POWERSTATION 2015

Tutorial Manual

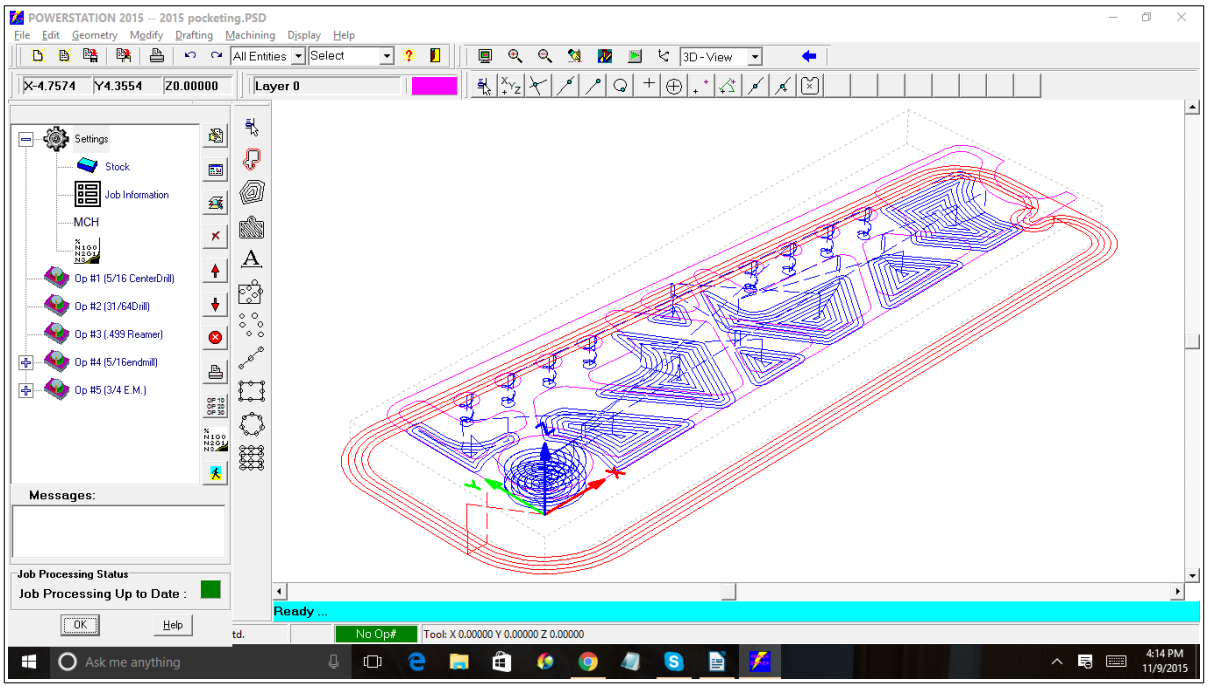


TABLE OF CONTENTS

INTRODUCTION

POWERSTATION represents a new direction in CAD/CAM software by breaking some long standing traditions:

- CAD/CAM is much more expensive then general software like word processors, spread sheets and data bases.
- The software has to come from several sources, since no single vendor can make both good CAD and CAM.
- To run CAD/CAM you MUST purchase the latest and most expensive computer hardware.
- Only the "Big shops" need and can afford the cost of CAD/CAM.

We at MICROCIMM felt it was time to shatter these myths:

- With the inexpensive, high quality tools available to a software developer these days, the time required to produce a quality program has been reduced substantially.
- MICROCIMM has brought together experts from both the CAD and CAM fields. And given them a common goal.
- While others say "You need better hardware to run our software" we say "We will make our software better, to run on your hardware."
- At our low prices, even the "One man shop" can benefit from high quality CAD/CAM.

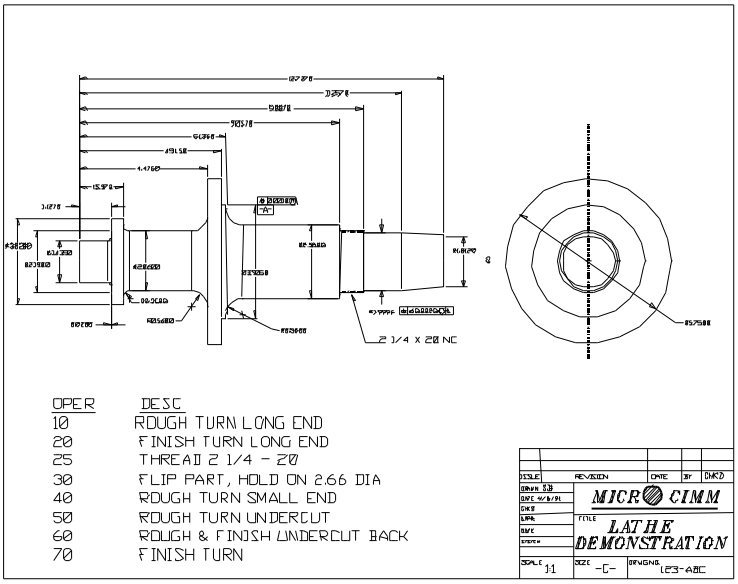
TECHNICAL INFORMATION ON POWERSTATION

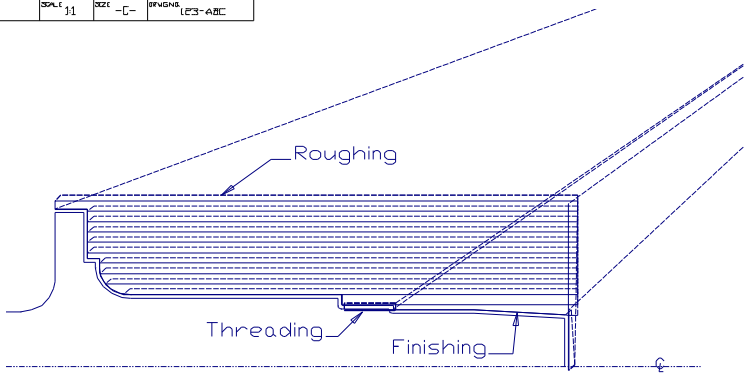
POWERSTATION consists of over one Million lines of OBJECT PASCAL. The compiler used is Borland internationals Delphi 5.0/Delphi 7.0, and Embarcadero® Technologies, Inc Delphi 2010.

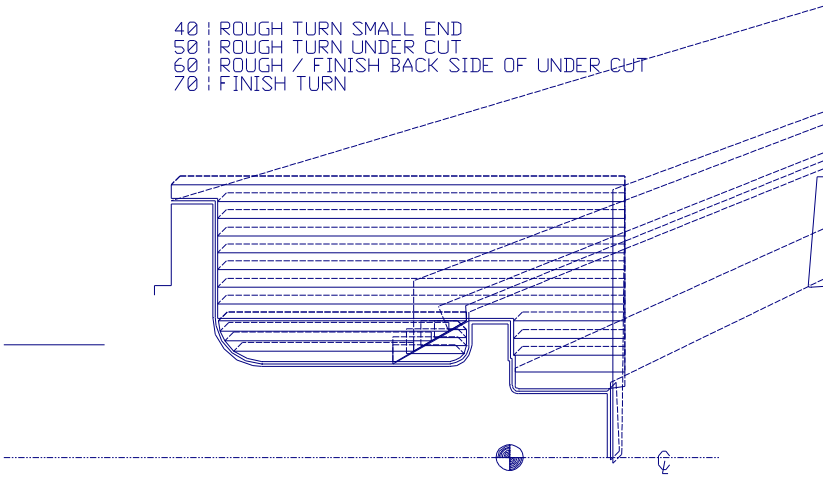
Examples:

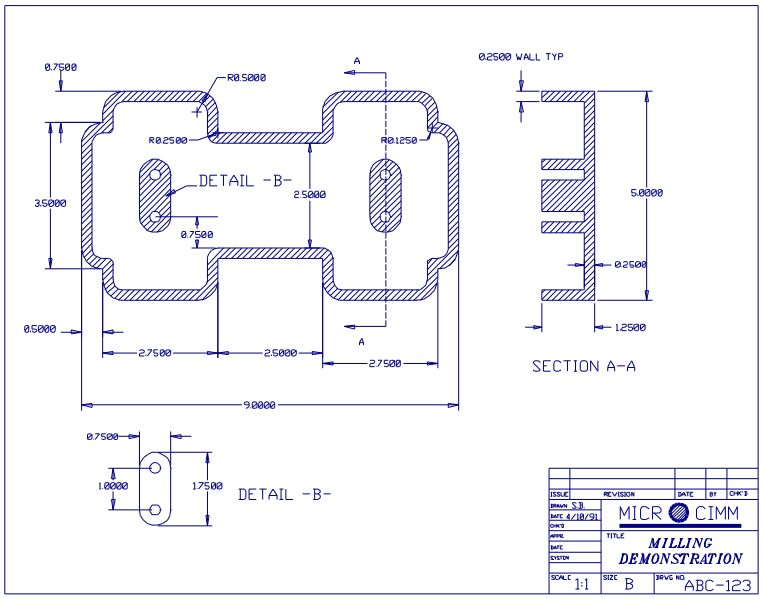
The following pages contain some sample POWERSTATION drawings.

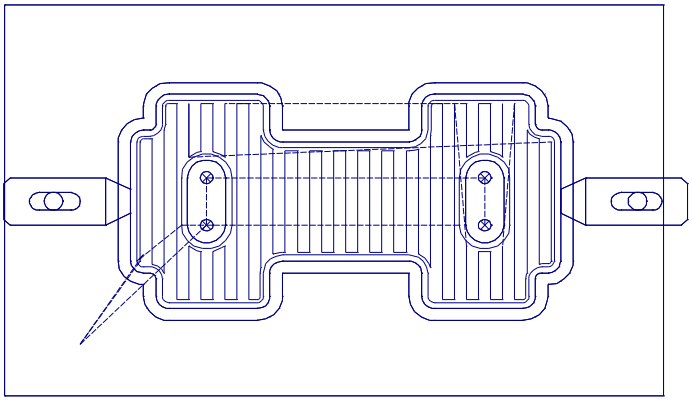
6 POWERSTATION








0.7500
R0.5000
A
0.2500 WALL TYP
R0.2500
R0.1250
DETAIL -B-
2.5000
5.0000
3.5000
0.7500
0.2500
0.5000
2.7500
2.5000
2.7500
A
1.2500
SECTION A-A
9.0000
0.7500
1.0000
1.7500
DETAIL -B-
ISSUE
REVISION
DATE
BY
CHK'D
DRAWN S.B.
DATE 4/18/91
CHK'D
APPR.
DATE
SYSTEM
MICR CIMM
TITLE
MILLING DEMONSTRATION
SCALE 1:1
SIZE B
DRVG NO ABC-123



OBTAINING TECHNICAL SUPPORT

Come visit us at WWW.MICROCIMM.COM for the latest support information

Email us at SUPPORT@MICROCIMM.COM

Please include:

- Company name
- Your name
- Software product name & version number (Start the program, and select “Help” then “About” for the version name/number)
- Complete detailed description of the problem
- Any files that are involved, such as:
  - Powerstation drawing file (*.PSD)
  - Powerstation Job file (*.PSJ)
  - NC-Code file (*.TAP),(*.NC)
  - DXF file (*.DXF)

For additional information on support services, please see the brochure that came in your software envelope.

Trademark and Copyright Acknowledgments

HARDWARE REQUIREMENTS

POWERSTATION 2015 was designed from the ground up to support the latest 32/64 bit versions of WINDOWS.

Basically the requirements are that of the installed operating system.

The following are some “practical” minimums:

CPU : Intel I3 or higher

Memory : 4 GB

Video : Graphics board and Display Adapter that supports 1024 X 768 resolution or higher.

Operating System: WINDOWS XP / 7 / 8 / 10

INSTALLATION AND SETUP

In order to begin using POWERSTATION it must first be installed on your computer.

MICROSOFT WINDOWS must be installed and running before POWERSTATION for WINDOWS can be installed.

Setup

Insert the POWERSTATION CD disk in the drive. After a few minutes you will be prompted for the default directory for installation*. The default setting of "C:\PS" should be fine for most all situations. If another directory or drive is desired, type in the full drive / path name at this time.

At this time POWERSTATION will automatically install all of its files, and create the proper start menu items for launching POWERSTATION and its associated programs. The default is under the group called "MICROCIMM".

* If the setup does not begin automatically, start an explorer window by holding the Windows key and pressing the letter "E". Double click on the CD drives icon, and finally double click on the file "SETUP.EXE"

BASICS

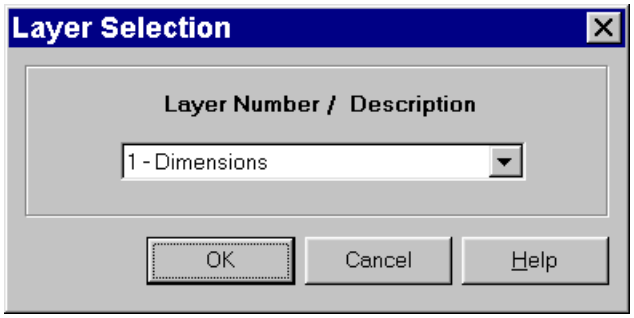
Entity Selection Menu

During the operation of POWERSTATION, frequently you will be asked to select the desired entity or entities (point, line, arc, etc.). For easy identification selected entities are changed from their current color and line style to a white color and a dashed line style. Generally you will be presented with the entity selection menu:

Command	"Hot Key"
All Entities	A
By-Layer	B
By-Type	T
Chain	C
Single	S
Window	W
Last	L
Remove	R
Z-Level	Z
Arc-Radius	D
Line-Length	N
By-Color	Y
Outside	O

ALL -- Selects every entity in the current drawing.

BY-LAYER -- Prompts you to select a layer from a drop down list, and selects all entities on the specified layer.



BY-TYPE -- Asks you to select a sample entity (Point, Line, Arc, Etc.) the automatically selects all entities of the selected type.

SINGLE -- Used to select individual entities, one at a time. The standard mouse cursor (+) will change to a small box shape. When picking an entity, position the box so that part or all of the desired entity is contained inside the box, then press the Left mouse button. After selecting all of the desired entities, press the right mouse button.

CHAIN -- The chain selection is used for rapid selection of connected entities that form a uninterrupted path. Note that lines and arcs are the only type of entities that can be chained.

Chaining works in the following manner:

1. First you will be asked to select the starting line or arc.The chaining will start at the opposite end of the entity you select. If an entity is found who's beginning or ending point is near (less then 0.0005 away (this number can be changed in the "FILE-PROJECT" section)) from the end of the current entity, it will also be selected. A small arrow will be drawn to show which direction the chaining went in.

2. This procedure will continue until no entities are found near the end of the current entity.

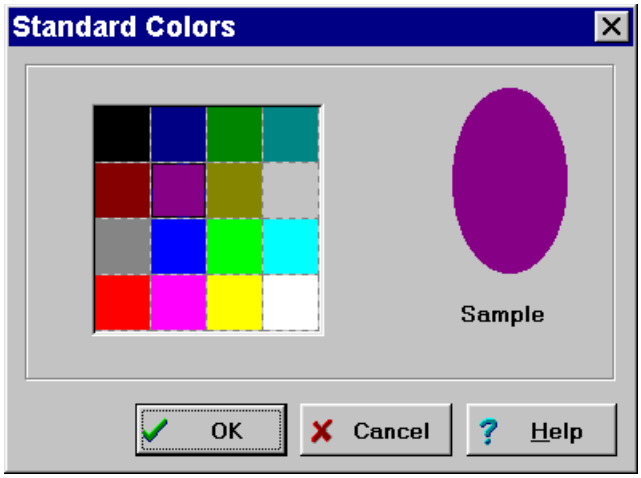
WINDOW -- Used to automatically select all entities inside of the selected window. You will be prompted to indicate (with the cursor) the two corners of the window.

LAST -- Easy selection of the last entity created. Not valid after and MODIFY, or MACHINING commands.

REMOVE -- Works exactly like the SINGLE selection above, except that the entities you pick are DE-selected (removed from the selection set).

OUTSIDE -- Works as the exact opposite of "WINDOW", selecting only entities who's end points fall outside of the specified window.

BY-COLOR -- Works exactly the same as "BY-LAYER". After selecting the color, all entities of that color will be selected.



ARC-RADIUS -- After prompting for a Radius value, Every arc with the specified radius will be selected. A good example of using this command would be to select all holes of a certain radius for drilling.

LINE-LENGTH -- By specifying the minimum / maximum length of lines to be selected. This command can also be used to select lines that extend way out of the drawing area. For example Shorter than 99999.0, and longer than 50.0 would remove any lines longer that 50 inches. You will be asked:

```
Find Lines with Length Shorter Than [10.0000] ?
Find Lines with Length Longer Than [0.0000] ?
```

Z-LEVEL -- Allows entities to be selected based on a -Z- level range that they fall within. This can be useful for modifying (erasing, moving, changing layers) wireframe geometry imported from other CAD systems.

```
Maximum -Z- Value [0.0000] ? / Minimum -Z- Value [-1.0000] ?
```

Entity Masks

At times the process of selecting an entity with a mouse may not be specific enough to return the desired entity. Example: Suppose you are trying to select a point that is the tangent point between a line and an arc. Simply placing the pick box over the point may select the line, arc, or point, depending on which entity is internally stored first. However by setting the proper mask, your cursor selection will be limited to the type of entity selected in the mask. To change the current mask, simply select a new Mask from the tool bar on the top of the display area (In the picture that follows, the mask box is the one that reads “All Entities”). IT IS A GOOD IDEA TO RETURN THE MASK SETTING BACK TO “ALL ENTITIES” AFTER THE CURRENT COMMAND. FORGETTING TO DO THIS CAN MAKE SUBSEQUENT COMMANDS APPEAR TO MALFUNCTION.

Hot Keys and Function Keys

Hot keys and function keys are used for speedy selection of commands. These commands are generally available all of the time with the exception of while you are answering a prompt.

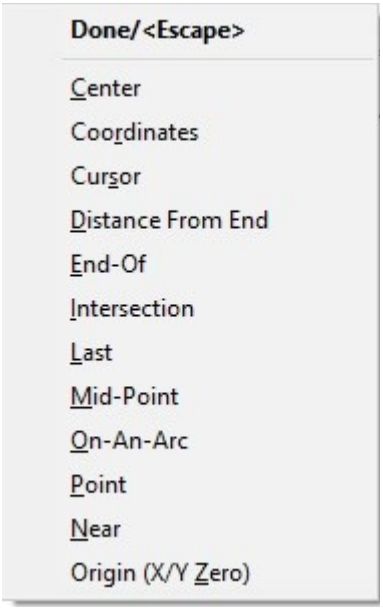
<F1>	**Help system**
<F5>	**Display Refresh**
<F7>	**Delete Last**
<F8>	**Current Color**
<F9>	**Layer Control Dialog**
<Ins>	**Repeats the last command (menu selection)**
<Ctrl>O	**Jumps directly into the “operations” Dialog.**
<Ctrl>P	**Print**
<Ctrl>C	**Copies the current image into the WINDOWS clipboard**
<Ctrl>N	**Run the Post processor**
<Ctrl>E	**Zoom Extents**
<Ctrl>R	**Zoom Previous**
<Ctrl>Z	**Zoom**
<Ctrl)V	**Starts the Solid Verification (PRO/PRO+ Only)**

Additional Hot keys will be described in the **"GET POINT"**, **"GET DISTANCE"**, and **"ENTITY SELECTION"** text.

GET POINT MENU

Within most all commands that require an X and Y coordinate you will be presented with the "GET POINT" menu. This menu provides a great deal of flexibility in specifying an X and Y location.

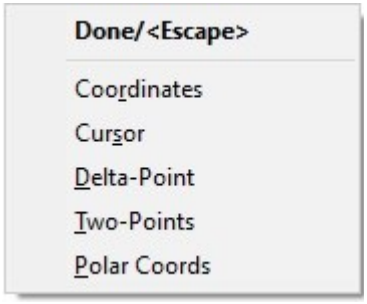
The menu displays:



Selection	Hot Key	Description
Center	C	Center point of an Arc or Ellipse
Coordinates	R	Prompts for the absolute X and Y coordinates.
Cursor	S	Displays a cross cursor for input of a screen position.
End-Of	E	Returns the End point of a Line, Arc, or Ellipse that is closest to the selection position.
Intersection	I	Finds the intersection of any combination of lines and arcs.
Last	L	Returns the X and Y location of the last screen location selected.
Mid-Point	M	Finds the Mid-point of a Line or Arc.
On-An-Arc	O	Calculates a point on an Arc or Ellipse, at a specified angle.
Point	P	Returns the X and Y location of a previously defined point.
Near	N	Calculates a point on an Arc or line near the selection point.
Origin (X/Y Zero)	Z	Enters the default origin of 0,0
Obround	B	Specifies the center of an obround shape (Power-fab only)

GET DISTANCE MENU

Within most all commands that require an X and Y distance you will be presented with the "GET DISTANCE" menu. This menu provides a great deal of flexibility in specifying an X and Y distance.



Selection	Hot Key	Description
Coordinates	**R**	Prompts for the absolute X and Y distances.
Cursor	**S**	Displays a cross cursor for input of two screen positions then calculates the true distance between these two positions.
Delta-Point	**F**	Prompts for a point using the "GET POINT" menu, then asks for an X then Y distance. The X and Y distance will be added to the selected point to calculate the total distance.
Two-Points	**T**	Prompts for two points using the "GET POINT" menu then calculates the difference between them.
Polar Coords	**P**	Prompts for a base point using the "GET POINT" menu. Additionally you will be prompted for the distance and angle from the base point.

Using the POWERSTATION HELP System

POWERSTATION – Learning to use the on-line help systems cannot be stressed enough, all of the information in the help files can be searched much faster then text based manuals, and we expect to be constantly updating the on-line help, while keeping manual revisions to a minimum. If you are not familiar with the features of the WINDOWS help system, in POWERSTATION select “Help”, then “Contents” to navigate through the various topic. “Help Search for Help on..” and type in the name of a topic. Pay special attention to the “Search” TAB, it can help you to find information just by knowing one of the key words involved, example; typing “DXF” in the “Search” page, and several areas of the manual where DXF files are discussed.

The on-line help files contain the detailed reference on every command / feature in POWERSTATION, this manual is intended primarily as an introduction/tutorial.

CALCULATED FIELDS

To simplify the entry of numbers, the standard Windows control “Text Box” has been extended to allow the entry of numbers as calculated expression.

The calculator function works as a twenty two function scientific calculator and is available any time throughout POWERSTATION whenever a number must be input. Please Note: All trigonometric functions are in radians.

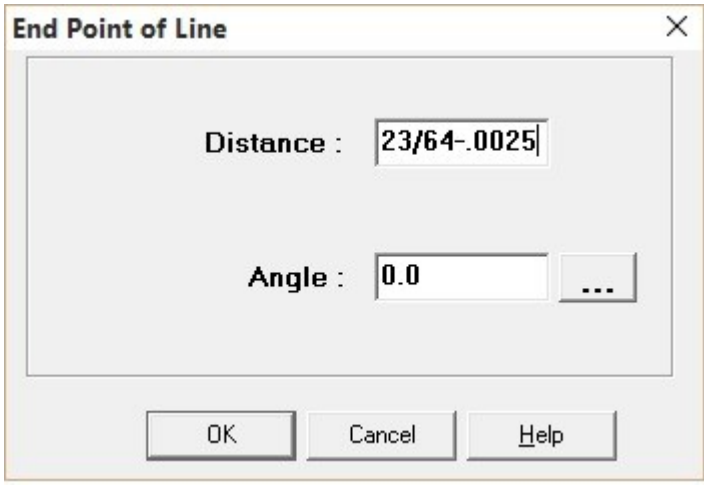
Sin Sine	**Cos** Cosine	**Tan** Tangent
Sec Secant	**CSC** Cosecant	**Cot** Cotangent
Asin Arc Sine	**Acos** Arc Cosine	**Atan** Arc Tang
Sinh Hyperbolic Sine	**Cosh** Hyperbolic Cosine	**Tanh** Hyperbolic Tangent
Sech Hyperbolic Secant	**Csch** Hyperbolic Cosecant	**Coth** Hyperbolic
Ln Natural Log	**Log** Log Base 10	**PI** 3.14159
Abs Absolute Value	**Sqrt** Square Root	**Exp** Exponential
Fact Factorial	**Sqr** Square (X*X)	

Rd Constant to convert Radians to Degrees 57.29578
Dr Constant to convert Degrees to radians 0.01745
Ft Constant to convert feet to inches 12.0
% Character to convert results from metric to inch mode
Standard Operator's (in order of precedence):

^ Exponential
*** Multiplication**
/ Division
+ Addition
- Subtraction

Parentheses ' () ', Square Brackets ' [] ', and Curly Brackets ' {} ' may be used to override the normal order of evaluation.

SIN(23DR)	(sine of 23 degrees)
Sqrt(SQR(3.5)+SQR(2))	(Square root of 3.5 squared plus 2 squared)
2 + (23/64) - .875	(2 plus 23/64 minus .875)
SIN(10+23/60+13/3600DR)	(Sine of 10 degrees23 minutes and 13 seconds)



Notice that the distance has been entered as “23/64-.0025”. When the input is moved to the next “text box” by pressing the <Tab> key or selecting a new box with the mouse, the “Distance” box will be changed to read “0.356875”.

ENTERING DIAMETERS

When prompted for a radius dimension, diameters can be enter simply by adding the letter “D” Example:

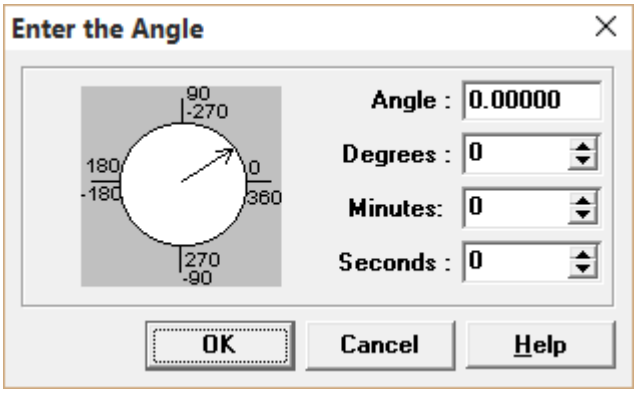
```
Enter the Radius [2.375D]
```

The radius value will be recorded as 2.375/2 or 1.1875.

Angle Entry

In POWERSTATION angles are measured in degrees from the three O'clock position. Counter clockwise angles are specified as positive values, and clockwise angles are specified as negative values. In other words +45 and -315 represent the same angle. If minutes and seconds are to be entered you enter them directly in the dialog. Example:

For 30 degrees, 12 minutes, 16 seconds type:



LAYERS

In a CAD/CAM system layers can be thought of as clear overlays that can be placed over the current drawing and removed at will. POWERSTATION supports a full 256 layers, each with it's own user selectable color, line style, and description, to allow a great deal of flexibility. Layers are commonly used to separate parts of the drawing which may "clutter up" the screen. Typically one might use one layer for the basic part shape, one layer for dimensions, one for the title block, one for each operation of the machining cycle. Each layer can be individually turned ON or OFF. When a layer is turned OFF any information stored on it is not drawn on the display screen and cannot be selected with the mouse (although it still exists in the drawing).

SCALE

For newcomers to CAD/CAM the drawing size and scale can be confusing. In traditional manual drafting, you must choose a scale at the start and everything you do revolves around this scale. In CAD/CAM the scale seen when viewing or plotting is insignificant, as it can be changed at any time with simple commands. A few point to remember:

1) It is generally a good practice to draw everything at its natural 1:1 scale, and use the "ZOOM" family of commands to change what is displayed on the screen.

2) Remember that the scale that he drawing is displayed at has no affect on the part coordinates. The only "scale" commands that in any way change the part coordinates is "BLOCK-MERGE-SCALE", and "MODIFY-SCALE".

3) No matter how large or small a part is it can always be blown-up to fit the entire screen (or plotted sheet).

4) No matter how small a detail is, you can "ZOOM-IN" until it is visible.

Additional Information

The libraries and bookstores contain a many volumes on CAD/CAM techniques. Although the exact commands used in other CAD systems may be different, the principals used to create drawings will be the same. These books will typically go much more into detail on the CAD drafting aspects then this manual.

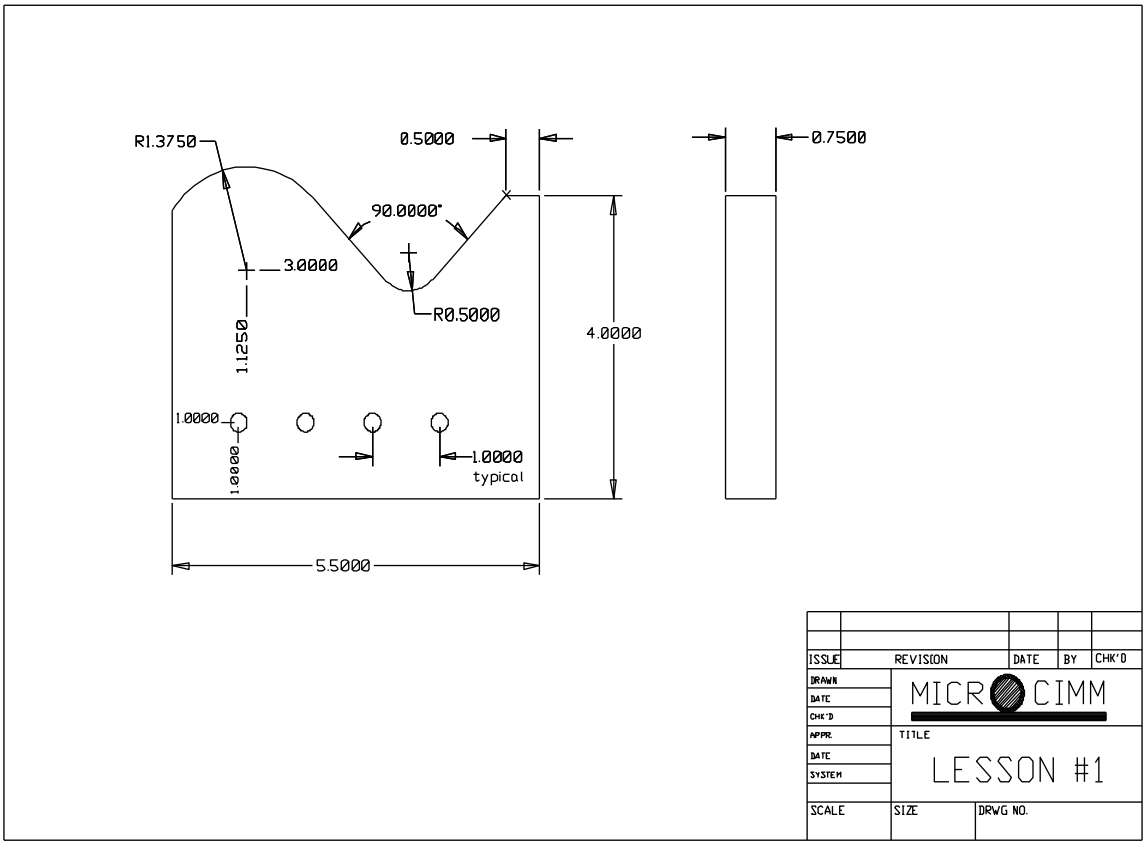


Figure 4-0 The sample drawing

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”

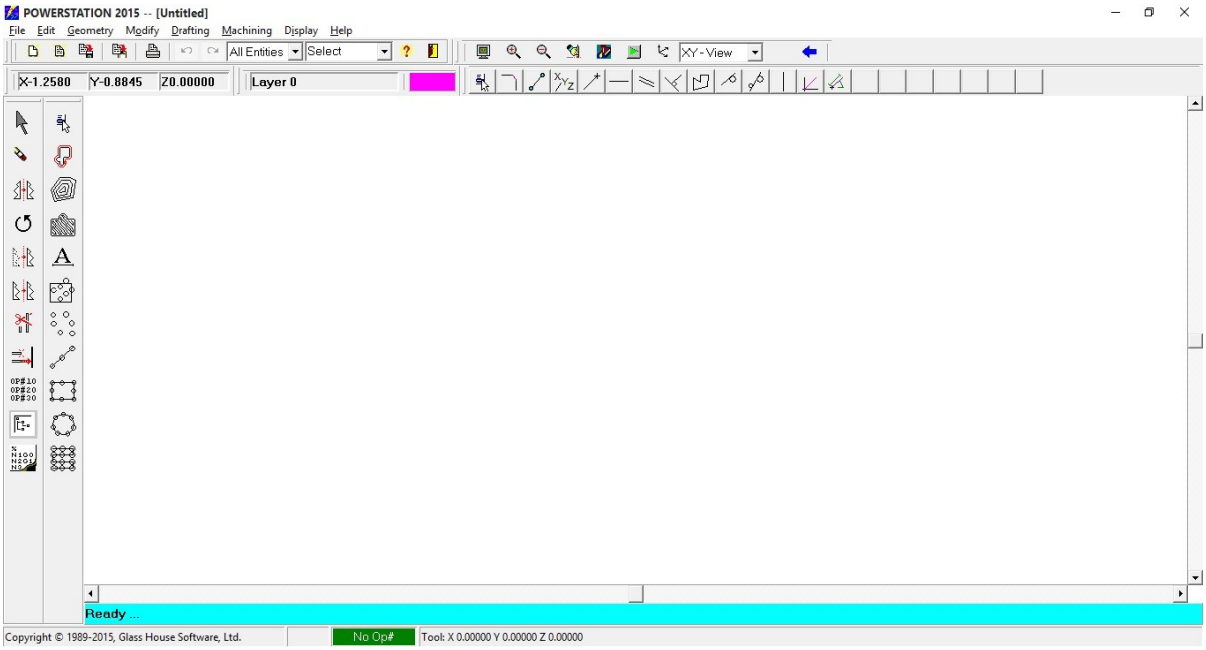


Figure 4-1 The opening screen

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" ,"Shapes", "Box"

At this point the above dialog will be displayed. For this example we need to change the width to 5.5 and the height to 4.0. Press [OK] with the mouse.

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

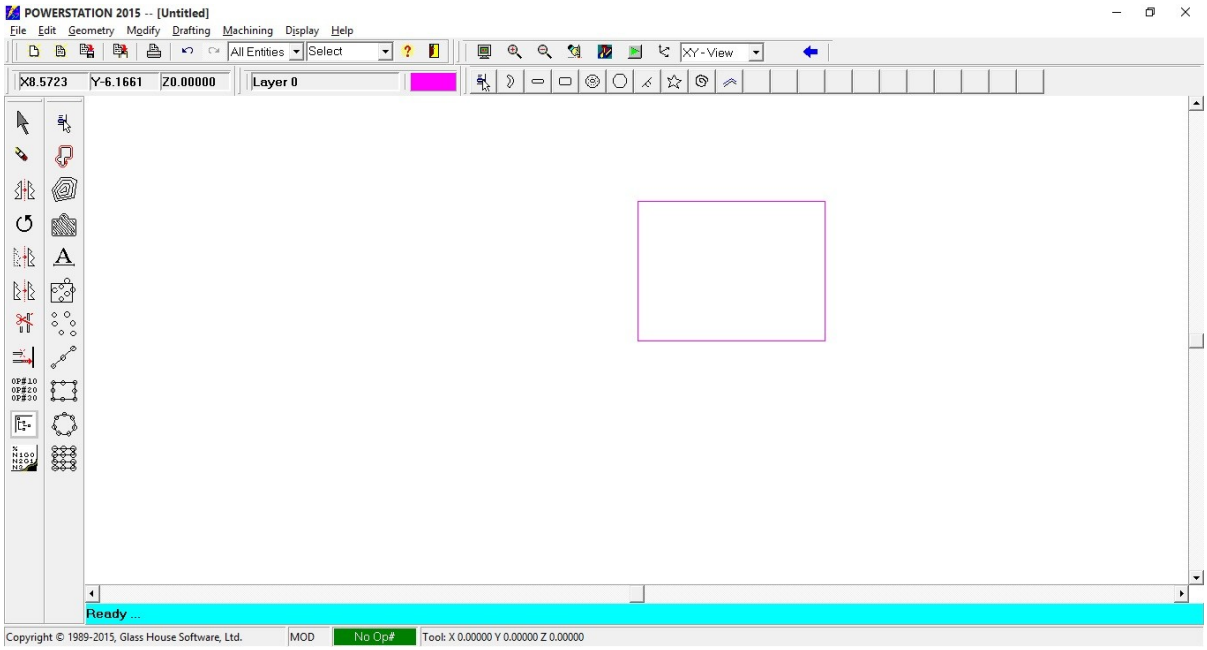


Figure 4-2 the basic part shape (box)

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 (or press <ctrl>E).

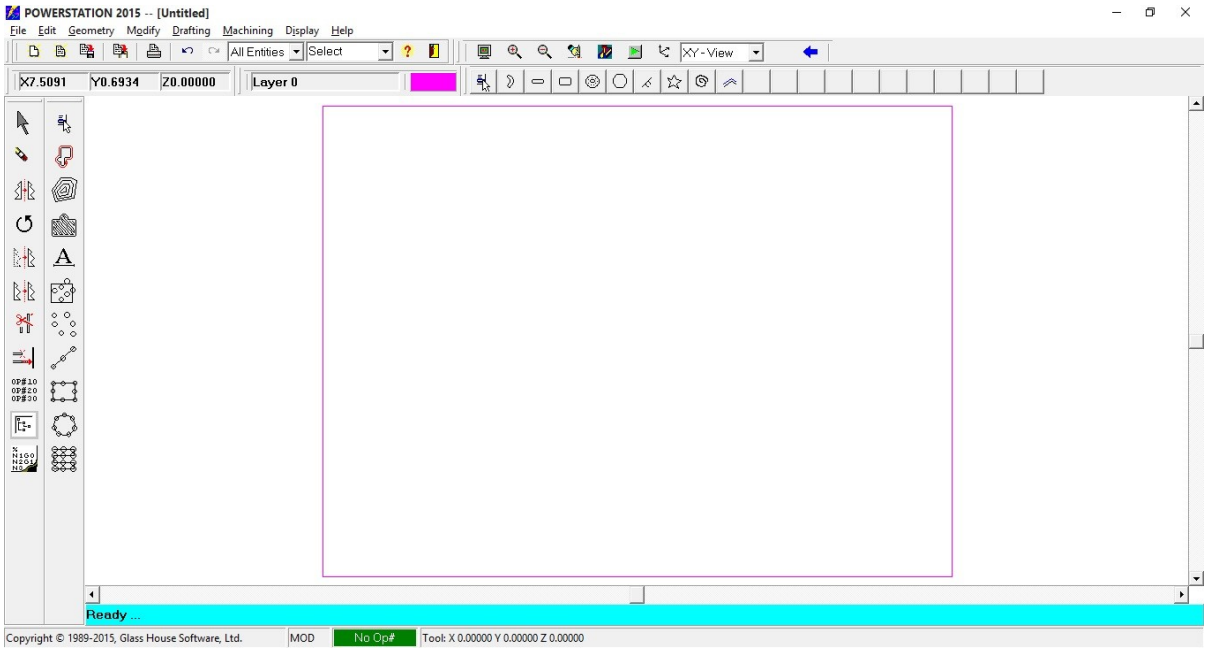


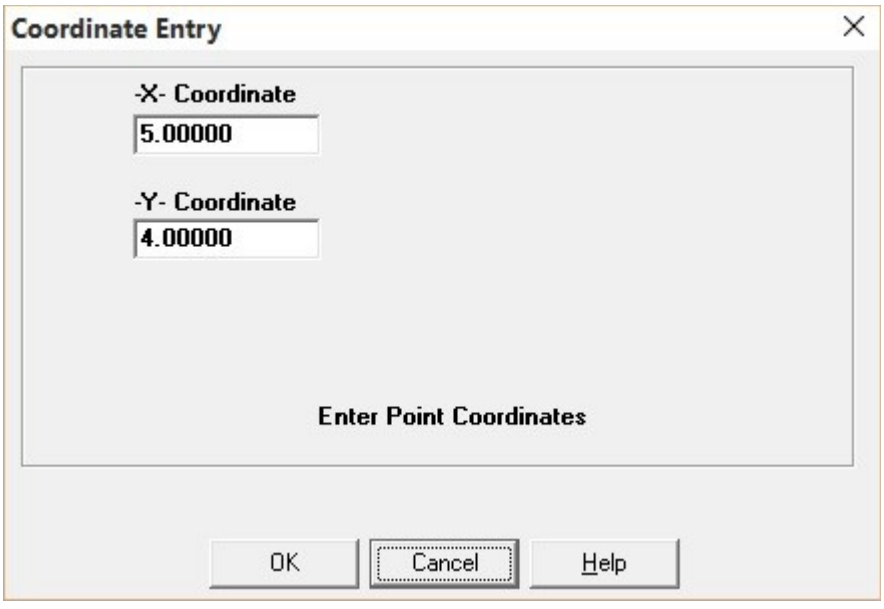
Figure 4-3 The display has been "Zoomed"

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

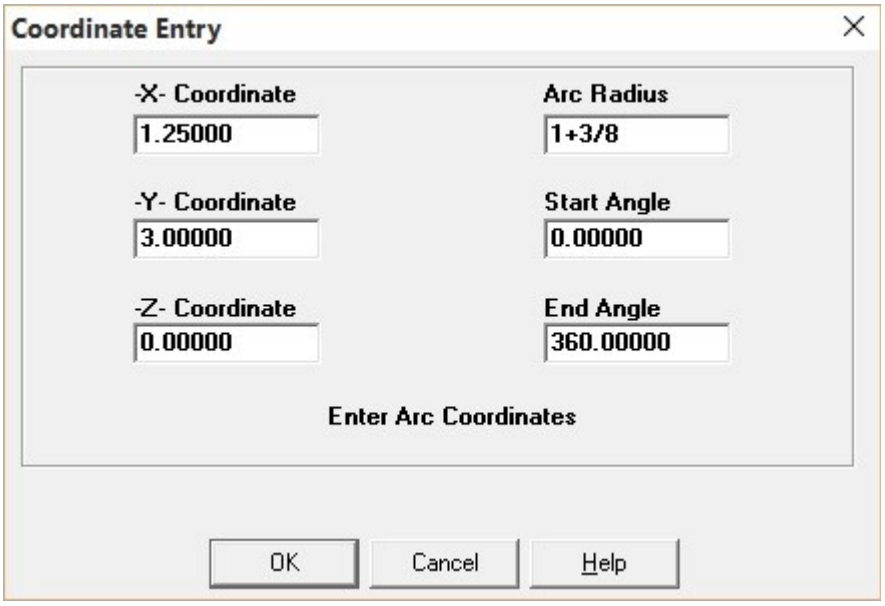
Looking ahead, we only need a small section of the line on the top. The remainder of the line will intersect with several other parts of the final geometry adding unnecessary difficulty to the clean-up “Trimming”. To do this we will change the end-point on the left side. Select “Modify”,”Trimming”,”Change Line End”.

Next, the Get Point menu (See chapter Basics) will be displayed. Select “Coordinates” from the menu. Enter in coordinates of X5.0, Y4.0.



STEP 6

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



Zoom Extents again and the display should now look like Figure 4-4.

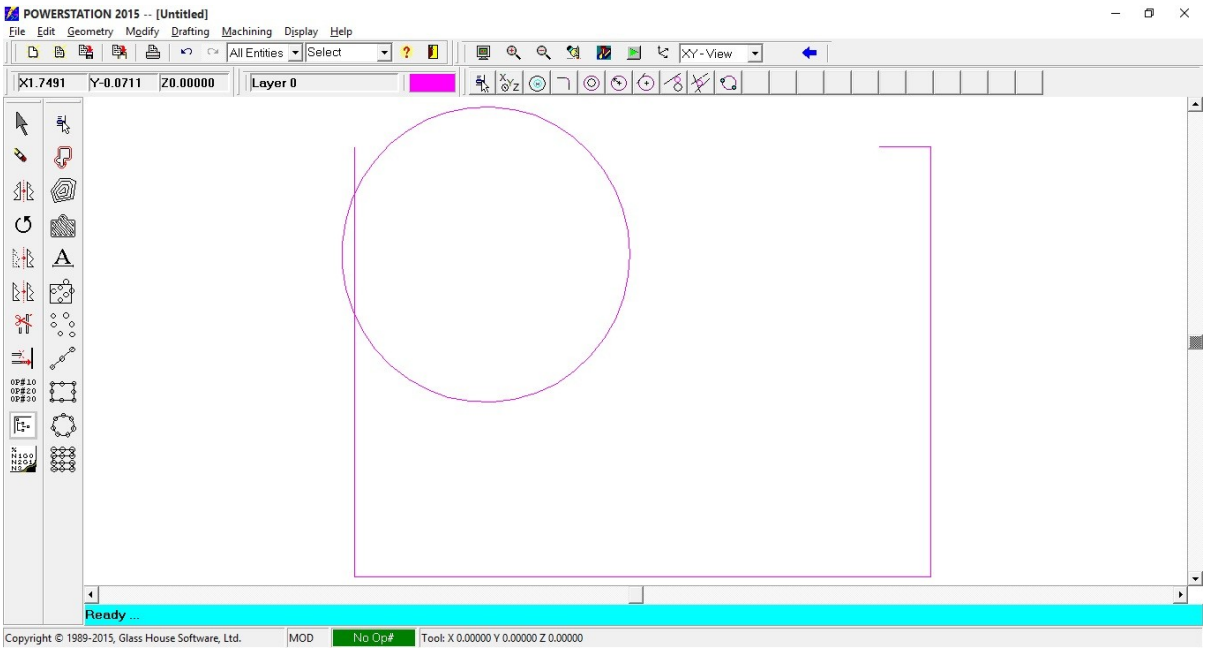


Figure 4-4 The first (1 3/8") arc

STEP 7

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.

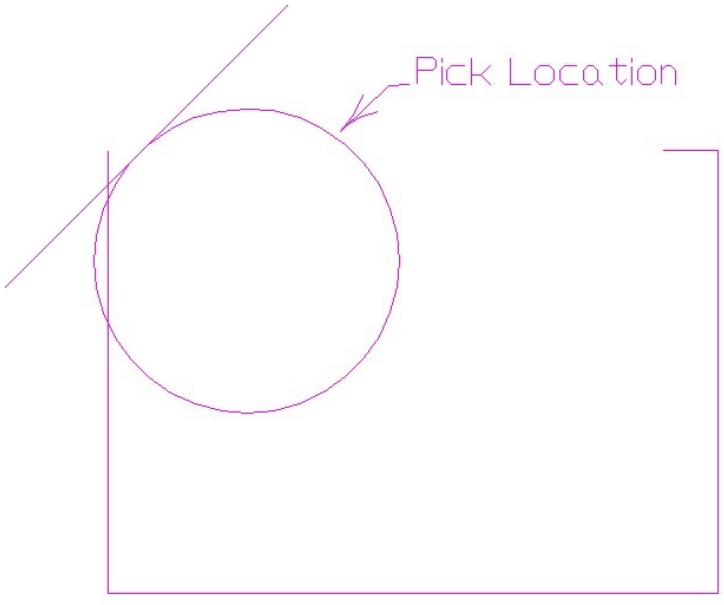
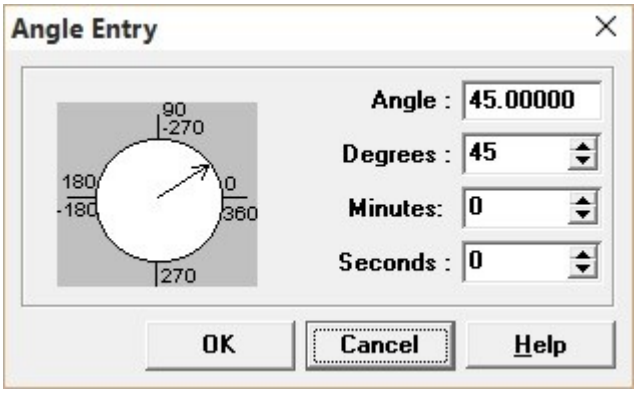


Figure 4-5 Creating the tangent line

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



Notice the wrong line has been created (see figure 4-5). Pressing the ⇐ icon on the tool bar **(Undo)** will remove it. This is a good time to see the effects of using the ⇐Undo and ⇒Redo commands. Repeat the steps to create the line, but use **-45** for the angle.
See figure 4-6

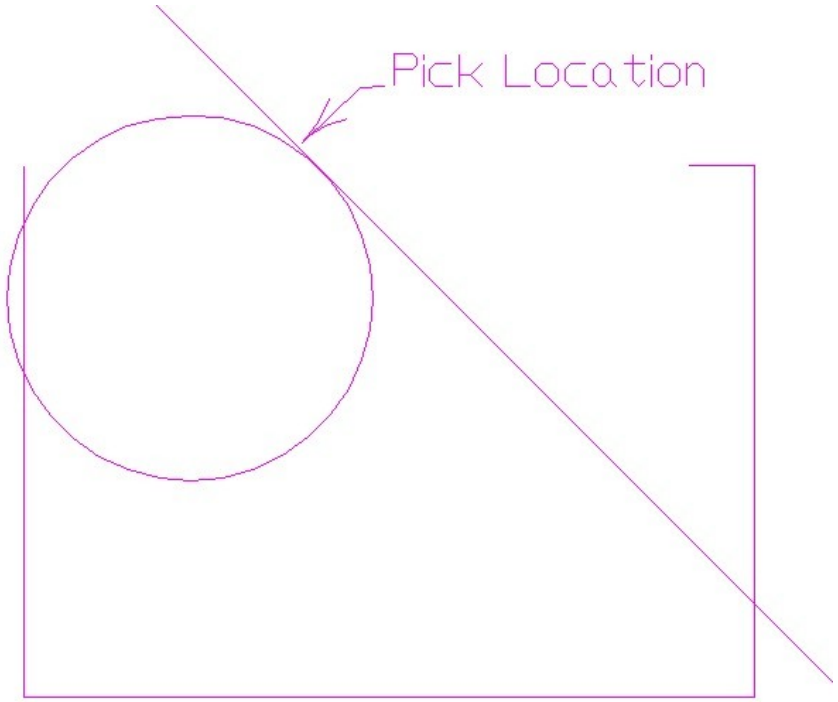
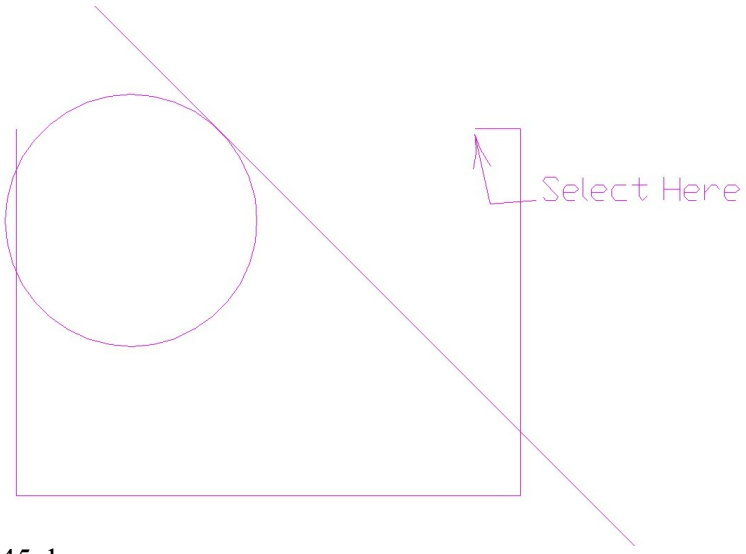


Figure 4-6 The corrected tangent line

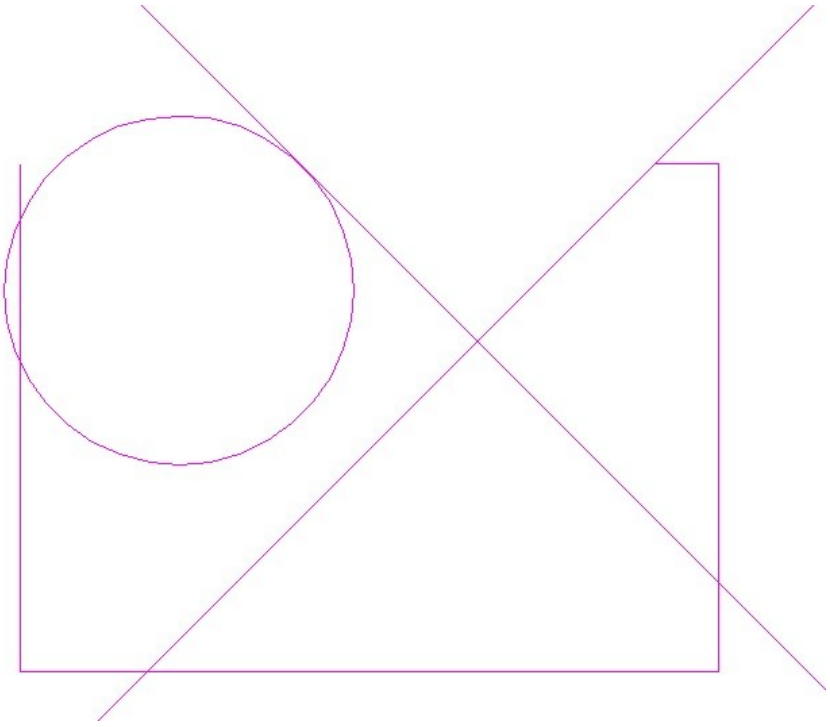
STEP 8

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, or <Escape> for Point Menu"**. Press the <Esc> key or click the right mouse button and select <Escape> from the menu.

The Get Point menu (See chapter Basics) will be displayed, select “End Of”, then select the left end of the line.



Next, enter in an angle of 45 degrees.



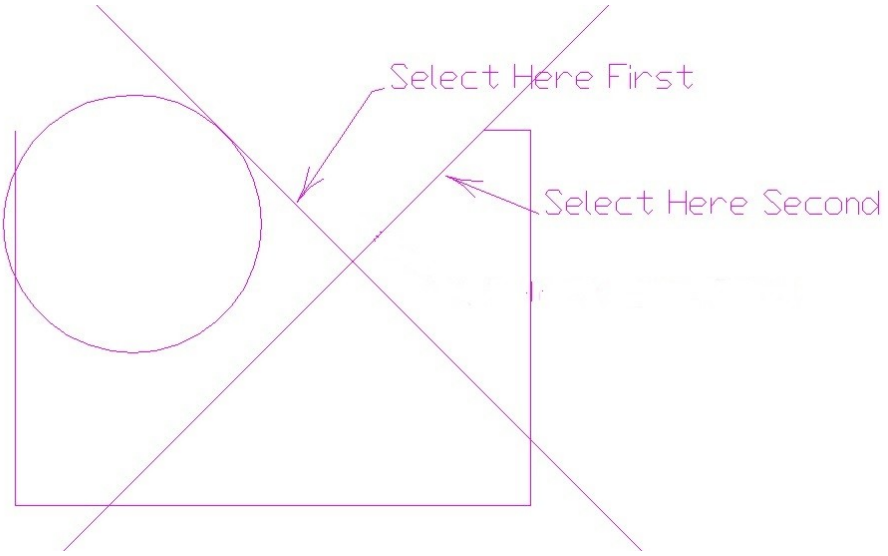
STEP 9

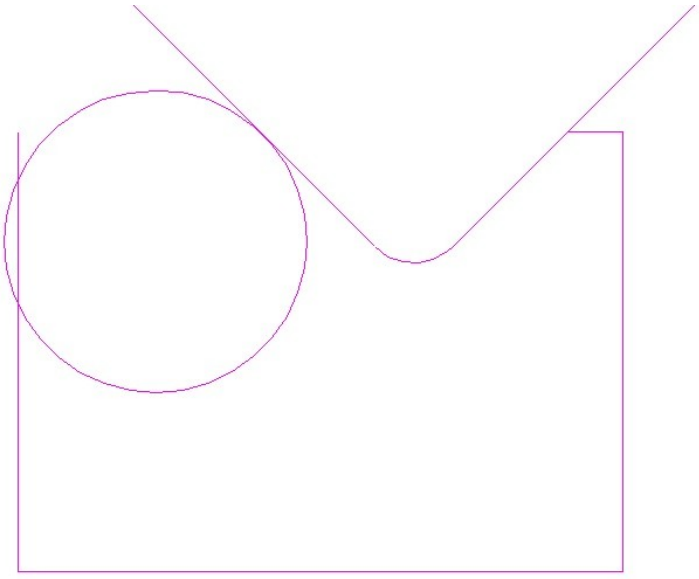
We need one more piece of geometry, the 1/2" radius on the top. To add this we will save a few steps and use the **"FILLET"** command, which will not only add the fillet, but will clean-up (Trim) the lines being filleted.

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
Enter the Radius [1.3750] ? .5<Enter>
```

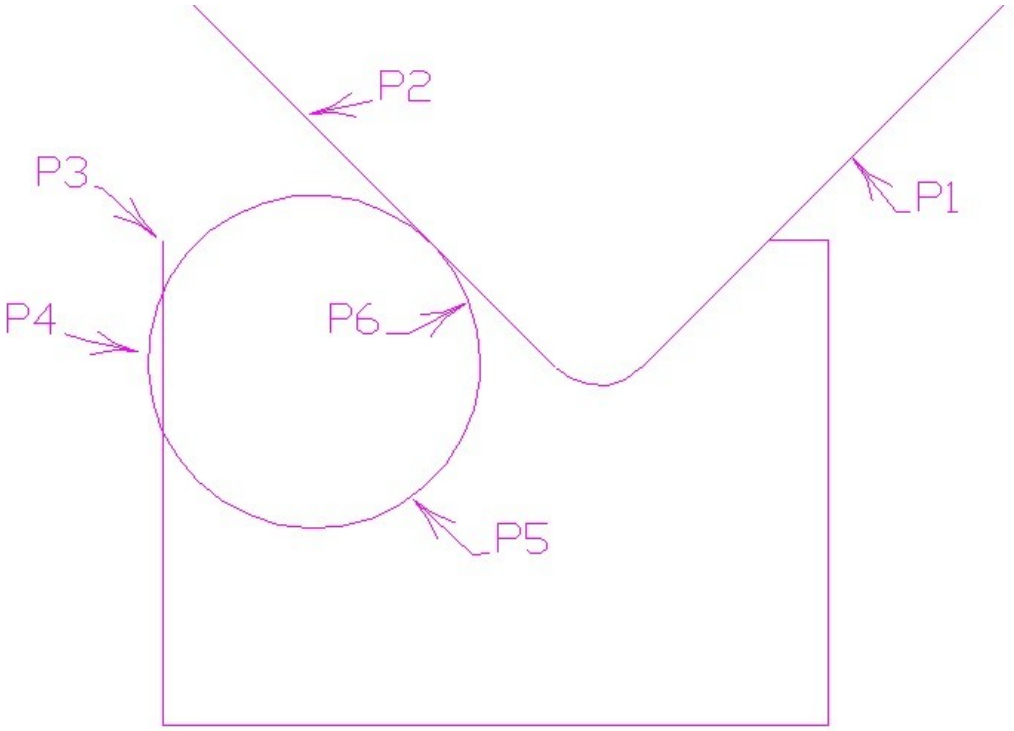
Note: Normally the entities to be filleted must be selected in a counter-clockwise order, or the wrong fillet will be generated. Note: This is not generally necessary when filleting lines, the system will generate the proper fillet automatically (This is NOT true when filleting lines to arcs, or arcs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





STEP 10

For Trimming, we will be using the “Modify”,”Trimming”,"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. Simply click (In Order) on the points in the following examples:



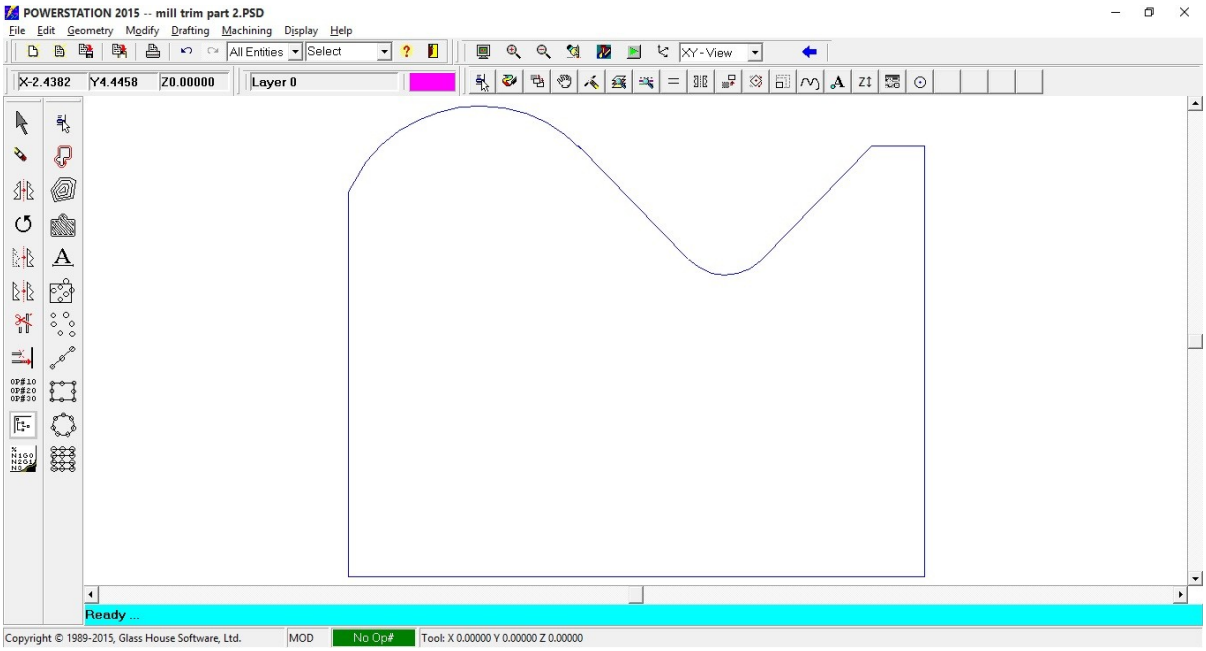
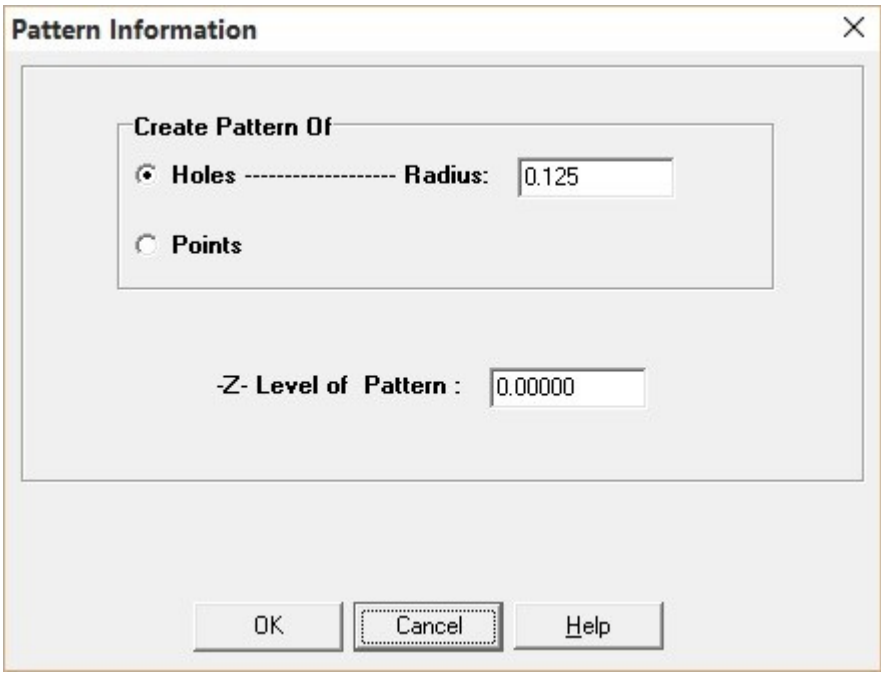


Figure 4-19 The part after trimming

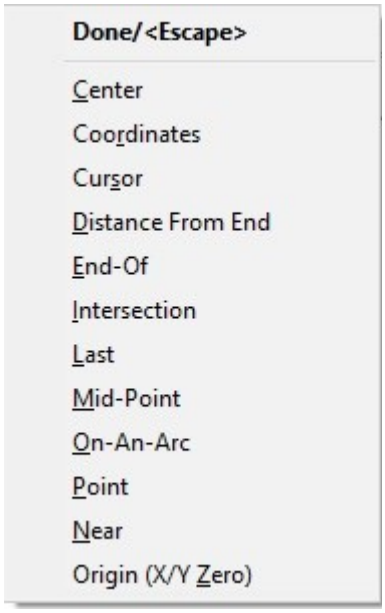
STEP 11

We will draw the drilled holes, using a “linear pattern'.

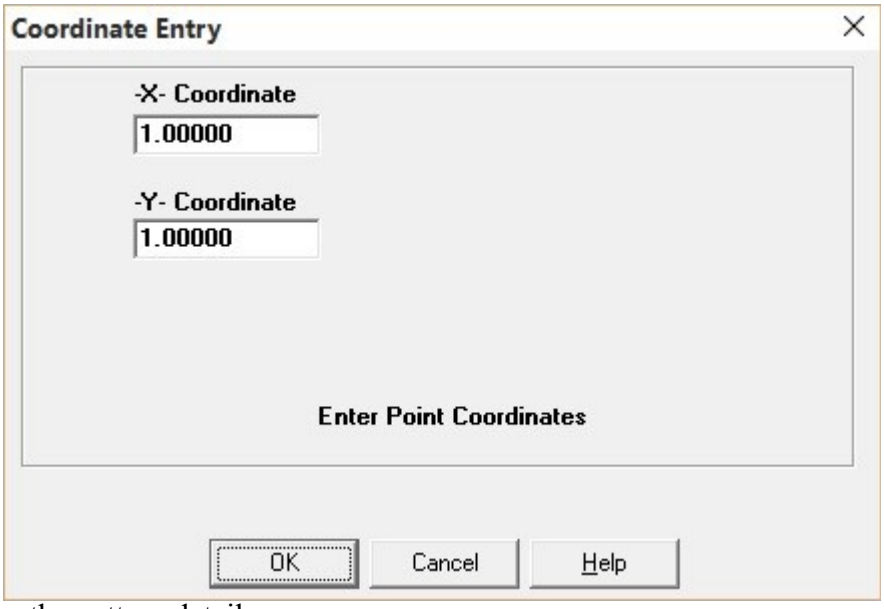
From the menu, select “Geometry”,”Pattern”,”Linear. You will be asked for the basics:



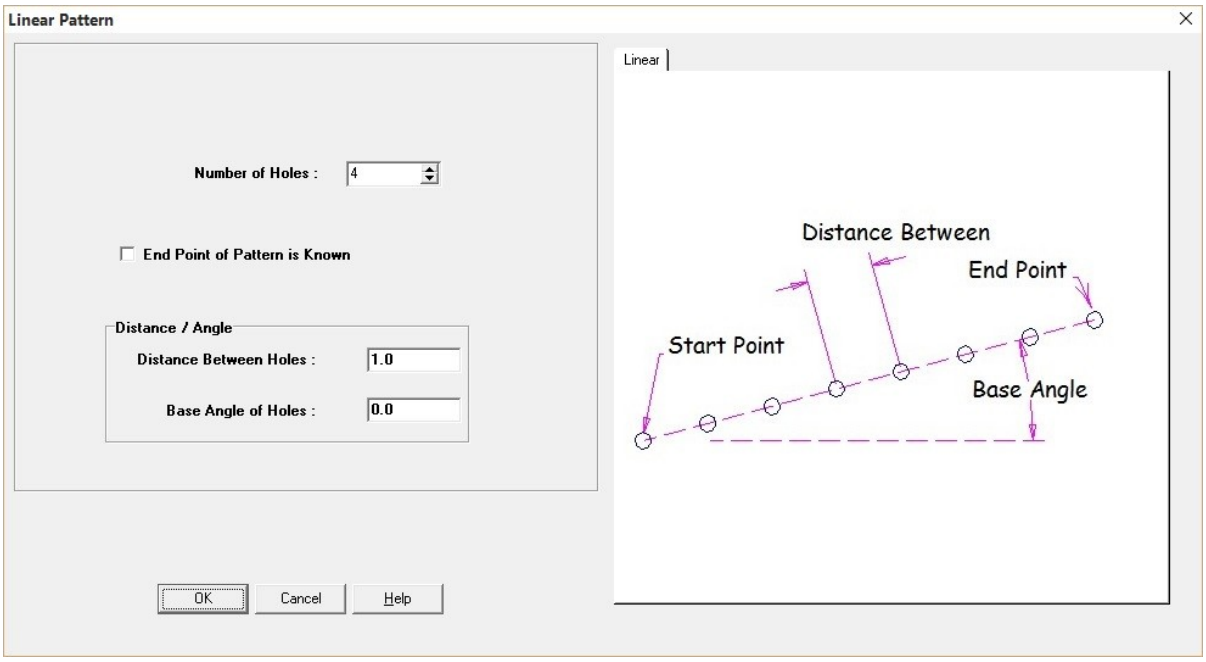
Next you will be asked to enter the start point of the pattern:

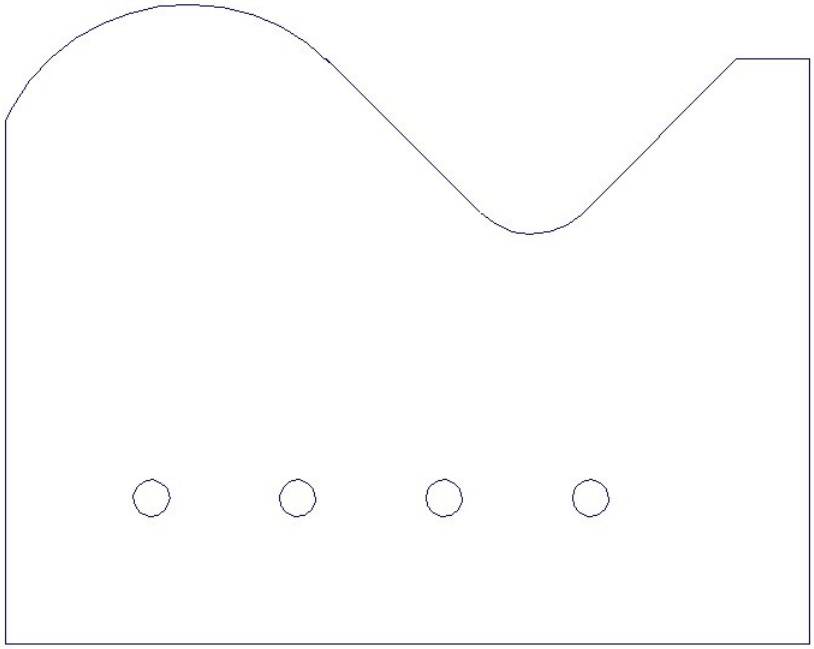


Select “Coordinates”, and enter the following:



Next we enter the pattern details:





The screen will now look like this

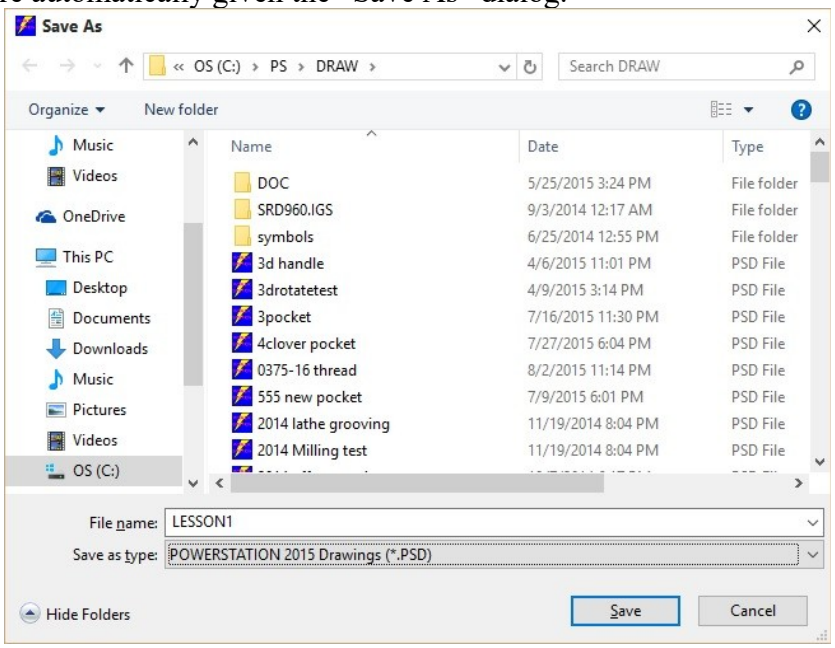
Note: This step is purely for drawing purposes, if all that you want to do is drill these holes, this step could be skipped.

STEP 12

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.



STEP 13

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 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.

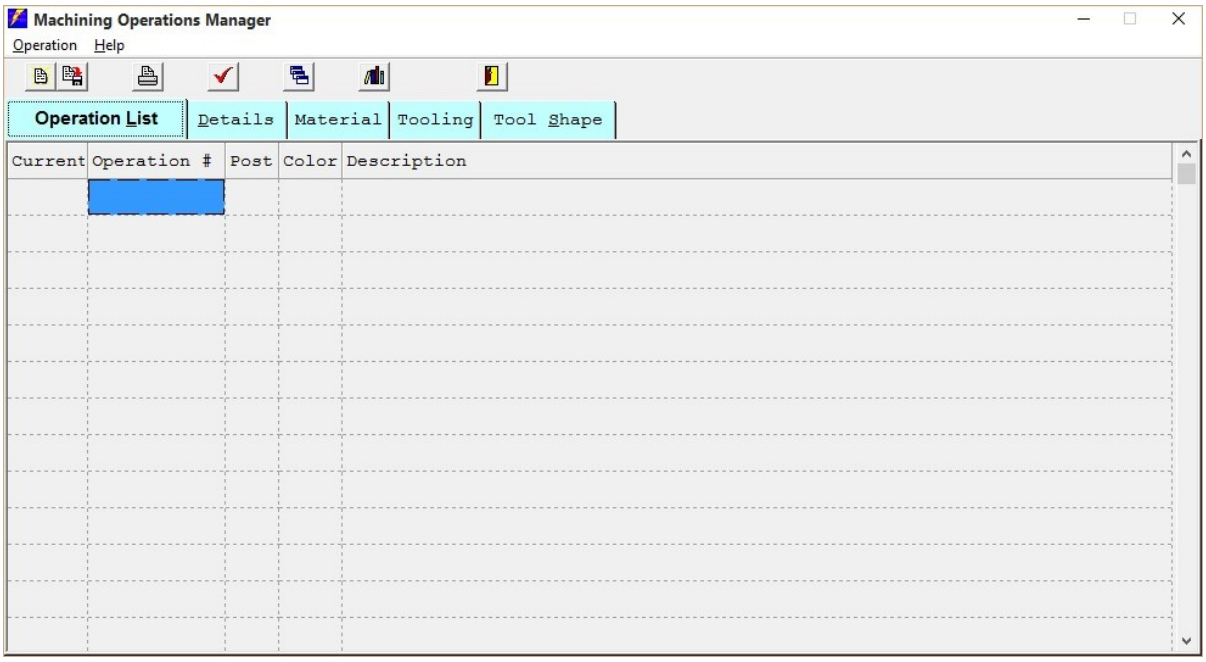


Figure 4-21a (The Machining Operations Manager)

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

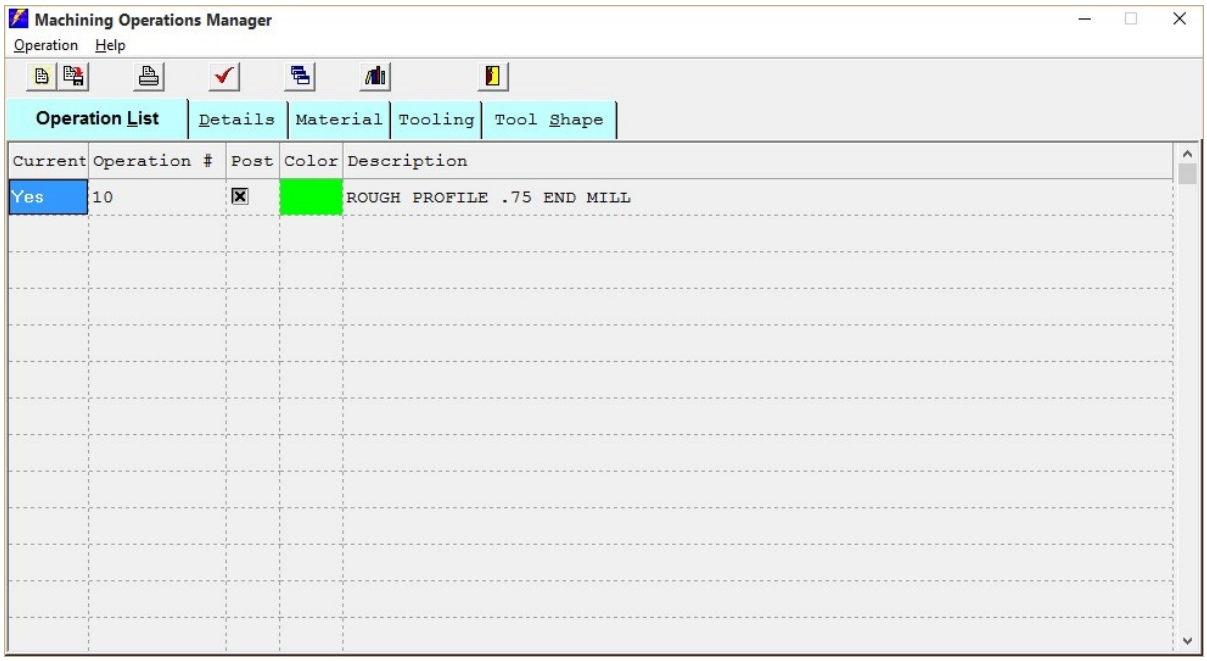


Figure 4-21b (Basic operation information)

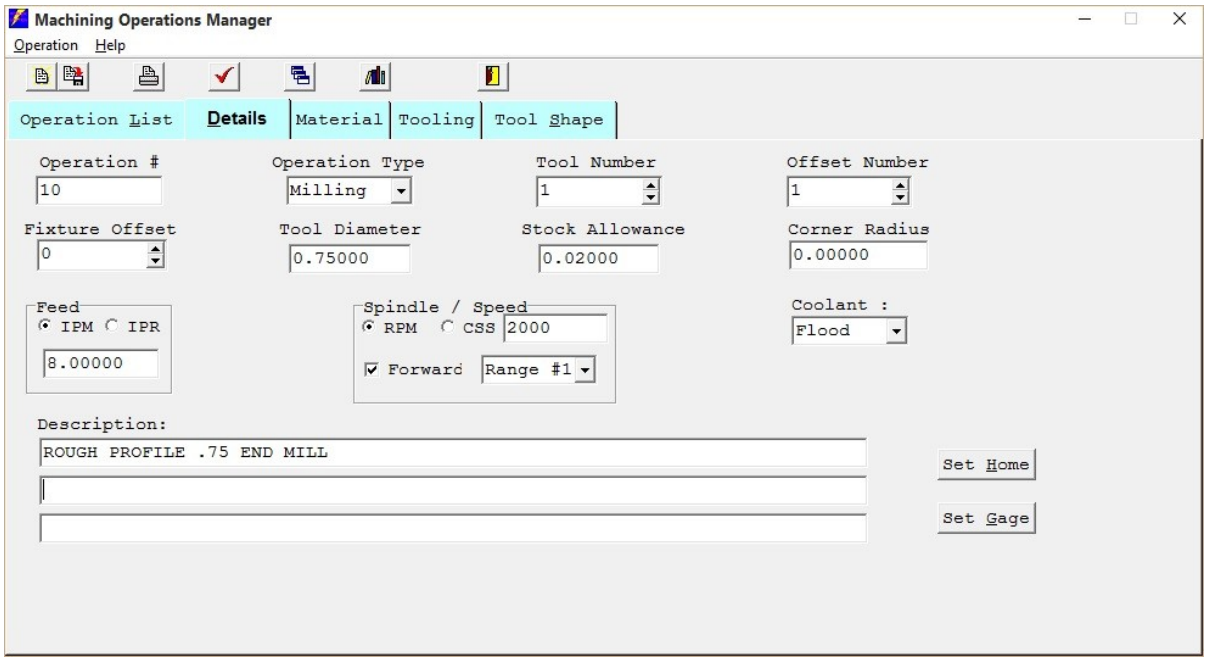


Figure 4-21c (Operation #10 Details)

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

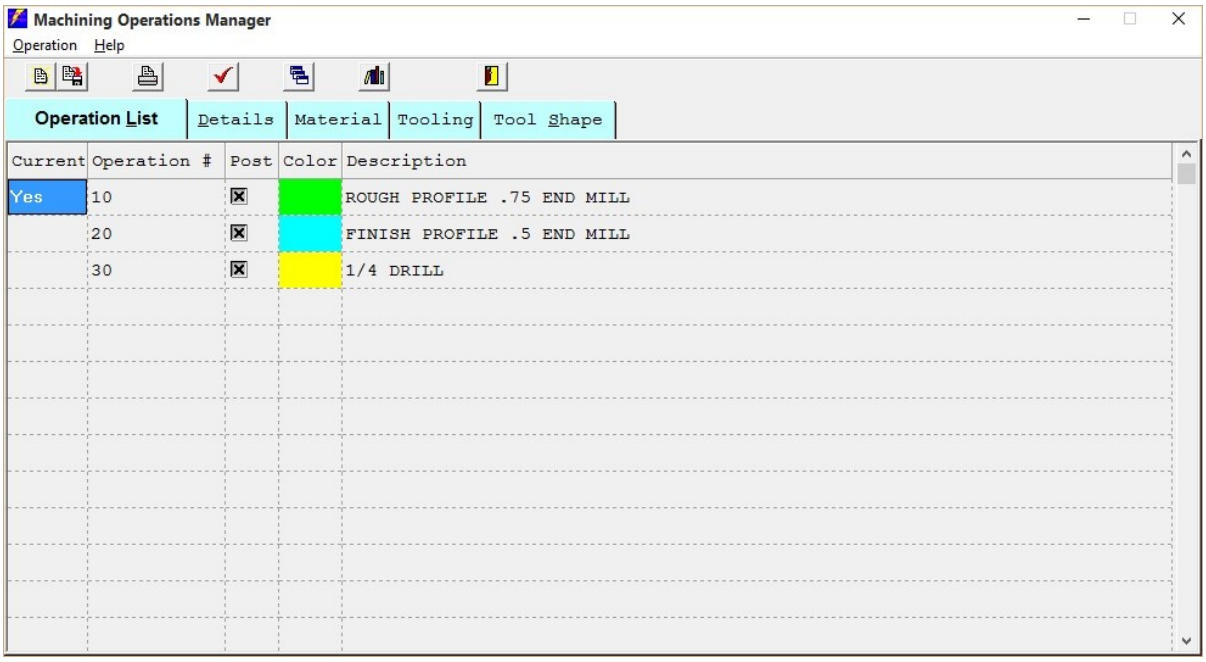


Figure 4-21d (All operations for the tutorial part)

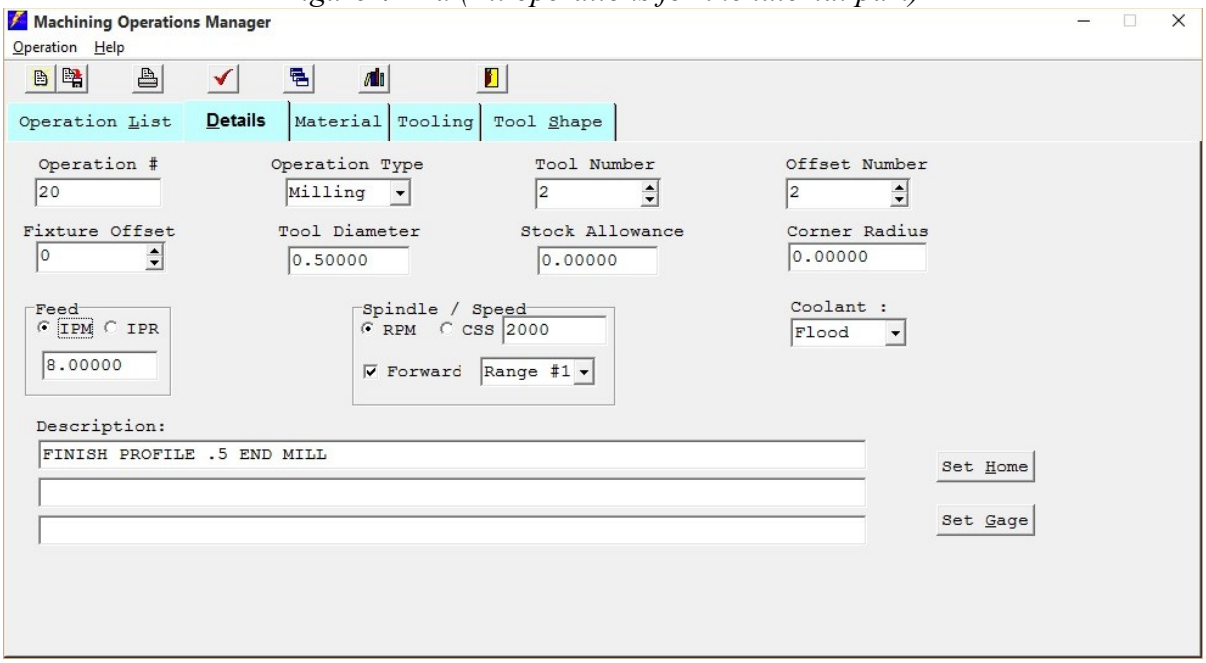


Figure 4-21e (Operation #20 for the tutorial part)

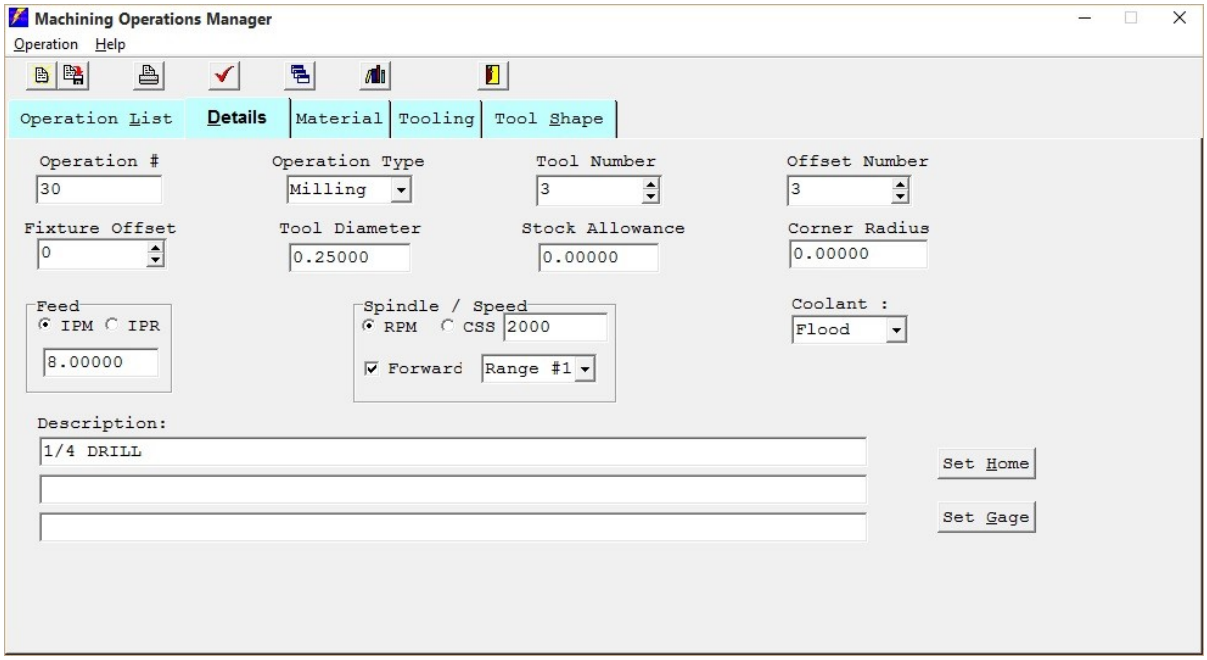


Figure 4-21f (Operation #30 for the tutorial part)

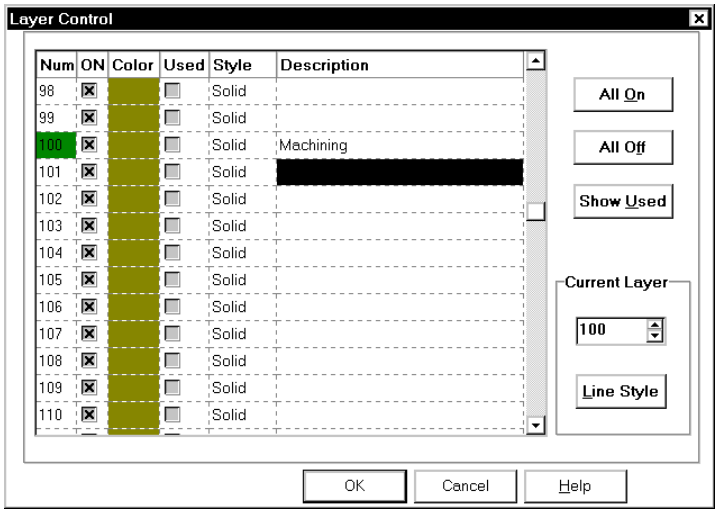
STEP 14

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 15 (Optional)

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 16

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”.

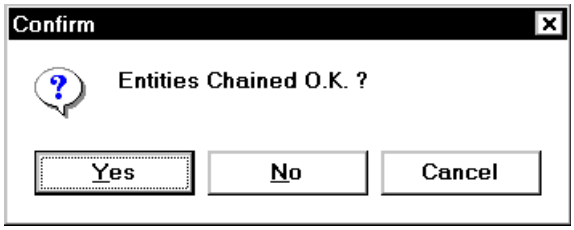
STEP 17

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 14, 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).

STEP 18

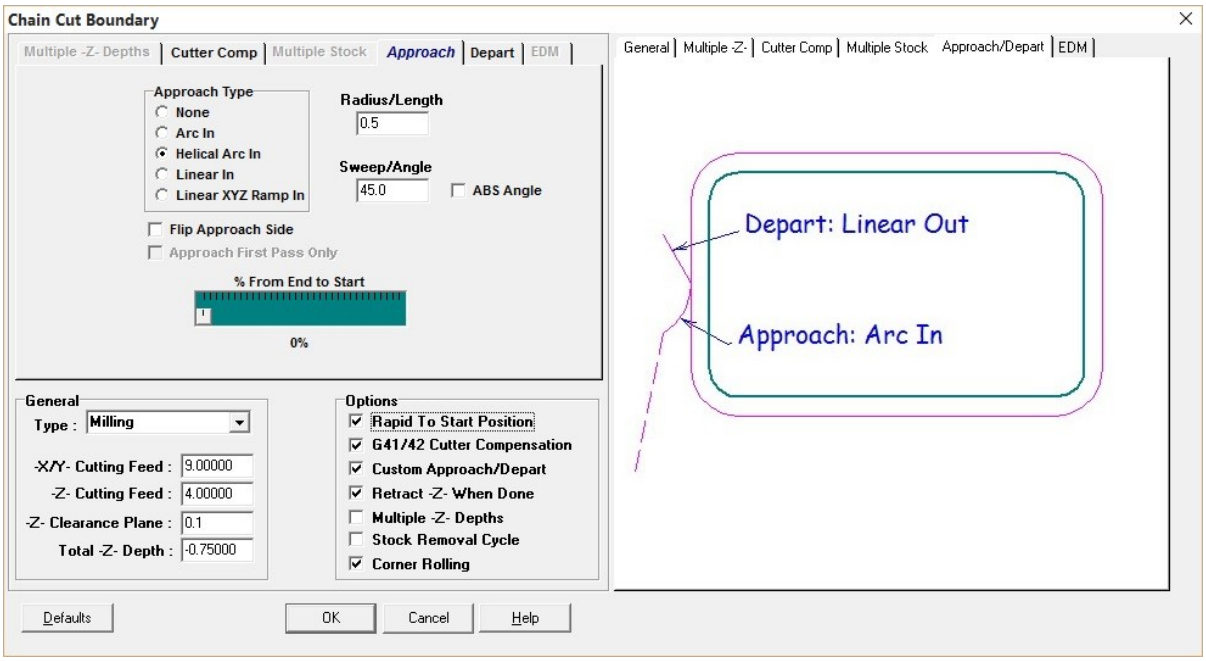
Now we will use "chain selection" to mill the outside profile. Select “Machining”, “Chain Cutting” from the menu. We will be asked:

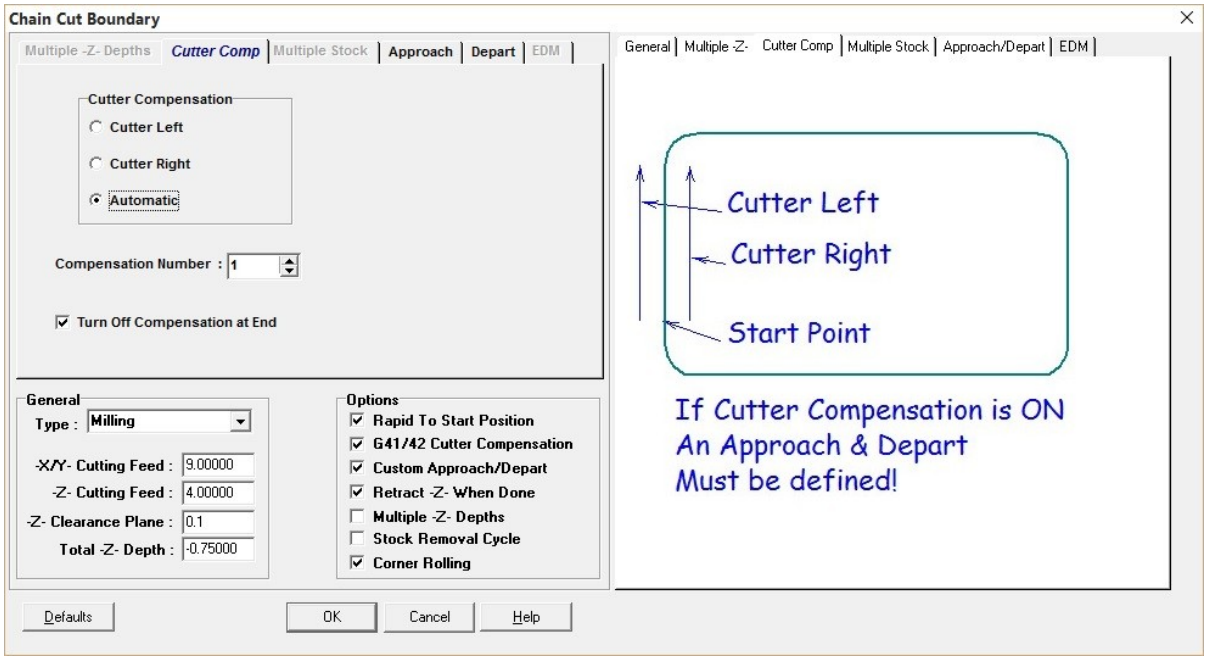
`Select starting line/arc (figure 4-24)`
`Select the Line or Arc to stop before, <Escape> for none`
(Press the <Esc> key, or click the right mouse button, and select <Escape>)



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

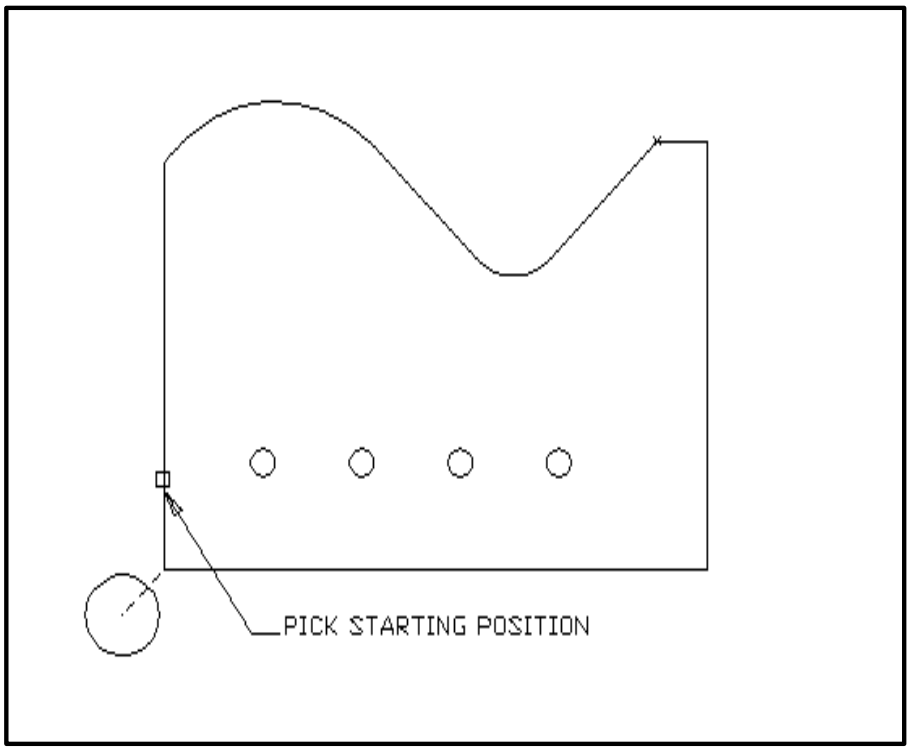


Figure 4-24 Selecting the starting line or arc

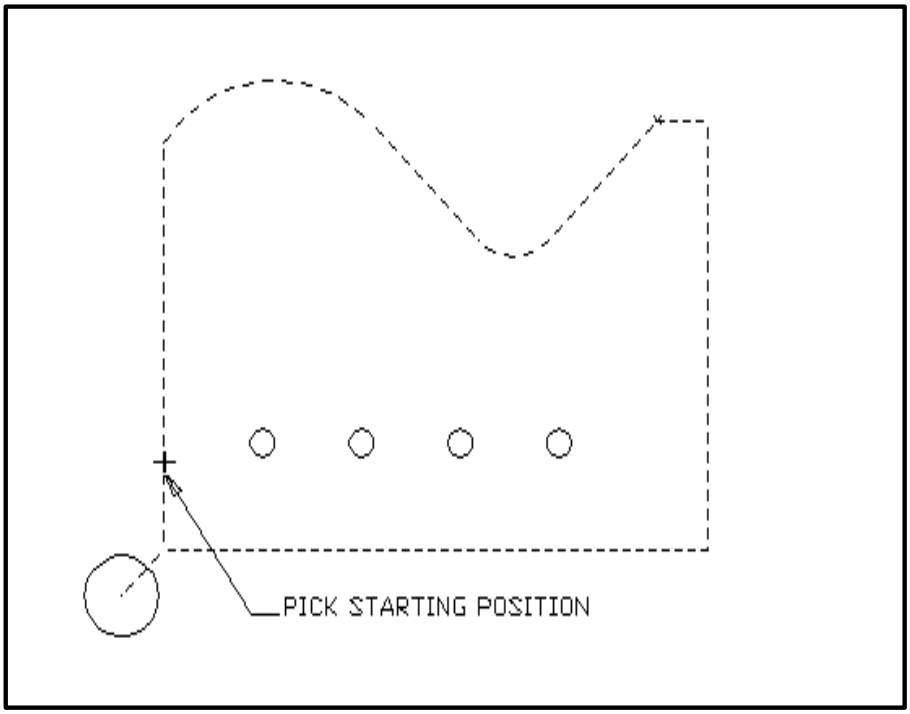


Figure 4-25 The entities have been chained

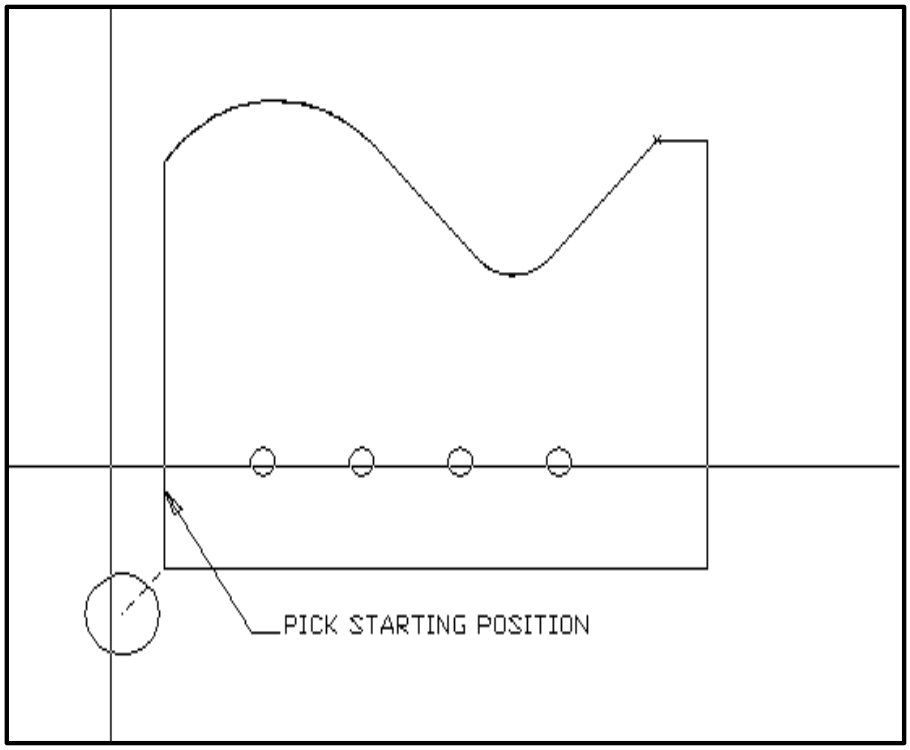


Figure 4-26 Picking the side to offset to

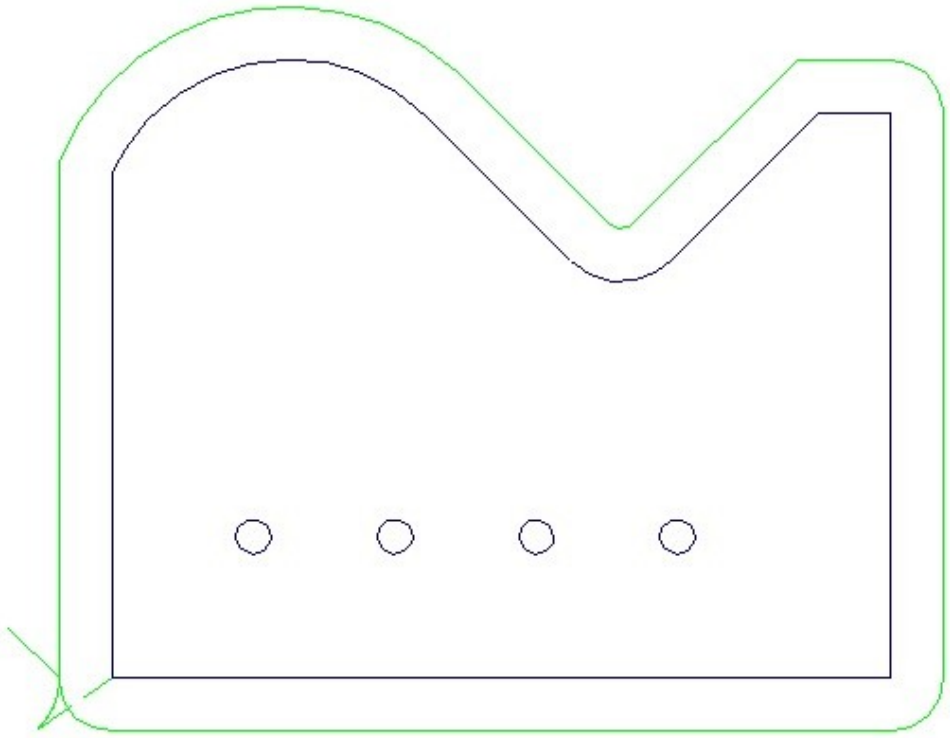


Figure 4-27 The first operation completed

STEP 19

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 step 18. 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 step 18.

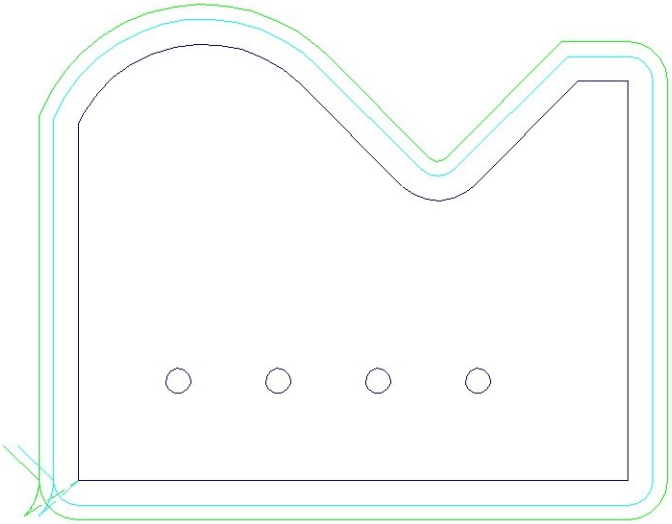
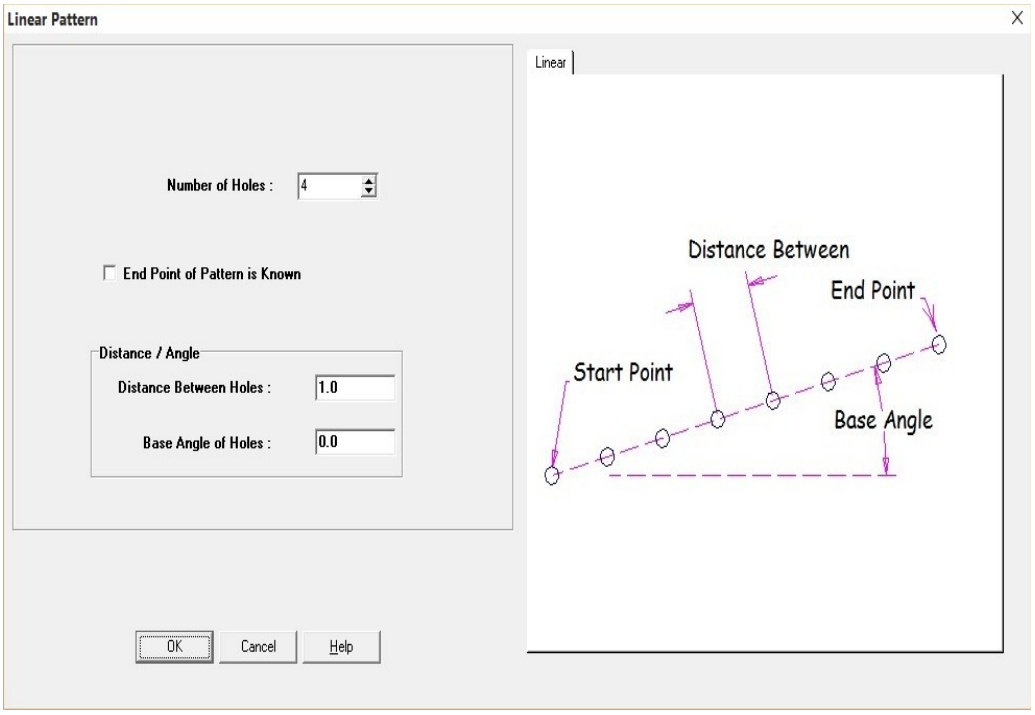


Figure 4-28 The second operation completed

STEP 20

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.

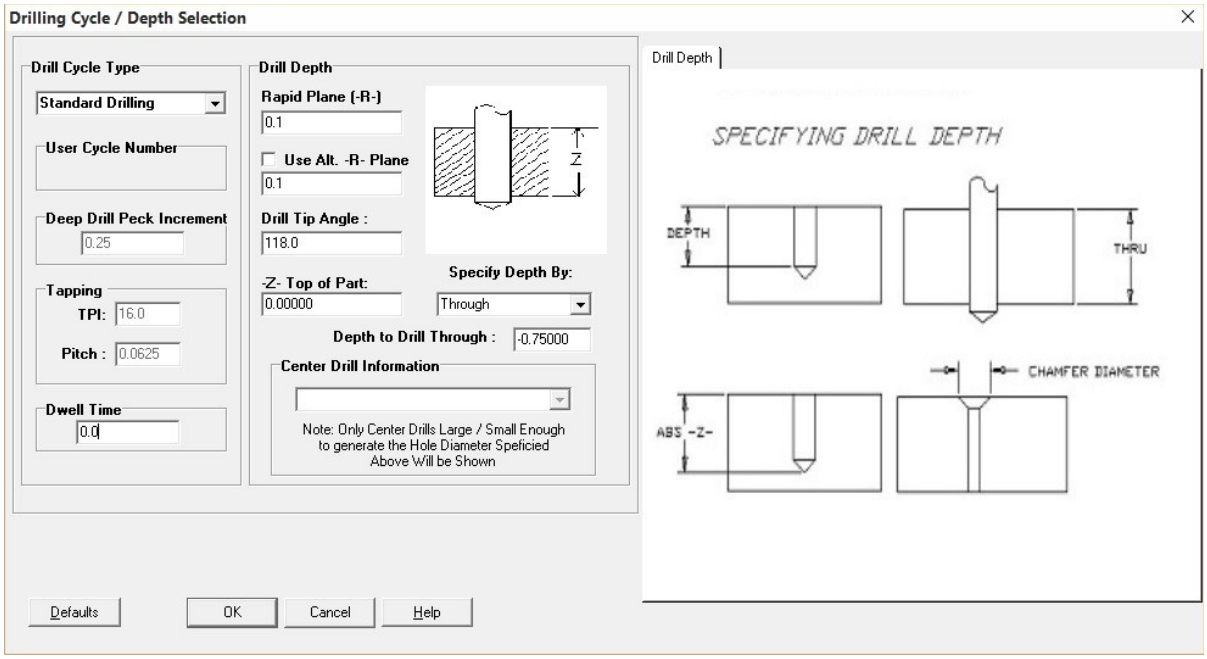


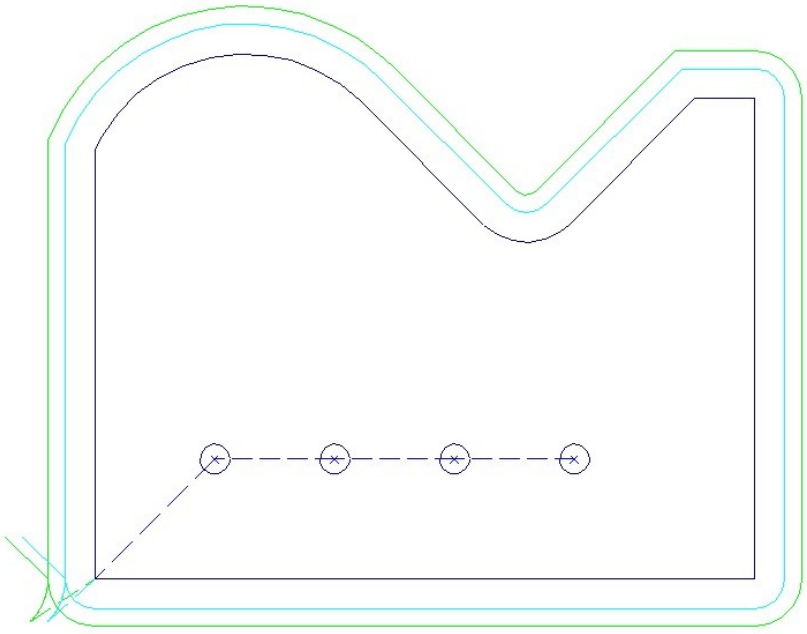
Fill in the “Linear Pattern” dialog as shown above.

Next you be given the "GET POINT" menu, (See chapter "basics" for more information on this menu). Select **"Center"** from the popup menu, then move the pick box on to the first hole and press the left mouse button.

STEP 20 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”.





Completed Drill Cycle

STEP 21

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 22

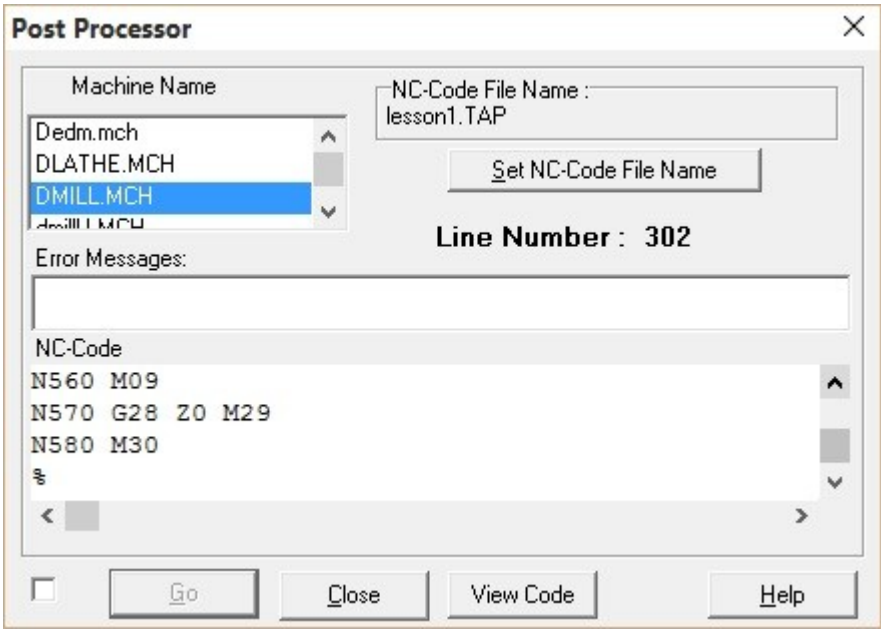
The final step in machining is to run the post processor. Select “Machining”, “Post (Generate NC-Code)” from the menu (or select the Post icon).

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 labeled [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.

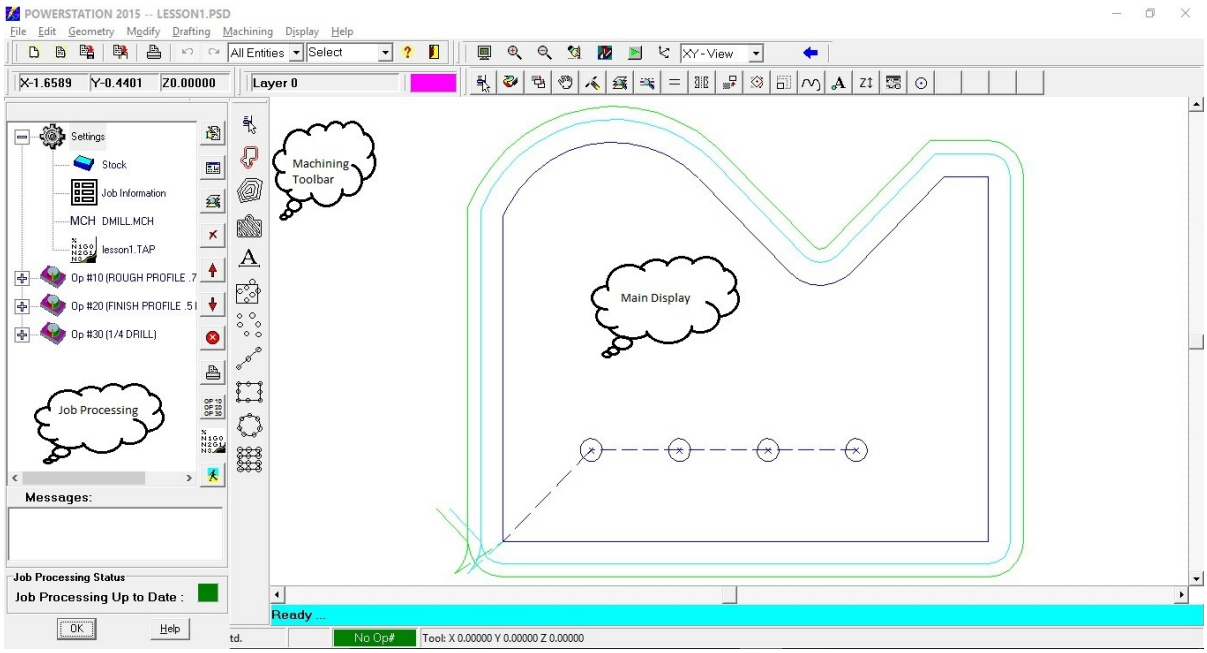
NOTE: You will need to add your own machine names to the post processor list. For information on add them, select “Help”, “Post Processor Setup”.



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

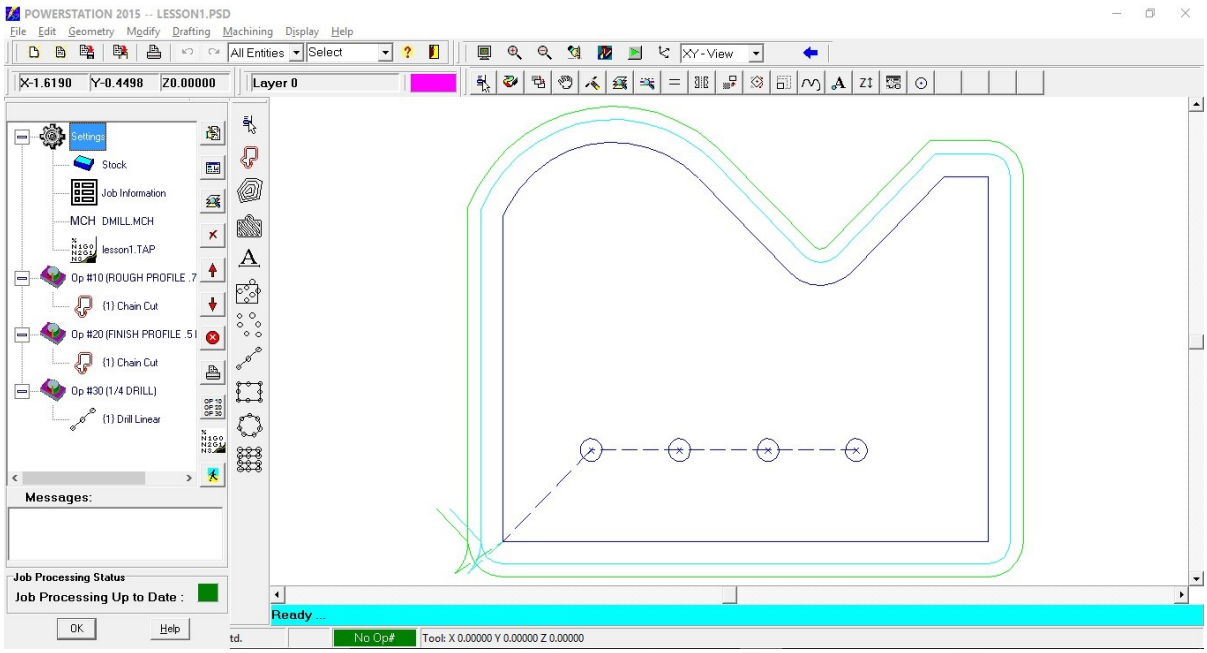
Step 23

Select “Machining-Job Processing”. The screen layout will change and following dialog will be displayed:



Step 24

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:

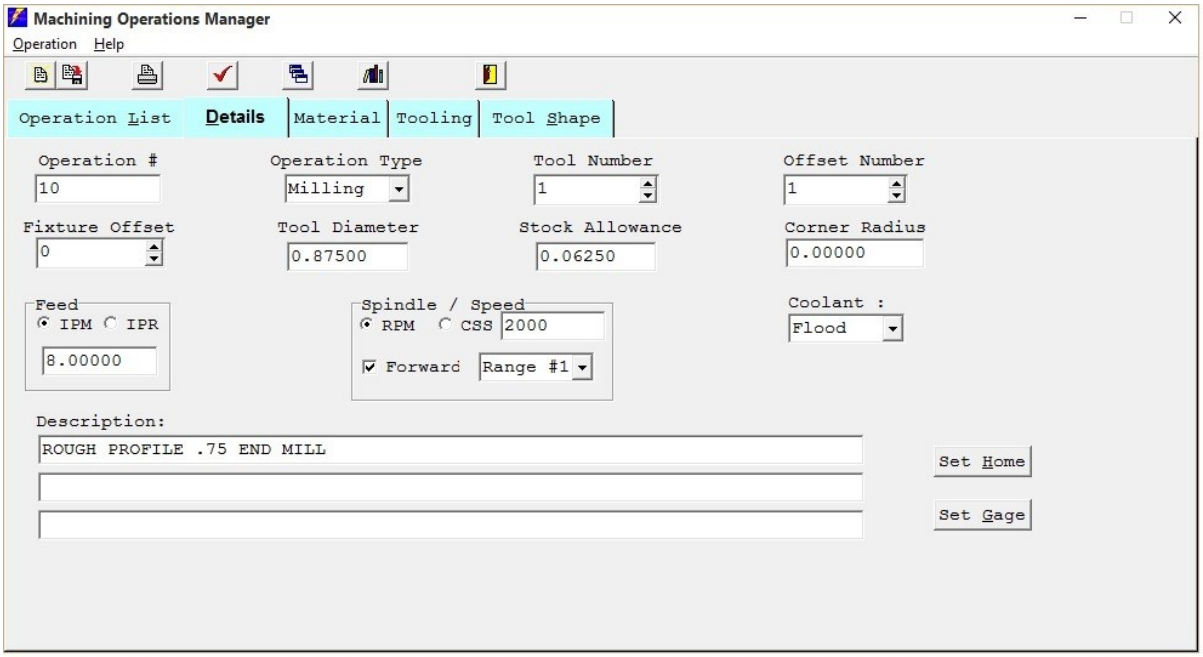


Step 25

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 -.75 .

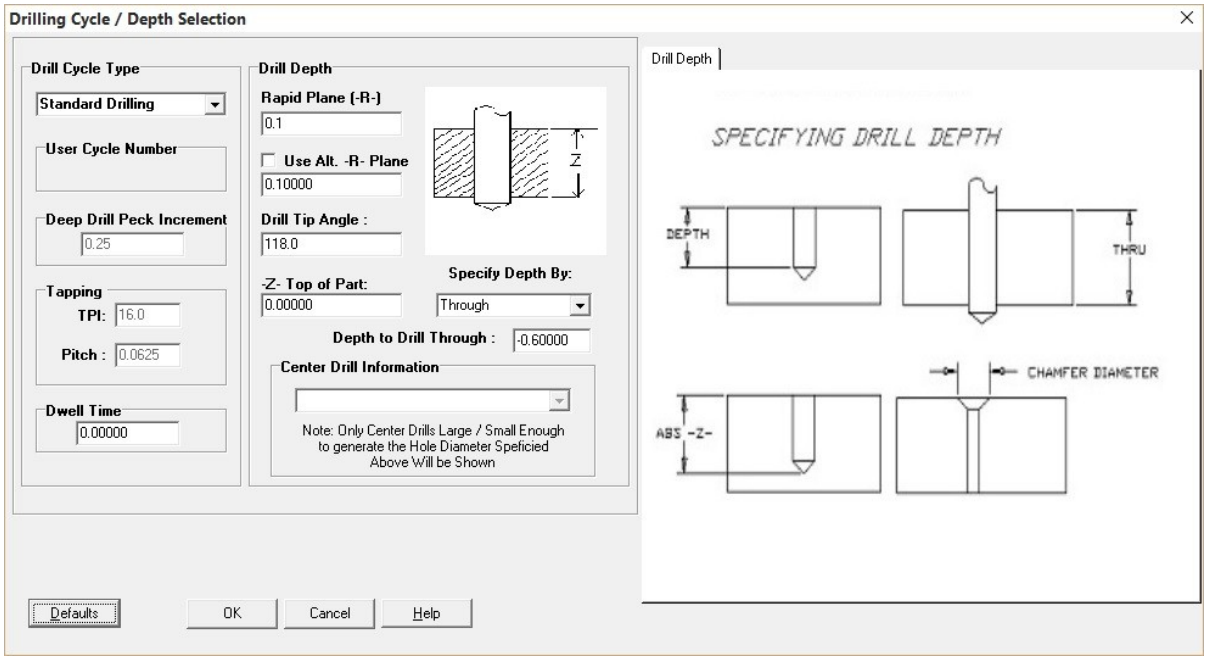
Step 26

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.



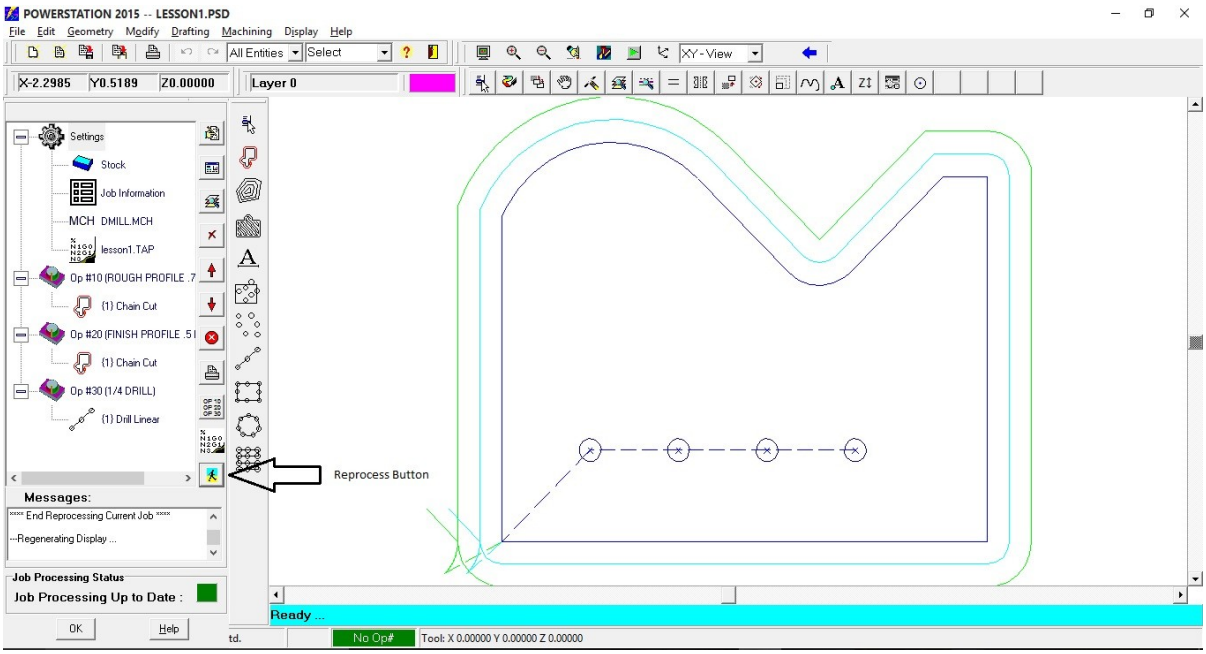
Step 27

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 -.75 to -.6, press [OK].



Step 28

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



Step 29

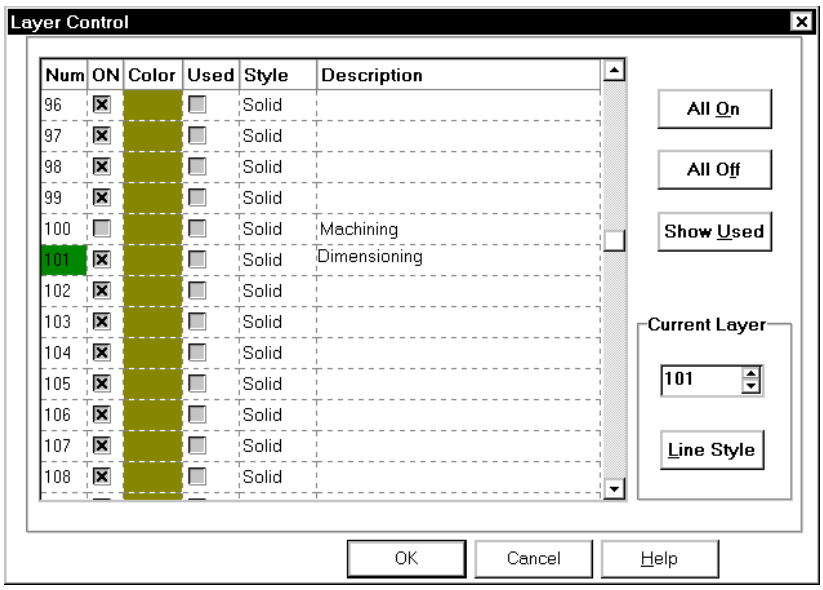
At this point, the new tool path is generated. To regenerate a new NC-Code file, simply repeat step #22 (Run the post processor).

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 30

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

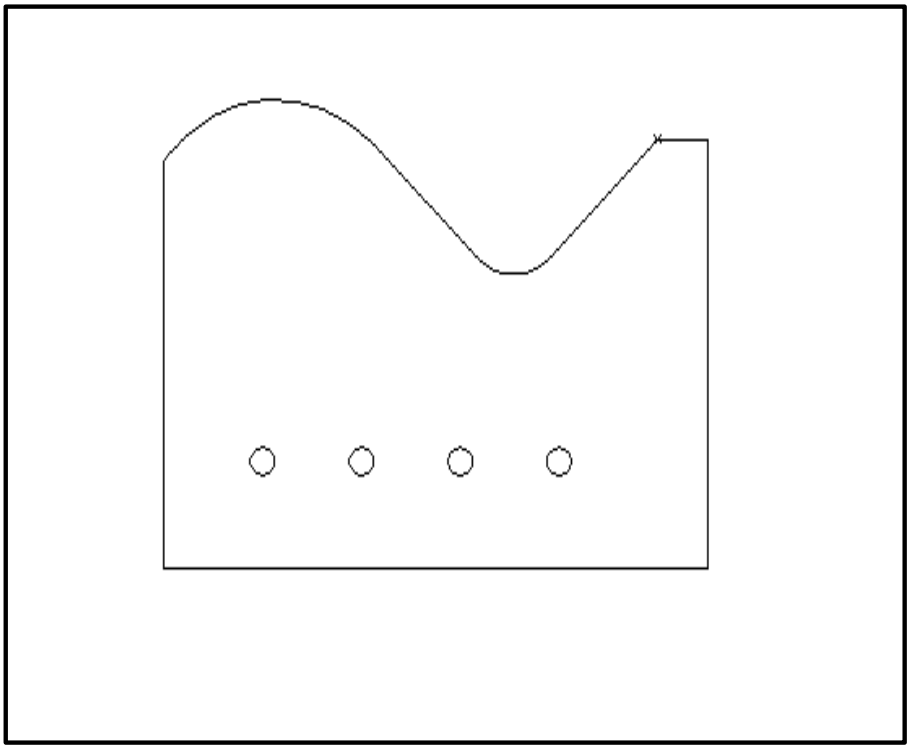


Figure 4-31 Display with the tool path layer off

STEP 31

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 32

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

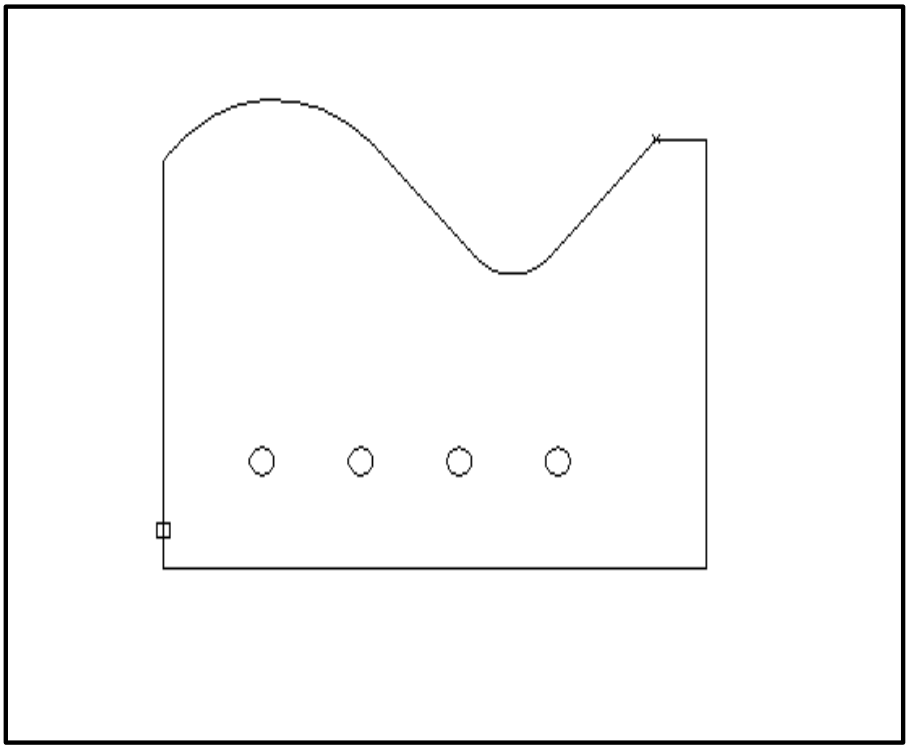


Figure 4-32 Extension line origin

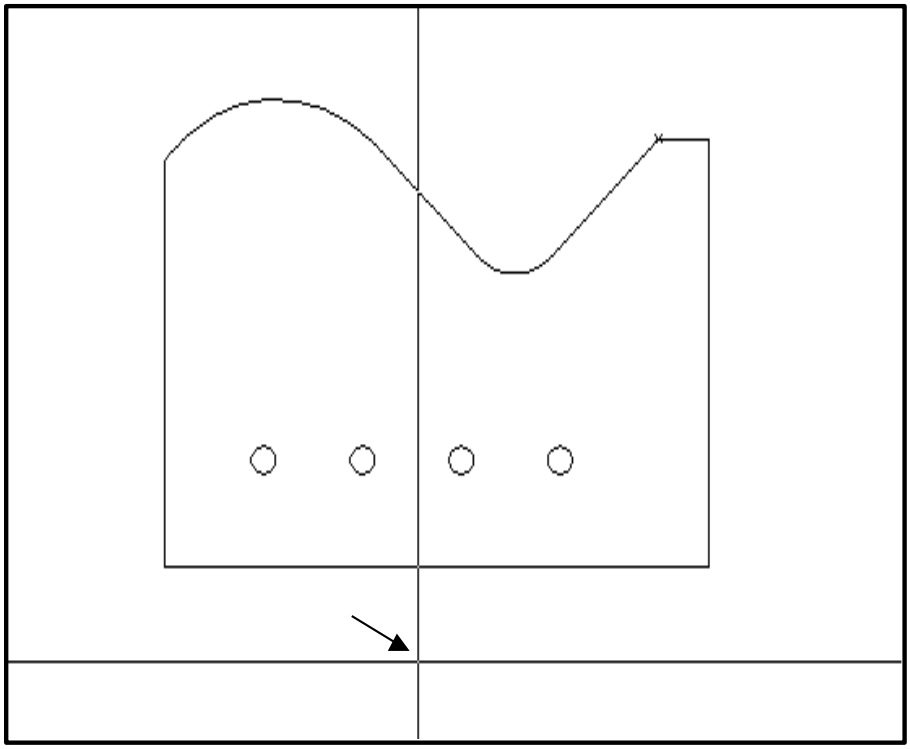


Figure 4-33 Indicating the approximate text point

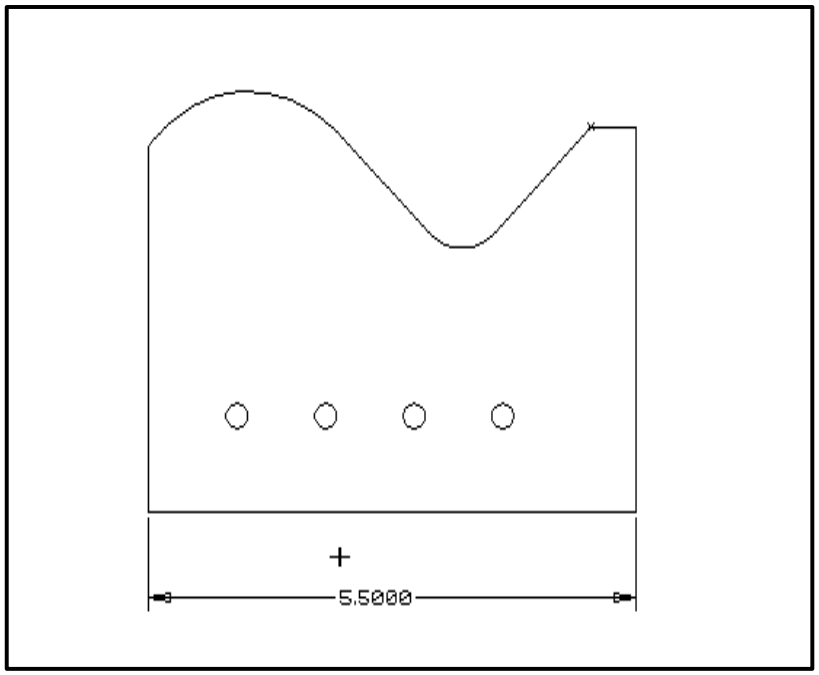


Figure 4-34 Dimensioning the overall width

STEP 33

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)

The dimension should look like figure 4-36

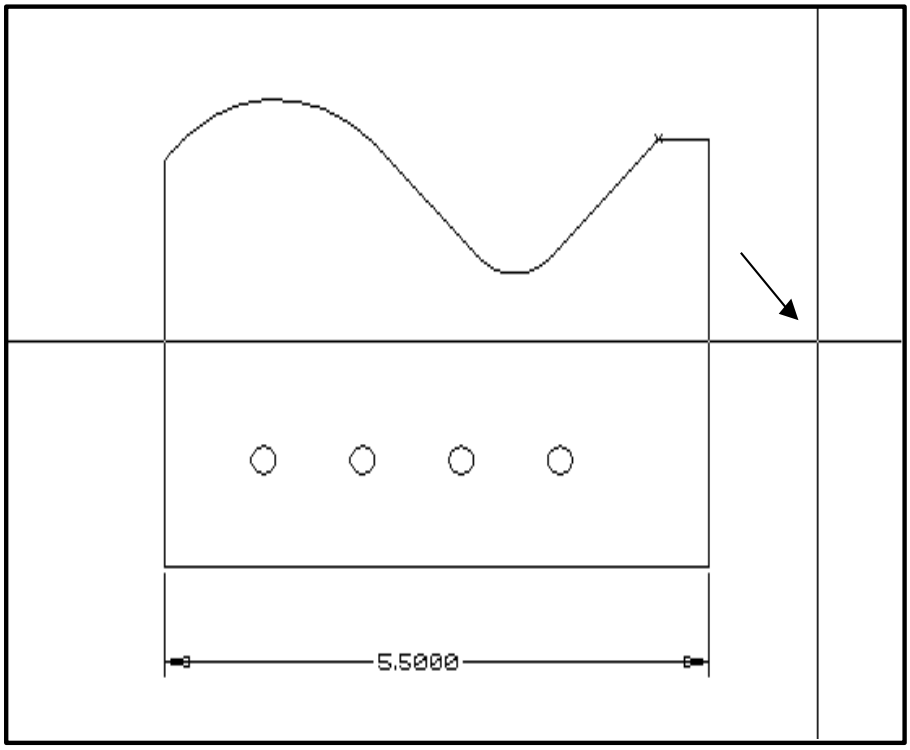


Figure 4-35 Constructing the vertical dimension

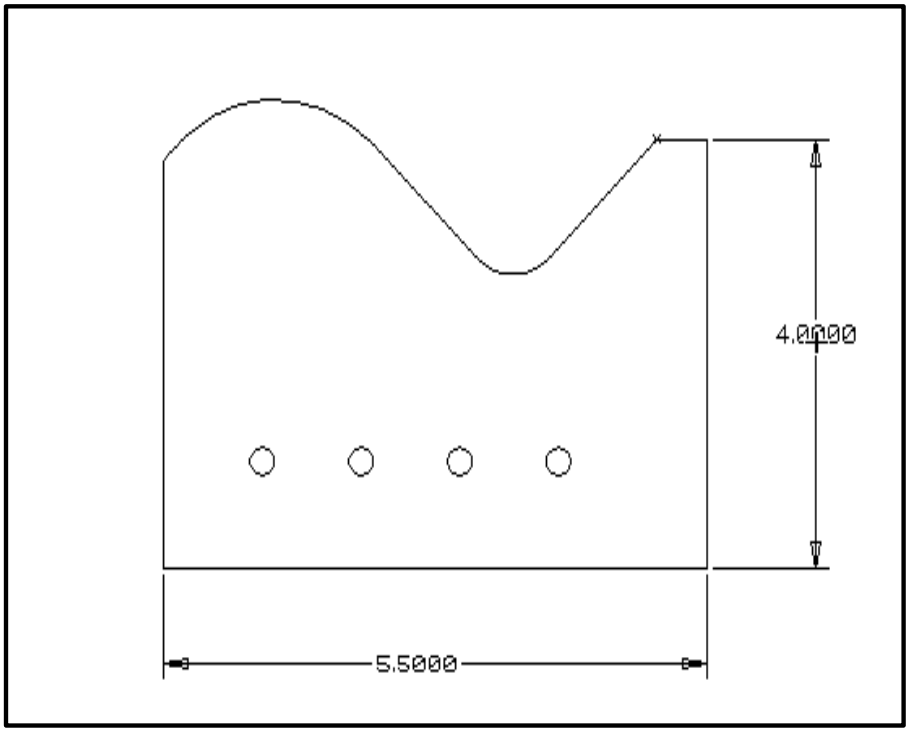


Figure 4-36 Dimensioning the overall height

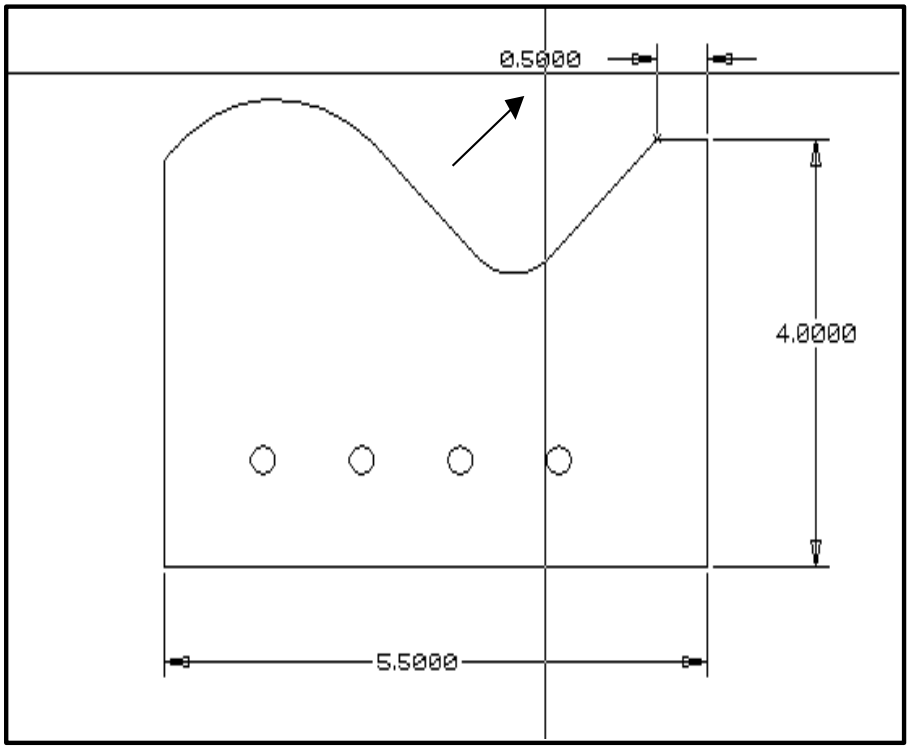


Figure 4-37 Dimensioning the top most line

STEP 34

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

STEP 35

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 36

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

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

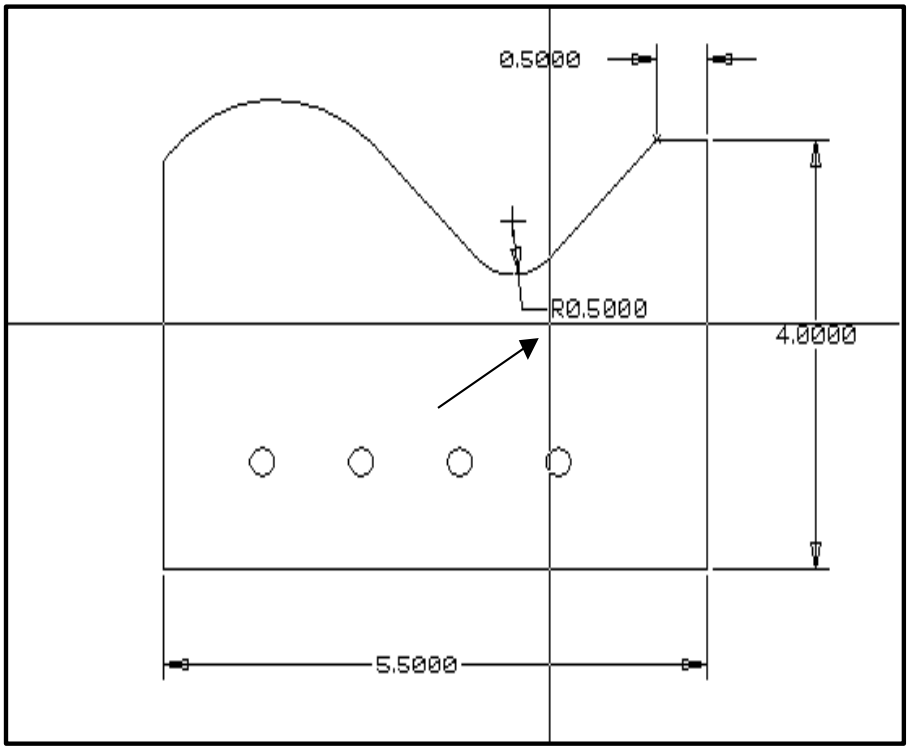


Figure 4-38 Dimensioning the 1/2" radius

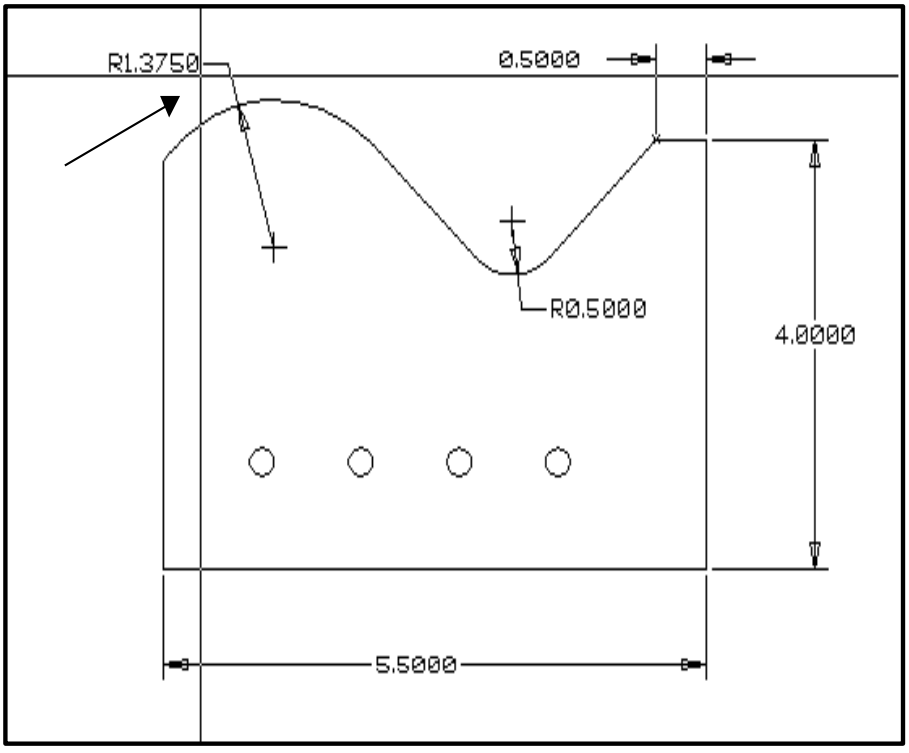


Figure 4-39 Dimensioning the 1 3/8" radius

STEP 37

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

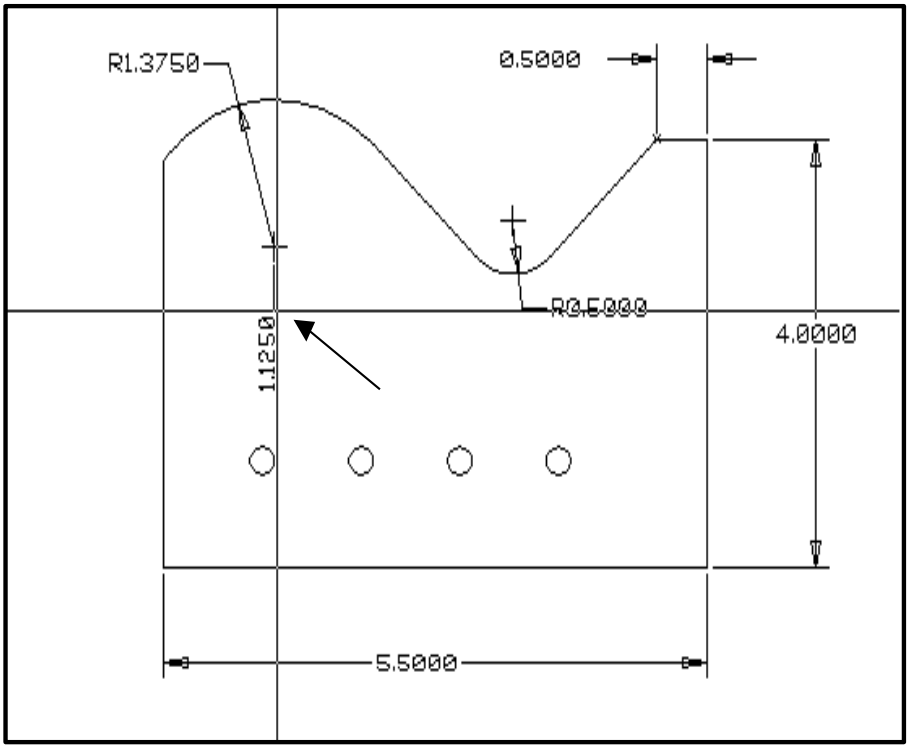


Figure 4-40 Dimensioning the X center of the 1 3/8" radius

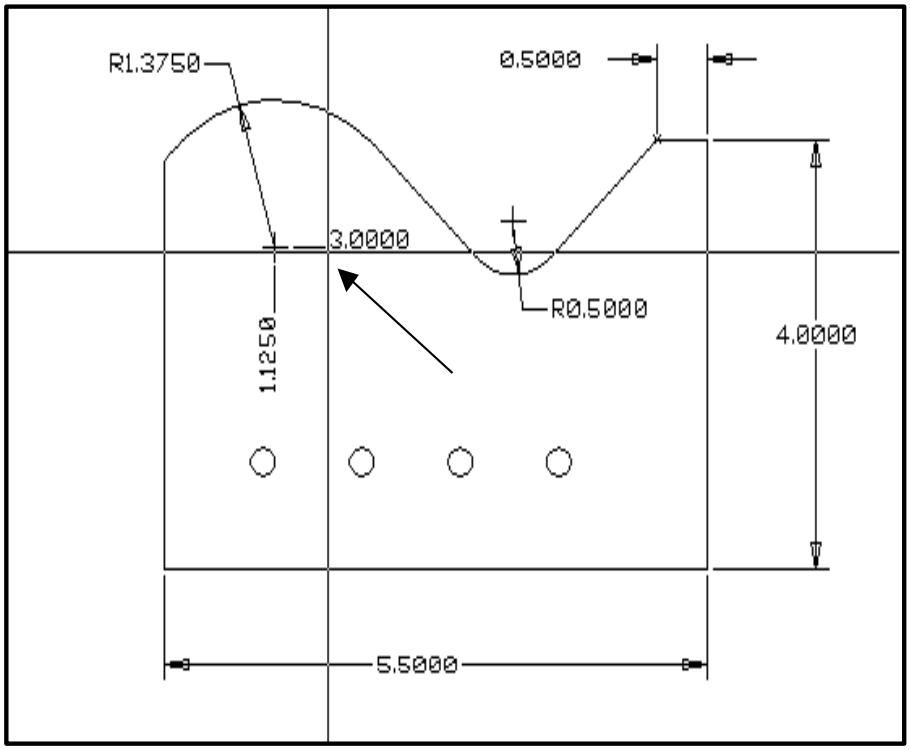


Figure 4-41 Dimensioning the Y center of the 1 3/8" radius

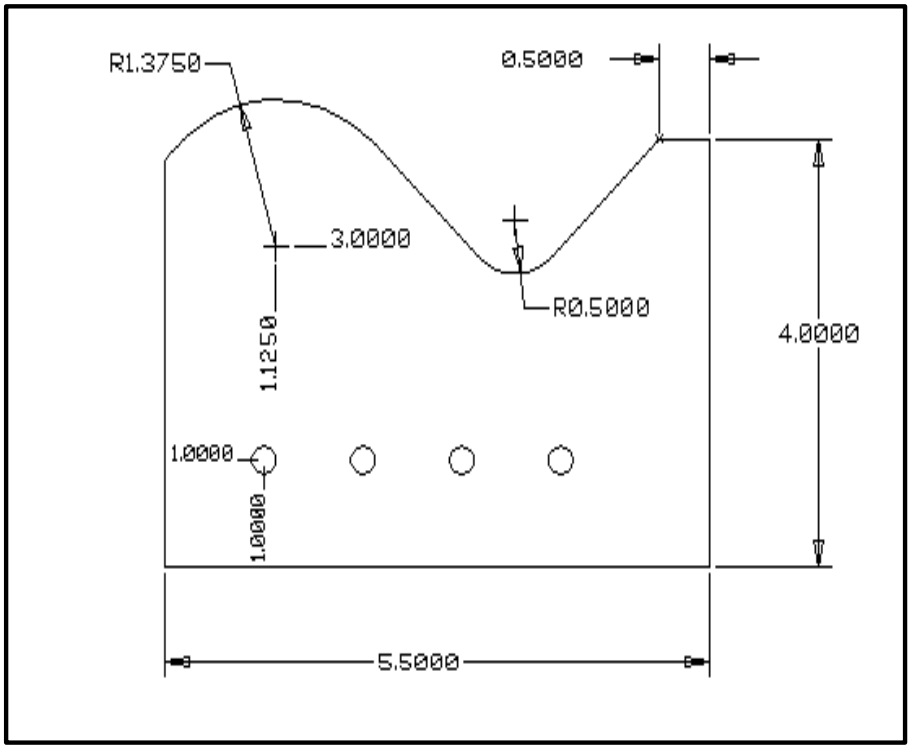


Figure 4-42 Ordinate dimensions

STEP 38

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

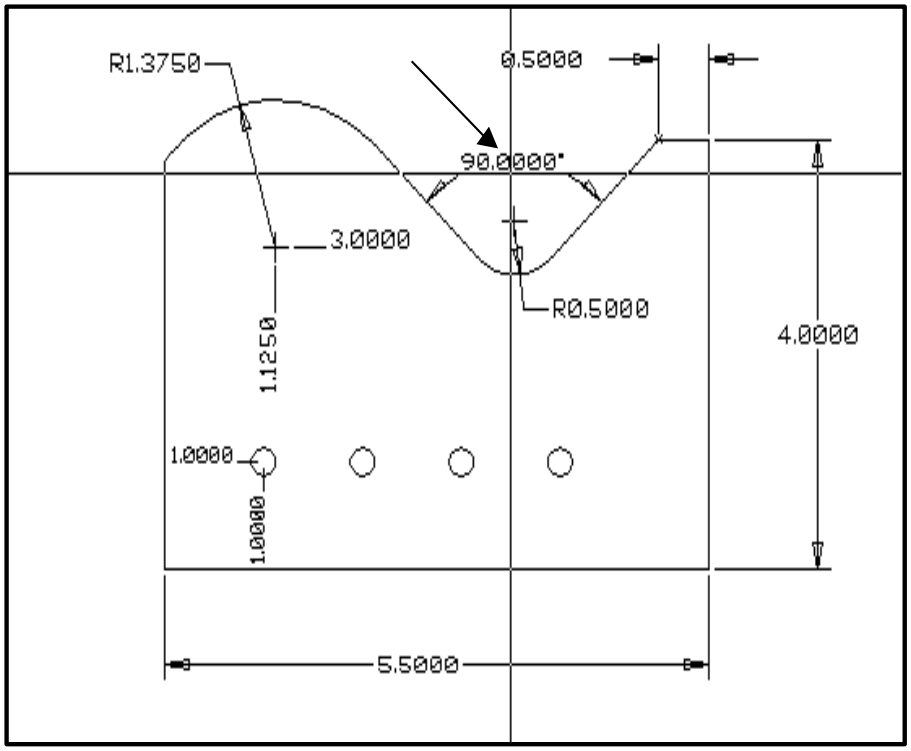


Figure 4-43 Constructing the angular dimension

STEP 39

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

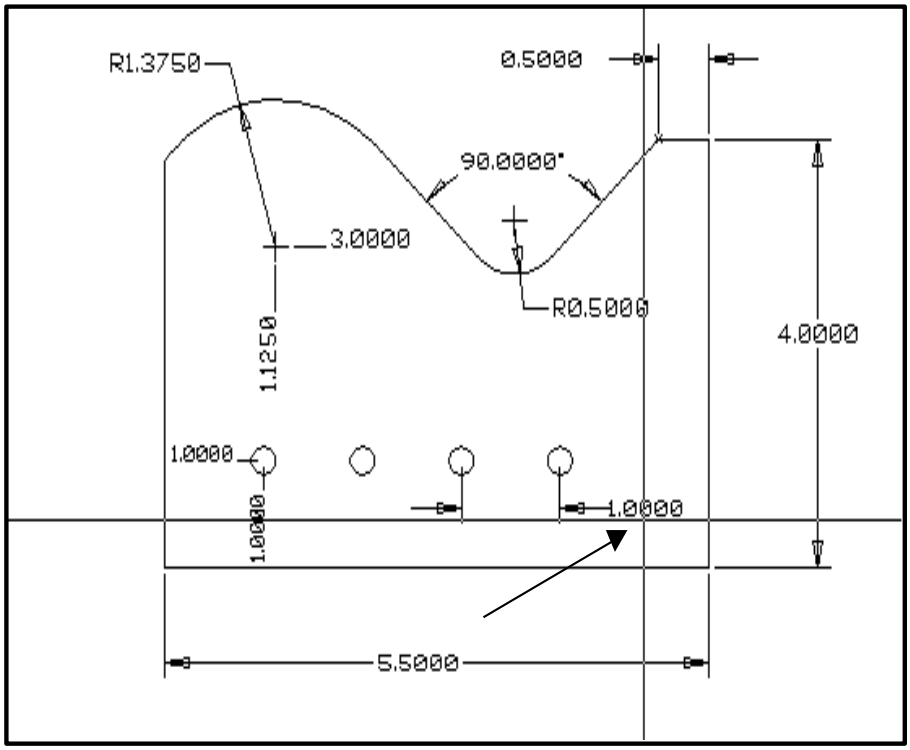
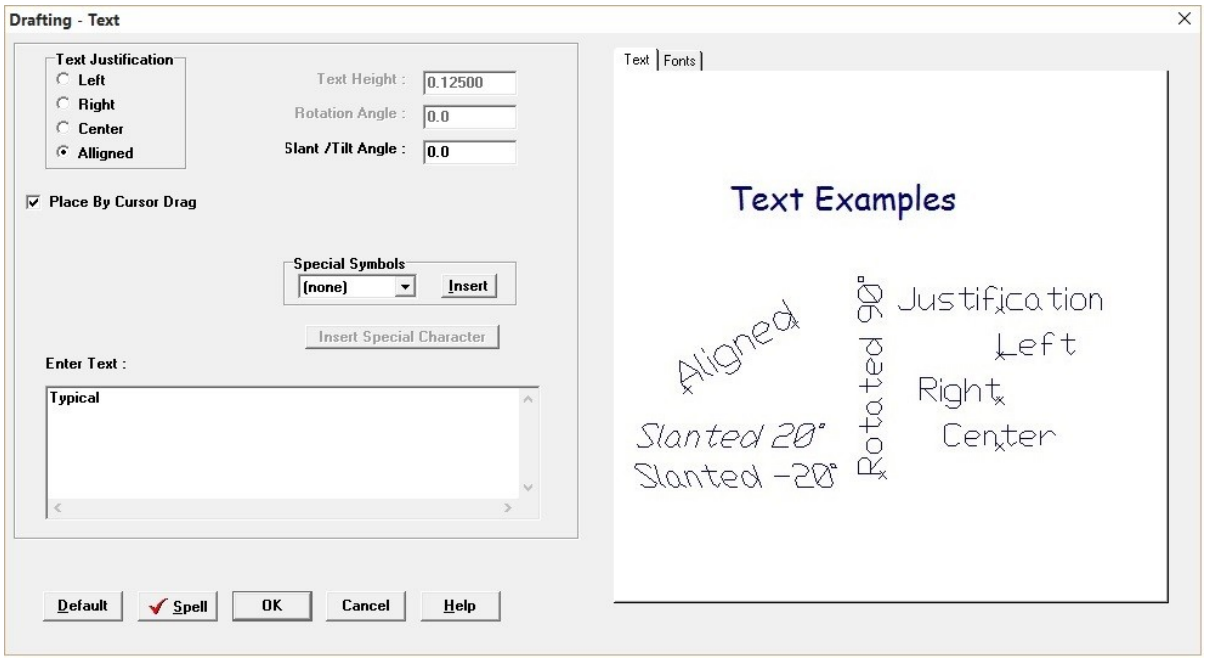


Figure 4-44 Dimensioning the distance between holes

STEP 40

Now we will add the text "typical" under the last dimension. Select **"TEXT"** from the sub menu.



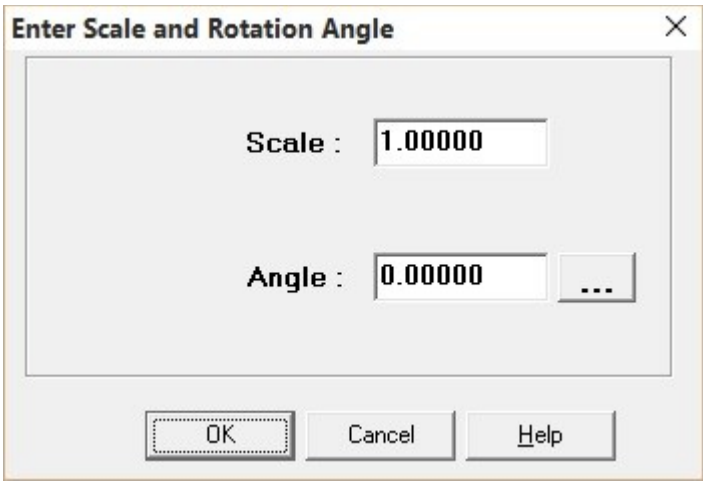
Next enter the approximate text start and end points as shown in fig 4-45

STEP 41

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- **2.5**
-Y- **-4.5**



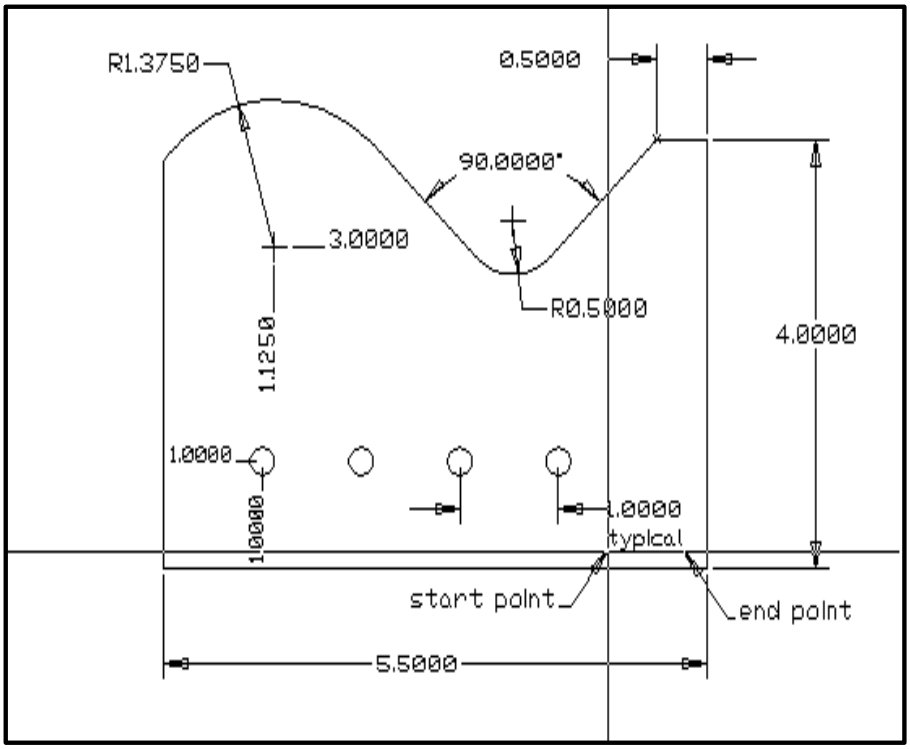


Figure 4-45 Adding text

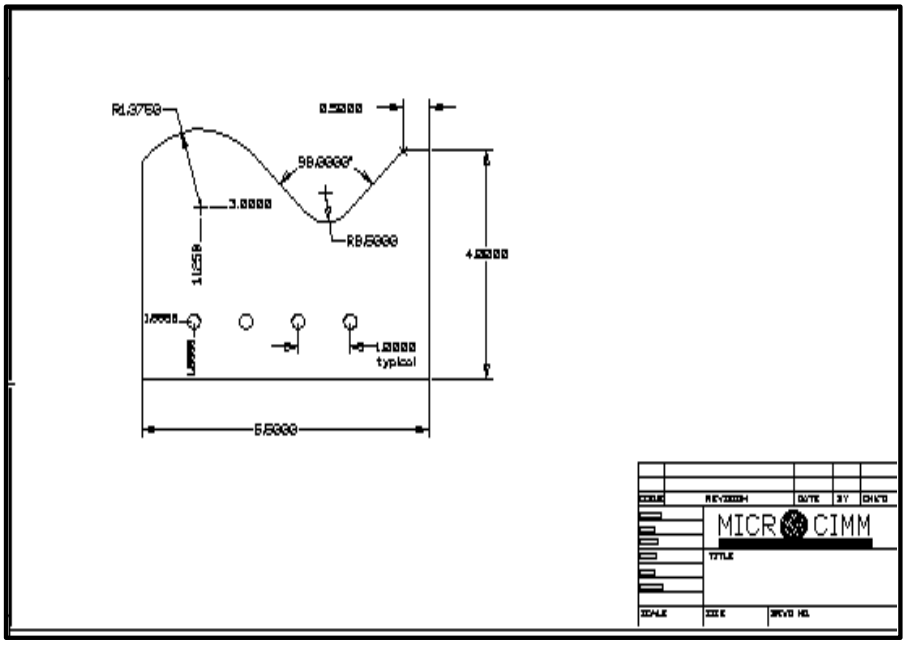


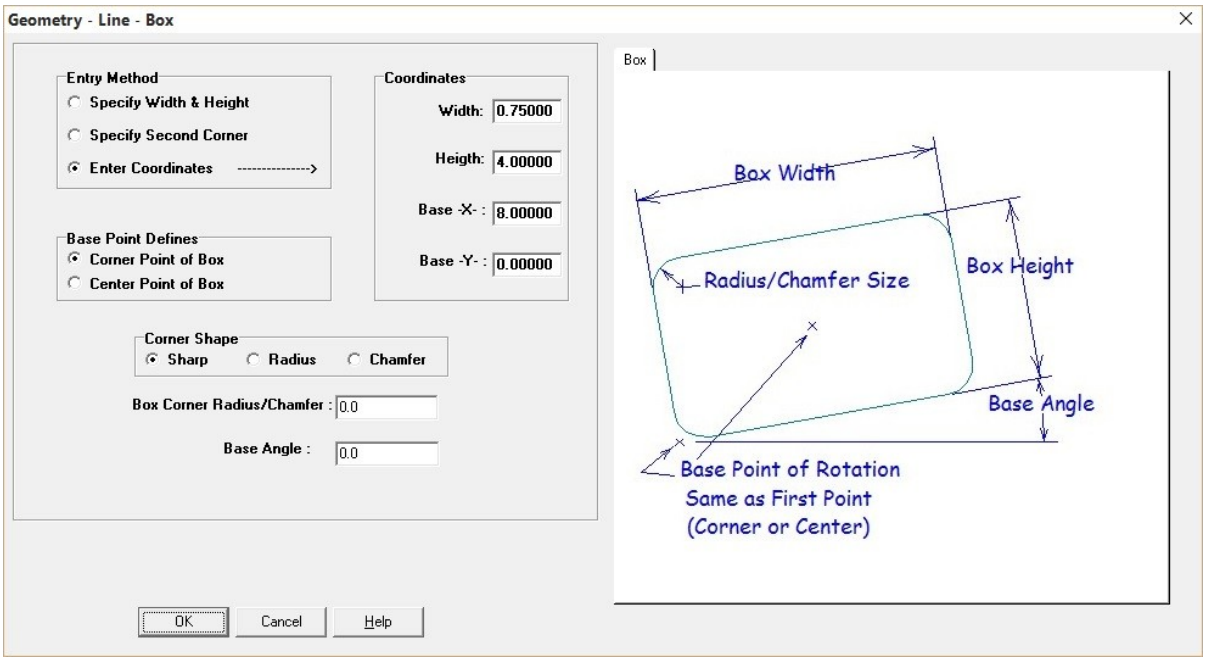
Figure 4-46 Title block and sheet frame

STEP 41 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 42

Now let's generate a side view. Select “Geometry”, “Shapes”, “Box” from the menu.



The display should now look like figure 4-47

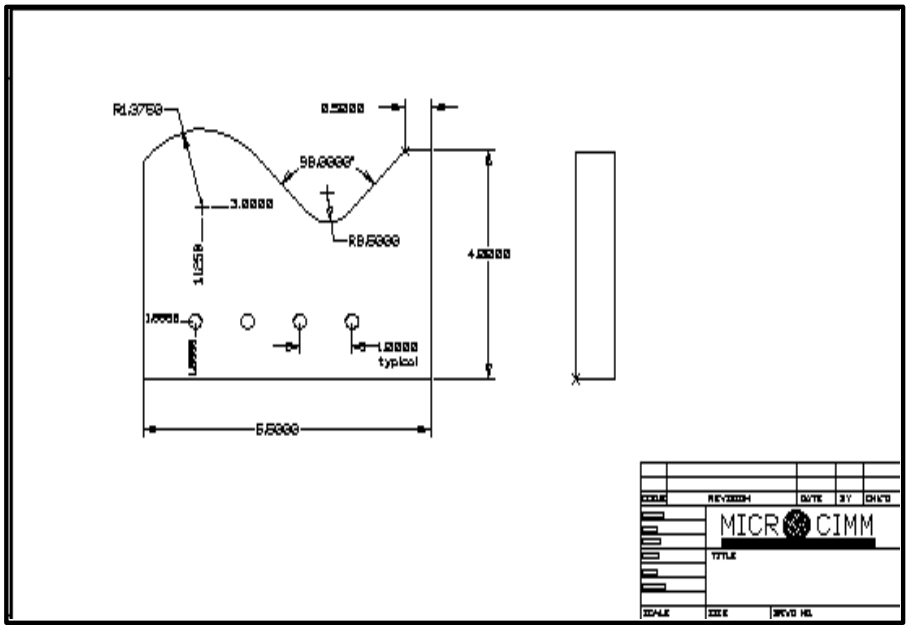


Figure 4-47 Adding a side view

STEP 43

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 44

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.

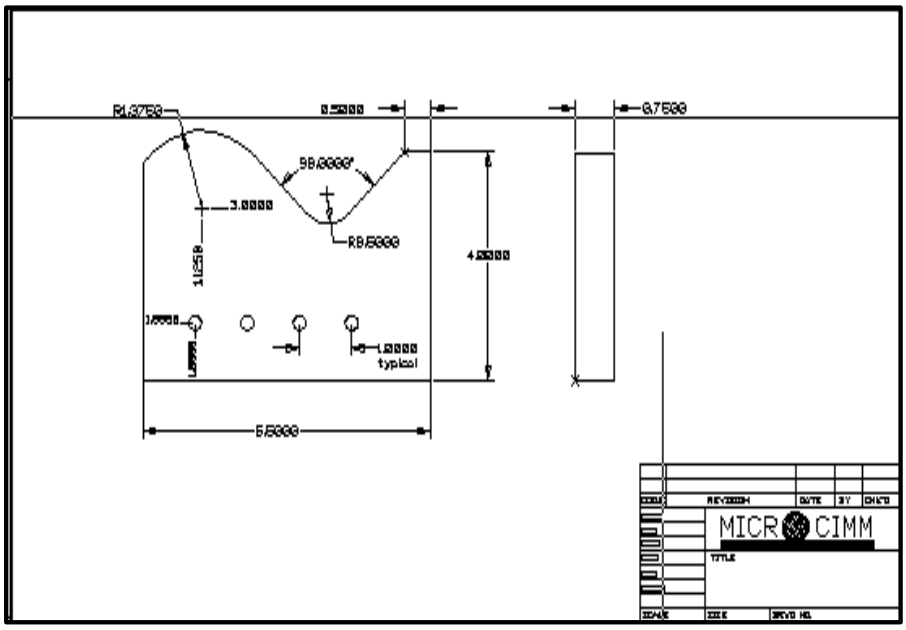


Figure 4-48 Dimensioning the part thickness

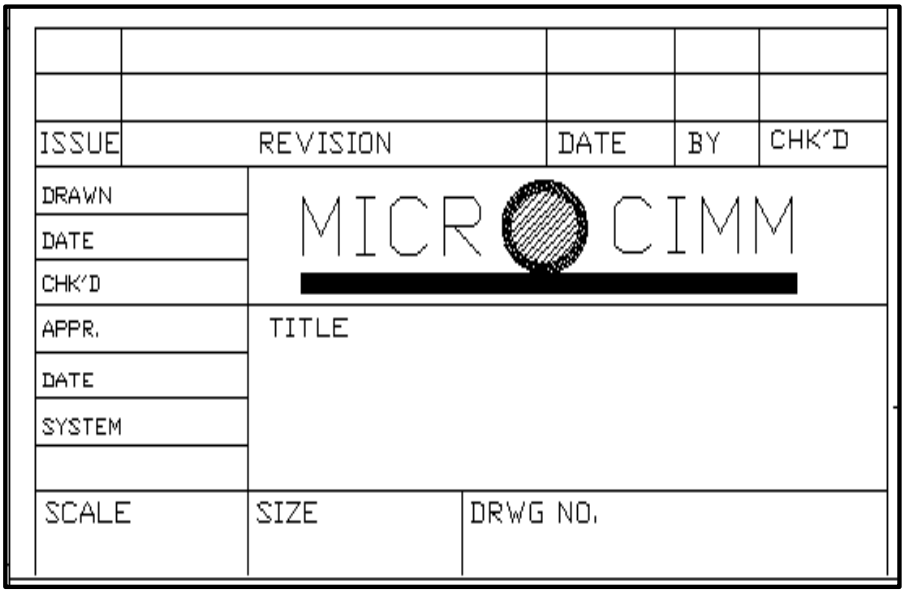
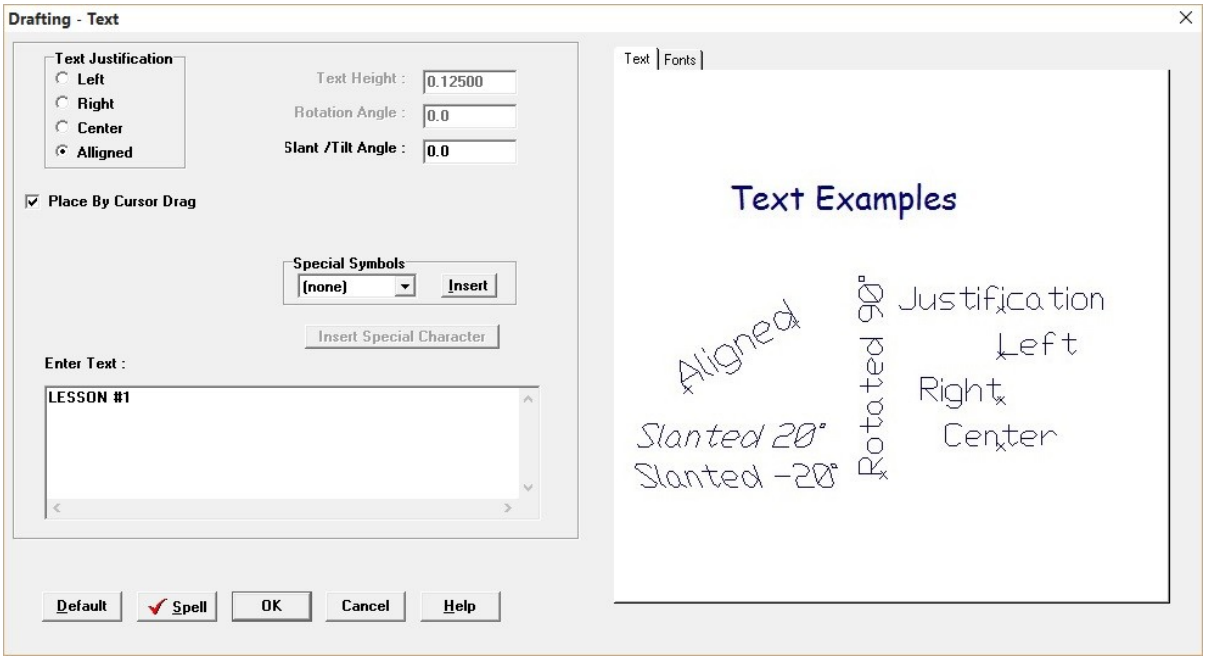


Figure 4-49 Zooming in on the title block

STEP 44 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 45

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

STEP 46

Save the drawing as in step 15.

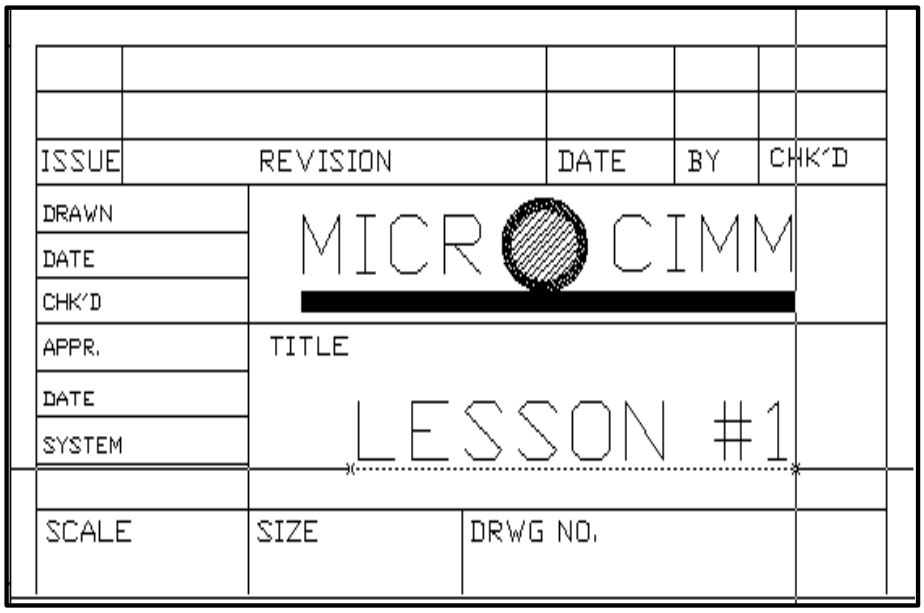


Figure 4-50 Adding the drawing name

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 47

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

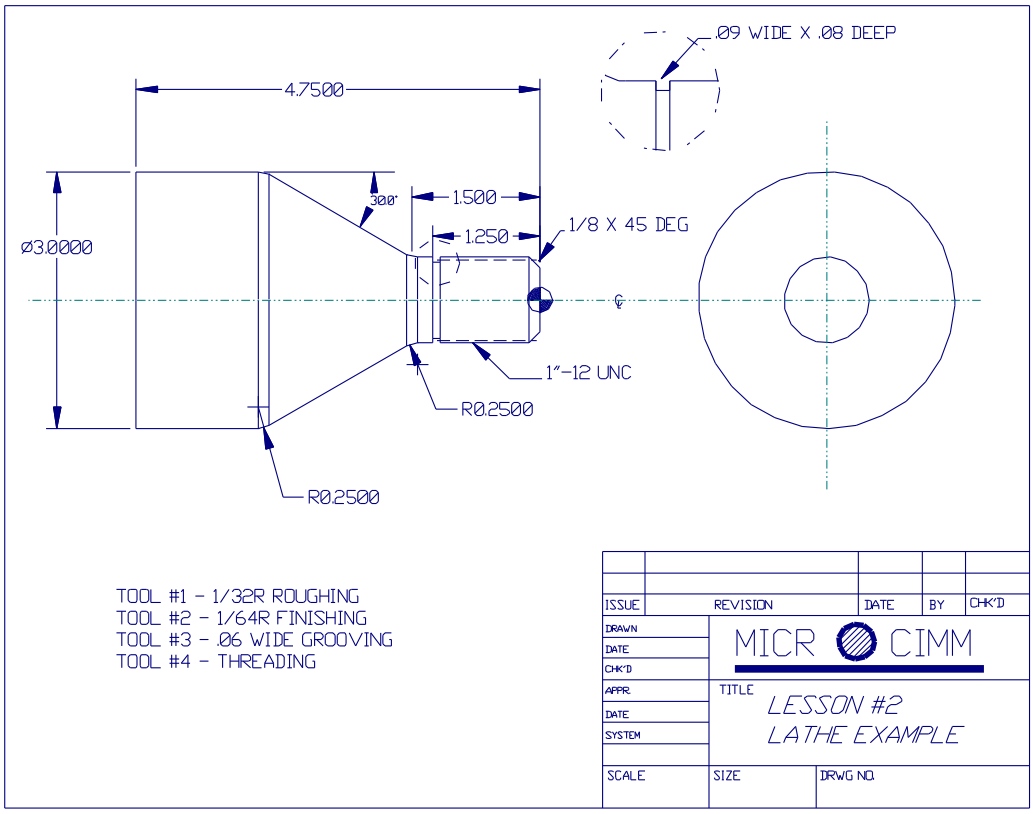


Figure 4-60 The Sample Lathe Drawing

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.

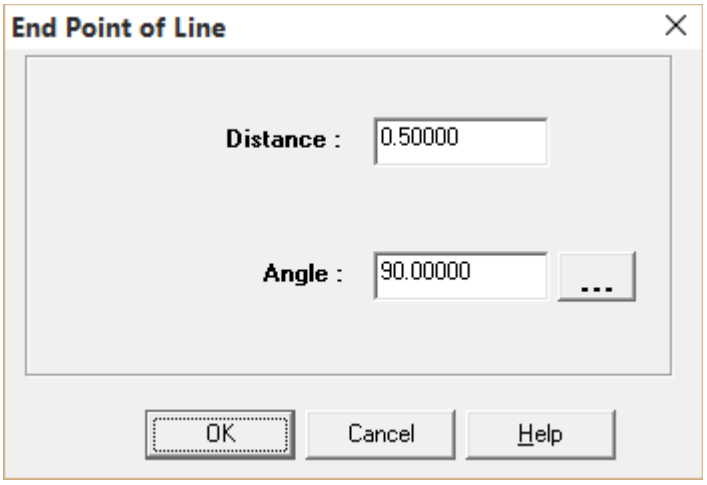
STEP 1

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

Note: *The following steps show one of many "long Hand" methods 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 the following steps, please take a look at step 10 it shows how to generate the same part geometry using far fewer steps, using the built in "Easy Geometry" command.*

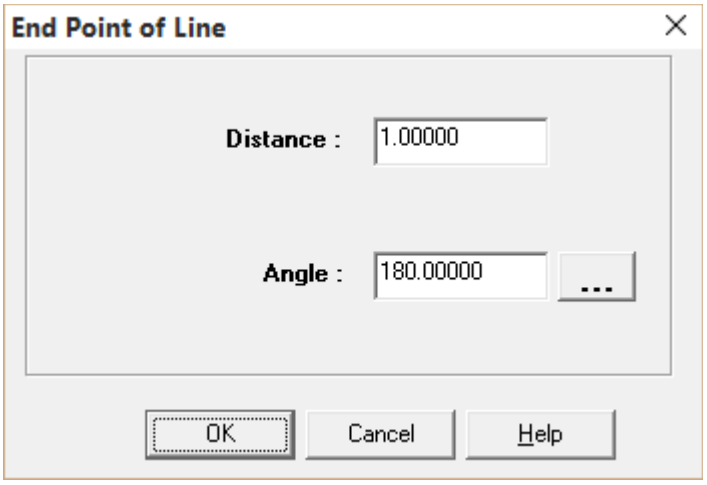
STEP 2

From the menu, select “Geometry-Line-Polar” The “Get Point” menu will be displayed, select “Origin (X/Y Zero)”. Next enter the polar coordinates for the end of the line. Note that the distance is .5 or ½ of the diameter.



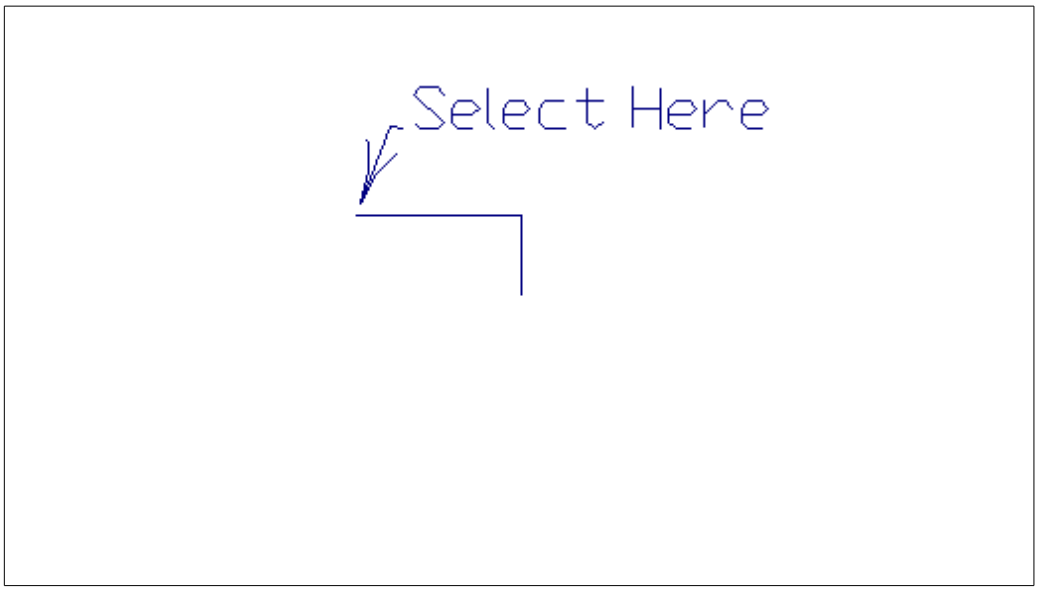
STEP 3

From the menu, select “Geometry-Line-Polar” The “Get Point” menu will be displayed, select “End of”, and select near the end of the line drawn in step 2.. Next enter the polar coordinates for the end of the line.

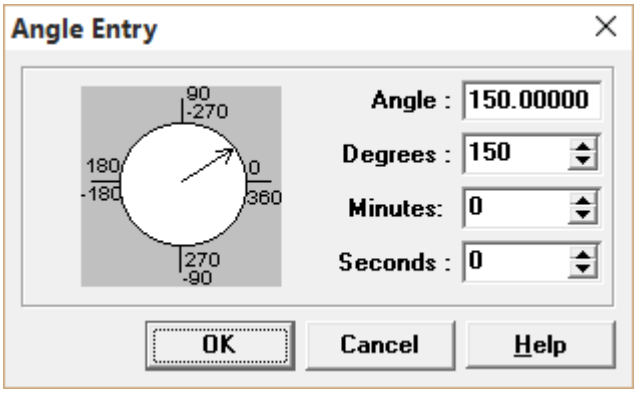


STEP 4

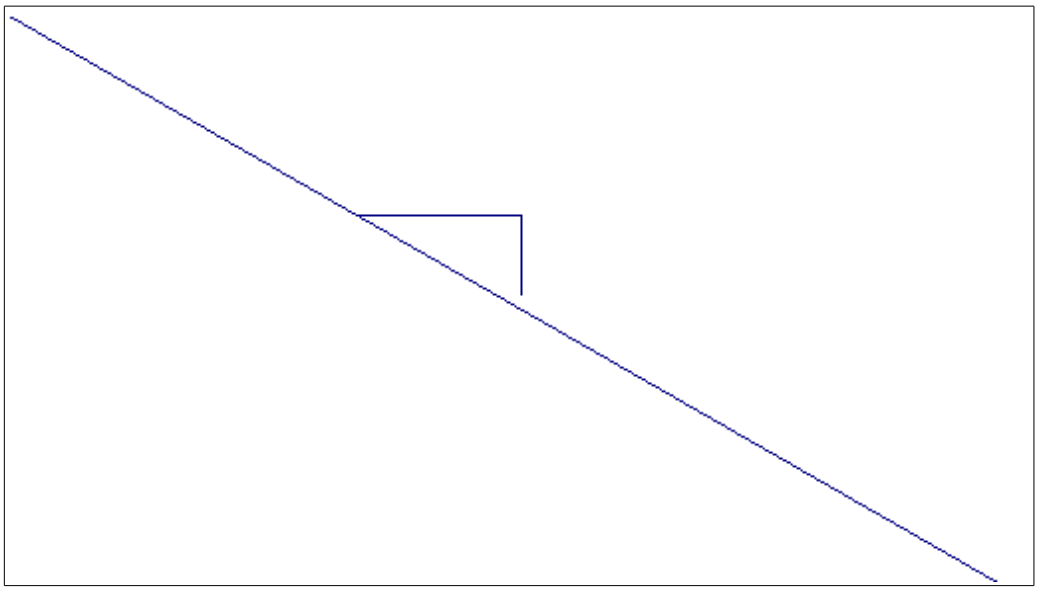
From the menu select “Display-Zoom In” to make the geometry a bit easier to see. Next from the menu, select “Geometry-Line-Tangent to One” The system will display “Select a Point or Arc, or <Escape> For Point Menu”. The “Get Point” menu will be displayed, select “End of”, and select near the end of the line drawn in step 3..



Next you will be asked to enter the angle of the line (150 or -30 are both correct):

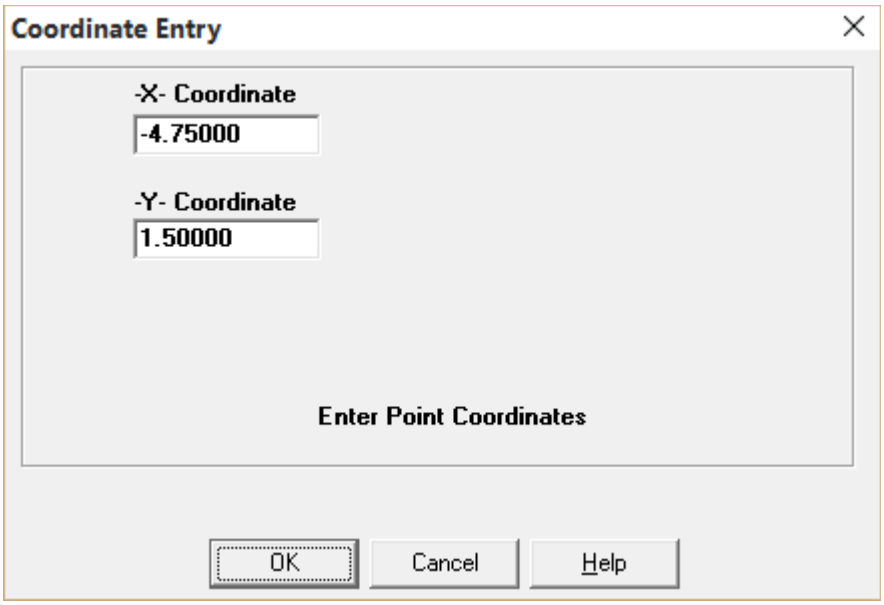


The display should look like this:



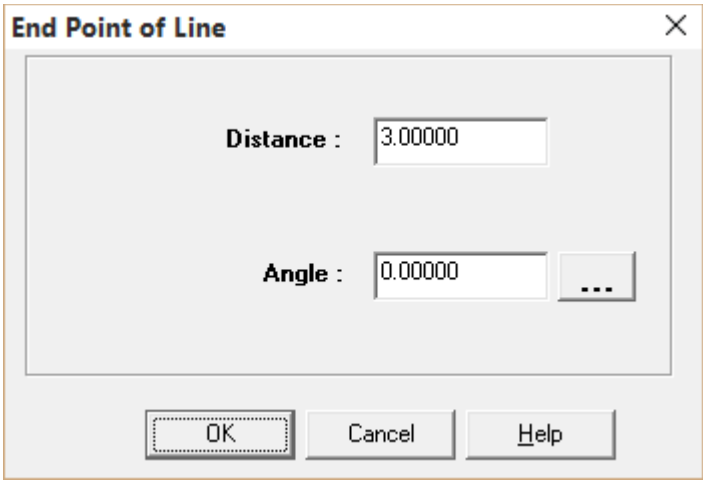
STEP 5

From the menu, select “Geometry-Line-Polar” The “Get Point” menu will be displayed, select “Coordinates”, and enter the following:

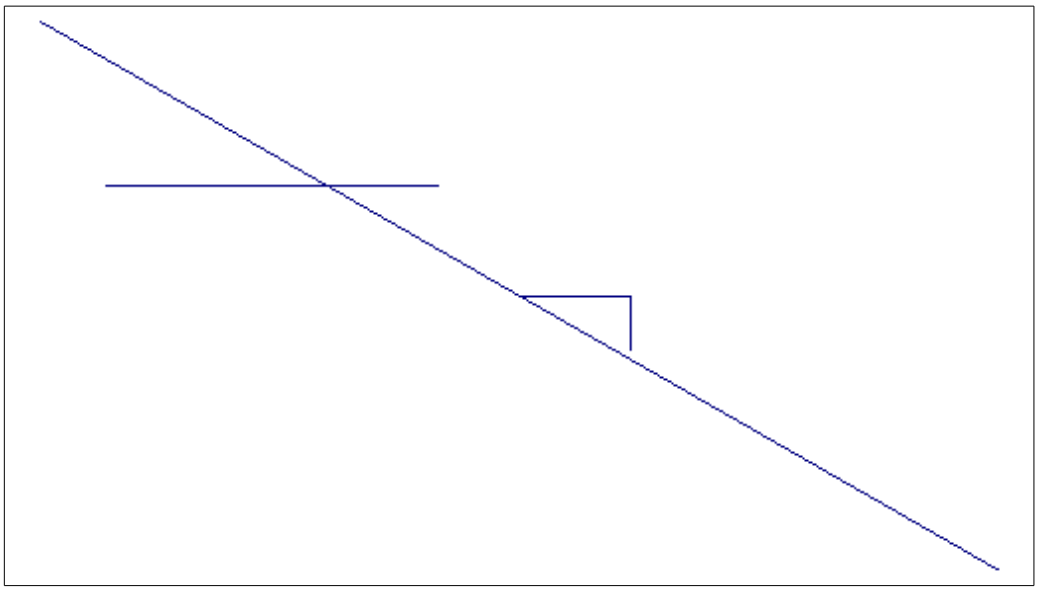


Notice that the -Y- is the diameter divided by 2.

Next enter the Polar coordinates:

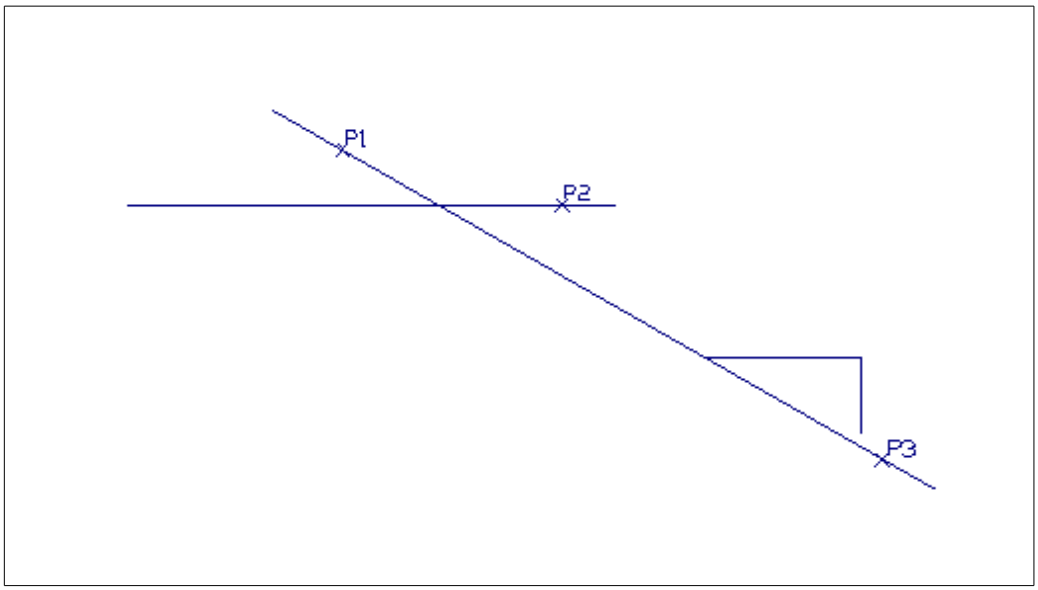


Select “Display-Zoom Extents” and the display should look like the following:



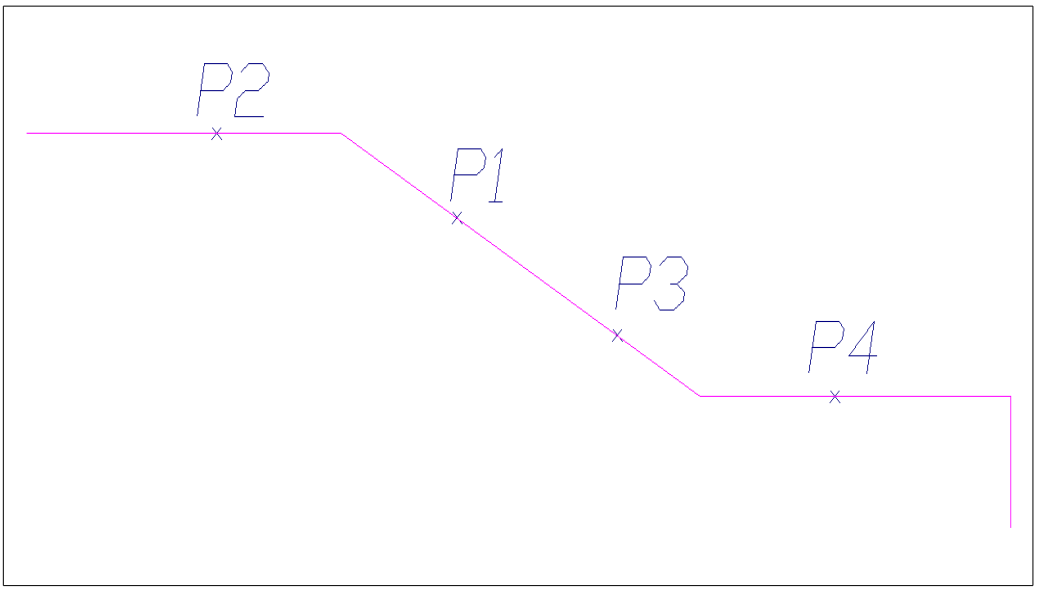
STEP 6

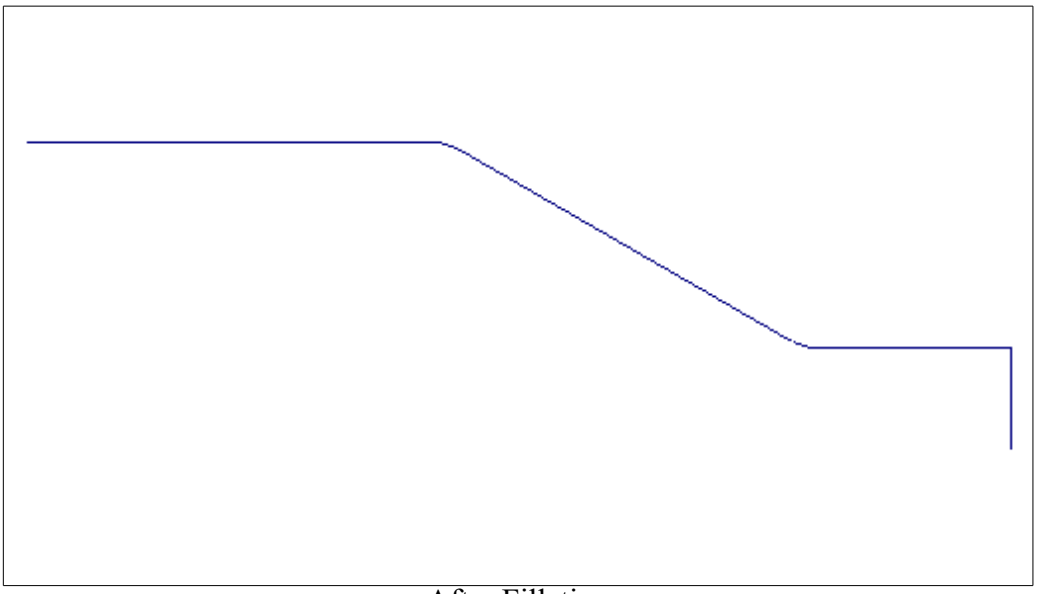
Next we will neaten up the geometry by trimming it. From the menu, select “Modify-Trimming-Easy Trim”. Clean up the geometry by selecting it at the three points indicated:



STEP 7

Next we will insert the two fillet radii. From the menu, select “Modify-Trimming-Fillet” You will be asked to select the two lines (P1 & P2 below) and then enter the radius (0.25). Repeat using points P3 and P4.

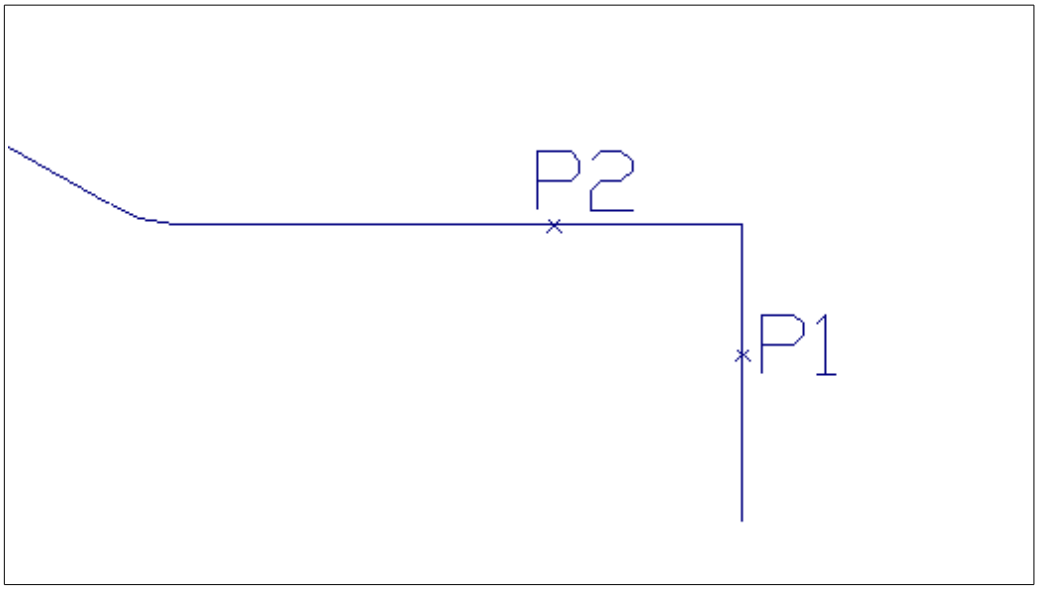




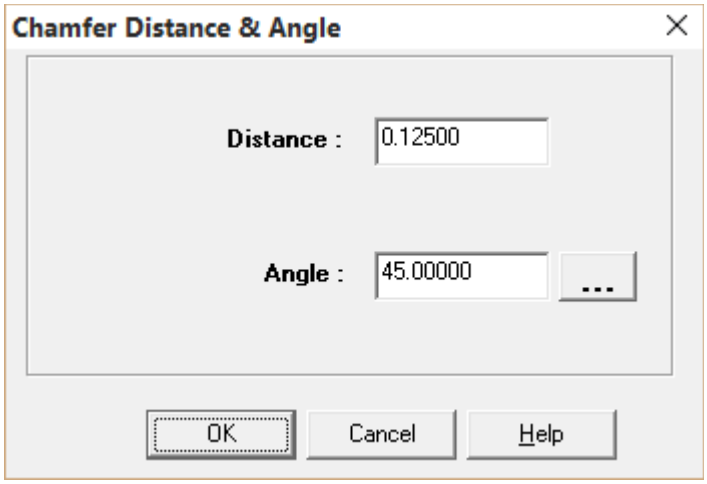
After Filleting

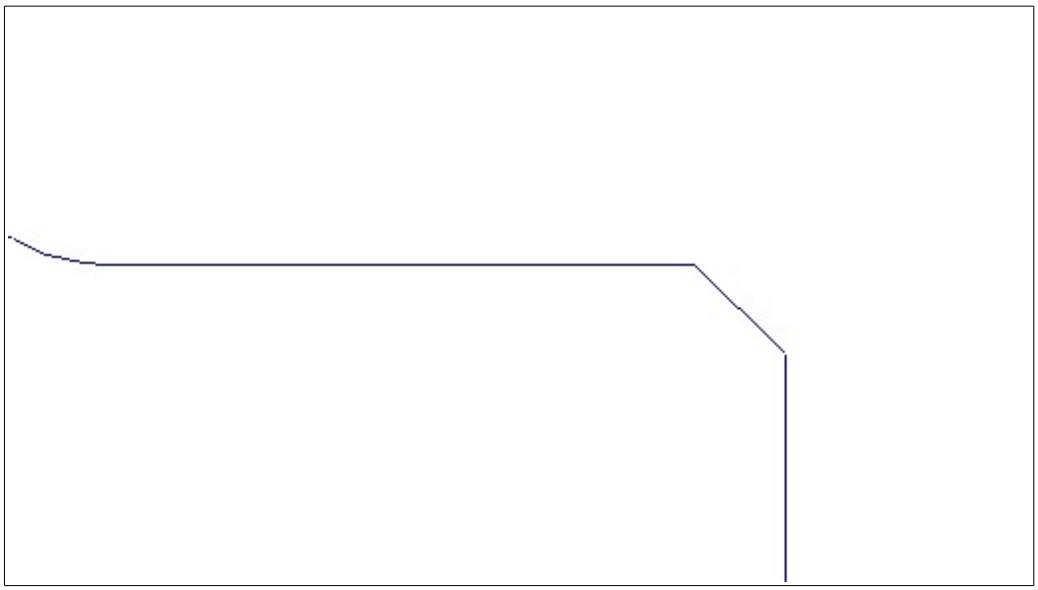
STEP 8

Finally we will add the Chamfer on the face. From the menu select “Modify-Trimming-Chamfer”. Select the line on the face (P1), and then the 1” diameter (P2).



Next enter the Chamfer distance & angle.





STEP 9

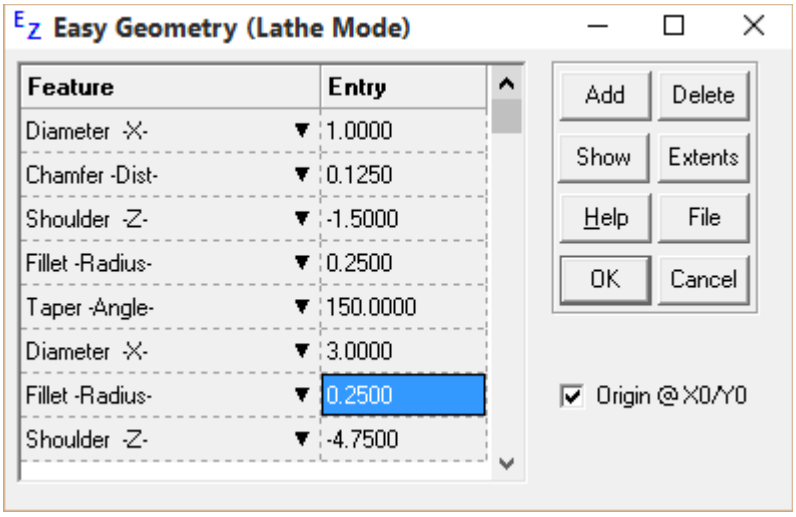
To be safe, lets save the file. From the menu, select “File-Save As” and enter “Lesson2”

STEP 10

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 2-8 as previously outlined).

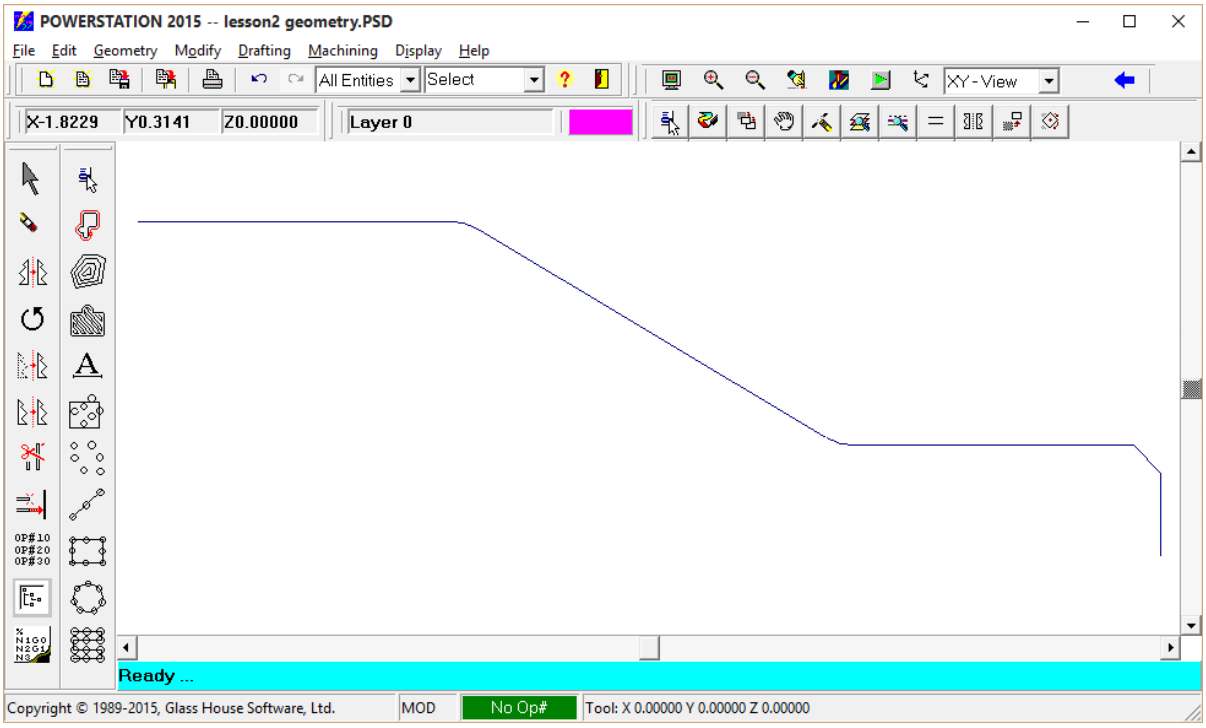
Select "Geometry-Easy-Geometry-Lathe Geometry" from the menu.

Enter the following settings into the Easy Geometry Dialog (Note, be sure to move the highlight to a different (any) other field after entering information into the "Entry" column. If you compare these entries to the sample drawing you will see that you can make quick work out of basic turning features.



STEP 11

No matter how you got here, this is what the geometry should look like at this point.



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.

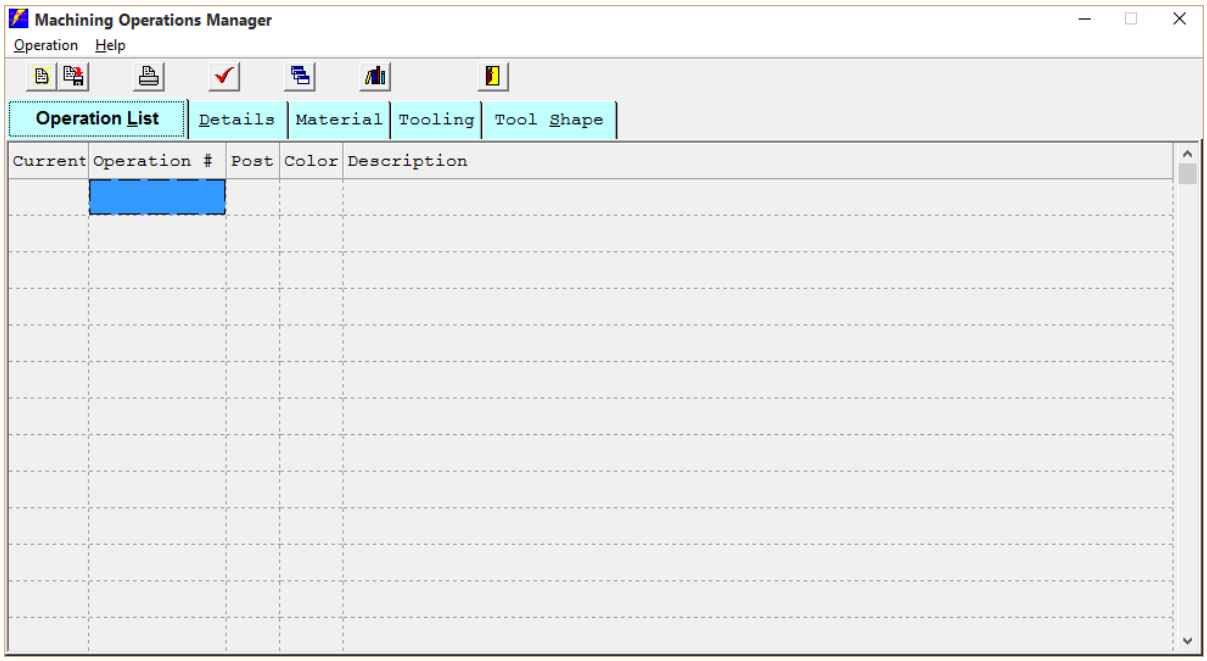


Figure 4-89a (The Machining Operations Manager)

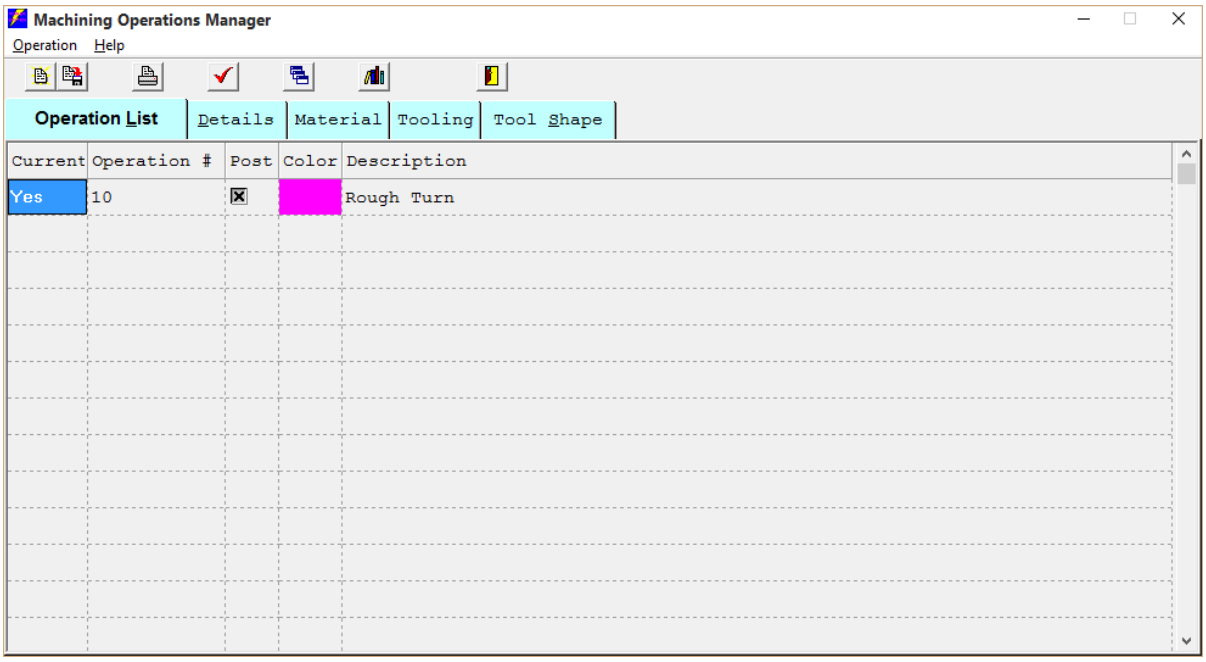


Figure 4-89b (Basic operation information)

To enter an operation:

1) Click the cursor in the “Op #” column, and type in the number of the operation (10).
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

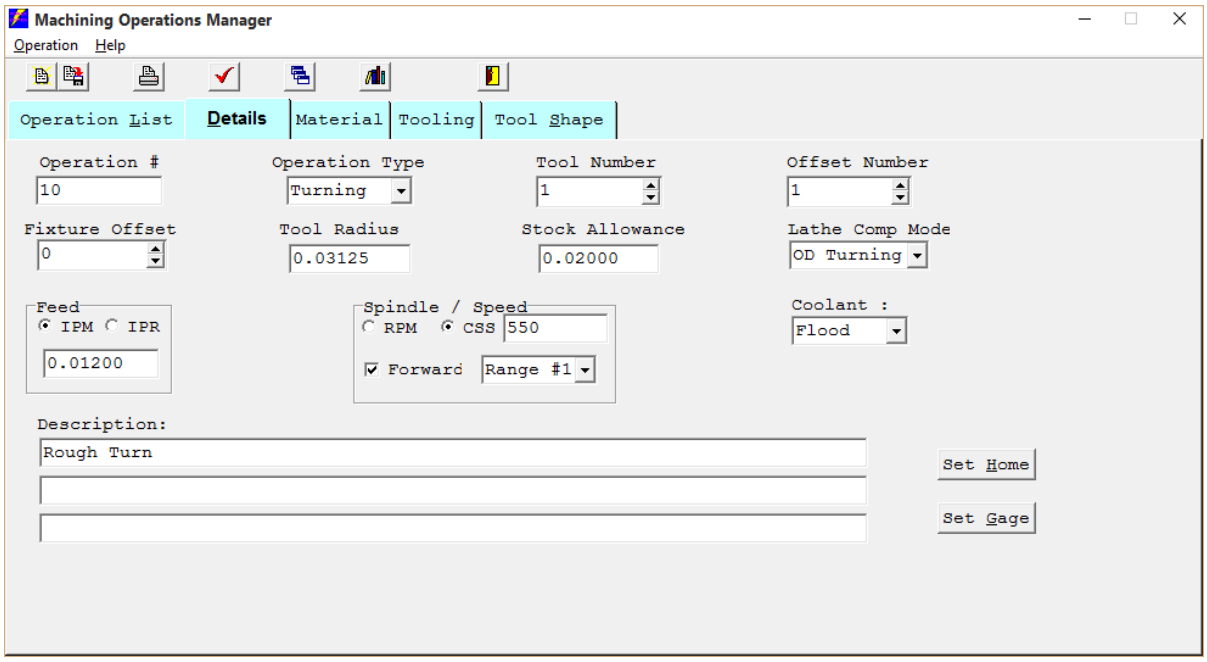


Figure 4-89c (Operation #10 Details)

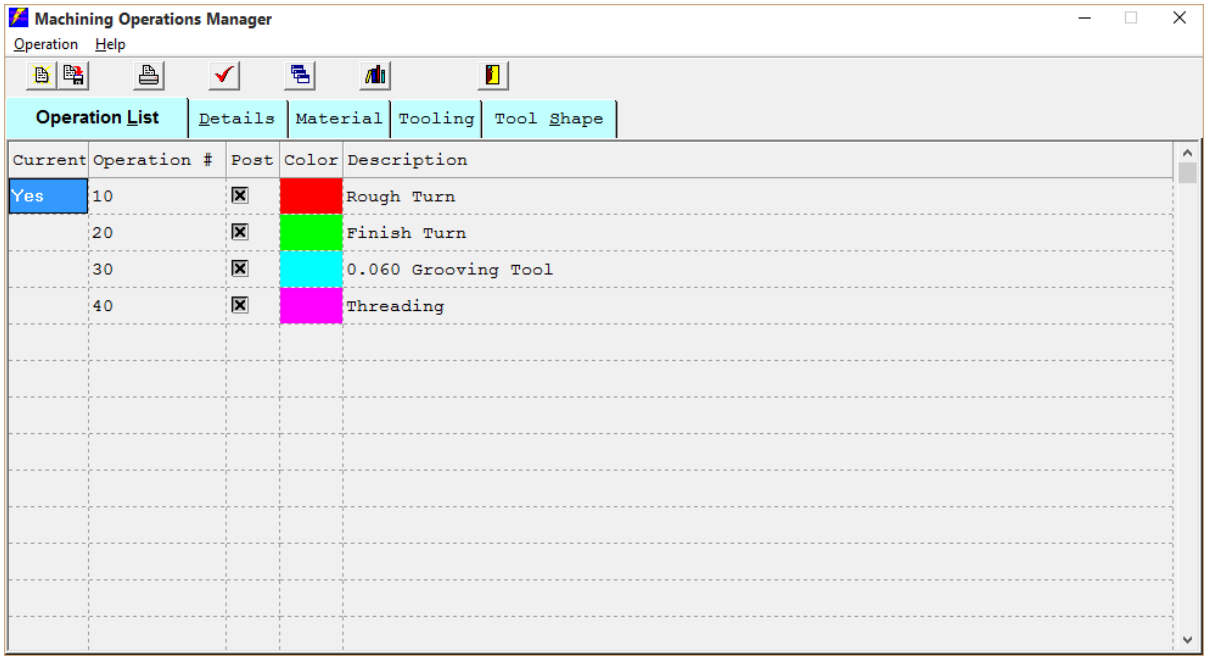


Figure 4-89d (All operations for the tutorial part)

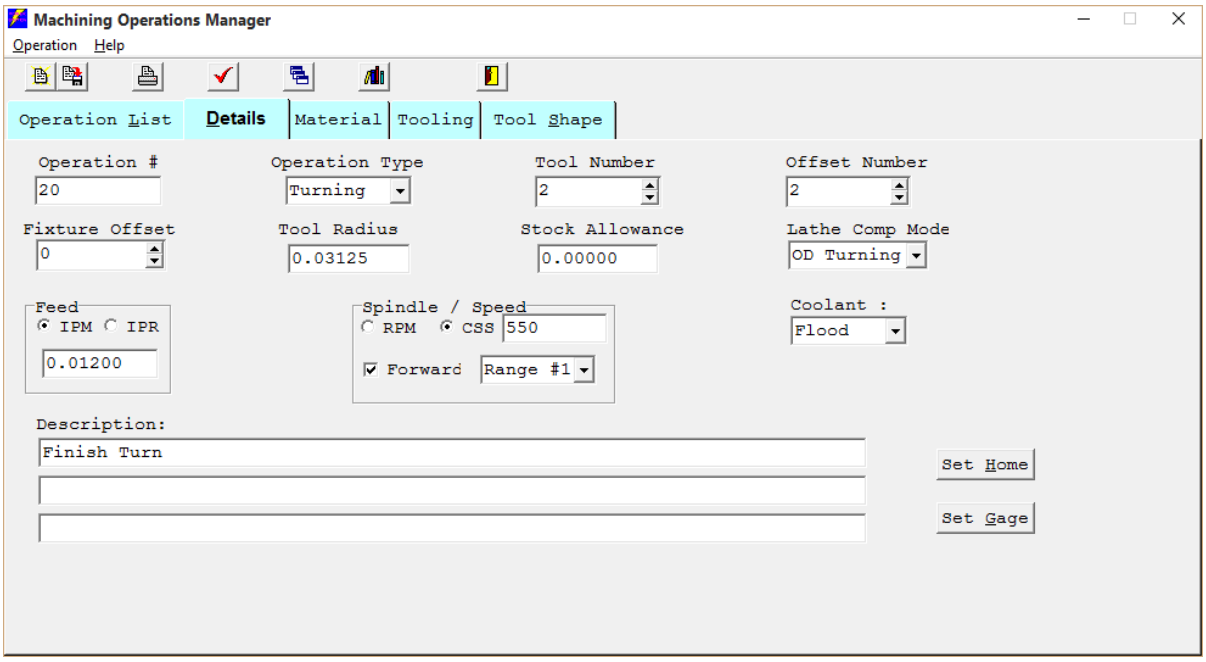


Figure 4-89e (Operation #20 for the tutorial part)

Next define the remaining operations, using the information displayed in figures 4-89 c,e,f & g.

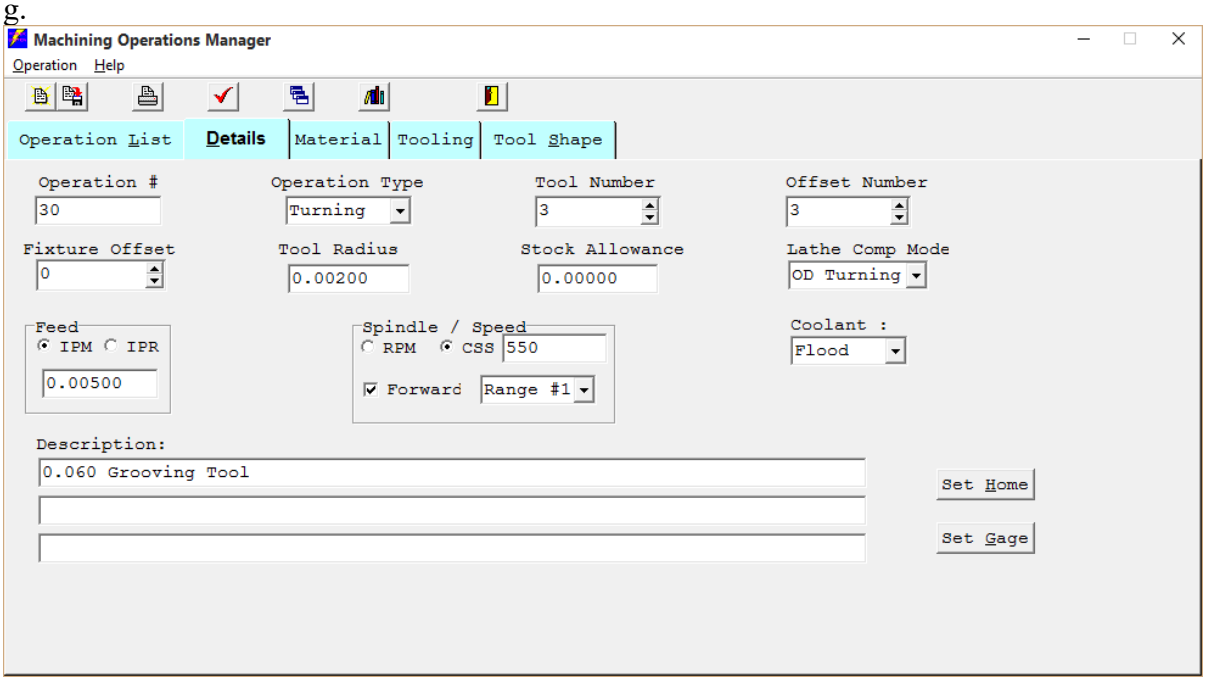


Figure 4-89f (Operation #30 for the tutorial part)

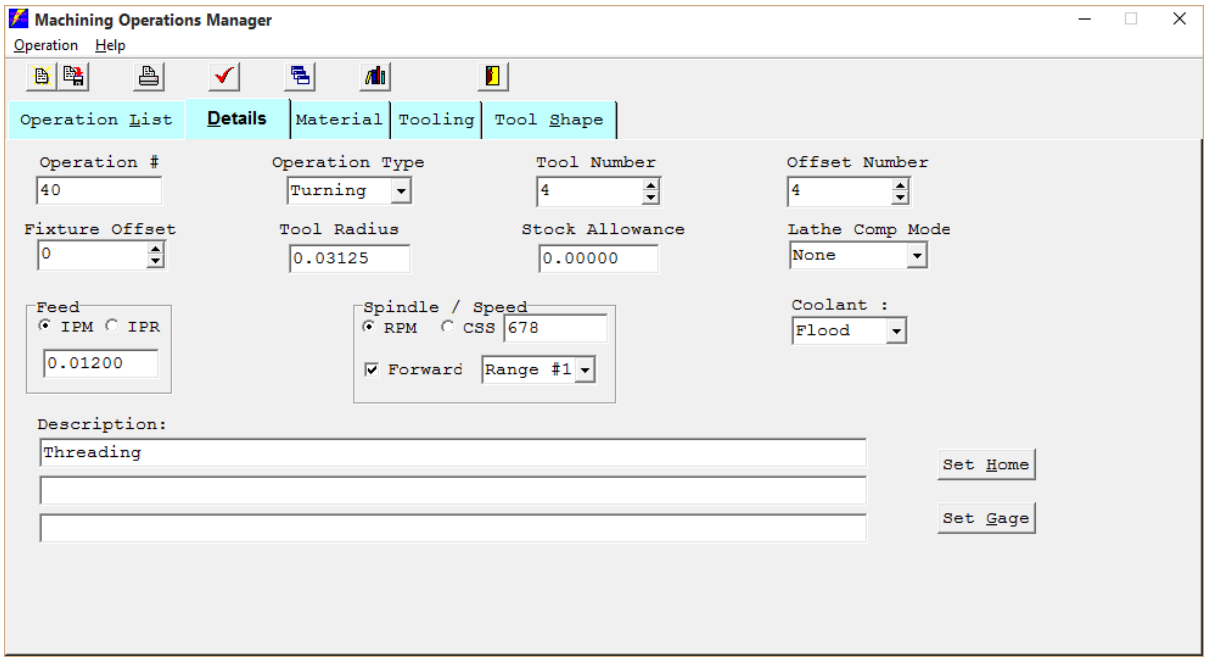


Figure 4-89g (Operation #40 for the tutorial part)

STEP 12

Next, select “Operation-Set Home For All” home position of X6 and Z6.

STEP 13

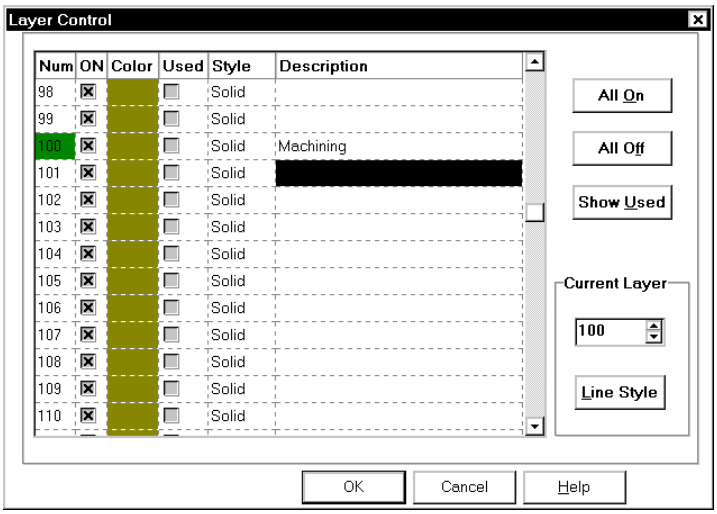
At this time it is a good idea to make a print out of the defined operations. To do this select “Operation”, “Print” from the menu.

STEP 14

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 15

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.



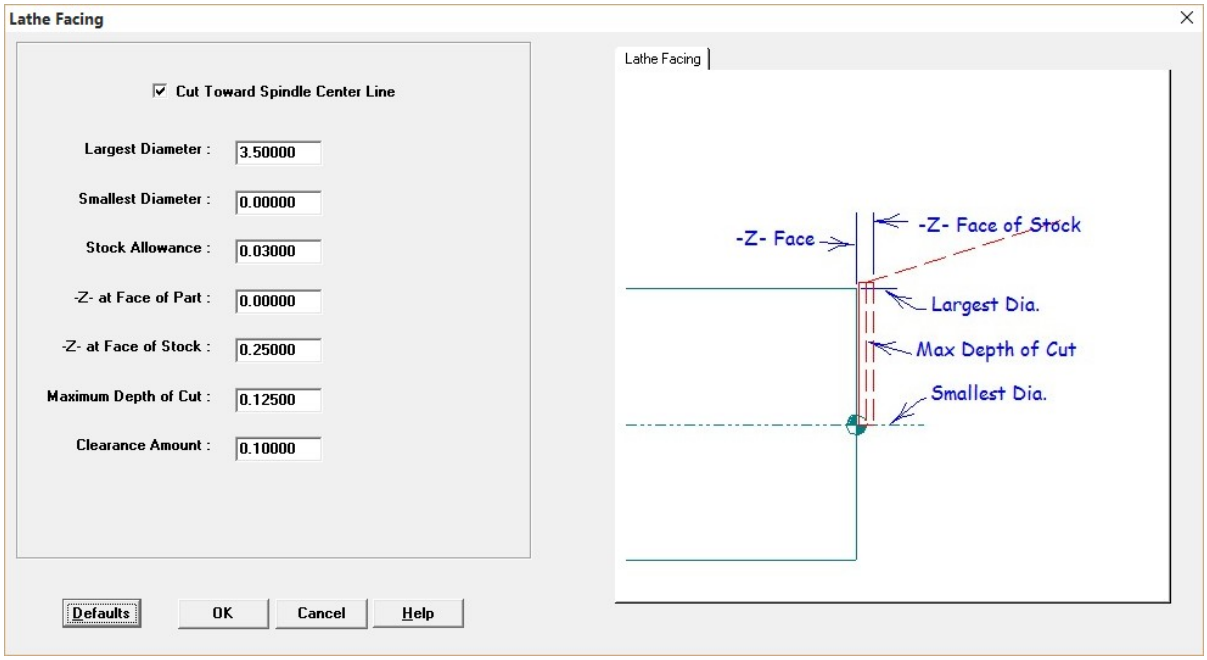
To change the current layer to 100, press the **<F9>** key (or select “Display”, “Layer-Control” from the menu), then click on “Current Layer” box and type: 100, click the cursor on the description box (The right most column in the layer 100 row), and enter in a description like “Machining”. Next press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

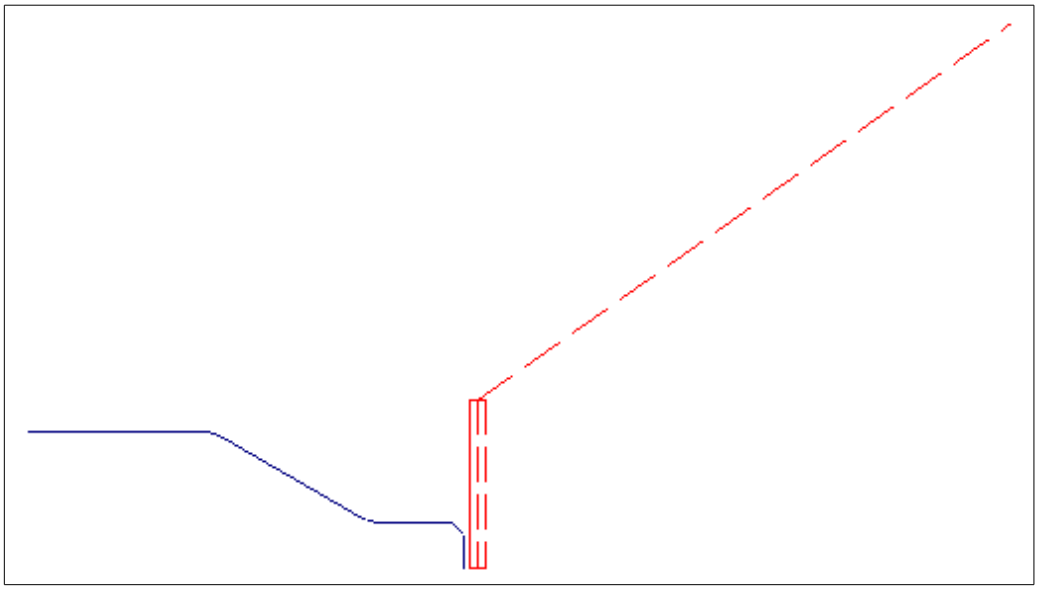
STEP 16

This is a good time to save our work as in step 9.

STEP 17

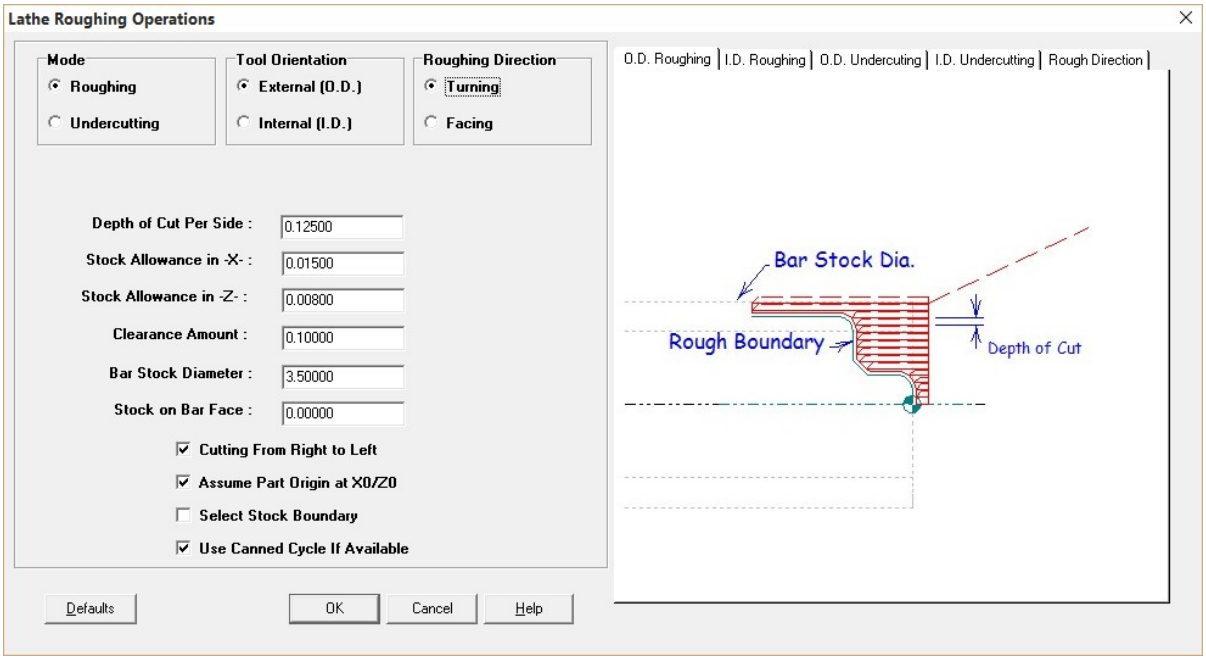
With our roughing tool, we will now face off , then rough turn the part. First the facing. Select **"Machining" - "Lathe Cycles – Lathe Facing"** from the menu. Lets assume a rough bar stock diameter of 3.5 and additional rough stock on the face of 0.25.





STEP 18

Now let's start the roughing cycle. Select “Machining”, “Lathe Cycles”, “Lathe Roughing/Undercutting” from the menu. The single most important rule for turning work is ***“IF THE TOOL IS NOT GOING TO TOUCH IT, THEN DO NOT SELECT IT”*** (In fact don’t even define it).



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

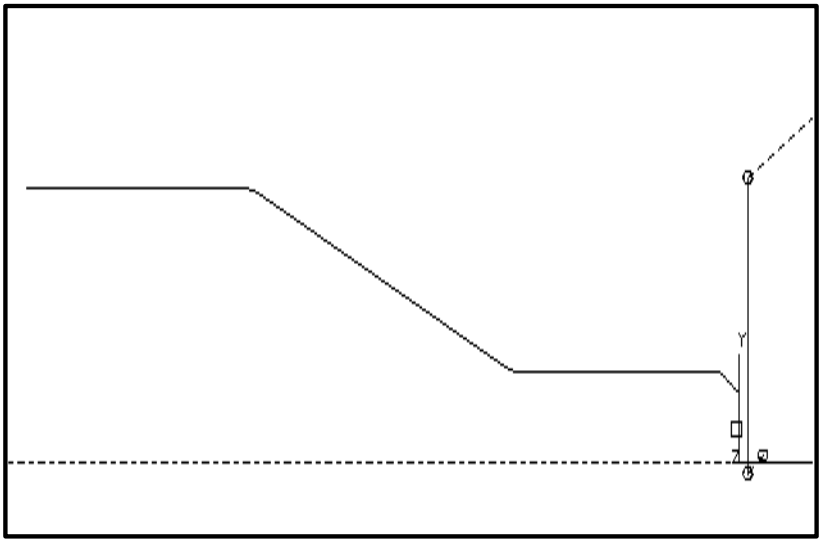
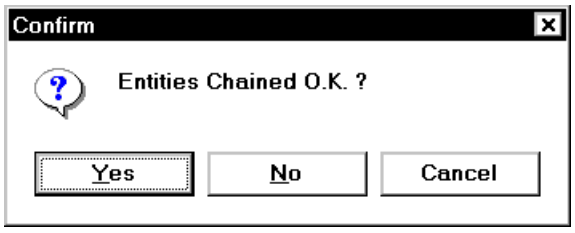


Figure 4-93 Selecting the starting point for roughing

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

Press the <Esc> key or 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.

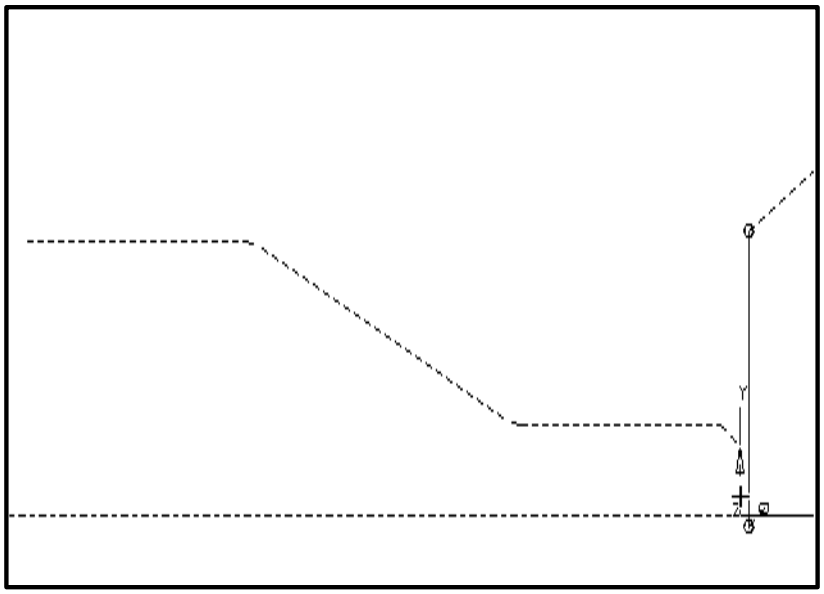


Figure 4-94 Proper chaining

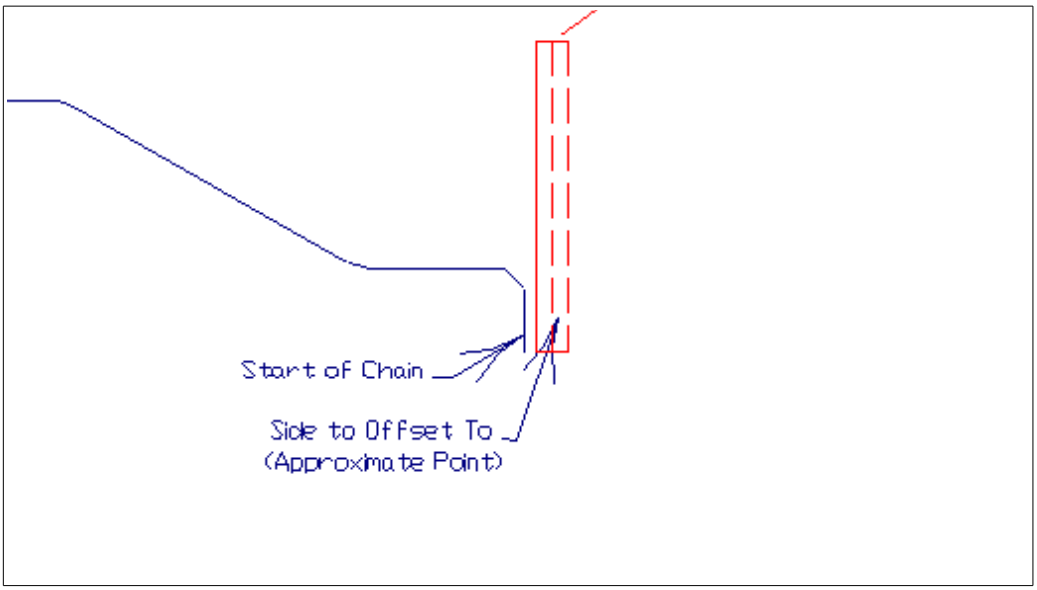


Figure 4-95 Picking the side to offset to

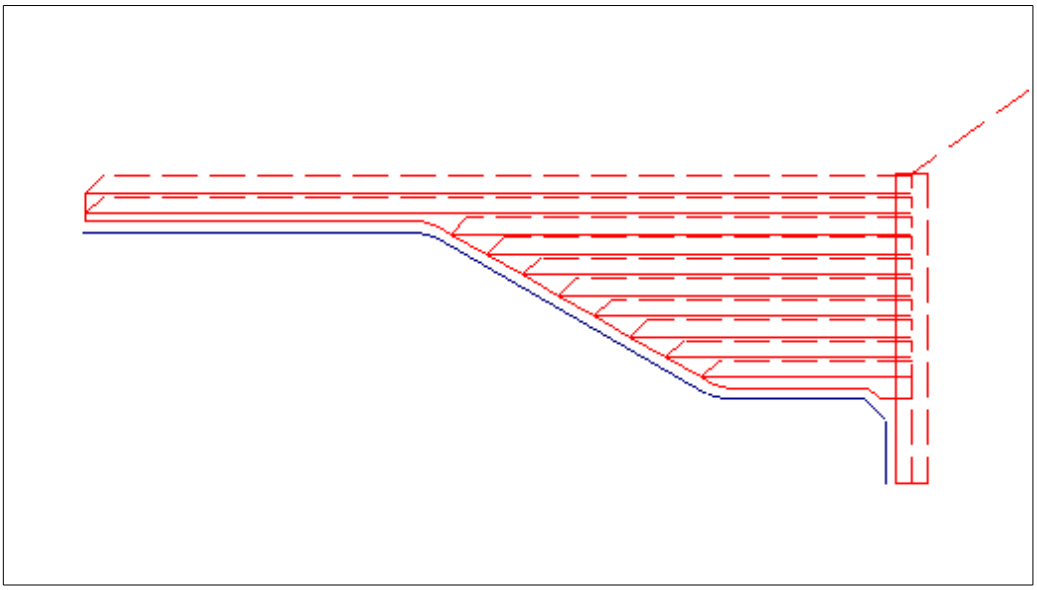


Figure 4-96 Roughing completed

STEP 19

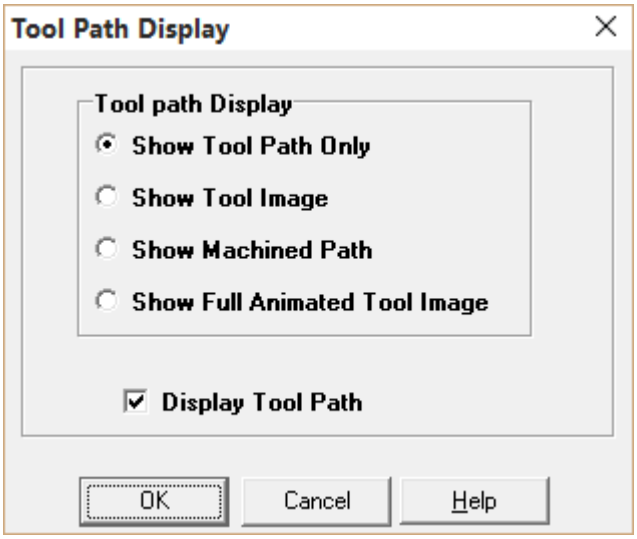
Next we will bring the tool back home. Select “Machining”, **"MOVE & CUT" - "GO HOME"** from the menu.

STEP 20

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

STEP 21

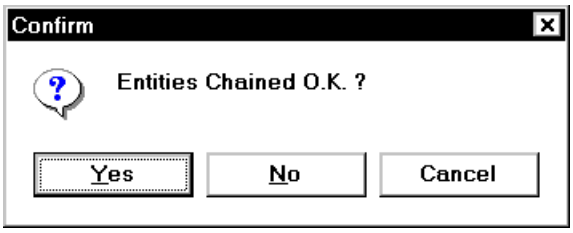
On many slower computers, the display of the tool images (small circles) can be very time consuming. If you want to turn them off, select **"DISPLAY" - "Tool Image"** from the menus.



STEP 22

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 <Esc> key or click 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)

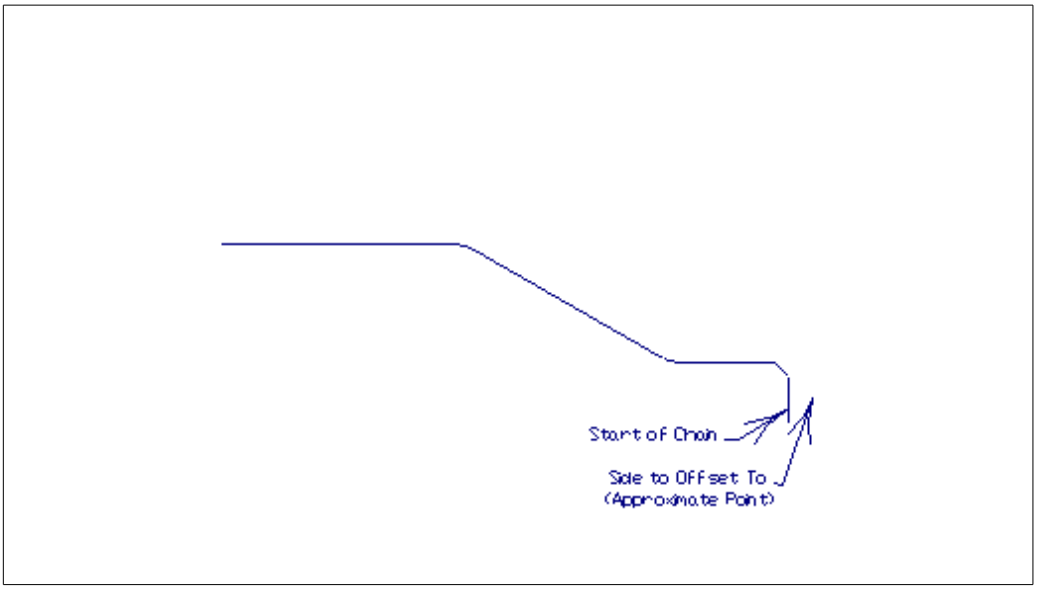
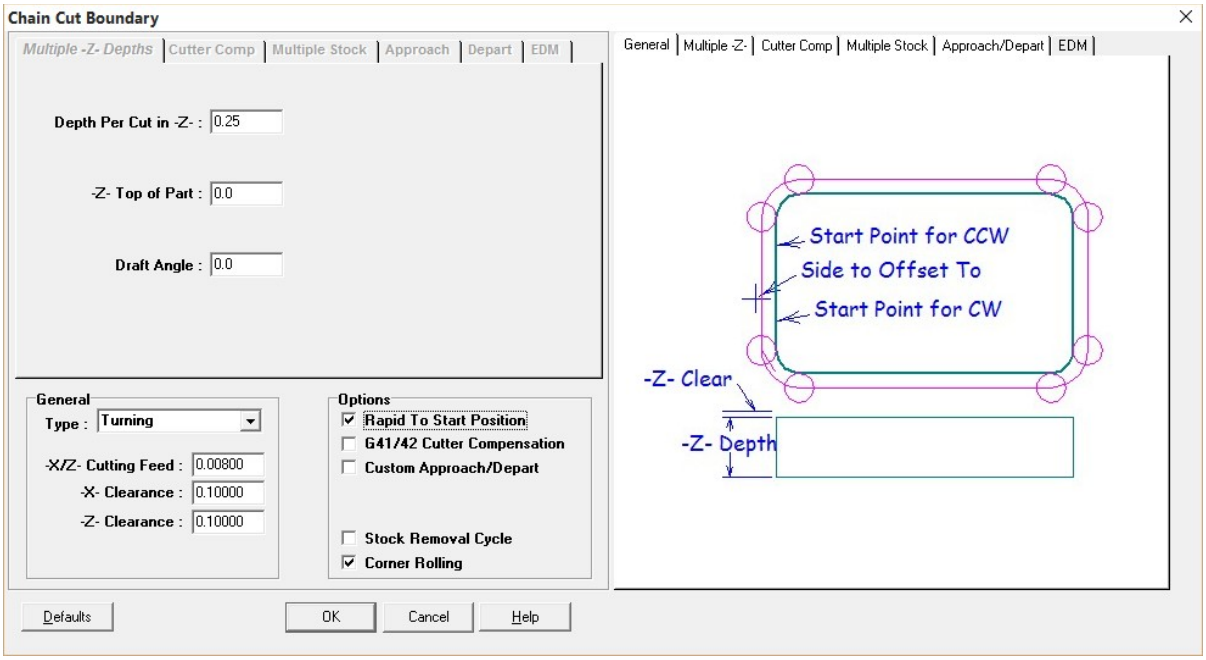
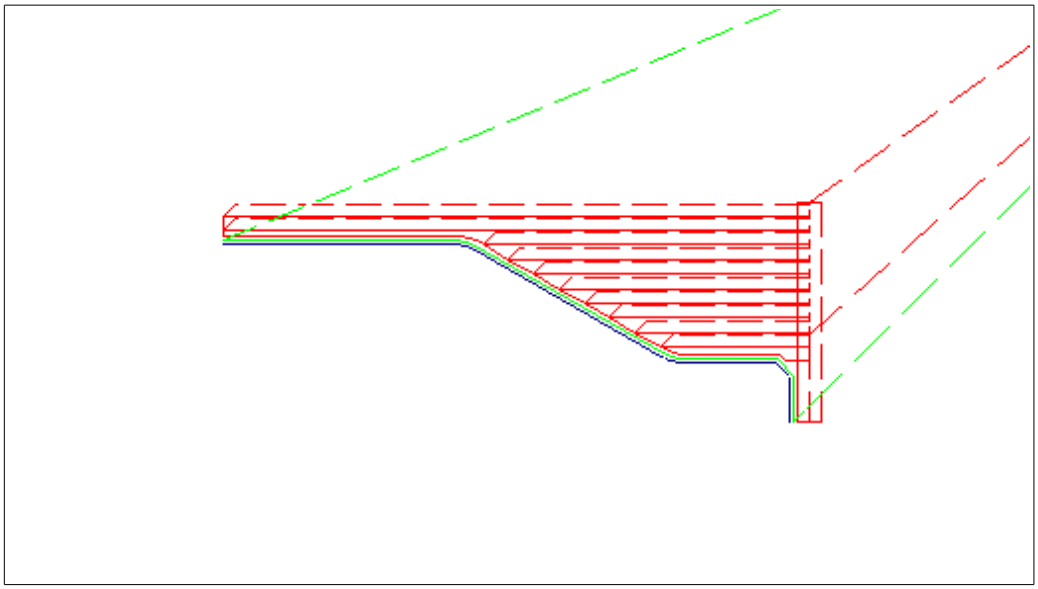


Figure 4-97 Selecting the start line for chaining



STEP 23

Next we will bring the tool back home. Select “Machining”, "MOVE & CUT" - "GO HOME" from the menus. The screen should look like this:

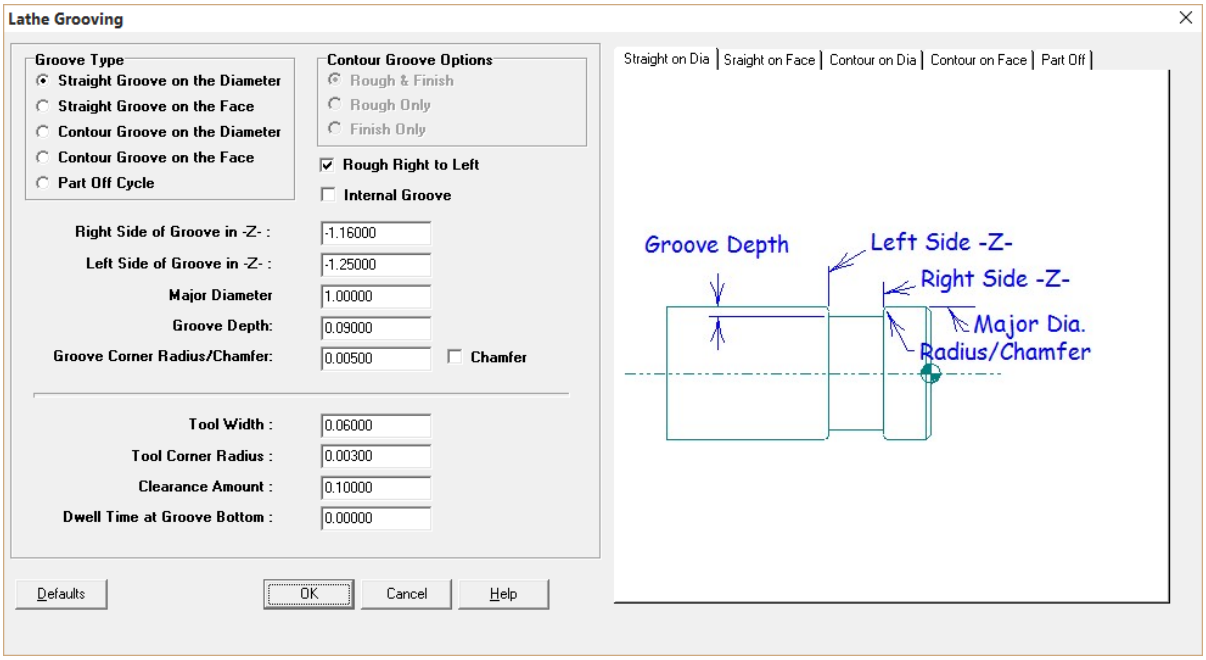


STEP 24

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

STEP 25

Now we will generate the groove. Select “Machining”, “Lathe Cycles”, “Lathe Grooving/Partoff” from the menu. 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- : -1.5+.09`
`Left Side of Groove in -Z-  : -1.5`

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 26

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

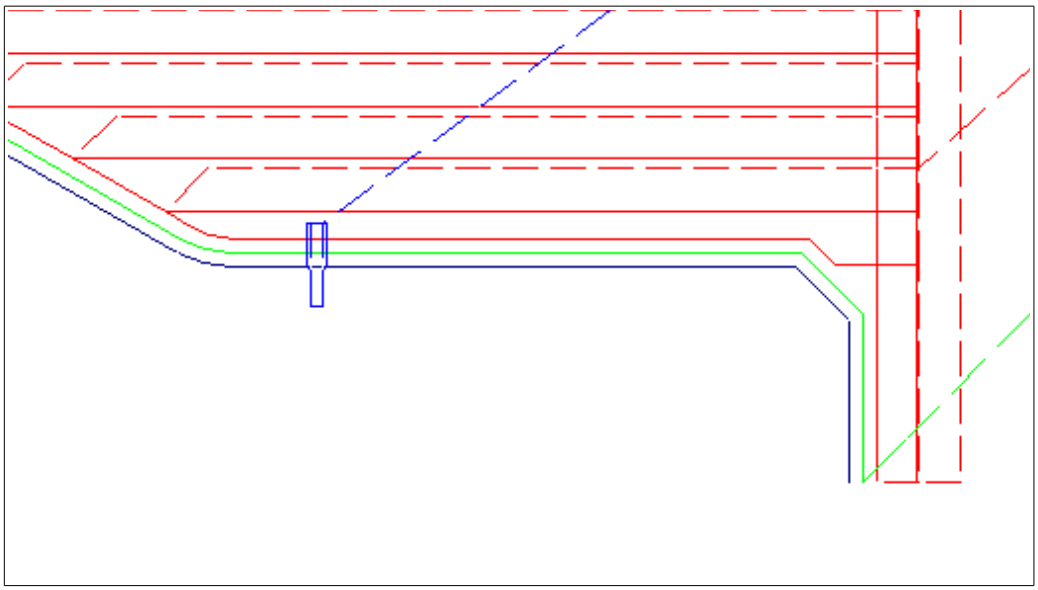


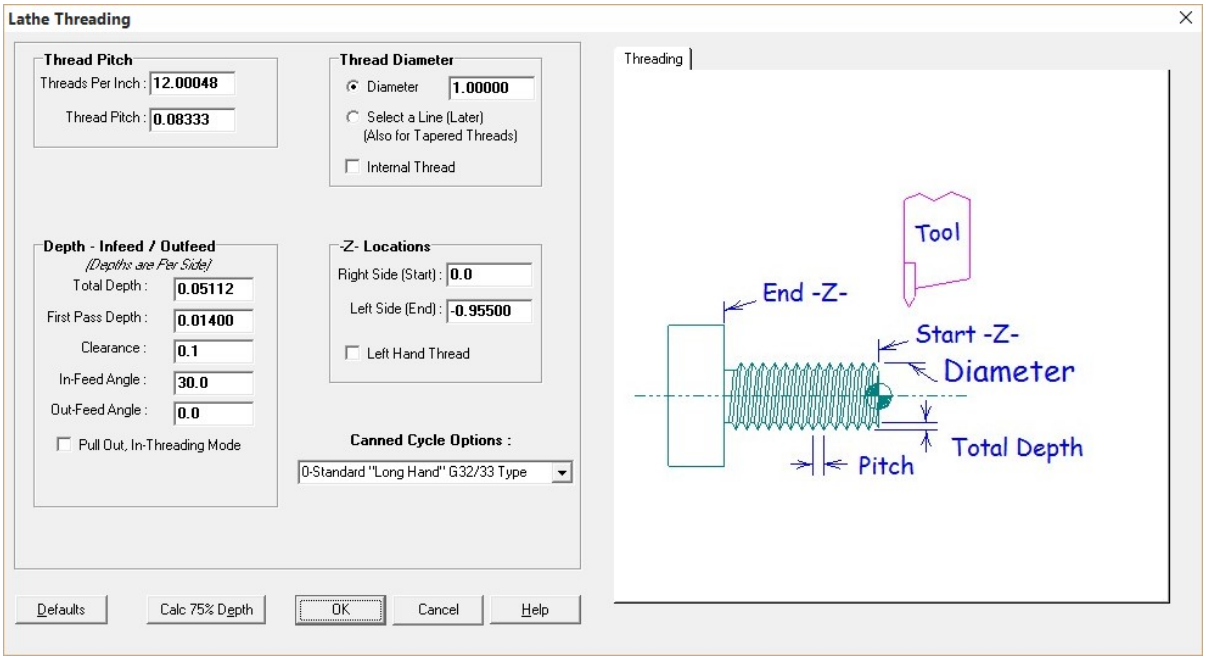
Figure 4-99 The grooving complete

STEP 27

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

STEP 28

Now we will add the thread. Select “Machining”, “Lathe Cycles”, “Lathe Threading” from the menu.



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 “ -1.25+.045” in order to thread into middle of the groove.

STEP 29

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

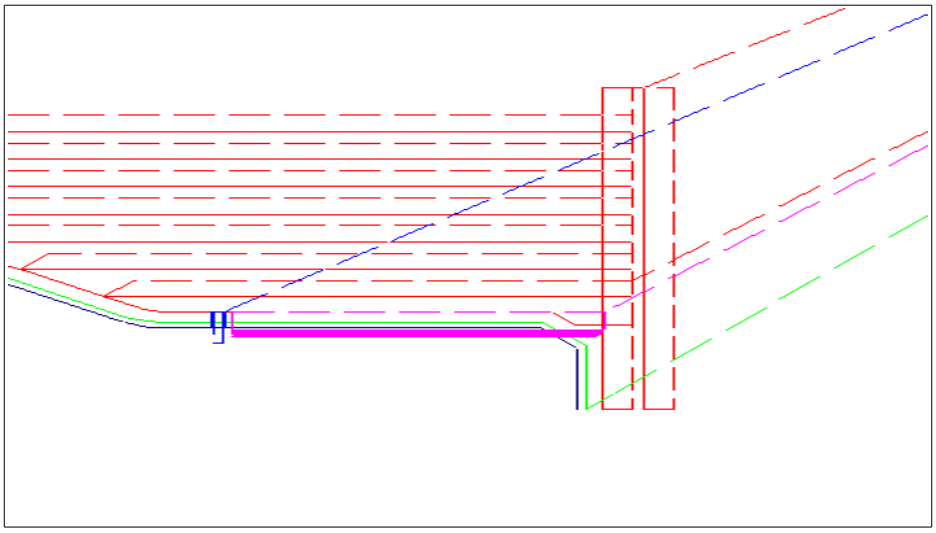


Figure 4-100 The Threading Complete

STEP 30

Once again, lets save the file as in step 9. After that, lets check the machining status. Select “Machining”, “Machining Status” from the main menu. When done viewing the status, press the [OK] Button.

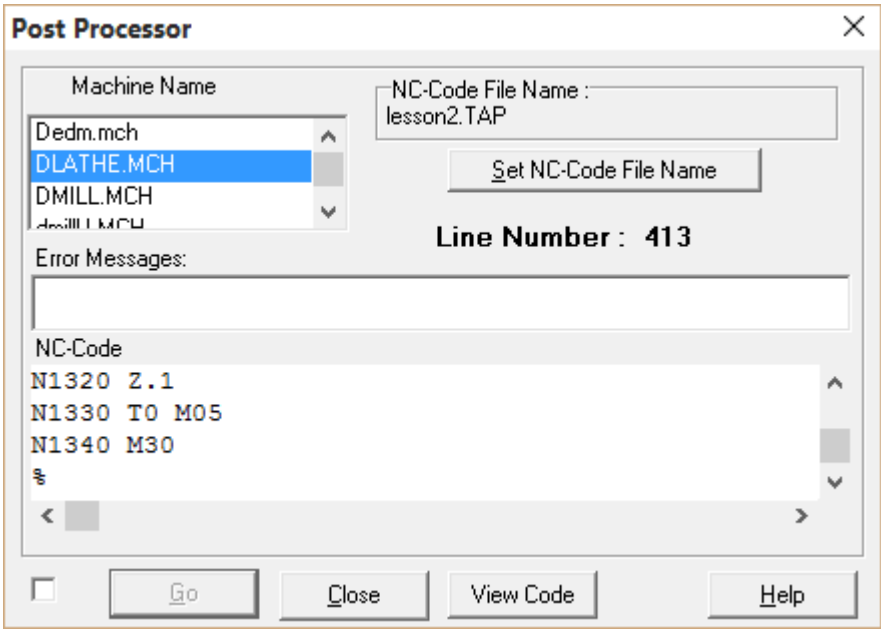
STEP 31

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 labeled [Set NC-Code File Name]. Enter “LESSON2” as the “File Name”, and press [OK].

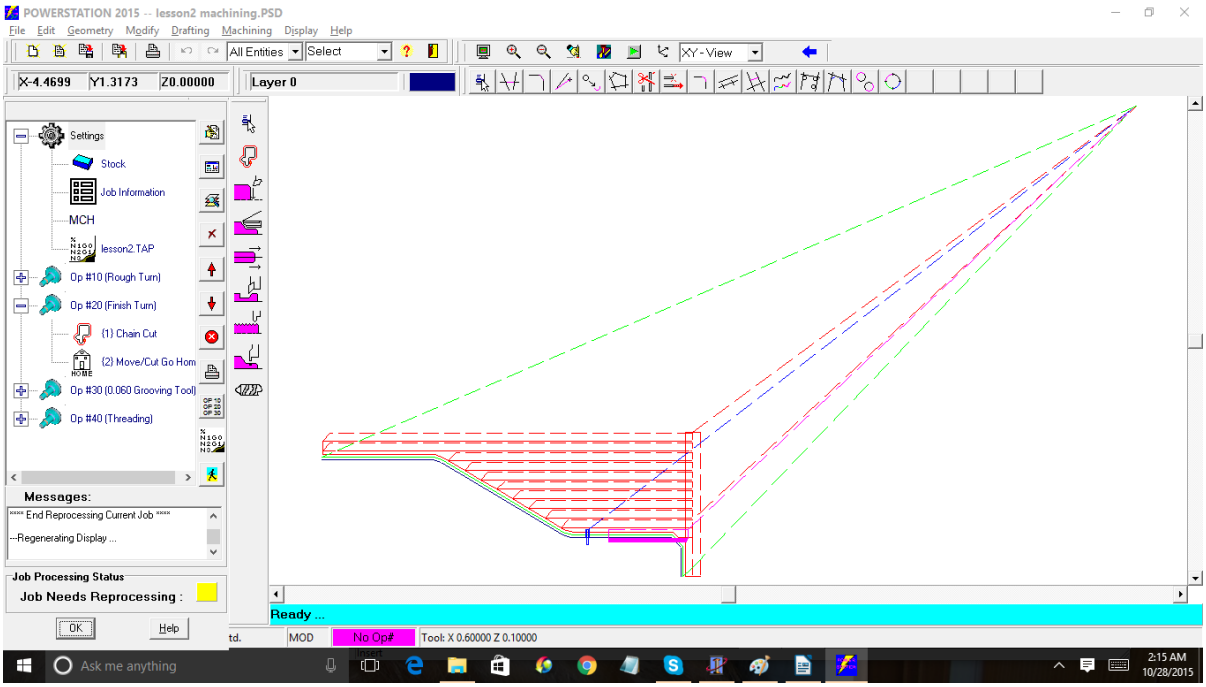
Press the [Go] button. The post processing now begins, and may take a few minutes. When done, press [Close] to leave the post processor.



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

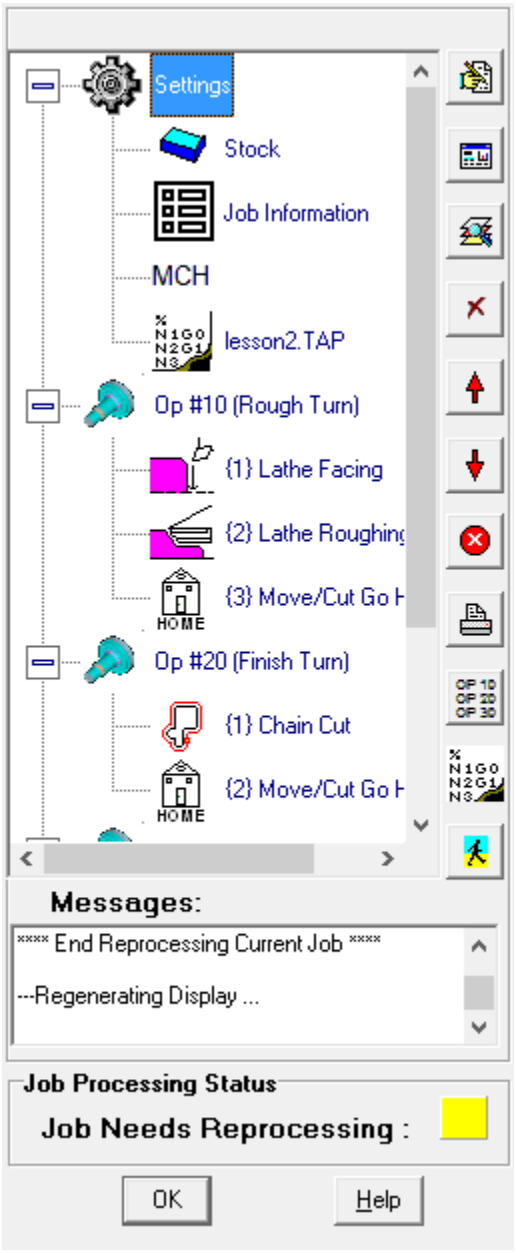
Step 32

Select “Machining-Job Processing”. The main display will change in order to accommodate the Job Processing Dialog.



Step 33

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:



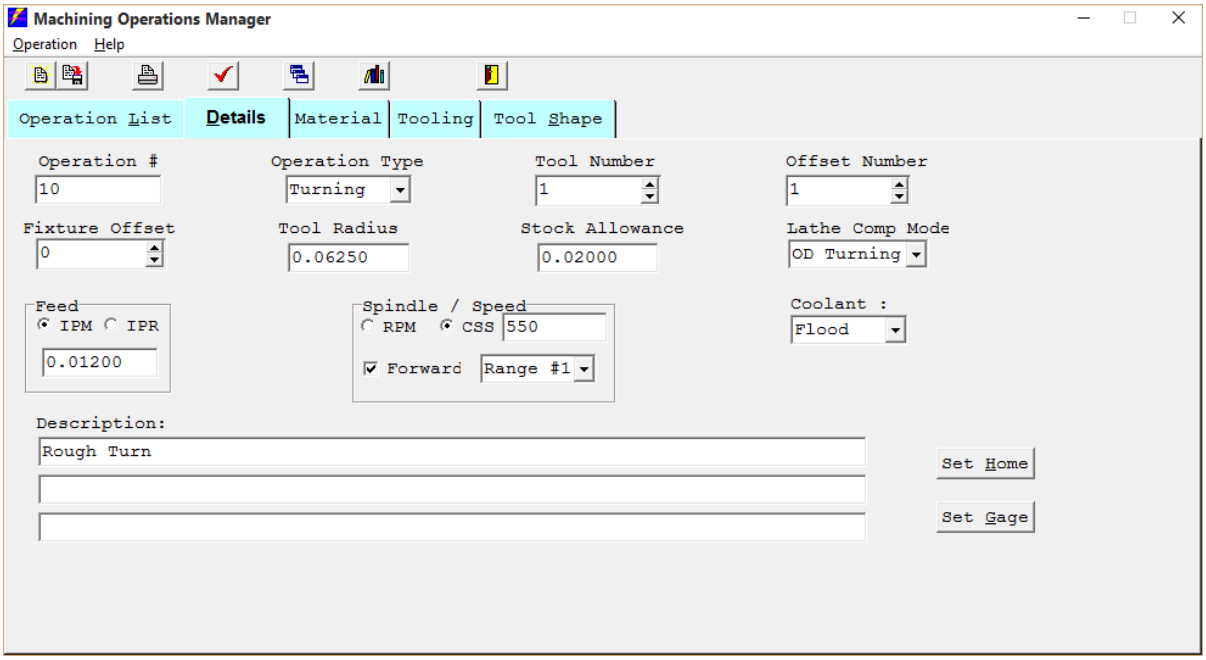
Step 34

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, 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

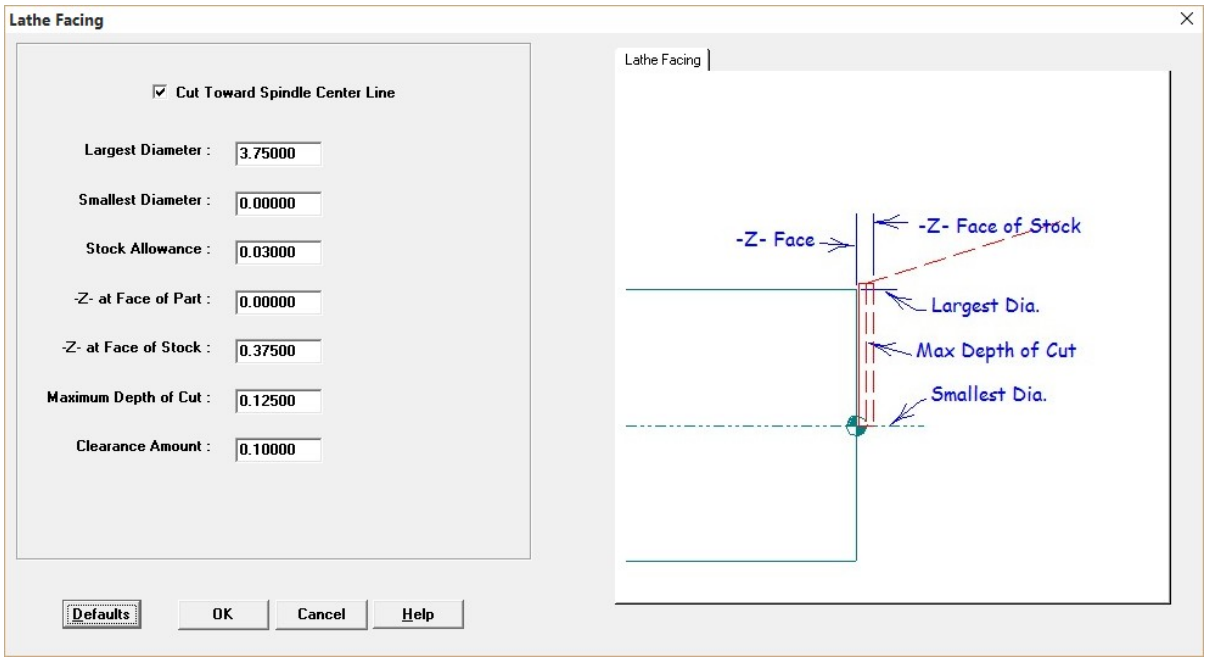
Step 35

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.



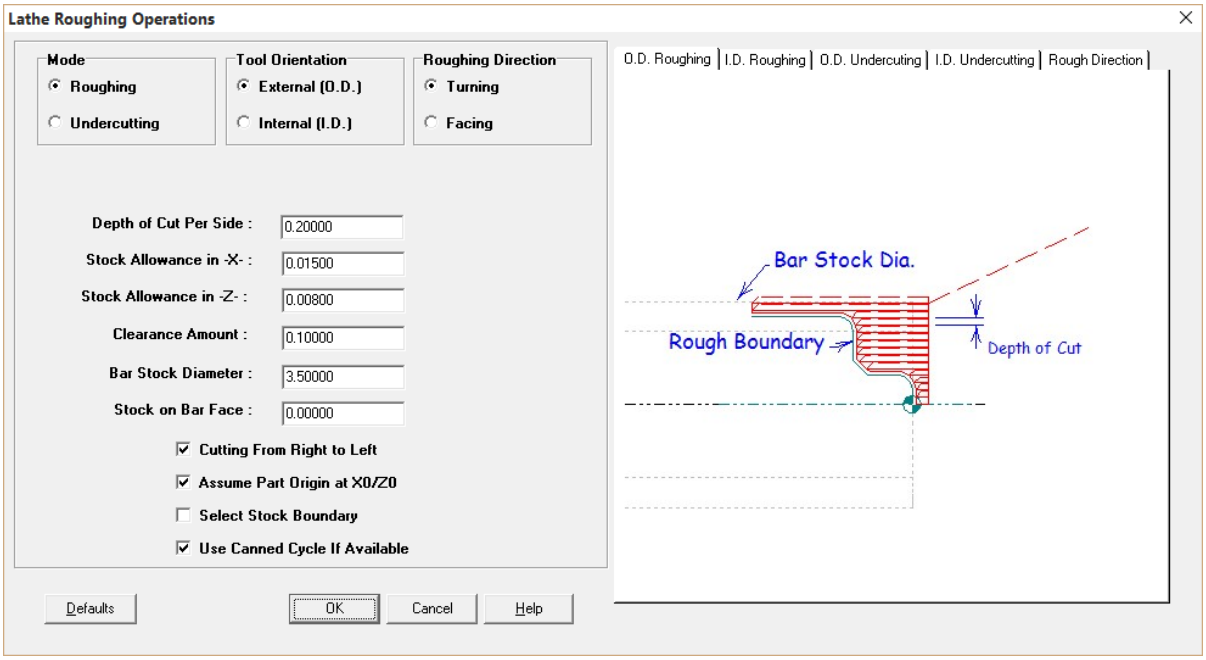
Step 36

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 37

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 38

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

Step 39

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

INDEX