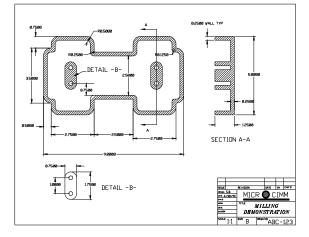
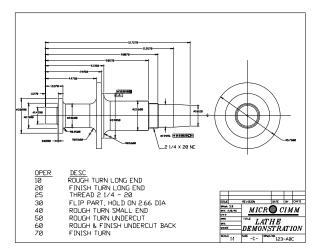
POWERSTATION 2002/XPERT

Tutorial Manual

Revision 6/10/2004

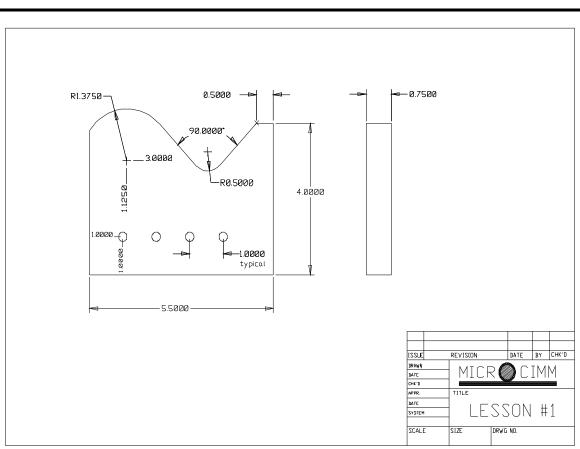
Copyright ©1991,2004 GLASS HOUSE SOFTWARE, LTD. All Rights Reserved





Reproduction or publication of the contents of this document in any manner, without express permission of Glass House Software, Ltd., is prohibited. No liability is assumed with respect to the use or consequential damages incurred from use of the information herein.

orials	5
LL tutorial	5
Drawing	
Machining	
Job Processing	
Drafting	39
THE tutorial	
Drawing	56
Machining	
Job Processing)3



MILL TUTORIAL

Figure 4-0 The sample drawing

The purpose of the tutorial is to guide you step by step through the creation of a POWERSTATION drawing. The drawing shown in figure 4-0 will be taken from the initial geometry creation, to machining, and finally dimensioned and plotted. Each step will be numbered for easy reference. The information you are required to type will be underlined. There are two tutorials, one for milling, and one for turning.

<u>STEP 1</u>

We begin by starting POWERSTATION. To do this: Press [Start] (on the WINDOWS Tool bar), then select "Programs", then "MICROCIMM", then "POWERSTATION"

<mark>//</mark> P0'	WERS	TATION 2000) — [Untitle	ed]									_	×
<u>F</u> ile	<u>E</u> dit	<u>G</u> eometry	Modify	<u>D</u> rafting	<u>M</u> achining	D <u>i</u> splay	<u>H</u> elp							
🖸		🕦 📴 🗎		 All Entitie: 	s 🔻 Select	- ?		<u>.</u>	⊕	Q	1	1	XY - View	•
X-8	1529	Y4.5689	Z0.00000	Laye	er O									
A	~ 1													-
7	4													
LOAD DXF	S.													
1	Q													
₿ ŀ ₿	₿•₿													
×	×,													
0P#10 0P#20 0P#30	P													
é														
A	(
X N1G0 N2G1 N3	B	•												-
-		Ready												_
Copyrig	ght©19	189-2000, Glass	House Soff	ware, Ltd.][]Т	ool:×0.00	0000	r 0.00	1000 Z	Z 0.00	000		1.

Figure 4-1 The opening screen

<u>STEP 2</u>

At this point the POWERSTATION opening screen will be displayed (Figure 4-1). We will begin by drawing a rectangular box which will be used as the basic shape of the part.

To create the box we select, from the main menu select "Geometry", "Line", "Box"

Geometry - Line - Box 🛛
Entry Method Specify Width & Height Specify Second Corner
Box Corner Radius : 0.0
Base Angle : 0.0
OK Cancel <u>H</u> elp

At this point the above dialog will be displayed. For this example we do not need to change any of the above defaults. Press [OK] with the mouse.

Next you will be presented with the standard "Get Point" popup menu. Select "Coordinates" from this menu.

Now you will be asked for the X and Y coordinates of one corner of the box.

Coordinate Entry	×
-X- Coordinate 0.00000	
Y- Coordinate 0.00000	
Enter Point Coordinates	
OK Cancel <u>H</u> elp	

Next you will be presented with the standard "Get Distance" menu. Select "Coordinates" from the popup menu and answer with:

Coordinate Entry	×
-X- Coordinate 5.50000 -Y- Coordinate 4.00000	
Enter X/Y Distances	
OK Cancel	Help

At this point the screen will look like figure 4-2.

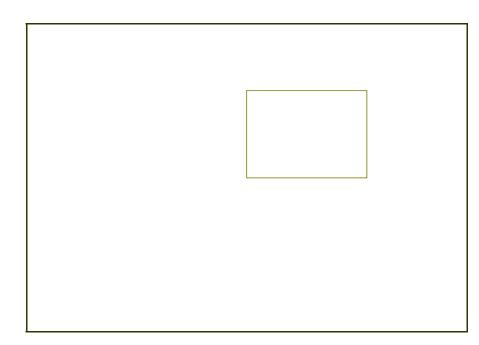


Figure 4-2 the basic part shape (box)

<u>STEP 4</u>

It will be easier to work if the box were more centered in the display screen. To do this we will use the **"Display-Zoom Extents"** menu command.

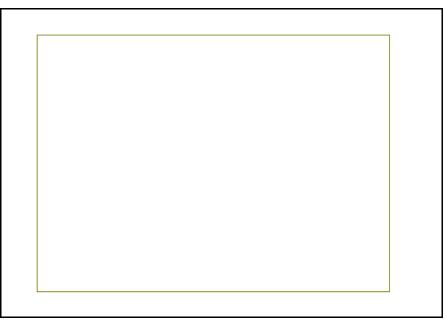


Figure 4-3 The display has been "Zoomed"

Figure 4-3 shows the new box location. Note that the fitting may change the position of the box on the display but not its size or location in space.

<u>STEP 5</u>

Next we will enter the 1 3/8" arc. Select "Geometry", "Arc", "Coordinates" from the main menu. You will be asked for the coordinates:

Coordinate Entry	×
-X- Coordinate	Arc Radius
1.12500	1+3/8
-Y- Coordinate	Start Angle
3.00000	0.00000
-Z- Coordinate	
0.00000	360.00000
Enter Arc	Coordinates
ОК	Cancel <u>H</u> elp

The display should now look like Figure 4-4.

<u>STEP 6</u>

Now lets create the line tangent to the 1 3/8" arc. First select "Geometry", "Line", "Tangent to One" from the menu.

You will now be asked to "Select a Point or ARC". You should now select the arc, anywhere on its upper right side. See figure 4-5.

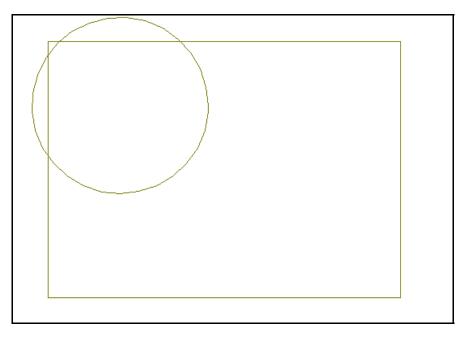


Figure 4-4 The first (1 3/8") arc

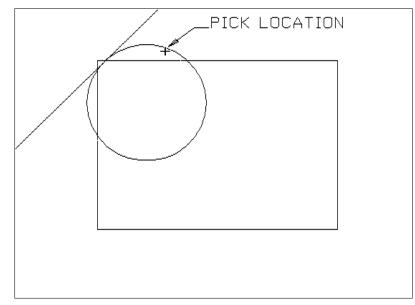


Figure 4-5 Creating the tangent line

After picking the approximate tangent location, you will be asked:

Angle : 45.000	1000
Degrees : 45	¢
Minutes: 0	•
Seconds : 0	4
Seconds : 0	

Notice the wrong line has been created (see figure 4-5). Pressing the Δ icon on the tool bar (Undo) will remove it. This is a good time to see the effects of using the Δ Undo and XRedo commands. Repeat the steps to create the line, but use -45 for the angle.

See figure 4-6

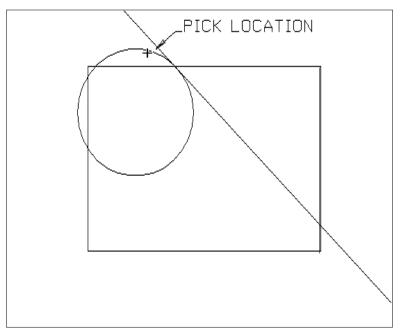


Figure 4-6 The corrected tangent line

<u>STEP 7</u>

Now for the other line at 45 degrees. To create this line we need to define a "construction" point first. Select "Geometry", Point", "Coordinates" the menu. You will be asked:

Coordinate Entry	×
-X- Coordinate 5.00000	
-Y- Coordinate 4.00000	
-Z- Coordinate 0.00000	
Enter	Point Coordinates
OK	Cancel <u>H</u> elp

The screen will now look like figure 4-7

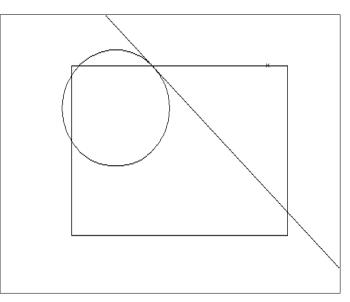


Figure 4-7 a construction point

Now we can create the line. Select "Geometry", "Line", "Tangent to One" from the menu. You will now be asked **"Select a Point or Arc"**. In a case like this you must be careful, if you simply place the pick box over the point, you do not know if you are picking the line or the point.

To be sure that you are picking the point you must use what is called a mask. To **"mask"** the selection so that only a point can possible be picked, on the tool bar, select "Point" from the "Entity Mask" drop down list (The list should currently be displaying "All Entities"), now place the cursor over the point, and press the left mouse button to select it. Note: After executing this command, it is a good idea to set the "Entity Mask" back to "All Entities". See figure 4-8.

Next you are asked:

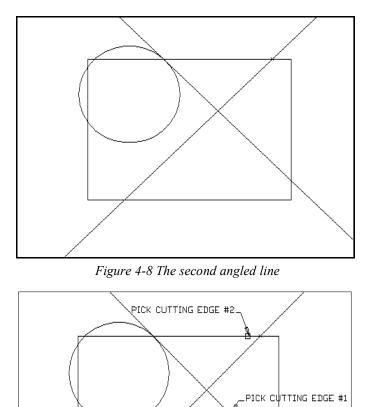
90 -270	Angle :	15.00000
	Degrees :	45 🚦
360	Minutes: [)
270 -90	Seconds : [)
	Seconds : [)

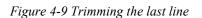
Note: A faster way to enter this would be to have skipped the "construction" point all together. Go directly to the "Line Tangent to One" command, when asked to select a point, press the right mouse button and select <Escape> from the menu. The standard "Point Entry" menu will be displayed, select "Coordinates" and enter in the point coordinates. Enter the line angle as shown above.

<u>STEP 8</u>

Now we can begin "trimming" the entities. The basic principal of trimming is to first select the entity to be trimmed, then two "cutting edges". The entity will be trimmed so that only the portion of the entity that is **between** the cutting edges will remain. <u>NOTE: In the following examples the terms "Cutting Edge" &</u> "Bounding Edge" will be used interchangeably.

First we will trim the last line entered. Select "Modify", "Trimming", "Trim/Extent" from the menu.





PICK LINE TO BE TRIMMED

STEP 8 continued . . .

Next you will be asked to "Select any Entity". For this you can pick any position along the line. Now the "Trim/Extend" command asks for the cutting edges:

"Bounding Edge #1, Select a Point, Line or Arc" "Bounding Edge #2, Select a Point, Line or Arc"

See figure 4-9 for where to pick, and figure 4-10 for the results.

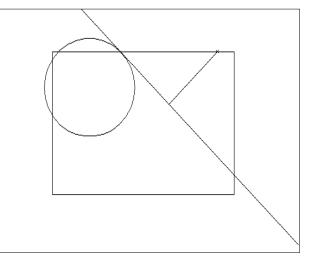


Figure 4-10 The trimmed line

<u>STEP 9</u>

Now we can trim two other lines in the same manner. See figures 4-11 and 4-12 for the pick locations, and figure 4-13 for the results. Remember, if you accidentally remove the wrong thing, simply use the "Undo" command to remove the mistake.

<u>STEP 10</u>

Trimming the line on the left is a bit more complicated as it intersects the 1 3/8" radius arc in two places. After you select the line to trim and the two cutting edges you will be asked to indicate which of the intersection to trim to. See figure 4-14. See figure 4-15 for the results.

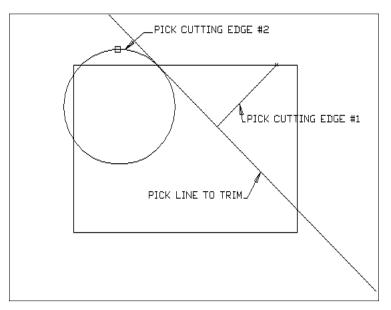


Figure 4-11 trimming lines

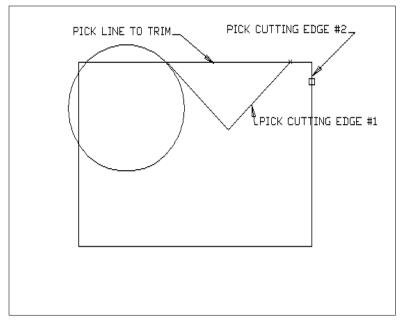


Figure 4-12 Trimming lines

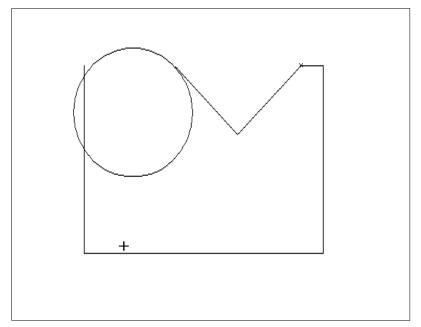


Figure 4-13 The trimmed lines

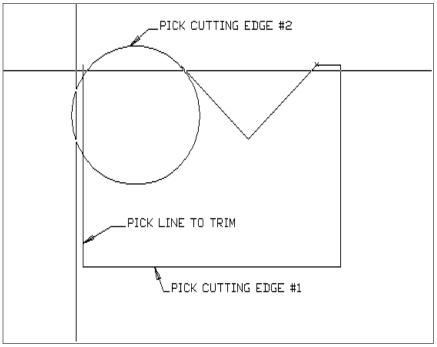


Figure 4-14 Trimming a line between a line and arc

<u>STEP 11</u>

Now we will trim the 1 3/8" arc. Follow trimming the arc carefully as selecting the cutting edges in proper order is extremely important. In this example the arc intersects cutting edge #2 in two places, you will be asked to specify which intersection to trim to. See fig 4-16 Note: If you make a mistake, and the results are not what you desire, simply press the "undo" Δ icon on the toolbar.

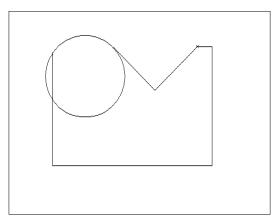
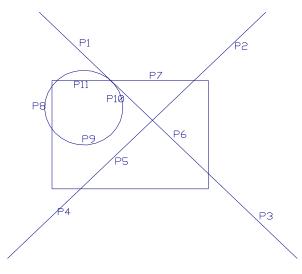


Figure 4-15 The trimmed line

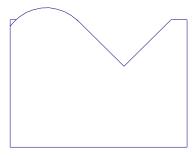
The display is really starting to look like the part we want see figure 4-17.

<u>Notes on Trimming:</u>

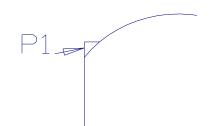
The Trim/Extend command was used for all of the above examples to demonstrate the versatility of this command. *It does not represent the easiest way to trim*. All of the above examples could have been done using the "EASY-TRIM" command. After selecting this command you would simply point (with the mouse) to the sections of lines or arcs that you want to remove. Feel free to try the example trimming using this command. Simply click (In Order) on the points in the following examples:



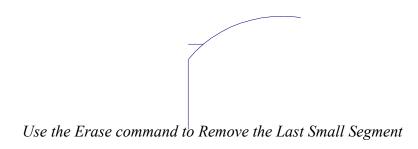
Easy Trim, Part #1



Results After Part #1



Removing the Small Section at Point #1





<u>STEP 12</u>

We need one more piece of geometry, the 1/2" radius on the top. To add this we will save a few steps and use the "FILLET" command.

Select "Geometry", "Arc", "Fillet" from the menu. You will now be asked:

```
First Item - Select a Point, Line or Arc
Second Item - Select a Point, Line or Arc
Indicate the approximate location
Enter the Radius [1.3750] ? <u>.5<Enter></u>
```

Note that the entities to be filleted must be selected in a counter-clockwise order, or the wrong fillet will be generated. Note: This is not generally necessary when filleting lines, the system will generate the proper fillet automatically (This is NOT true when filleting lines to arcs, or arcs to arcs). (See figure 4-18)

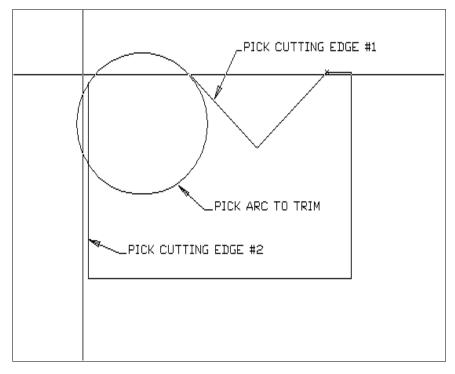


Figure 4-16 Trimming an arc

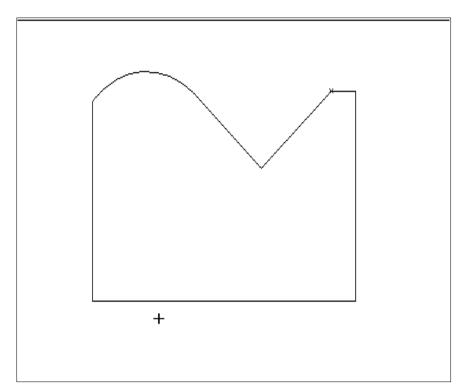


Figure 4-17 All geometry has been trimmed

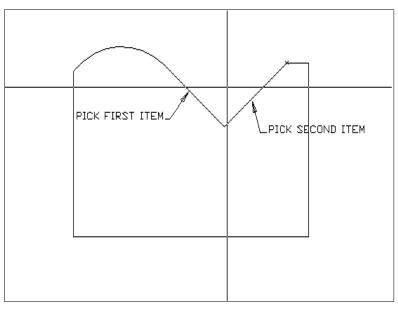


Figure 4-18 Filleting to lines

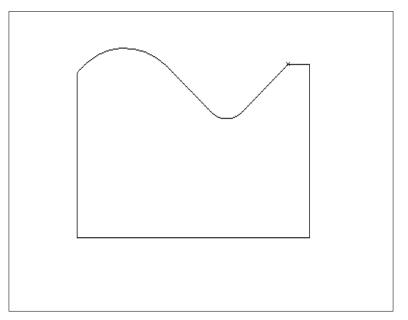


Figure 4-19 The part after filleting

<u>STEP 13</u>

Now lets draw the drilled holes. Select "Geometry", "Arc", then "Coordinates" from the menu (see figure 4-20). You will be asked:

Coordinate Entry	×
-X- Coordinate	Arc Radius
1.00000	0.12500
-Y- Coordinate	Start Angle
1.00000	0.00000
-Z- Coordinate	End Angle
0.00000	360.00000
Enter Arc	Coordinates
OK	Cancel <u>H</u> elp

<u>STEP 14</u>

Now let's copy the first hole to generate the other three holes. Select "Modify", then "Copy" from the menu. From the **"Entity Selection menu"** select **"SINGLE"** and place the "Bulls-eye Pick Cursor" anywhere on the first hole, and press the left mouse button. Next press the Right Mouse button and select "<Escape>" from the popup menu. Now select Done/Escape from the "Entity Selection" popup menu to exit the **"Entity selection mode"**.

Next the "copy entities" dialog will be displayed. We want to generate a grid pattern of holes, with a single row, and 4 columns. So fill in the dialog as shown (You will notice that both the circular pattern, and single copy sections of the dialog are disabled once "Grid" is selected).

Copy Entities	×
Copy Type O Single Copy O Circular Pattern O Grid pattern	Circular Pattern Number of Copies : 1
Single Copy	Grid Pattern
New Layer : 0 -	Number of Columns : 4
	OK Cancel <u>H</u> elp

Press the [OK] Button.

Next you will be asked for the X,Y Distance Between Columns and Rows

Select "Coordinates" from the popup menu, and enter:

Coordinate Entry X
-X- Coordinate 1.00000
-Y- Coordinate 0.00000
Enter X/Y Distances
OK Cancel <u>H</u> elp

The screen will now look like figure 4-21

Note:	This	step	is pure	elv fo	or drawi	ng	purp	oses,	if al	l that	you	want	to do	is a	drill t	hese	holes,	this ste	p could	<u>be</u>
<u>skipp</u>	<u>ed.</u>	-	-								•							-		

<u>STEP 15</u>

At this point the part geometry has been generated and it is a good time to save the work done so far. You should save your work every few minutes, so if anything goes wrong (power-out, disk fails, etc.) not all of your work will be lost.

To save the drawing, select "File", then "Save" from the menu. At this point you will be asked:

Save As							? ×
Save <u>i</u> n:	🔄 Draw		•	£	d	 	
Enversion of the second		ostest2.PSD ut1.PSD					
File <u>n</u> ame:	LESSON1					<u>S</u> ave	9
Save as <u>t</u> ype:	POWERSTA	TION Drawings		-		Cance	el

Enter LESSON1 as the file name, and press [Save]. (Notice, that since the file has never been saved, you were automatically given the "Save As" dialog.

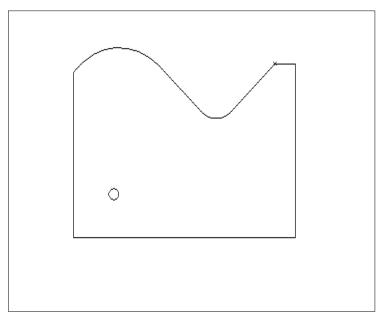


Figure 4-20 The first hole

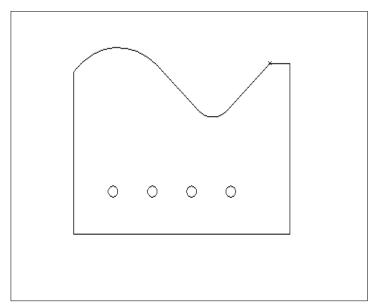


Figure 4-21 The copied holes

<u>STEP 16</u>

MACHINING

POWERSTATION uses a method of operation modeled after the way a machinist thinks. We call this method "OPERATION ORIENTED". NOTE: POWER-CAD users, please skip ahead to step #27 (Dimensioning).

The first step is to define one more operations that will take place. The second step is to select the operation that you want to work on, and finally do the machining for that operation.

Note that the operations must be specified in the order that you want the part machined in, but they do not have to be selected & worked on in any specific order. POWERSTATION will automatically generate the operations in the originally defined sequence no matter what order the operations were selected in.

If you make a mistake while doing any machining operations, first try pressing the <F7> (DELETE-LAST) key. If this does not produce the desired results, simply select the "Machining", "Tool Path|Edit", "Delete-Operation" command to delete all tool motion from the current operation, re-select the operation and try again.

Select "**Machining**", "**Operations**" from the menu (or press the operations ICON on the icon bar (located on the left side of the screen). Not sure which is the "operations" icon ? Simply hold the mouse cursor over the icon in about two seconds, a "Hint" will be displayed describing the function of the icon.

K Machining Operations	_ 0	×
Operation Help		
Current Op # Post Color Description	Operation List Detail	Tòl
	s Material	J

Figure 4-21a (The Machining Operations Manager)

To enter an operation:

- 1) Click the cursor in the "Op #" column, and type in the number of the operation.
- 2) Double click the mouse on the "Color" column, and select the desired color.
- 3) Click the cursor on the "Description" column, and enter a description of the operation
- 4) Double click on the first "Current" column, and the word "Yes" will appear. This indicates that this is now the currently selected operation.
- 5) At this point the dialog should look like figure 4-21b
- 6) Click on the tab that reads "Details". This will switch to the "details" page where you enter the specifics on the currently selected operation (tool number, diameter, etc..). See figure 4-21c

<u>/</u>	Machin peratio	ing O n	peratio	ons			×
	<u>B</u> 📑	A	l	d i:			
	Currer	Op #	Post	Color	Description		
	Yes	10	X		ROUGH PROFILE .75 END MILL	ę	
				, , , +		eratio	Material
				 		Operation List	1.81
			+	 		Ĩ	
		, 	+	 - 			
			+	 		Ш	
			+	- 		Details	Tooling
		- -	+	 		6	£90
			+	 			
			<u>.</u>	 	▼		

Figure 4-21b (Basic operation information)

Machining Opera	tions	
🗕 🛤 🗎	Alla 📔	
Operation #	Operation Type Tool Number Offset Number	Operation
1	0.75000 0.05000 0.00000	tion List
⊙ IPM ○ IPR 10.00000	 ● RPM ○ CSS 2200.00000 Flood ▼ Forward Range #1 ▼ Set Home 	
Description:	Set <u>G</u> age	
ROUGH PROFILE	.75 END MILL	<u>D</u> etails
	Eigene 4.21s (On suntion #10 Details)	

Figure 4-21c (Operation #10 Details)

	lachin eratio	ing Op n	eratio	ons			×
	¥ 🛤		l	rd o			
C	Currer	Op #	Post	Color	Description		
Y	'es	10	×		ROUGH PROFILE .75 END MILL	р р	
		20	×		FINISH PROFILE .5 END MILL	erati	Meteriel
		30	X		1/4" DRILL	Operation <u>L</u> ist	rigl
						st	
				.			
				.			
				 		Details	Tooling
			 	 		ils.	ting:
				1 1 1 1 -			
				 	▼		

Figure 4-21d (All operations for the tutorial part)

Machining Opera	tions	
Operation	/di	
Operation # 20 🗣 Fixture Offset	Operation Type Tool Number Offset Num Milling Tool Diameter Stock Allowance Corner Rad	■ pp
Feed O IPM O IPR	0.50000 0.00000 0.00000 Spindle / Speed Coolant ⊙ RPM ○ CSS 3000.00000 Flood	
7.50000 Description:	✓ Forward Range #2 ▼ Set Hom Set Gage	4
FINISH PROFIL	E .5 END MILL	Details

Figure 4-21e (Operation #20 for the tutorial part)

Next define the remaining operations, using the information displayed in figures 4-21d,e, & f.

Machining Opera	tions	- 🗆 X
🛛 🙀 📑	/du 🔰	
Operation # 30	Operation Type Tool Number Offset Number	Оре
Fixture Offset	Tool Diameter Stock Allowance Corner Radius 0.25000 0.00000 0.00000	Material Operation <u>L</u>
Feed O IPM O IPR	Spindle / Speed Coolant :	List
5.00000	▼ Forward Range #1 ▼ Set <u>Home</u> Set <u>G</u> age	
Description:		_ =
		Details

Figure 4-21f (Operation #30 for the tutorial part)

<u>STEP 17</u>

At this time it is a good idea to make a print out of the defined operations. To do this select "Operation", "Print" from the menu. Next we need to make operation #10 active. To do this, double click on the "Current" column for operation #10, then exit from the Machining Operations Manager.

<u>STEP 18</u>

It is a good practice to place your machining on a different layer than the part geometry. While we are dimensioning the part we really do not need to see the tool path, so placing it on its own layer gives us an easy way to "hide" the tool path.

To change the current layer to 100, press the $\langle F9 \rangle$ key (or select "Display", "Layer-Control" from the menu), then click on "Current Layer" box and type: <u>100</u>, click the cursor on the description box (The right most column in the layer 100 row), and enter in a description like "Machining". Next press the [OK] button.

<u>STEP 19</u>

Num	ON	Color	Used	Style	Description 🔺
98	X			Solid	All <u>O</u> n
99	X			Solid	
100	X			Solid	Machining All Off
101	X			Solid	
102	X			Solid	Show Used
103	X			Solid	
104	×			Solid	
105	X			Solid	Current Layer
106	X			Solid	· · · · · · · · · · · · · · · · · · ·
107	×			Solid	100 🜩
108	×			Solid	
109	×			Solid	Line Style
110	X			Solid	▼
	<u> </u>		- <u></u>	F	

To be safe let's save the file again. This time, just press the right mouse button with the cursor anywhere in the main display area. A short menu will popup, Select "Save File".

<u>STEP 20</u>

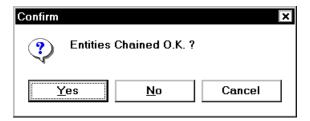
Now we can start rough machining the profile. Before any machining can be done we must select an operation. This should have been done at the end of step 17, but just in case you missed it, select "Machining", "Operations". Double click the mouse in the "Current" column of operation #10, then select "Operation", "Exit" (Or press the exit ICON).

<u>STEPS 21/22</u>

Now we will use "chain selection" to mill the outside profile. Select "Machining", "Move & Cut", "Chain" from the menu. We will be asked:

Select starting line/arc (figure 4-24)

Select the Line or Arc to stop before, <Escape> for none (press the right mouse button, and select <Escape>)



Press the [Yes] Button (figure 4-25)

Pick the side to offset to (figure 4-26)

General Type : Milling 🔽	Multiple -Z- Depths Cutter Comp Multiple Stock Approach Depart
-X/Y- Cutting Feed : 4.00000 -Z- Cutting Feed : 2.50000 -Z- Clearance Plane : 0.10000 Total -Z- Depth : -0.75000 Dptions ☑ Rapid To Start Position	Approach Type O None O Arc In O Helical Arc In O Linear XYZ Ramp In Approach Type Radius/Length 0.5 Sweep/Angle 225.00000 V ABS Angle
 ☑ G41/42 Cutter Compensation ☑ Custom Approach/Depart ☑ Retract -Z- When Done ☑ Multiple -Z- Depths ☑ Stock Removal Cycle 	% From End to Start

General Type: Milling 🔽	Multiple Stock Approach Depart Multiple -Z- Depths Cutter Comp
-X/Y- Cutting Feed : 4.00000 -Z- Cutting Feed : 2.50000 -Z- Clearance Plane : 0.10000 Total -Z- Depth : -0.75000	Cutter Compensation C Cutter Left C Cutter Right
Options	O Automatic
 Rapid To Start Position G41/42 Cutter Compensation Custom Approach/Depart 	Compensation Number : 1
 Retract -Z- When Done Multiple -Z- Depths Stock Removal Cycle 	Turn Off Compensation at End

Enter the information (-Z- feed, clearance, and total depth, etc..) as displayed above. Note; Depending on your preference, you can either leave the "Depart" settings to their default of "none", or set them the same as the "Approach" settings. The display will look like figure 4-27

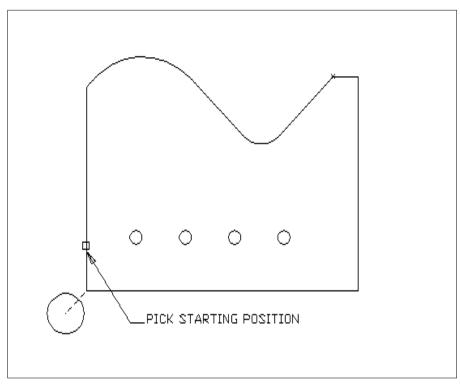


Figure 4-24 Selecting the starting line or arc

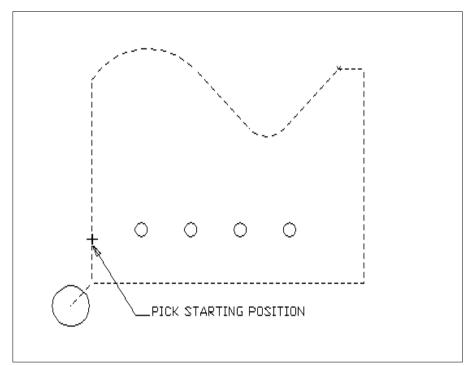


Figure 4-25 The entities have been chained

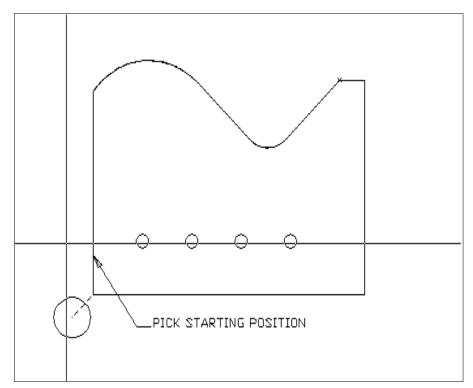


Figure 4-26 Picking the side to offset to

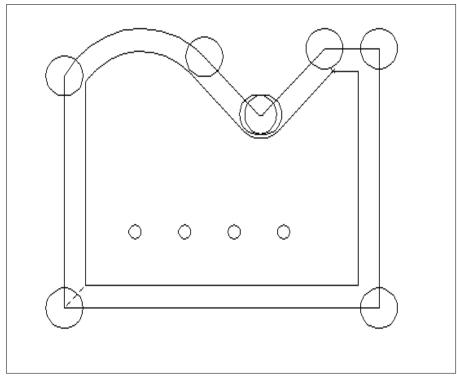


Figure 4-27 The first operation completed

<u>STEP 23</u>

Select operation #20 by selecting "Machining", "Operations" from the menu (or clicking the "Operations" icon). Double click on the "Current" column, in the row for operation #20. Exit the Machining Operations Manager.

Next repeat steps 20 through 22. When done the display will now look like figure 4-28.

Note: To finish with the same tool, do not select a new operation (#20), simply select "Machining", "Tool Path Edit", "Stock Allowance", and change the stock allowance to 0.0 before repeating steps 20 through 22.

<u>STEP 24</u>

Now for the drilling. Select operation #30 by selecting "Machining", "Operations" from the menu (or clicking the "Operations" icon). Double click on the "Current" column, in the row for operation #30. Exit the Machining Operations Manager.

Select "Machining", "Drilling", "Linear" from the menu.

Next you be given the "GET POINT" menu, (See chapter "basics" for more information on this menu) and be asked:

Start Point for Linear Pattern

Select "Center" from the popup menu, then move the pick box on to the first hole and press the left mouse button.

Linear Pattern	×					
Number of Holes :	4					
End Point of Pattern is K	nown					
Distance / Angle Distance Between Holes	: [1.0					
Base Angle of Holes : 0.0						
OK Car	ncel <u>H</u> elp					

Fill in the "Linear Pattern" dialog as shown above.

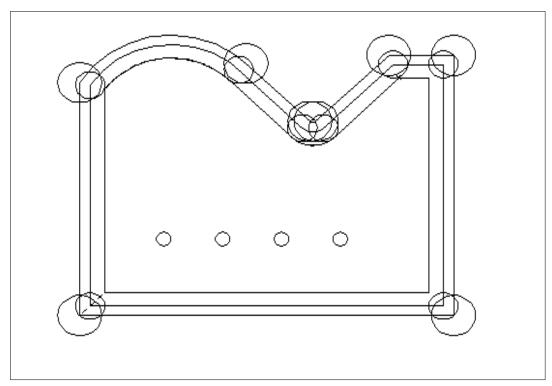


Figure 4-28 The second operation completed

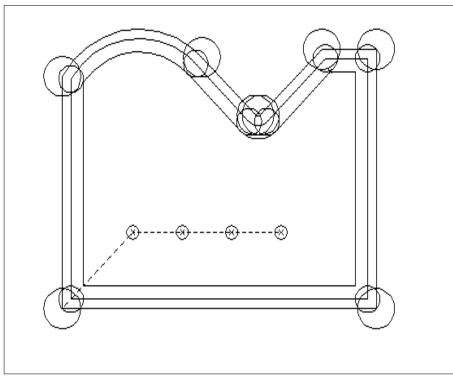


Figure 4-29 Drilling a linear pattern

STEP 24 continued

Next you are asked to select the type of drilling cycle, and the drilling depth. Select "Through" from the "Depth Specifier" list, and enter -.75 as the "Depth to Drill Through".

Drill Cycle Type	Drill Depth
Standard Drilling	Depth Specifier : Through Rapid Plane (-R-)
Deep Drill Peck Increment	0.1 Drill Tip Angle : 118.0
Tapping TPI: 16.0	Depth to Drill Through : -0.75000
Pitch : 0.0625	Center Drill Information
Dwell Time	Note: Only Center Drills Large / Small Enough to generate the Hole Diameter Speficied Above Will be Shown
Defaults	OK Cancel <u>H</u> elp



34 POWERSTATION <u>STEP 25</u>

Once again, let's save the file as in step 15. After that, let's check the machining status. Select "Machining", "Machining Status". When done viewing the status, press the [OK] button.

<u>STEP 26</u>

The final step in machining is to run the post processor. Select "Machining", "Post (Generate NC-Code)" from the menu (or select the Post icon).

From the list of machines in the upper left hand corner of the dialog box, move the highlight to "DMILL.MCH".

Next we will set the name of the NC-tape file to be created. Press the button labeler [Set NC-Code File Name]. Enter "LESSON1" as the "File Name", and press [OK].

Press the [Go] button. The post processing now begins, and may take a few minutes. When done, press [Close] to leave the post processor.

NOTE: You will need to add your own machine names to the post processor. For information on add them, select "Help", "Post Processor Setup".

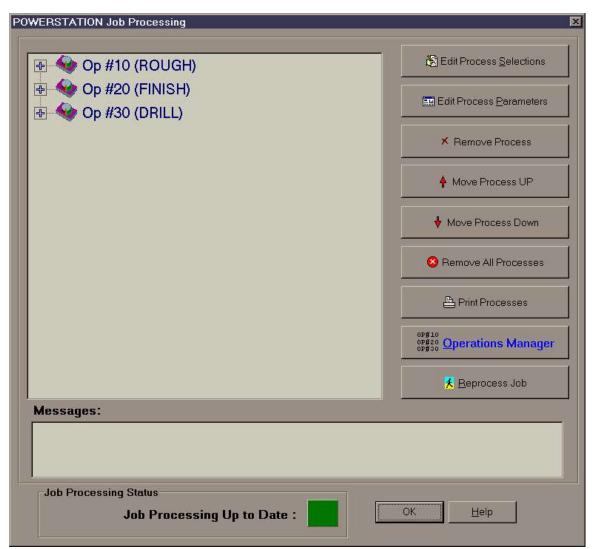
Machine Name DLATHE.MCH DMILL.MCH	-NC-Code File Name : LESSON1.TAP
SUBMILL.MCH Set NC-Code File Name Error Messages: Line Number : 216	
NC-Code N490 G80 N500 M09 N510 G28 Z0 M29 N520 M30 %	

Now that the tutorial part is finished, lets see how the new "Job Processing" feature can help with making changes.

NOTE !!! The Following steps 45-51 are NOT available/possible in the XPERT Version of POWERSTATION.

<u>Step 45</u>

Select "Machining-Job Processing". The following dialog will be displayed:



<u>Step 46</u>

In the Job Processing Dialog, in the "Tree area" (Upper left) one at a time, click on the Plus "+" signs. The dialog will now look as follows:

POWERSTATION Job Processing	X		
Op #10 (ROUGH)	😫 Edit Process Selections		
□ ↓ Chain Cut □ ↓ Op #20 (FINISH)	Edit Process Parameters		
Chain Cut	★ Remove Process		
Drill Linear	A Move Process UP		
	♦ Move Process Down		
	Remove All Processes		
	Print Processes		
	OP#10 OP#20 OP#30 Operations Manager		
	<u> R</u> eprocess Job		
Messages:			
Job Processing Status Job Processing Up to Date :	OK <u>H</u> elp		

<u>Step 47</u>

For this example, we will make three common changes and show how quickly we can regenerate the tool path and the NC-Code. For this example, we will say that operation #10 which is currently a $\frac{3}{4}$ " end mill, leaving .05 stock, need to be changed to use a $\frac{7}{8}$ " end mill leaving .062 stock. In operation #30 the drilling needs to be done to a depth of Z-.6 rather than the current Z of -.5.

<u>Step 48</u>

In the Job Processing dialog, click on the [Operations Manager] button. Select operation #10, then click on the "Details" page. Change the details page to look as follows, then close the operations manager.

Mill Tutorial 37

	111111 1
Machining Operations Manager	
Operation Help	
Operation # Operation Typ Tool Number Offset Number 10 Milling 1 1 1 Fixture Offset Tool Diameter Stock Allowanc Corner Radius 0.87500 0.06200 0.00000 Feed Spindle / Speed Coolant : * Flood * 10.00000 * Flood * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * <t< td=""><td>Tooling Tool Shape Operation List Details Material</td></t<>	Tooling Tool Shape Operation List Details Material

<u>Step 49</u>

On the Job Processing dialog, double click on the line under operation #30 that reads "Drill Linear". First the linear drilling dialog will be displayed, press [OK] to leave the current settings. Next the Drill Depth dialog will be shown. Click on the box that reads "Absolute –Z- Position to Drill To" and change it from -.5 to -.6, press [OK].

Drill Cycle Type	Drill Depth
Standard Drilling	Depth Specifier : Absolute -Z-Location Rapid Plane (-R-) 0.1
Deep Drill Peck Increment 0.25 Tapping TPI: 16.0	Drill Tip Angle : 118.0 -Z- Top of Part: 0.00000 Absolute -Z- Position to Drill To : -0.60000
Pitch : 0.0625	Center Drill Information
Dwell Time	Note: Only Center Drills Large / Small Enough to generate the Hole Diameter Speficied Above Will be Shown

38 POWERSTATION <u>Step 50</u>

In the Job Processing Dialog, press the [Reprocess Job] button. In a few seconds the entire tool path will be regenerated.

<u>Step 51</u>

At this point, the new tool path is generated. To regenerate a new NC-Code file, simply repeat step #26 (Run the post processor).

DIMENSIONING

With the machining done, we will probably want to produce a dimensioned drawing. We will add dimensions, a title block, and a side view to the drawing, then make a hard copy plot on the printer.

<u>STEP 27</u>

While dimensioning we do not want to see the tool path, so we will turn off its layer, and set the a new default layer (#101) for the dimensioning.

Start by pressing the $\leq F9 \geq$ key to display the layer control dialog. First to change the current layer, type 101 in the "Current Layer" box, Click on the "description" column for layer 101 and enter "dimensioning". Finally click on the box displayed in the "ON" column, in the Layer 100 row, until the "X" is not displayed in the box.

Num	ON	Color	Used	Style	Description	▲	
96	×			Solid	 		All <u>O</u> n
97	×			Solid			
98	×			Solid	T		All O <u>f</u> f
99	×			Solid			
100				Solid	Machining		Show <u>U</u> sed
101	×			Solid	Dimensioning] 	
102	×			Solid	T		
103	×			Solid	T		Current Layer-
104	×			Solid	T		
105	×			Solid	T		101 🌻
106	×			Solid	T		
107	×			Solid	T		Line Style
108	×			Solid	T		
					T	<u>\</u>	L

At this point the display should look like figure 4-31

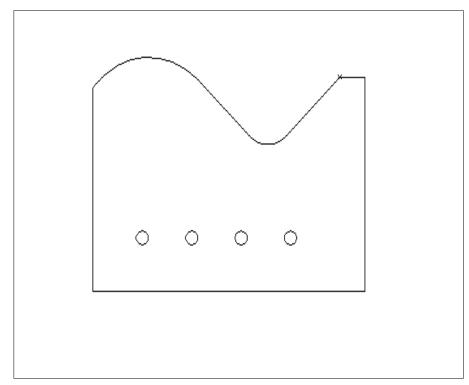


Figure 4-31 Display with the tool path layer off

<u>STEP 28</u>

On a color display, it is generally appealing to have the dimensions displayed in a different color that the part, so press the $\leq F8 \geq$ key and select a new color.

<u>STEP 29</u>

First we will dimension the overall width of the part. Select "**DRAFTING**" from the main menu, then "**DIMENSION**" then "**Horizontal**" from the sub menu.

For the **"First Extension Line Origin"** select **"END-OF"** from the sub menu then pick the lower left corner of the part (figure 4-32). Next select **"END-OF"** and pick the lower right corner of the part.

Indicate the approximate text point (See figure 4-33)

The dimension should look like figure 4-34

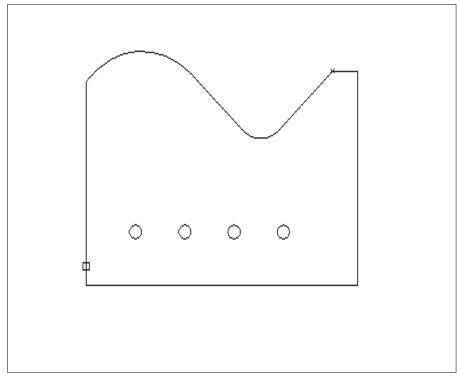


Figure 4-32 Extension line origin

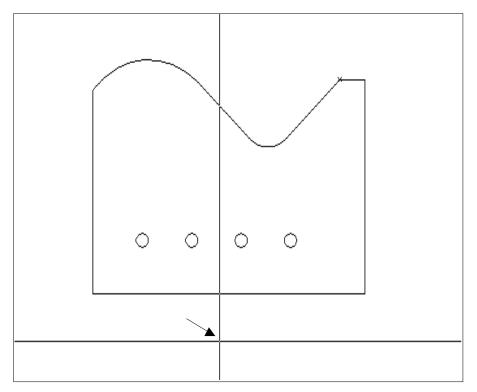


Figure 4-33 Indicating the approximate text point

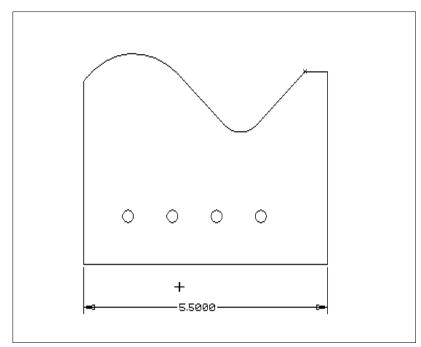


Figure 4-34 Dimensioning the overall width

<u>STEP 30</u>

Now we will dimension the overall height. Select "DIMENSION" then "Vertical" from the sub menu.

For the **"First Extension Line Origin"** select **"END-OF"** from the sub menu then pick the lower right corner of the part (figure 4-35). Next select **"END-OF"** and pick the upper right corner of the part.

Indicate the approximate text point (See figure 4-35)

Diameter Dimension [N/Y] ? <<u>Enter></u>

The dimension should look like figure 4-36

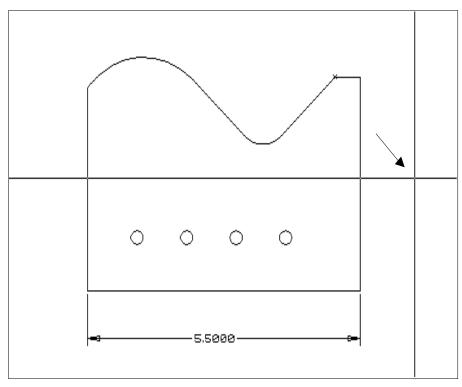


Figure 4-35 Constructing the vertical dimension

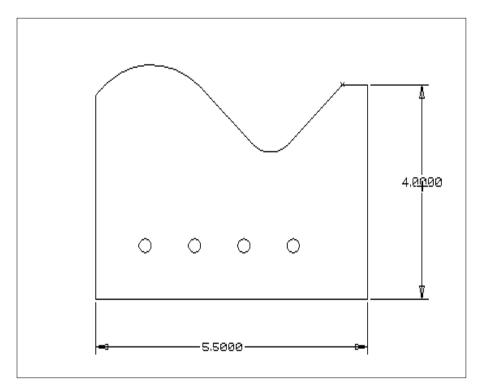


Figure 4-36 Dimensioning the overall height

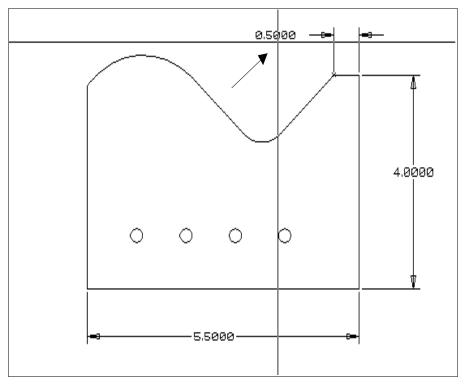


Figure 4-37 Dimensioning the top most line

<u>STEP 31</u>

Try to dimension the top most line of the part using step 29 and figure 4-37, as examples.

<u>STEP 32</u>

Now we will add the radius dimensions. Select "**DIMENSION**" then "**RADIUS**" from the sub menu. When asked to "**Select a Arc**", move the pick box over the 1/2" radius and press the left mouse button.

Indicate the approximate text point as shown in figure 4-38

<u>STEP 33</u>

Dimension the 1 3/8" radius as in step 32.

Indicate the approximate text point as shown in figure 4-39

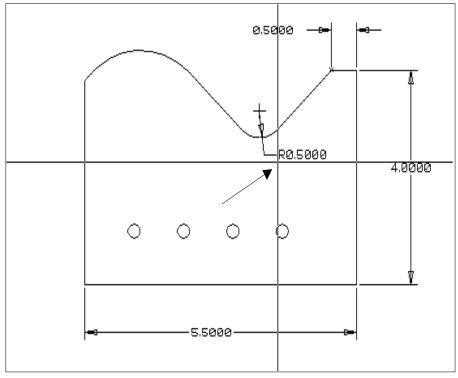


Figure 4-38 Dimensioning the 1/2" radius

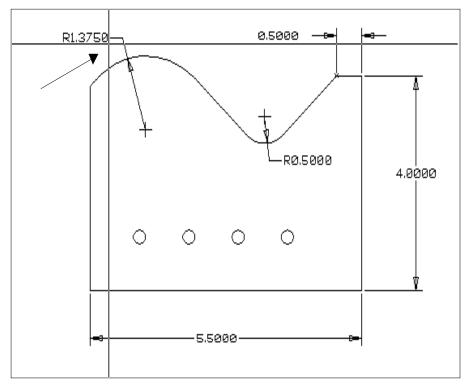


Figure 4-39 Dimensioning the 1 3/8" radius

<u>STEP 34</u>

Now let's dimension the center of the 1 3/8" radius and center of the first hole. Select "**DIMENSION**" then "**ORDINATE**" from the sub menu.

You will be given the "GET POINT" menu to select a base point for the ordinate dimensions. Select "END-OF" from the sub menu and pick the lower left corner of the part.

The message area of the screen will read:

"Point for ordinate Dimension, Right button when done"

Select "END-OF" from the sub menu and pick the bottom of the center mark. Indicate the approximate text point as shown in figure 4-40

Next select "END-OF" from the sub menu and pick the right most point on the arc center mark. Indicate the approximate text point as shown in figure 4-41

Next dimension the first hole by selecting "CENTER" from the sub menu for both the X and Y dimensions.

Press the right mouse button and select <Escape> from the popup menu to end the ordinate dimensioning mode.

The part should look like figure 4-42

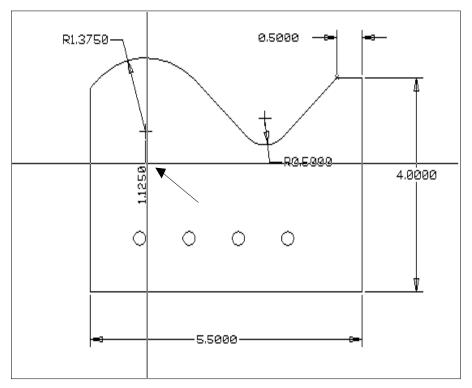


Figure 4-40 Dimensioning the X center of the 1 3/8" radius

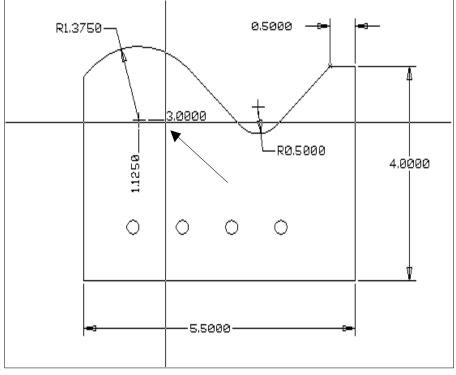


Figure 4-41 Dimensioning the Y center of the 1 3/8" radius

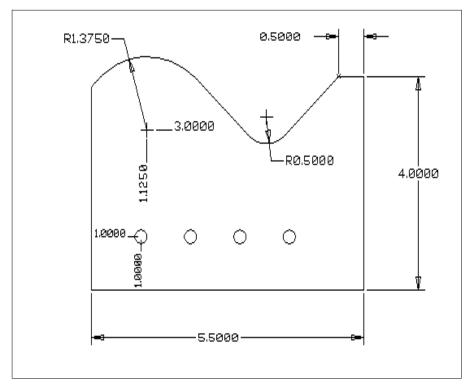


Figure 4-42 Ordinate dimensions

<u>STEP 35</u>

Now let's dimension the angle. Select "DIMENSION" then "ANGULAR" from the sub menu.

Select Line #1 (select the line to the right of the 1/2 radius)

Select Line #2 (select the line to the left of the 1/2 radius)

Indicate the approximate text point as shown in fig 4-43

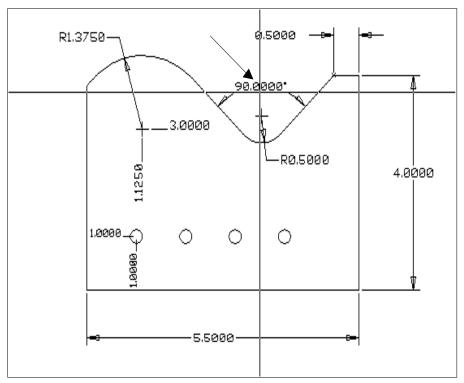


Figure 4-43 Constructing the angular dimension

<u>STEP 36</u>

Now we will dimension the distance between the holes. Select "DIMENSION" then "HORIZ" from the sub menu.

For the **"First Extension Line Origin"** select **"CENTER"** from the sub menu, and pick hole #3. For the **"Second Extension Line Origin"** select **"CENTER"** from the sub menu, and pick hole #4. Next indicate the approximate text point as shown in figure 4-44

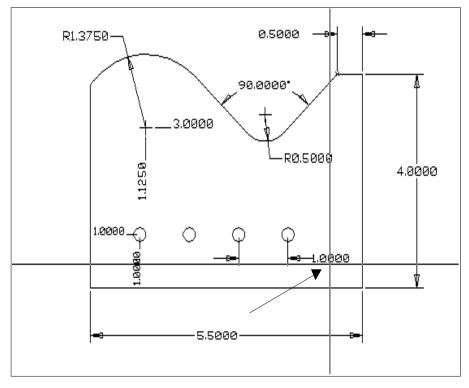


Figure 4-44 Dimensioning the distance between holes

<u>STEP 37</u>

Now we will add the text "typical" under the last dimension. Select "TEXT" from the sub menu. You will be asked:

```
Text: A)Ligned, L)eft, R)ight,
or C)enter Justified [L/R/C/A] ? <u>A<Enter></u>
```

Next enter the approximate text start and end points as shown in fig 4-45

Finally enter the text "Typical".

<u>STEP 38</u>

Now to add a finished look to our drawing, let's merge in a title block and sheet frame. Select "GEOMETRY" from the main menu, then "BLOCKS" then "MERGE" from the sub menu. A standard WINDOWS file open dialog will appear select the file named "SIZEB.PSB", then press the [OK] button.

When asked "Select the Block Insertion Point", select "COORDS" from the sub menu, and answer:

```
-X- value [0.0000] ? <u>-2.5<Enter></u>
-Y- value [0.0000] ? <u>-4.5<Enter></u>
Enter the Angle for Rotation [0.0000] ? <u><Enter></u>
Scale Factor [1.0000] ? <u><Enter></u>
```

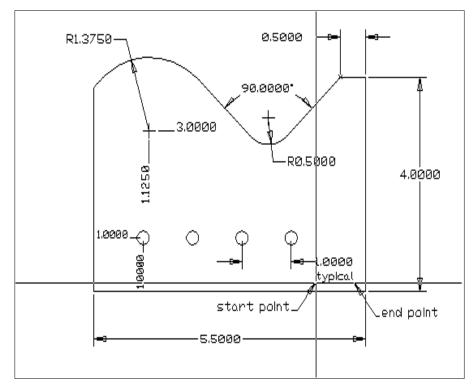


Figure 4-45 Adding text

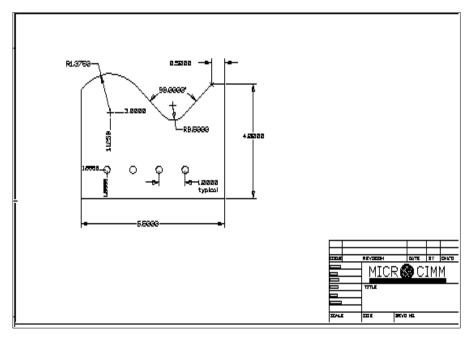


Figure 4-46 Title block and sheet frame

STEP 38 continued . . .

Notice that we are seeing very little of the block that has been merged. This is because most of it extends beyond the current display. To fit the entire drawing on the display, press the right mouse button anywhere within the main display area, and select "Zoom Extents" from the popup menu. See figure 4-46

<u>STEP 39</u>

Now let's generate a side view. Select "Geometry", "Line", "Box" from the menu.

When asked "First Corner of Box", select "Cursor" from the sub menu. Then move the cross hair cursor to the right of the part a few inches and up and down until it is close to the bottom line of the part. Press the left mouse button to enter this position.

Next when asked "Width & Height of Box", select "Coordinates" from the popup menu, and answer the questions:

```
-X- value [0.0000] ? <u>.75</u>
-Y- value [0.0000] ? <u>4</u>
```

The display should now look like figure 4-47

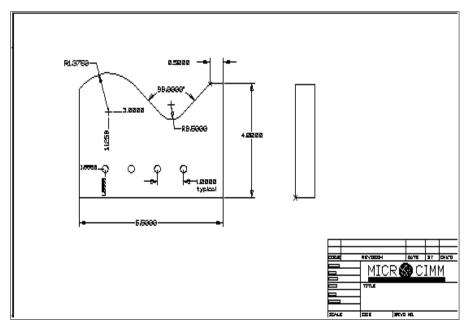


Figure 4-47 Adding a side view

<u>STEP 40</u>

Now we will dimension the part thickness using the side view we generated. Select "**DRAFTING**" from the main menu, then "**DIMENSION**" then "**Horizontal**" from the sub menu.

When asked for the "First Extension Line Origin", select "END-OF" from the sub menu, and pick the top left corner of the side view.

When asked for the "Second Extension Line Origin", select "END-OF" from the sub menu, and pick the top right corner of the side view.

Next indicate the approximate text point as shown in figure 4-48

<u>STEP 41</u>

Now we will add the drawing name to the title block. This will be easier to do if we "ZOOM IN" on the title block. To zoom, press the right mouse button, anywhere within the main display area, and select "Zoom" from the popup menu. Using the mouse, select any two diagonal corners of the title block.

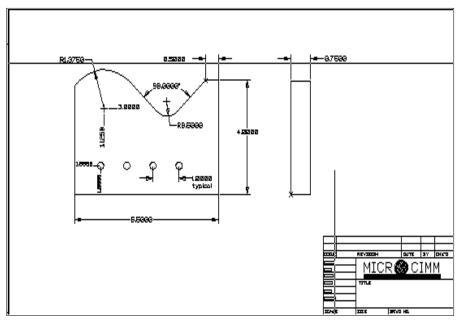


Figure 4-48 Dimensioning the part thickness

IZZUE	REVISION	DATE	ΒY	СНК′Ъ
DRAWN DATE CHK'D	_ MICR	0 C	ΙM	Μ
APPR. DATE SYSTEM				
SCALE	SIZE DR	WG NO,		

Figure 4-49 Zooming in on the title block

STEP 41 continued ...

To add the text, select the **"DRAFTING"** from the main menu, then **"TEXT"** from the sub menu. You be given the text entry dialog:

-Text Justification— O Left	Text Height : 0.25000
O Right	Rotation Angle : 0.00000
O Center ⊙ Alligned	Slant /Tilt Angle : 0.00000
Enter Text :	Special Symbols (none) <u>I</u> nsert
DRAFTING	
4	

Select the text start point and end point as shown in figure 4-50.

<u>STEP 42</u>

To return the display to the previous view, press the right mouse button, and select "Zoom previous" from the popup menu.

<u>STEP 43</u>

Save the drawing as in step 15.

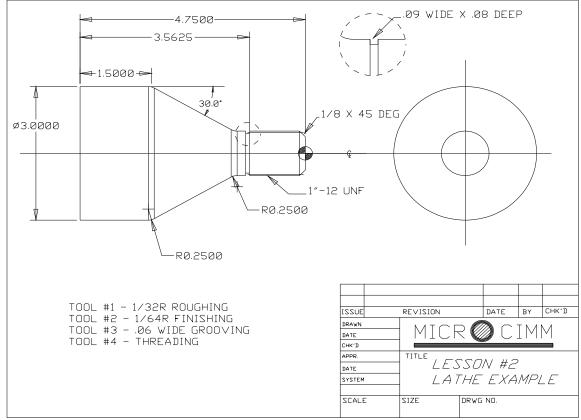
ISSUE	REVISION	DATE	ΒY	Снкл		
DRAWN			тки	М		
DATE	_ MIC	MICR 🏈 CIMM				
СНК′Д						
APPR.	TITLE					
DATE			1 1	1		
SYSTEM		ISSON	1	F⊥l		
	(1		
SCALE	SIZE	DRWG NO,				

Figure 4-50 Adding the drawing name

Note: For information on how to permanently change the default title block to have your own company name/logo, please see the "Applications Cookbook". <u>STEP 44</u>

Finally let's print the drawing. To print, select "File", "Print" from the menu.

The "display" file generation will take a few seconds. When done, the printing will start.



LATHE TUTORIAL

Figure 4-60 The Sample Lathe Drawing

The purpose of the tutorial is to guide you step by step through the creation and machining of a lathe part. The drawing shown in figure 4-60 will be taken from the initial geometry creation, to machining, and finally generate the NC-Code. Each step will be numbered for easy reference. The information you are required to type will be underlined.

<u>STEP 1</u>

We begin by starting POWERSTATION. To do this: Press [Start] (on the WINDOWS Tool bar), then select "Programs", then "MICROCIMM", then "POWERSTATION"

<u>STEP 2</u>

First we will ask POWERSTATION to display the coordinate axis, so we have a simple reference as to where the 0,0 point is. To turn the AXIS on, select "Display", "Axis" from the main menu.

<u>STEP 3</u>

If instead of simple axis lines, you would like to see a labeled ruler on the screen, select "Display", "Axis" (to turn of the axis display), then select "Display", "Ruler" to turn on the ruler.

	STATION 2000			Mashisian	Diseler	Hele			-	
<u>File</u> dit	<u>G</u> eometry	Modify	Drafting	Machining s ▼ Select	D <u>i</u> splay	Help	@		🛯 🗶 XY - View	T
×12.7357		zo.00000	Laye				200	·		
▶ ←	2									-
LOAD DXF					4					
₫ <u>₿</u> (5										
₿ • ₿ ₿ • ₿										
👫 🐳					13	\rightarrow				
0P#10 0P#20 0P#30										
A 🔛										
X N1G0 N2G1 N3										Ŧ
	Image: A state of the state									ł
Copyright@1	989-2000, Glass	House Soft	ware, Ltd.		T	ool: X 0.00000	Y 0.0000) Z 0.00	1000	1

Figure 4-62 The initial display showing the axis

Note: The following steps 4 through 21 show the "long Hand" method of developing the part geometry and trimming it. These methods are shown primarily as an example of how to build geometry step by step. This method has application to everything from the simplest to the most complex parts. However: In a case like the sample lathe part we are working on here, POWERSTATION has a much simpler and quicker way of generating the part geometry. After completing steps 4-21, please take a look at step 21A it shows how to generate the same part geometry using far fewer steps, using the built in "Easy Geometry" command.

<u>STEP 4</u>

Next we will create a vertical line for the front face of the part. Select "Geometry", "Line", "Vertical" from the menus. You will then be prompted:

Vertical Distance [0.00000] ? <Enter>

See figure 4-63.

Next we will create a horizontal line for the 1 inch diameter. Select "GEOMETRY" Then "LINE" and then "Horizontal" from the menus. You will then be prompted:

Horizontal Distance [0.00000] ? .5 or 1/2<Enter>

Note that all turned diameters are entered as a radius value. In the above example the radius was simple to calculate so .5 was the answer. In place of typing in the radius, you may simply type in the diameter followed by "/2" (divide by two).

See figure 4-64.

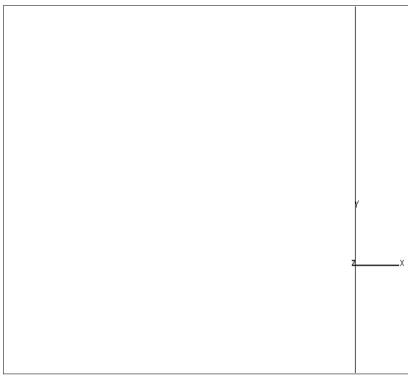


Figure 4-63 Defining the Front Face

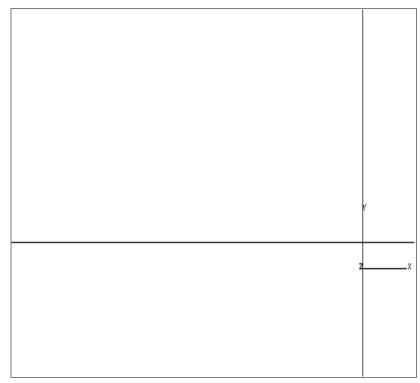


Figure 4-64 One inch diameter

<u>STEP 6</u>

Building the 30 degree line. There are several methods we could use to build this line. We will first create a construction point, and then pass the 30 degree line through it.

Select "GEOMETRY" - "POINT" - "Coordinates" from the menus. You will be prompted:

-X- Coordinate			
-3.25000			
-Y- Coordinate			
1.50000			
-Z- Coordinate			
0.00000			
Enter Po	nt Coordi	nates	
Enter Po	nt Coordi	nates	

Note: The -X- Coordinate can be entered as: -4.75+1.5And the -Y- Coordinate can be entered as: 3/2

See figure 4-65.

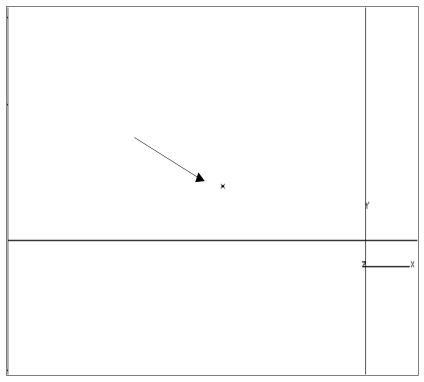


Figure 4-65 The construction point

Next select "Geometry", "LINE" - "Tangent to One" from the menus. You will be prompted:

Select a Point or Arc, or <Esc> for Point Menu

At this time we will select the Point we just made. Figure 4-66.

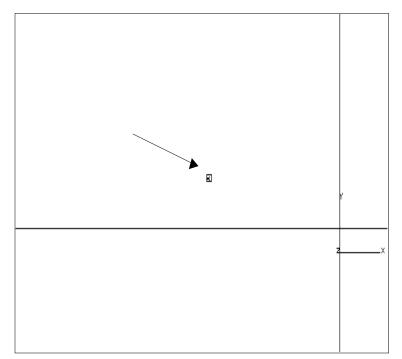


Figure 4-66 Selecting the point

Enter the Angle [0.00000] ? -30<Enter>

The display will now look like figure 4-67.

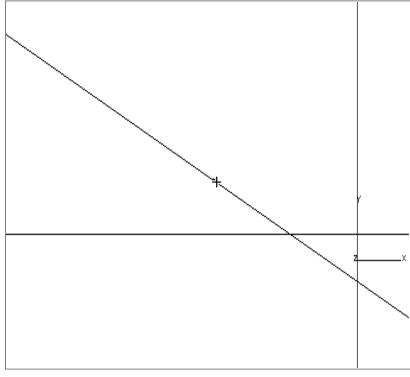


Figure 4-67 The 30 degree line

<u>STEP 7</u>

At this point it is a good time to save the work done so far. You should save your work every few minutes, so if anything goes wrong (power-out, disk fails, etc.) not all of your work will be lost. To save the drawing, select "File", "Save" from the main menu. Enter "LESSON2" in the file name box, and press the [OK] button.

Save As Save <u>i</u> n:	🔄 Draw		•	È		? ×
Demodim. Fonts.psd Lathe1.psd Lathe2.psd Lathe3.psd Mill1.psd Mill2.psd	ste Stuti.	ame.psd ist2.PSD PSD				
File <u>n</u> ame: Save as <u>t</u> ype:	LESSON2 POWERSTATIO	N Drawings		•	<u>S</u> ave Cance	

<u>STEP 8</u>

Next we will build the 3 inch diameter. We could use the "LINE"-"HORIZ" command that we used in creating the 1 inch diameter. For this example we will use a different method, using the point and "GEOMETRY" - "LINE" - "Tangent to One".

Select a Point or Arc, or <Esc> for Point Menu

Here we have a condition that you will face many times in the future. When you position the cursor over the point and press the left mouse button there is no way of knowing if the Point or Line will be selected. This is where the MASKS (See chapter BASICS) come into play.

By using a Point mask (Selecting "Point" in the Entity Mask list on the tool bar) we are telling the system **"IGNORE ANY ENTITY TYPE OTHER THAN A POINT"**. In this way we can be sure that we will be getting the point. Note: After executing this command, it is a good idea to set the "Entity Mask" back to "All Entities".

Enter the Angle [0.00000] ? <Enter>

See figure 4-68.

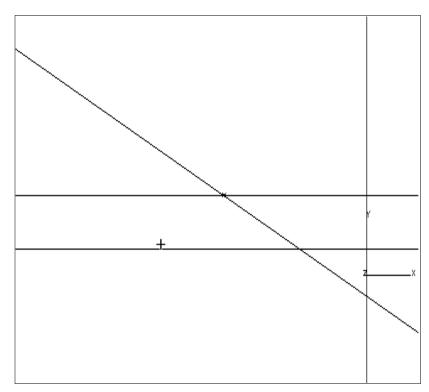


Figure 4-68 The 3 inch diameter

<u>STEP 9</u>

Now we will make the back face (We will be erasing this line later). Select "GEOMETRY" - "LINE" - "Vertical" from the menus.

Enter the Vertical distance [0.00000] ? -4.75<Enter>

See Figure 6-69.

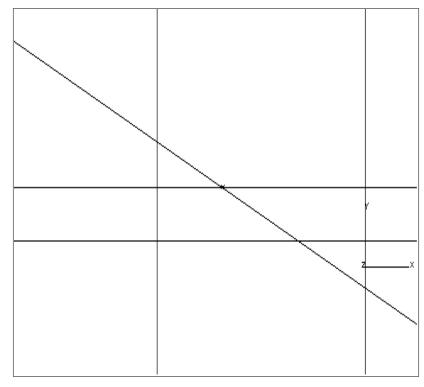


Figure 4-69 The back face

<u>STEP 10</u>

Now we will add a center line to the geometry. Although not required for machining, it will provide a needed edge for trimming, and a helpful visual reference. Select "GEOMETRY" - "LINE" - "Horizontal" from the menus.

Enter the Horizontal distance [0.00000] ? <Enter>

<u>STEP 11</u>

To further enhance the visual aide of the center line, we will now modify the line style so that it appears as a proper center line. Select "**Modify**" - "Line-Style" from the menus. Now select "SINGLE" from the entity selection method menu.

Pick the center line as in figure 4-70.

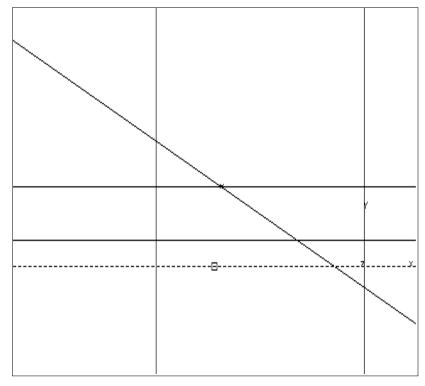


Figure 4-70 Picking the center line

STEP 11 CONTINUED ...

Press the right mouse button, and select <Escape> from the popup menu.

Next select Done/<Escape> from the selection menu. Now the line style dialog will be displayed. Change the line style to "Center", and press [OK].

Line Style	×
Line Style : Center — – – – – – Line Width (# of 1/64 increments) 1	▼ Line Width (Decimal) 0.01563
OK	Cancel <u>H</u> elp

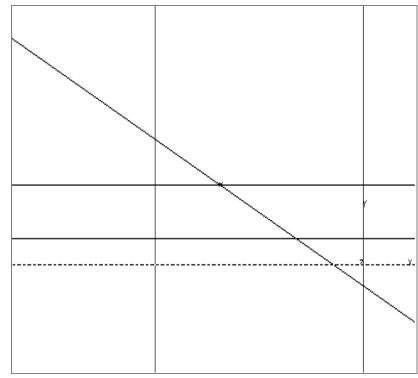


Figure 4-71 The completed center line

<u>STEP 12</u>

Now we will start to clean up the geometry until it looks like the part we are making. We will do this with the Trim/Extend and Break commands. The Trim/Extend and Break commands are found in the "MODIFY-Trimming" menu, and address the problem of cleaning up the geometry from two different directions.

With the **Trim/Extend** command you will first be asked to select the entity to trim, and then to select two cutting edges (Points, Lines, or Arcs). The entity to be trimmed will be **trimmed or extended** until it spans between the two cutting edges.

With the **Break** command, you will first be asked to select the cutting edge or edges (Lines or Arcs). You will now enter a loop where you simply point to the section of an entity that you want to remove. If the entity you select intersects one or more of the cutting edges, the section where you selected the entity will be removed, even if this requires splitting the entity into two pieces. Note that the BREAK command always removes, and will never add (Extend) the element selected.

We will start by trimming the face of the part. Select "**MODIFY -TRIMMING-Trim/Extend**" from the menus. At the prompts, select the entities as shown in figures 4-72 through 4-74. <u>NOTE: In the following examples the terms "Cutting Edge" & "Bounding Edge" will be used interchangeably.</u>

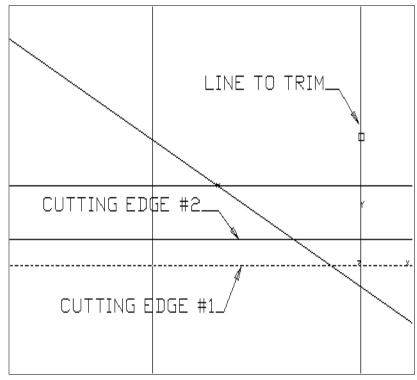


Figure 4-72 Selecting the line to trim

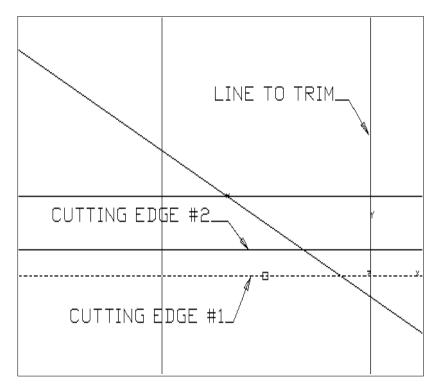


Figure 4-73 Selecting cutting edge #1

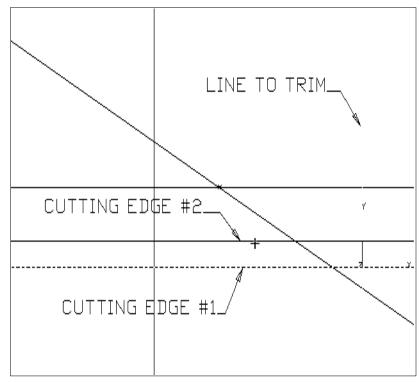


Figure 4-74 Front face after trimming

<u>STEP 13</u>

Trimming the 1 inch diameter. At this time we will use one of the **"HOT KEYS"** as defined in the chapter **"BASICS"**. It is a very common case where you may want to trim several entities, one after the other. Rather than going through the steps of selecting the **"MODIFY-TRIMMING-Trim/Extent"** command from the menus, you can simply press the **<Insert>** key to repeat the last command. The command can also be repeated by pressing the right mouse button, and selecting "Repeat Last Command" from the popup menu.

Now we will trim the 1 inch diameter as shown in figures 4-75 and 4-76.

<u>Notes on Trimming:</u>

The Trim/Extend command was used for all of the above examples to demonstrate the versatility of this command. It does not represent the easiest way to trim. All of the above examples could have been done using the "EASY-TRIM" command. After selecting this command you would simply point (with the mouse) to the sections of lines or arcs that you want to remove. Feel free to try the example trimming using this command.

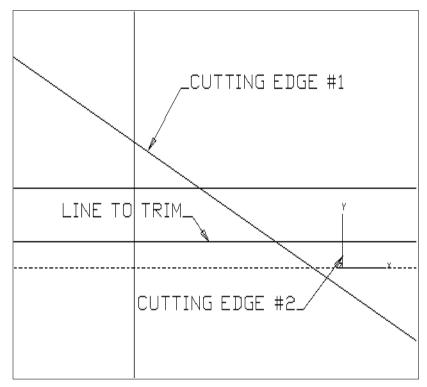


Figure 4-75 Trimming the 1 inch diameter

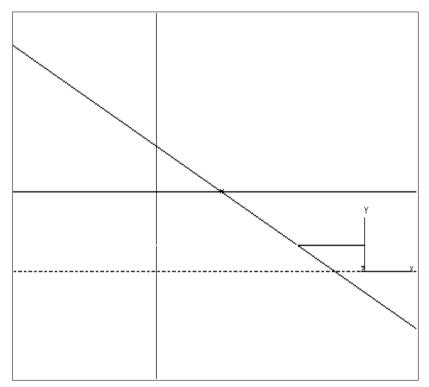


Figure 4-76 After trimming the 1 inch diameter

<u>STEP 14</u>

Just to show the difference between BREAK and TRIM/EXTEND we will be using the BREAK command on the 30 degree line. The TRIM/EXTEND could have also been used.

Select "MODIFY-TRIMMING-BREAK" from the menus.

Select cutting edges as in figure 4-77. Now select the section of the 30 degree line that you wish to remove. (Figures 4-78/79) A note on figure 4-79: if you select a location too close to the intersection with the 1 inch diameter, the BREAK command will not know exactly what to do, and so it will ignore your selection. If this is the case, keep trying locations further to the right until the line section is removed.

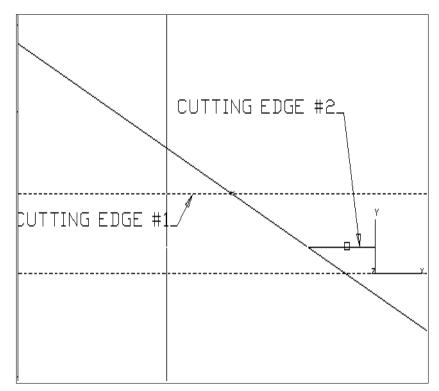


Figure 4-77 The cutting edges for breaking the 30 degree line

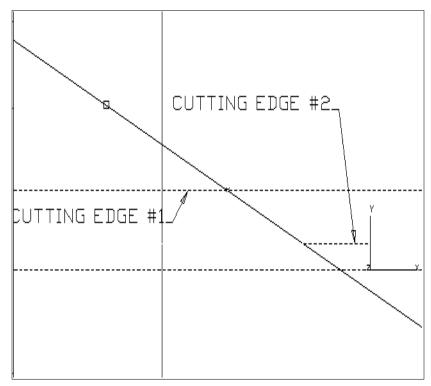


Figure 4-78 Breaking the top half of the 30 degree line

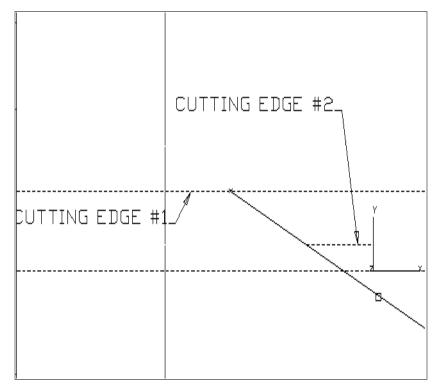


Figure 4-79 Breaking the bottom half of the 30 degree line

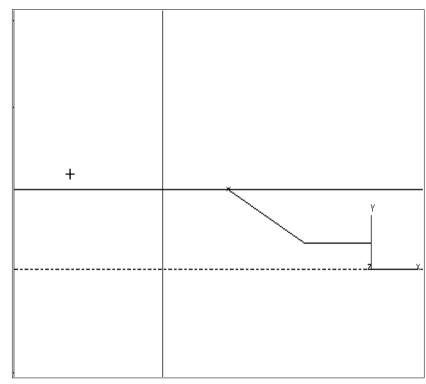


Figure 4-80 The 30 degree line after breaking

<u>STEP 15</u>

To trim the 3 inch diameter, select **"MODIFY-TRIMMING-TRIM/EXTEND"** from the menus, then select the cutting edges and line to trim as shown in figure 4-81.

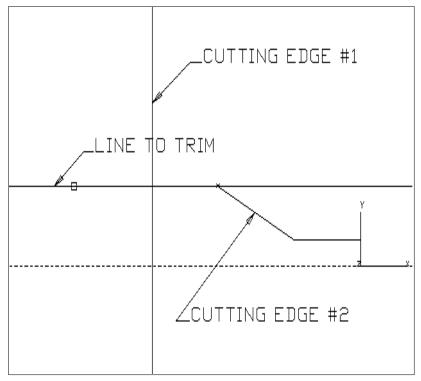


Figure 4-81 Cutting edges, for trimming the 3 inch diameter

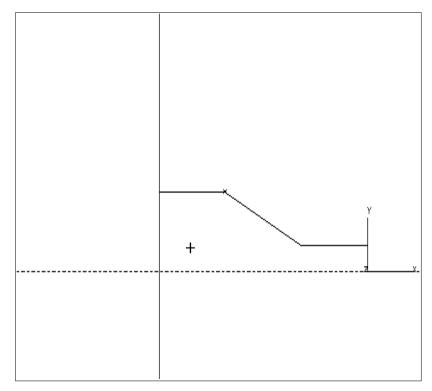


Figure 4-82 The 3 inch diameter after trimming

<u>STEP 16</u>

For now we do not need the vertical line that makes the back face, so we will erase it. Select "**MODIFY**" - "**ERASE**" from the menus. Next select "**SINGLE**" from the entity selection menu. Now select the line as shown in figure 4-83.

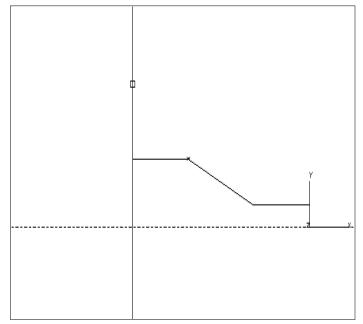
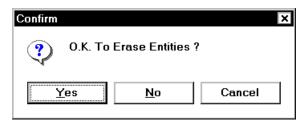


Figure 4-83 Selecting the line to erase

Finally press the right mouse button select <Escape>, then select "Done/Escape" from the selection menu, and answer the confirm prompt:



<u>STEP 17</u>

To make the display more readable, we will ZOOM in on just the area we will be working on. Select **"DISPLAY" - "ZOOM"** from the menus. With this command, you are asked to select two locations that make up the extents of what you want to see on the screen. Pick the locations in a manner similar to the ones in figure 4-84. The results of the zoom are in figure 4-85.

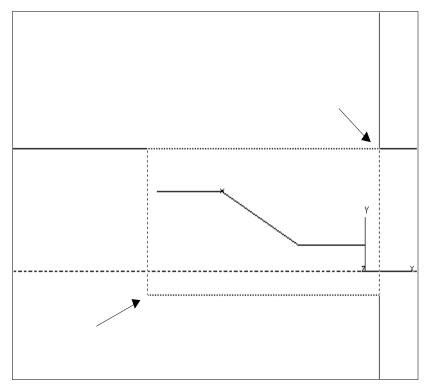


Figure 4-84 The zoom window

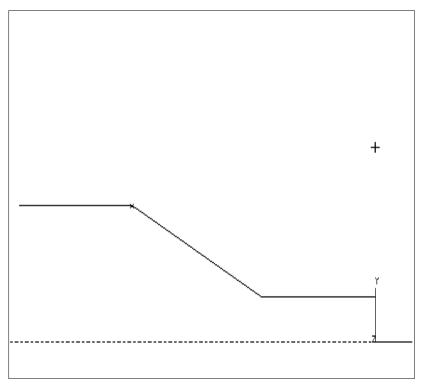


Figure 4-85 After the zoom command

<u>STEP 18</u>

Next we will add the 1/8 chamfer the 1 inch diameter. Select **"GEOMETRY" - "LINE" - "CHAMFER"** from the menus. Start by selecting the lines in the order shown in figure 4-86.

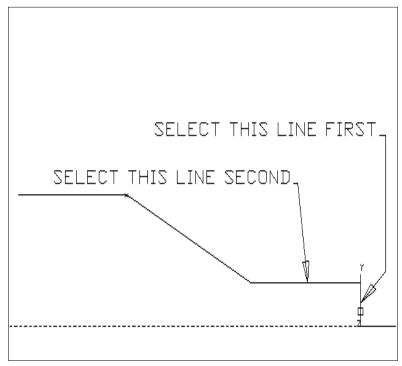


Figure 4-86 Selecting the lines to chamfer

Next you will be asked :

Chamfer Distance [0.25000] ? <u>.125<Enter></u> Chamfer Angle [45.00000] ? <u><Enter></u>

The part should now look like figure 4-87.

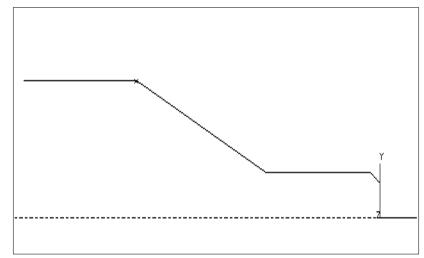


Figure 4-87 The completed chamfer

<u>STEP 19</u>

Now we will add the .25 fillet between the 1 inch diameter, and the 30 degree line. Select "GEOMETRY" - "ARC" - "FILLET" from the menus.

Select the lines to fillet in the order shown in figure 4-88. Note: When filleting, the order in which you pick the lines / arcs is critical (it must always by counter-clockwise). Filleting two lines is an exception, the system will automatically place them in the correct order. If you do however get bad results, you can always use the "Undo" command (From the Tool bar or Edit Menu) to reverse the changes.

Next you will be asked:

Enter the Radius [0.25000] ? .25<Enter>

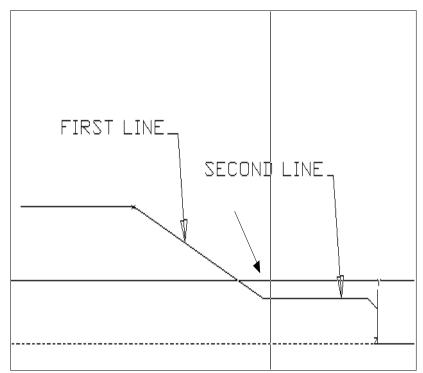


Figure 4-88 Selecting the lines to fillet and indicating the approximate point

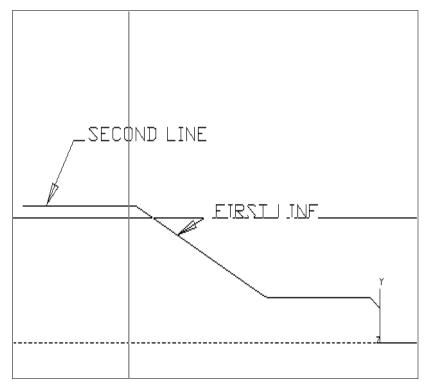


Figure 4-89 Selecting the lines to fillet and indicating the approximate point

<u>STEP 20</u>

Press the <Ins> key to repeat the FILLET command, then select the lines to be filleted as shown in figure 4-89.

Enter the Radius [0.25000] ? <u>.25<Enter></u>

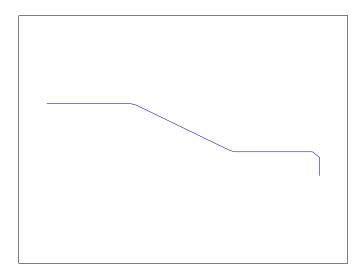
<u>STEP 21</u>

To be safe, lets save the file as we did in step 7.

<u>STEP 21 A</u>

The following is an alternate method for building the exact same geometry, using the "Easy Geometry" feature. (This following procedure can be done IN PLACE of steps 4-21 as previously outlined).

- a) Select "Geometry-Easy-Geometry-Lathe Geometry" from the menu.
- b) When asked to select the "Start point for easy geometry", select "Origin (X/Y) Zero" from the menu.
- c) From the Easy Geometry menu, select "Diameter" and enter 1 for the diameter
- d) Select "Chamfer" from the menu and enter the size of .125
- e) Select "Shoulder" from the menu, and enter in -1.518 (Yes we cheated to get this number which is not on the print). To find this number, enter in the construction point and the angled line as shown in step #6, Enter in the 1" diameter, then use the "Geometry-Inquire-Identify Point" with the "Intersection" option to find the coordinates to this point.
- f) Select "Radius" and enter a radius of .25
- g) Select "Taper" and enter an angle of 150 degrees
- h) Select "Diameter" and enter 3.0
- i) Select "Radius" and enter .25 (Should already be there as the default)
- j) Select "Shoulder" and enter -4.75
- k) Select "Done/<Escape>"



<u>STEP 22</u>

MACHINING

POWERSTATION uses a method of operation modeled after the way a machinist thinks. We call this method "OPERATION ORIENTED". Note: The remainder of this section does not apply to POWER-CAD users. The first step is to define one more operations that will take place. The second step is to select the operation that you want to work on, and finally do the machining for that operation.

Note that the operations must be specified in the order that you want the part machined in, but they do not have to be selected & worked on in any specific order. POWERSTATION will automatically generate the operations in the originally defined sequence no matter what order the operations were selected in.

If you make a mistake while doing any machining operations, first try pressing the <F7> (DELETE-LAST) key. If this does not produce the desired results, simply select the "Machining", "Tool Path|Edit", "Delete-Operation" command to delete all tool motion from the current operation, re-select the operation and try again.

Select "**Machining**", "**Operations**" from the menu (or press the operations ICON on the icon bar (located on the left side of the screen). Not sure which is the "operations" icon ? Simply hold the mouse cursor over the icon in about two seconds, a "Hint" will be displayed describing the function of the icon.

<mark>差</mark> Machining Op	perations				_ 🗆 X
<u>Operation</u>			 		
👲 🙀 🕒	/dla 🔰				
Currer Op #	Post Color Desc	ription		-	
			 		Material Operation L
			 		加速也会定立地上 Operation List
	*** * ****		 		De 1
	· - - - - - - - - - - - - -		 		Tooling Details

Figure 4-89a (The Machining Operations Manager)

To enter an operation:

- 1. Click the cursor in the "Op #" column, and type in the number of the operation.
- 2. Double click the mouse on the "Color" column, and select the desired color.
- 3. Click the cursor on the "Description" column, and enter a description of the operation
- 4. Double click on the first "Current" column, and the word "Yes" will appear. This indicates that this is now the currently selected operation.
- 5. At this point the dialog should look like figure 4-89b
- 6. Click on the tab that reads "Details". This will switch to the "details" page where you enter the specifics on the currently selected operation (tool number, diameter, etc..). See figure 4-89c
- 7. When entering the details for each of the operations, press the [Home] button and enter a home position of 6 and 6.

Machining Operations		×
Operation # Operation Type Tool Number Offset Number	ope	
Fixture Offset Tool Radius Stock Allowance Lathe Comp Mode 0 • 0.03125 0.03000 OD Turning •	Operation]	加速在在工业员
Feed Coolant : ○ IPM ⊙ IPR ○ RPM ⊙ CSS 500.00000	List	T
0.00800 Forward Range #2 Set Home Description:		
Roughing	Details	Tooling
	.,	ē

Figure 4-89b (Basic operation information)

Figure 4-89c (Operation #10 Details) Figure 4-89d (All operations for the tutorial part)

Lathe Tutorial 79

Machin		eratio	ons		_ 🗆	×
)peratio	n		7 1 0			
		1				
Currer	Op #	Post	Color	Description		
Yes	10	X		Roughing	ę	L
	20	X		Finish	erati	1966.6
	30	X		.06 Wide Grooving	Operation List	加度均差加工具工
	40	X		Threading	ist	ľ
		¦ ! !	 ! !			
	 ! !	 	1 		Г	ŕ
	+ + 1	 	+ 1 1			
	+	 - 	+		le le	١. E
					Detai	SULTOOI.
	 - 	 - - 	 - 		μ	a di
	 	 - 	 			

Machining Operations Operation	_ 🗆 🗙
Operation # Operation Type Tool Number Of 20 Turning V 2 2 2	fset Number
Fixture Offset Tool Radius Stock Allowance Late 0 • 0.03125 0.00000 OD	Turning V
O IPM ⊙ IPR O RPM ⊙ CSS 650.00000 [foolant :
	Set <u>H</u> ome
Description: L Finish	

Figure 4-89e (Operation #20 for the tutorial part)

Next define the remaining operations, using the information displayed in

<mark>A</mark> Machining Opera	tions	
🗕 🛱	/di 🔟	
Operation # 30 Fixture Offset Feed O IPM O IPR 0.00500 Description:	Operation Type Tool Number Offset Number Turning	Operation List
.06 Wide Groom		<u>D</u> etails

figures 4-89 c,e,f & g.

Figure 4-89f (Operation #30 for the tutorial part)

Machining Operations		×
Operation # Operation Type Tool Number Offset Number	Ope	
Fixture Offset Tool Radius Stock Allowance Lathe Comp Mode 0 • 0.00000 0.00000	Operation]	開始たまで主要1
Feed Coolant : ○ IPM ⊙ IPR ○ RPM ○ CSS 625.00000 Flood ▼	List	1
0.01000 Forward Range #1 Set Home		П
Description:		4
	<u>D</u> etails	Tooling

Figure 4-89g (Operation #40 for the tutorial part)

<u>STEP 23</u>

At this time it is a good idea to make a print out of the defined operations. To do this select "Operation", "Print" from the menu. Next we need to make operation #10 active. To do this, double click on the "Current" column for operation #10, then exit from the Machining Operations Manager.

<u>STEP 24</u>

It is a good practice to place your machining on a different layer than the part geometry. While we are dimensioning the part we really do not need to see the tool path, so placing it on its own layer gives us an easy way to "hide" the tool path.

To change the current layer to 100, press the $\langle F9 \rangle$ key (or select "Display", "Layer-Control" from the menu), then click on "Current Layer" box and type: <u>100</u>, click the cursor on the description box (The right most column in the layer 100 row), and enter in a description like "Machining". Next press the [OK] button.

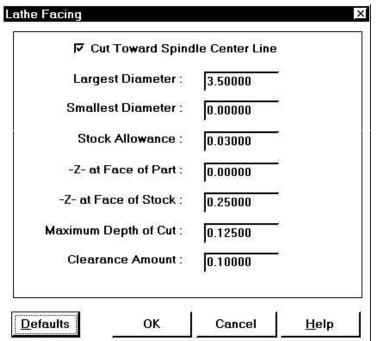
Num	ON	Color	Used	Style	Description 🔺
98	X			Solid	All <u>O</u> n
99	X			Solid	
100	X			Solid	Machining All Off
101	X			Solid	
102	X			Solid	Show Used
103	X			Solid	
104	X			Solid	- T
105	X			Solid	Current Layer-
106	X			Solid	
107	X			Solid	100 🜩
108	X			Solid	
109	X			Solid	Line Style
110	X			Solid	· · · · · · · · · · · · · · · · · · ·
				<u> </u>	· Ŧ 1 🔟 L

<u>STEP 25</u>

This is a good time to save our work as in step 7.

<u>STEP 26</u>

With our roughing tool, we will now face off, then rough turn the part. First the facing. Select "**MOVE & CUT**" - "Lathe Face" from the menus. Lets assume a rough bar stock diameter of 3.5 and additional rough stock on the face of 0.25. You will be asked:



<u>STEP 27</u>

There is no step 27. (We thought that you needed a break)

<u>STEP 28</u>

Now let's start the roughing cycle. Select "Machining", "Lathe Roughing/Undercutting" from the menus. The single most important rule is *"IF THE TOOL IS NOT GOING TO TOUCH IT, THEN DO NOT SELECT IT*" (In fact don't even define it). The system will ask:

Mode	Tool Orientation	Roughing Direction
Roughing	C External (O.D.)	e Turning
C Undercutting	C Internal (I.D.)	C Facing
Depth of Cut P	er Side : 0.125	Cutting From Right to Left
Stock Allowand	ce in -X- : 0.03000	- ▼ Assume Part Origin at X0/2(
Stock Allowanc	e in -Z- : 0.01000	- Г Select Stock Boundary
Clearance /	Amount : 0.10000	-
Bar Stock D	ameter : 3.25000	-
Stock on B	ar Face : 0.05000	-

Select the starting line or arc of the boundary to rough (See figure 4-93.

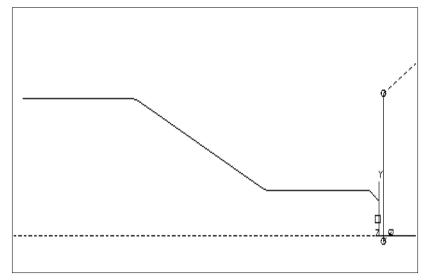
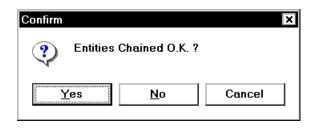


Figure 4-93 Selecting the starting point for roughing

Select the line or arc to stop before, <Esc> for none

Press the right mouse button, and select <Escape> from the menu.



Note: You must select at least two entities (lines or arcs)

At this point the display looks like figure 4-94, and you will be asked to indicate the side of the boundary to offset to. Select a position just to the right of the front face and the center line.

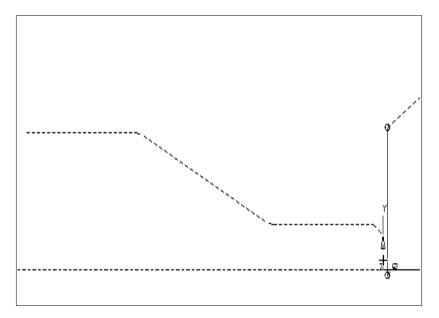


Figure 4-94 Proper chaining

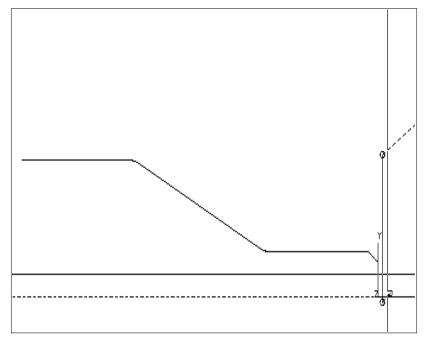


Figure 4-95 Picking the side to offset to

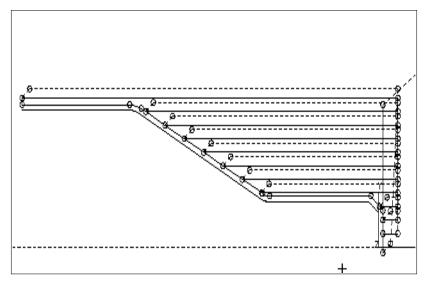


Figure 4-96 Roughing completed

<u>STEP 29</u>

Next we will bring the tool back home. Select "Machining", "MOVE & CUT" - "GO HOME" from the menus.

[X] Rapid Move

<u>STEP 30</u>

Select operation 20 from the "Machining", "Operations" Dialog.

<u>STEP 31</u>

On many slower computers, the display of the tool images (small circles) can be very time consuming. If you want to turn them off, select "**DISPLAY**" - "Tool Image" from the menus.

ool path Display Show Tool Path Only	
) Show Tool Image) Show Machined Path) Show Full Animated Too	l Image
🗹 Display Tool Path	

<u>STEP 32</u>

Finish cutting the O.D.. Select "Machining", "MOVE & CUT" - "CHAIN" from the menus.

Select the starting Line or Arc (See figure 4-97). Select the line or arc to stop before, <Esc> for none (Press the right mouse button, and select <Escape> from the popup menu)

Confirm		×
? Entities	Chained O.K. ?	
<u>Y</u> es	No	Cancel

Pick the Side to Offset to

(Move the cursor just to the right of the front face)

Lathe Tutorial 87

-X/Z- Cutting Feed : 0.00500 -X- Clearance : 0.10000 -Z- Clearance : 0.10000 Options I Rapid To Start Position G41/42 Cutter Compensation Cutter Compensation Cutter Right Automatic Compensation Number : 1	General Type : Turning	Multiple Stock Approach Depart Multiple -Z- Depths Cutter Comp
	-X- Clearance : 0.10000 -Z- Clearance : 0.10000 Options Rapid To Start Position G41/42 Cutter Compensation	 Cutter Left Cutter Right Automatic
		✓ Turn Off Compensation at End

See figure 4-98

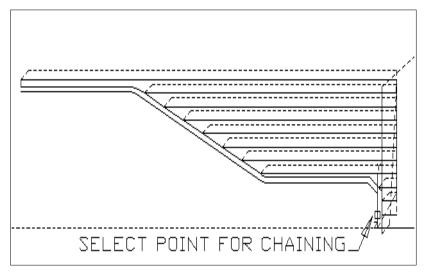


Figure 4-97 Selecting the start line for chaining

<u>STEP 33</u>

Next we will bring the tool back home. Select "Machining", "MOVE & CUT" - "GO HOME" from the menus.

[X] Rapid Move ?

<u>STEP 34</u>

Select operation 30 from the "Machining", "Operations" Dialog.

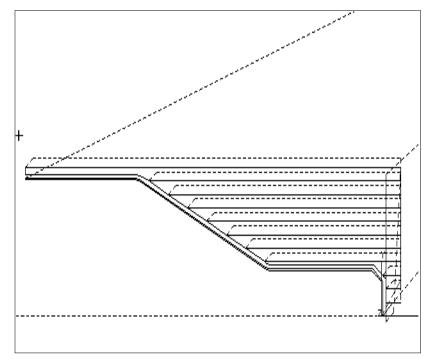


Figure 4-98 The Finish pass complete

<u>STEP 35</u>

🗖 Grooving On a Face	☑ Rough Right to Left □ Internal Groove
Right Side of Groove in -Z- :	-1.18480
Left Side of Groove in -Z- :	-1.27750
Major Diameter	1.00000
Groove Depth :	0.08000
Groove Corner radius :	0.00200
Tool Width :	0.06000
Tool Corner Radius :	0.00500
Clearance Amount :	0.10000
Dwell Time at Groove Bottom :	0.00000

Now we will generate the groove. Select "Machining", "Grooving" from the menus. The dialog will be displayed:

Note: From the way in which the groove is dimensioned, it is probably easiest to enter the -Z- sides of the groove in the following manner.

Right Side of Groove in -Z- : <u>-4.75+3.5625</u> Left Side of Groove in -Z- : <u>-4.75+3.5625-.09</u>

These expressions can always be replaced by the exact decimal number if you know it. If you do not know it, why not let the computer do the math for you ?

<u>STEP 36</u>

Next we will bring the tool back home. Select "MOVE & CUT" - "GO HOME" from the menus.

[X] Rapid Move ?

<u>STEP 37</u>

Select operation 40 from the "Machining", "Operations" Dialog.

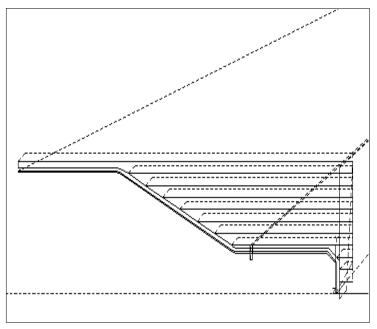


Figure 4-99 The grooving complete

<u>STEP 38</u>

Thread Pitch	Thread Diameter
Threads Per Inch : 12.00000	Diameter 1.00000
Thread Pitch : 0.08333	O Select a Line (Later) (Also for Tapered Threads)
Depth - Infeed / Outfeed	🗖 Internal Thread
(Depths are Per Side)	
Total Depth : 0.05112	-Z- Locations
First Pass Depth : 0.01400	Right Side (Start) : 0.00000
Clearance : 0.10000	Left Side (End) : -1.23250
In-Feed Angle : 30.00000	
Out-Feed Angle : 0.00000	Canned Cycle Options :
🗖 Pull Out, In-Threading Mode	0-Standard "Long Hand" G32/33 Type

Now we will add the thread. Select "Machining", "Lathe Threading" from the menus.

STEP 38 CONTINUED ...

Notes on threading: After entering in the threads per inch, press the "Calc 75% Depth" button. This will take the current thread pitch and calculate the proper depth for a 75% of a standard 60 degree thread. Also, when entering the "Left Side (End)", again try entering the location as an expression "-4.75+3.5625-.045" in order to thread into middle of the groove.

<u>STEP 39</u>

Next we will bring the tool back home. Select "MOVE & CUT" - "GO HOME" from the menus.

[X] Rapid Move ?

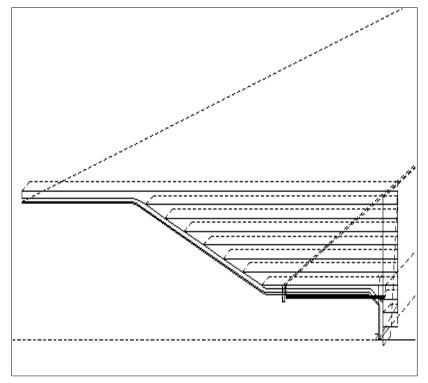


Figure 4-100 The Threading Complete

<u>STEP 40</u>

Once again, lets save the file as in step 7. After that, lets check the machining status. Select "Machining", "Machining Status" from the main menu. When done viewing the status, press the [OK] Button.

<u>STEP 41</u>

The final step in machining is to run the post processor. Select "Machining", "Post (Generate NC-Code)" from the menu (or select the Post icon).

From the list of machines in the upper left hand corner of the dialog box, move the highlight to "DLATHE.MCH".

Next we will set the name of the NC-tape file to be created. Press the button labeler [Set NC-Code File Name]. Enter "LESSON2" as the "File Name", and press [OK].

Press the [Go] button. The post processing now begins, and may take a few minutes. When done, press [Close] to leave the post processor.

Post Processor	×
Machine Name DLATHE.MCH DMILL.MCH SUBMILL.MCH	NC-Code File Name : lesson2.TAP <u>S</u> et NC-Code File Name
Error Messages:	Line Number : 262
NC-Code N0950 G00 X1.2 N0960 Z.1 N0970 T0 M05 N0980 M30 %	• •
	<u>C</u> lose <u>H</u> elp

Now that the tutorial part is finished, lets see how the new "Job Processing" feature can help with making changes.

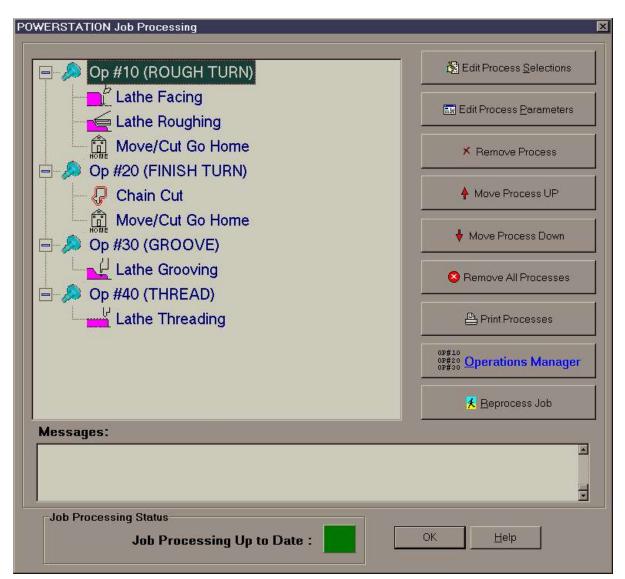
NOTE **!!!** The Following steps are NOT available/possible in the XPERT Version of POWERSTATION. <u>Step 42</u>

Select "Machining-Job Processing". The following dialog will be displayed:

POWERSTATION Job Processing	×
P 🔊 Op #10 (ROUGH TURN)	Edit Process Selections
 ⊕ - Ø Op #20 (FINISH TURN) ⊕ - Ø Op #30 (GROOVE) 	Edit Process Parameters
🔁 🔎 Op #40 (THREAD)	× Remove Process
	A Move Process UP
	Move Process Down
	S Remove All Processes
	Print Processes
	OP#10 OP#20 OP#30 Operations Manager
Managana	Reprocess Job
Messages:	
	<u>.</u>
Job Processing Status Job Processing Up to Date :	ОК <u>Н</u> еlp

<u>Step 43</u>

In the Job Processing Dialog, in the "Tree area" (Upper left) one at a time, click on the Plus "+" signs. The dialog will now look as follows:



<u>Step 44</u>

For this example, we will make some common changes and show how quickly we can regenerate the tool path and the NC-Code. For this example, lets assume the following changes:

- 1) The tool nose radius on the roughing tool was changed to 1/16
- 2) The a rough bar stock diameter turned out to be 3.75 not the expected 3.5
- 3) The additional rough stock on the face turned out to be .375 not the expected 0.25.
- 4) The depth of cut for the roughing need to be increased to 0.2

<u>Step 45</u>

In the Job Processing dialog, click on the [Operations Manager] button. Select operation #10, then click on the "Details" page. Change the details page to look as follows, then close the operations manager.

Machining Operations Manager			
Operation # Operation Typ 10 • Turning • Fixture Offse Tool Radius 1 • 0.06250 Feed Spindle / C IPM © IPR 0.00600 Forward Description: ROUGH TURN	0.03000 Speed CSS 500.00000	Offset Number 1 • Lathe Comp Mode OD Turning • Coolant : Flood • Set Home Set <u>G</u> age	Tooling Tool <u>Shape</u> Operation <u>List</u> Details Material

<u>Step 46</u>

On the Job Processing dialog, double click on the line under operation #10 that reads "Lathe Facing". Change the settings to the following to reflect the new bar stock size.

Cut Toward Spind	lle Center Line	
Largest Diameter :	3.75000	
Smallest Diameter :	0.00000	
Stock Allowance :	0.03000	
-Z- at Face of Part :	0.05000	
-Z- at Face of Stock :	0.37500	
Maximum Depth of Cut :	0.12500	
Clearance Amount :	0.10000	

<u>Step 47</u>

On the Job Processing dialog, double click on the line under operation #10 that reads "Lathe Roughing". Change the settings to the following to reflect the new bar stock size and the new depth of cut.

Mode	To	ol Orientation	Roughing Direction	
Roughing	External (0.D.)		Turning	
O Undercutting	O Internal (I.D.)		© Facing	
Depth of Cut P	er Side :	0.20000	Cutting From Right to Left	
Stock Allowanc	:e in -X- :	0.03000	Assume Part Origin at X0/Z0	
Stock Allowanc	e in -Z- :	0.01000	- Select Stock Boundary	
Clearance A	Amount :	0.10000		
Bar Stock Di	iameter :	3.75000		
Stock on B	ar Face :	0.05000	1	

<u>Step 48</u>

In the Job Processing Dialog, press the [Reprocess Job] button. In a few seconds the entire tool path will be regenerated.

<u>Step 49</u>

At this point, the new tool path is generated. To regenerate a new NC-Code file, simply repeat step #41 (Run the post processor).

Please see the MILL TUTORIAL for examples of drafting, dimensioning, and printing your sample part