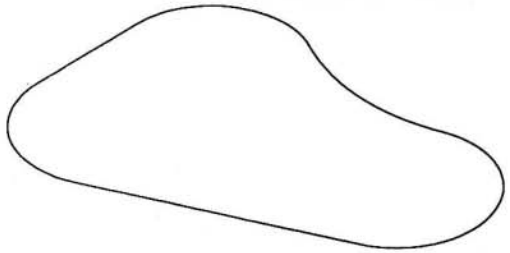
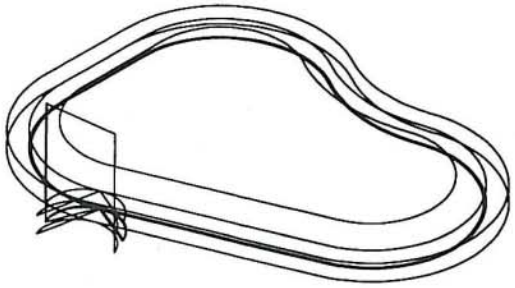
Project 7 - 1. Contouring external profile

Use the contour function to generate toolpaths to cut a profile shown in Figure 7.20. The total depth of cut is 0.3-in. The side cut amount is 0.3-in. The geometry file is ***proj5.***



a. Part geometry



b. Toolpaths

Figure 7.20

Procedure outline:

1. Use the ½-in. end mill which is #34 in the tool library.
2. Use the geometry file ***proj5.***
3. Use the contour function to generate contouring toolpaths.
4. Add an arc moves to both leadin and leadout of each tool pass.
5. Set the amount to remove at 0.3" for both on sides and in Z.
6. Select aluminum material (#24) to automatically calculate machining parameter values.
7. Save the toolpath file as ***contour1.***

Step 1. Load the geometry file *proj5.*

Select **File → Open**
File Name: ***proj5***
The geometry drawing should appear as in Figure 7.21.

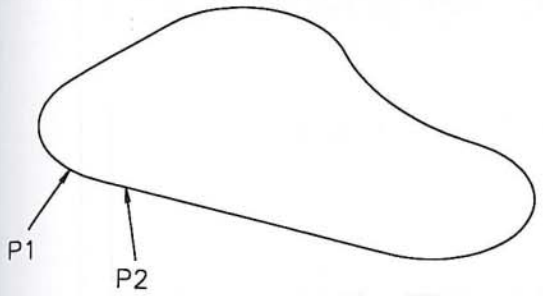


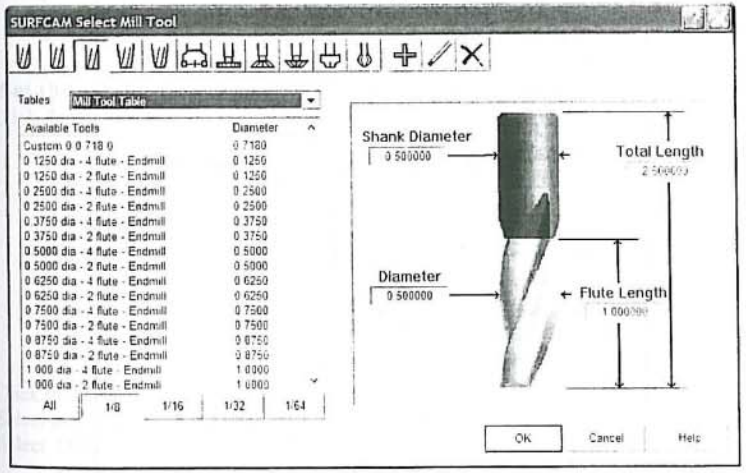
Figure 7.21

Step 2. Initiate the contour function. Use the pick points in Figure 7.21.

Select **NC → 2 axis → Contour → Chain**
Pick **P1** to select a beginning element
Pick **P2** to select an ending element
Select **Done** to complete the selection of contour

Step 3. Select the tool.

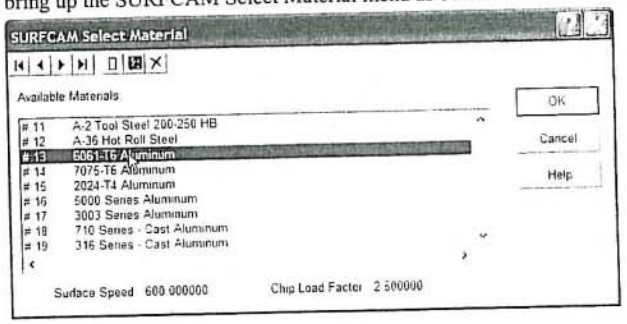
The Tool Information menu should appear on the screen.
Click on the **Tool** box button
The SURFCAM Select Mill Tool menu will appear as below.



Click the end mill icon on the top of the menu
Click **tool #32** (1/2 inch HSS end mill) from the tool list

Step 4. Select material type to be cut.

Click on the **Material** box in the Tool information menu to bring up the SURFCAM Select Material menu as below.

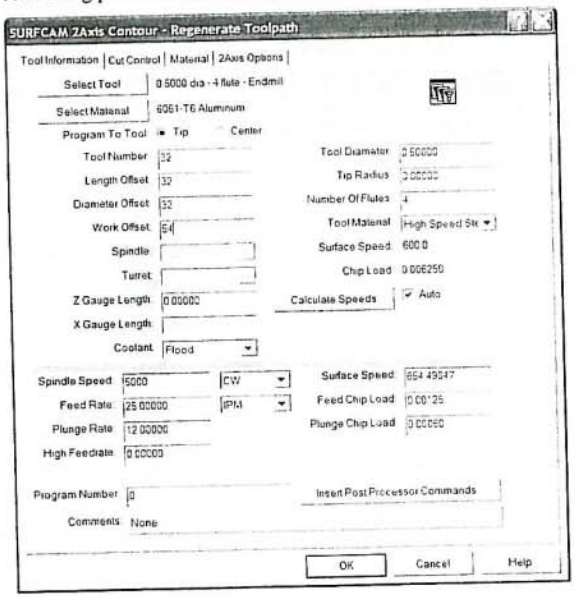


Scroll the available materials list to show **#13Aluminum**, click to select it
Select **OK** to return back to the tool information menu

The value of the three machining parameters located at the bottom of the tool information menu will be calculated automatically from the given cutter and material data. You can input a new value to override them.

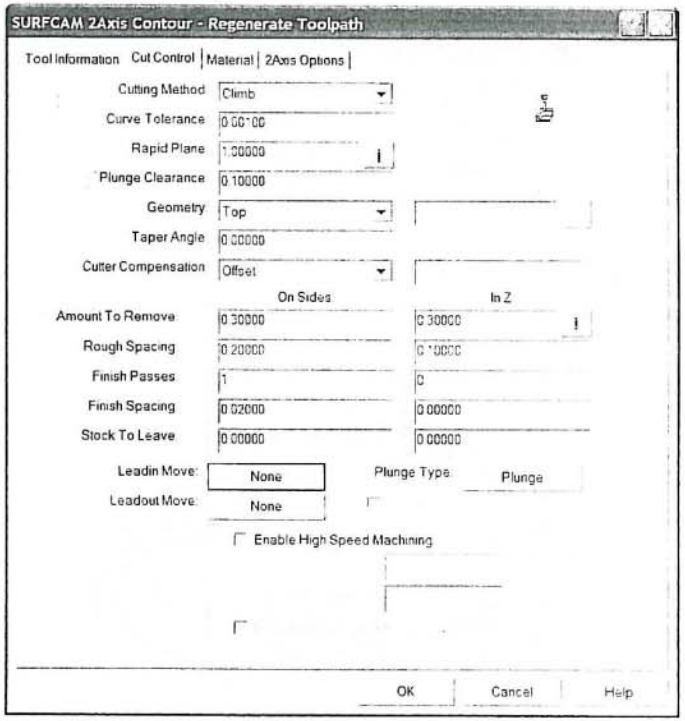
Step 5. Define the tool information parameters.

Set the remaining parameters in the tool information menu as shown next.



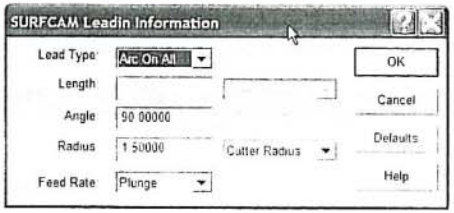
Step 6. Set the Cut Control parameters as shown here.

Click on the Cut Control tab to advance to the cut control menu.



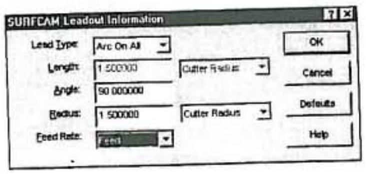
Step 7. Set the Leadin and Leadout Move parameters.

Click the Leadin Move button, the Leadin information menu should appear as below.



Click the **Lead Type** button to bring up the five options
Select the **Arc On All** option
Select **OK** to return back to the Cut Control menu

Click the **Leadout Move** button, and then set the menu as here.



Select **OK** to return back to the Cut Control menu.
Select **OK**
Pick any point *outside* the contour at the "Select Side to contour" prompt

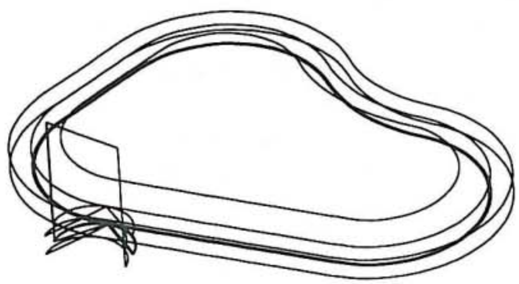
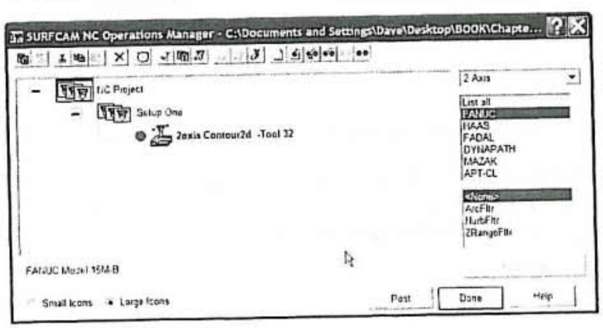


Figure 7.22 The rough and finish toolpaths are generated

Select **Accept** to retain this toolpath

Step 8. Run the postprocessor.

Click the **SURFCAM Operations Manager** icon to bring up the menu as below.



Select **Post**

The toolpaths are processed by the specified post processor and the result appears on the SEdit II utility. A partial part program is listed here for reference.

```
%
O0
(Surfware, Inc.)
([illegible] Corsa Ave)
(Westlake, [illegible])
G17 G40 G80 G90
T32 M6
M3 S5000
G00 G54 X2.6181 Y3.0097
G43 Z1. H32
M8
G00 Z0.1
G01 Z-0.1 F12.0
G02 X3.0251 Y3.0796 I0.407 J-1.1501 F25.0
X4.2451 Y1.8596 I0 J-1.22
X3.5018 Y0.7266 I-1.22 J0
G01 X1.4767 Y-0.123
G02 X1. Y-0.22 I-0.4767 J1.123
X-0.22 Y1. I0 J1.22
.
.
.
.
M9
G90 G00 G49 Z0 M5
X0 Y0
M30
%
```

Step 9. Save the toolpaths as *contour1*.

Select **File → Save As**
File Name: ***contour1***
Select **Save** to save this model file.

Pocket Function

The pocket function is used to generate a set of toolpaths to (1) remove the material enclosed by a closed contour, (2) face a flat surface, and (3) rough cut a local pocket. The Cut Control menu for the pocket function is shown below.

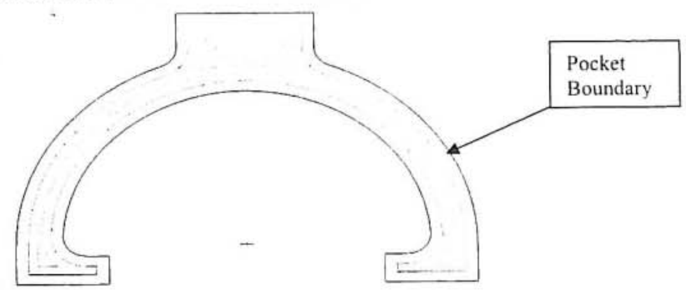


Figure 7.23

Rough and Finish Passes

The rough and finish passes in the Sides direction are automatically calculated from the given values for the machining parameters. The rough and finish passes in the Z direction are determined the same way as they are for the contour function.

Pocket Cut Modes

SURFCAM uses three pocket cut modes to generate the pocketing toolpaths:

- **Zigzag Mode** This mode generates a set of back and forth linear toolpaths in the specified orientation to rough the pocket (refer to Figure 7.24).

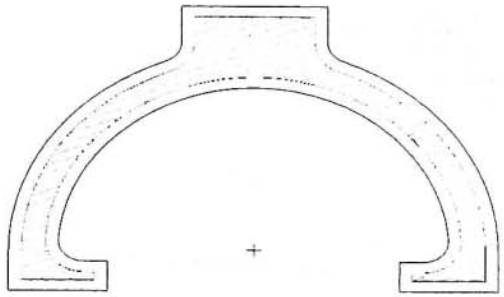


Figure 7.24 Zigzag mode

- **Zig Mode** This mode generates a set of linear toolpaths similar to the zigzag method. The difference is that this is a one-way method. It generates toolpaths that cut in the same direction. They make a pass, then retract in the Z direction, rapid to another starting point and take another pass in the specified direction. The toolpath pattern is repeated until the pocket cutting is completed (Figure 7.25).

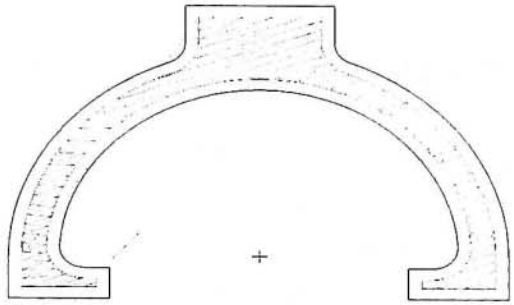


Figure 7.25

- **Spiral Mode** The spiral mode starts the cutter at the pocket center or specified pocket start point and spirals it to the wall of the pocket (Figure 7.26).

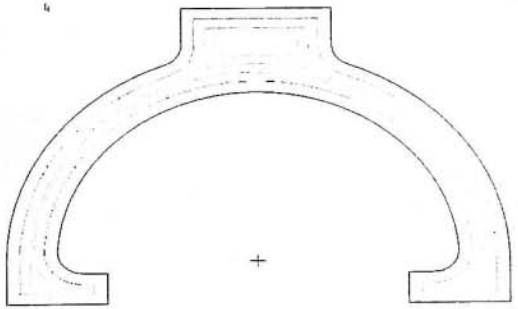


Figure 7.26 Spiral mode

Pocketing with Islands

Islands are areas inside the pocket boundary that are not intended to be machined during pocketing. The islands must be closed contours. SURFCAM is capable of handling multiple profiles commonly found in work pieces. The number one rule is to have one outside boundary and *(unlimited)* closed inside boundaries as shown in Figure 7.27.

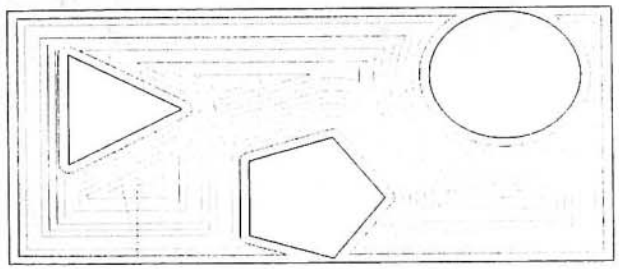


Figure 7.27 Pocketing with islands