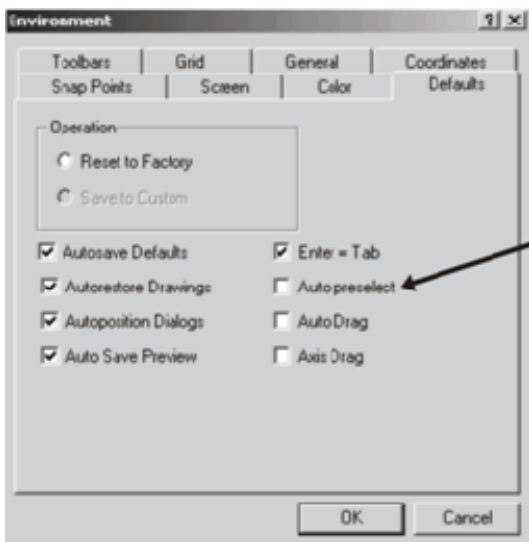


BobCAD-CAM V20 TAPERED WALL POCKETING

This lesson is going to teach you how to use the Version 20 system for tapered wall pocketing.

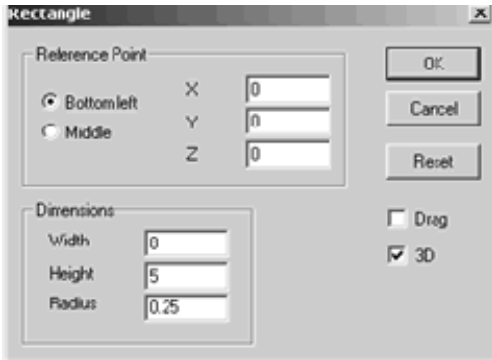
NOTE: BEFORE DOING THIS LESSON, PLEASE TURN THE “AUTO-Preselect” option OFF. Do this by selecting the FILE menu and then ENVIRONMENT. Choose the DEFAULTS tab and un-check this option. Thank you.



By un-checking the Auto Preselect option, the geometry You draw will NOT appear selected.

STEP 1

Start this lesson with a NEW (blank) drawing screen. Go directly to the OTHER menu and select RECTANGLE which is the first item of the OTHER menu.



Make the following selections and entries:

- Select BOTTOM LEFT as the reference point.
- Leave X, Y and Z all set at 0.
- Un-check the DRAG option. It is OK if the 3D option is checked even though we did not enter a Z value.
- Under DIMENTIONS enter 8 for the WIDTH (this is along the X axis), enter 5 for the HEIGHT (this is along the Y axis) and .25 for the Radius as we want .25 on all for corners of this geometry.

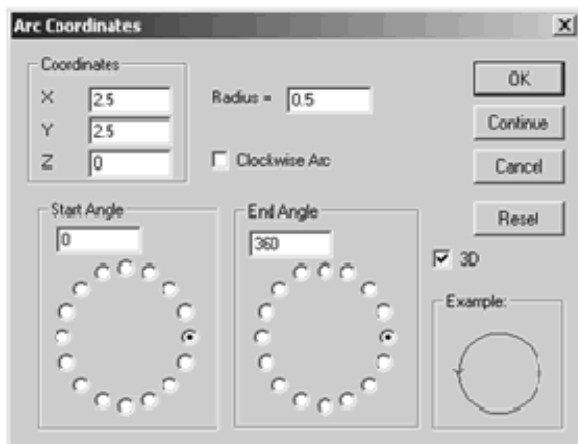
Click OK to exit the rectangle box and draw this geometry on the screen.

Now click on the VIEW ALL icon which is located along the TOP toolbar of the screen. This is the icon that features a magnifying glass with the letter “A” in it.



STEP 2

Now we are going to draw 2 circles that we will be using as our islands. Go to the ARC menu and select COORDINATE CENTER.



Do the following:

- Enter 2.5 for X.
- Enter 2.5 for Y.

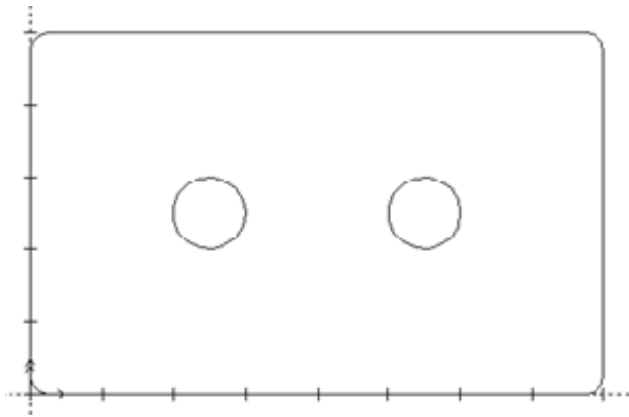
- Z should be 0.
- Enter .5 for the Radius.
- Enter a Start Angle of 0.
- Enter an End Angle of 360.

Click CONTINUE.

- Enter 5.5 for X.
- Leave Y at 2.5.
- Z should still be 0.
- Leave the Radius at .5.
- The Start and End Angles should be left the same.

Click OK.

The result:

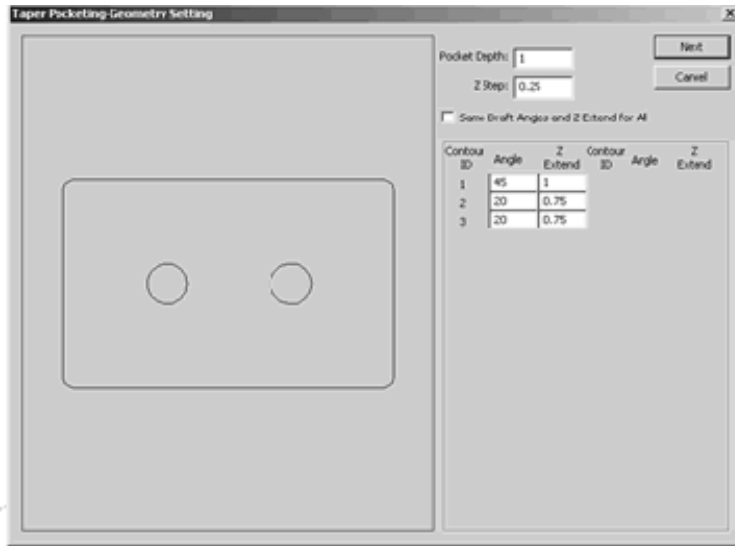


STEP 3

Go to the OTHER menu and select POCKET. The Pocket options box will appear offering 5 different styles of pocket toolpath. Select **TAPER SPIRAL** by clicking on the icon/button.

You will be immediately prompted by the software to select the pocket contour and any islands. Click OK to close this box and then click on the **OUTSIDE** contour **FIRST**. Simply place your cursor anywhere on the outside geometry and left-click your mouse. Then click on both circles as they are the islands.

When you have done this, **RIGHT-CLICK** your mouse to open the Tapered Pocketing box.



This new box will allow you to control the following:

- How deep the pocket will be entirely.
- The size of each toolpath step going down toward the bottom.
- The tapered angle of the outside walls as well as the depth that the taper will begin and/or end which is the “Z Extend” option.
- The tapered angle of each island that you may have and at what depth the taper will begin and/or end which is the “Z Extend” option.

You can see in the image above that the geometry that you have on the CAD screen will be shown. The Angle and Z Extend options in this box will reflect the sequence in which you selected them originally. In other words, the outside contour will always be listed as “1” in the angle/z extend section and so on.

NOTE: You do NOT need to put a minus symbol (-) next to the POCKET DEPTH value nor do you need to do this for the Z STEP value. If the draft/tapered angles are going to be the same for the outside contour and for any islands, check this option.



STEP 4

Make the following entries:

- The POCKET DEPTH is 1.
- The Z STEP is .25.
- Under Contour ID, the 1st angle will be 45, the 2nd angle will be 20 and the 3rd angle will be 20 as well.

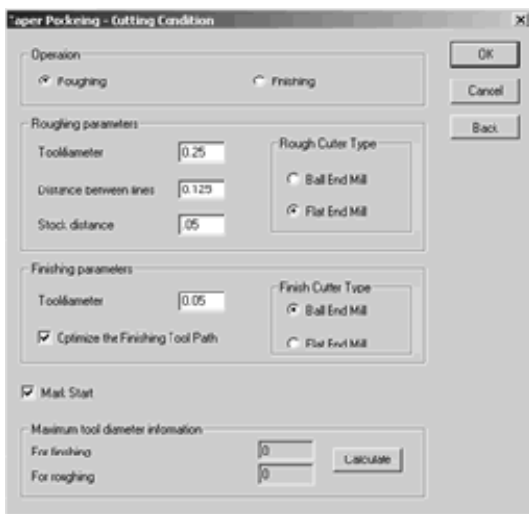
Contour ID	Angle	Z Extend	Contour ID	Angle	Z Extend
1	45	1			
2	20	0.75			
3	20	0.75			

- The 1st Z Extend will be 1. This will apply a 45 degree taper for the full depth of the part.
- The 2nd Z Extend will be .75. This means that we will pocket down .25 without a taper and then the rest of the island wall will have a 20 degree taper to it.
- The 3rd Z Extend will also be .75. Once again, this means that we will pocket down .25 without a taper and then the rest of the island wall will have a 20 degree taper to it.

Now click NEXT.

STEP 5

Now you will have the Taper Pocketing box on your screen. First off, select the ROUGHING option under “Operation” as we will be roughing out this pocket first. Under ROUGH CUTTER TYPE, select Flat End Mill.



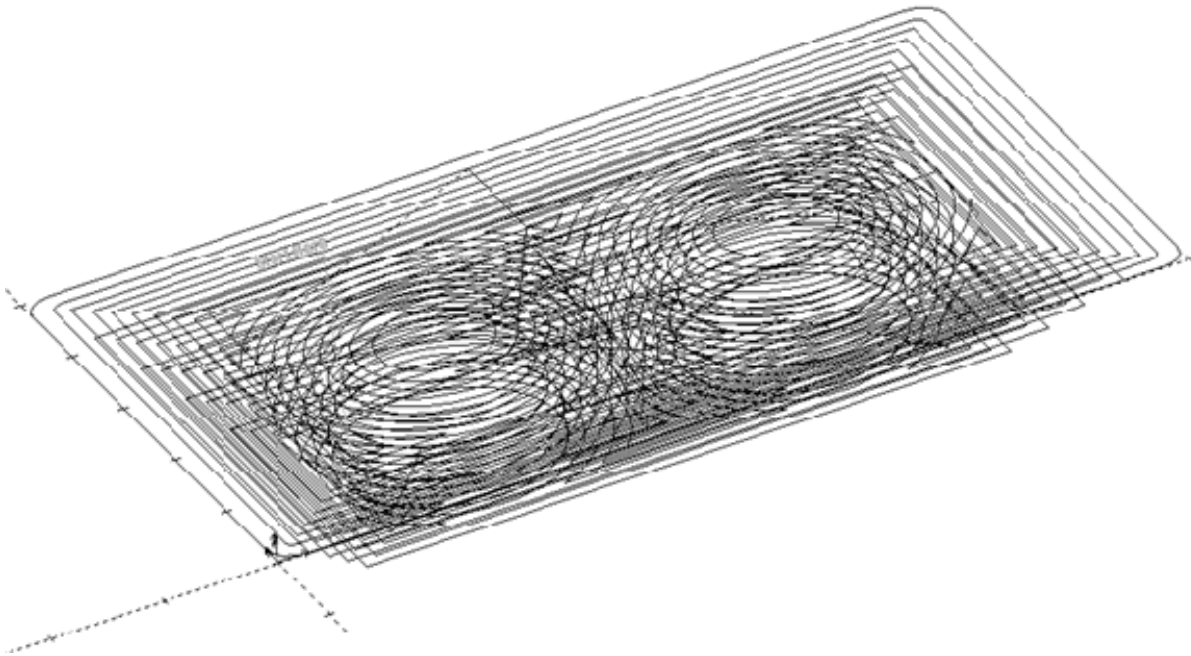
Under Roughing Parameters enter the following:

- Enter .25 for the Tool Diameter.
- Enter .125 for the Distance Between Lines as this is our tool step-over. This compensates for the tool.
- Enter .05 for the Stock Distance as we want to leave some stock around the walls for our finish pass.

Now click the OK button to generate the toolpath for this program.

Now go to the 3D menu and **Turn 3D ON**.

The result:



STEP 6

Go to the Special/NC-CAM menu and choose INSERT NC. The Insert NC Object box will appear offering available post processor configurations. Click on the post called, “Fanuc 6M” and then click OK to open the CAM side of the software. In BobCAD-CAM software, the CAD is always on the left and the CAM will always open on the right side.

On the CAM side, you will see a tool/icon bar that runs down the wall. Locate the 3D button and click on it so that this button is selected. We’re doing this as this is now designated as 3D toolpath in the software.



OK, let’s take a look at this toolpath on your screen. Notice the green vertical line that also says “start here.” We now need to perform a chain selection starting at the top of this green line. Do not click on “Start Here.” You will actually be clicking on the green line itself.

Do this now by going to the EDIT menu, then Select Entities and choose CHAIN. Now place your cursor right on that green vertical starting line and click on it **ONCE**. Now move your cursor towards the **BOTTOM** of that line and you will see that the directional arrow on the line itself will be pointing downward. Once you have that directional arrow, at the midpoint of the line pointing down, click your **LEFT** mouse button **ONE** more time and then hit the F3 key on your keyboard to select the entire toolpath properly. You see, the toolpath is also a chain and this is why we have chosen to use the chain selection feature.

STEP 7

Now that the toolpath is selected and ready to be machined, double check that you have clicked on the 3D button on the **CAM** side wall. Good, now go to the **TOOL** menu on the CAM side and choose **TOOL CHANGE**.



In the Next Tool Number space, enter 1. For the Tool Description enter .25 and then click OK. This will add the codes to NC Editor. Now select the **U/D** button from the CAM wall.



This opened the **TOOL DEPTH SETTINGS** box. All we are concerned with for this operation is the **RAPID PLANE** and **Material Top**. Enter .25 for the **RAPID PLANE**, which is our clearance. Enter 0 for the **Material Top** and leave the cutting depth at 0 as well as we have already entered this in the wizard. Make sure the **Enable** check-box for **Automatic Roughing** is not checked and click **OK**.

Now go directly to the **MACHINE** menu on the CAM side and select **AUTO**. This will produce the complete G-Code program for the roughing.

We're not done. Move your cursor to the CAD screen where you see the words, "Start Here" and while holding down your **CONTROL** key (Ctrl), click on "Start Here." Now the toolpath and "Start Here" will be selected in red (unless you have clicked your mouse anywhere else). With these entities and text selected, go to the CAD side **CHANGE** menu and choose **ATTRIBUTES**.

STEP 8

Select the **GENERAL** tab and under "Layer" type in **Roughing Toolpath** and click **OK**. Now, before clicking anywhere else, go to the **CHANGE** menu again and select **BLANK** to blank this layer out and off of the drawing screen.

You should now have the original geometry on the screen.

sure the Mark Start option is checked and then go ahead and click OK to create the toolpath on the screen.

STEP 10

Now we want to check a few things. First, make sure that the 3D button on the CAM wall is selected. Second, click on the U/D button and ensure that your RAPID PLANE is set at .25 still and that Material Top is 0 as well as the Cutting Depth. We are not concerned with these as we already told the software what our depth is and we have already told the software what our Z-Step is. Click OK to exit the Tool Depth Settings Box.



Now go to the Tool menu on the CAM side and select TOOL CHANGE. Now enter 2 for the NEXT TOOL NUMBER and .25 BALL for the Description. Then click OK to insert the tool change into the code.

STEP 11

OK. Now we are ready to select this toolpath and proceed with the finish. Do this now by going to the EDIT menu, then Select Entities and choose CHAIN. Now place your cursor right on that green vertical starting line and click on it **ONCE**. Now move your cursor towards the **BOTTOM** of that line and you will see that the directional arrow on the line itself will be pointing downward. Once you have that directional arrow, at the midpoint of the line pointing down, click your **LEFT** mouse button **ONE** more time and then hit the **F3** key on your keyboard to select the entire toolpath properly. You see, the toolpath is also a chain and this is why we have chosen to use the chain selection feature.

IMPORTANT: *At this time look to see what the number is for the last line of G-Code and note this on a piece of paper. This is important as we will be using this information when setting up the tools for solid simulation at the end of this program.*

```
BobCAD20.0 - [NC for Fanuc 6M in Tape... X
File Edit Machine Contour Cycle Tool Macro
Setup Window Help
N5940G2X3.4207Y1.55I-1.323J0
N5950G1X4.5793
N5960G2Y3.45I0.9207J0.95
N5970G1X4.0914
N5980G1Y3.575
N5990G1X3.0287
N6000G2X3.698Y2.5I-0.5287J-1
N6010G2X3.0287Y1.425I-1.198J
N6020G1X4.9713
N6030G2Y3.575I0.5287J1.075
N6040G1X4.0914
N6050G1Y3.7
N6060G1X1.3
N6070G1Y1.3
N6080G1X5.7
N6090G1Y3.7
N6100G1X4.0914
N6110G1Y3.825
N6120G1X1.175
N6130G1Y1.175
N6140G1X5.825
N6150G1Y3.825
N6160G1X4.0914 G40
N6170G0Z0.25
N6180G0G28G91G40G30Z0.0
N6190M6
N6200 T02 (.25 Ball)
F
```

Write this number down on a piece of paper. This is where the tool change is for the finishing tool.

OK. With the toolpath selected, go ahead over to the CAM side and select the MACHINE menu at the top and then click on AUTO. This will generate the g-code program for the finish. Good Job!

STEP 12

Now let's go ahead and setup the tools for the solid simulation. To do this you need to scroll up to the first line of G-Code. Now click your mouse just on the left side of the first letter N. Hold down your SHIFT key and scroll down through the code (while holding down your shift key) to the number that you wrote down on that piece of paper. You can use the down arrow key on your key-pad, your mouse or by clicking and dragging the scroll bar on the right side wall of the CAM-G-Code Editor.

We are selecting all of the code that we used the first tool to create.

When you have reached this line of code with the same number that you wrote down, **STOP** scrolling downward and release your shift key. Go directly to the EDIT menu on the CAM side and select **TOOL**.

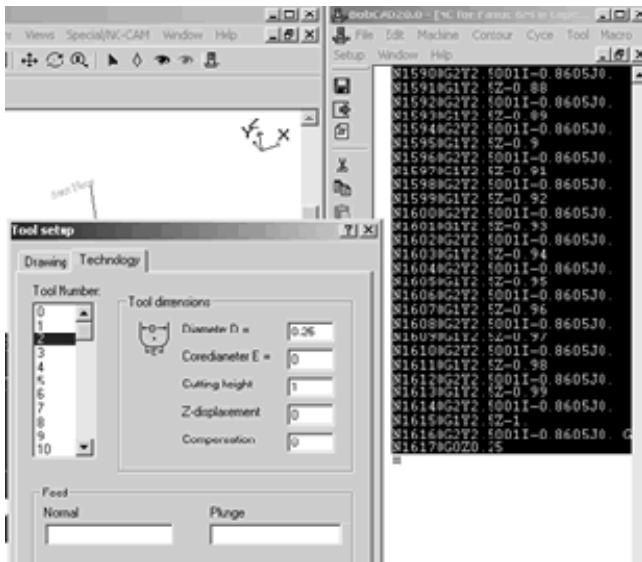
The screenshot displays the BobCAD20.0 interface. On the left, the 'Tool setup' dialog box is open, showing the 'Technology' tab. The 'Tool Number' is set to 1. The 'Tool dimensions' section includes: Diameter D = 0.25, Corediameter E = 0.25, Cutting height = 1, Z-displacement = 0, and Compensation = 0. The 'Feed' section has 'Normal' and 'Flunge' options. The 'Parameters' section has a 'Name' dropdown, a 'Type' dropdown set to 'String', and a 'Value' input field with a 'Delete' button. On the right, the CAM-G-Code Editor shows a list of G-code lines, with the first line 'N5820G2X4.Y2.9736I-1.4086J-0' highlighted. A 'Start Tool' label points to the first line of code.

Select the TECHNOLOGY tab from the Tool Setup box. Under, "Tool Number" click on 1. For the DIAMETER enter .25. Because we used a flat tool for the roughing we need to enter .25 for the CORE DIAMETER. Enter 1 for the CUTTING HEIGHT. Click OK at the bottom.

STEP 13

Now we need to do the same action starting from the next un-selected line of code. First click next to the letter N (which stands for "Number.") and while holding down your SHIFT key, scroll down to the last line of code. Make sure that you have highlighted the last line of code fully. When finished, go to the CAM EDIT menu and select TOOL again.

Select the TECHNOLOGY tab and this time click on tool 2 under Tool Number.

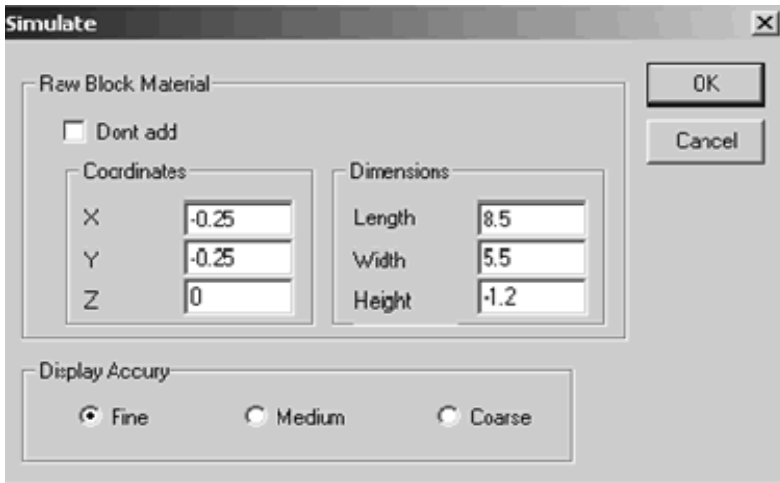


Enter .25 for the Diameter. Enter 0 for the Core Diameter and 1 for the Cutting Height. Now click OK at the bottom.

Alright. Now that the tools for simulation are setup you can go to the CAM EDIT menu and choose SELECT ALL. All of the code will be selected.

STEP 14

Go back to the CAM EDIT menu and select SIMULATE.

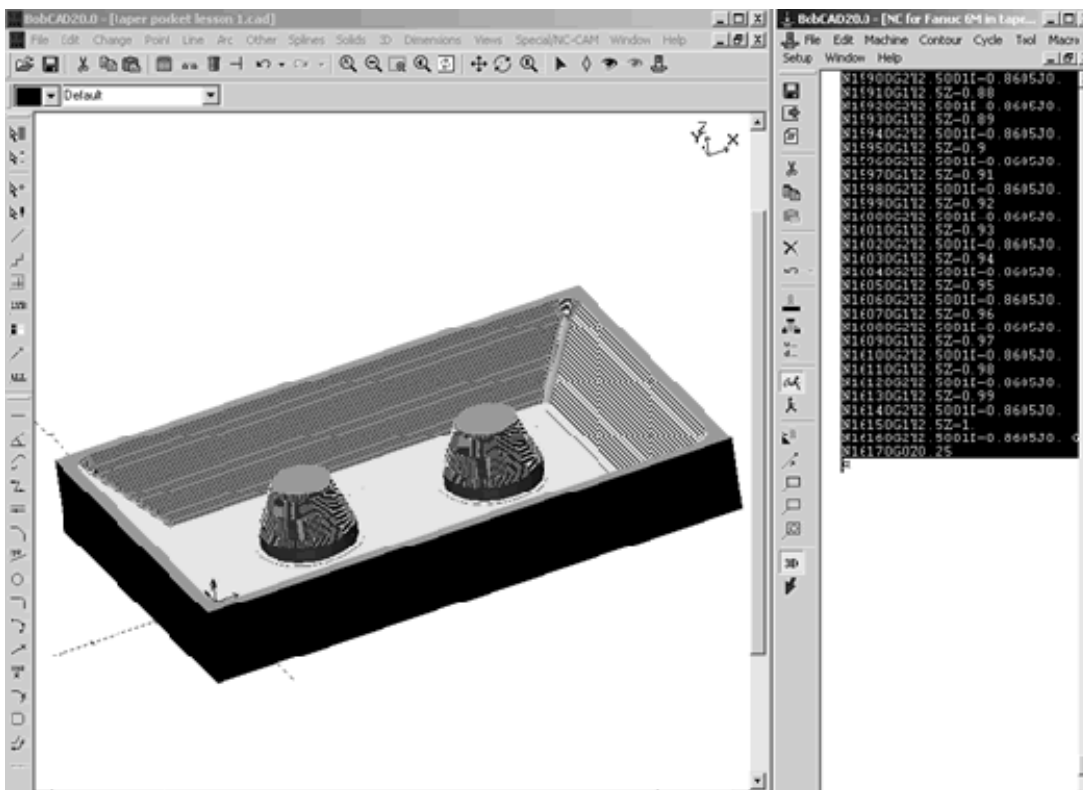


Under COORDINATES enter -.25 for X, -.25 for Y and leave Z at 0.

Under DIMENSIONS enter 8.5 for the Length, 5.5 for the Width and -1.2 for the Height. Check the FINE option for the display accuracy and click OK.

This will begin the simulation on the CAD side of the screen. You will see the roughing cycle first and then you will see the finishing occur.

The simulation result:



Note that in the simulation process, flat tools are simulated using the tip of the tool and ball mill cutters are simulated using the center of the tool.

You have completed this lesson.