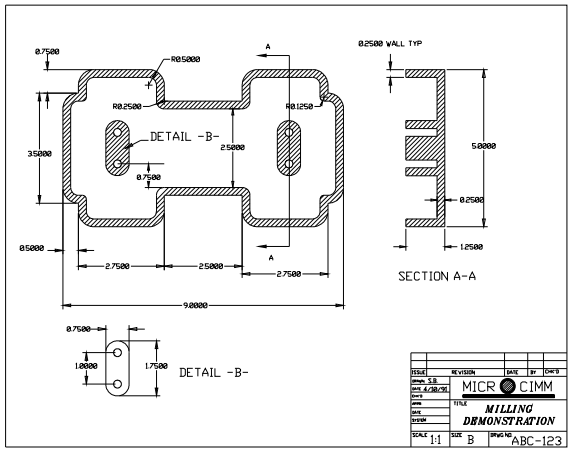
POWERSTATION 2002/XPERT

Tutorial Manual

Revision 6/10/2004



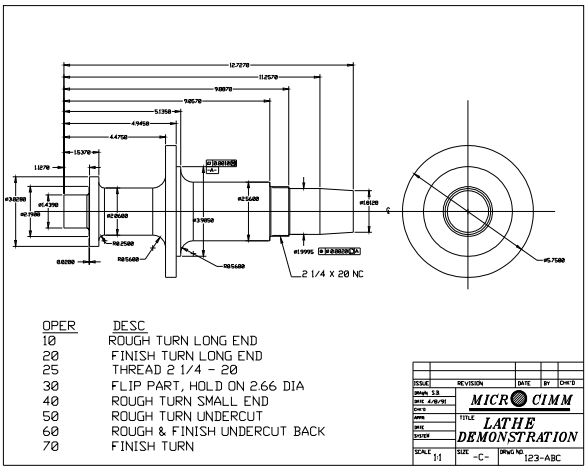


TABLE OF CONTENTS

MILL TUTORIAL

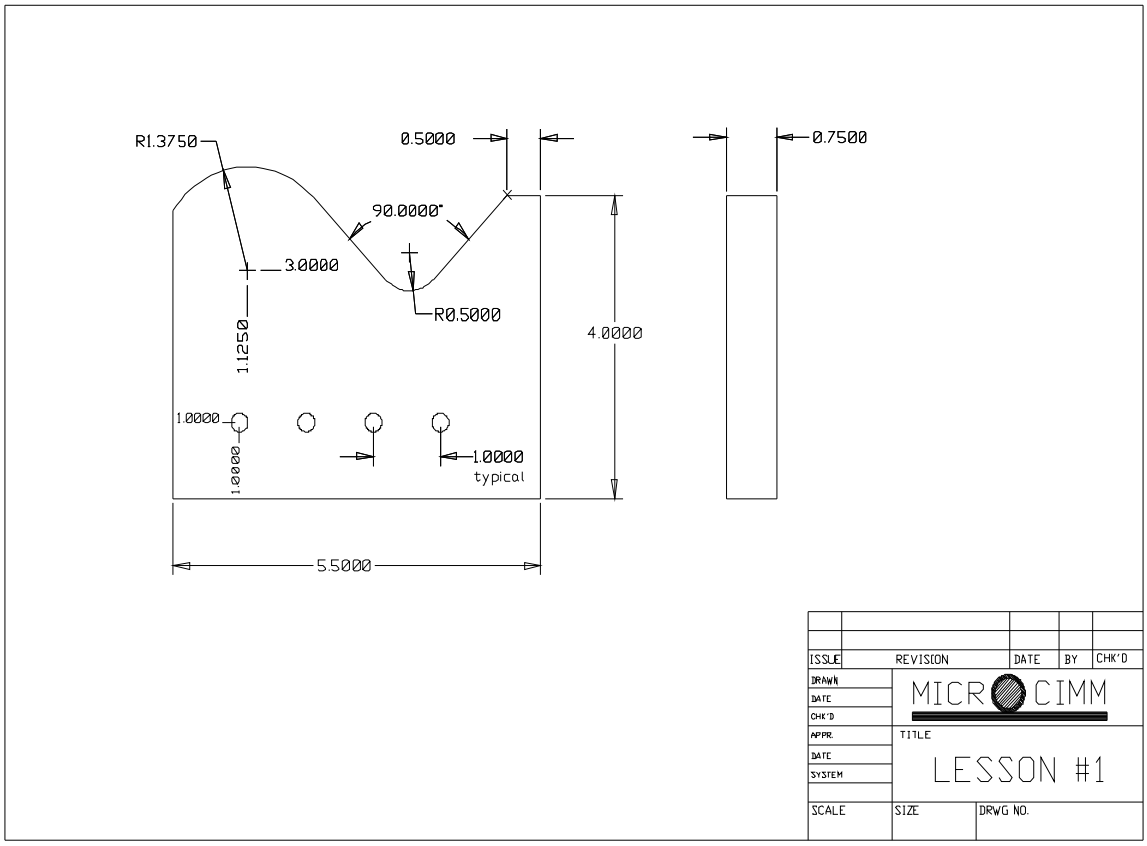


Figure 4-0 The sample drawing

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.

STEP 1

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

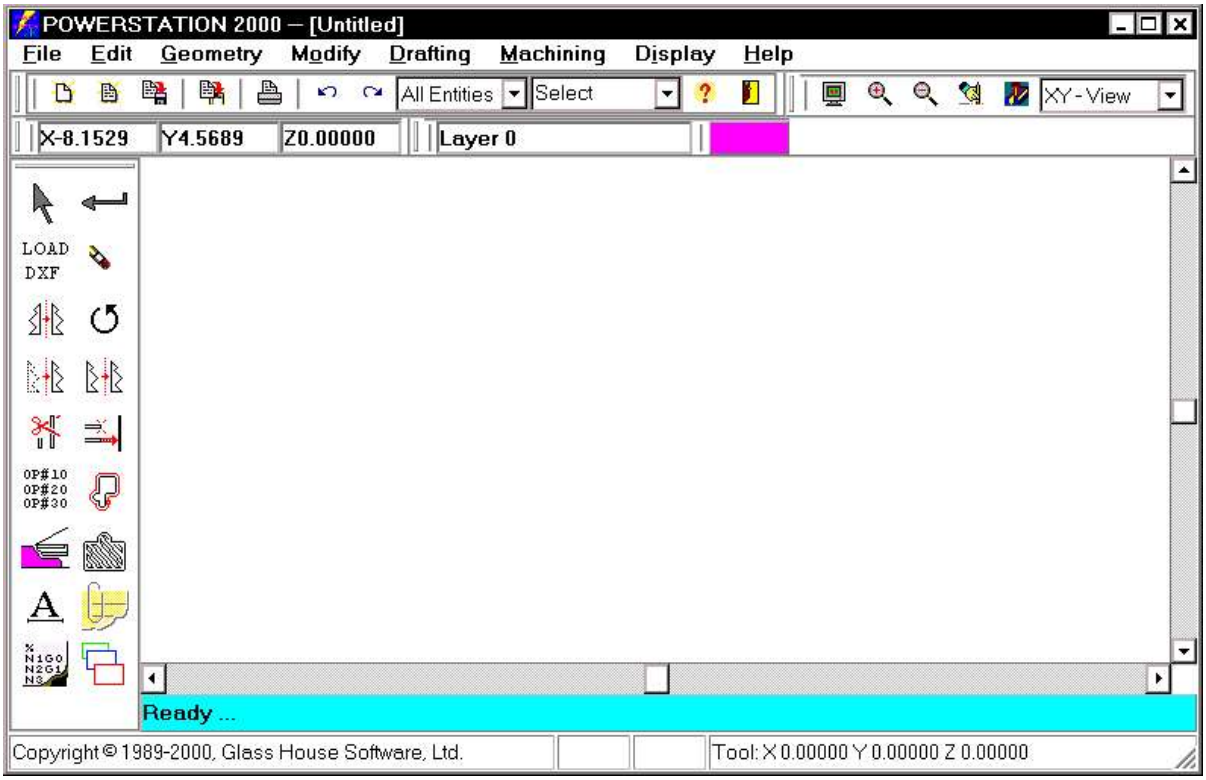
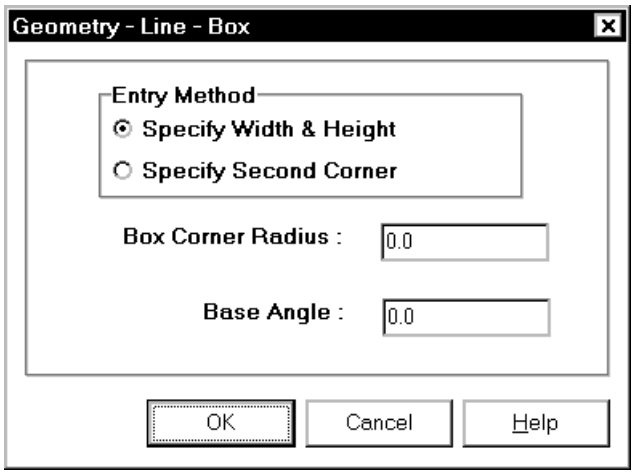


Figure 4-1 The opening screen

STEP 2

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.

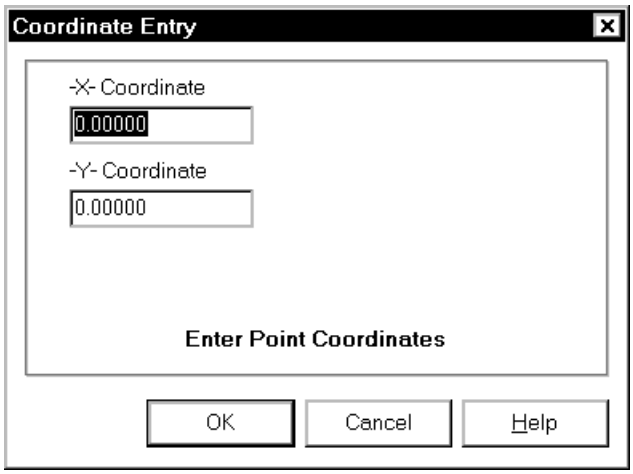
To create the box we select, from the main menu select "Geometry" ,"Line", "Box"



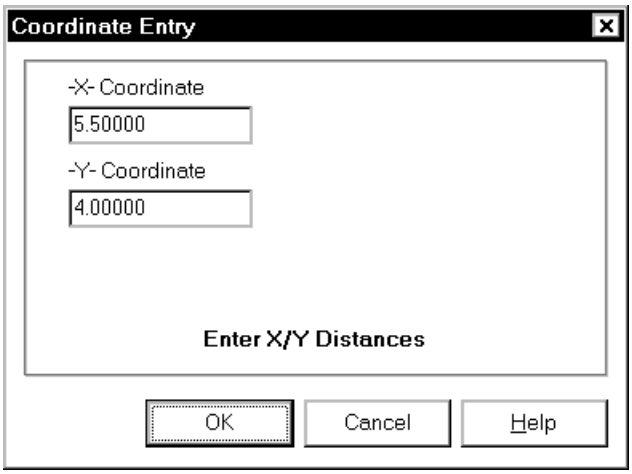
At this point the above dialog will be displayed. For this example we do not need to change any of the above defaults. Press [OK] with the mouse.

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.



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



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

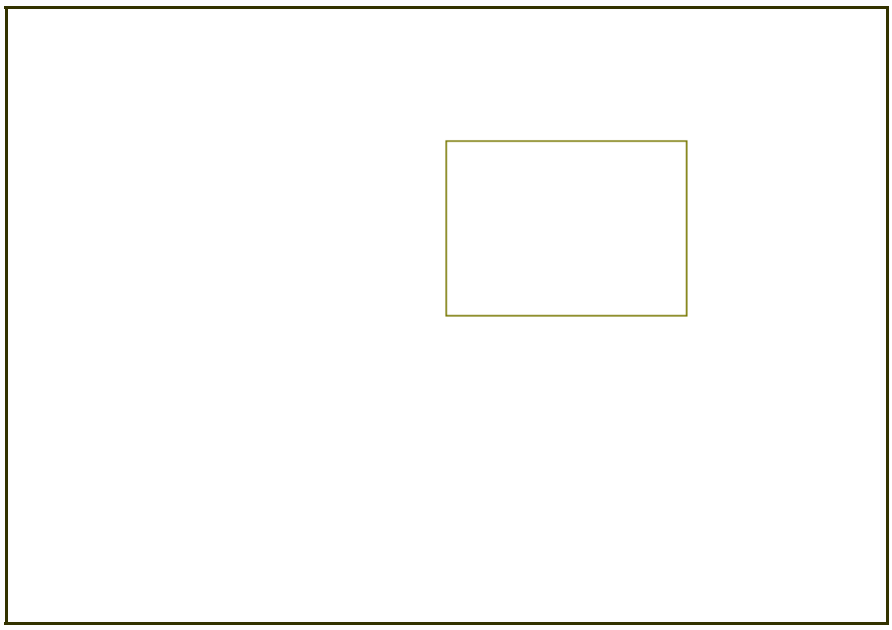


Figure 4-2 the basic part shape (box)

STEP 4

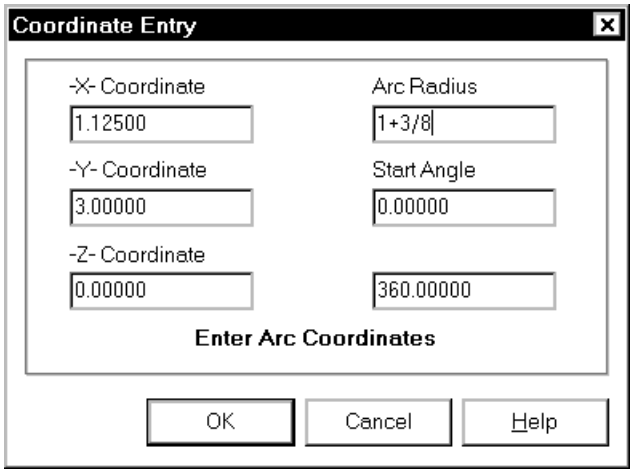
It will be easier to work if the box were more centered in the display screen. To do this we will use the **"Display-Zoom Extents"** menu command.

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.

STEP 5

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



The display should now look like Figure 4-4.

STEP 6

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.

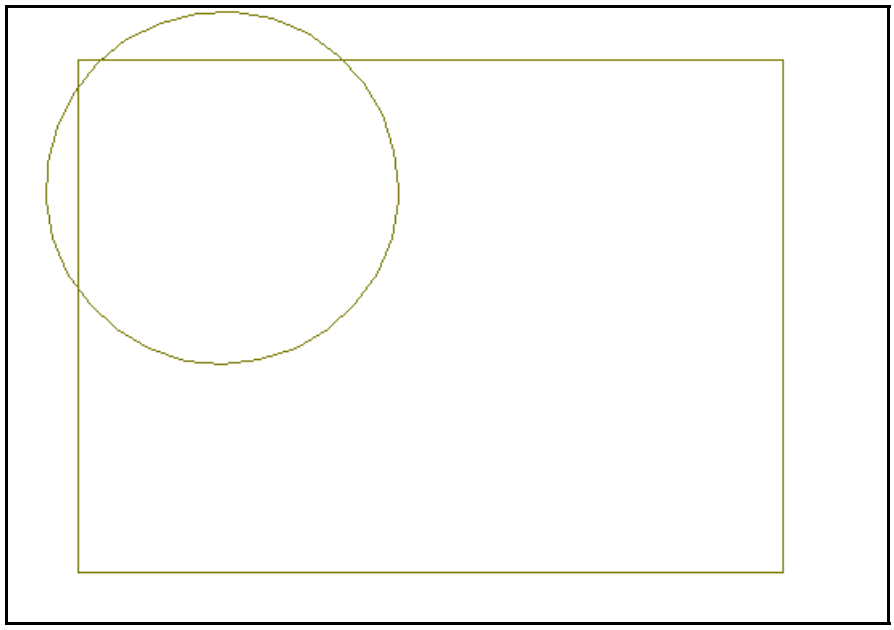


Figure 4-4 The first (1 3/8") arc

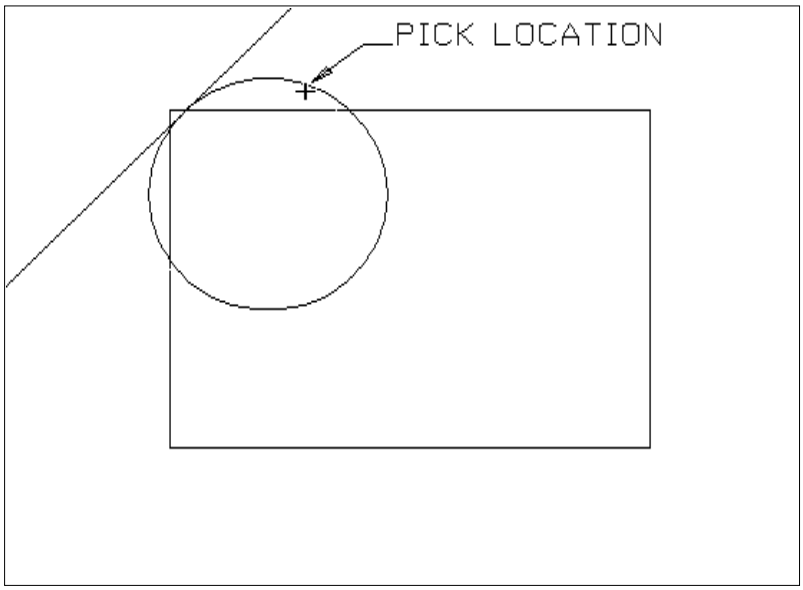
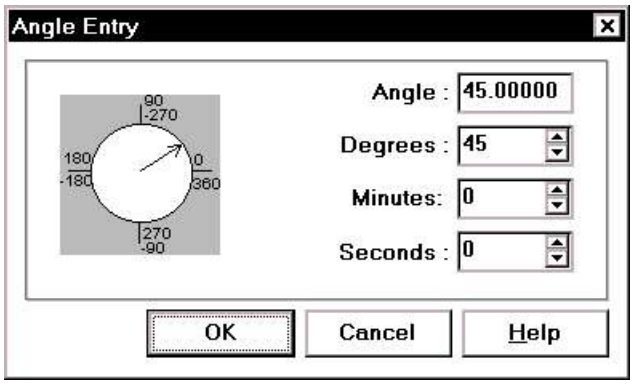


Figure 4-5 Creating the tangent line

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



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 ΔUndo and XRedo commands. Repeat the steps to create the line, but use **-45** for the angle.
See figure 4-6

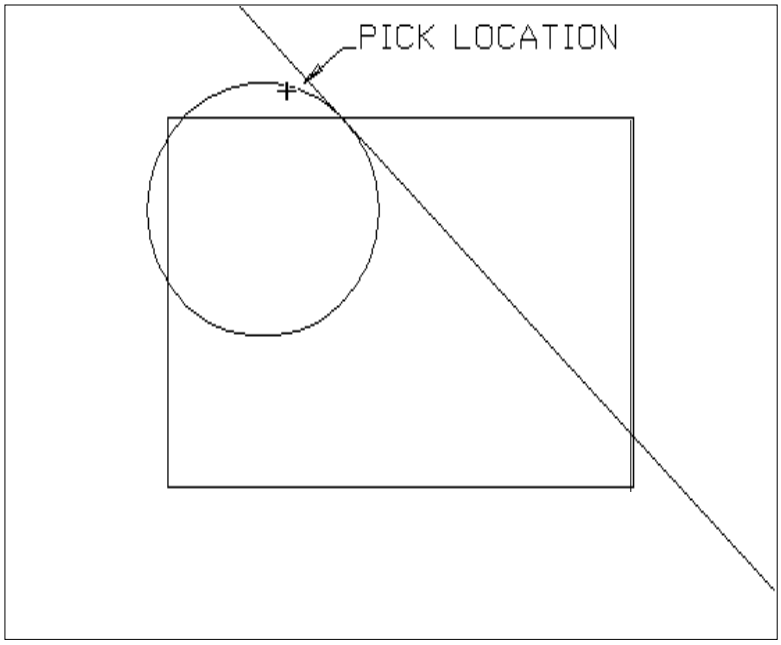
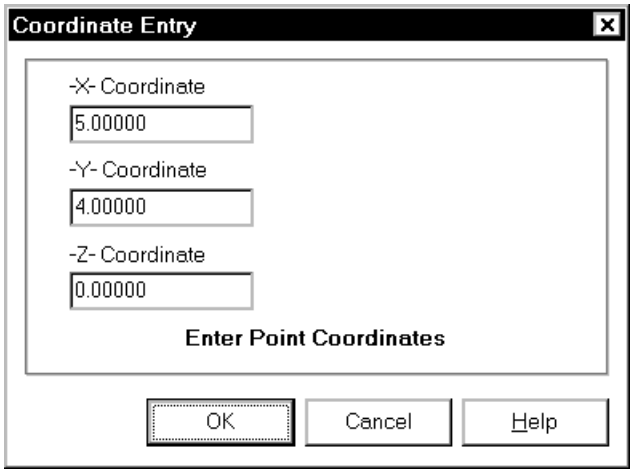


Figure 4-6 The corrected tangent line

STEP 7

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:



The screen will now look like figure 4-7

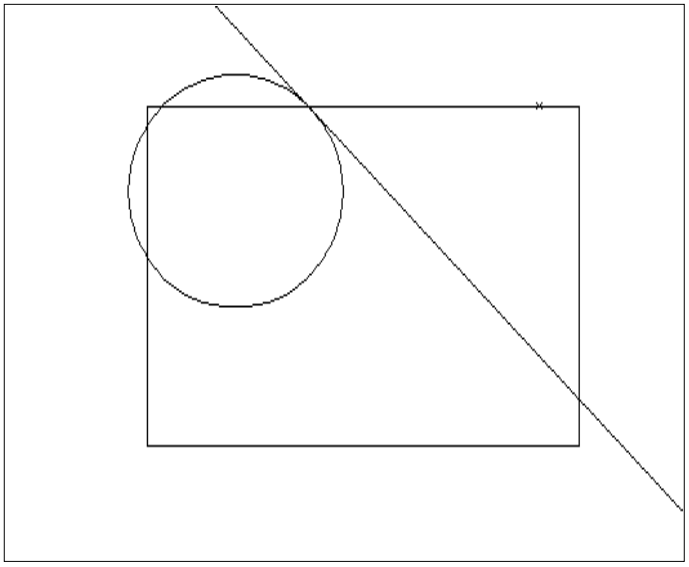
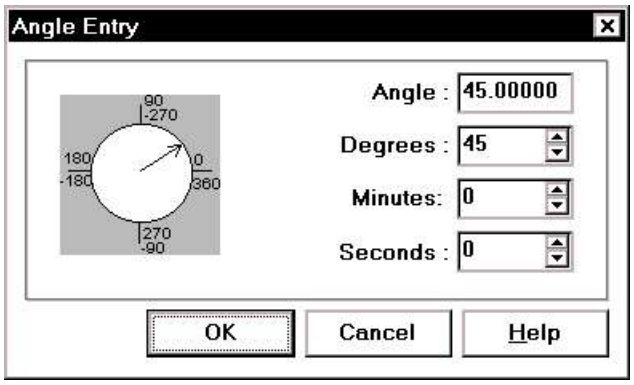


Figure 4-7 a construction point

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:



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.

STEP 8

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. *NOTE: In the following examples the terms "Cutting Edge" & "Bounding Edge" will be used interchangeably.*

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

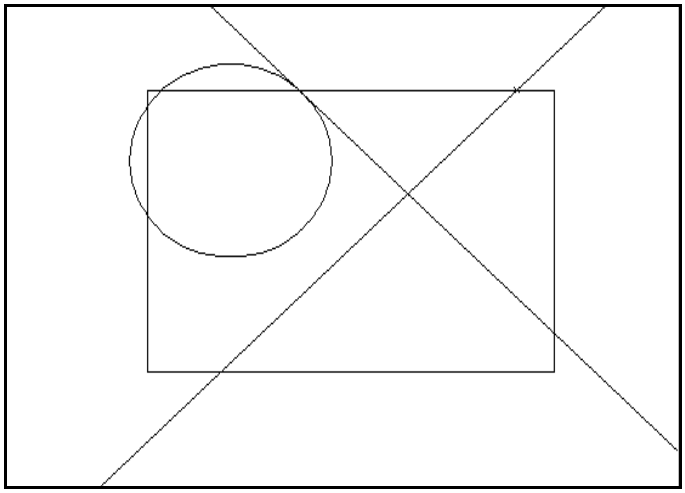


Figure 4-8 The second angled line

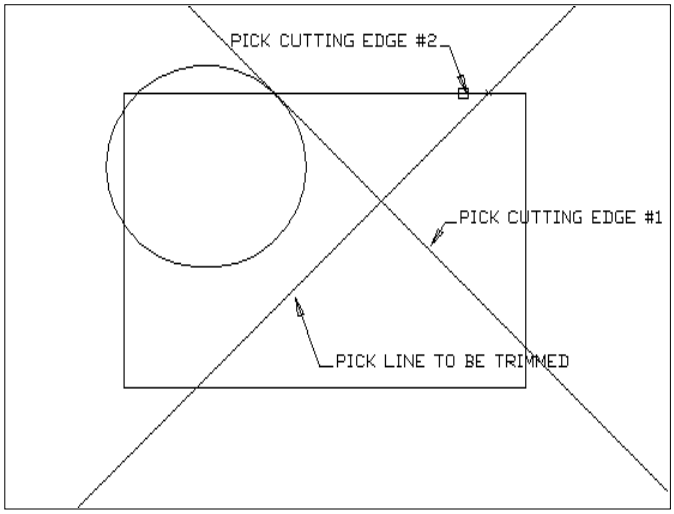


Figure 4-9 Trimming the last line

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.

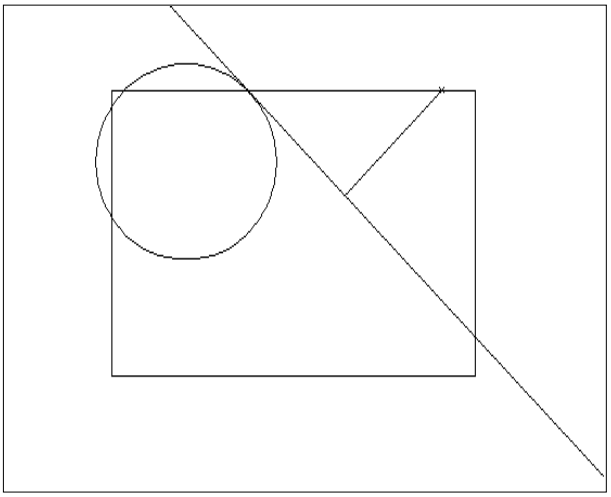


Figure 4-10 The trimmed line

STEP 9

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.

STEP 10

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.

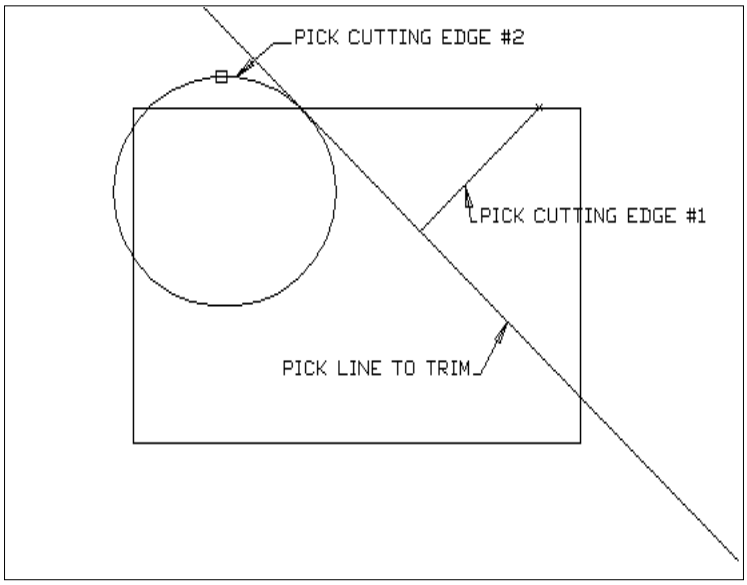


Figure 4-11 trimming lines

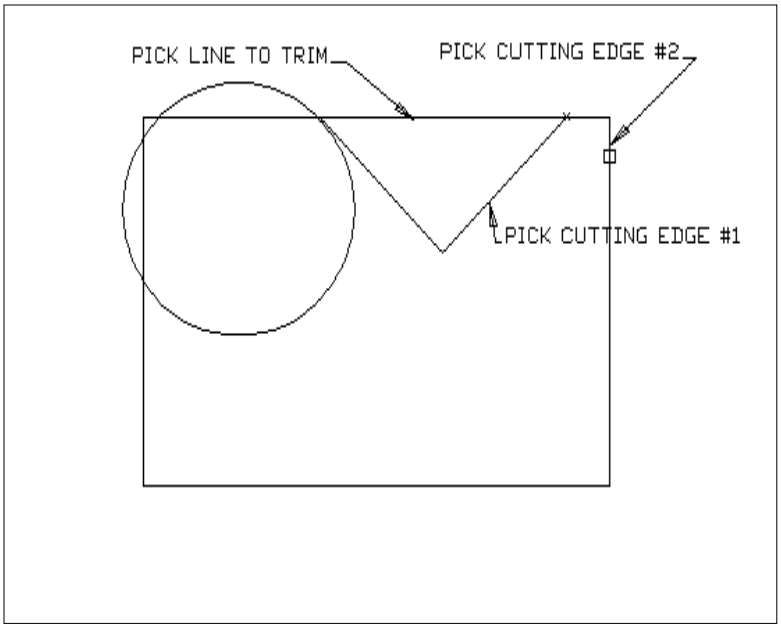


Figure 4-12 Trimming lines

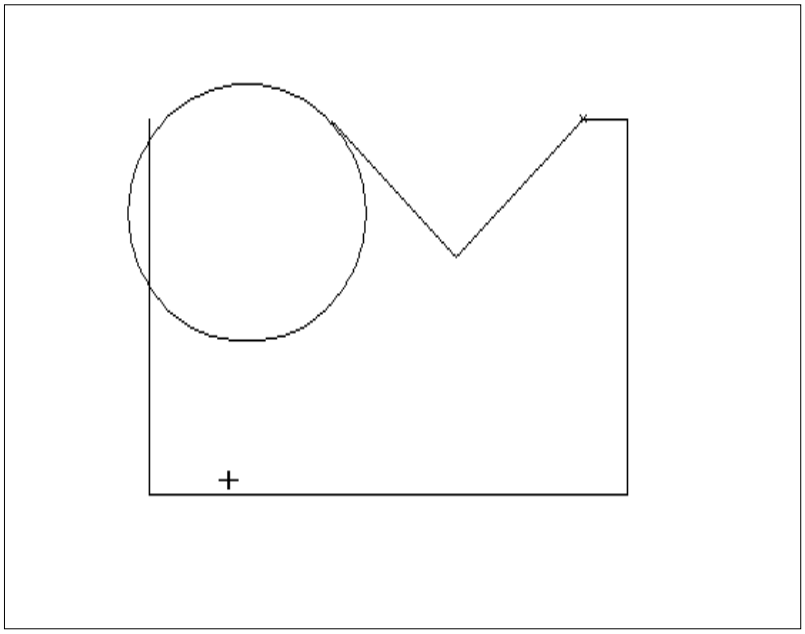


Figure 4-13 The trimmed lines

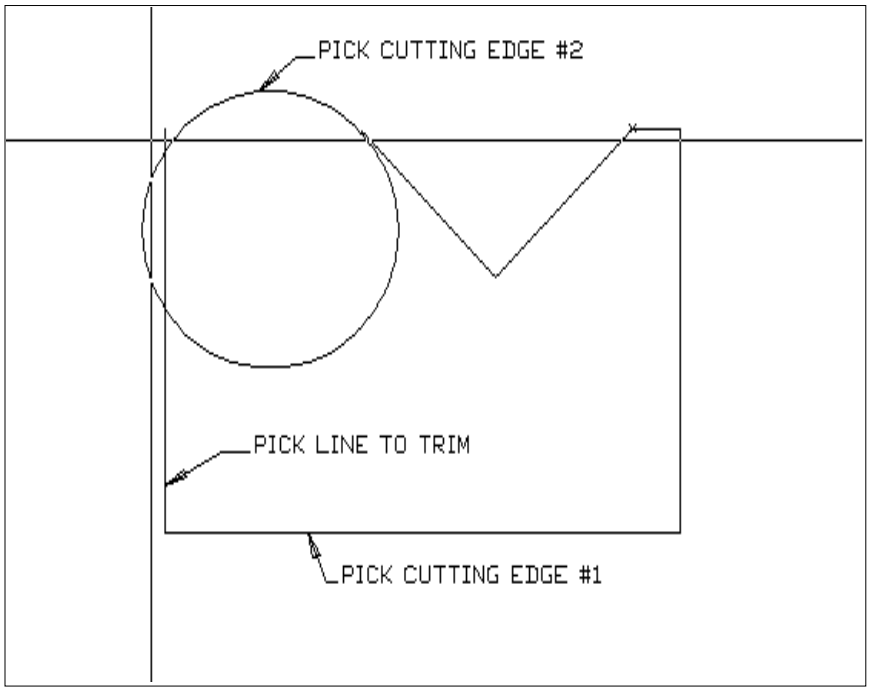


Figure 4-14 Trimming a line between a line and arc

STEP 11

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.

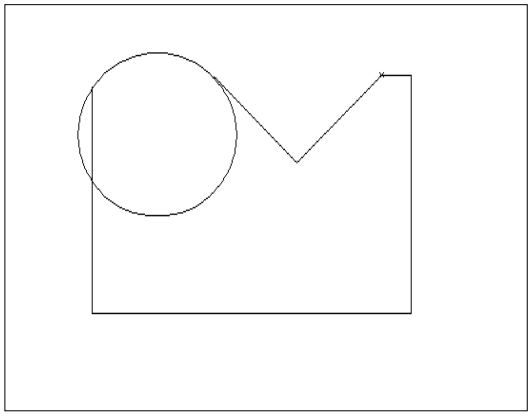
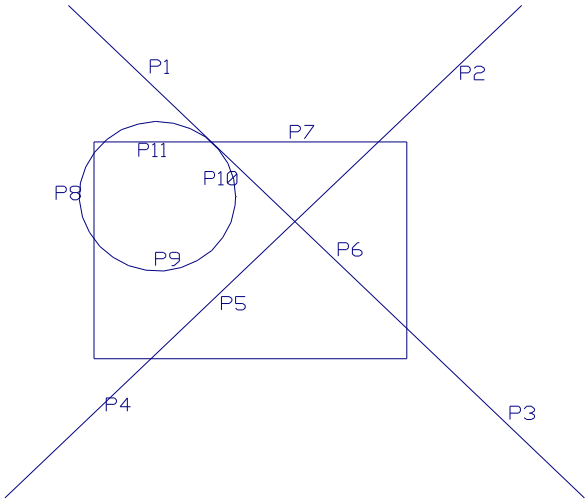


Figure 4-15 The trimmed line

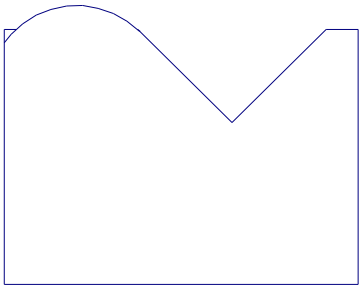
The display is really starting to look like the part we want see figure 4-17.

Notes on Trimming:

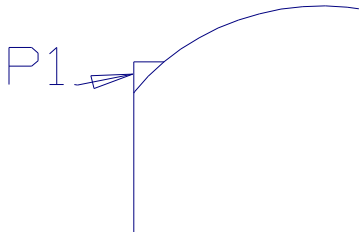
The Trim/Extend command was used for all of the above examples to demonstrate the versatility of this command. *It does not represent the easiest way to trim*. All of the above examples could have been done using the "EASY-TRIM" command. After selecting this command you would simply point (with the mouse) 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:



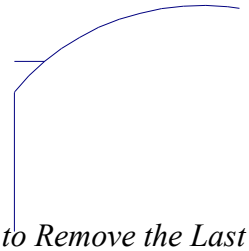
Easy Trim, Part #1



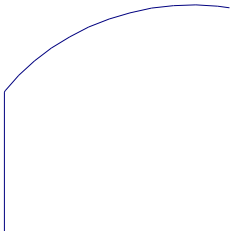
Results After Part #1



Removing the Small Section at Point #1



Use the Erase command to Remove the Last Small Segment



Final Cleaned up Corner

STEP 12

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] ? .5<Enter>
```

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)

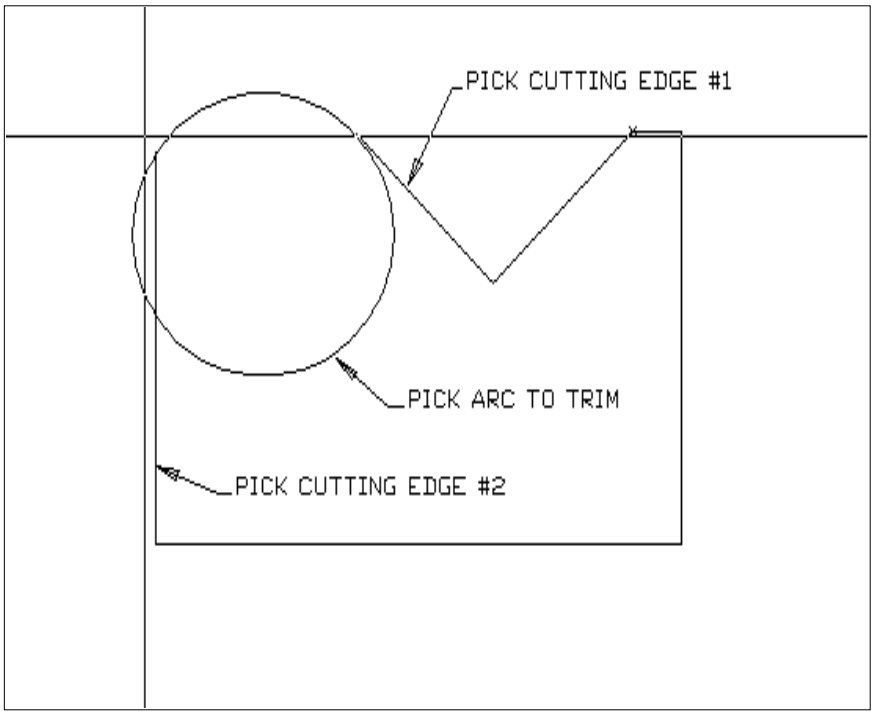


Figure 4-16 Trimming an arc

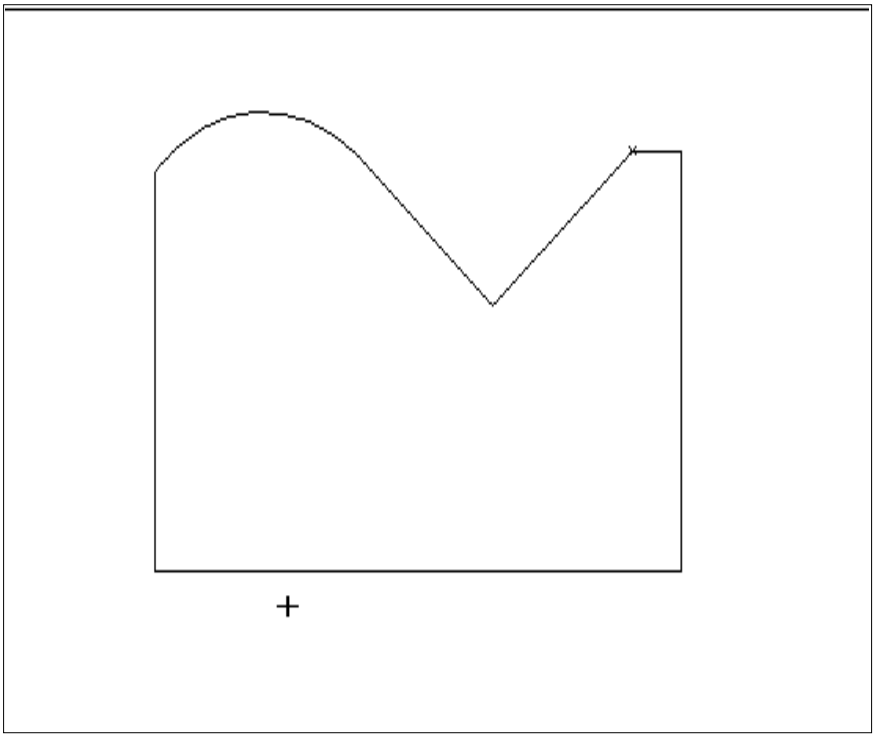


Figure 4-17 All geometry has been trimmed

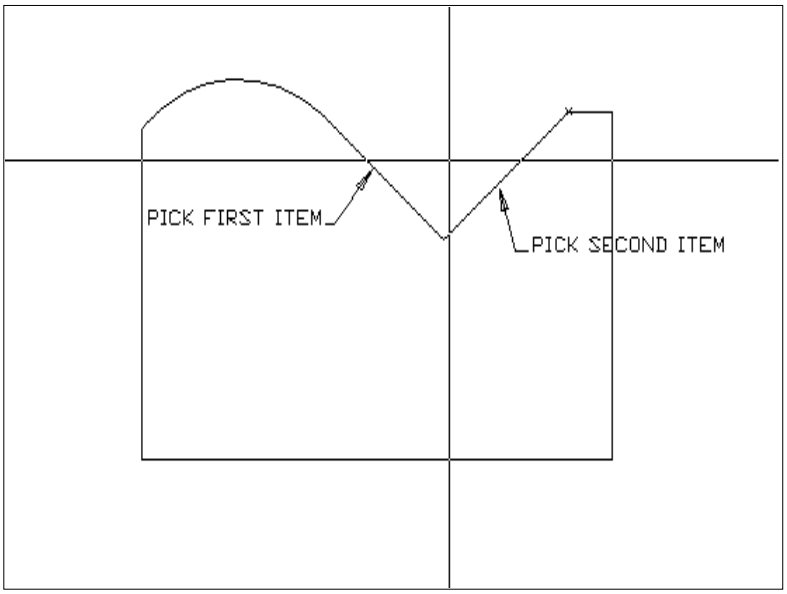


Figure 4-18 Filleting to lines

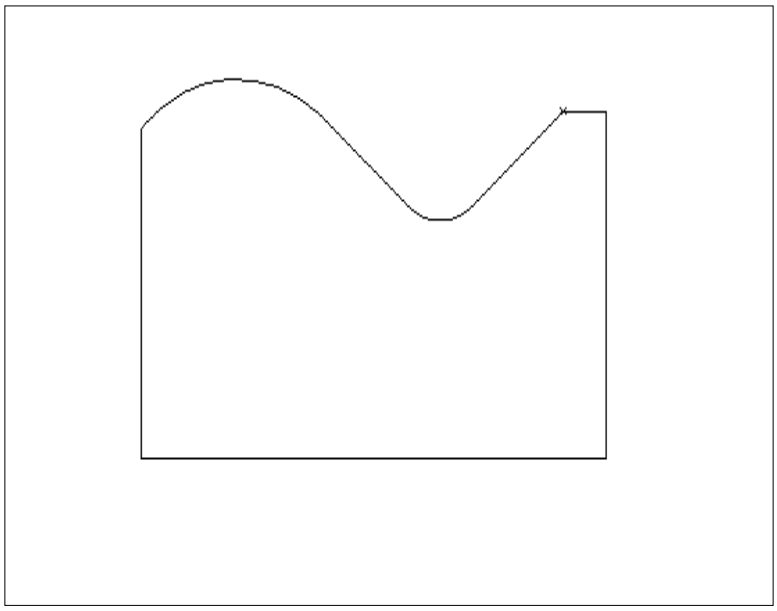
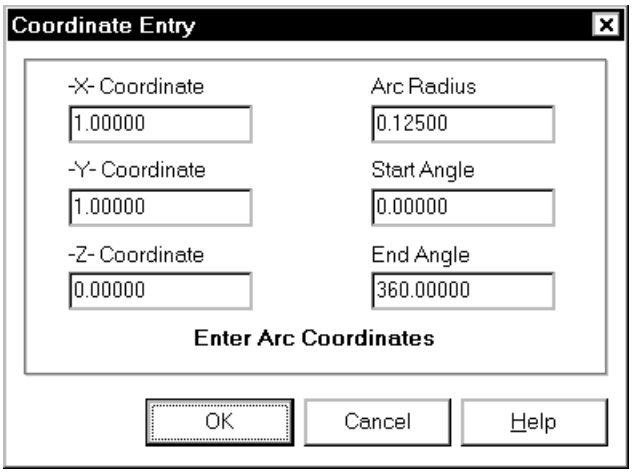


Figure 4-19 The part after filleting

STEP 13

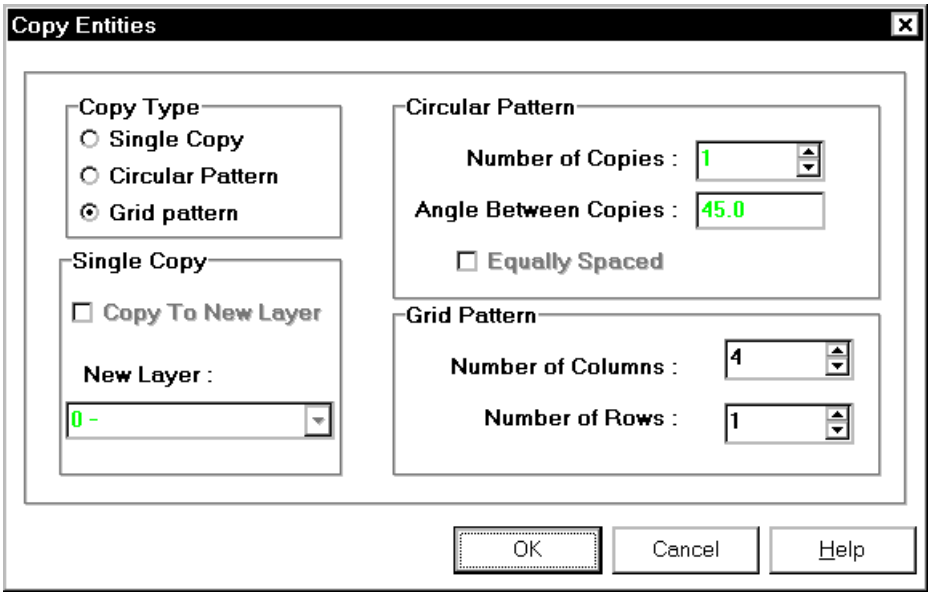
Now lets draw the drilled holes. Select “Geometry”, “Arc”, then “Coordinates” from the menu (see figure 4-20). You will be asked:



STEP 14

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".**

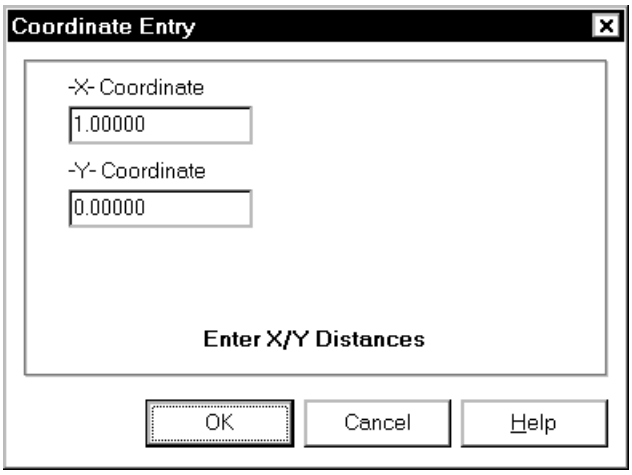
Next the “copy entities” dialog will be displayed. We want to generate a grid pattern of holes, with a single row, and 4 columns. So fill in the dialog as shown (You will notice that both the circular pattern, and single copy sections of the dialog are disabled once “Grid” is selected).



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:



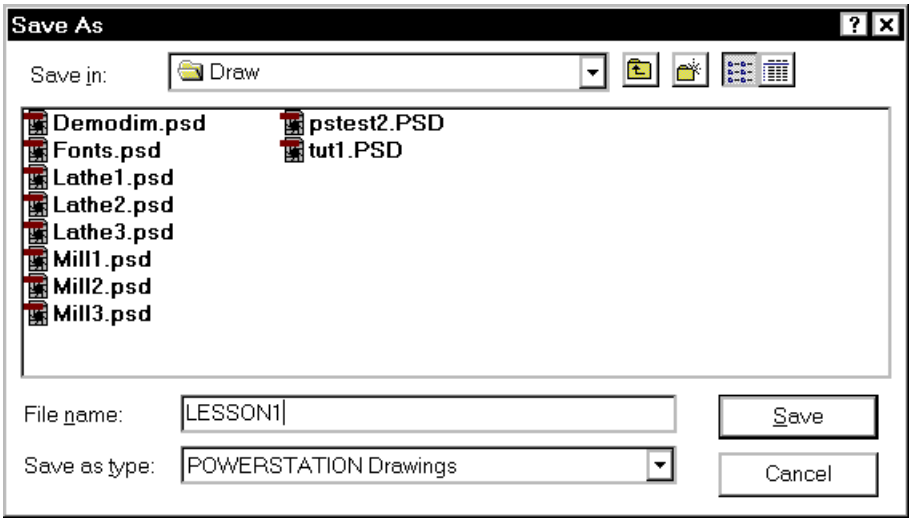
The screen will now look like figure 4-21

Note: This step is purely for drawing purposes, if all that you want to do is drill these holes, this step could be skipped.

STEP 15

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:



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.

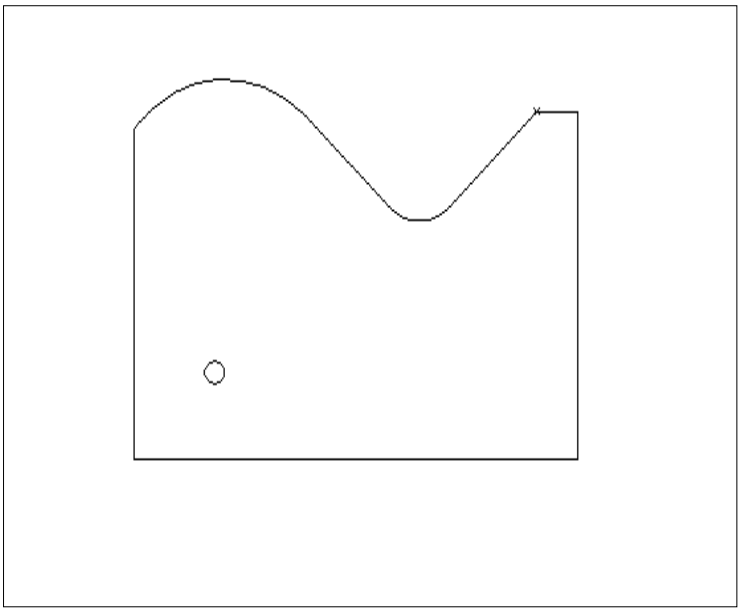


Figure 4-20 The first hole

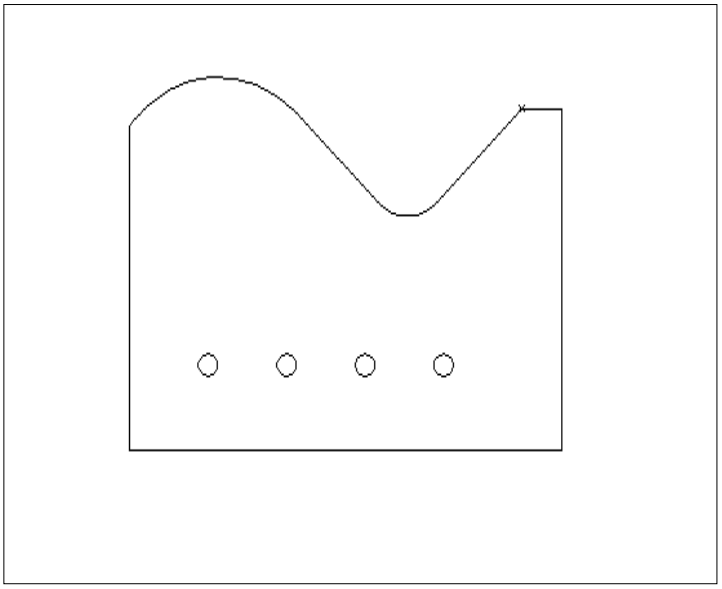


Figure 4-21 The copied holes

STEP 16

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.

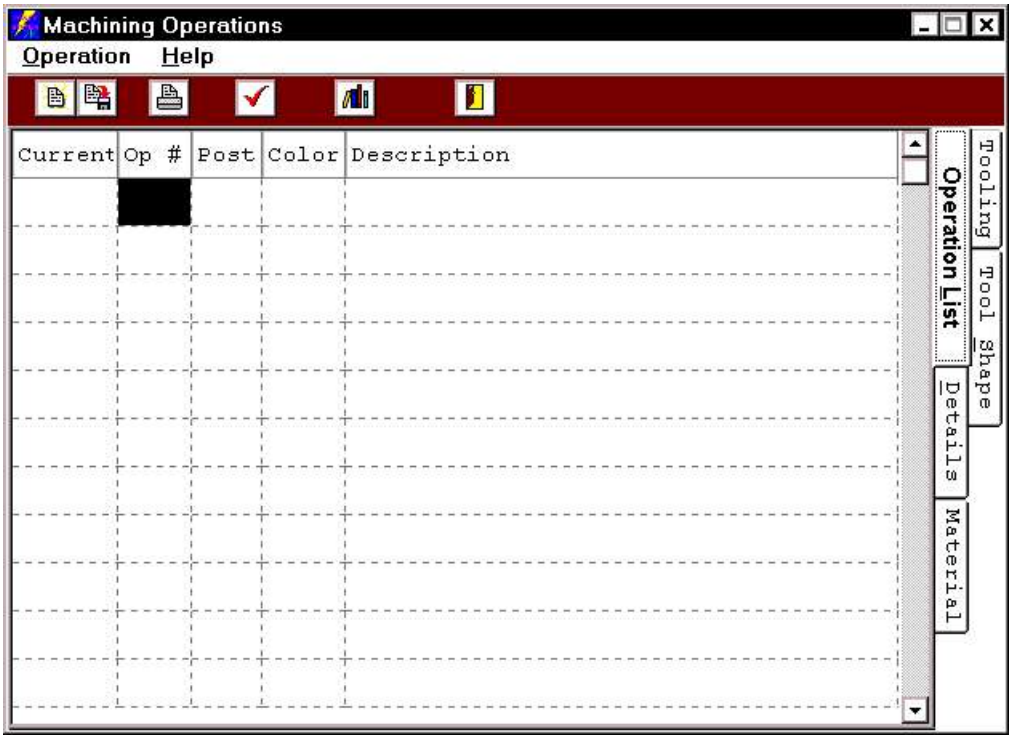


Figure 4-21a (The Machining Operations Manager)

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

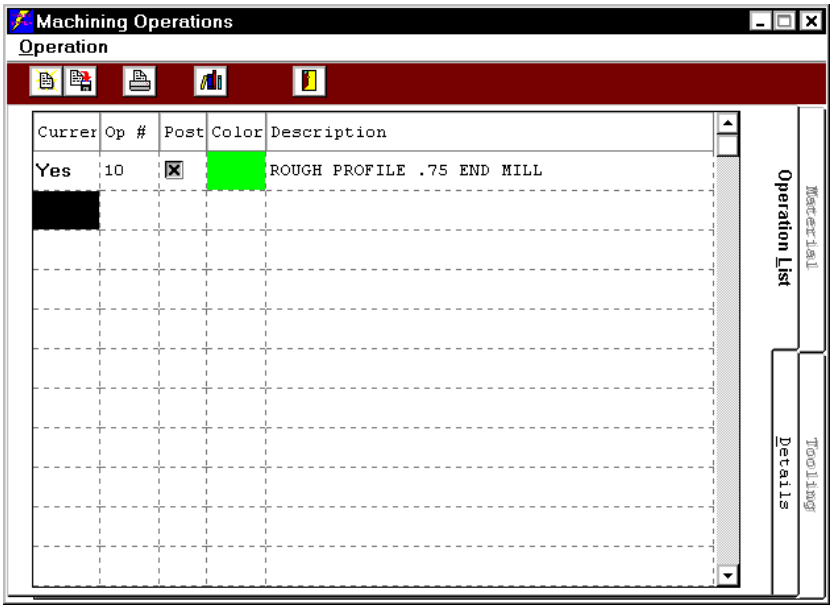


Figure 4-21b (Basic operation information)

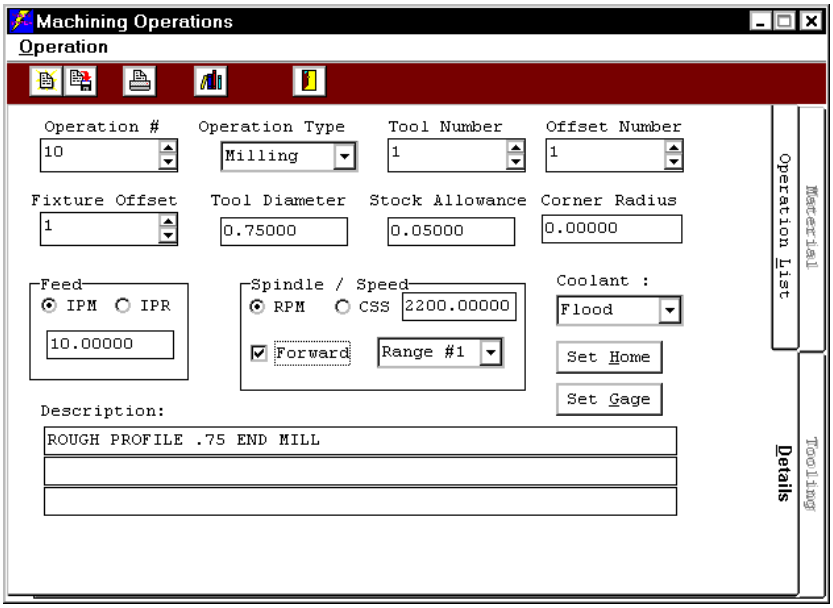


Figure 4-21c (Operation #10 Details)

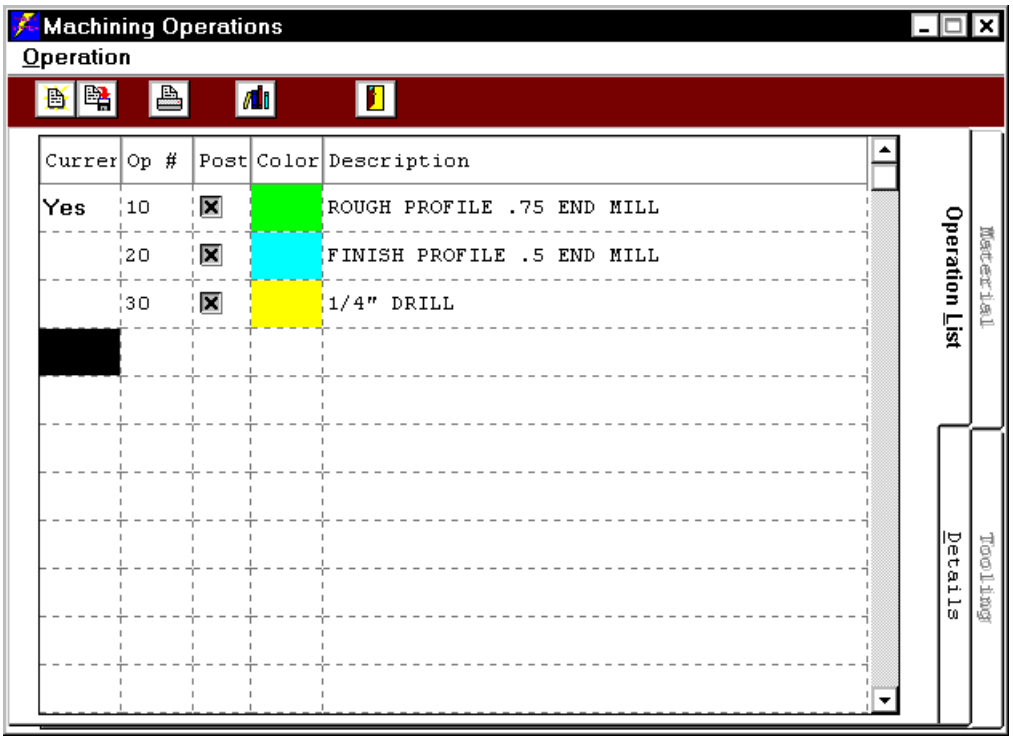


Figure 4-21d (All operations for the tutorial part)

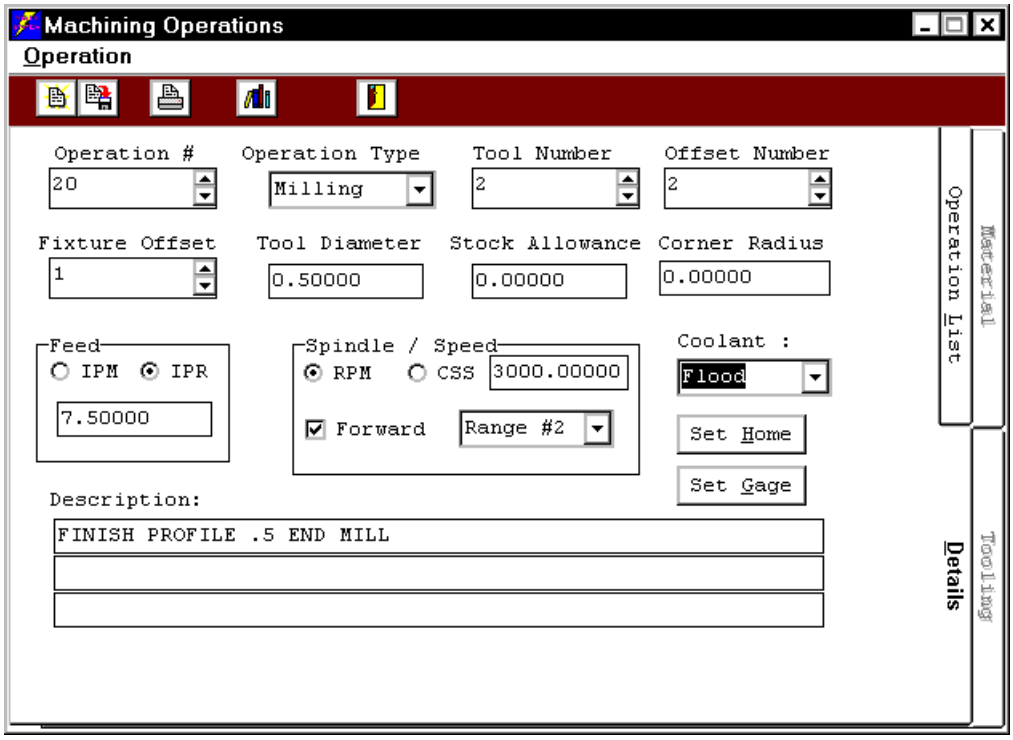


Figure 4-21e (Operation #20 for the tutorial part)

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

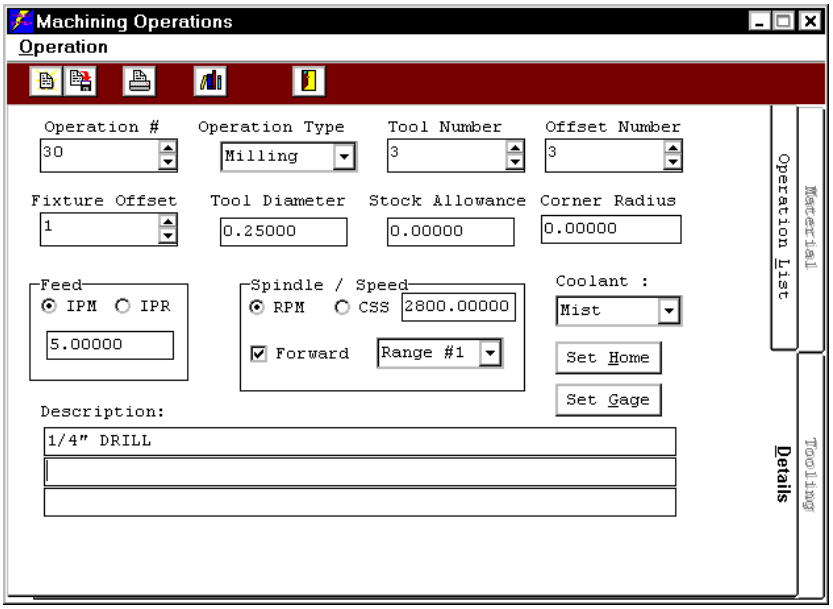


Figure 4-21f (Operation #30 for the tutorial part)

STEP 17

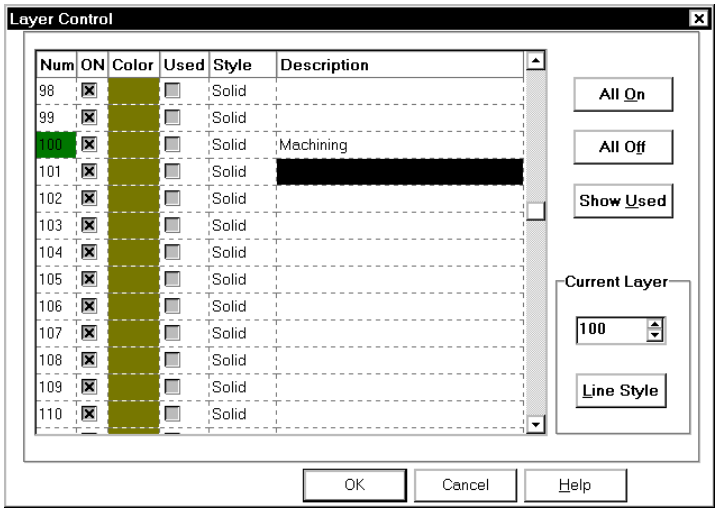
At this time it is a good idea to make a print out of the defined operations. To do this select “Operation”, “Print” from the menu. Next we need to make operation #10 active. To do this, double click on the “Current” column for operation #10, then exit from the Machining Operations Manager.

STEP 18

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 **<F9>** key (or select “Display”, “Layer-Control” from the menu), then click on “Current Layer” box and type: 100, 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.

STEP 19



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”.

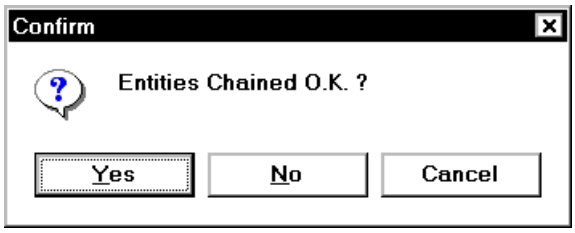
STEP 20

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).

STEPS 21/22

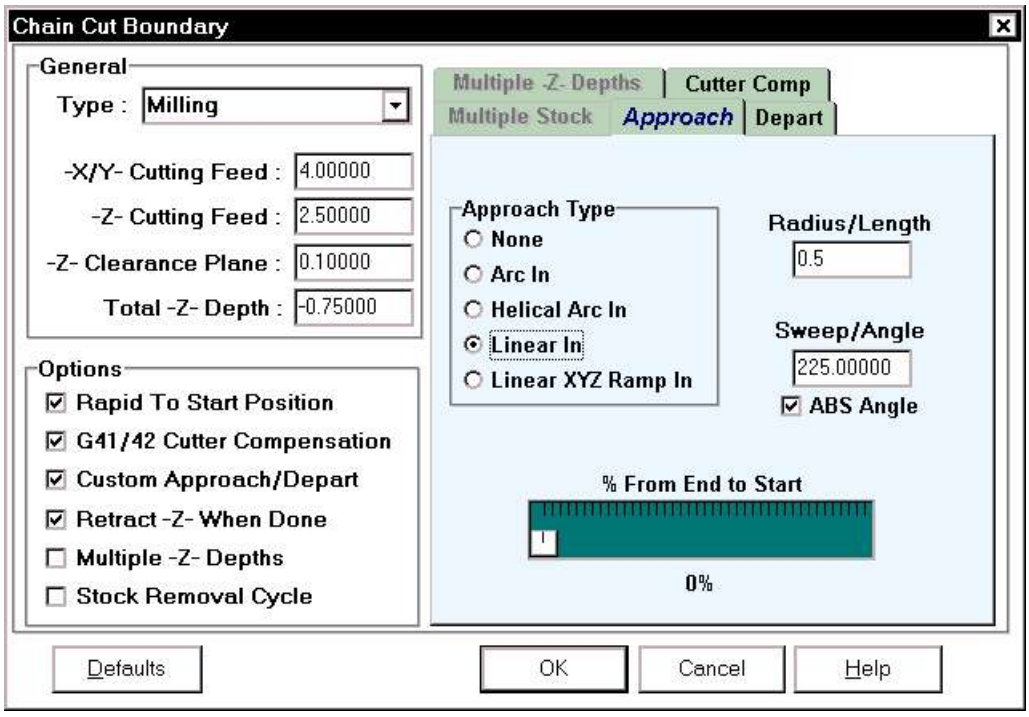
Now we will use "chain selection" to mill the outside profile. Select “Machining”, “Move & Cut”, “Chain” from the menu. We will be asked:

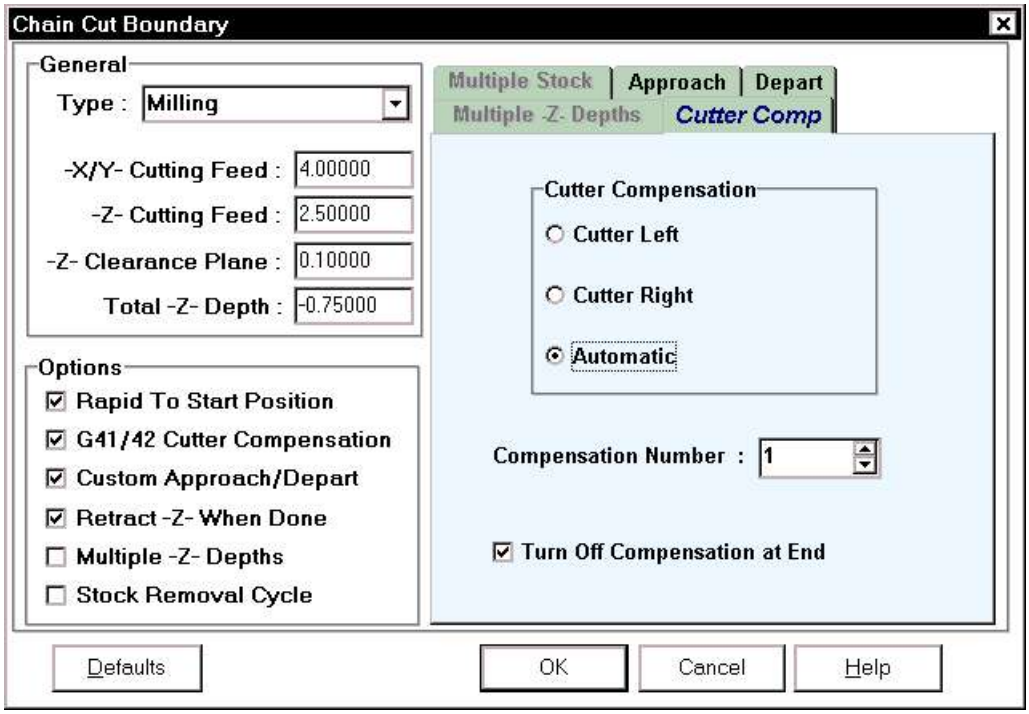
`Select starting line/arc (figure 4-24)`
`Select the Line or Arc to stop before, <Escape> for none`
(press the right mouse button, and select <Escape>)



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

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





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

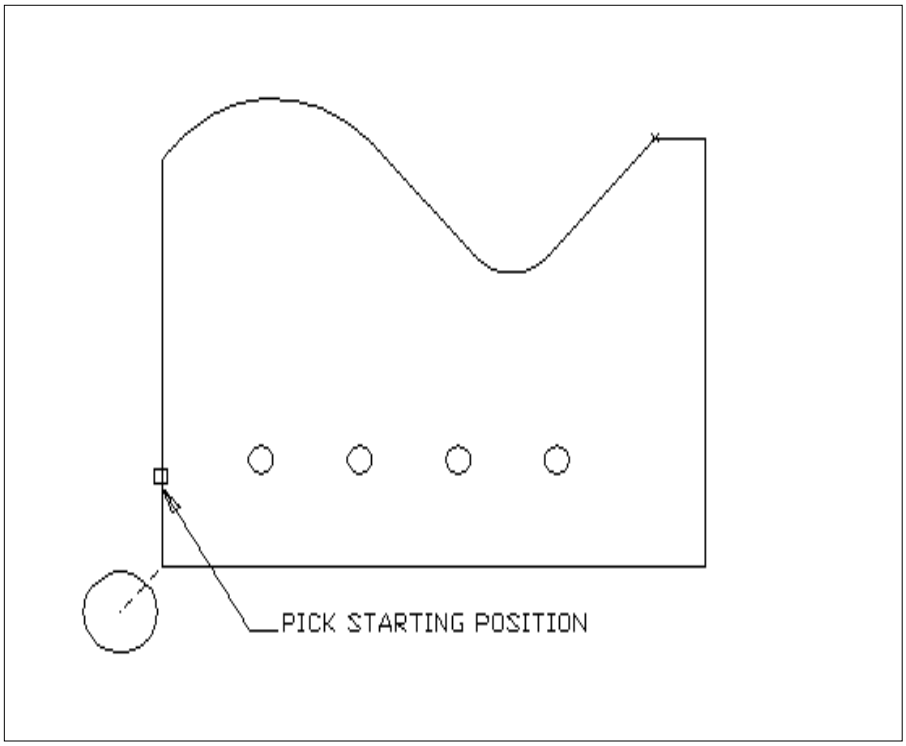


Figure 4-24 Selecting the starting line or arc

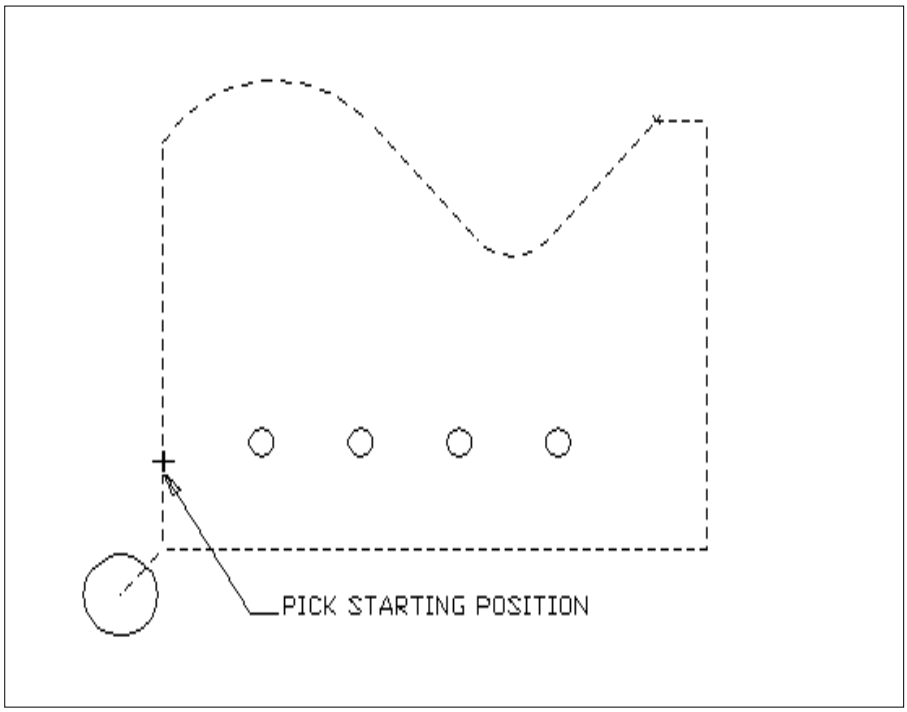


Figure 4-25 The entities have been chained

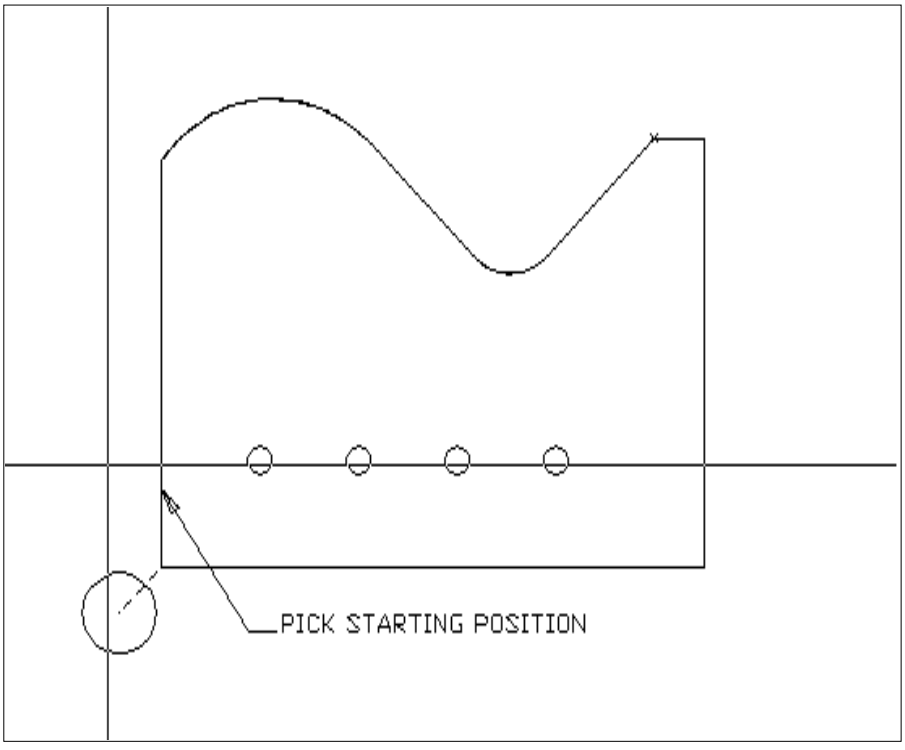


Figure 4-26 Picking the side to offset to

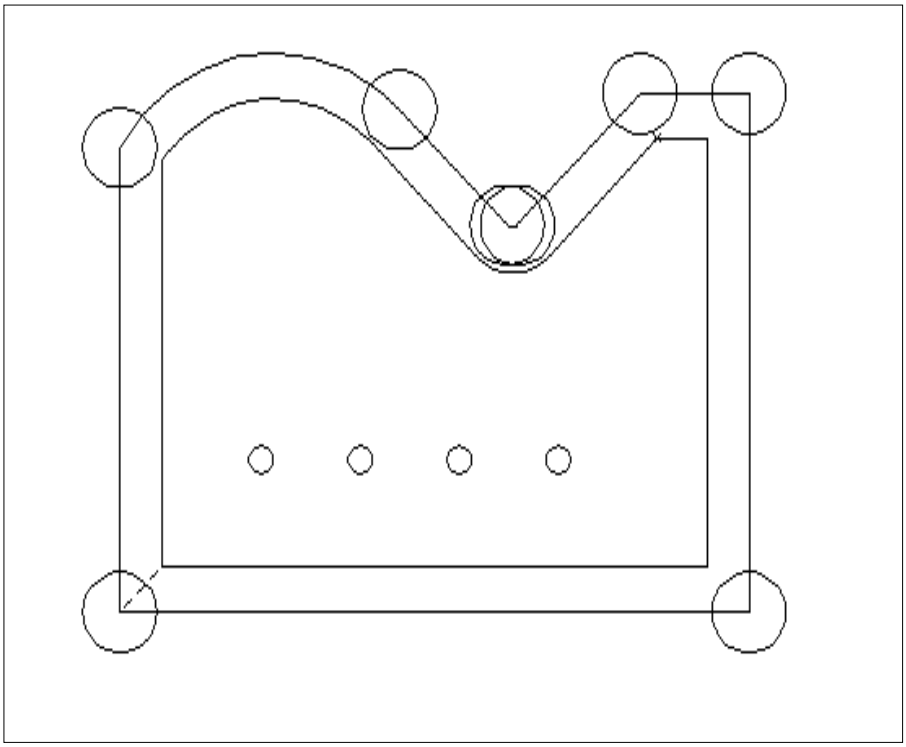


Figure 4-27 The first operation completed

STEP 23

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.

STEP 24

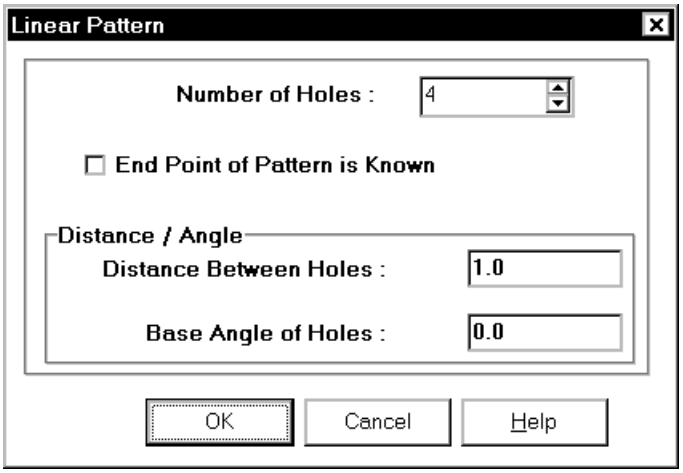
Now for the drilling. Select operation #30 by selecting “Machining”, “Operations” from the menu (or clicking the “Operations” icon). Double click on the “Current” column, in the row for operation #30. Exit the Machining Operations Manager.

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.



Fill in the “Linear Pattern” dialog as shown above.

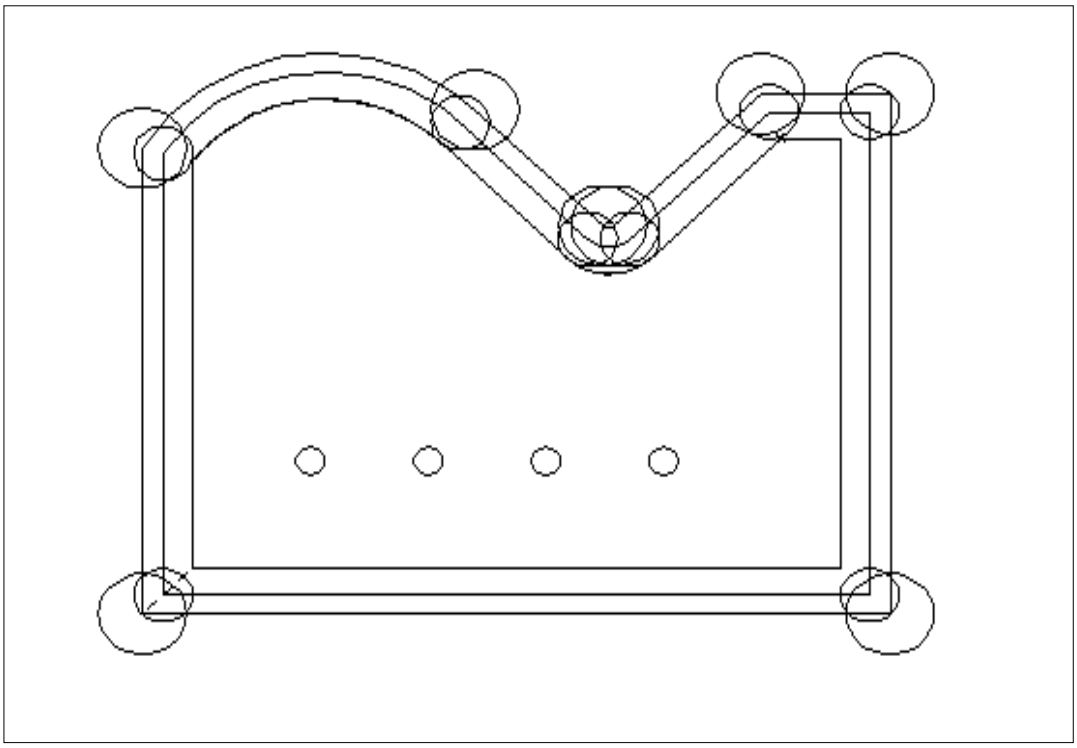


Figure 4-28 The second operation completed

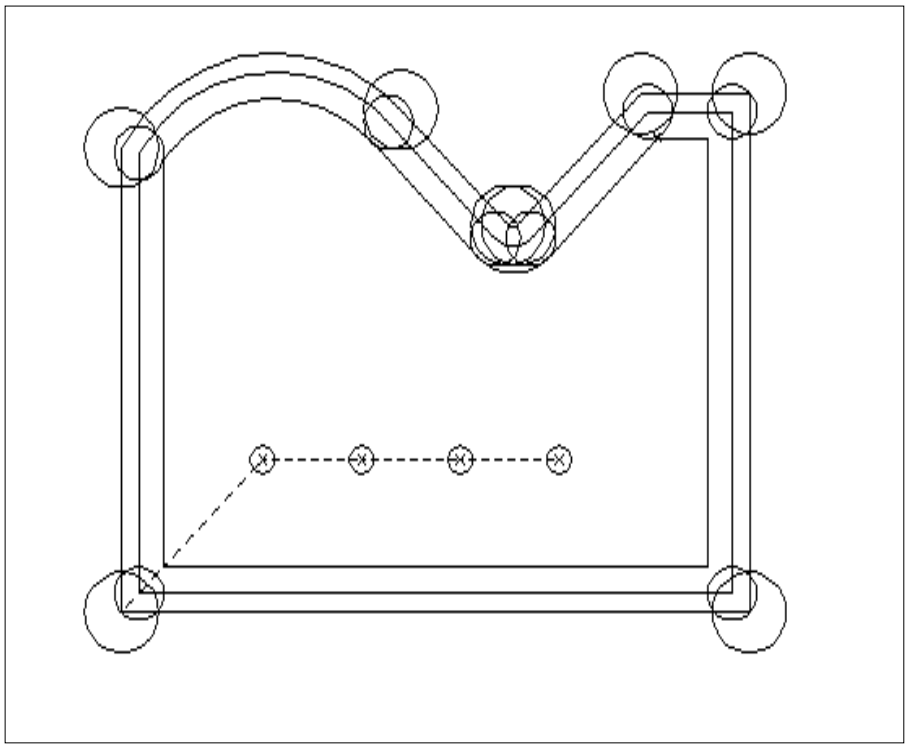
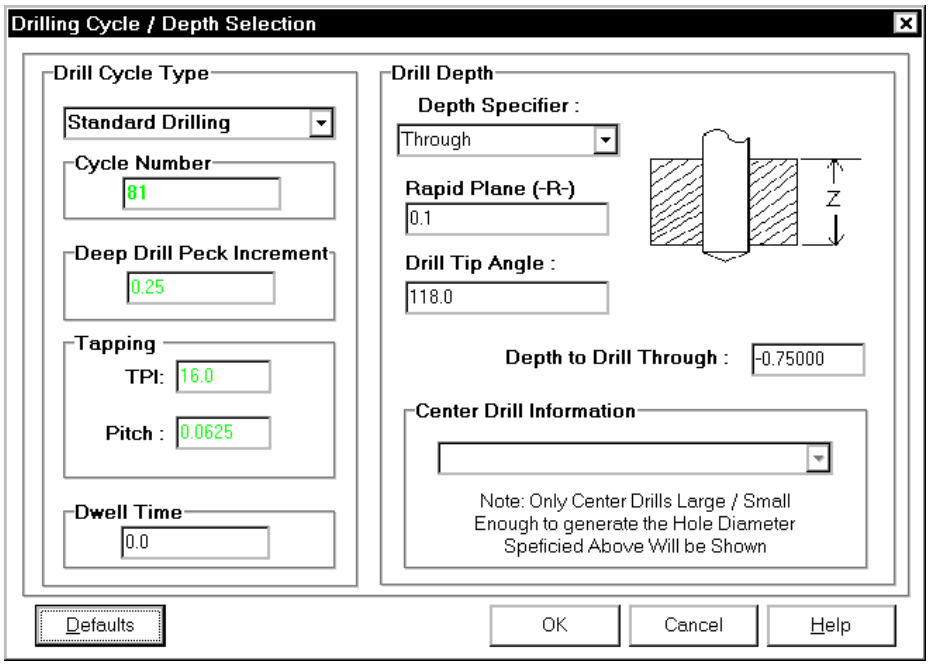


Figure 4-29 Drilling a linear pattern

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”.



See figure 4-29

STEP 25

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.

STEP 26

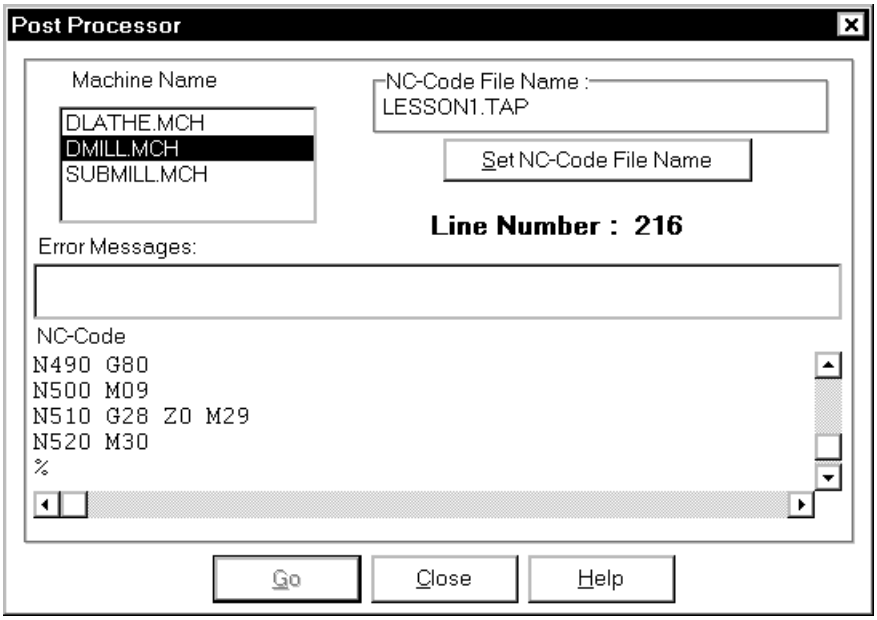
The final step in machining is to run the post processor. Select “Machining”, “Post (Generate NC-Code)” from the menu (or select the Post icon).

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”.

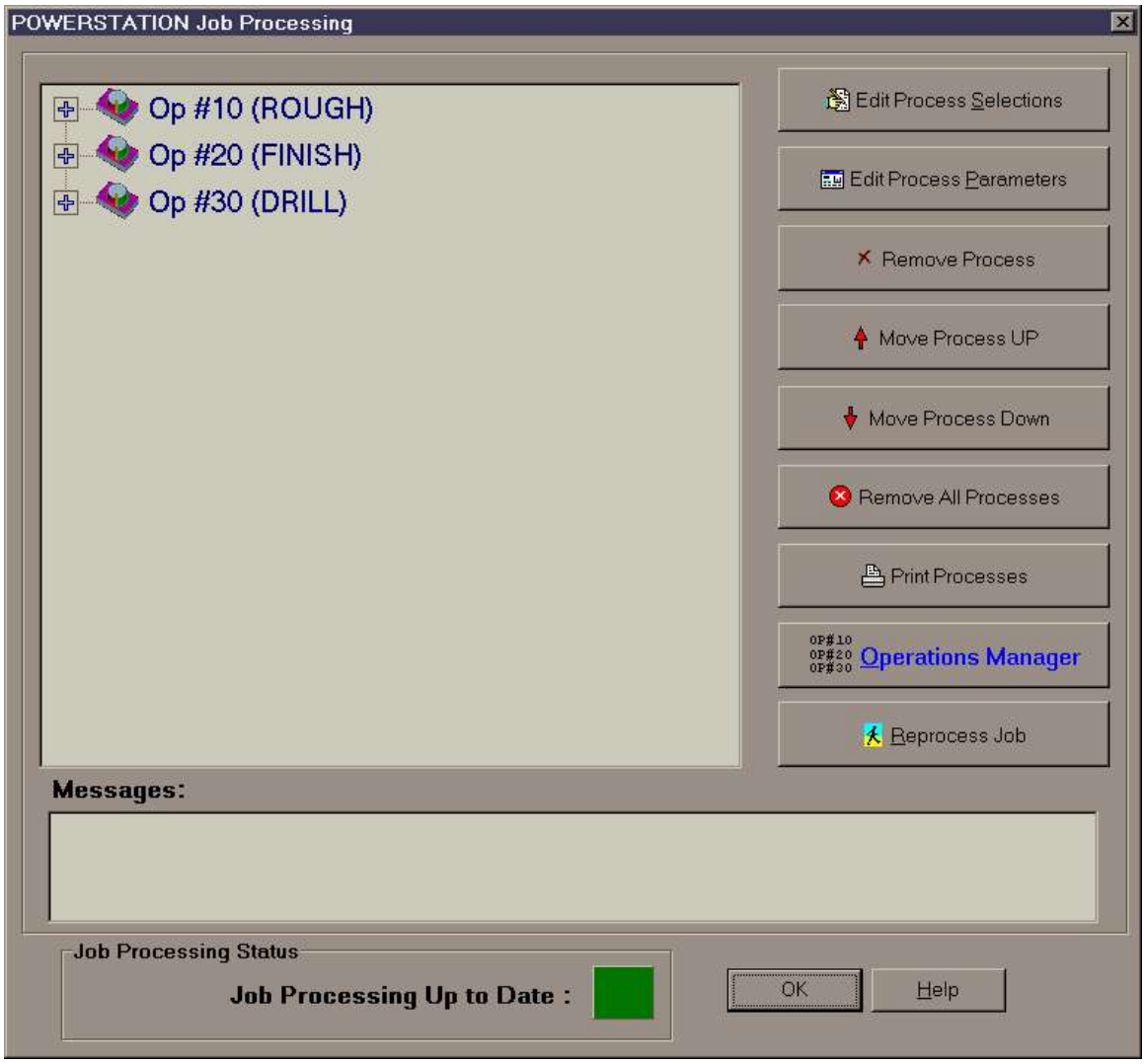


Now that the tutorial part is finished, lets see how the new “Job Processing” feature can help with making changes.

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

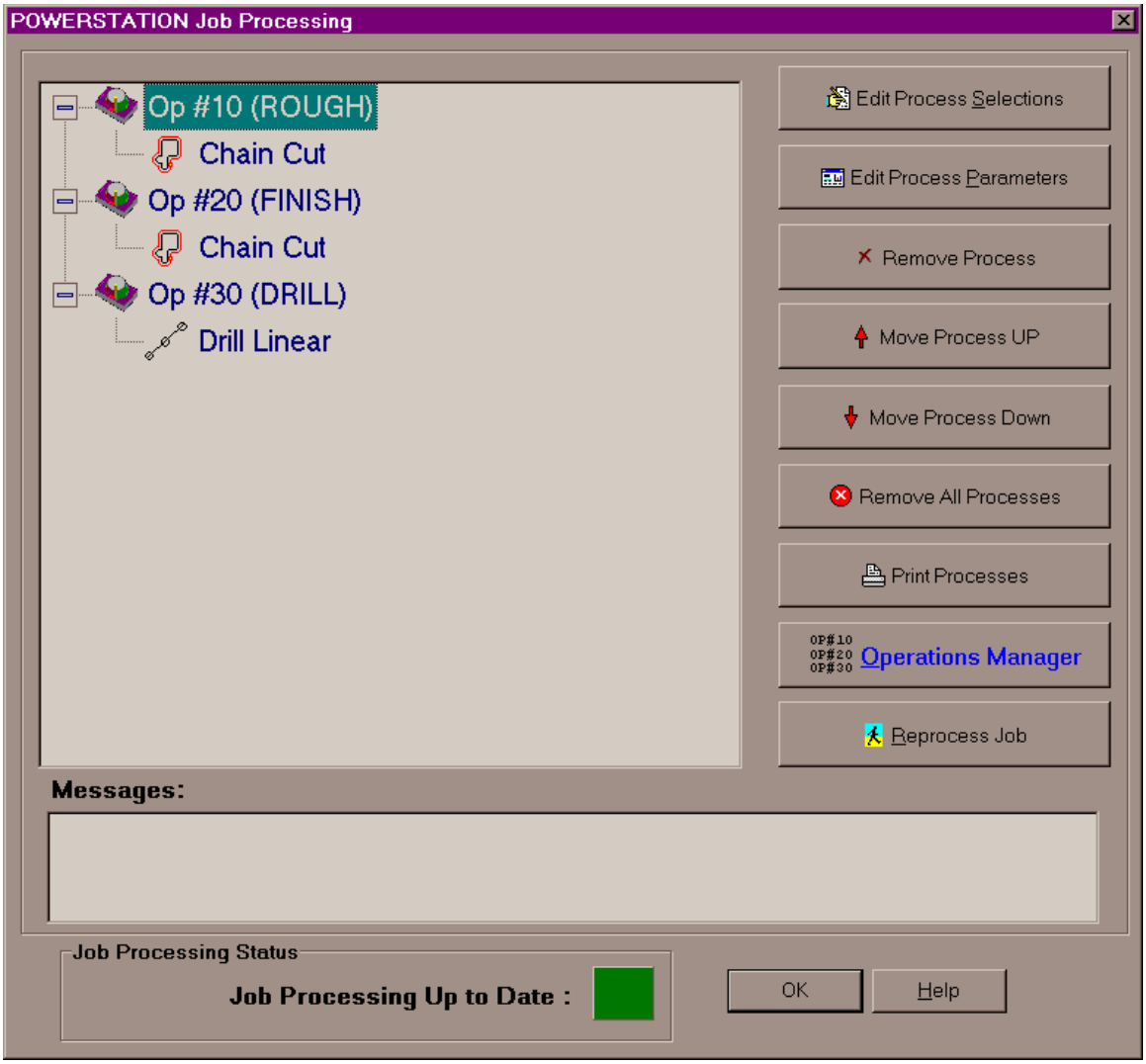
Step 45

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



Step 46

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:

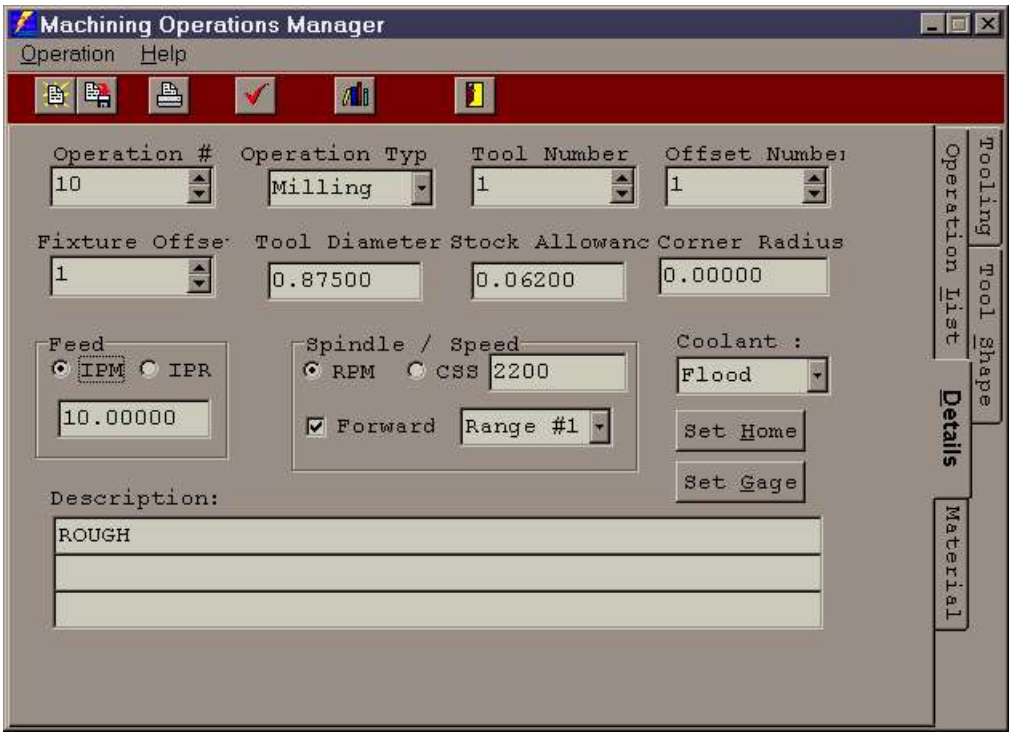


Step 47

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 ¾" end mill, leaving .05 stock, need to be changed to use a 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 .

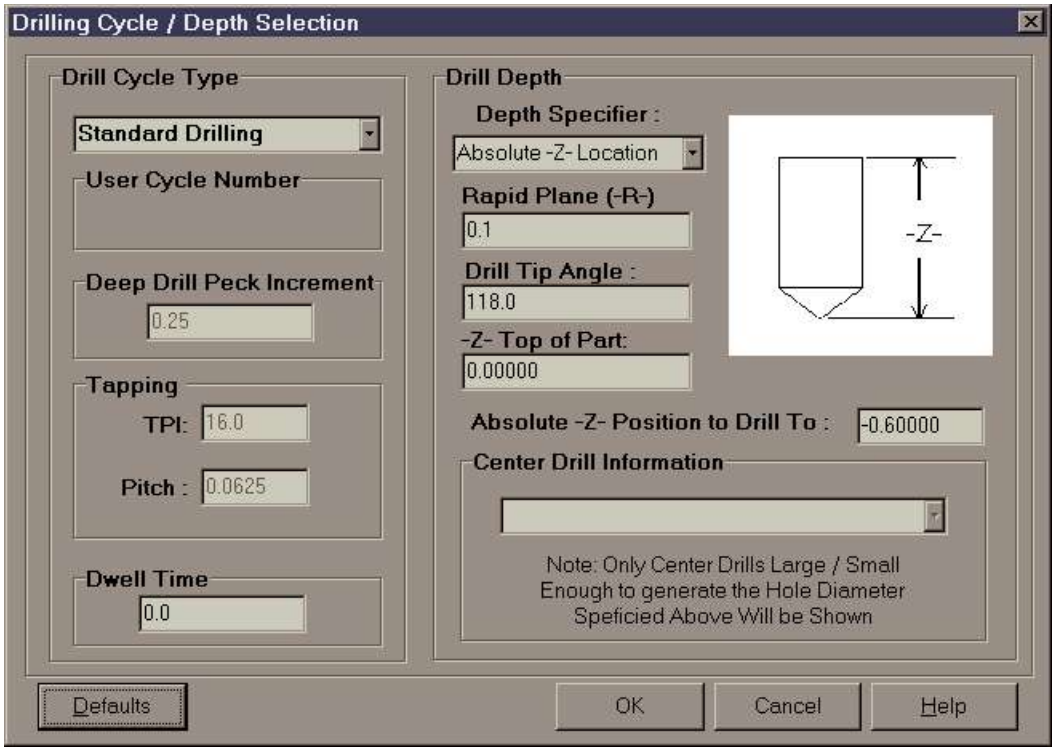
Step 48

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.



Step 49

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].



Step 50

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

Step 51

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

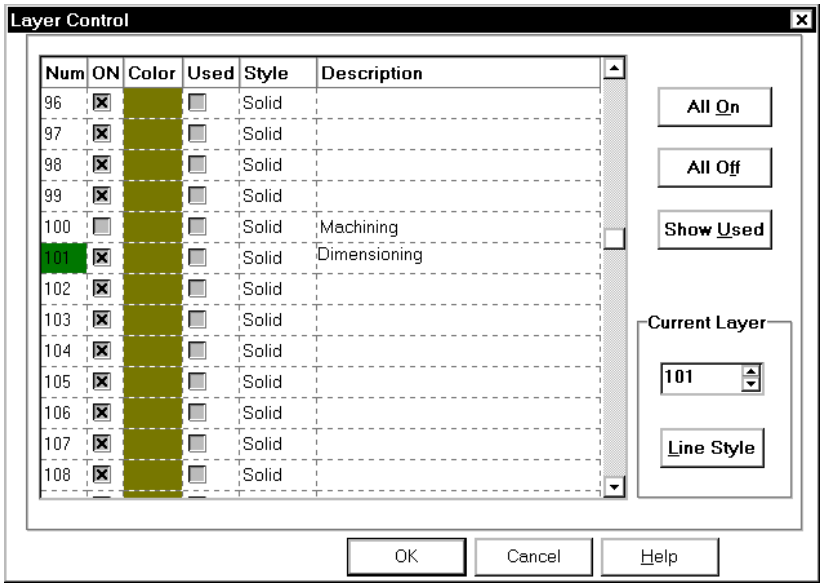
DIMENSIONING

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.

STEP 27

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 **<F9>** 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.



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

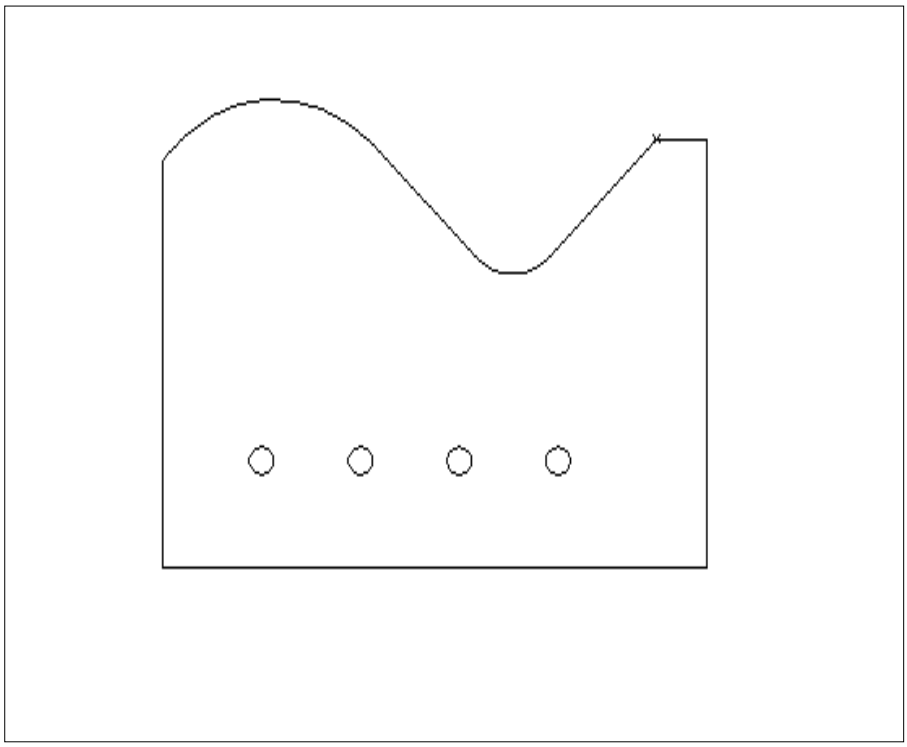


Figure 4-31 Display with the tool path layer off

STEP 28

On a color display, it is generally appealing to have the dimensions displayed in a different color that the part, so press the **<F8>** key and select a new color.

STEP 29

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

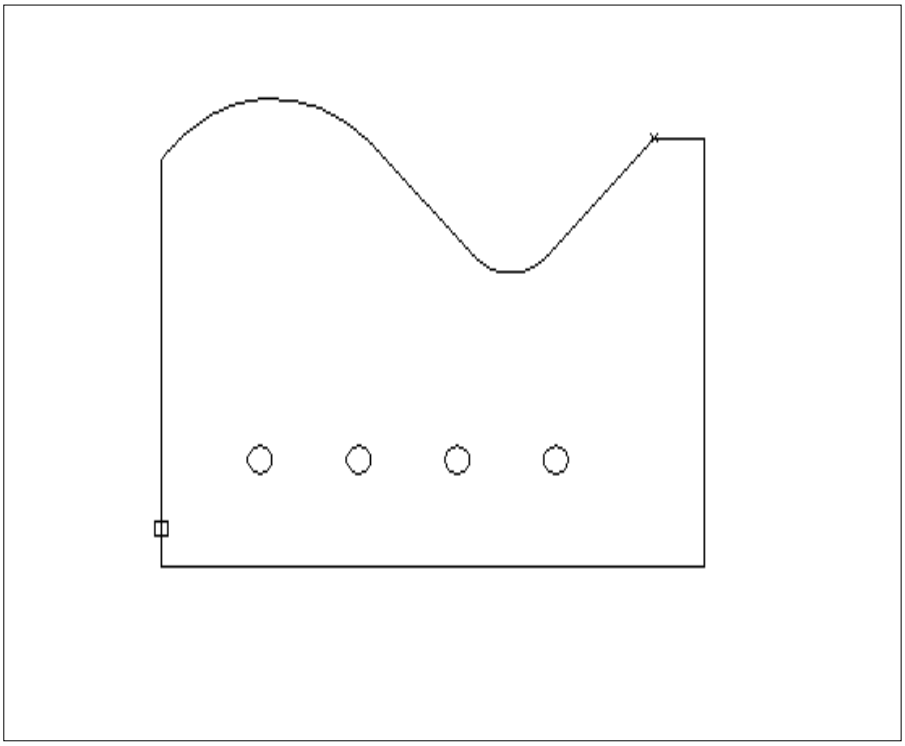


Figure 4-32 Extension line origin

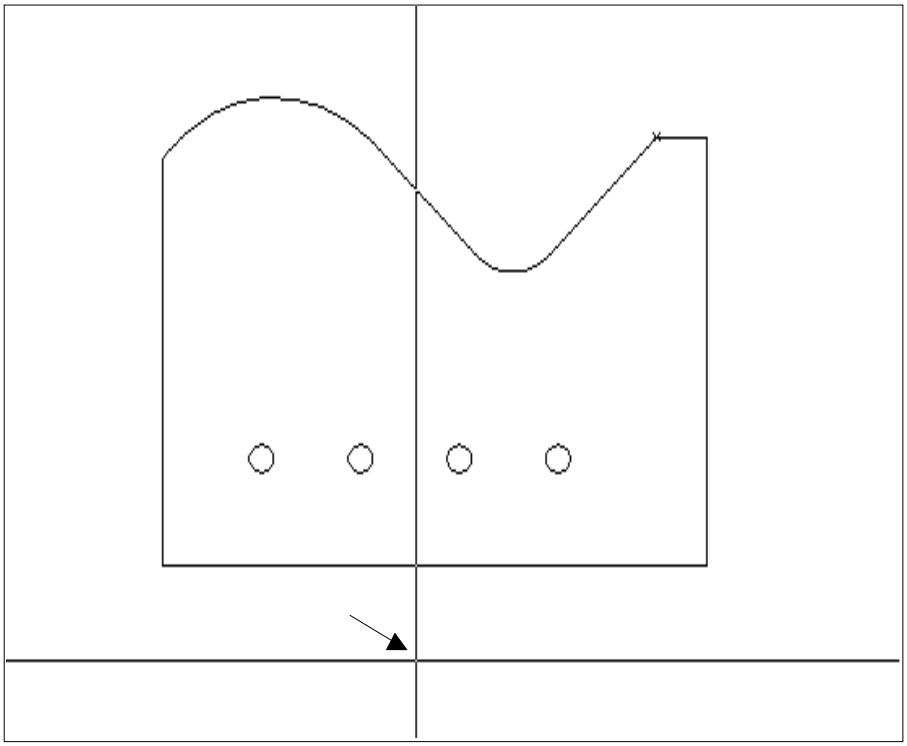


Figure 4-33 Indicating the approximate text point

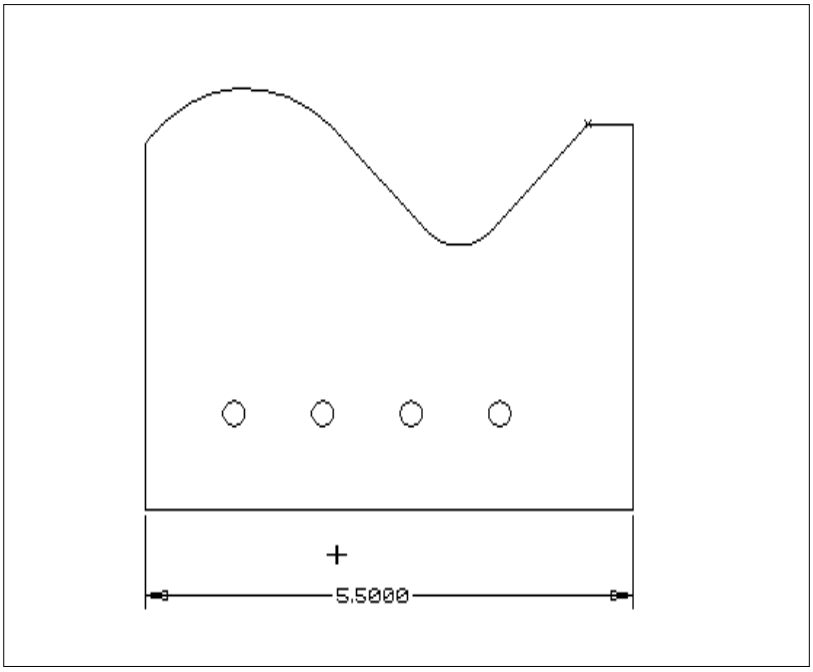


Figure 4-34 Dimensioning the overall width

STEP 30

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] ? <Enter>

The dimension should look like figure 4-36

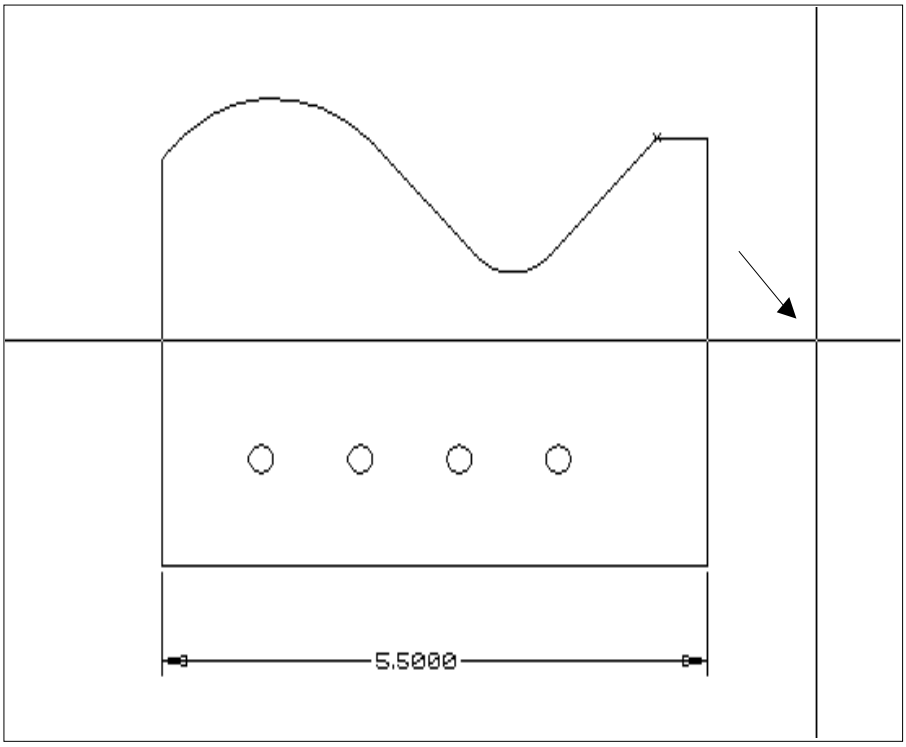


Figure 4-35 Constructing the vertical dimension

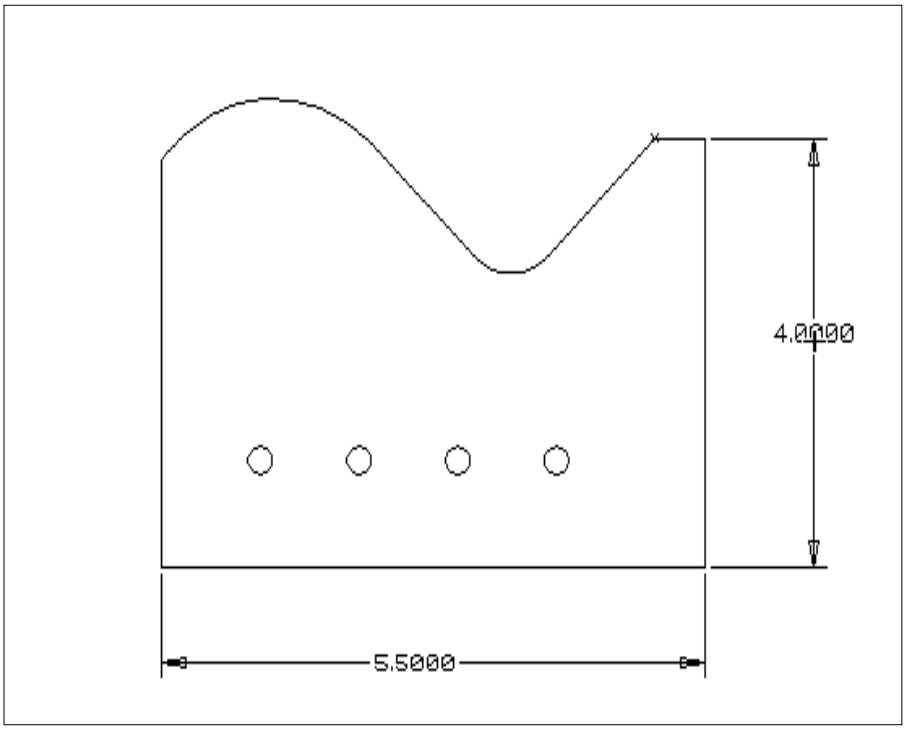


Figure 4-36 Dimensioning the overall height

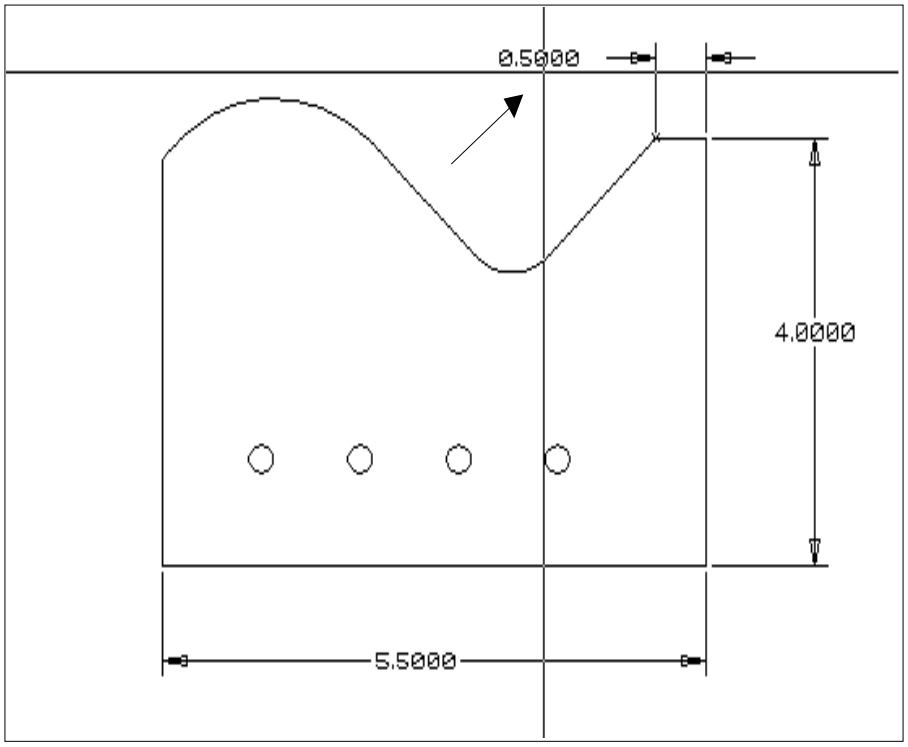


Figure 4-37 Dimensioning the top most line

STEP 31

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

STEP 32

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

STEP 33

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

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

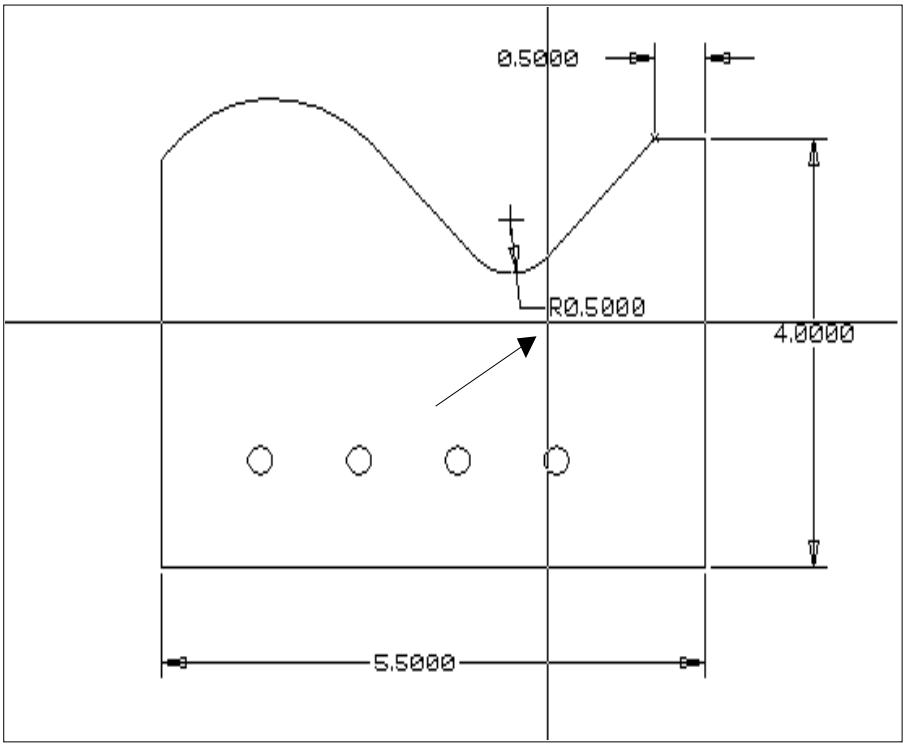


Figure 4-38 Dimensioning the 1/2" radius

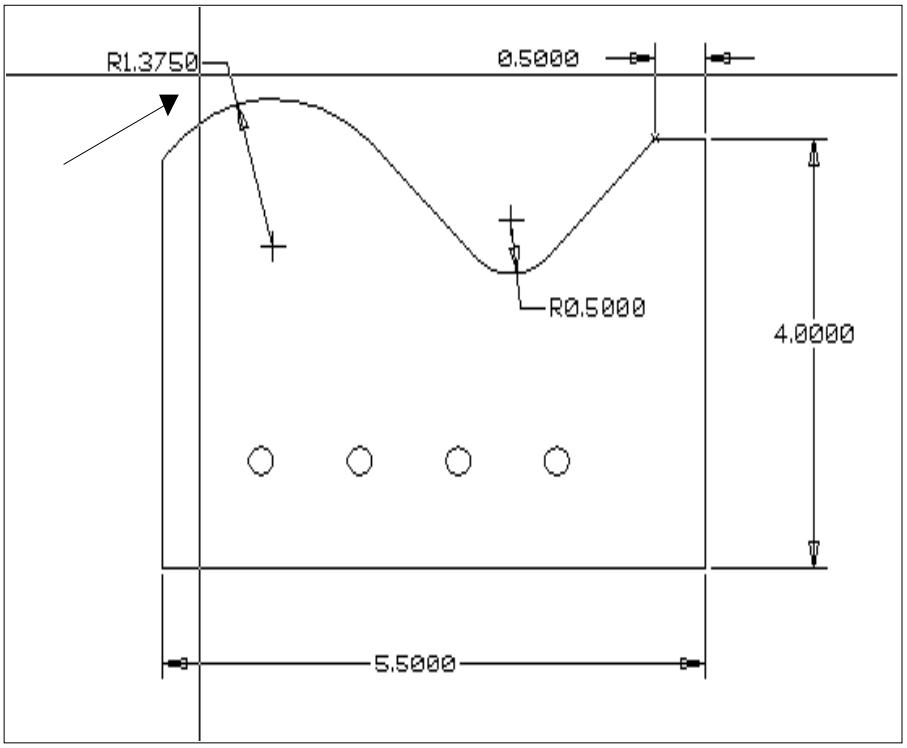


Figure 4-39 Dimensioning the 1 3/8" radius

STEP 34

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

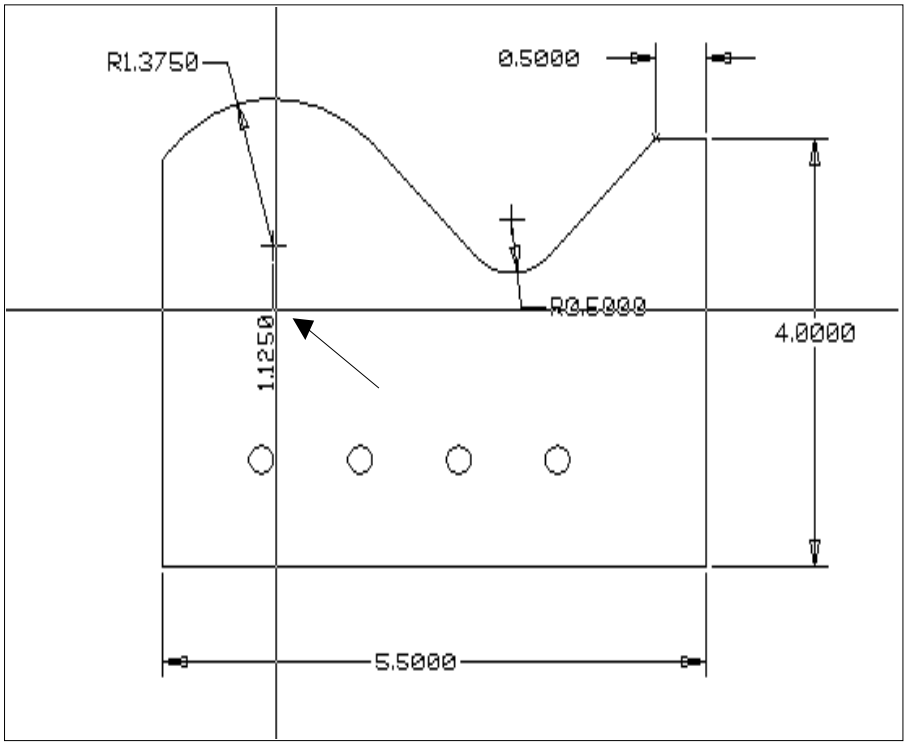


Figure 4-40 Dimensioning the X center of the 1 3/8" radius

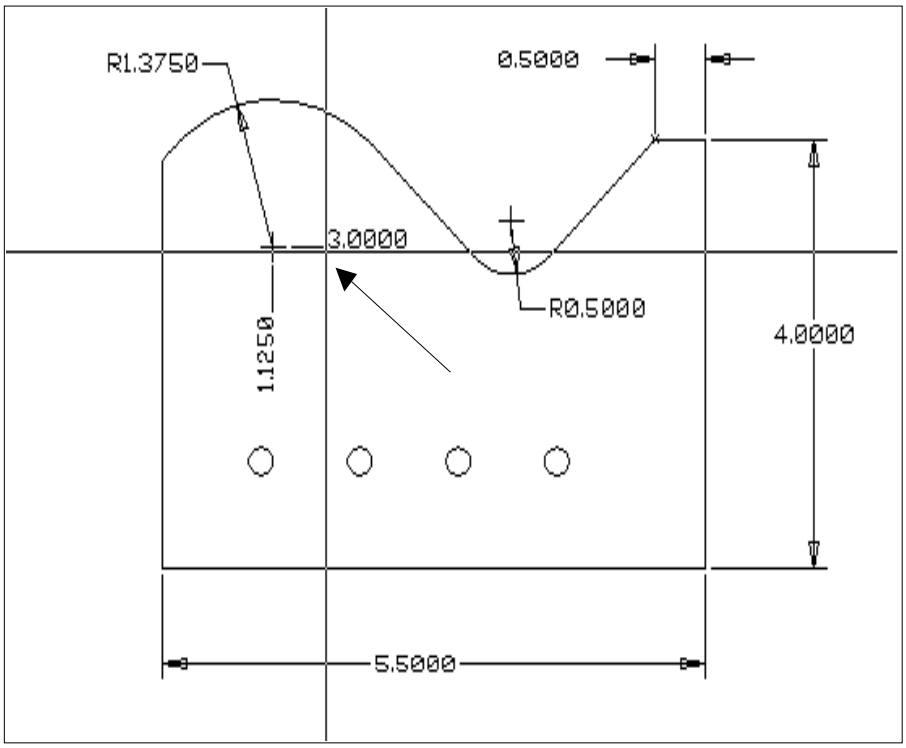


Figure 4-41 Dimensioning the Y center of the 1 3/8" radius

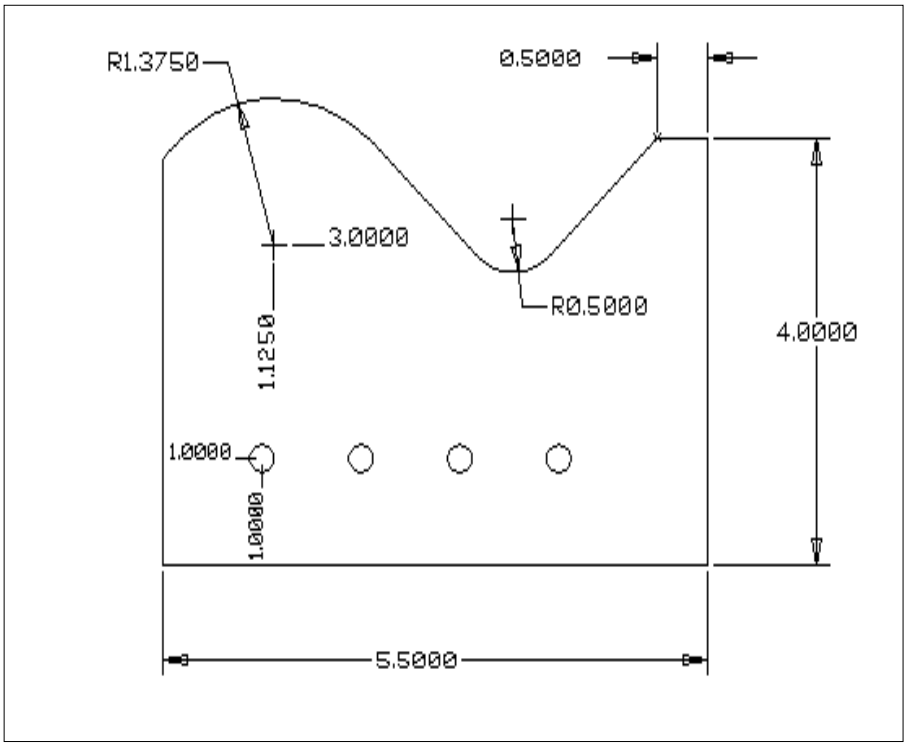


Figure 4-42 Ordinate dimensions

STEP 35

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

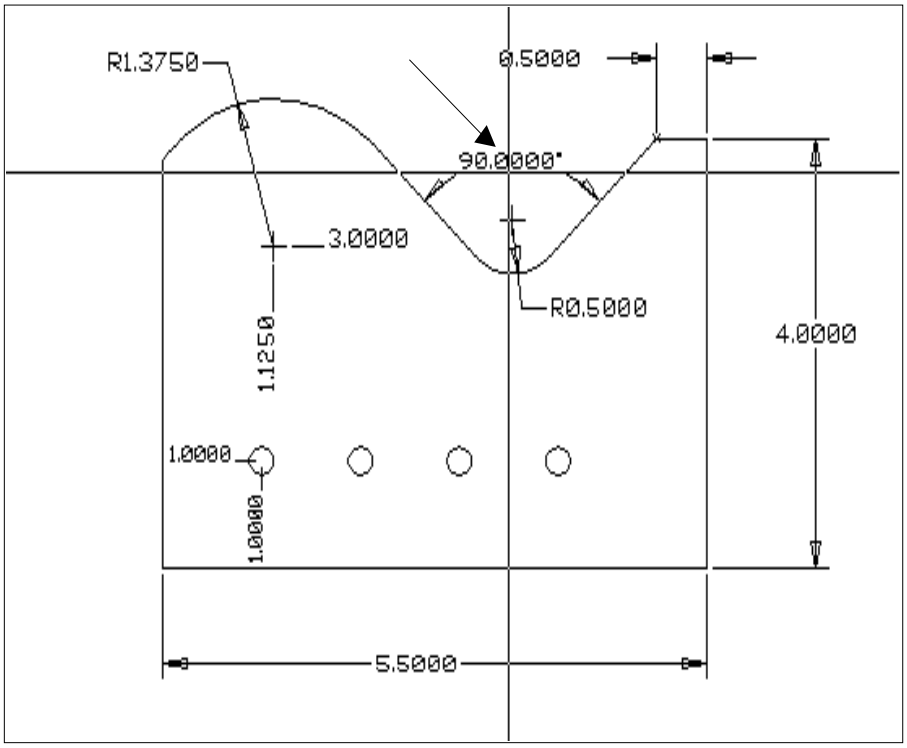


Figure 4-43 Constructing the angular dimension

STEP 36

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

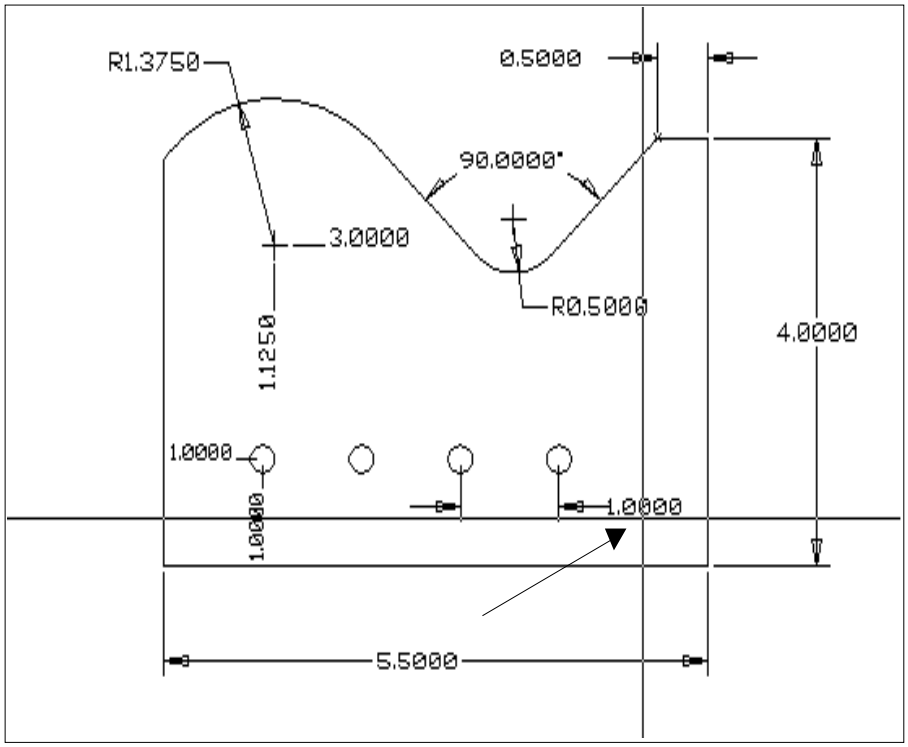


Figure 4-44 Dimensioning the distance between holes

STEP 37

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] ? A<Enter>
```

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

Finally enter the text “Typical”.

STEP 38

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

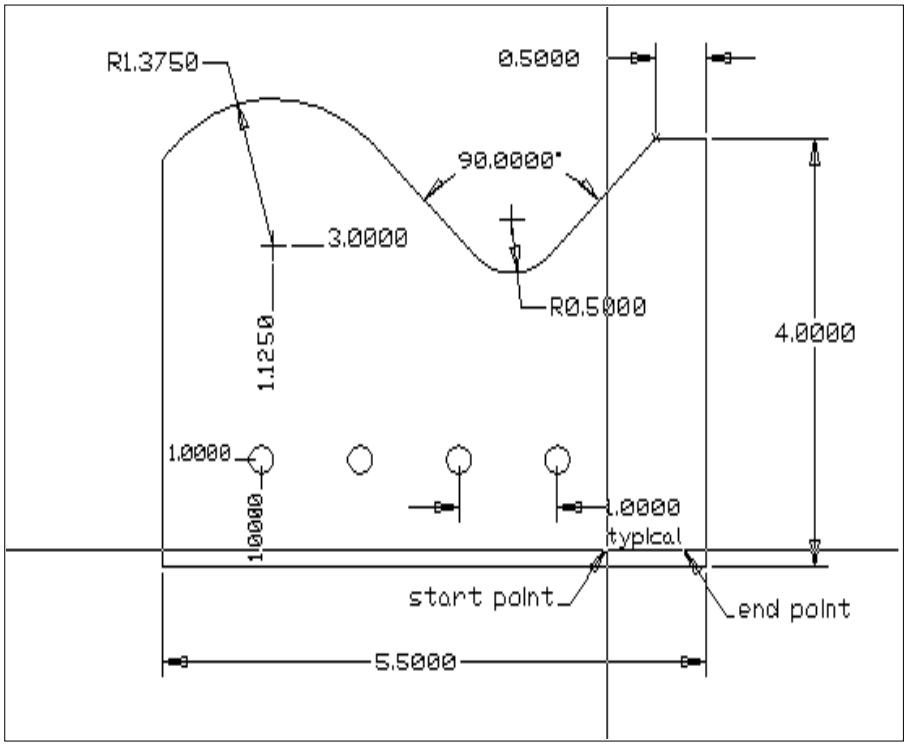


Figure 4-45 Adding text

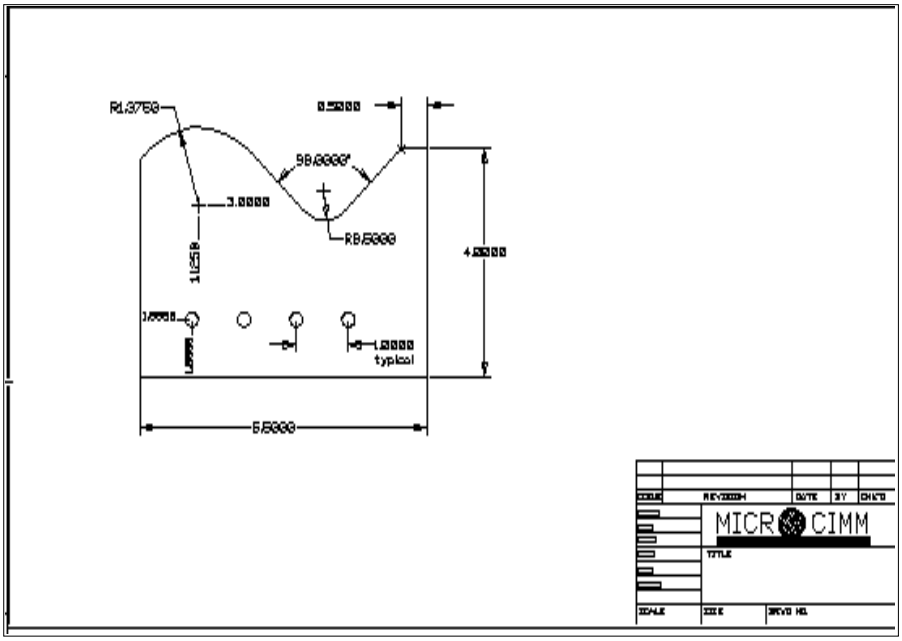


Figure 4-46 Title block and sheet frame

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

STEP 39

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] ? .75
-Y- value [0.0000] ? 4
```

The display should now look like figure 4-47

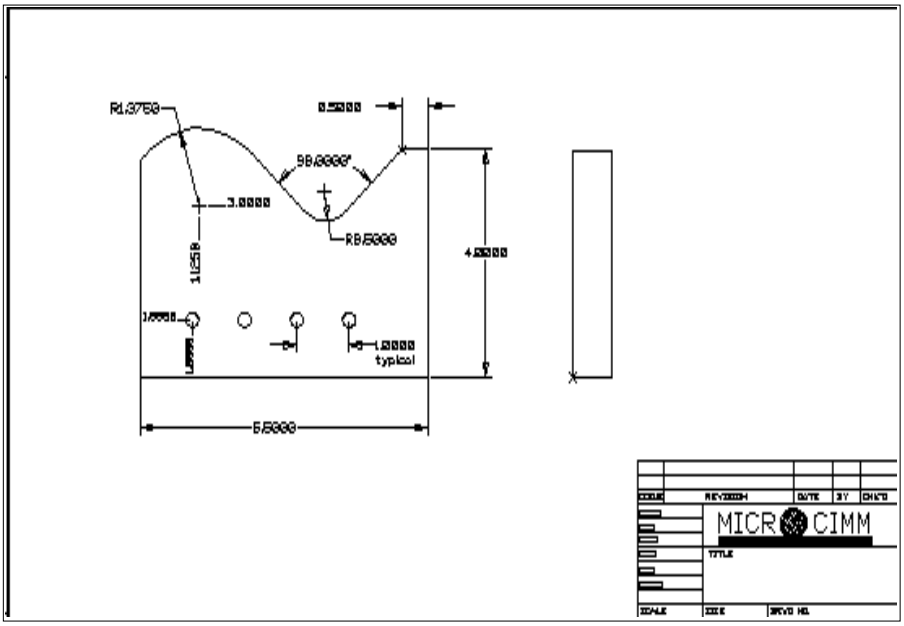


Figure 4-47 Adding a side view

STEP 40

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

STEP 41

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.

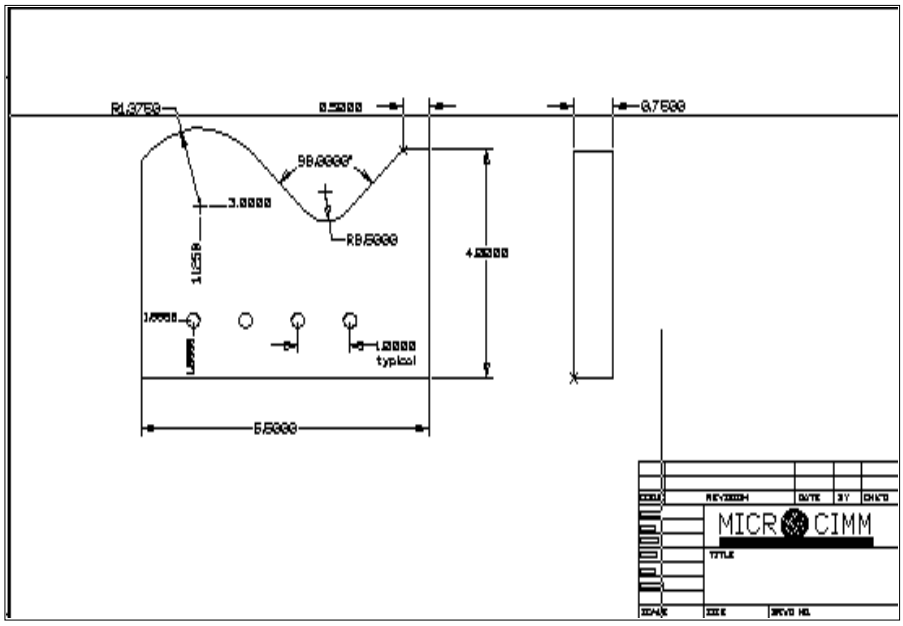


Figure 4-48 Dimensioning the part thickness

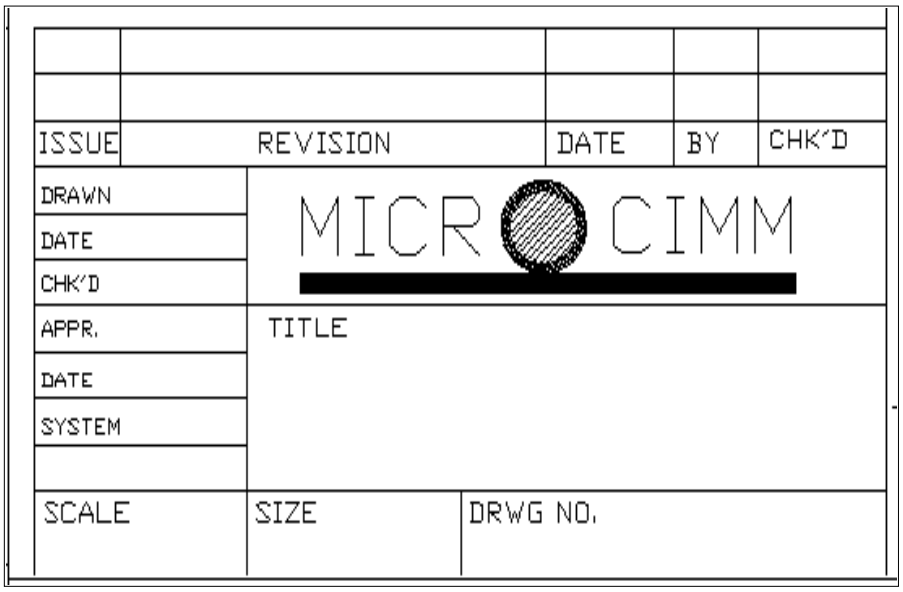
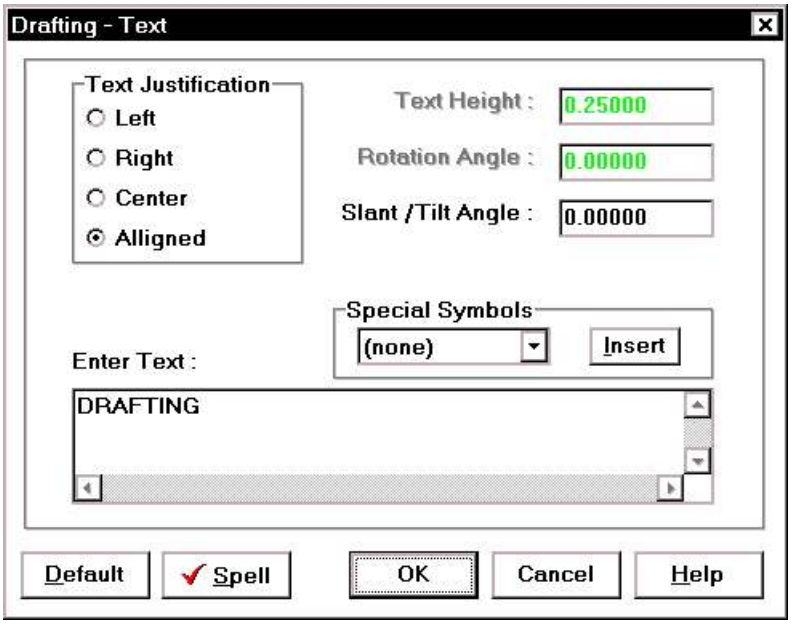


Figure 4-49 Zooming in on the title block

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:



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

STEP 42

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

STEP 43

Save the drawing as in step 15.

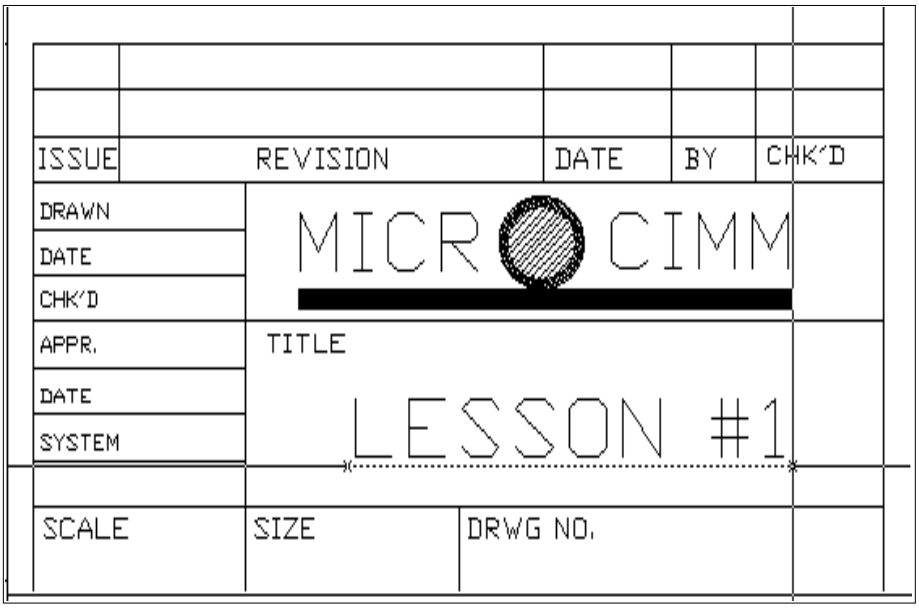


Figure 4-50 Adding the drawing name

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

STEP 44

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

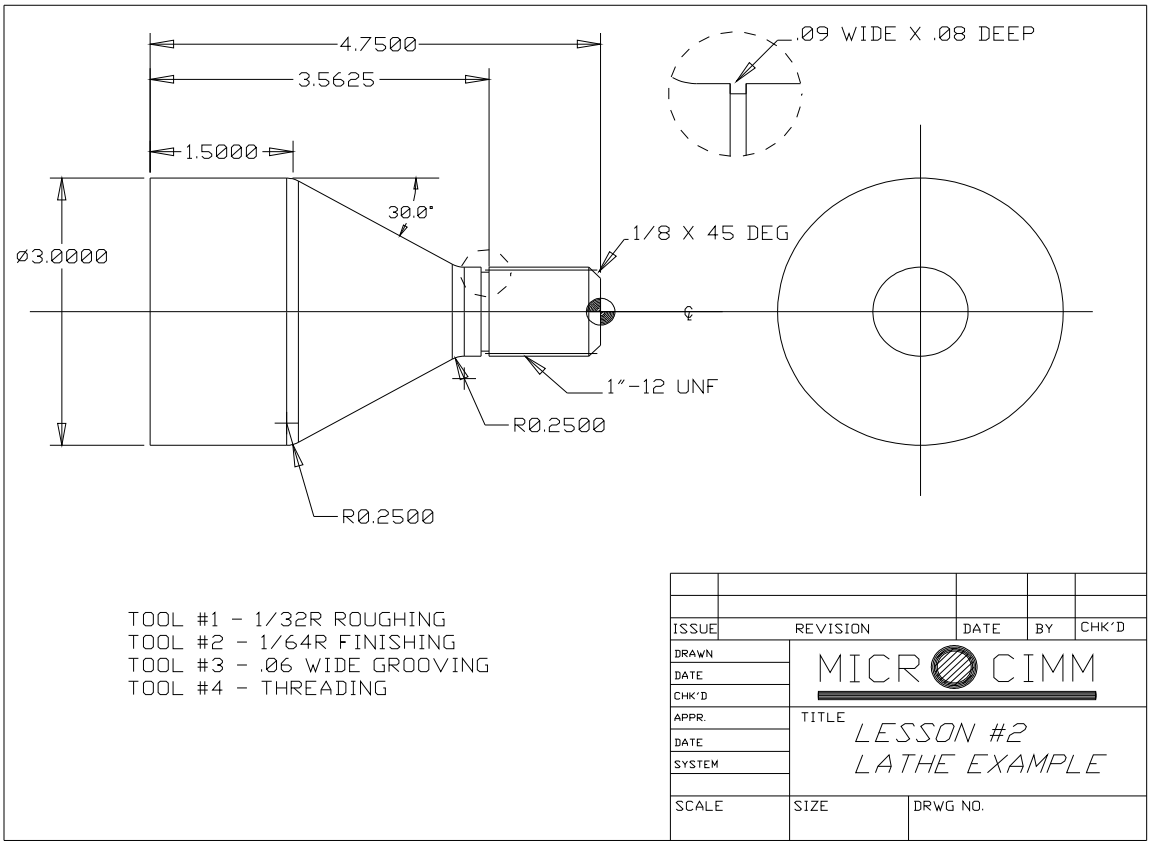


Figure 4-60 The Sample Lathe Drawing

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.

STEP 1

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

STEP 2

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.

STEP 3

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.

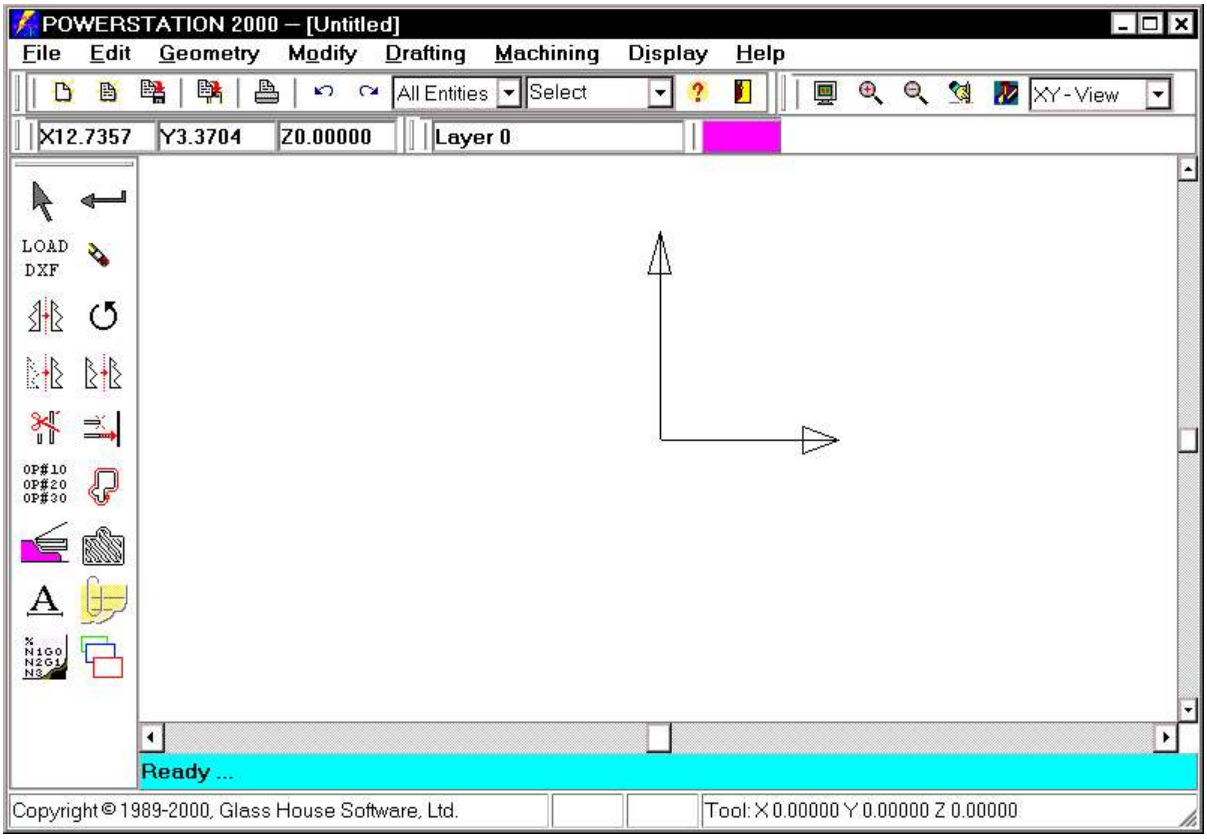


Figure 4-62 The initial display showing the axis

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.*

STEP 4

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.

STEP 5

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.

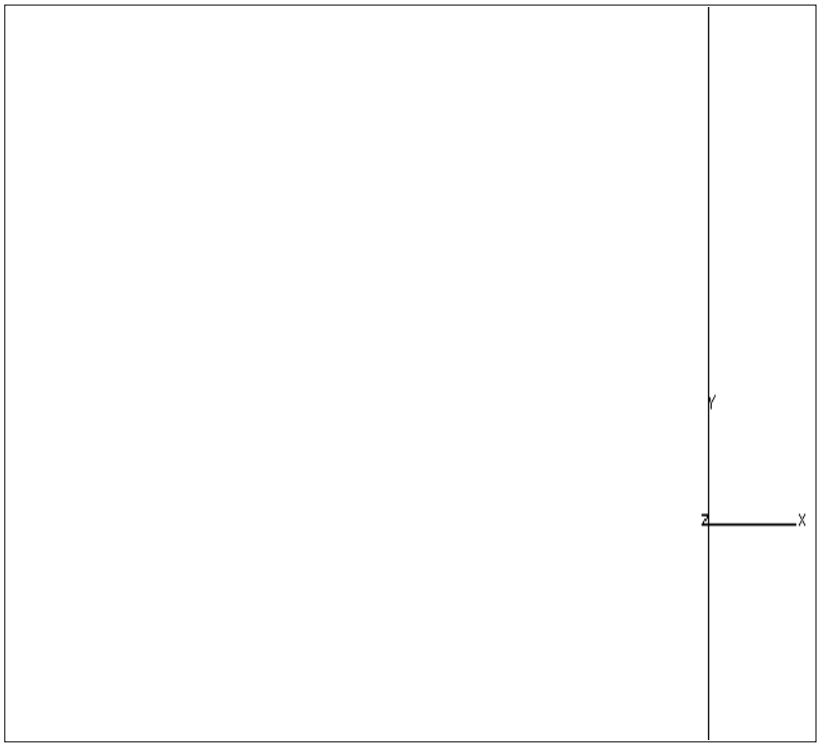


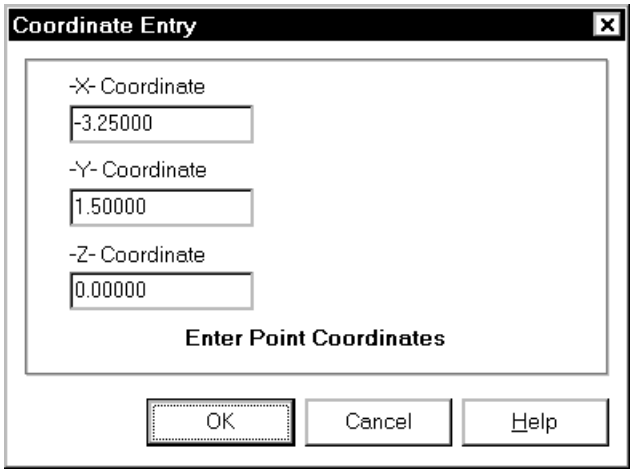
Figure 4-63 Defining the Front Face

Figure 4-64 One inch diameter

STEP 6

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:



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

See figure 4-65.

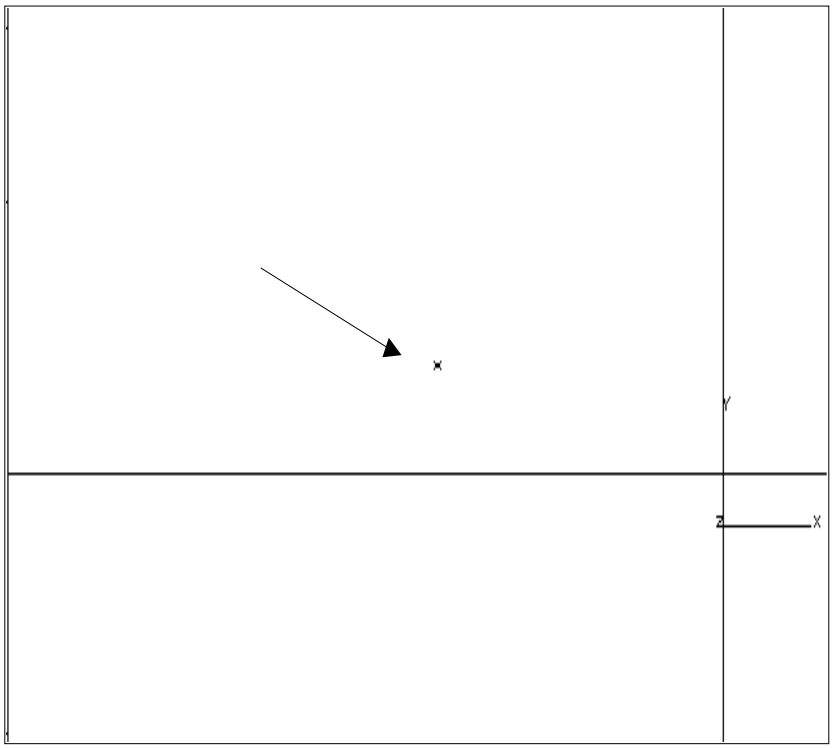


Figure 4-65 The construction point

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.

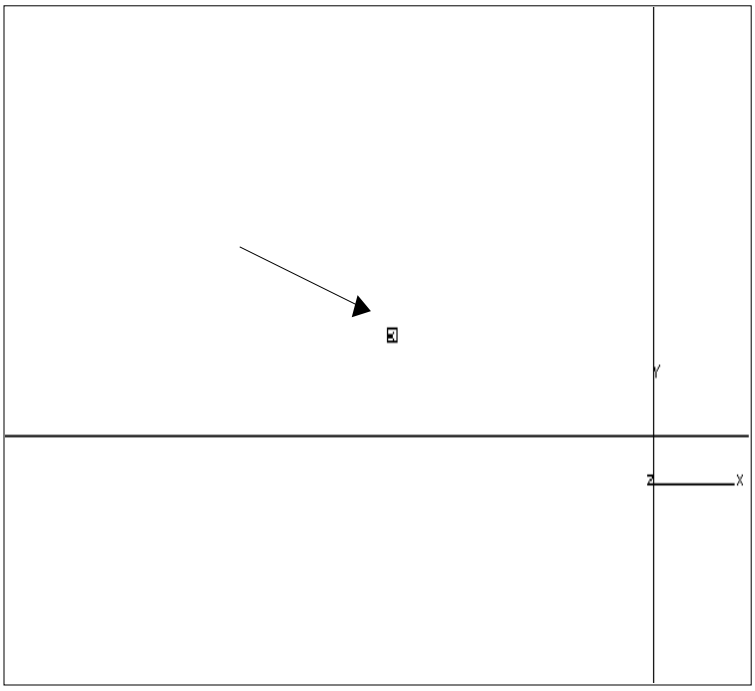


Figure 4-66 Selecting the point

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

The display will now look like figure 4-67.

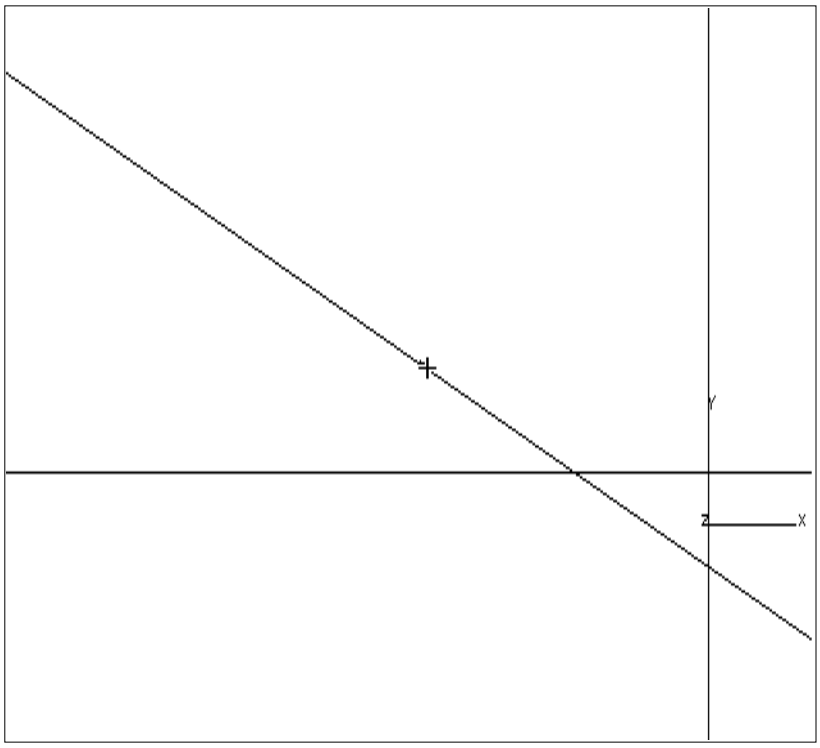
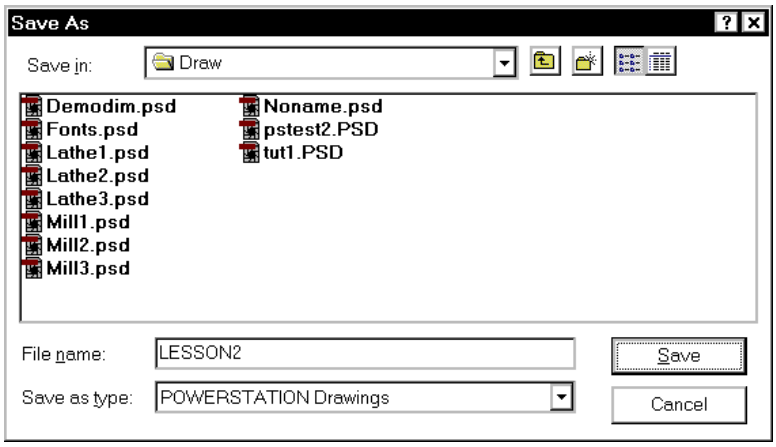


Figure 4-67 The 30 degree line

STEP 7

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.



STEP 8

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.

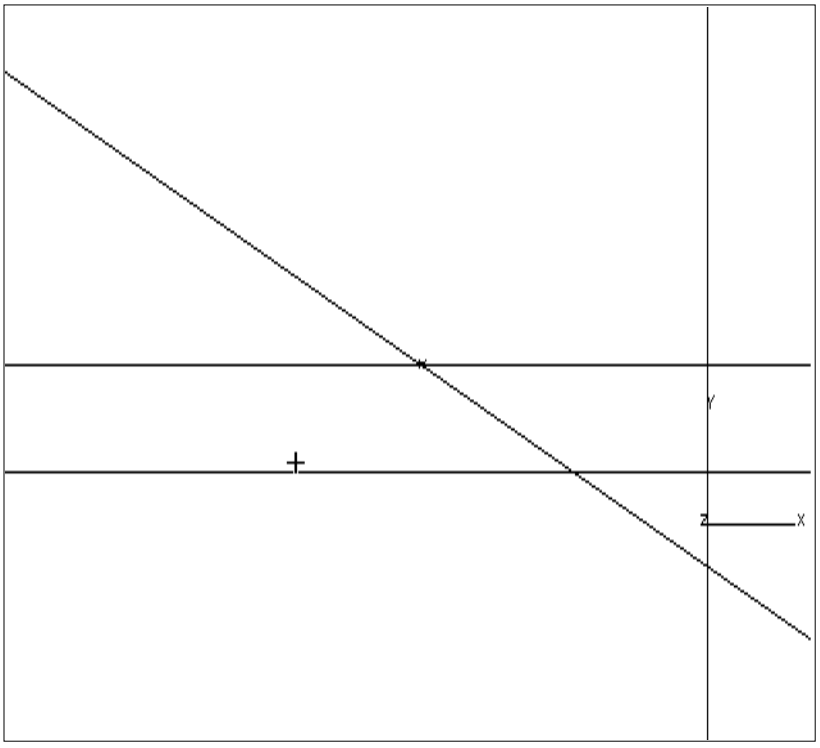


Figure 4-68 The 3 inch diameter

STEP 9

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.

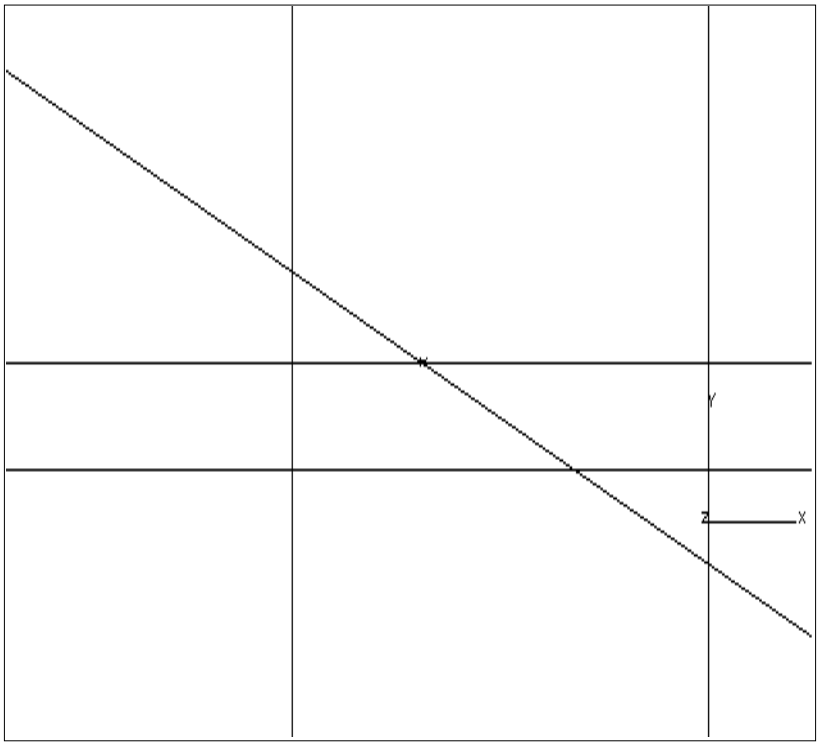


Figure 4-69 The back face

STEP 10

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>

STEP 11

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.

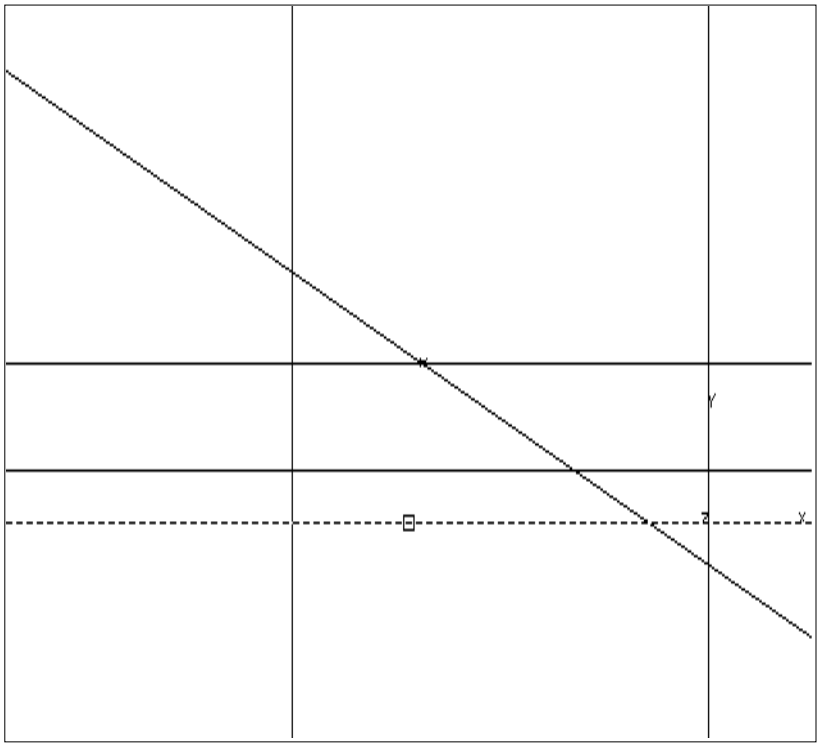
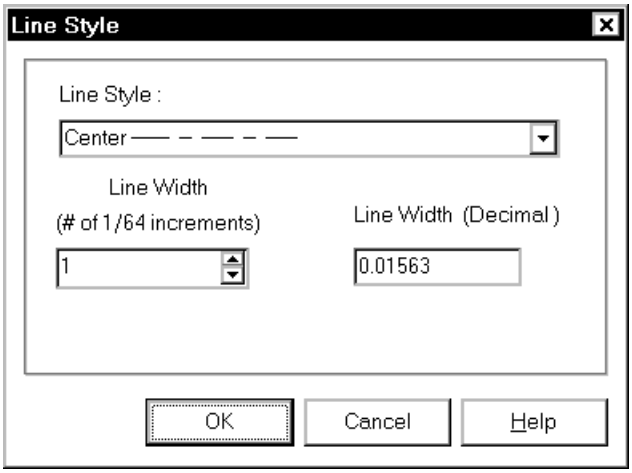


Figure 4-70 Picking the center line

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].



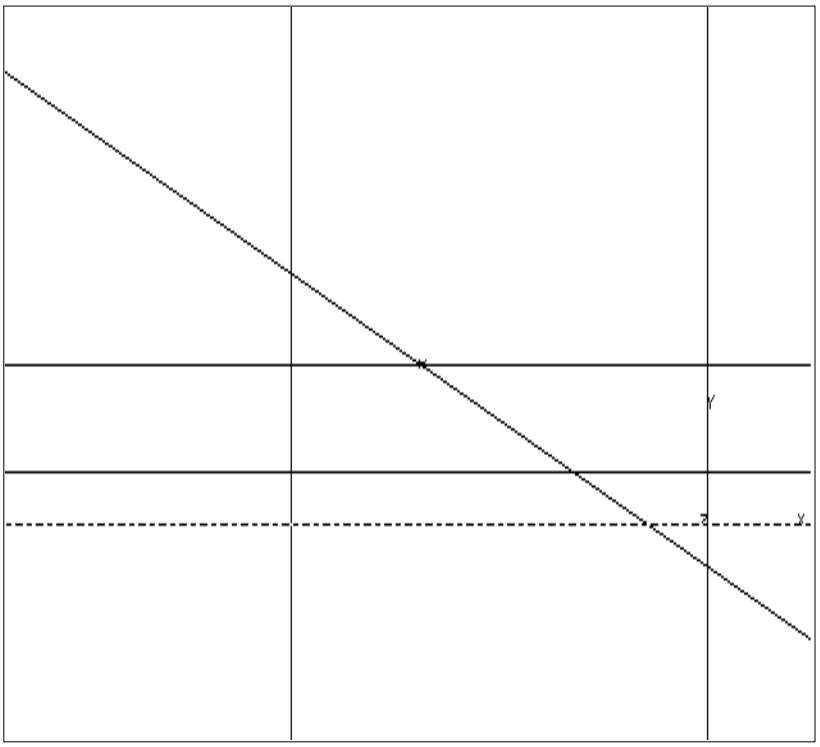


Figure 4-71 The completed center line

STEP 12

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. *NOTE: In the following examples the terms "Cutting Edge" & "Bounding Edge" will be used interchangeably.*

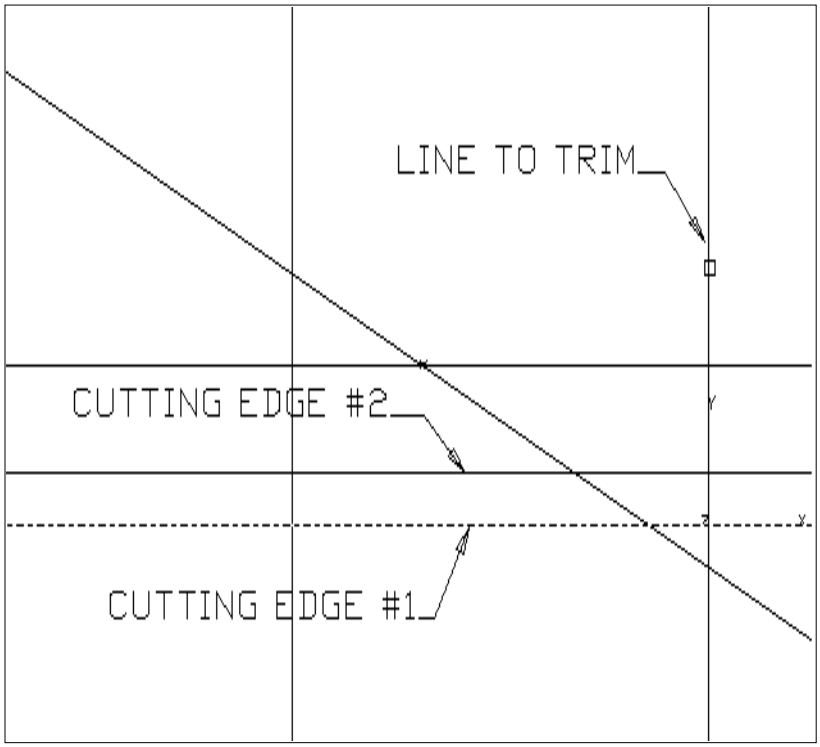


Figure 4-72 Selecting the line to trim

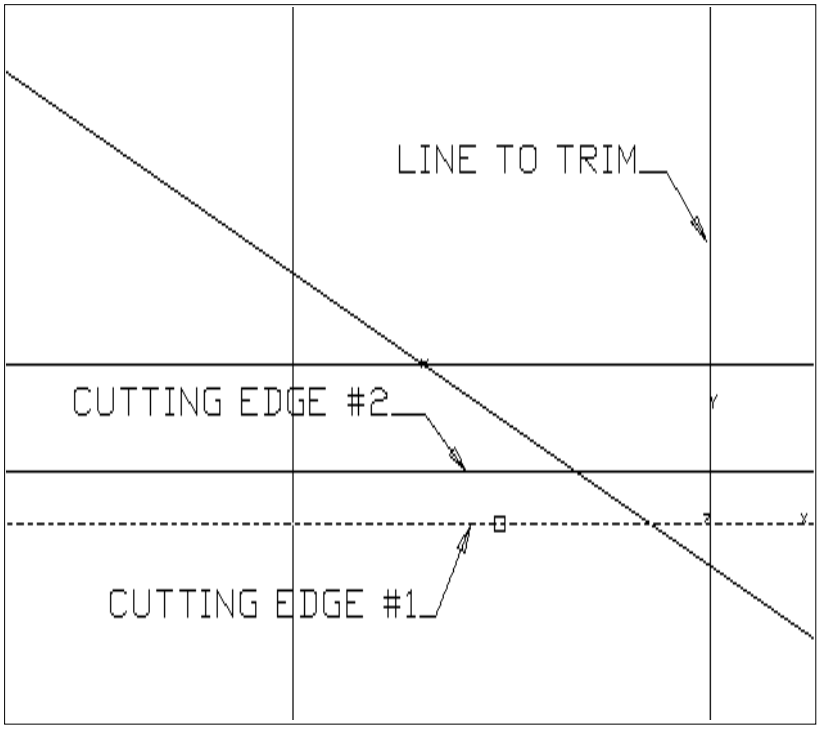


Figure 4-73 Selecting cutting edge #1

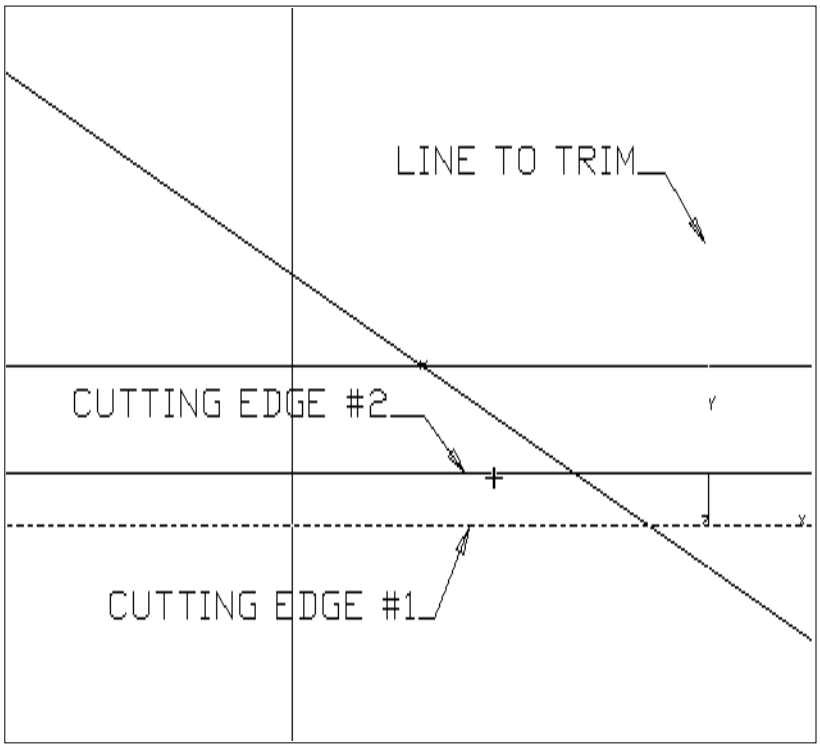


Figure 4-74 Front face after trimming

STEP 13

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.

Notes on Trimming:

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.

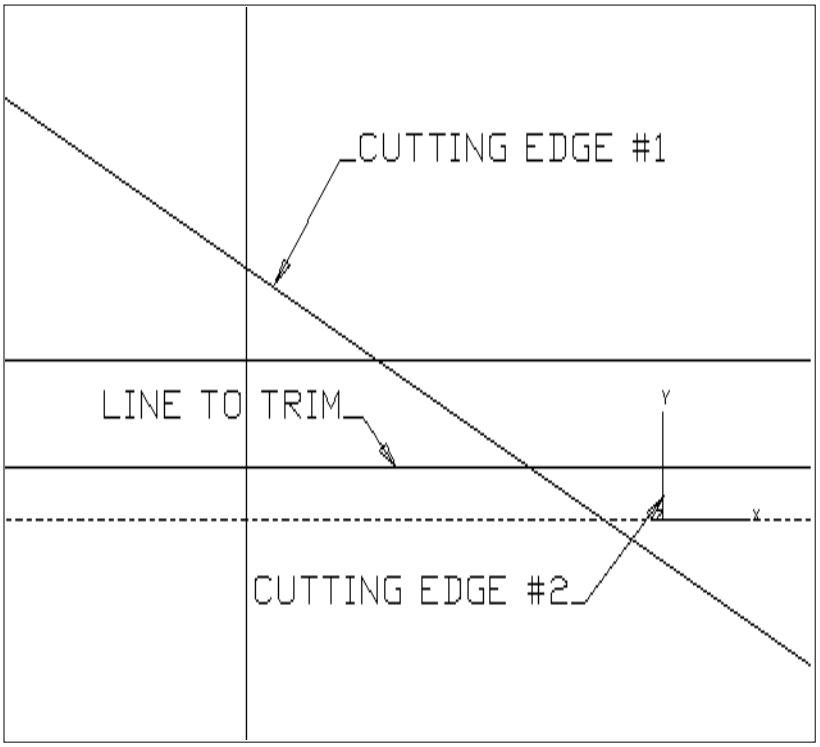


Figure 4-75 Trimming the 1 inch diameter

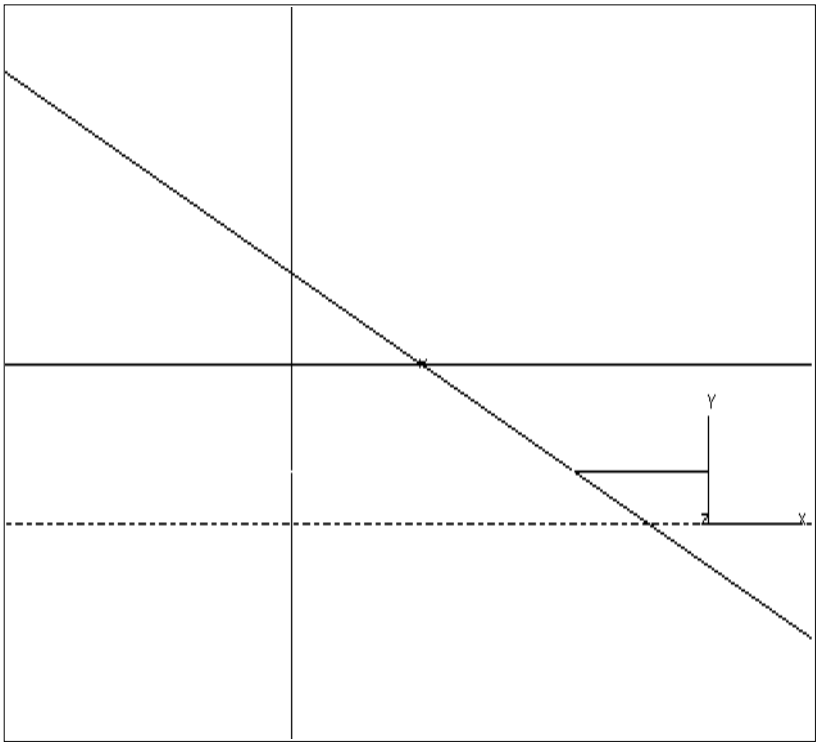


Figure 4-76 After trimming the 1 inch diameter

STEP 14

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.

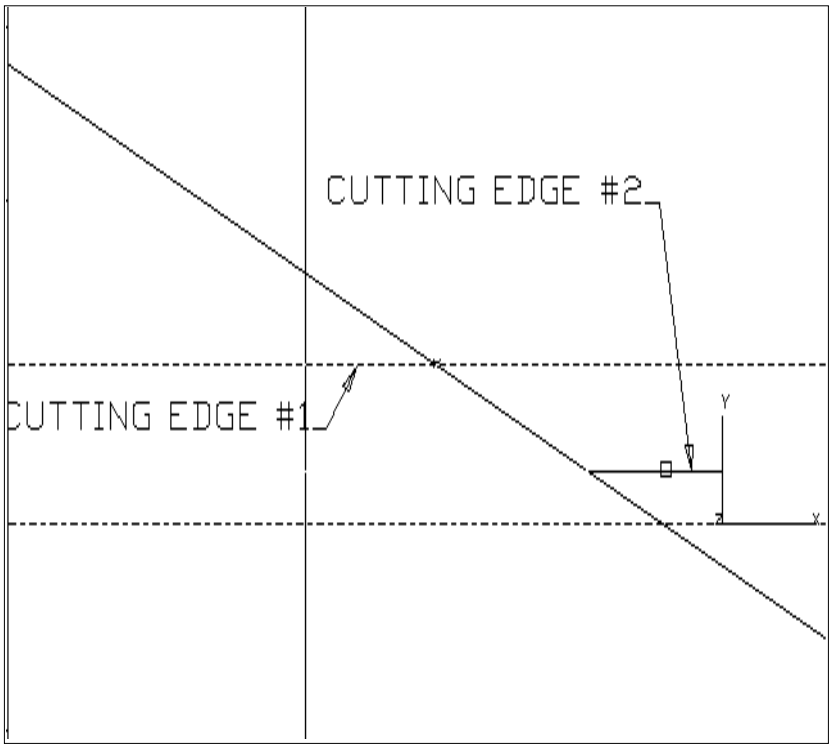


Figure 4-77 The cutting edges for breaking the 30 degree line

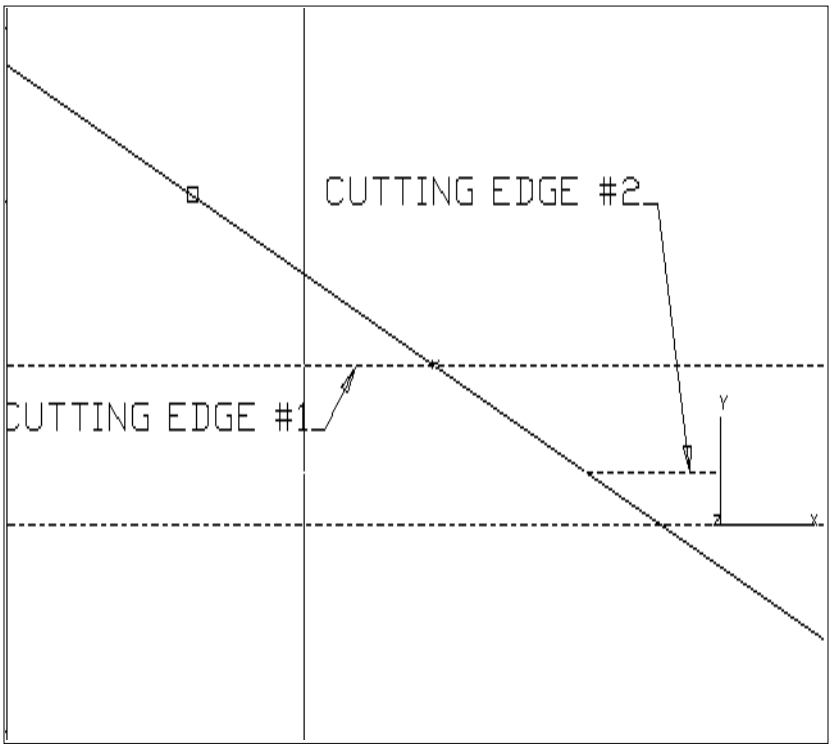


Figure 4-78 Breaking the top half of the 30 degree line

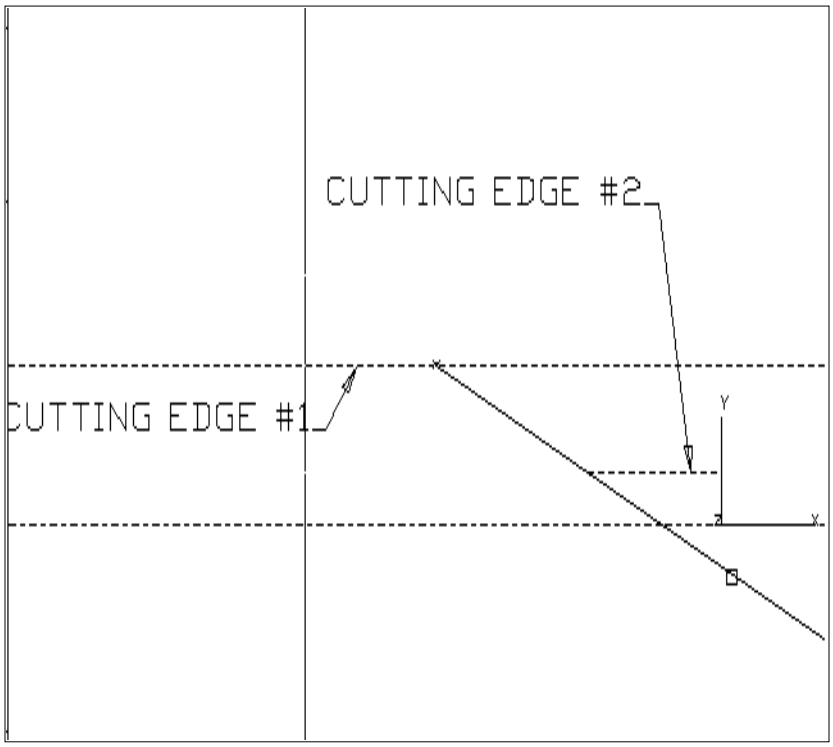


Figure 4-79 Breaking the bottom half of the 30 degree line

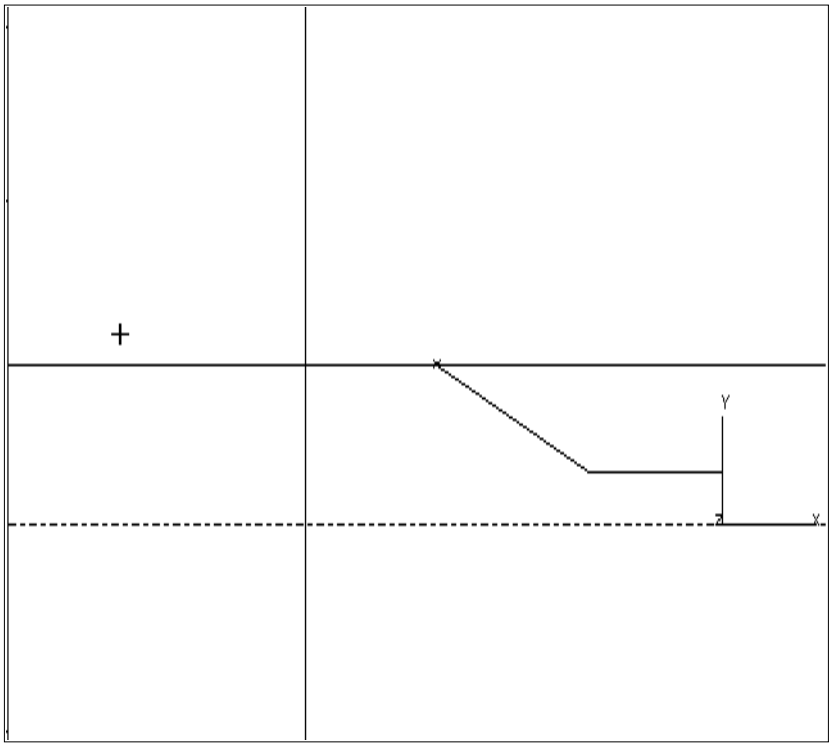


Figure 4-80 The 30 degree line after breaking

STEP 15

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.

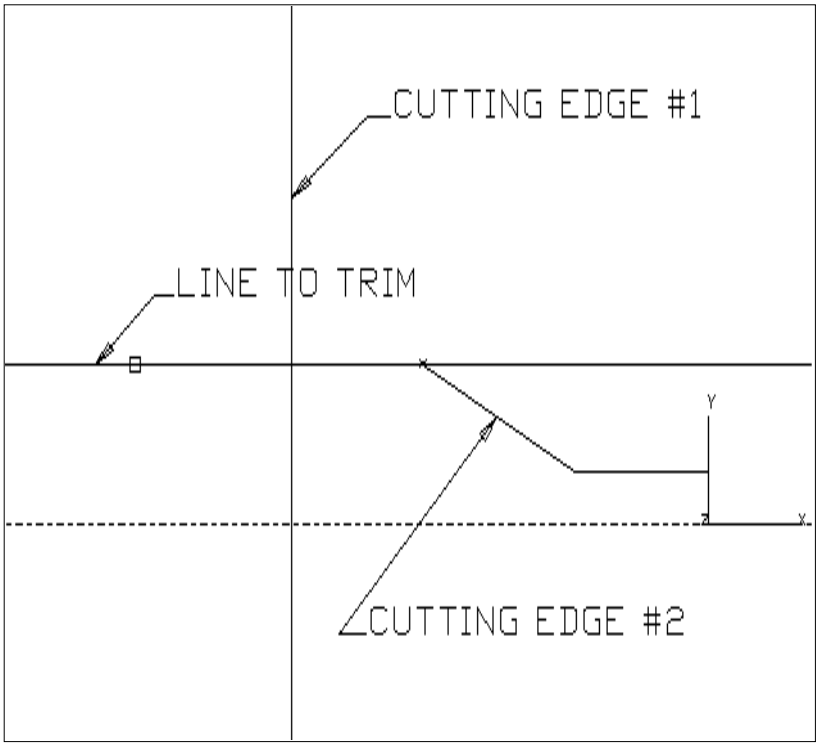


Figure 4-81 Cutting edges, for trimming the 3 inch diameter

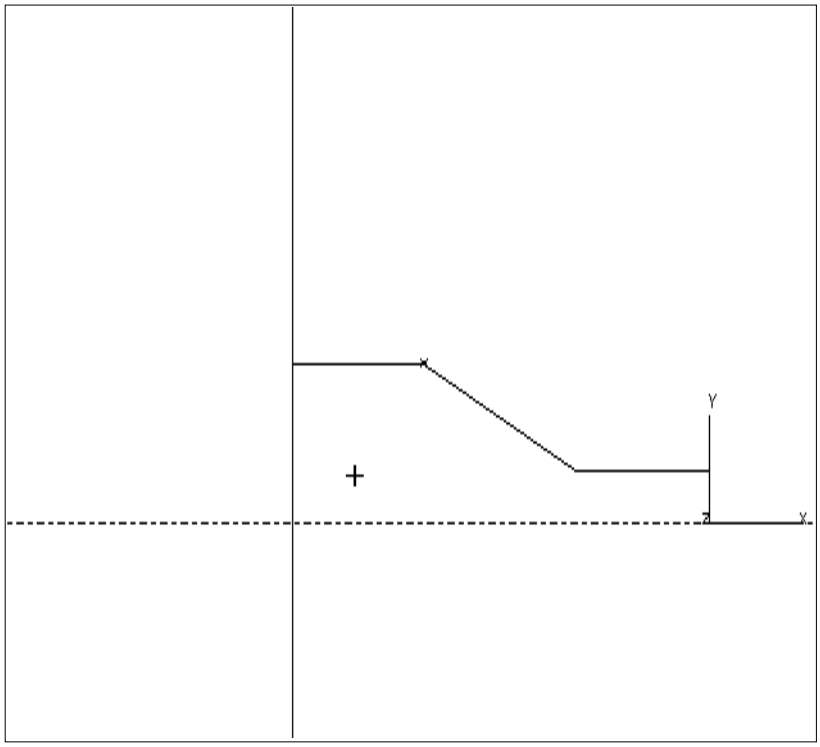


Figure 4-82 The 3 inch diameter after trimming

STEP 16

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.

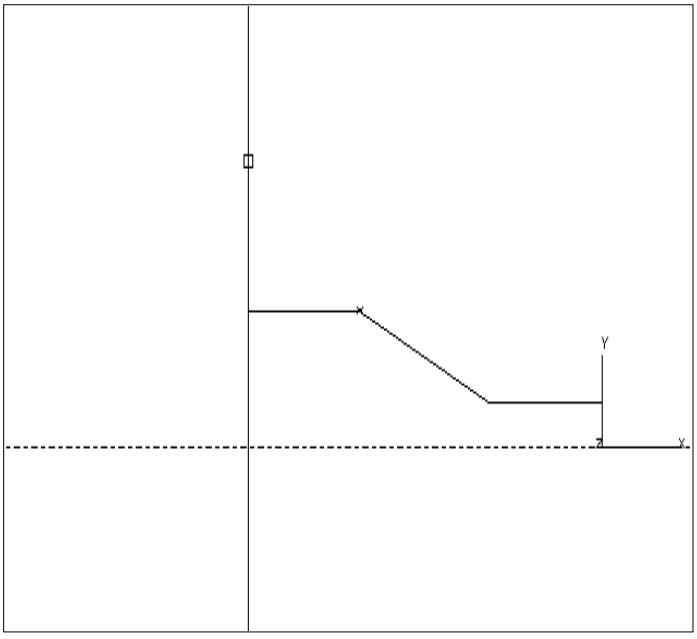
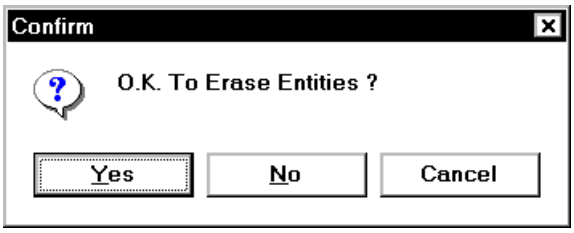


Figure 4-83 Selecting the line to erase

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



STEP 17

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.

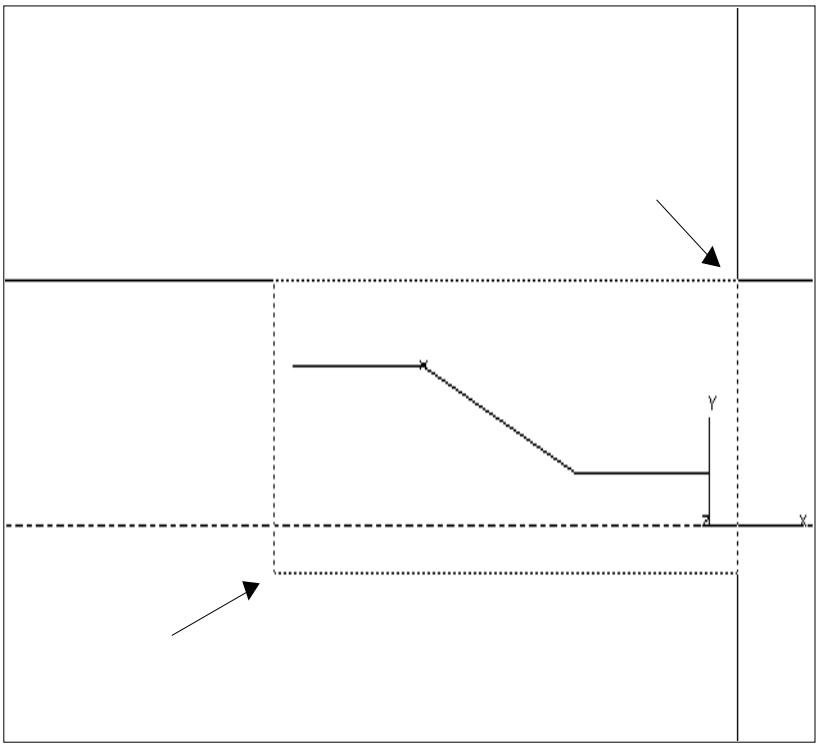


Figure 4-84 The zoom window

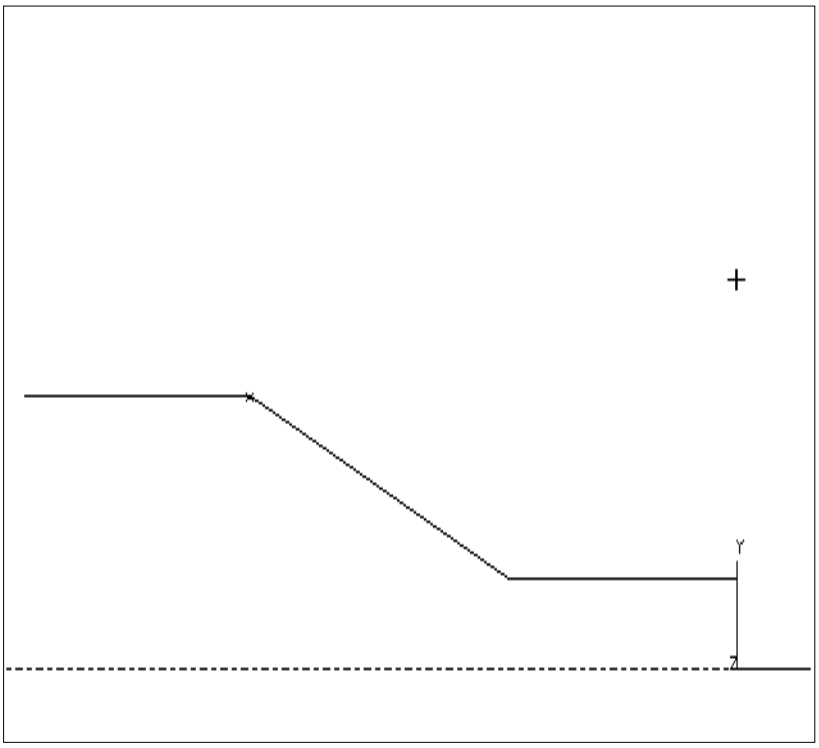


Figure 4-85 After the zoom command

STEP 18

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.

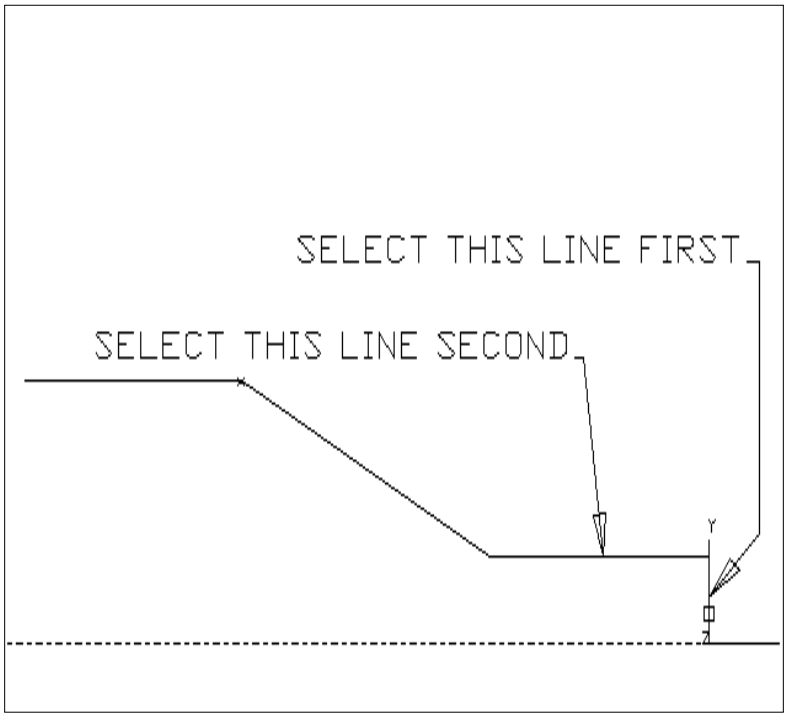


Figure 4-86 Selecting the lines to chamfer

Next you will be asked :

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

The part should now look like figure 4-87.

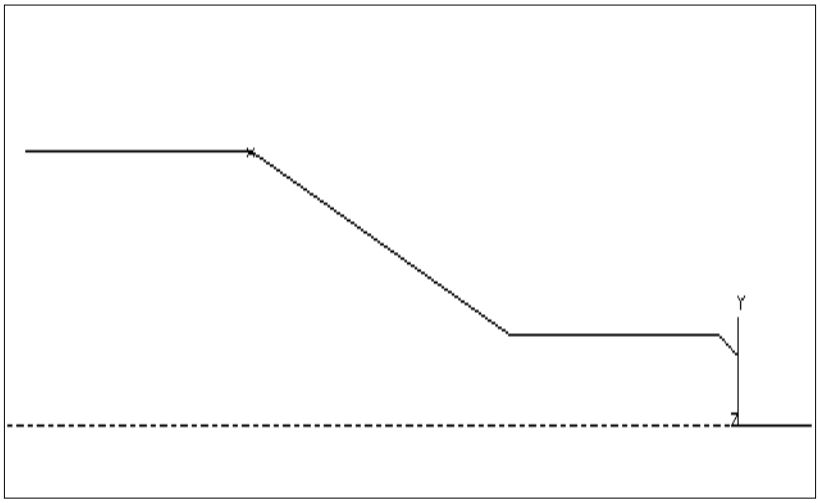


Figure 4-87 The completed chamfer

STEP 19

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>

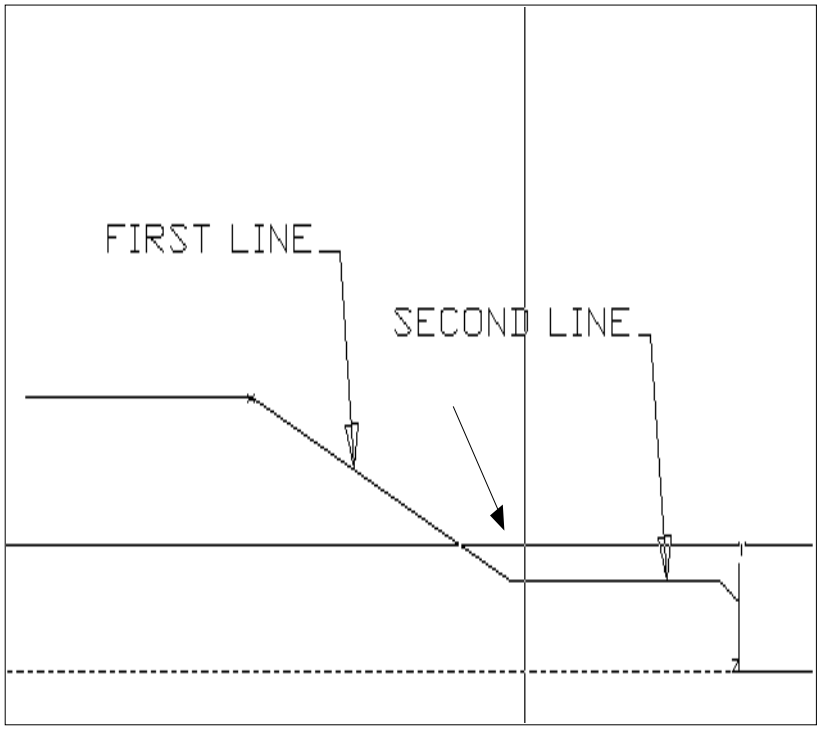


Figure 4-88 Selecting the lines to fillet and indicating the approximate point

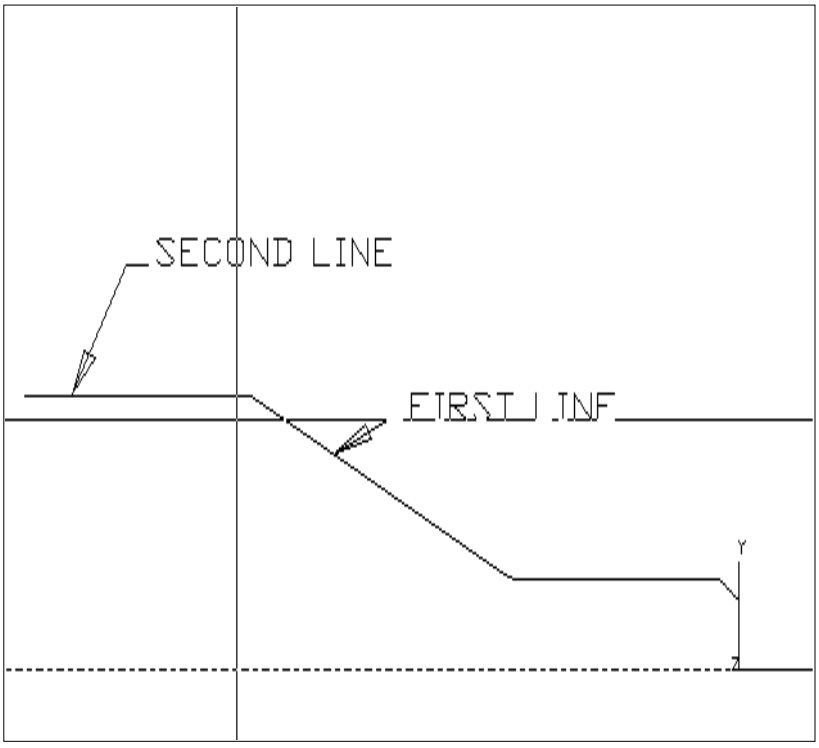


Figure 4-89 Selecting the lines to fillet and indicating the approximate point

STEP 20

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] ? .25<Enter>

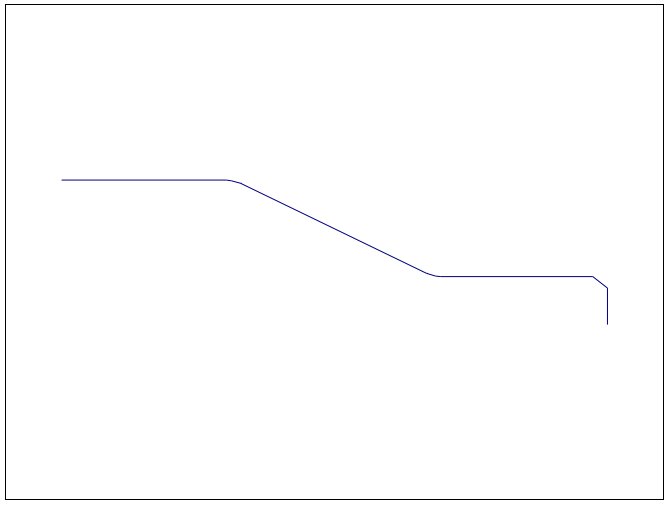
STEP 21

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

STEP 21 A

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>"



STEP 22

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.

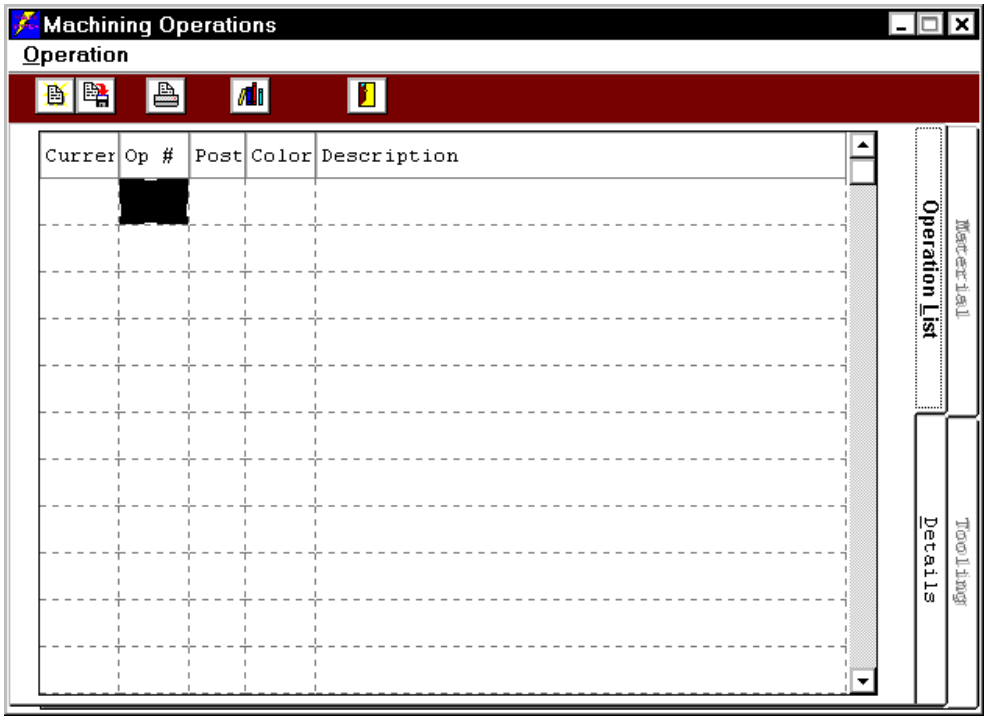


Figure 4-89a (The Machining Operations Manager)

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.

Figure 4-89b (Basic operation information)

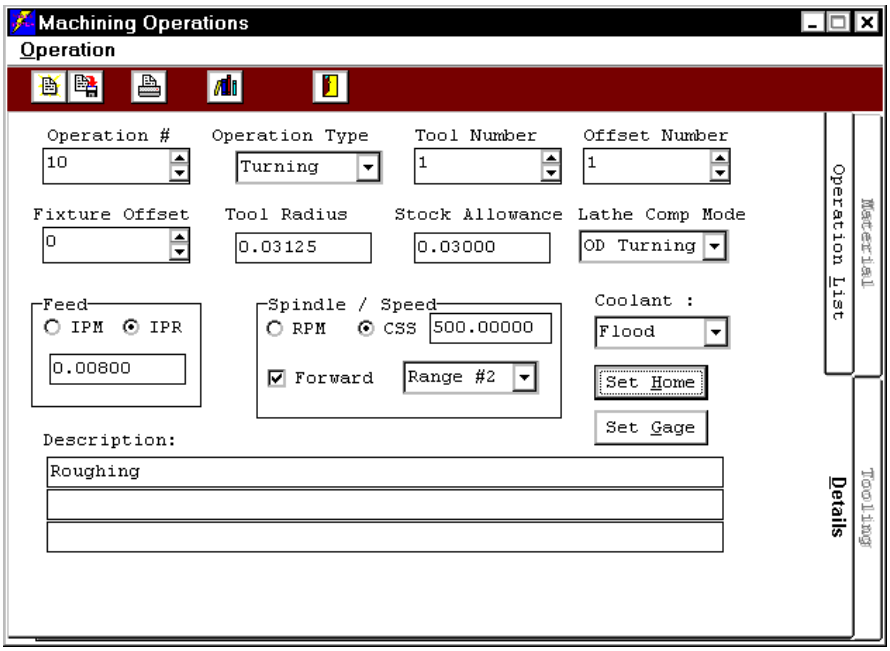


Figure 4-89c (Operation #10 Details)
Figure 4-89d (All operations for the tutorial part)

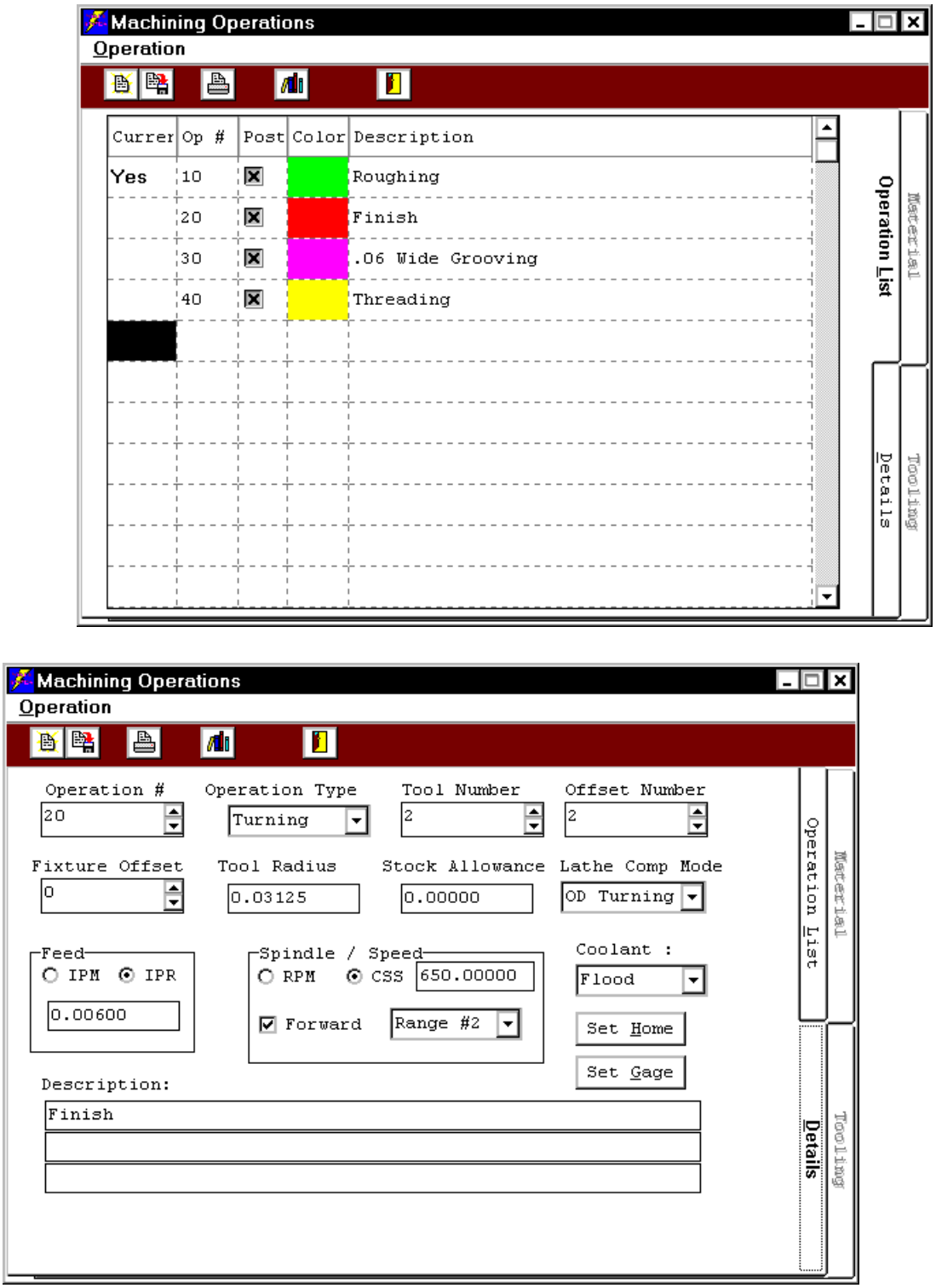
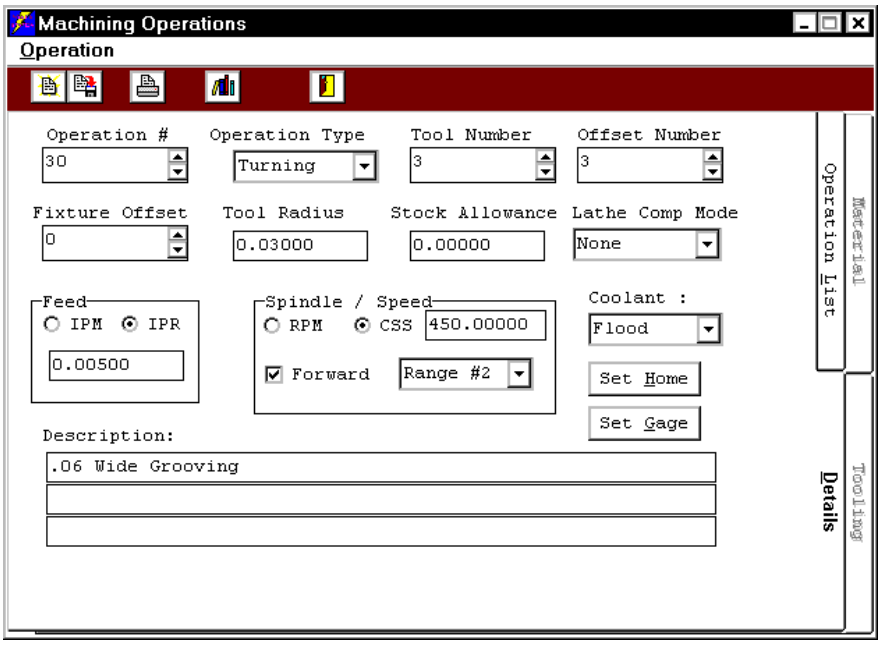


Figure 4-89e (Operation #20 for the tutorial part)

Next define the remaining operations, using the information displayed in



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

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

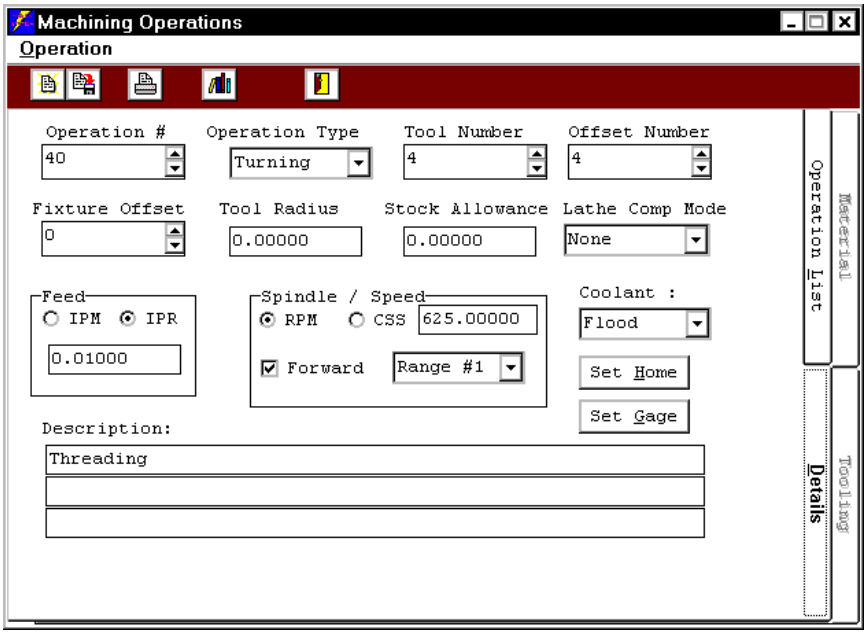


Figure 4-89g (Operation #40 for the tutorial part)

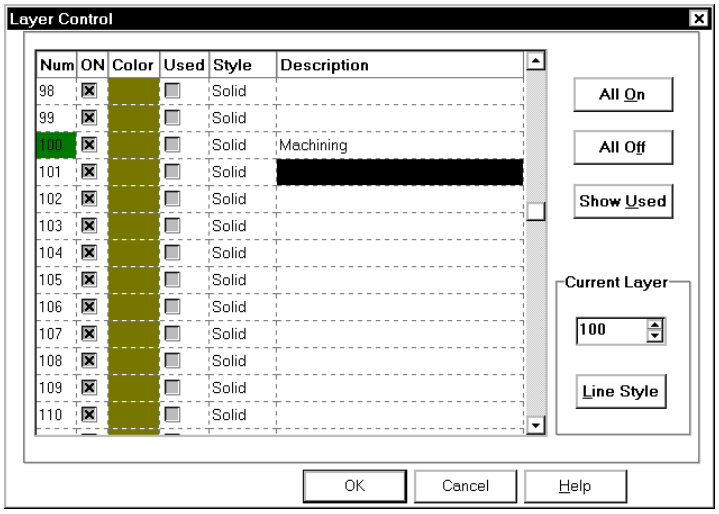
STEP 23

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.

STEP 24

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 **<F9>** key (or select “Display”, “Layer-Control” from the menu), then click on “Current Layer” box and type: 100, 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.

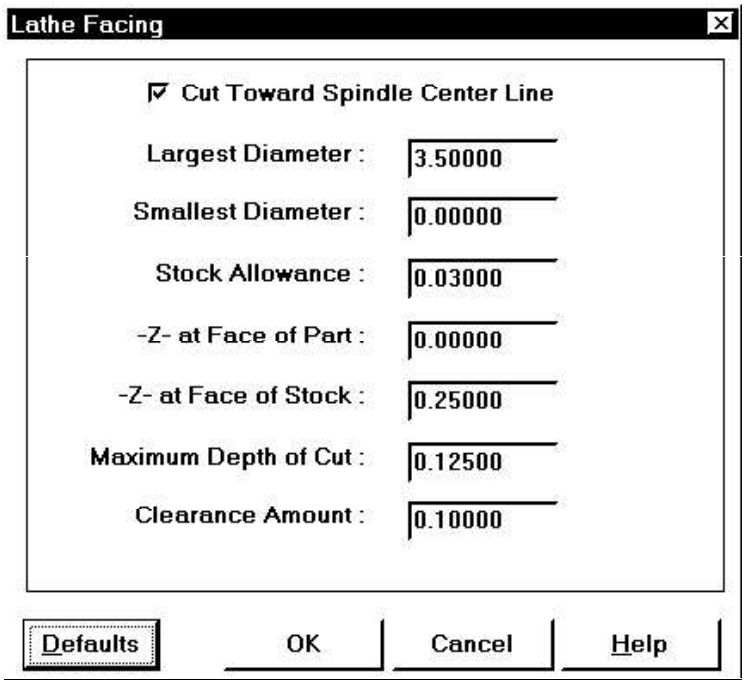


STEP 25

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

STEP 26

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:

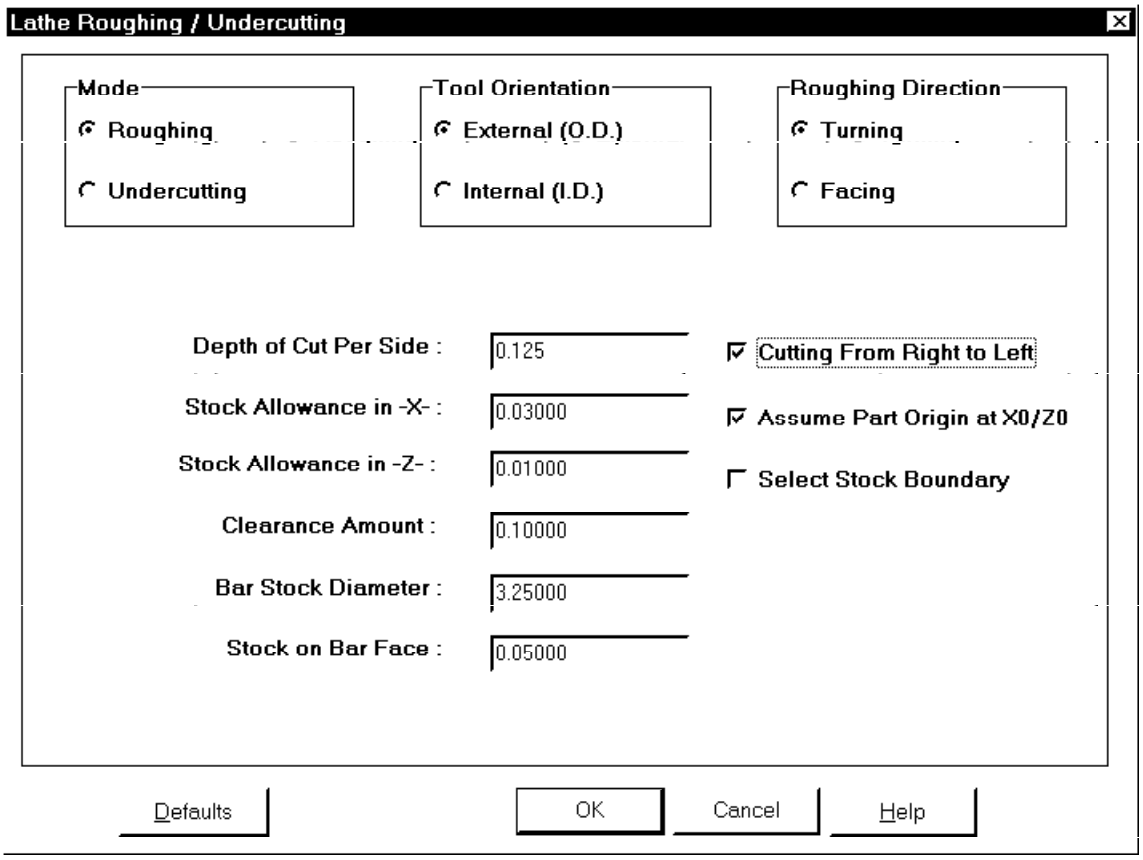


STEP 27

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

STEP 28

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:



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

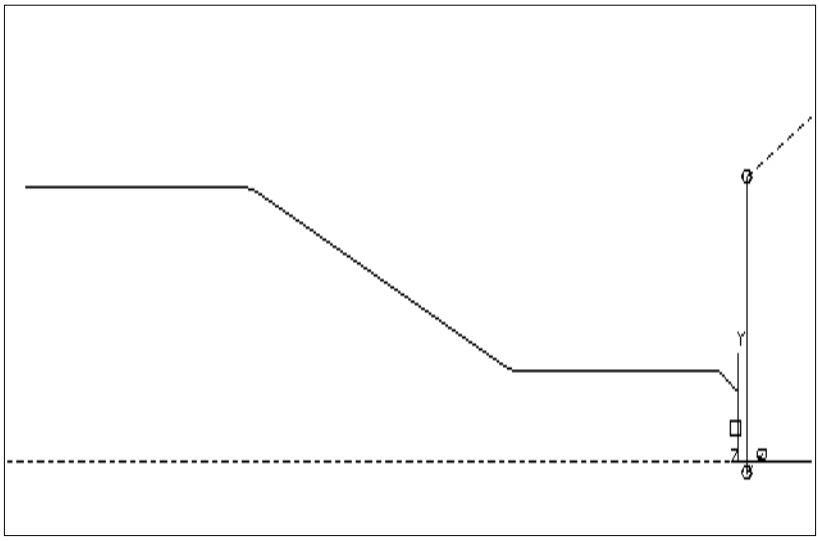
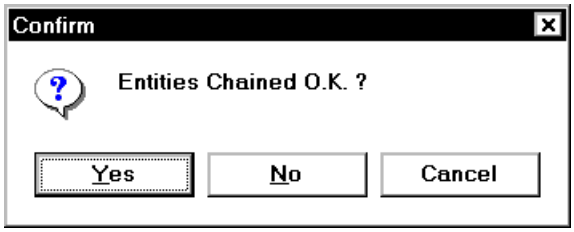


Figure 4-93 Selecting the starting point for roughing

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.

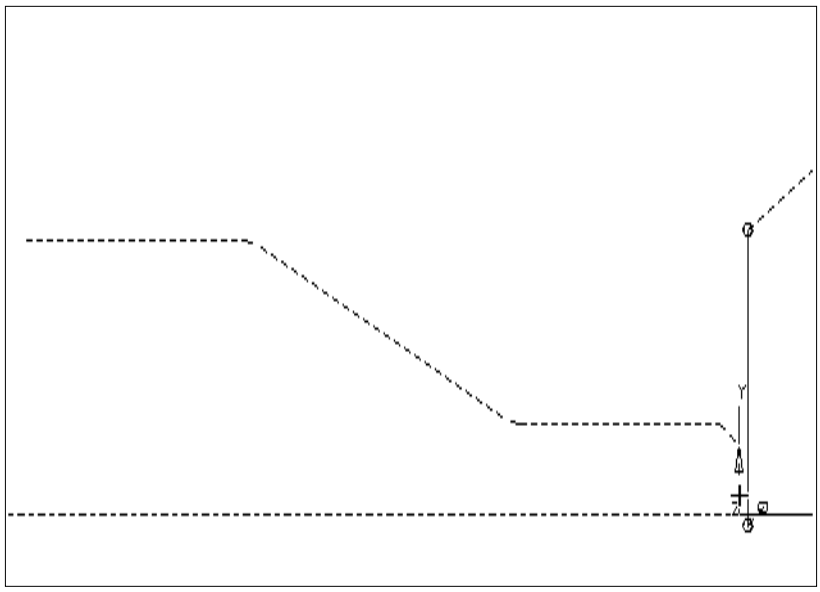


Figure 4-94 Proper chaining

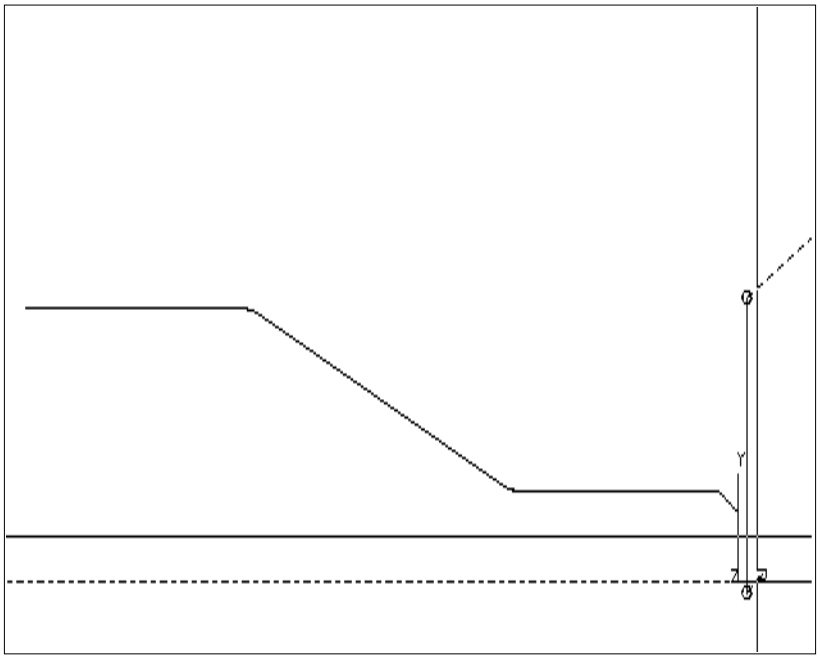


Figure 4-95 Picking the side to offset to

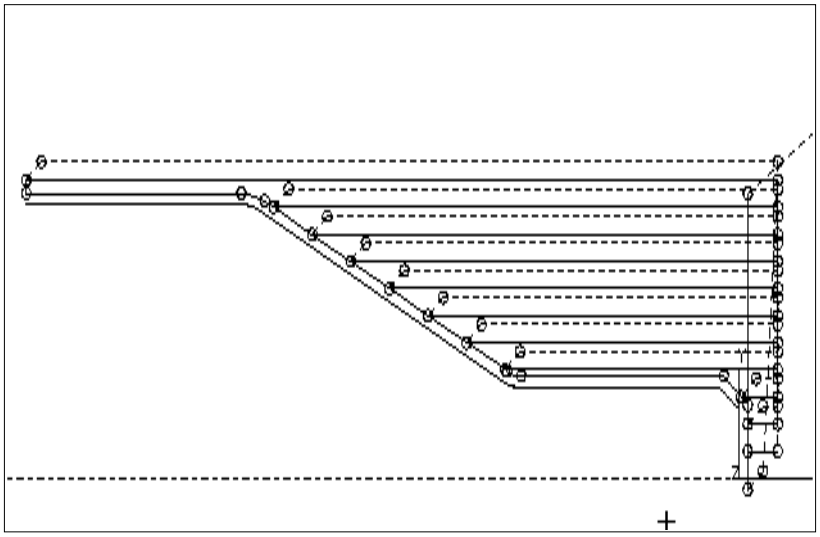


Figure 4-96 Roughing completed

STEP 29

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

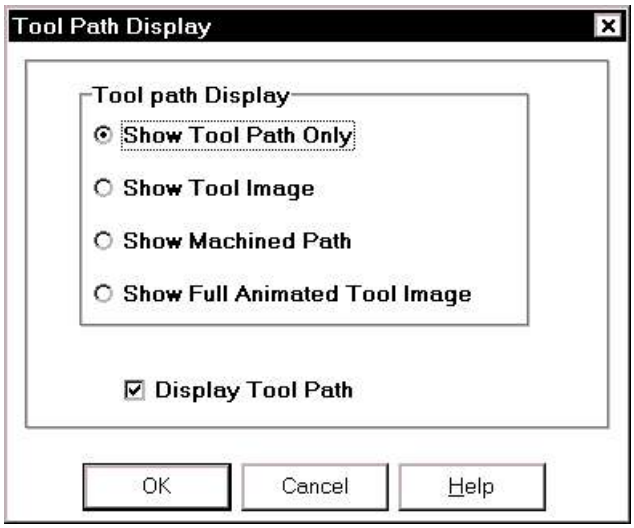
`[X] Rapid Move`

STEP 30

Select operation 20 from the “Machining”, “Operations” Dialog.

STEP 31

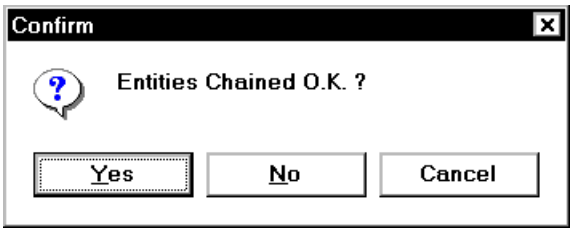
On many slower computers, the display of the tool images (small circles) can be very time consuming. If you want to turn them off, select **"DISPLAY" - "Tool Image"** from the menus.



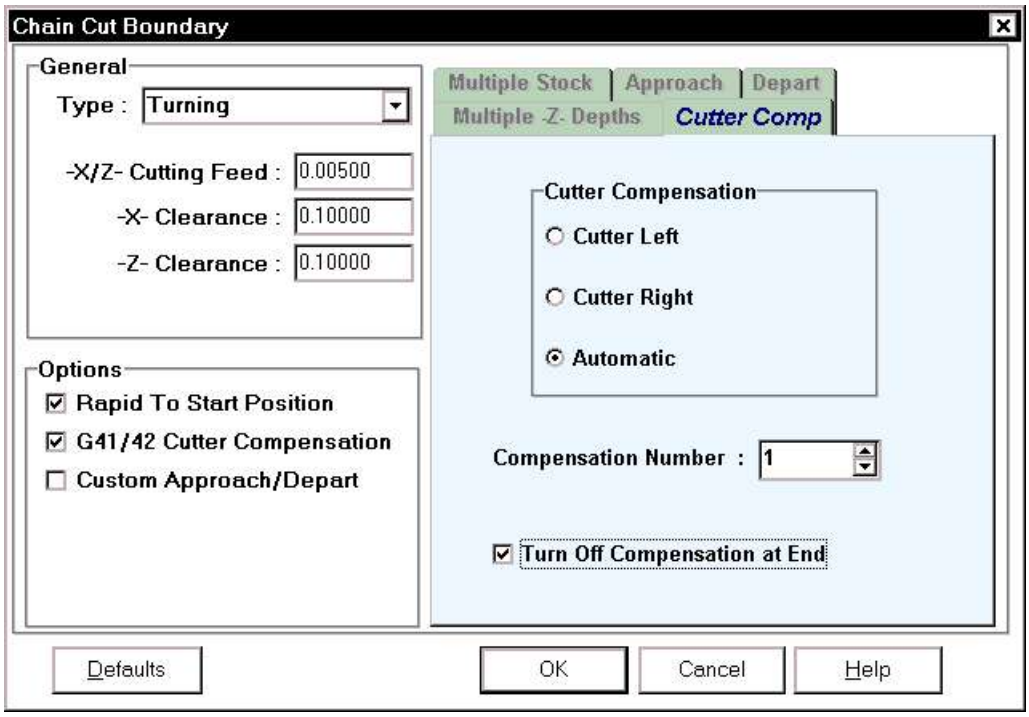
STEP 32

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)



`Pick the Side to Offset to`
(Move the cursor just to the right of the front face)



See figure 4-98

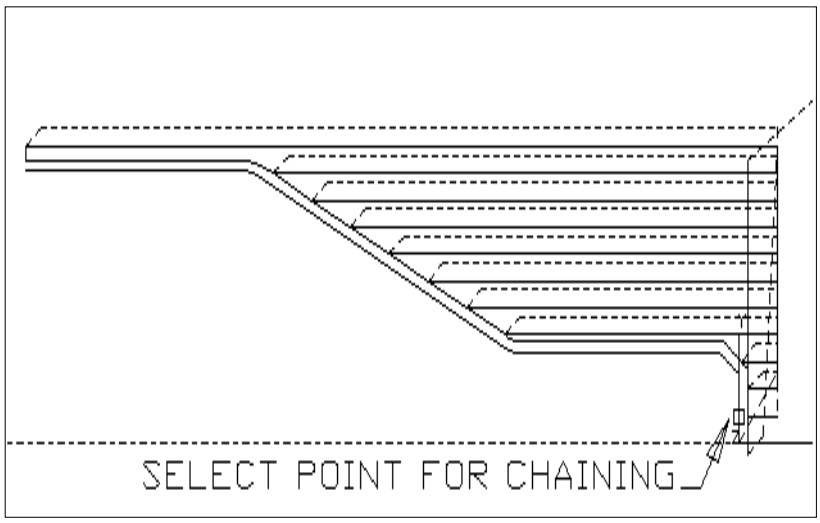


Figure 4-97 Selecting the start line for chaining

STEP 33

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

`[X] Rapid Move ?`

STEP 34

Select operation 30 from the “Machining”, “Operations” Dialog.

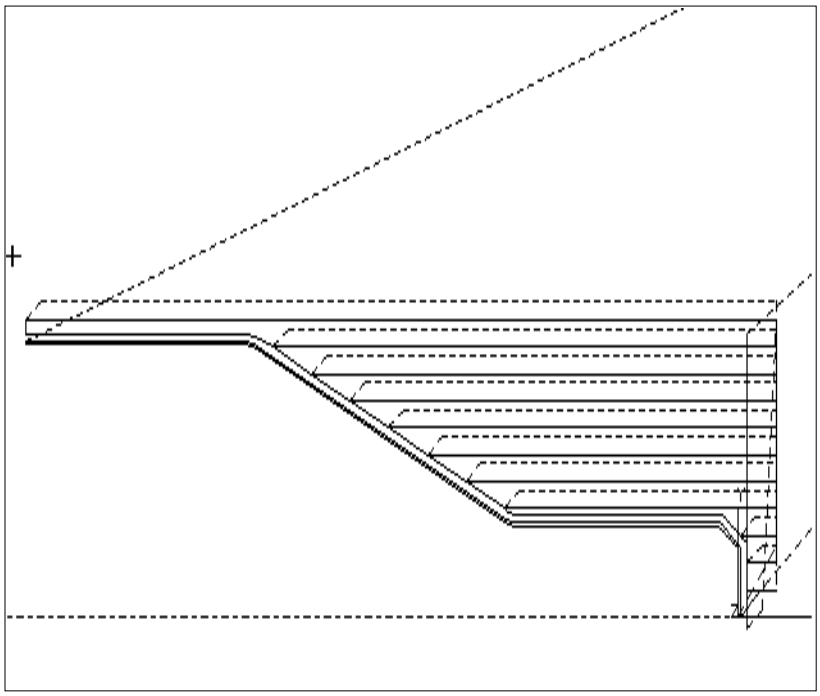
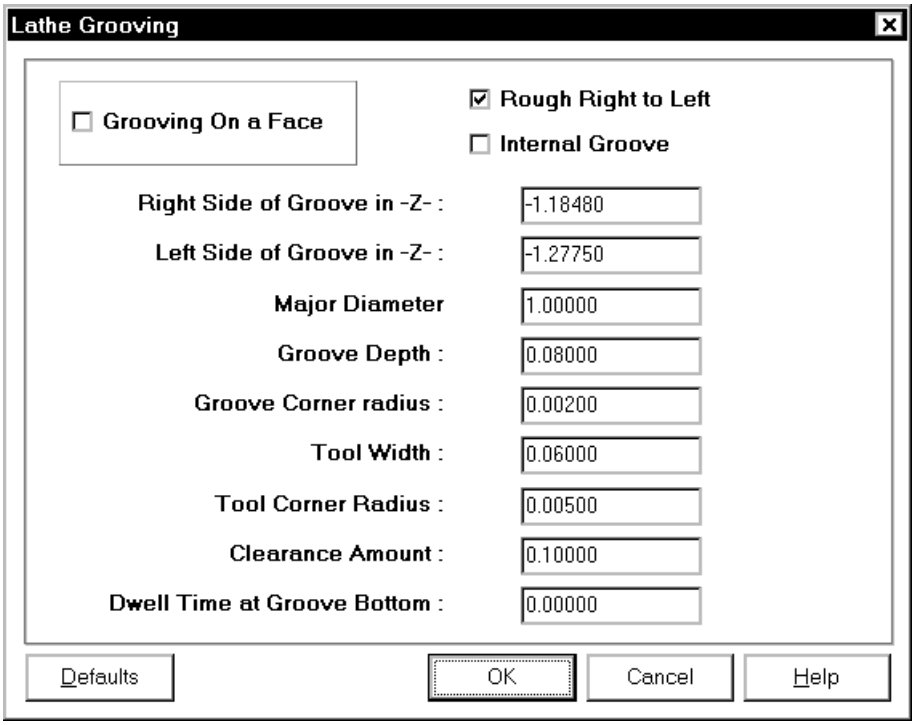


Figure 4-98 The Finish pass complete

STEP 35

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- : -4.75+3.5625
Left Side of Groove in -Z- : -4.75+3.5625-.09

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 ?

STEP 36

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

[X] Rapid Move ?

STEP 37

Select operation 40 from the “Machining”, “Operations” Dialog.

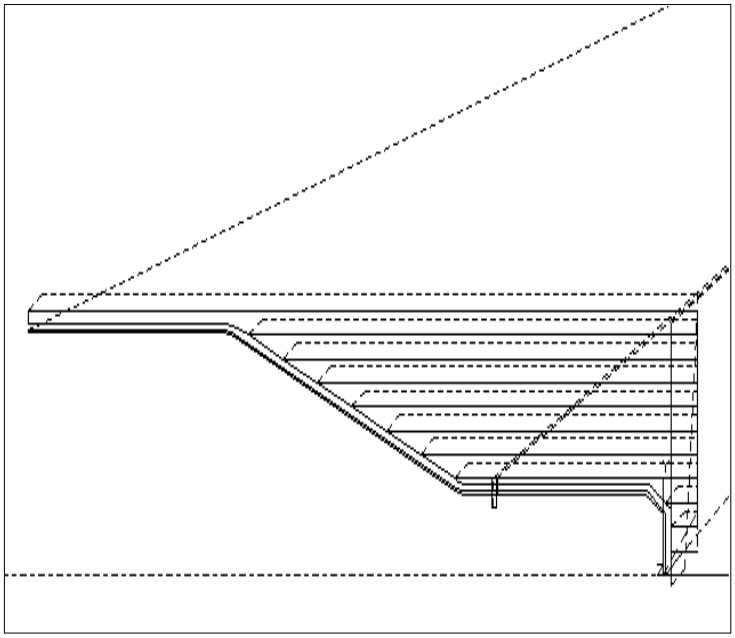
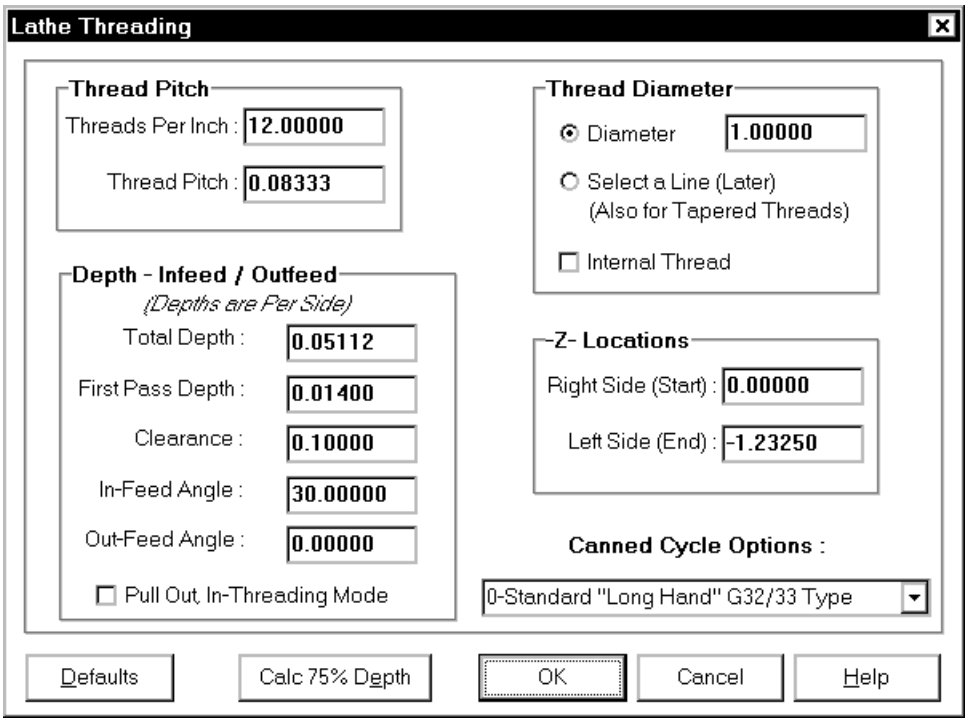


Figure 4-99 The grooving complete

STEP 38



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.

STEP 39

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

```
[X] Rapid Move ?
```

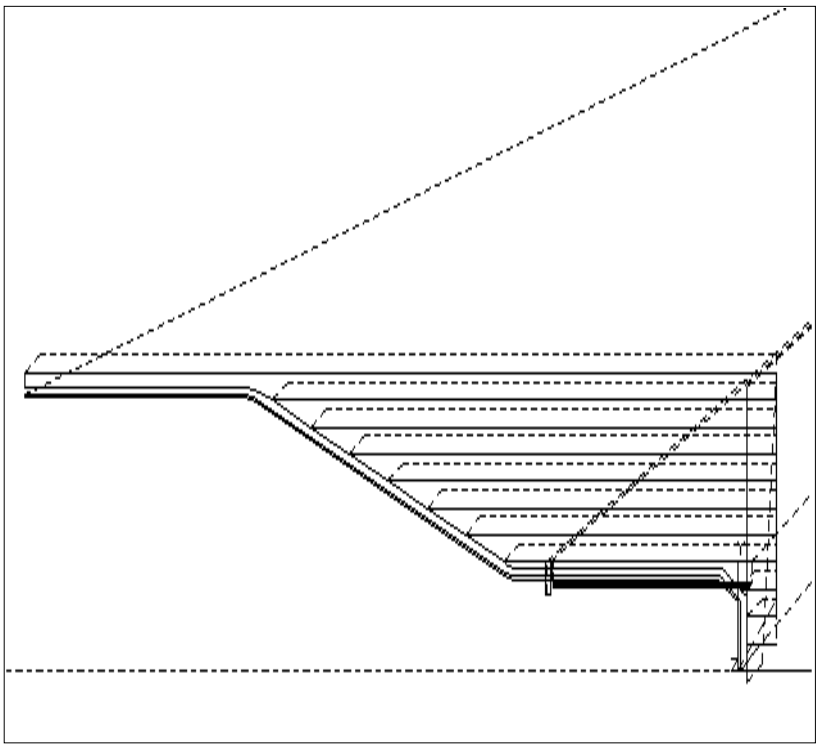


Figure 4-100 The Threading Complete

STEP 40

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.

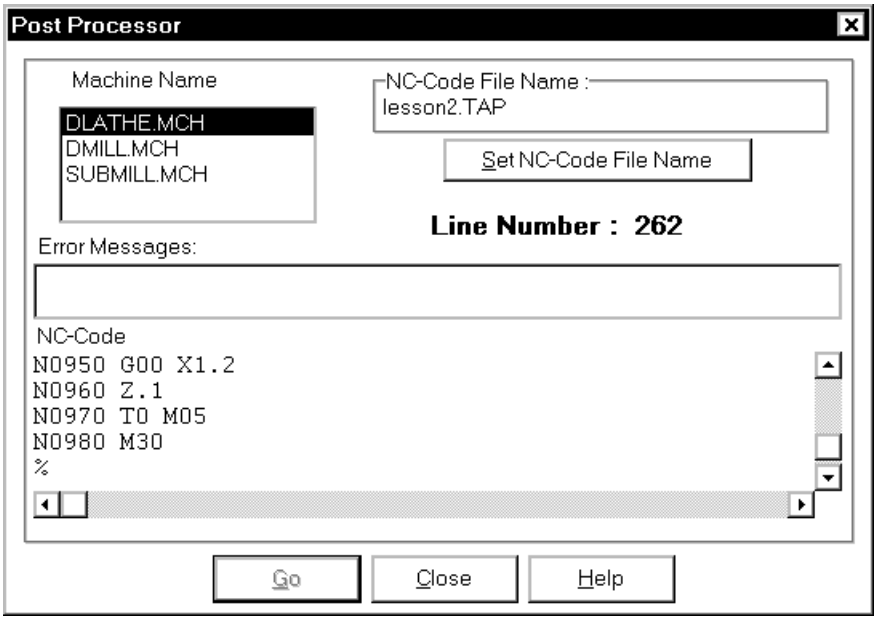
STEP 41

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.

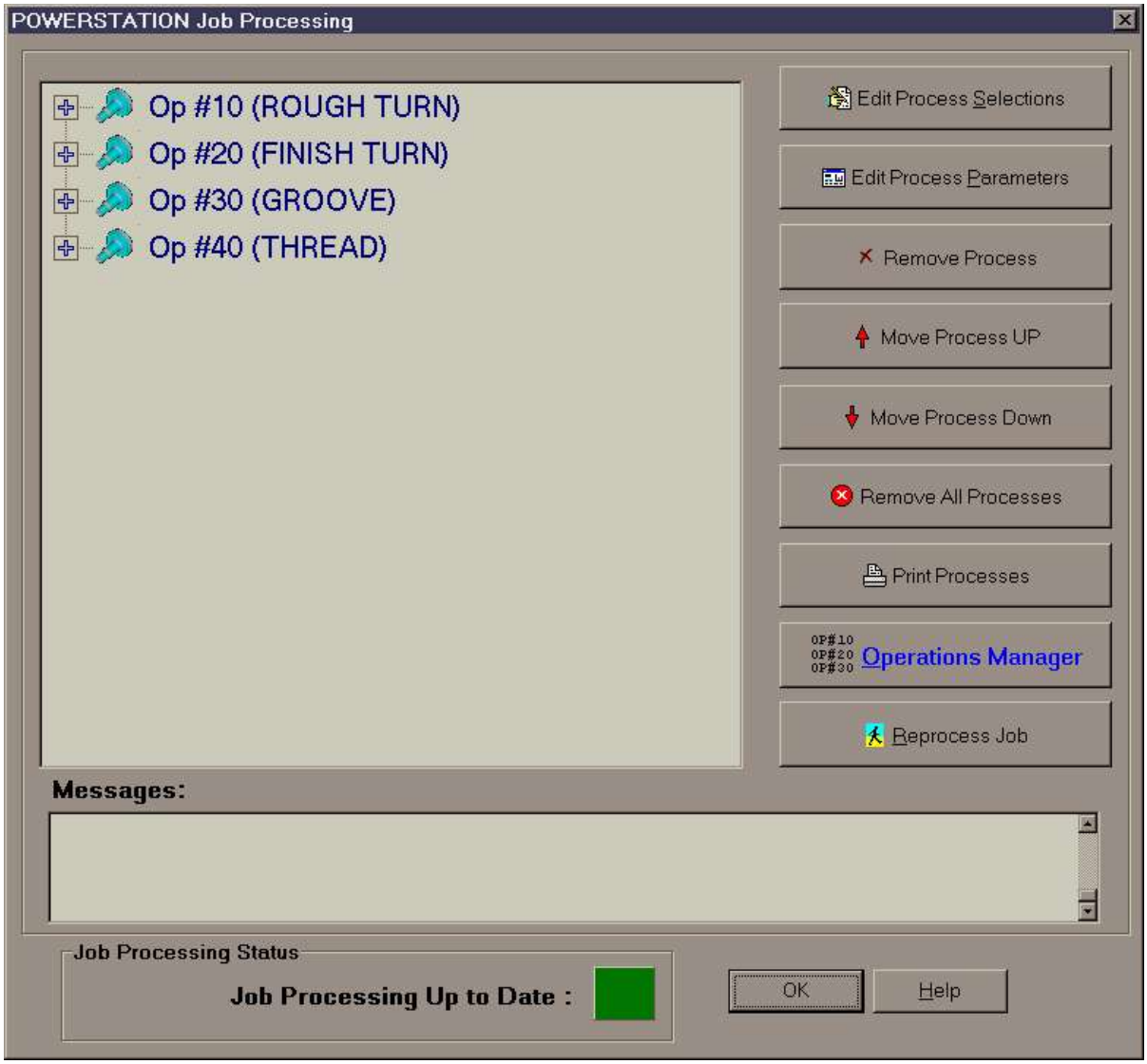


Now that the tutorial part is finished, lets see how the new “Job Processing” feature can help with making changes.

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

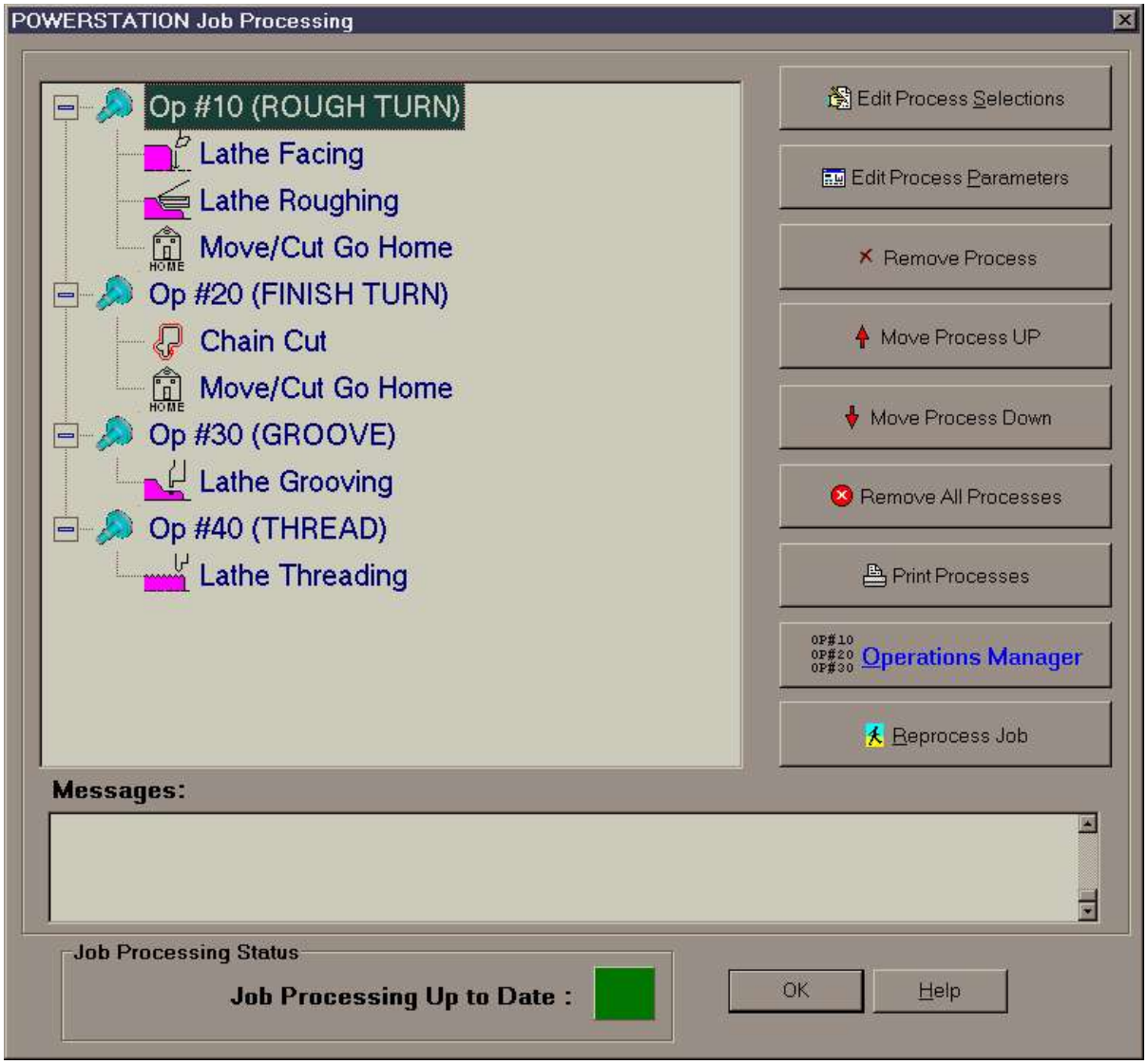
Step 42

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



Step 43

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:



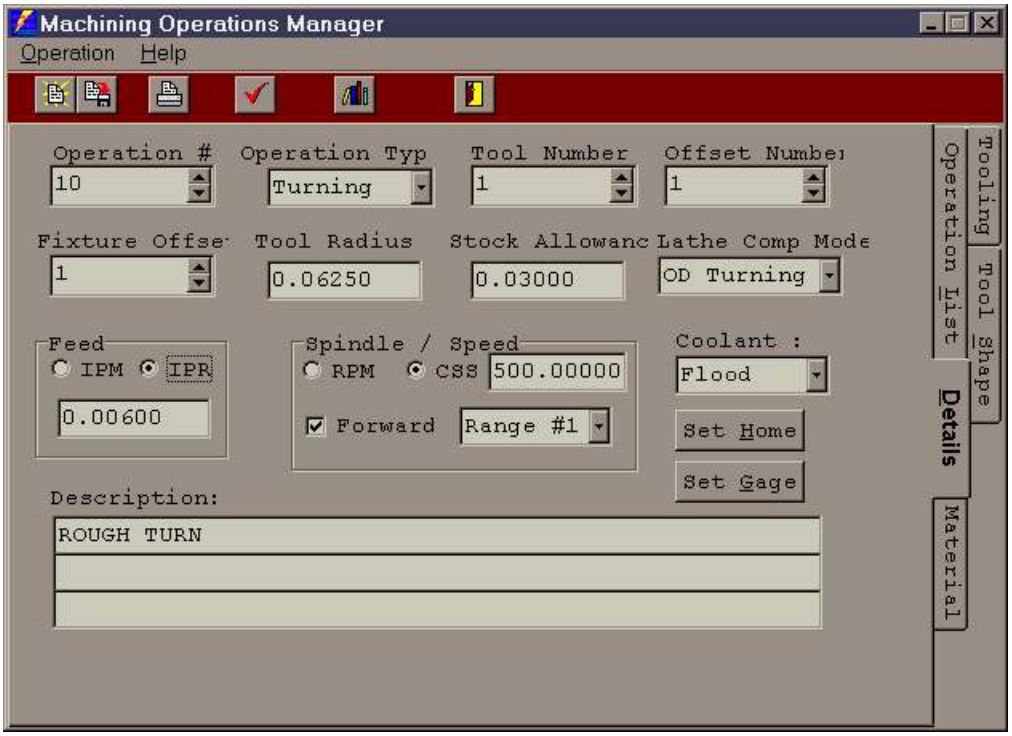
Step 44

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

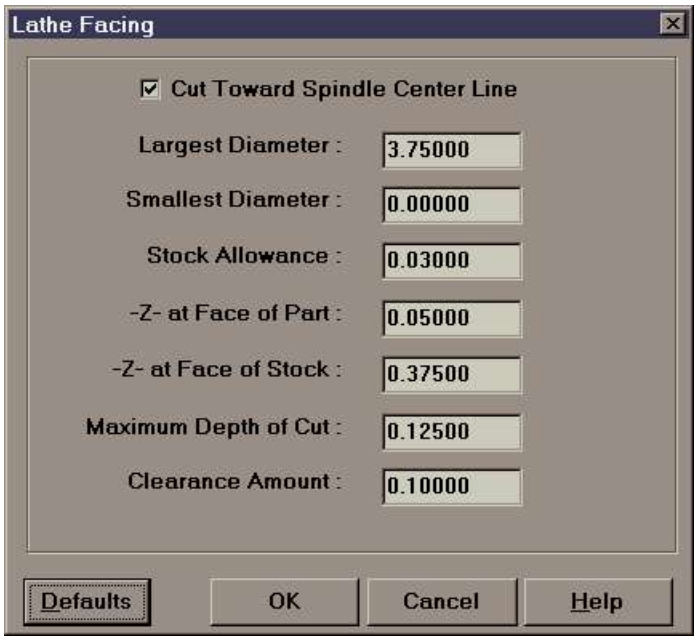
Step 45

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.



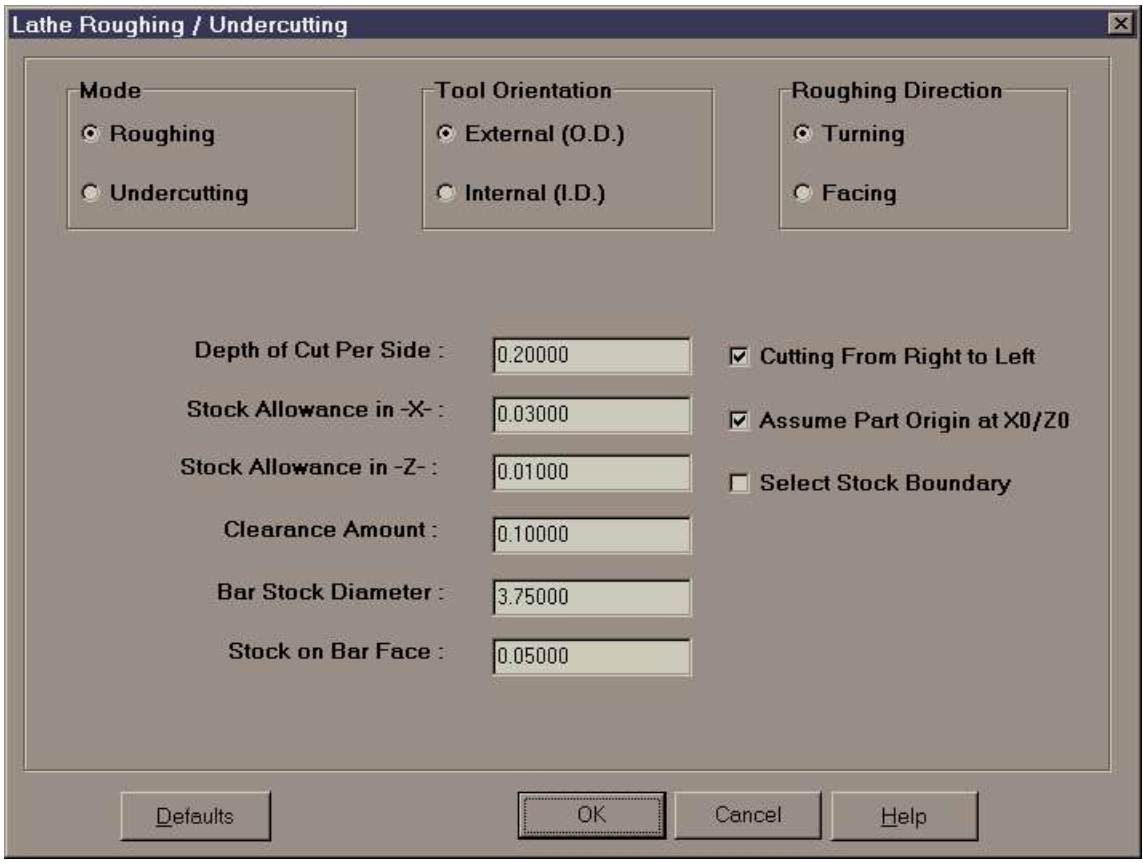
Step 46

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.



Step 47

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.



Step 48

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

Step 49

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