NC Programming Lab

Goal

You will create NC words and use those words to create an NC part program from a part drawing. You will verify the NC code tool paths and machine a part on a CNC milling machine.

Materials Required

- Graph paper and an NC coding sheet (provided).
- 2” x 3” raw material stock (provided)
- Milling machine (in lab)
- 1/8” end mill

Activities

- Plot the coordinates for the sample part (USA STAR) on a graph using the part drawing provided. Then, construct an NC program to machine the part, and enter the NC coding for the program in the coding sheet provided. Type the code into a “text” file in NC block format.

Introduction and Steps

There are several steps involved in writing an NC program:

1. Sketch the part
2. Draw the part to scale, on graph paper
3. Determine the stock size and select the tool.
4. Determine the feed rate and depth of cut.
5. Determine how to cut the part.
6. Enter the code on a coding sheet.
7. Transfer this code to a “text” file (use a diskette or USB drive.)
8. Verify that your code produces an acceptable part.

1. Sketch The Part

Your first task is to sketch the part you would like to machine. Sketching the part gives you your first idea of what the part will look like. You should begin to consider the size, or the dimensions of the part. You will also decide what material you will use and how to machine it. (If you are looking at a part drawing, a sketch may not be necessary.) Let us consider creating the NC code for a part called "ABAR" shown in Figure 34.

2. Draw the Part to Scale on Graph Paper

Once you have a visual idea of the part you would like to make, you have to draw the part to scale on graph paper (see Figure 35.) The drawing should show the outline of the workpiece plus a representation of the finished part. Next, you should plot, or locate, the part's coordinates. The coordinates must be precise so you know the part's exact size. Because they are easier to calculate, it is best to use decimals instead of fractions to plot the coordinates; for instance, use .5 instead of 1/2.

Since the workpiece will be placed in the machine's clamp with the longest side along the X axis, the part is drawn in the same manner.
Figure 34. ABAR

This is the sketch for our sample ABAR.NC program.

Figure 35. Sample part plotted on graph paper
3. Determine the Stock Size and Select the Tool

When choosing the stock for a part, always try to use the smallest piece possible to eliminate waste. For the sample part stock, you will be provided material that is approximately a total size of 3" x 2" x 1 ½".

When selecting the cutting tool to use for machining a part, consider the profile of the milled surface you want. Different kinds of end mills produce different surfaces. For example, use center-cutting end mills to cut flat surfaces and start cavities, and ball shaped end mills to cut curved surfaces. After selecting a tool type, the tool size must be determined. As a general suggestion, select the largest size available of an appropriate tool; large tools can remove more material than small tools can in the same period of time. For the sample part tool, we will use a 1/8" diameter center-cutting end mill.

4. Determine the Feed Rate and Depth of Cut

The feed rate depends on the stock material. The sample part shown here was designed to be machined from a soft material. The maximum recommended feed rate for should be no more than 9 ipm (inches per minute). If we had planned to machine the sample part from a harder material, like steel, we would have used a slower feed rate (1 ipm). Use a feed rate of 9 ipm to cut the sample part.

The depth of cut depends on the tool size and the type of material being machined. The depth of cut for soft material should be no more than .2 inches deep. For the sample part, choose to make the cuts at a depth of .15 inch (-.15 on the Z axis). You should be able to find the feed rate, spindle speed and depth of cut information you need in the table below.

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>TOOLS (Inches)</th>
<th>1/8</th>
<th>3/16</th>
<th>1/4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acrylic</td>
<td>6 2000 .06</td>
<td>6</td>
<td>2</td>
<td>6</td>
</tr>
<tr>
<td>Aluminum</td>
<td>3 2000 .03</td>
<td>2.5</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Brass</td>
<td>4 2000 .03</td>
<td>3.5</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>Delrin</td>
<td>5 2000 .1</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Machinable Wax</td>
<td>9 2000 .2</td>
<td>9</td>
<td>9</td>
<td>9</td>
</tr>
<tr>
<td>Steel</td>
<td>1 1500 .015</td>
<td>.7</td>
<td>.015</td>
<td>.5</td>
</tr>
<tr>
<td>Wood</td>
<td>5 2000 .15</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
</tbody>
</table>

This table will help you determine approximate maximum values for feed rates, spindle speeds and depth of cut for different materials and tool diameters on the milling machine tool. Experience and experimentation will help you to choose the best values for your particular needs.

5. Determine How to Cut the Part

Before you can write the NC code, you must determine how you will go about cutting the part. This is a very logical process. You simply map-out each action of the cutter by deciding where to start cutting, where to insert the tool, where to extract the tool, when to move to a new
cutting position, and so on. When you have finished mapping out this sequence, it will be very easy to translate the information into NC code. The following example shows how the cutting sequence for the ABAR program planned; the A-contour was cut first and the bar-shaped pocket second.

Cut the A-Contour (refer to Figure 36.)

- The cutter moves to the Start Point. The cutter makes a rapid traverse movement above the workpiece, not touching the workpiece.
- When the cutter reaches the correct X and Y coordinates above point A (X1, Y.5), the spindle turns on.
- The cutter is inserted into the workpiece at point A to the depth (Z-.15) we decided on earlier.
- The cutter moves according to the coordinates on the scaled drawing. It moves up the leading edge of the A-contour to point B (X2.5, Y1), then down the trailing edge to point C (X1, Y1.5).
- The cutter lifts out of the workpiece (Z .1).
- The cutter moves to the next insertion point. The cutter makes a rapid traverse move to the X and Y coordinates of point D (X1.6, Y1.3)
- The cutter is inserted into the workpiece (Z-.15) to the same depth as before.
- The cutter moves to point E (X1.6, Y.7).
- The cutter lifts out of the workpiece (Z .1).
- The cutter moves to point F (X .7, Y1.5). Point F is the start point for the bar-pocket.

Figure 36. Producing the A-countour

Cut the Bar-Pocket

Note: in cutting the pocket, you have to make sure that you overlap each pass and not leave material between the passes. The incremental movement must be calculated and depends on the cutter diameter. The maximum movement should be equal to the cutter diameter.

- The cutter is inserted at point G (Z-.15).
• The outline of the pocket is cut. The cutter moves from point G (X .7, Y1.5) to points H (X .5, Y1.5), J (X .5, Y .5), K (X .7, Y .5) and back to G.

Figure 37. Pocket outline

• The cutter moves to point L (X .6, Y1.5).
• The cutter moves to point M (X .6, Y .5), removing the center of the pocket.
• The cutter lifts out of the workpiece (Z .1).
• The cutter returns to the original start point and the spindle is turned off. The program ends here.

Figure 38. Pocket middle

6. Enter the Code on a Coding Sheet

Now that we have the information necessary to mill the part (the stock size, the part coordinates, the feed rate, the cutting sequence), we can use this information to create the NC program codes and write the codes on an NC coding sheet.

An NC coding sheet is a simple form for writing NC code. After the code is written on this sheet, it is used as a guide for entering the code into the computer. The instructions on the coding sheet should be in this order:

1. Move the tool to its starting position.
2. Specify the programming mode (absolute or incremental).
3. Turn on the spindle.
4. Specify the feed rate.
5. Perform each cut and rapid traverse motion according to the cutting sequence and coordinates on the graph paper drawing.
6. Return the tool to the starting position.
7. Turn the spindle off.

The coding sheet for the ABAR program is shown in Figure 39. All the necessary address characters are given as column headings. Each parameter is placed under its appropriate heading.

For example, the first block in the program has a zero under the N heading, the numbers zero and 90 under the G heading, and a value of .1 under the Z heading. The comment "Rapid, Absolute, To Z Start Point" is under the Comments heading as a reminder of what we want this block of NC code to do.

So, the first block of NC code reads:

NOG0G90Z.1; MOVE TOOL AWAY FROM WORKPIECE

When writing an NC program, there are a few things to remember:

- To make a strictly horizontal movement with the tool, place the X or Y coordinate in a block separate from the Z coordinate.
- To make a strictly vertical movement with the tool, place the Z coordinate in a block separate from the X and Y coordinates.
- To move the tool in a diagonal line, place both the X and Y coordinates in the same block.
- To make rapid traverse moves, the cutter must first move in the +Z direction, away from the workpiece.
- Before cutting, the spindle is turned on and the cutter is inserted into the workpiece in the -Z direction.

Remember: It is very important to remember the positive and negative Z axis directions and to be aware of the tool's position at all times when milling. If the Z coordinates in your program are incorrect, the cutter could plunge into the workpiece, possibly breaking the cutter and damaging the workpiece as well.

Now that you know how to create a part program, you can try writing one on your own. Provided is a part drawing with dimensions, a graph to plot the coordinates on, and a coding sheet. Your task is to transfer the drawing to graph paper, plot the coordinates, determine the cutting sequence and write the NC program.

Do not worry; it is not as hard as it looks. Here is a clue: you only have to cut five straight lines to mill the star, without lifting the cutter out of the workpiece until you're done, then cut the letters.
**Figure 39. Sample worksheet for ABAR**

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
<th>S</th>
<th>M</th>
<th>COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0.90</td>
<td></td>
<td>.1</td>
<td></td>
<td></td>
<td>2000</td>
<td>MOVE TOOL AWAY FROM STOCK</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>3</td>
<td>3</td>
<td>SPINDLE SPEED 2000 RPM, SPINDLE ON</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>1</td>
<td>.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>MOVE TO POINT A</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>1</td>
<td>-.02</td>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td>LINEAR, INSERT TOOL, RATE 6 IPM</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>2.5</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT B</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>1</td>
<td>1.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT C</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>0</td>
<td></td>
<td>.1</td>
<td></td>
<td></td>
<td></td>
<td>EXTRACT TOOL, RAPID TO POINT D</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td>1.6</td>
<td>1.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>1</td>
<td></td>
<td>-.02</td>
<td></td>
<td></td>
<td></td>
<td>LINEAR, INSERT TOOL</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td>.7</td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT E</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>0</td>
<td></td>
<td>.1</td>
<td></td>
<td></td>
<td></td>
<td>EXTRACT TOOL, RAPID TO POCKET</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>.7</td>
<td>1.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td>1</td>
<td></td>
<td>-.02</td>
<td></td>
<td></td>
<td></td>
<td>LINEAR, INSERT TOOL AT POINT G</td>
</tr>
<tr>
<td>13</td>
<td></td>
<td>.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT H</td>
</tr>
<tr>
<td>14</td>
<td></td>
<td>.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT J</td>
</tr>
<tr>
<td>15</td>
<td></td>
<td>.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT K</td>
</tr>
<tr>
<td>16</td>
<td></td>
<td>1.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT G</td>
</tr>
<tr>
<td>17</td>
<td></td>
<td>.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT L</td>
</tr>
<tr>
<td>18</td>
<td></td>
<td>.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CUT TO POINT M</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0</td>
<td></td>
<td>.1</td>
<td></td>
<td></td>
<td></td>
<td>EXTRACT TOOL, RAPID TO START</td>
</tr>
<tr>
<td>20</td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>5</td>
<td></td>
<td></td>
<td>SPINDLE OFF</td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td>END OF PROGRAM</td>
</tr>
</tbody>
</table>

Sheet 1 of 1
USA Star

We have designed this part to be milled on a 3" x 2" x 1 ½ " piece of stock using a 1/8" end mill. The stock material is a soft material. Determine a feed rate, spindle speed and depth of cut. Note the origin and the orientation of the X and Y axes.

A graph and coding sheet are provided for you to draw the part. You may wish to begin by placing a small dot at the coordinate for each point on the star, and writing in the coordinates. Then you can draw the cutting lines from dot to dot. When you've finished, do the same for each letter.

When you plan the sequence for each cut, do not forget to include the Z axis moves, when to insert and when to extract the cutter, when to turn the spindle on and off, and when to set the feed rate.

After your coding is complete, you must create a computer file containing the blocks of NC code and make sure to save this as a “text” file. A MS Word document file will not work. MS Notepad creates text files. If you use MW Word, you can create the text file by choosing “txt” option in the Save dialog box.
Figure 41. NC graph paper
Figure 42. NC coding sheet