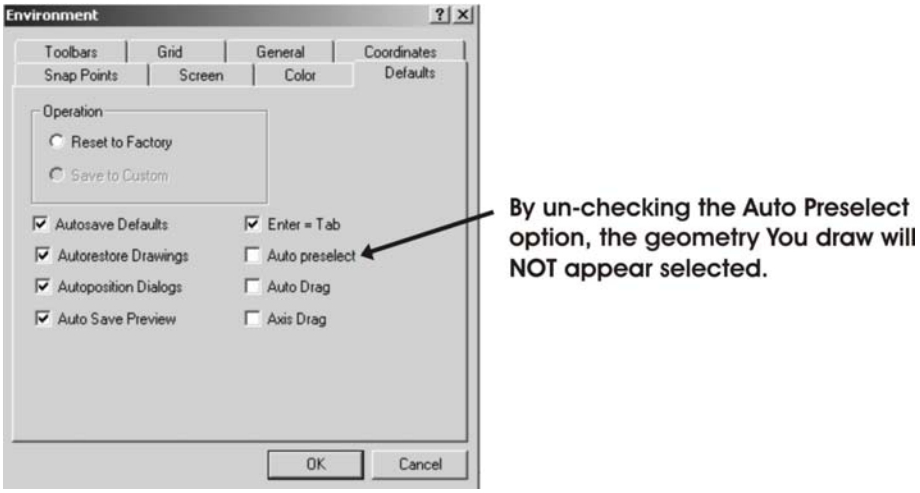
PROFILE TRAINING LESSON:

IMPORTANT: V21 has a global environmental option that is called, “Auto Pre-select.” This allows you to draw geometry whether freehand or by using coordinate values and have the geometry appear on the CAD screen “selected” and ready to be used for functions that are considered Object/Action. This simply allows you to draw shapes and not have to select them first to use a function. For advanced operators, this can help save time. In this lesson, we would like you to turn this option OFF to prevent any discrepancies executing the steps of this lesson. To do this, go to the main FILE menu and select ENVIRONMENT. Choose the DEFAULTS TAB and UN-CHECK the Auto Pre-select button if it is checked. Now click OK to exit the dialog.

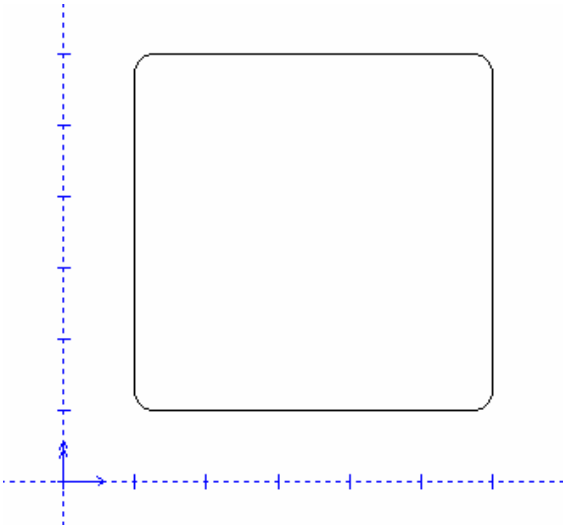


STEP 1
Start this lesson with a NEW CAD drawing screen.

Go to the OTHER menu and select RECTANGLE. Select Bottom Left under the section called, “Mode.” Select Fillet as the Corner Type and enter .25 as the radius. We will be using the **bottom left** corner of the rectangle as our point of reference. Now enter 1 for X, 1 for Y and leave the Z value set at 0.

Now, go ahead and enter a width of 5 and a height of 5. This is going to be a basic square shape with radius corners.

Now click OK to draw the shape.

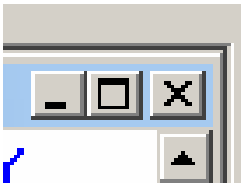


STEP 2
Now we are going to draw a point on the screen to indicate our start position for the tools approach into the profile cut. This is not mandatory for profiling. However it is a new option that we need you to be aware of.

Go to the POINT menu and select COORDINATES. In the box that appears, enter 2 for X and leave Y and Z at 0. Click OK to draw the point.

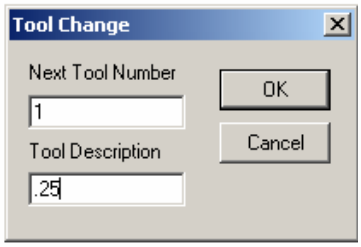
STEP 3
Now go to the Special/NC CAM main menu and select INSERT NC. This opens the Insert NC Data box and is where you will select the post processor. For this lesson select the FANUC 6M configuration and click OK to open the CAM.

NOTE: If your CAD window splits into 2 or more CAD windows it is because you opened a new file without fully closing out the last window. Simply use the Maximize option in the upper right corner of the CAD window we are working in. This is the middle, square looking button.



This will open the drawing window that we are working in to full view. Now select the VIEW ALL icon from the main CAD toolbar. This is the small magnifying glass with the letter, “A” inside of it.

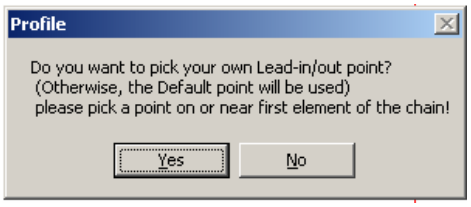
STEP 4
Go to the CAM side of the software and select the TOOL menu and then TOOL CHANGE. Under Next Tool Number enter 1. Under Tool Description enter .25.



Now click OK to insert this data into the NC Editor. The NC Editor is similar to a standard word processor allowing you to type in it and make editing changes manually as needed. This is where the entire program is generated.

STEP 5
Select the MACHINE menu from the CAM side and then PROFILE. Now you need to place your cursor on the BOTTOM line of the profile closest to the point and click your left mouse button one time to select it. Now move your cursor toward the left end of the line so that the directional arrow at the midpoint of the line is pointing towards the left. Now click your left mouse button ONE time to indicate this direction and then hit the F3 key on your keyboard to select the entire profile chain.

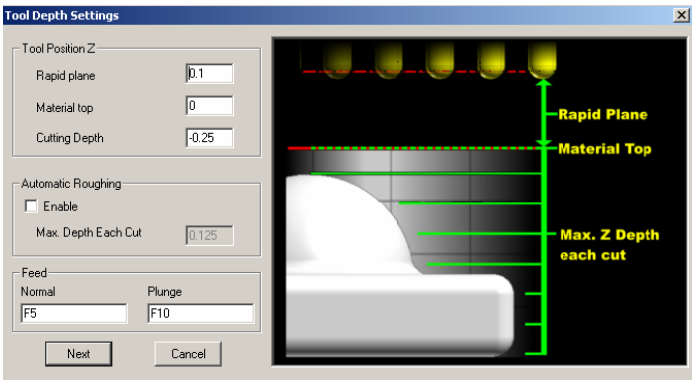
The profile box will automatically appear asking you if you would like to pick your own lead-in/lead-out point for this profile or not. Because we drew a start point for this we will select YES.



The “Default point” referred to in this box will be the starting point of the line you clicked on if you were to choose NO. In any case, select the YES button.

Go directly to the point that you created and click on it to open the Tool Depth Settings box.

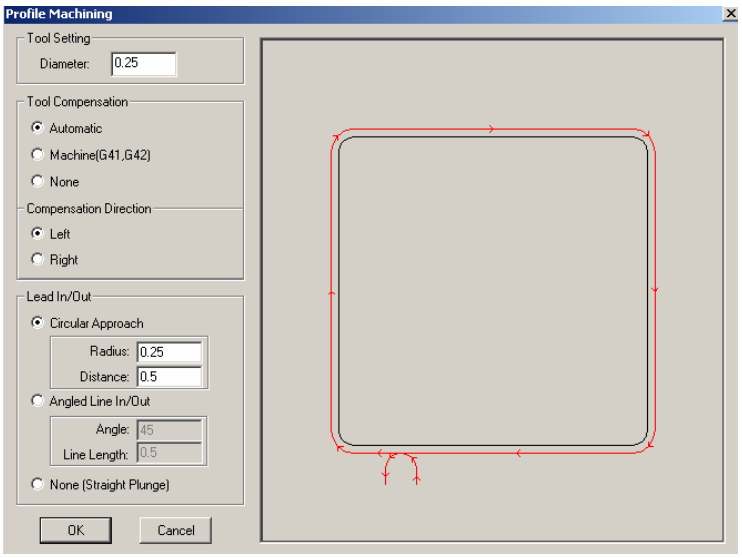
STEP 6
Have a look at this new box.



Under Tool Position Z you have your Rapid Plane. Enter .1 for this as it represents the tools clearance above the part during rapid moves. Your material top for this part if 0 and the CUTTING DEPTH is -.25. We are not going to be roughing this profile so do NOT check the Enable option under Automatic Roughing. In the FEED section enter F5 for NORMAL and F10 for the Plunge.

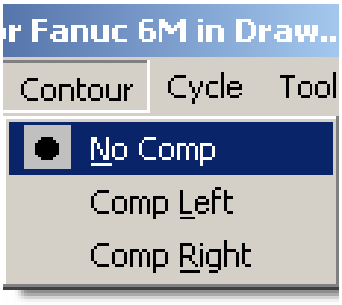
Now select NEXT.

STEP 7
Now you will have the Profile Milling command box. Let’s take a brief break from this lesson and have a look at this box.



The Profile Milling command box gives you control over the following:

- **The tool diameter** that you will be using. Everything in this box is associative, meaning that if you make a change, the preview will reflect the changes made. If you change the size of the tools diameter and hit your enter key the preview will reflect this modification.
- **The tool compensation**. By choosing **Automatic**, you can use the right or left options in this box and the software will automatically create an offset on the left side or right side of the original geometry. The lead-in and lead-out will be added to this offset unless you select the "Straight Plunge" option. Basically, the software does the offset for you based on half the distance of the size tool you use. By choosing **Machine G41/42** you are telling the software that you would prefer to cut the original profile geometry using right or left compensation (G41 or G42) and that this G41 or G42 code will be added to the g-code program created in BobCAD. By choosing **NONE** you are indicating that you are not going to use compensation at all and will be using the drawn geometry for cutting with the center of the tool being used. This is the same as using the Auto Cut function in BobCAD and whatever you have selected in the Contour menu will be added to the code.



This is important to understand when using these options. Some post processors are setup differently when it comes to the Contour menu in the CAM. Some posts will add "Normal" and some will list compensation as in the image above that is setup for the Fanuc 6M post.

If you have something selected in the contour menu and use the profile wizard for compensation, the profile wizard will over ride this menu command unless you select NONE in the comp section of the profile wizard. Therefore, you should check the contour menu in the CAM to see what is selected.

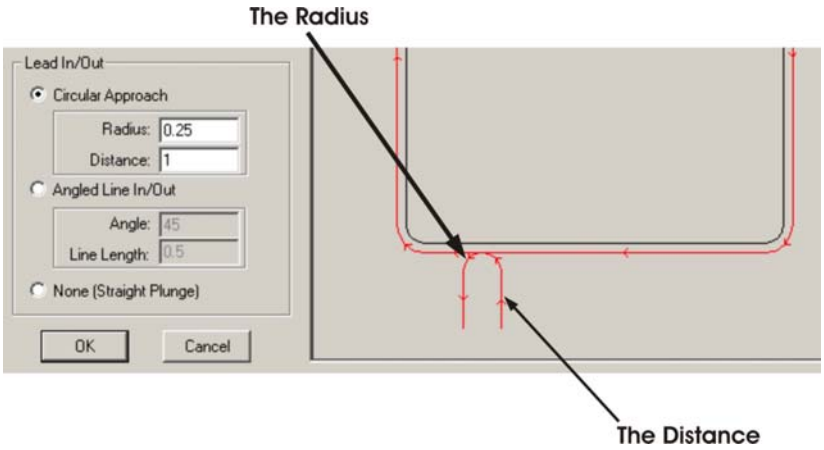
Here it is again. If you use a comp option in the contour menu and choose NONE in the profile wizard, the option selected in the contour menu will be used when the software creates the g-code program. If you use compensation in the profile wizard it will over ride the option selected in the contour menu. It's that simple.

You see, you do not always have to use the profile wizard for profiling in BobCAD. You can simply create manual offsets for your tool (half the distance of the tool size), use the Tool Depth Settings and then the Auto Cut function in the machine menu along with the comp option you need selected in the contour menu if your post processor configuration supports it.

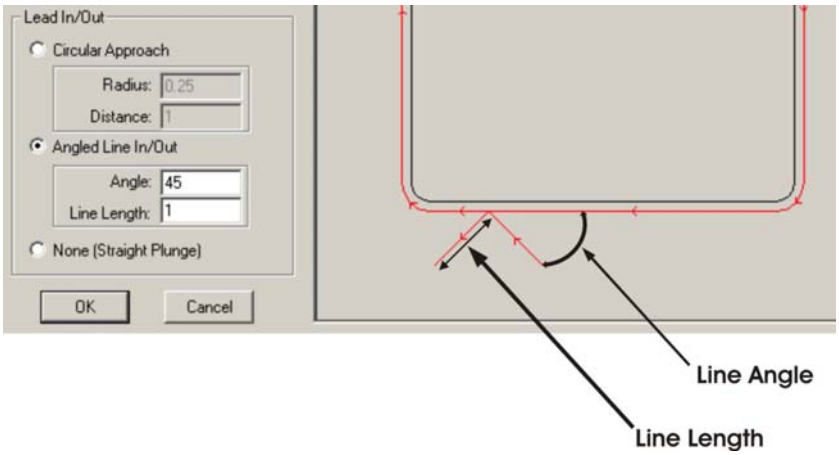
Just remember one important thing, when you are using G41 or G42 comp from the CONTOUR menu and will be using the Auto Cut option, you will want to create a lead-in and lead-out manually by using the approach/depart function from the OTHER menu of the software.

- **Compensation Direction**. You have two options, Right and Left.
- **Lead-In/Out.** This section offers you a **circular/radial** approach and depart or an angled line approach and departure for your tool. The radius value allows you to adjust the overall radial approach as well as the departure and the "Distance" option offers you the ability to shorten or extend the length of the approach and departure of the tool.

Here's an example of the **Circular approach and departure** below.



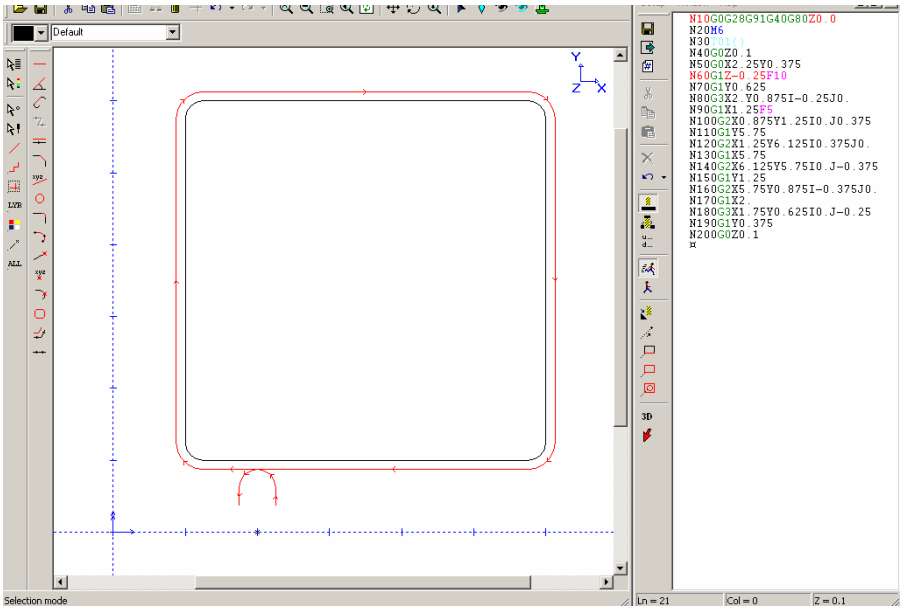
You also have the ability to create an angled line type approach and departure for the tool. The "**Angle**" value is simply the angle of the lead-in and lead-out line itself and the "Line Length" option determines how long or short you want these lines to be. Let's take a look at this in the example below.



The **NONE** option refers to straight plunge ONLY.
OK, let's get back to the lesson now that you understand how to use this box. Go ahead and enter .25 for Diameter under TOOL SETTING.

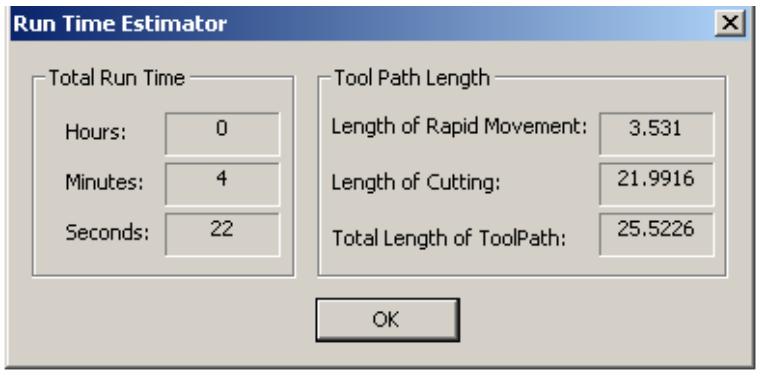
STEP 8
Select Automatic under the Tool Compensation. Under Compensation Direction choose the LEFT Option. Under Lead-In/Out select the CIRCULAR approach option. Now enter .25 for the Radius and .5 for the Distance.

Now select OK to produce the toolpath and your G-Code program.



STEP 9

Now go to the EDIT menu on the CAM side and choose Select ALL. This will highlight all of the code. Now go back to the EDIT menu on the CAM side and select RUN TIME ESTIMATE. A box will appear Asking if the Maximum Feed Rate is 50 inches or millimeters per minute. You can change this value if you want but if you select OK the estimator box will appear next.



Congratulations! You have completed this lesson.