9.10 Reverse Function

This function is used to reverse the direction of the toolpaths in an INC file. The **Plunge Distance** parameter specifies the plunge distance for the reversed toolpaths.

9.11 Review Questions

1. Describe how SURFCAM organizes toolpaths.
2. List and briefly describe the four file types SURFCAM uses to contain the data.
3. Describe the NC Operations Manager.
4. List the two ways to access the NC Operations manager dialog box.
5. What are the three main areas in the NC Operations Manager dialog box?
6. Describe how to edit toolpaths by individual elements.
7. List the five types of toolpath information that can be edited.
8. Describe how to edit toolpaths by sections or project.
9. Explain the procedure for changing the sequence of operations.
10. Describe the purpose of toolpath transformation functions.
11. List the ten toolpath transformation functions.
12. What are the advantages of using toolpath transformation functions?
13. When should you use the translate function?
14. What is a limitation on the use of the translate function?
15. When should you use the rotate function?
16. Use drawings to show how the construction view affects the result of toolpath rotation.
17. Use drawings to show the difference between using the rotate function with and without using retain orientation.
18. When should you use the scale uniform function?
19. What are the two important factors in scaling toolpaths?

20. Describe the difference between the scale uniform function and the scale XYZ function.
21. When should you use the mirror function?
22. Use drawings to show the effect of the mirroring plane on the copied toolpaths.
23. When should you use the rectangular array function?
24. When should you use the circular array function?
25. When should you use the indexed array function?
26. Describe the difference between the circular array function and indexed array function.
27. The pocket module is used twice to generate toolpaths for pocketing the larger circular pocket and pocketing one of the seven small pockets as shown in Figure 9.13.

The toolpaths for the small pockets are copied to six locations by using the circular array function. The part geometry file name is ***rotate-1*** which is created in problem 4 of chapter 6.

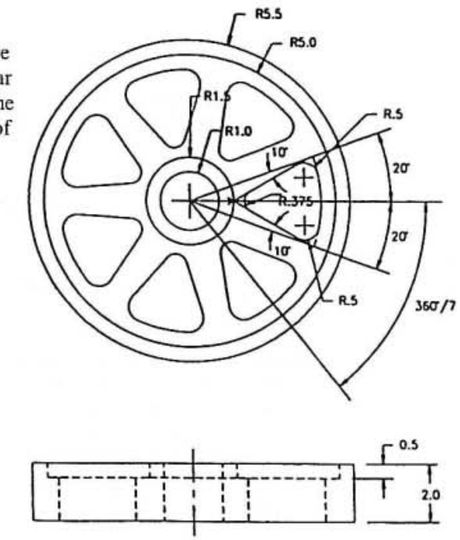

Figure 9.13

Tips:
1. Use the pocket function to generate toolpaths to cut the large pocket at a depth of Z-0.5". Use a 10" diameter circle as the outside contour and 3" diameter circle as the island.
2. Use the pocket function to generate toolpaths for cutting one of the seven small pockets using the following machining conditions:
   Cutter diameter = **0.5"**
   Roughing passes spacing = **0.3"**
   Number of finish passes = **1**
   Finish passes spacing = **0.05"**
   Depth cuts: Rough **2** cuts at **0.5"** Finish: **1** cut at **0.5"**
3. Use the circular array function to copy the toolpaths for the small pockets to the other six locations.

28. The scale function is used to scale the toolpaths for pocketing a rectangle for cutting two smaller rectangular pockets as shown in Figure 9.14. Some tips are listed after the figure.

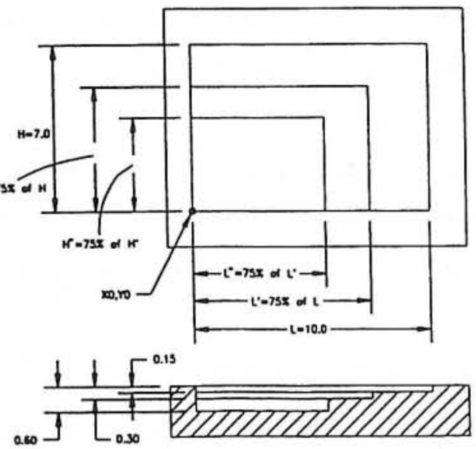

Figure 9.14

Tips:

1. Create a 10" x 7" rectangle.
2. Use the pocket function to generate toolpaths to cut the pocket using the following machining data:

   Cutter diameter = **0.75"**
   Cutting method: **Spiral inside out**
   Roughing passes spacing = **0.5"**
   Number of finish passes = **1**
   Finish passes spacing = **0.1"**
   Pocket depth = **-0.15"**
3. Use the scale uniform function to scale the original toolpaths and copy to two locations. The scaling data is:

Scale about the Origin:	Axis	Scaling factor
	X	**0.75**
	Y	**0.75**
	Z	**2.0**

29. Two identical workpieces are to be produced in one machine setup as shown in Figure 9.15. The set of toolpaths for cutting one part is copied to another location as specified by using the translation function. The part geometry file name is ***exer-6*** which is created in problem 23 of chapter 5. The generation of the toolpaths for machining the part includes using contour, pocket and drill functions.

Tips:

1. The machining sequence starts with the contour function for cutting the external profile, followed by the pocket function for pocketing the slot pocket, and finally using the drill function the drill four holes.

   Cutters: 1" end mill for contouring
   0.5" end mill for pocketing
   1/2" drill for drilling four holes

2. Use the translate function to copy the toolpaths to a new location.

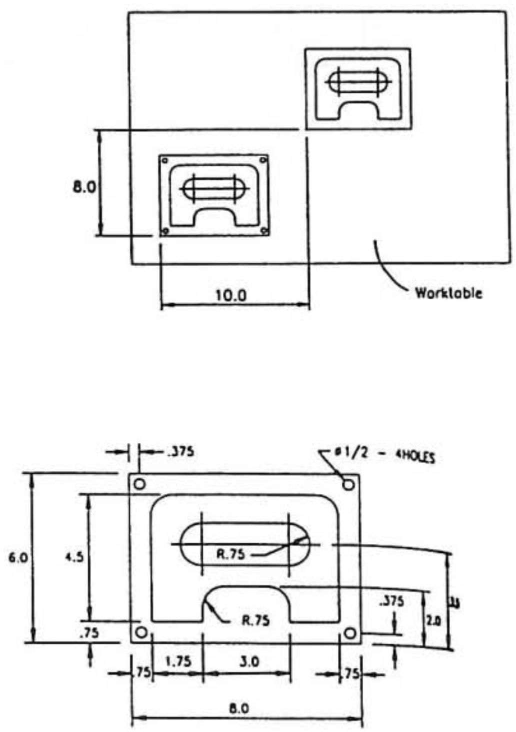

Figure 9.15

30. Use the rectangular array function to generate the toolpaths for cutting four identical parts in one part program. This allows four parts to be machined in one machine setup. The workholding layout of these four parts is shown in Figure 9.16. The part geometry file is ***exer-6*** that is identical to the file used in problem 29. Use the instructions given in problem 29 to generate the original toolpaths.

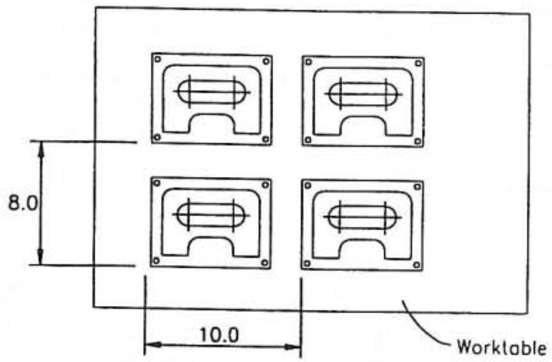

Figure 9.16

Chapter 10
Creating Part Geometry for Lathes

Highlights of this chapter:

1. Basics of lathe coordinate systems
2. Basics of lathe program zero
3. Basics of lathe radius and diameter dimensioning
4. Creating part geometry for lathes

10.1 Overview

Many machining operations can be performed in a CNC lathe. These operations include **facing, turning, profiling, grooving, drilling, boring, threading, chamfering, cutting off,** and **milling** (in mill-turn machines). This chapter presents how to create part geometry for CNC lathes. It begins with lathe coordinate systems, and is followed by the selection of program zero, radius or diameter programming, and several examples of creating part geometry.

The parts for lathe operation are designed in 2D geometry. The parts are turned on two axis lathes. SURFCAM Velocity III is compatible with any lathe axis orientation. The user sets the axis orientation. The actual axis orientation and direction is controlled by the post processor. You can create the standard output for many manufacturers with the sample post processors that are supplied. See the Post Processor for the procedure for configuring the different post processors. For display purposes, all references to the tool orientation are based on the turret that is at the rear of the part. Front lathe machine code is produced by the post processor.

You can generate NC programs in a local coordinate system. Therefore, you can machine different sections of the work with independent machining orientations—a necessity with many parts. Each Lathe operation has a separate dialog box. That box is displayed after you select the operation and identify the geometry on which it will be done. After you click the OK button on a Lathe dialog box, the prompt asks you to select a "Lathe Retract Point" and "Lathe Clearance Point."

Retract-Clearance Points

The **retract point** is the point where the toolpath begins and ends. The lathe retract point is normally the point where the tool changes are made.

The lathe **clearance point** defines a level or distance from the Z-axis. The tool does a rapid move from the retract point to this level at the lathe cycle start. The tool does a rapid move from this level to the retract point at the end of the lathe cycle. These rapid moves are perpendicular to the Z-axis. The lathe ID operations normally use lathe clearance points.

Note: If you do not need a retract point, click Done. If you do not need a clearance point, click Done as this control point may be output automatically in your postprocessor.