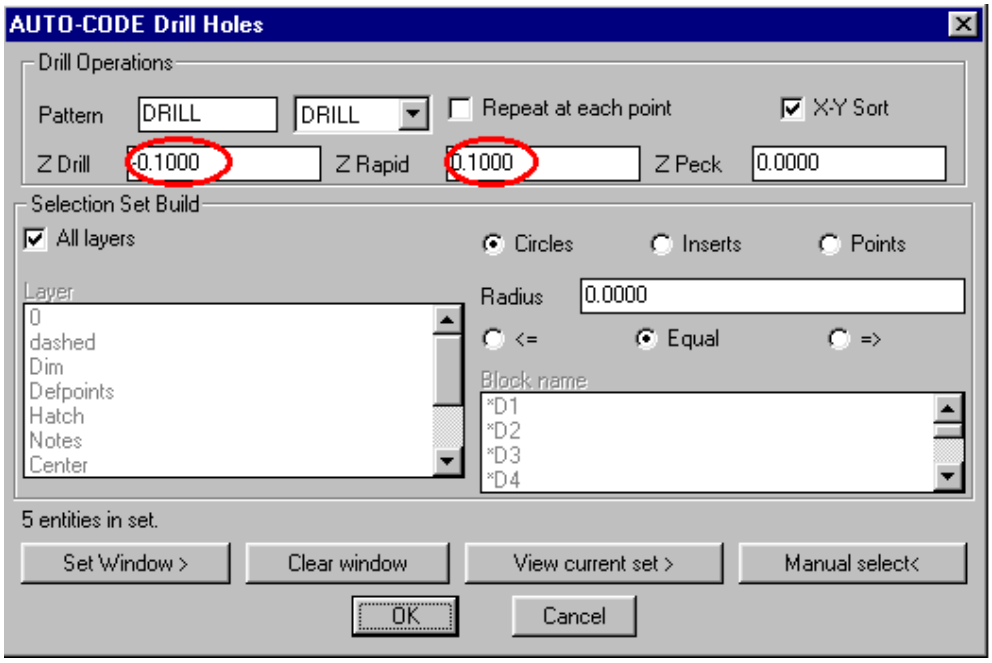
To see how this works, we will continue in the tutorial. Currently, we have a drill tool defined, so the next step is to select the drill options.

Click on the **Drill/pocket** option in the radio button choices to the left of the dialog box. A different set of icons will appear in the graphic windows to the right.



Next select the drill icon (upper left) to display the drill dialog box with the drill options already filled in. The icons present a quick way to launch the Route and Drill dialog boxes with values supplied as defaults. The idea is to provide you with maximum flexibility in using the interface to do the programming you need to do.

 The Drill icon leads to the drill details dialog box. Drilling in AUTO-CODE is simple when the drawing already contains circles marking the locations of the drill centers.



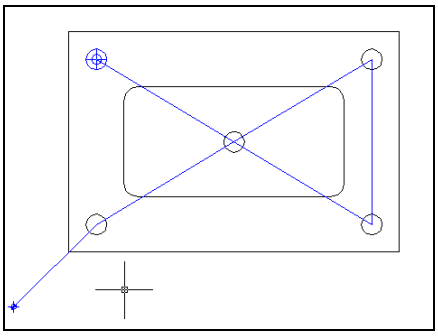
When the drill icon is selected, the drill pattern will automatically be selected. Now we need to correct the Z values for our application.

Z Drill	-0.5
Z Rapid	1.0

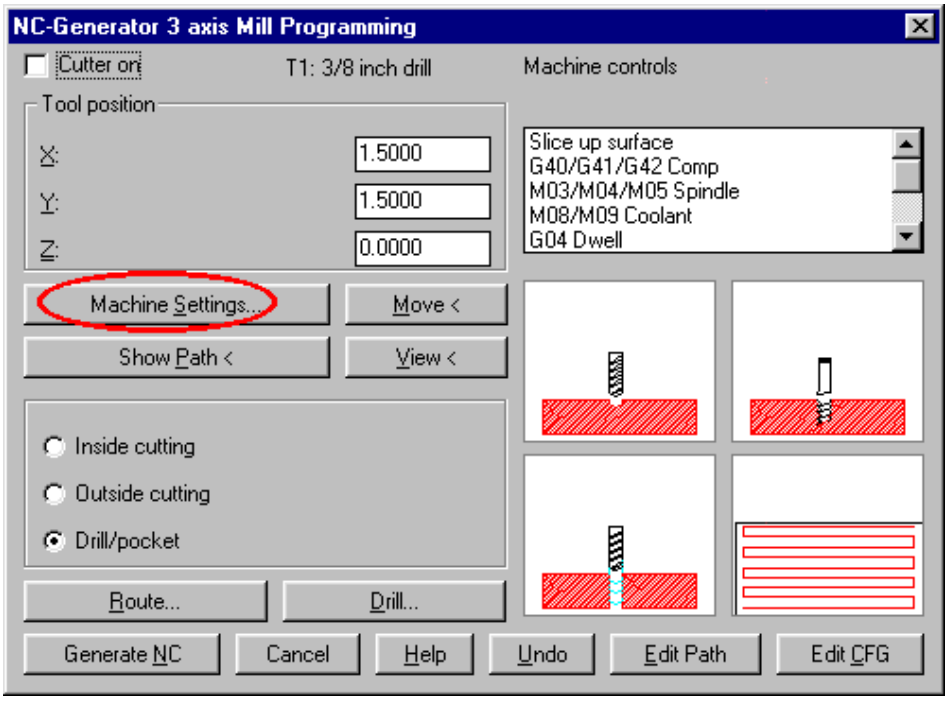
Note that the message in the bottom left of the window indicates that five objects have already been selected. These are the circles in the drawing. The central part of the dialog box allows us to build a selection set of objects for drilling.

For the tutorial, the five circles are exactly what we want. Make sure the X-Y Sort toggle located in the upper right is enabled (a check mark is in the box) and then press the **OK** button.

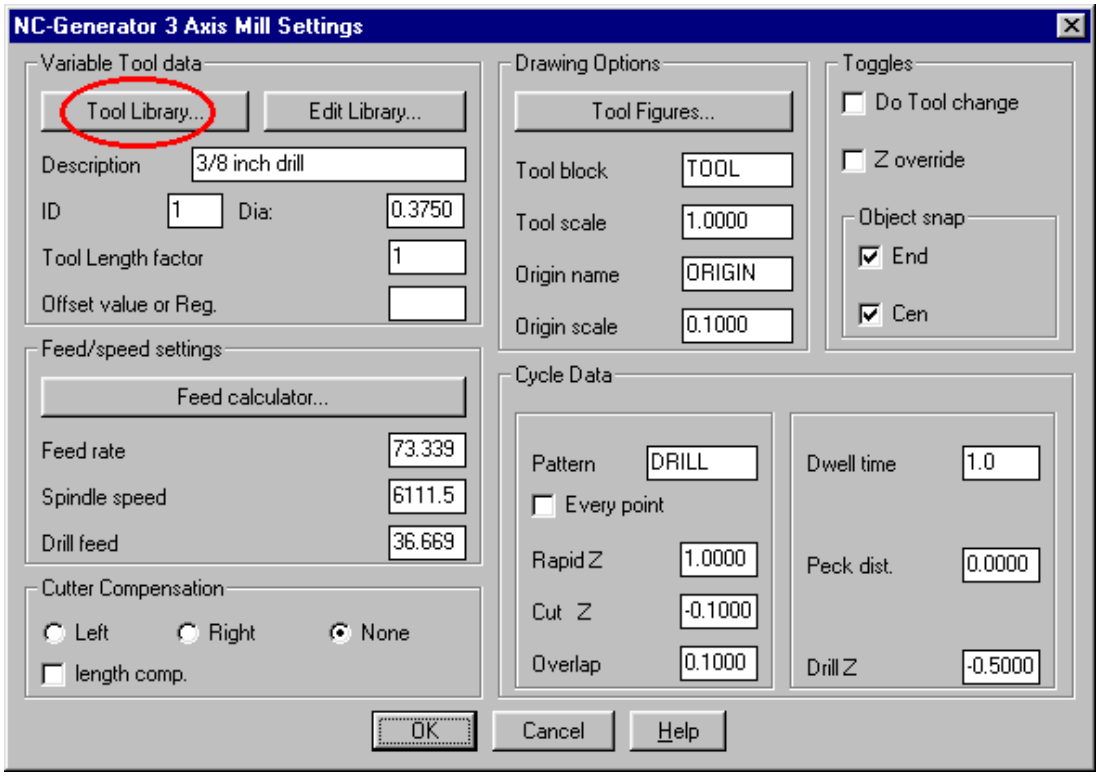
The drill points will be added to the program and control will return to the main dialog box.



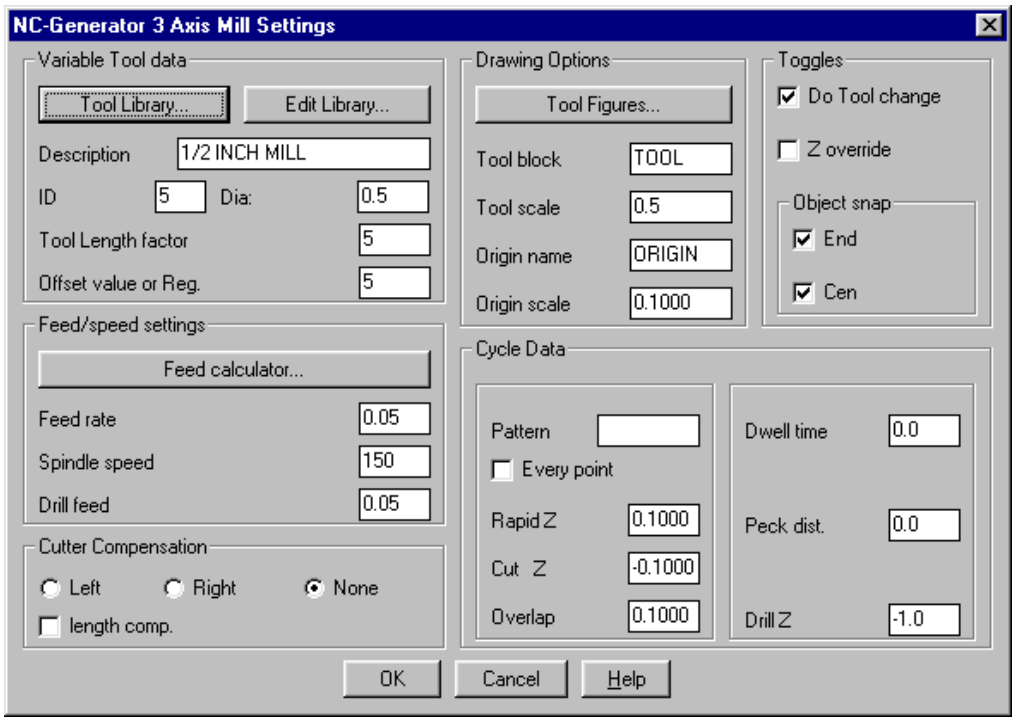
From the main dialog box, select **Machine Settings** to bring up the tool details dialog box.



Click on the **Tool Library** button located on the upper left. A directory listing of existing TL files will be presented. TL files are tool library files. You can create your own using the **Edit Library** button and by supplying the values you want to be made available for various tools.



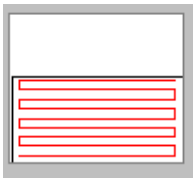
From the list of TL files, select the **END1-2.TL** file. This is a half-inch milling tool defined just for demonstration purposes. After opening the HLF-BALL tool library, AUTO-CODE will insert the values for the tool size and other parameters from the values in the file.



You can have any number of TL files in your AUTO-CODE directory. They can be used to automatically fill in the field values of the settings dialog box.

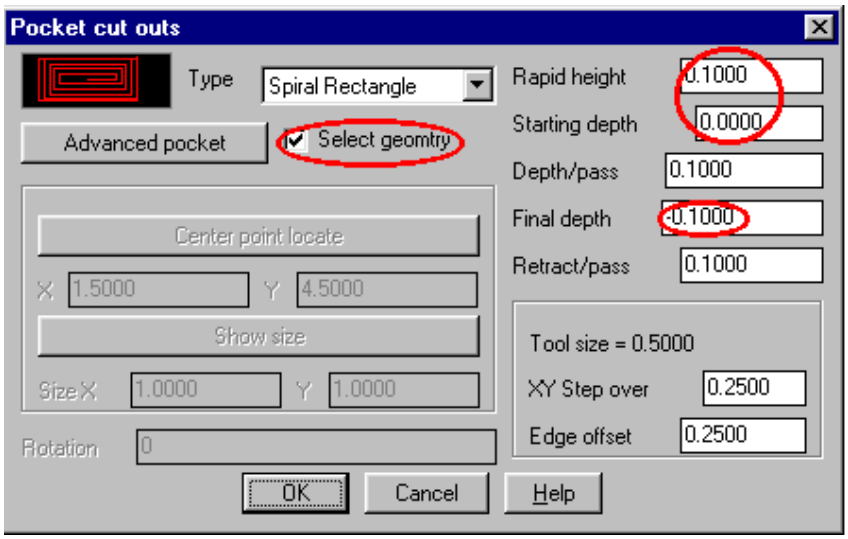
Select the **OK** button to output the code to load the half-inch mill and to return to the main dialog box.

The next operation to be performed is a spiral cut out. We will start at the center and work our way around the geometry to spiral cut the rectangular area in the middle of the plate. AUTO-CODE has a series of pocket cut macros that make this task easy.



Select the Pockets icon from the main dialog box.

This will bring up the pockets dialog box. AUTO-CODE has a variety of standard pocket cut out shapes it supports as well as the ability to create pocket cut out geometry for irregular shapes.



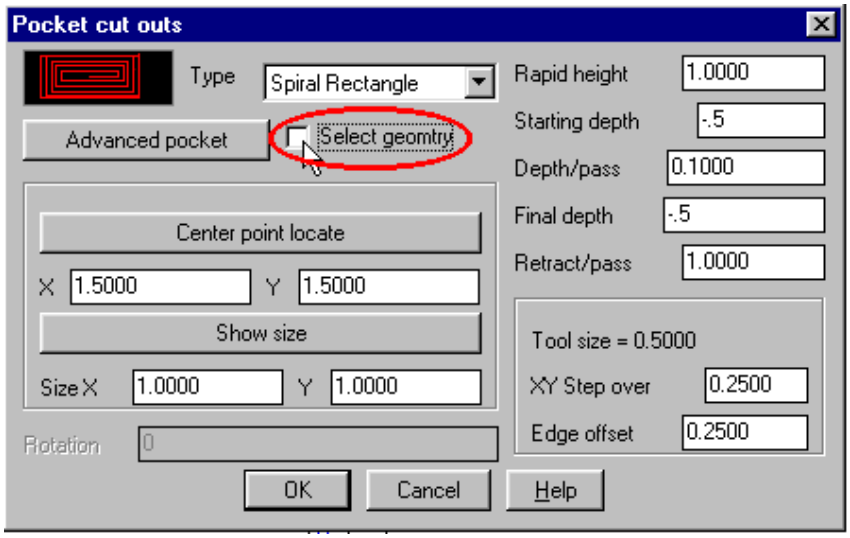
You can select different types of pockets by picking from the pull down list or by picking on the icon to step through the same list.

In the upper right are the pocket cutting parameters. We need to adjust a couple of them to match our machining requirements.

Rapid height	1.0
Starting depth	-0.5
Final depth	-0.5

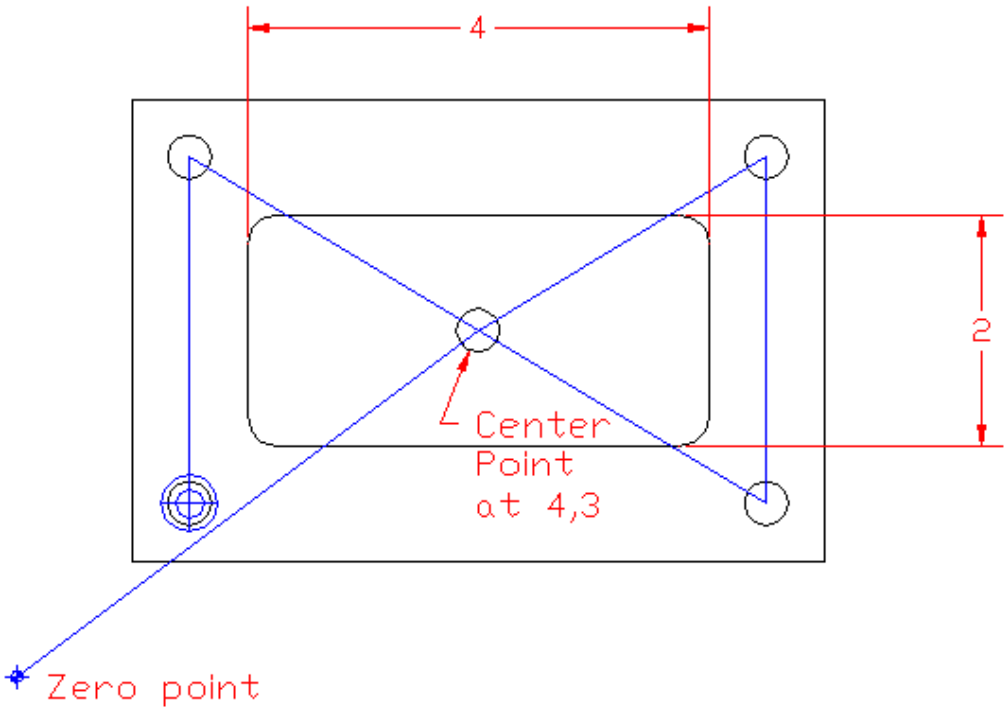
Since the starting and final depths are the same, the process will ignore the Depth/pass and Retract/pass values.

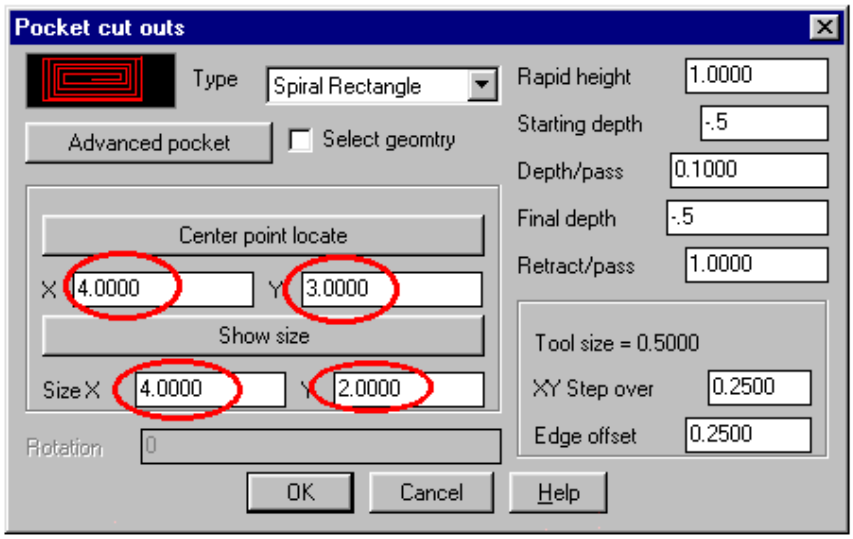
Next toggle the **Select Geometry** option to off so that the bottom left of the dialog box becomes visible.



We can locate the center point and show the sizes of the rectangle cut out. Click on the **Center point locate** button to bring up the graphics screen. Either pick the center point of the cut out area or enter the coordinates of the center point.

Next select the **Show Size** button and either enter the sizes (4 for the X axis, 2 for the Y axis) or show the distances by locating the extreme points of the rectangular area.





Once all the information has been entered for the cut out, select the **OK** button.

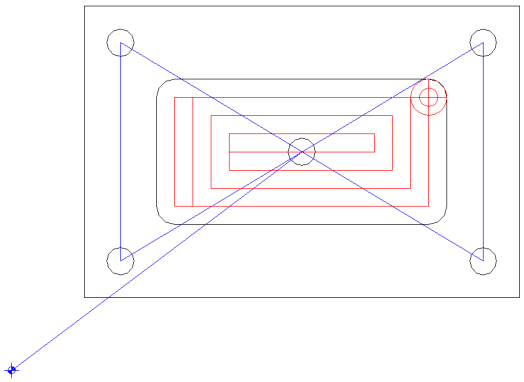
```
Cut direction: CW or <CCW>?
```

The cutting direction will then be requested. For the spiral cut you can opt to cut in a clockwise or counterclockwise direction. Press the **Enter** key to accept the default direction.

AUTO-CODE will generate the tool path to cut out the rectangle.

The cut will start at the center of the rectangle and go across to the left. The cutter will then reverse direction and cut back towards the right, advance for a quarter inch, then cut back to the left. The pattern is repeated in a spiral pattern until the entire rectangle has been cut.

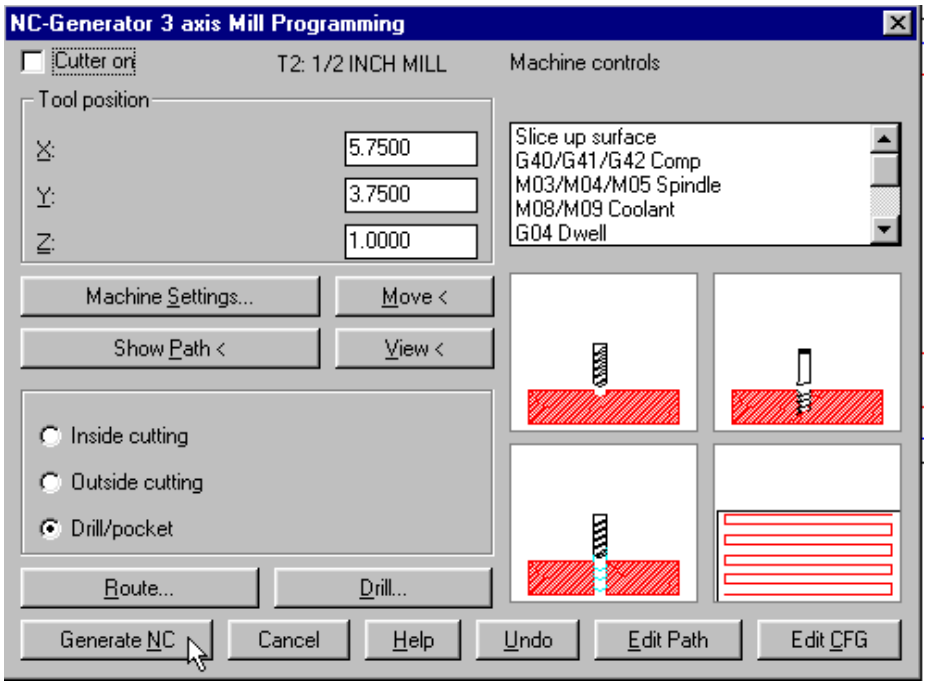
At the end of the cut the tool remains in the part and the operator is given the chance to have the cutter retract.



```
Retract to rapid height? <Yes>
```

Press the **Enter** key to have the tool retract to the rapid height. The pocket cut out dialog box will appear again.

Select the **Cancel** button to return to the AUTO-CODE main dialog box.



Pick the **Generate NC** button to create the NC output code.

The system will ask if you want to rapid to a finishing point. Locate or type the coordinates for the zero point (0,0).

```
Rapid position to finish point <Yes>?
```

Press **Enter** to provide a rapid movement to the end of the program.

```
End or Center point [X,Y,Z] <press Enter to
exit>:0,0
```

Type **0,0** to provide the ending point of the tool after this last operation. You can also show the point graphically on the screen. After typing the zero point, the prompt is repeated. Press **Enter** to move onward.

```
Run ending pattern <Yes>?
```

An ending pattern is provided in the 3D Mill interface. This is the sequence of M and G codes that wrap up a program file. Press the **Enter** key to output these codes at this time. The option is provided so that you can generate partial programs to be inserted into larger existing programs.

A dialog box will be presented for the entry of a file name to output. The default file name will be the name of the drawing in the directory it came from. Click on the **Save** button to use this name.

The NC file will be created in the directory named.

The drawing editor will return and you can now draw other things in AutoCAD, view the tooling path just defined, or view the NC file. To view the file, select the Text Editor option from the AUTO-CODE pull down menu.

Summary:

Tutorial One introduced the basic operation of AUTO-CODE; how to start the program, select or define a tool and how to perform a basic machining operation such as drilling and rectangular pocket cut outs.