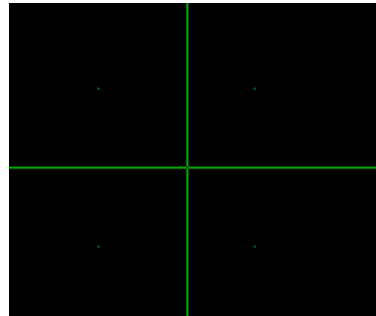
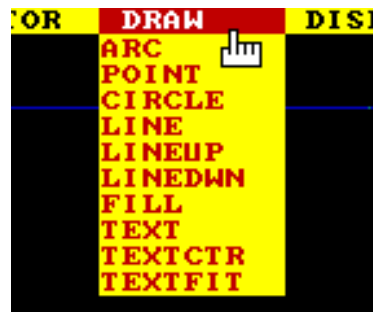


Locating menus/ Selecting commands

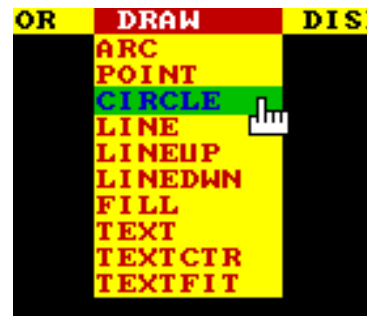
The green crosshair is moved by the mouse over the black graphic display area of the screen.



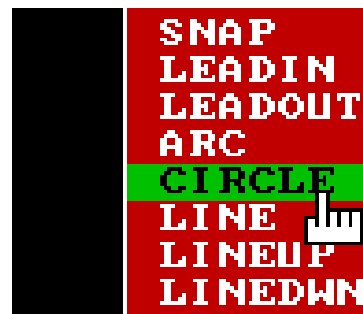
Moving the crosshair to the top of the screen activates titles of pull down menus and their commands



The hand pointer is moved by the mouse to select a menu. A command on the list can be selected with the hand pointer

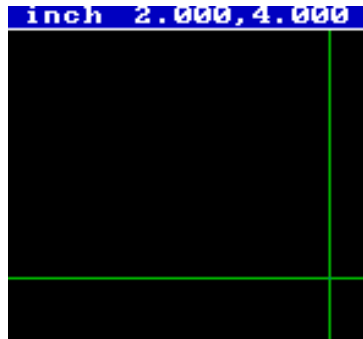


At the right side of the screen is the quick select menu. The hand pointer appears when the crosshair is moved into the menu area.

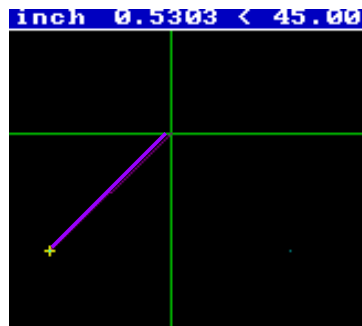


Crosshair coordinates/Tool coordinates/Motors status/Command line

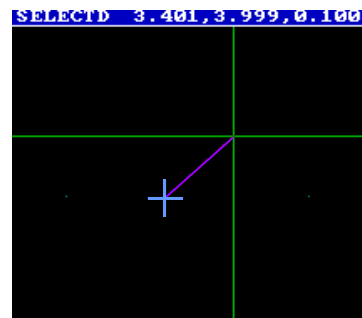
As the crosshair is moved on the black drawing area, the X and Y axis position is displayed on the blue bar at the top of the screen.



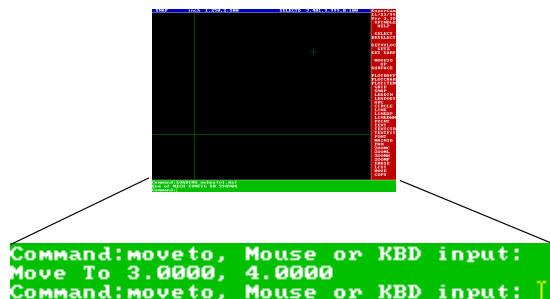
When you select a draw command and are moving the crosshair, the blue bar will change to the line length and number of degrees from the start point.



The top right blue bar displays the X, Y, and Z position of the machine cutting tool. The word "SELECTED" on the bar indicates that the stepper motors are energized and the machine is locked electronically into position.



The command line is at the bottom of the screen. It displays what the program is doing step by step. It is important to refer to the command line often as it will change with each selected command. Value settings are displayed on the command line as entered with the key-



Command line entries/Grid setting/Snap setting

The “ENTER” key on the keyboard will pass you through settings on the command line. If you wish to type in a new value, the new setting will be registered by pressing the enter key. All entries on the command line are in decimal units.

```
Move To 3.5625, 3.6875
Command:setzdepth, Z cut depth <0.1250 inch>: .250
Command:
```

The “GRID” is a pattern of dots on the screen that assist you in seeing the position and size of screen objects. Keyboard entry changes the size of the grid.



```
- <toggle|enter>
- <toggle|enter>
Spacing < ON 1.0000 inch >:
```

Command line shows the grid is now set for a one inch spacing. Accept or type new value. Press “ENTER”.

The “SETGRID” Command turns the grid on or off.

```
Grid Flag OFF - <toggle|enter>
Grid Flag ON - <toggle|enter>
Command:
```

Press the space bar to toggle the grid on or off. Press enter to set.



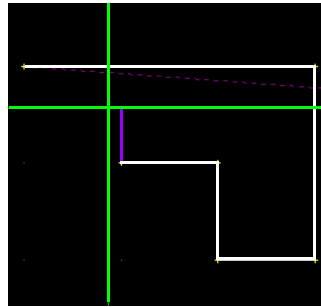
The “SNAP” setting causes the green crosshair to move in increments of set units. For example, a value of one sixteenth inch will only allow lines to be drawn on 1/16 inch intervals. Set any value you want on the command line. Settings are in decimal units.



Top left corner of screen shows if snap is on or off. The “F9”

Ortho/Tool display/Clear buffer/Command line and mouse buttons

The "ORTHO" command only allows lines to be drawn vertically or horizontally. It assures lines will be drawn "true".

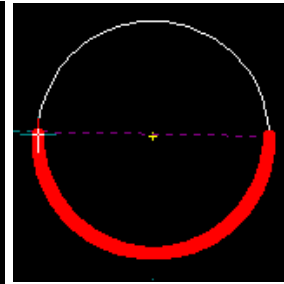
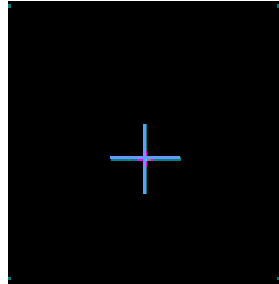


ORTHO

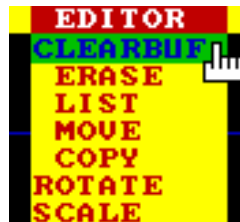
"F8" will toggle the ortho feature on or off.

The light blue crosshair represents the position of the cutting device on your machine. It remains blue when traveling and turns red when cutting.

"SHIFT F2" toggles the display of the tool movement on or off. Refresh the screen with F1.



When you draw on the screen, the lines are temporary, and will be lost unless saved with a file name. You can play with making lines, circles, arcs etc. To erase the the whole screen and start over you clear the buffer.



Respond to the command line with a "yes" by clicking the right mouse button.

Limits
Clear Drawing Buffer! <N|Y>

The command line often asks for a "yes" or "no" as in the illustration above. The position of the "N" and the "Y" on the command line indicates which mouse button to use.

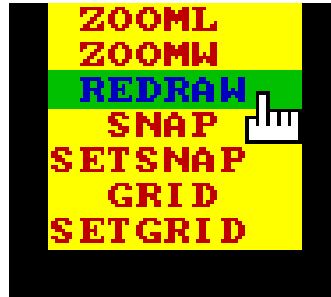
<N|Y>

LEFT MOUSE
BUTTON

RIGHT MOUSE
BUTTON

Redraw/Set mechanical limits/Zoom limits

The “REDRAW” command and the F1 key do the same function. They clear the screen of residual path lines and points.



In SuperCam, the word “limits” means the mechanical cutting range of the device you are operating. You must enter the dimensions with the “SETLIMIT” command. We consider 0,0, to be the lower left corner.



Press the “ENTER” key to accept.

Limits lower left <0.0000,0.0000>

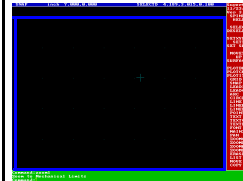
The first value on the command line is length (X axis) and the second is height (Y axis).

```
Upper Right <7.0000,5.0000> : 10.
Command:setmlmt, Set Mechanizm Li
Upper Right <10.0000,8.0000> :
```

The old limits are 10 in. by 8 in. This example is to change to 7 in. by 5 in.

```
mlmt, Set Mechanizm Limits lower
<10.0000,8.0000> : 7.000,5.000
```

Values are typed. Press “ENTER”.



The blue border shows the new limits.

The zoom limits command shows the area of the mechanical limits instantly.



Zoom center/Zoom Window/Zoom Previous

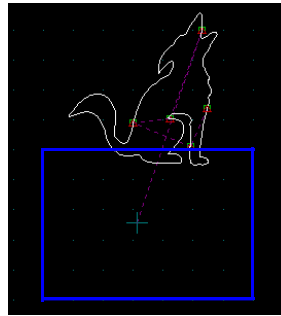
The zoom center command allows you to see beyond the field of the mechanical limits you have set.



```
Command: zoomc, Zoom Center, Point:  
Height <5.338>: 10.00  
Command:
```

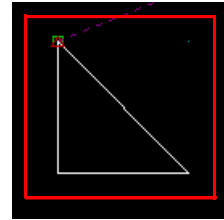
After clicking on the command you are asked to select a point. Place the cross-hair in the center of the screen and click. Enter a number larger than the one given for height. Press “ENTER”.

You can enter larger numbers like 20, 50, 100, to see larger areas outside your mechanical limits. Sometimes when you import a file from Autocad or Coral Draw, the drawing will be outside the mechanical limits.



Example of imported drawing that is outside the machine limits. The move command can drag the object into the limits.

The zoom window command allows you to select a specific area on the screen to zoom in on. Left click on a point at the edge of the area you want to magnify, and move the mouse across it. A red window frame will grow from the selected point. When-



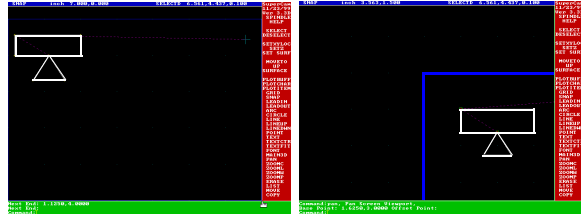
the window surrounds the object, left click and the area will fill the screen.

The zoom previous command will take you back to the view you had before you created the current view. Repeated selection will move you back through several views.



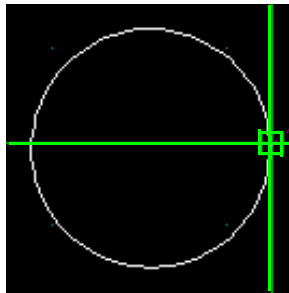
Pan/Move/Copy

The pan command is a viewing assist. Using pan is like moving a large drawing around on a table so the part that interests you, is in your work area. Select a point near the edge of the screen by clicking. Then move the crosshair to the center of the screen and click.



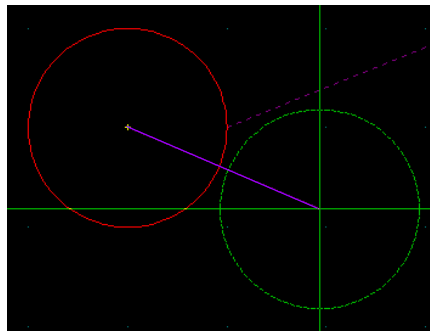
Notice that when you pan, you can be looking beyond the machine limits.

The move command makes it possible to move objects around on the screen and also, to bring objects that are outside the machine limits, into the work area. Note that the F5 key must be toggled "ON" to drag objects.

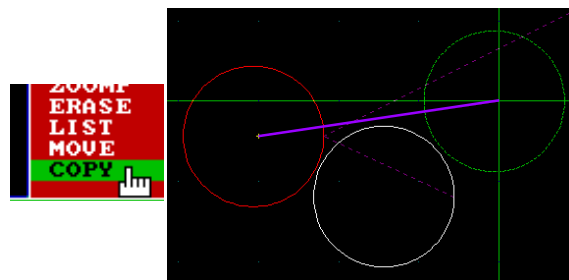


The crosshair will have a square displayed. You must place the crosshair over a line in the object to select it.

The lines in the object will turn red when you click the left mouse button. You may not get it the first time. Move the crosshair slightly and click again until the lines turn red. Right click the mouse to end the selection. The command line asks for a base point. Click in the center of the object and drag it to where you want to locate it. Left click to set the object in place.

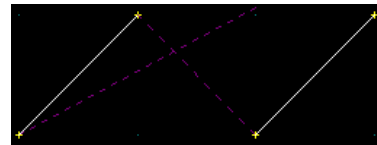
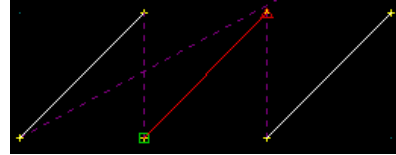


The copy command works much like the move command. Select the item with the crosshair and left click. End the selection by right clicking. Click in the object to select the base point. Move the crosshair to the point you want the object copied to, and left

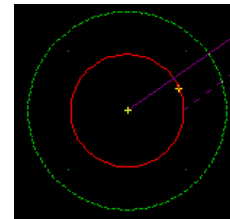
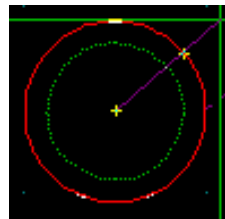


Erase/scale/rotate/toolwidth

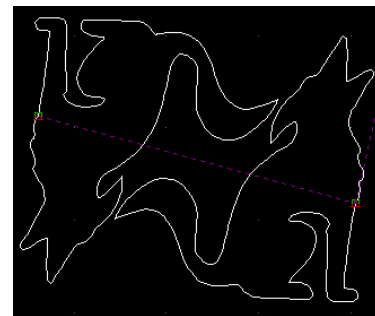
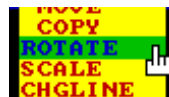
The erase command allows you to eliminate an item in your drawing. Use the crosshair to select the item. Item lines will turn red. Right click to end the selection. Answer "YES" to the command line question.



The scale command makes it possible to re-size an item. Use the crosshair, left click to select, right click to end selection. Left click inside the object as the base point. Move the mouse away from the point. Object sizes from small to large.



Rotate an object by using crosshair and left clicking on a line in the object. End selection by right clicking. Select the base point inside the object. Move the mouse around the pivot. Place the object by left clicking. In this example, the object has been copied, moved and rotated.



The toolwidth command displays the path of the cutter, showing the area being cut out by a specific tool diameter. You can see exactly what material will be removed. It is most often used to "dry run" a pattern to see the results on the screen, before cutting. Example of 3 different line sizes.

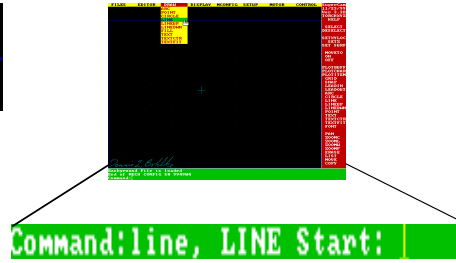


Type the decimal diameter of the cutter on the command line. Press enter.

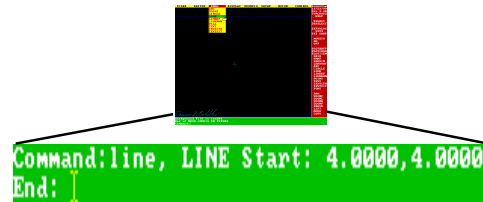
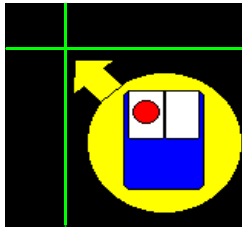


Drawing lines/

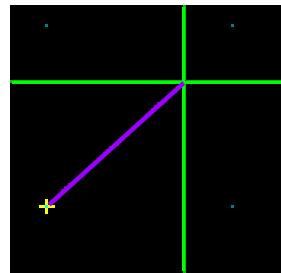
You can select the line command from either the pull down draw menu, or the quick select menu.



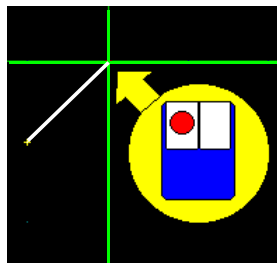
Move the mouse to direct the crosshair to the point where you want the line to start. Left click on the point.



As you move the mouse away from the start point, a purple line will connect the mouse and the start point.

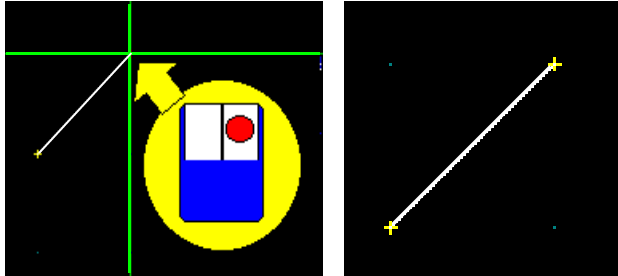


Left click where you want the line to end.

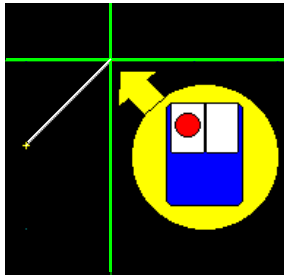


Drawing Lines

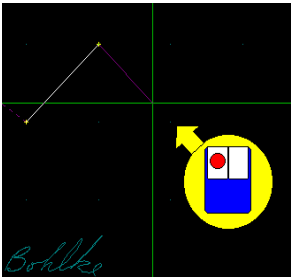
The purple line is still active so you can draw another line from the end of the first. End the selection by right clicking.



You can draw connected lines by simply left clicking from point to point.

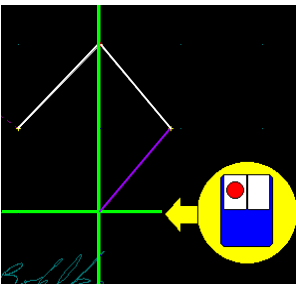


Left click to draw a second line that is connected to the first.



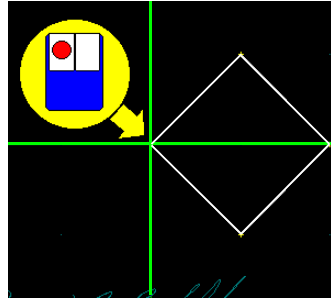
Left click again to make line 3.

(continued on page 11)

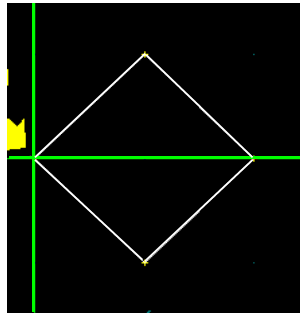


Drawing lines-groups as one item/lines as single items.

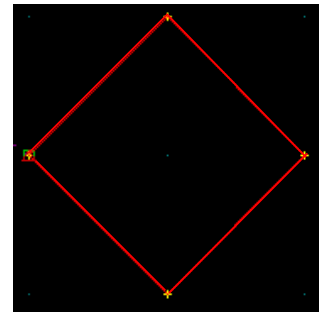
Left clicking makes line 4.



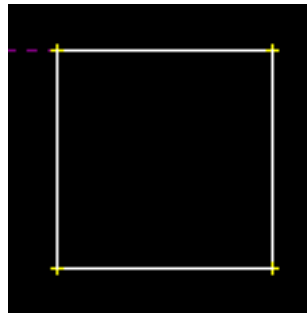
Right click to end line selection. You have created four connected lines which are seen by the program to be one item.



In the future, selecting on any of the four lines will bring all four lines of the object into selection.

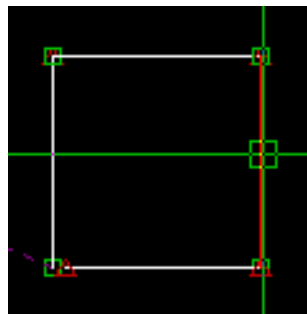


The four lines shown here were each individually drawn by selecting the line command to draw each one. When drawing in Super-Cam, it helps to remember that your machine is going

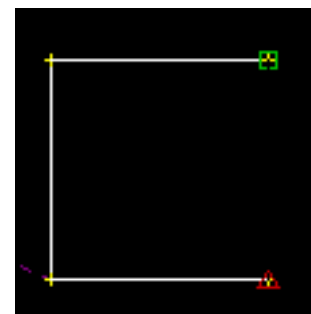


to cut items in the series you drew them in. You can change the order in which items are cut at any time you wish with the "chngorder" command.

The square made from four separate line entities is easily edited. In this example, the erase command is active. The line

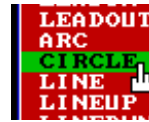


Completing the erase command removes the single line.

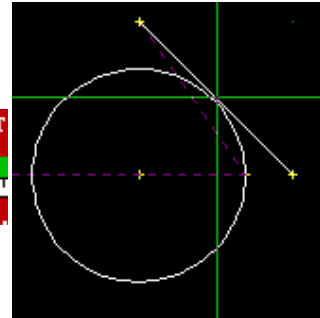


Circle/Arc

The circle command allows you to determine the size of the circle you draw in two ways. Place the crosshair where you want the center to be and left click. Then move the mouse to where you want the circle edge to be and left click. The second way is to select where the center of the circle is to be and then type in the length of the radius on the command line and press the “enter” key.



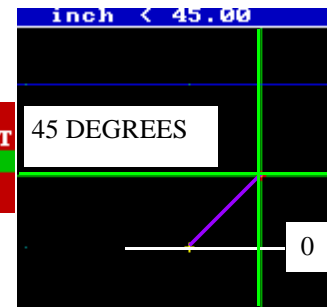
LEADOUT
ARC
CIRCLE
LINE
LINEUP



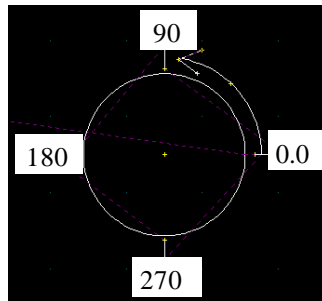
The arc command makes it possible to draw pieces of a circle of the size and length desired. Choose the center point by left clicking. The command line asks for the start angle. As the crosshair is moved away from the start point, the top blue screen will show the line angle from zero. The angle relationship is shown be-



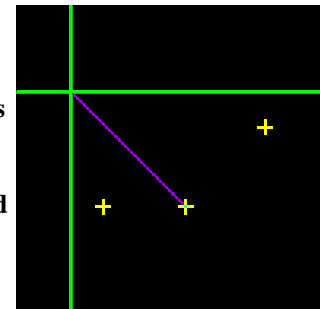
LEADIN
LEADOUT
ARC
CIRCLE
LINE



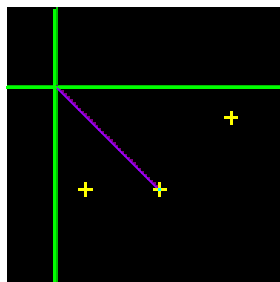
The start angle is selected by rotating the crosshair around the center point. Rotating counter clockwise increases the number of degrees from zero degrees. Left click to select.



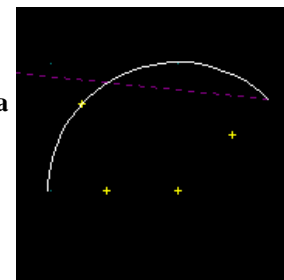
Next, select the end angle by clicking. Example, start angle is 45 degrees, end angle is 180 degrees. Command line asks for radius.



Moving the crosshair outward increases the length of the radius and will increase the size of the arc. You can view the length of the radius line on the blue line at the top left of the screen.



Left click to set the radius. You can type a decimal value for the length of the radius if you would rather. Press “enter”.



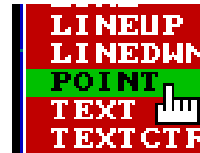
Arcdir/Point/Lineup and Linedown/Change line

The Arcdir command makes possible setting the direction of angle increase to clockwise or counter-clockwise. When drawing arcs, you may prefer to use a clockwise increase.



```
End of MECH CONFIG S# 994904  
Command:setarcdir, Arc Tool Paths <Toggle!Accept>: CCW  
Arc Tool Paths <Toggle!Accept>: CW
```

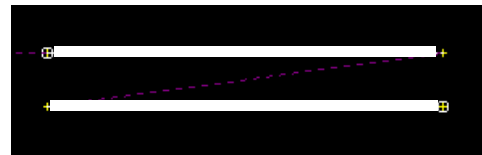
The point command is used to direct the machine in drilling, cutting or burning holes. It sets where the tool will plunge or burn. It does not determine the size or depth of the hole. Tool size and setup do that. Left click to set the point. Right click to renew



the command and left click to set the next point. After selecting a point you can move the mouse into any menu to select a new command.

The lineup and linedown commands make possible the cutting of lines that slope down into the material during the cut, or start down in the material and slope up. This makes 3D sculpting possible using the Z axis.

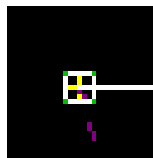
LINE UP
LINE DOWN



LINE UP ■ TOOL PLUNGES AND COMES UP.

LINE DOWN ■ TOOL STARTS AT SURFACE AND SLOPE CUTS TO MAX CUT DEPTH.

The white box at the end of a line indicates the down position of the tool.



The chgline command will convert a lineup and linedown item into a regular line that cuts at an even depth.



Chglndwn/Chglnup/Swapend

The **chglndwn** command will convert a regular line into one that starts at the surface and slopes to end at the designated cut depth.

SCALE
CHGLINE
CHGLNDWN
CHGLNUP
SWAPEND



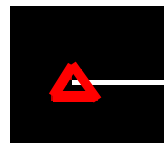
The **chglnup** command will convert a regular line into one that plunges to cut depth and slopes up to end at the surface.

CHGLINE
CHGLNDWN
CHGLNUP
SWAPEND
CHGORDER

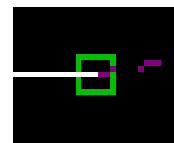


The **swapend** command will change the direction of cut. Select item and reply “yes” to the command line. The green box indicates the line start point and the

CHGLNDWN
CHGLNUP
SWAPEND
CHGORDER
LISTFRNT
JOIN



Red triangle shows where line ends.



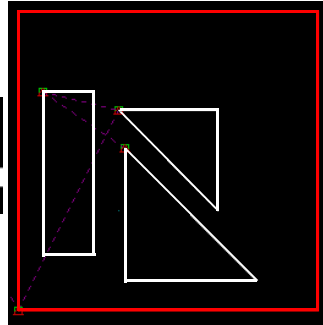
Green square shows where line starts.



Fill/Tool paths

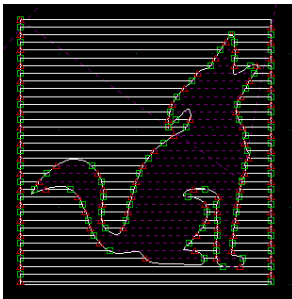
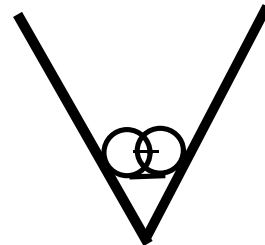
The fill command will make a cut pattern that removes material in the background. A border must exist with objects inside it for the fill to work. Artistic letters such as those created in Coral Draw can be made to rise above the background as in sand-blasting. Select a line in the border. The border turns red.

LINEUP
LINEDWN
FILL
TEXT
TEXTCTR

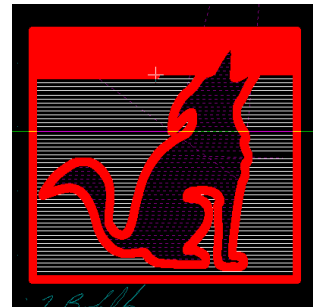


```
Command:fill, Fill Closed Item, Select: item 0 33 - line 2,4,5,063,4.063,5,1,2,1,2,40
item 0 25 - line 2,4,5,4,5,1,2,1,2,40 Set Fill Increment <0.125>:
Set Fill Increment <0.100>:.125 Set Fill Smallest Move <0.125>:
```

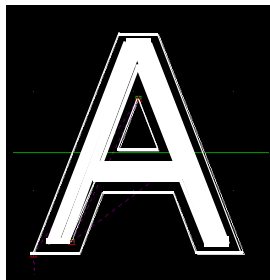
The command line asks for the fill increment. This is the path the cutting tool will move on. Use the radius of the cutting tool for this setting. If the cutter were .250 the increment setting would be .125. Next the command line asks for the smallest move. To prevent cutting into an object while removing fill, make the smallest move equal



A raster line pattern will appear within the border surrounding the object. This is the toolpath the machine will cut on to remove the background fill. Example on the right is an object directly filled without a contour tool path surrounding the object first.



The “A” has a contour tool path around and inside. The offset distance for the path is determined by the size of the cutting tool to be used. The offset contour is equal to the radius of the cutter.

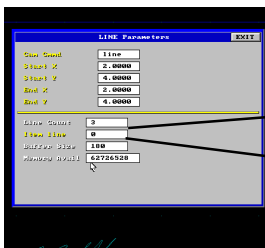
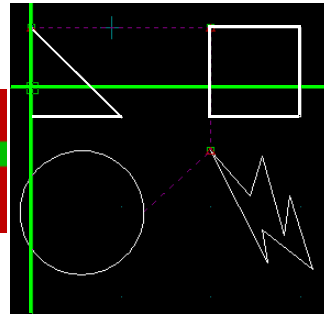


Contouring features can be found in Coral Draw. There are offset abilities in AutoCad also. Contour offset lines must be made around objects before the fill is applied.

List/Listfront

The list command will tell you the order of an item in a list the program assembles. This is the move and cut sequence the machine will execute. After clicking the command, the command line asks you to select an object. Once selected with the crosshair, a screen will appear. In this example, the triangle is selected.

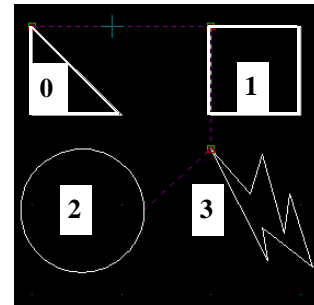
ZOOMP
ERASE
LIST
MOVE
COPY



The triangle is item zero. The list begins with zero and builds from there. Read the item line to see the position of

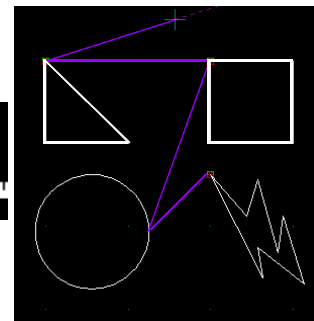
Item line 0

the item on the list. In the example on the right, the numbers show the order the items were drawn in and the order the machine will cut them in.

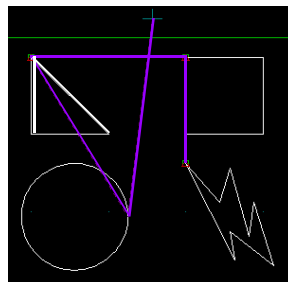


The listfrnt command makes it possible to select an item and place it at the beginning of the list so that it becomes zero. Notice the purple line connecting the objects. From its' start position, you can easily see the order the machine will move in. (0) triangle, (1) square (2)circle and (3)bolt.

CHGLNUP
SWAPEND
CHGORDER
LISTFRNT
JOIN



After selecting the listfrnt command, use the crosshair to select the object you wish to cut first. Immediately after the selection, the cutting order will change. Now, it will be circle, trian-



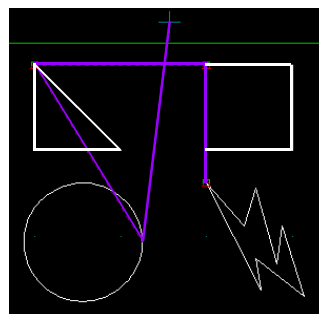
Item line 0 CIRCLE
Item line 1 TRIANGLE
Item line 2 SQUARE
Item line 3 BOLT

Change order/List front

The `chgorder` command allows you to select an item and insert it in the list under the previous item. The selection sequence is: the first selected item will follow the next selected

```
CHGLNUP
SWAPEND
CHGORDER
LISTFRNT
JOIN
```

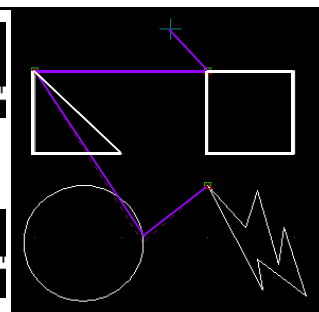
follow this item. We can change the list order from the one shown. Let's make the square first, triangle next, then the circle and the bolt last.



Use the `listfrnt` command and select the square to be the first item in the list. Then select the `chgorder` command. select the triangle and then the square. Now, the triangle is next in the list and follows the square. Again select the `chgorder` command. Select the circle then the triangle. Again select `chgorder`, select the bolt and then the circle. The bolt will follow

```
CHGLNUP
SWAPEND
CHGORDER
LISTFRNT
JOIN
```

```
CHGLNUP
SWAPEND
CHGORDER
LISTFRNT
JOIN
```



Text/Font

Selecting the text command causes the command line to ask for the start point of the first word. Select a point on the screen and left click.

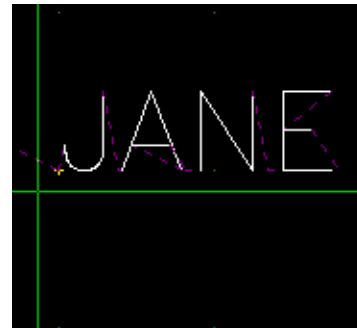


```
Background File is loaded
End of MECH CONFIG S# 994904
Command:text Start Point: , height<0.500>:
```

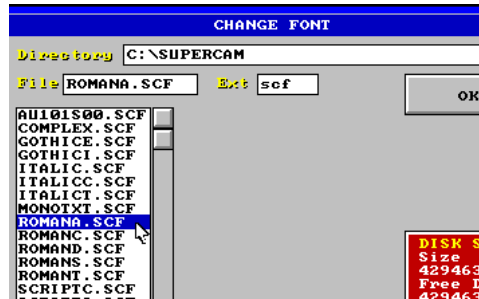
Next, the command line wants to know how tall the letters you want to engrave will be. The setting on the line is now one half inch. Press enter to accept or type in a new value and press enter.

```
End of MECH CONFIG S# 994904
Command:text Start Point: , height<0.500>:
text: JANE
```

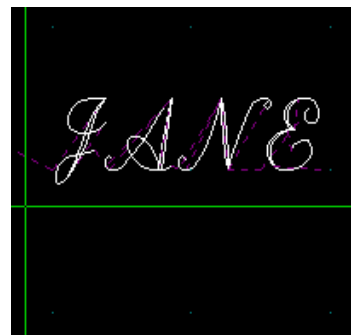
Having selected the height of the text, the command line asks that you type in the word or words you want to engrave. After typing, press enter and the text will appear on the screen. The text will appear above and to the right of the start point. It will be displayed in whatever font style is currently selected.



SuperCam contains fonts you can select for engraving. They create center line tool paths. Selecting the font command will bring up a menu screen. Left click on the font style you want and click the “ok” button.



A new font is selected. The text is changed from Romana to Scriptc.



Text center/Text fit/Reversing text

The text center command is an assist to placing a word or line of text in the middle of an area. Left click on the point you want to be center. Type height for text and a ratio value for height to length. Press enter. Type text and press enter.

```
Background File is loaded
End of MECH CONFIG S# 994904
Command:textctr Bottom Center: , Height(0.500): , Ratio(0.750):
```



The text fit command makes it possible to fit a word or line into a specific area.

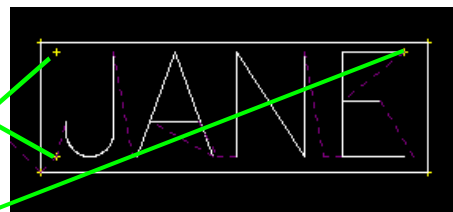


In the above example, a border has been drawn. Now, we want to place a name inside the border using “textfit”.

```
Background File is loaded
End of MECH CONFIG S# 994904
Command:textfit, bottom left:
```

```
Background File is loaded
End of MECH CONFIG S# 994904
Command:textfit, bottom left: , top:
```

```
End of MECH CONFIG S# 994904
Command:textfit, bottom left: , top: , end:
text:
```



Select the bottom left, top left and end at top right. Type text and press enter.

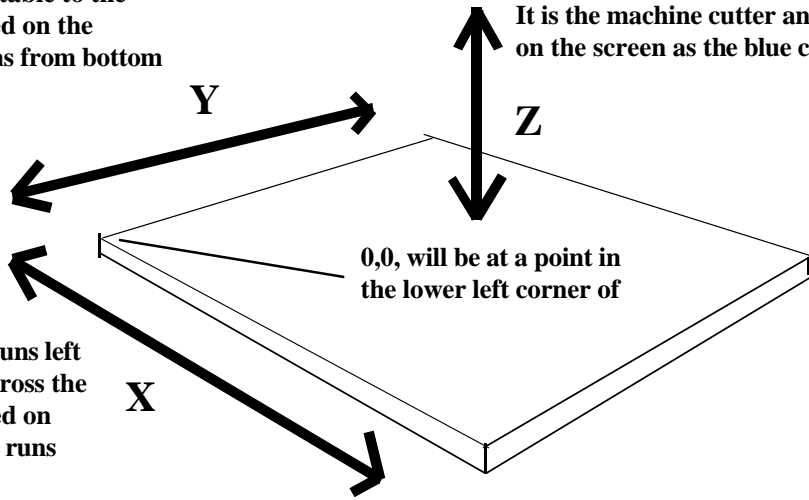
Reversing Text – Occasionally you may want to engrave on clear acrylic plastic from the backside. The words need to be reversed for engraving so they read correctly from the uncut surface. Instead of selecting the bottom left, select the bottom right and continue to top and on to end, which is top left.



BASIC MOTION CONTROL

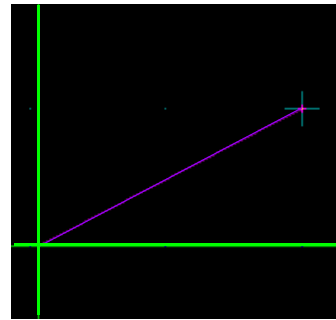
The Y axis runs from the front of the table to the back. Viewed on the screen it runs from bottom to top.

The Z axis runs up and down, away from and towards the table surface. It is the machine cutter and is viewed on the screen as the blue crosshair.

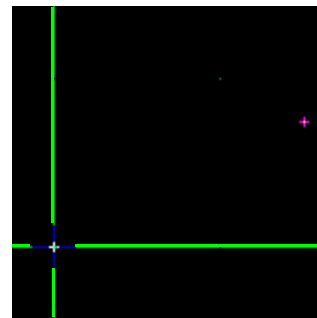


The X axis runs left and right across the table. Viewed on the screen it runs

The “Move To” command is one of the most often used in the program. It enables you to move the cutter to any point within the mechanical limits. The move will be in travel mode, with the cutter above the material. Select the command. As you move the mouse, notice the purple line connecting to the blue crosshair.

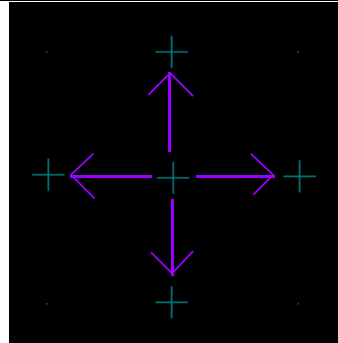


Left click on the point you want the machine cutter to go to. You will see the blue crosshair travel on your screen as the machine is moving. It will stop at the point you have selected. Be careful when moving the cutting head by taking into account any clamp or work holding piece that may be in the path you have selected.



Jog with keyboard/Jog increment/Jog increment Z/Plot buffer

The jog command moves the machine an incremental distance. Jog is done with the keyboard arrow keys. Jog is useful for exact cutter positioning and in moving the cutter away from the work area, depending on the amount of jog that has been set for each incremental move.



```
SETLIMIT Background File is loaded
LIMITFLG
JOGINCR End of MECH CONFIG S# 994904
JOGINCRZ Command:setjog Set X & Y Jog Increment <0.200>:|
SETLDLOC
```

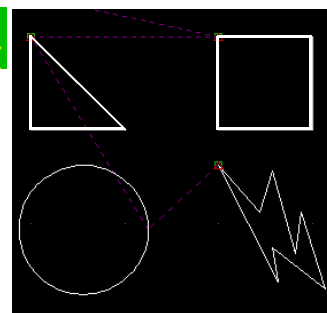
The jogincr command allows you to determine the distance of the jog. After selecting the command, the command line shows the present distance setting. Accept the present setting by pressing enter, or type a new value and press enter.

```
LIMITFLG Command:moveto, Mouse or KBD input:
JOGINCR Move To 4.0000, 3.0000
JOGINCRZ Command:setjogz Set Z Jog Increment <0.200>:|
SETLDLOC
MANUALZ
```

The jogincrz is to set the increment distance for the Z axis, for the up and down movement of the cutter. To jog Z, use the page up and page down keys.

```
PLOTBUFF Command:setzdepth, Z cut depth <0.5000 inch>: .125
PLOTCH Command:plotbuff on xytable, line_cnt 4
PLOTITER Yes - Cut to Depth 0.125000 - No to Repeat to Depth <N:Y>|
GRID
```

Plot buffer selects everything on the screen for cutting. A “yes” answer sets the machine to cut directly to the depth value set by the “Z depth” command. The machine will begin cutting immediately.



Plot buffer/Plot item

Attempting to cut too deep in a single pass causes stress on the cutting tool and the machine. It is often the case, that a router or milling machine must make several passes over the material. With each pass, the cut depth is increased deeper into the material. You are given the choice between single or multiple pass cutting. In the example of “plot buff” on the last page, if you had responded with a “no”, the command line would have asked for the final depth of cut and the depth of each pass (incremental) cut.

```
Command:plotbuff on xytable, line_cnt 4  
Yes - Cut to Depth 0.125000 - No to Repeat to Depth <N:Y>No  
Enter Final Cut Depth <1.000>: .500
```

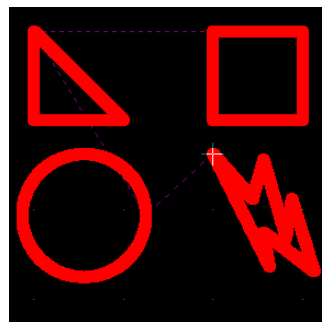
A “no” response causes the command line to ask for the final cut depth. Type and enter.

```
Enter Final Cut Depth <0.500>:  
Enter Incremental Cut Size <0.010>: .100  
PLOT OK <N:Y>?
```



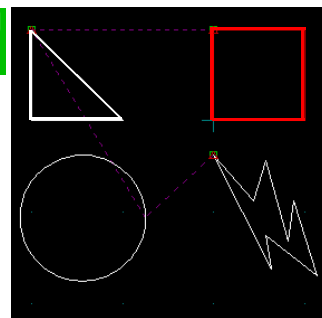
Next, it will ask for the cut depth of each pass cut. Type and enter.

In this example, the machine will go over the whole pattern 5 times, each time going one tenth of an inch deeper until the final depth of one half inch is achieved.

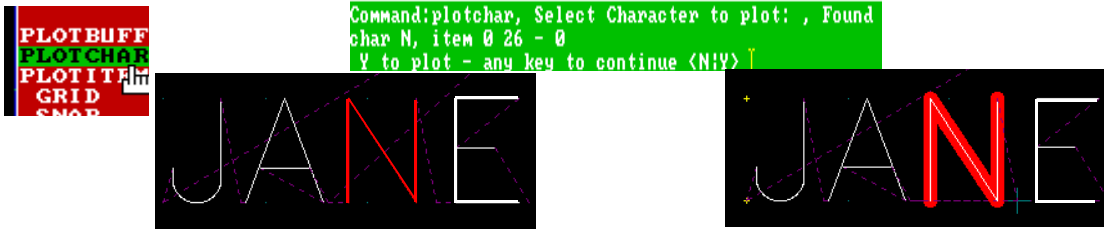


```
PLOTBUFF item 0 25 - line 4,4,5,4,5,3,4,3,4,4@  
PLOTCHAR Select Item <nxt!end>:  
PLOTITEM PLOT OK <N:Y>?  
GRID  
SNAP
```

The plot item command makes it possible to select a single item for cutting. The square is selected with the crosshair. The item turns red. Click the right button to end the selection. Right click to plot. Machine will start cutting.



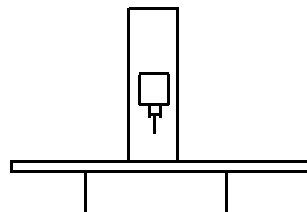
Plot character



The plot character command selects individual text letters. Select the letter to cut with crosshair. End selection by right clicking. Right click to plot. Machine cuts.

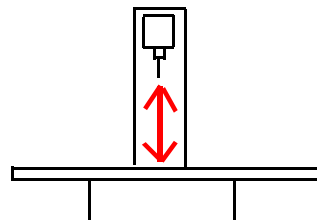
Set Z limit/Set Z

Two commands are used to set up the relationship of the cutting head to the machine table. First, mount an average length cutting tool in the chuck or collet. Use the “page up” and “page down” keys to jog the cutting head up and down.

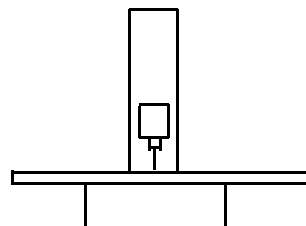


Now, use the “page up” key to move the cutting head to its upper mechanical limit. Leave a little space before you actually bump into the physical limit of travel.

Measure the distance from the table to the tip of the cutting tool. Say the total distance is 4 and 1/2 inches. This is the range of the Z axis.



Return the cutting tool to the table surface. The table is 0.00 and as the tool moves up, the numbers climb to the number that is your maximum limit. In the above example, 4.5 inches.

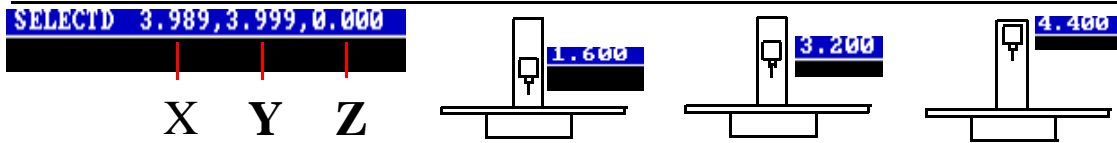


```
Command: SETZLMT
Z Limit Maximum <3.0000> : 4.500
Z Limit Minimum <-1.0000> : 0.000
```

“setzlimit” is a keyboard command. Type it in and press enter. The command line asks for Z limit maximum. Type 4.5 inches—press enter. Next, type value for Z minimum, which is 0.00 and enter. Now select the “setz” command. The current position of the tool tip is on the table. “setz” will allow you to assign a number value to the present Z position. Type 0.00 and press enter. (continued on next page)

SETXYLOC
SETZ
SET SU

Set Z/Set surface



You can monitor the position of the tool tip by viewing the blue bar with the white numbers at the top right of the screen. As you move the tool tip up the Z axis, you can read its position on the Z status line.

The positive number line can be used in all cutting applications. SuperCam was designed with cut depths being positive numbers. This works well because of a command we will explain called “setsurf”, and stands for set surface. Once the position of the surface of the material for cutting is recorded, all Z moves are derived from the surface value as the base reference. Cut depths are subtracted from the surface value.



A special word about “setz”. Leave it alone! You have created a fundamental relationship between your machine and the computer. Don’t change it. On the last page, “setz” was used as a set up tool. It created a common registration point between the machine and the program. Arbitrarily moving the Z position can lead to big problems. To do so is to shift the registration and could result in driving the cutter into the table.

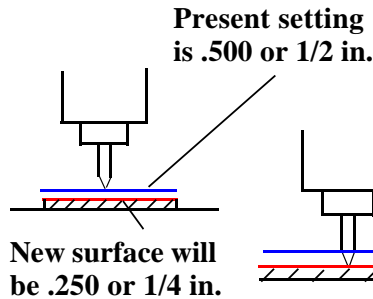


The “setsurf” command is used to physically lower the cutter tip until it nearly touches the material. Select “setsurf”. The left mouse button moves the tool down and the right button moves it up. Move the tool tip down, approaching the material. When you are satisfied that the tip is as close to the surface as possible without touching it, press enter. The surface number is recorded in the program. The tool moves to the previous surface position.

(continued next page)

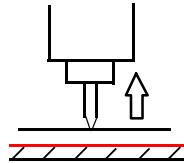
Surface/Up/Z altitude

As an example, say you were cutting 1/2 inch thick material and the surface has a setting of .500. You replace it with 1/4 inch material. You now need to set the new surface.

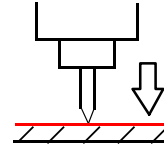


Select "setsurf". Left click in short intervals. You can make very small moves until the tip nearly touches the surface.

Press the enter key. The new surface number is recorded in the program.



Select the "surface" command. The tool will go down to the new position for 1/4 inch stock.



The tool retracts to the last surface position.



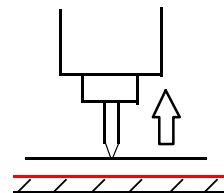
Selecting the "up" command will move the tool up from the surface to the Z altitude position. This is the distance the tool is away from the surface when traveling between cuts.



IG S# 994904
Z Altitude During Tool Movements (0.2000 inch): .1000

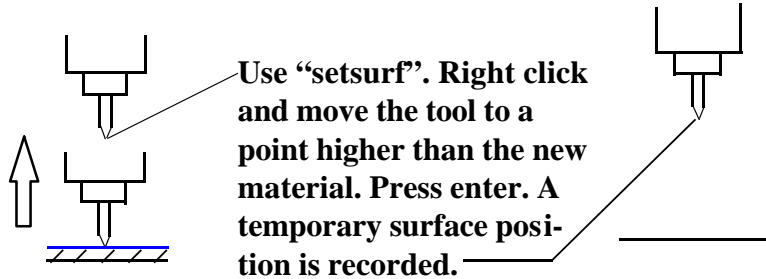
Select the "zalt" command. Accept the present value or type in a new one and press enter.

In the example above, we set surface on a new piece of material that was thinner than the previous. After pressing the enter key, the tool moved up to the previous surface position. Because the tool will return to its last surface position immediately after recording the new surface, care must be taken when loading a piece of material that is higher than the last thickness being cut. (continued next page)

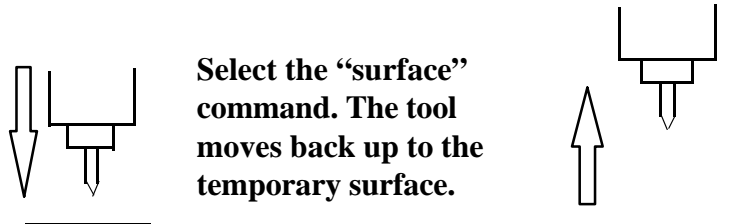


Conditions for setting surface/Zmax

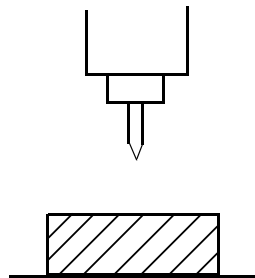
When setting a new surface that is higher than the last, create a temporary fictional surface above the new piece of material.



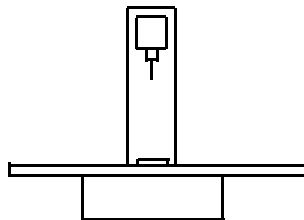
After pressing enter, the tool returns to the last surface position.



Load the new piece. Use the "setsurf" command and record the surface of the new material.

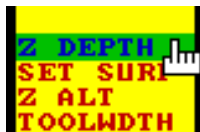


Use the "surface" and "up" commands to double check readiness for cutting. This procedure also should be used when changing to a longer cutting tool. Otherwise, you risk driving the cutter into the material when setting surface.



The "zmax" command will move the tool all the way up to the maximum height you have pre-set with the "setzlm" keyboard command.

Z depth/Down/Z travel/Z cut



```
round File is loaded  
f MECH CONFIG S# 994904  
Command:setzdpth, Z cut depth <0.1250 inch>: .06250
```

The “z depth” command allows you to enter a value for the depth you want the cutter to enter the material when cutting. In this example, the old cut depth was 1/8 th inch. The new depth will be 1/16 th inch. After the value is typed, press enter.



The “down” command moves the tool to the cut depth which has been set by the “Z depth” command. Obviously, this is a position below the material surface, so take care to have the cutter turned on when using this command.



```
Set XY Starting Travel Rate (0.041 - 1.668) in/sec <0.500>:  
Command:setztrvl  
Set Z travel rate (0.024 - 1.000) in/sec 0.500 ;
```

The “z travel” command sets the speed the machine will move when approaching or retracting from the surface of the material. The command line displays the recommended speed range. Accept the existing speed or type in a new value. Press enter.



```
End of MECH CONFIG S# 994904  
Command:setzcut  
Set Z Cut Rate (0.024 - 1.000) in/sec 0.500 ;
```

The “z cut” command determines the speed the tool will move when cutting downward into the material. The acceptable speed range is shown on the command line. Accept or type a new speed and press enter.

Manual Z

```
Command:setmanualz, Manual Z flag <Toggle;Enter> :AUTO  
Manual Z flag <Toggle;Enter> :MANUAL  
Manual Z flag <Toggle;Enter> :AUTO
```

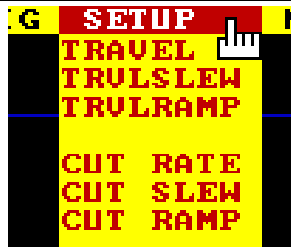


The “manual z” command is useful for those who are using a two axis system and do not have stepper motor control of Z. The program will stop whenever a manual adjustment of Z is required. Toggle between “manual” and “auto” with the spacebar.

X,Y, Motion/Start, Ramp and Slew Speeds

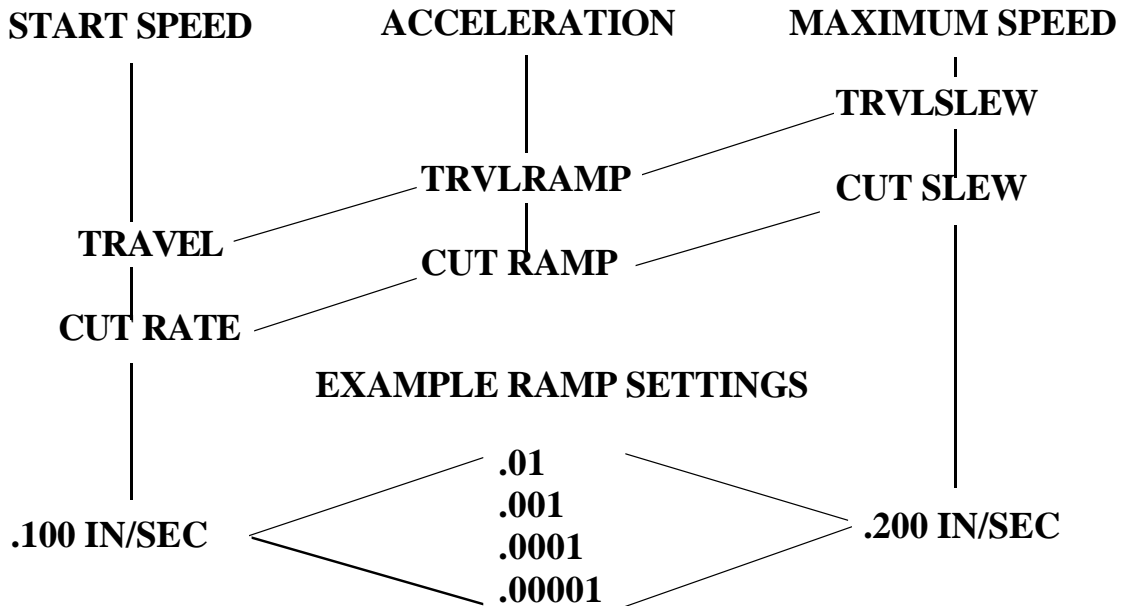
The purpose of varying the speed of the cutting head when either traveling or cutting, is to deal with the weight of the X and Y mechanics. Just as you can't move from zero to sixty miles an hour in an automobile suddenly, heavy XYZ systems need to start and stop with consideration for inertia. Thus, the concept of ramping is that of acceleration and deceleration at the beginning and end of machine moves. To accomplish this, a movement in time is expressed as a "start speed", a "ramp rate" and a "slew speed". Large machines create a greater concern for ramping than small ones. Even so, ramping can save wear and tear on

The setup menu shows a group of commands that work together in determining the speeds the machine will move in the X and Y axes.



Controls X,Y, travel.

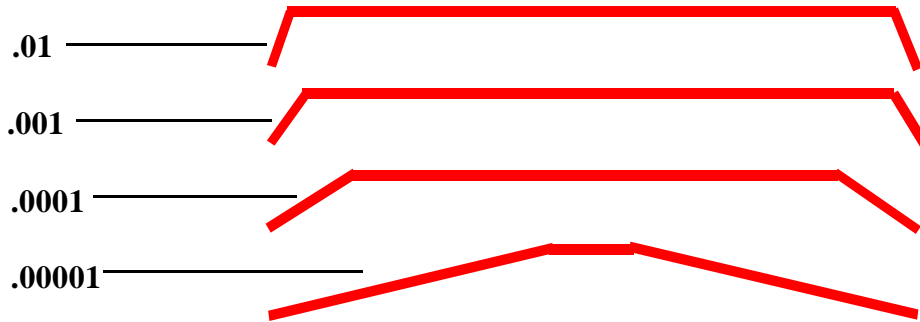
Controls X,Y, cutting.



We recommend the start speed be one half that of the slew speed.

(continued next page)

Ramp slopes illustrated/Machine configuration



In the above illustration, the general concept of ramp slope is shown. The red lines represent the amount of ramping that will happen in a machine move when going from cut or travel speed to slew speed.

At the bottom of the “setup” menu is the “mconfig” command. This stands for machine configuration. Many settings can be made on this screen without needing to select the individual commands associated with each. To change a value on the screen, left click on the box, backspace to clear, type a new value and press enter.



| EXIT | LIMITS | Configure Mechanism | DELAYS | PORT |
|--------------|---------------------------------------|----------------------|-----------------------------------|-------------------------------------|
| Z Altitude | <input type="text" value="0.1000"/> | inch Units | <input type="text" value="inch"/> | |
| Matl Surface | <input type="text" value="0.0000"/> | inch Display Flag | <input type="text" value="ON"/> | |
| Cut Depth | <input type="text" value="0.1250"/> | inch | | |
| TRAVEL Speed | <input type="text" value="0.2500"/> | 0.041 - 1.668 in/sec | | |
| TRAVEL Slew | <input type="text" value="0.5000"/> | 0.041 - 2.502 in/sec | | |
| TRAVEL Ramp | <input type="text" value="0.001000"/> | (in/sec)/step | | |
| Cut Speed | <input type="text" value="0.2500"/> | 0.041 - 1.668 in/sec | | |
| Cut Slew | <input type="text" value="0.5000"/> | 0.041 - 2.502 in/sec | | |
| Cut Ramp | <input type="text" value="0.000100"/> | (in/sec)/step | | |
| Z Travel | <input type="text" value="0.5000"/> | 0.024 - 1.000 in/sec | | |
| Z Cut | <input type="text" value="0.5000"/> | 0.024 - 1.000 in/sec | | |
| Toolwidth | <input type="text" value="0.1250"/> | inch | | |
| XY Jog Incr | <input type="text" value="0.2000"/> | inch | Z Jog Incr | <input type="text" value="0.2000"/> |

(Continued on next page)

English-Metric Speed and Grid Settings

The “units” button at the top right corner selects the measurement system to be used. In this example, measurement is in inches. The blue lettering expresses machine speeds. The first expresses inches of movement per second. The second, inches of movement per minute.



Millimeters per second.



Centimeters per second.



If centimeters were selected, it would show on the blue line at the top of the screen. The grid must be set to centimeters. Select the “grid” command.

```
Background File is loaded
End of MECH CONFIG S# 994904
Command:grid, Spacing < ON 2.5400 cm >: 1
```

Type a value of 1 and press enter.

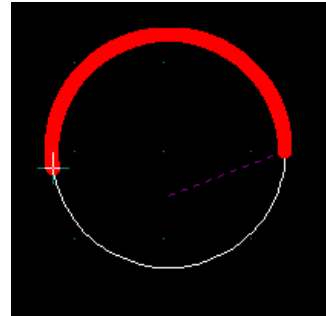
```
Command:grid, Spacing < ON 2.5400 cm >: 1
Grid Z Positon <2.540000 cm >: 1
Command:
```

Again, type 1 for Z and press enter.

You can return to the inch unit base using the same procedure. After selecting “inch” on the configuration screen, select the “grid” command and type a value of 1 for XY and again for Z.

Configuration /Screen display/Port mode/Motion delays

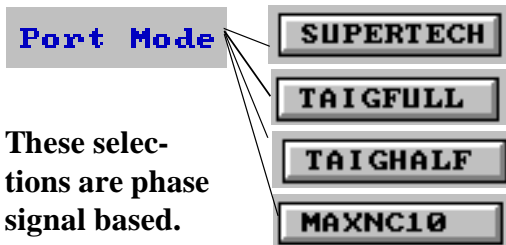
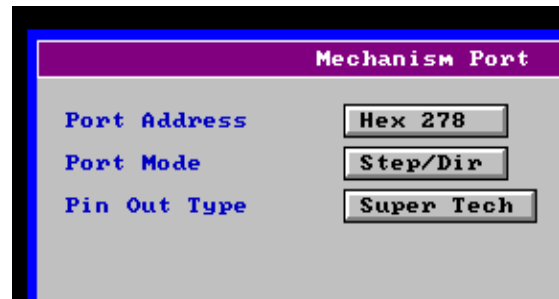
The display flag on the configuration screen is the same command as the keyboard shift F2. During cutting, viewing tool movement may not be needed. Screen viewing, can at times, slow machine action.



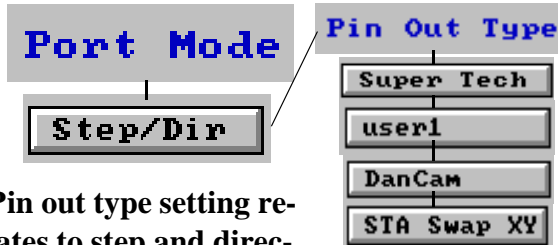
The top right corner button activates the port screen.

15 port addresses are available. The most commonly used are:

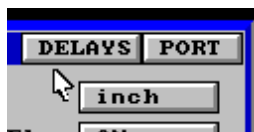
- HEX 3BC
- HEX 378
- HEX 278



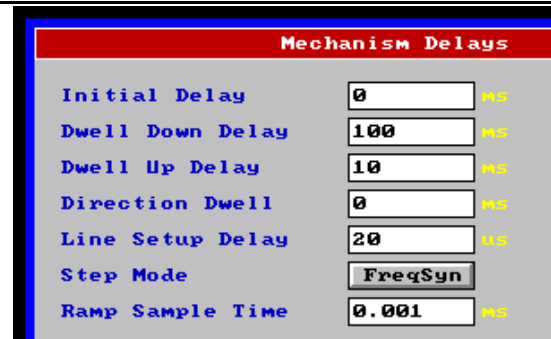
These selections are phase signal based. No additional selection is nec-



Pin out type setting relates to step and direction signals.

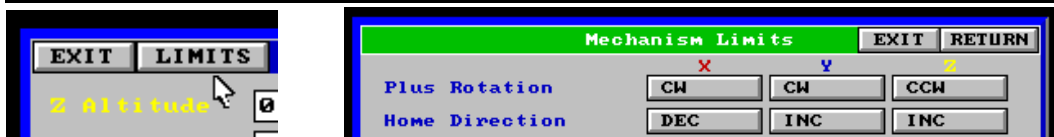


Selection of the "DELAYS" button on the configuration screen, makes possible the insertion of small time pauses between specific moves. Values are in milliseconds. (Continued next page)



Configuration screen/Motion delays/Limit switches setup/Home

| | | |
|------------------|--|---|
| Initial Delay | <input type="text" value="0"/> MS | After the machine has moved into position to cut, a delay can be inserted before it will |
| Dwell Down Delay | <input type="text" value="100"/> MS | After the tool plunges to cut, a delay can be inserted to allow the tool to settle in before cut- |
| Dwell Up Delay | <input type="text" value="10"/> MS | At the end of the cut, the tool can be paused before retracting. |
| Direction Dwell | <input type="text" value="0"/> MS | The tool will pause before changing direction. |
| Line Setup Delay | <input type="text" value="20"/> MS | Relates to the length of the parallel port cable from computer to controller. 20 ms sufficient for 50 Ft. length. |
| Step Mode | <input type="button" value="FreqSyn"/> | Computer must be 200 MHZ pentium to utilize frequency synthesis. Helps machine run smoother and faster. Clicking on the button will bring up "CLASSIC" for older computers. The ramp sample time is for frequency synthesis only. |
| Ramp Sample Time | <input type="text" value="0.001"/> MS | |



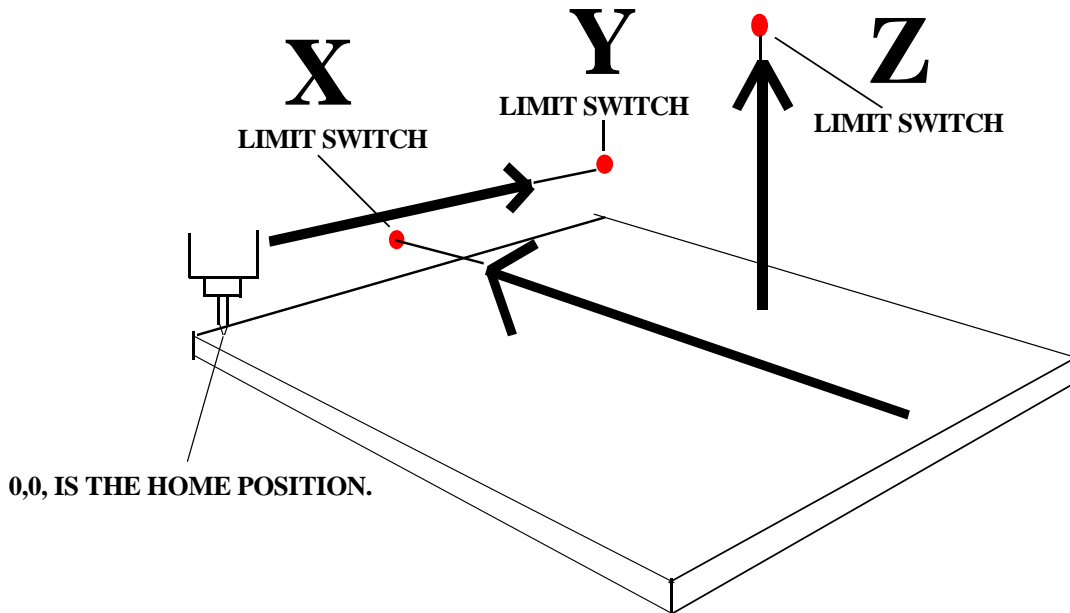
Select the limits button. The mechanism limits screen appears. Plus rotation means a positive gain will be made on the number line for that axis. You can reverse motor direction between clockwise and counter-clockwise to accomplish that end, depending on the machine.



The home direction is the direction the machine will move in to encounter the limit switches. Incremental is going up the number line. Decremental is going down on the number line. In this example, when the "home" command is selected, the axis motors are instructed to seek the limit switches. X will seek decrementally, Y and Z will seek incrementally.

(Continued next page)

Configuration screen/Limit switches/Home



Selecting the “home” command sends the mechanism moving to each of the limit switches. X is moving decrementally, Y and Z move incrementally.

| | X | Y | Z |
|------------------|--------|---------|--------|
| Home Switches At | 0.0000 | 16.0000 | 2.0000 |

As the limit switch in each axis is touched, the signals tell the program that these points in space have been reached.

| | X | Y | Z |
|------------|--------|--------|--------|
| Home Go To | 0.0000 | 0.0000 | 2.0000 |

Next, these are the destination points the motors will move to. This is the home position. Since the limit switches are usually close to the motors, X will not move because the limit switch is its home position. Now Y will move 16 inches away from its limit switch to the front of the table and Z will remain elevated for travel.

Set limits/Set load location/Step size, start & max per sec/Backlash compensation

| | X | Y | Z | |
|---|-----------|--------|--------|--------|
|  MCONFIG S COMPFLAG SETCOMP SETHPINC SETLIMIT LIMITFLG JOGINCR | Limit Max | 7.0000 | 5.0000 | 4.5000 |
| | Limit Min | 0.0000 | 0.0000 | 0.0000 |

A shortcut to using the “setlimit” command is to enter the values on the machine configuration screen. Be careful about the Z limit settings as they relate to the Z position as determined by “set z”.

| | X | Y | Z | |
|--|---------------|--------|--------|--------|
|  JOGINCRZ SETLDLOC MANUALZ ADAPTER | Load Location | 7.0000 | 5.0000 | 0.0000 |

The load location is a point away from the material mounting area. It is a point the cutter is moved to in order to remove finished work and load new material.

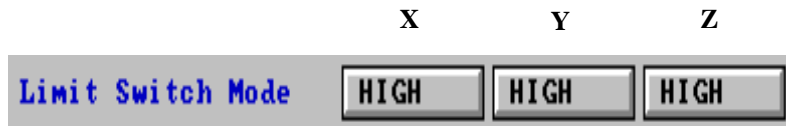
| | X | Y | Z | |
|---|-----------------|----------|----------|----------|
| Recommended settings for stepper motor operation. | Max Steps/Sec | 1200 | 1200 | 1000 |
| | Start Steps/Sec | 800 | 800 | 800 |
| | Step Size | 0.002085 | 0.002085 | 0.001250 |

| | X | Y | Z | |
|--|---------------|----------|----------|----------|
|  MCONFIG SET COMPFLAG SETCOMP SETHPINC | Backlash Comp | 0.000000 | 0.000000 | 0.000000 |

Due to wear in the lead screw nuts, some “slop” will occur when the carriage reverses direction. The variation can be measured and entered into the program. It will compensate for the backlash error. This can be done with the “set comp” command. Below, backlash compensation can be turned on or off with the “compflag” command or the button on the configuration screen.

| | | |
|---|--------------|-----|
|  MCONFIG SI COMPFLAG SETCOMP SETHPINC | Compensation | OFF |
|---|--------------|-----|

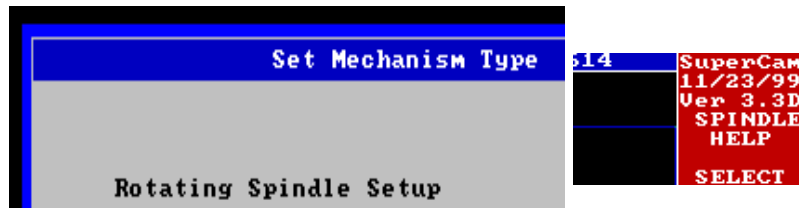
Switch mode/Table limits on-off/Mechanism type/



The limit switch mode allows alternative settings, depending on the type of switch that is being used, either open or closed when touched. Note that synchronized Y motors require high settings. Use switches that send + 5 volts to the computer when engaged.



The machine limits can be turned on or off with the “limitflg” command or the button in the lower left corner of the limits screen. If you need to move the cutter beyond the mechanical limits, you can turn the limits off and move beyond the boundary.



The keyboard command, “setmech” will bring up the mechanism type screen. Press the spacebar to move through the possible selections for machine type. Press enter to select. On the SuperCam main screen, the machine type is displayed at the top of the quick select menu.

Two Axis Plasma Torch



In this example, a two axis plasma torch is selected. Notice that the quick select menu is reconfigured with “on and “off” commands.

Relays/Setcords/Status screen/M-status screen

It is possible to control two relays with SuperCam. Relay “A” is toggled on and off with the keyboard “Home” key. Relay “B” is toggled on and off with the keyboard “End”

CNC

Note that the relays are active in the step/direction port mode only. They are accessed during machine operation by running cnc text files or when operating a plasma torch set up.

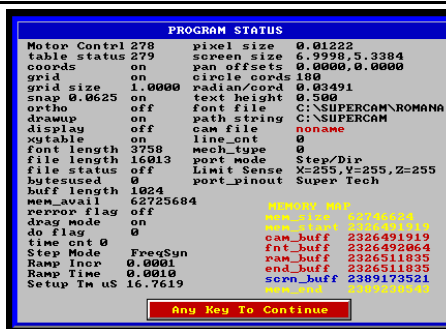
“setcords” is a keyboard command. It sets the number of slices a pie would be cut into, or the number of divisions in a circle or arc. Circles and arcs are combinations of short

```
Command: SETCORDS
Machine Cords < 180>:
Display Cords < 90>:
```

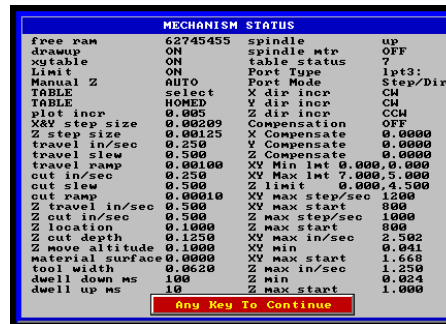
There are two settings. The first is the number of segments the machine will cut in doing a circle or arc. The second is the screen display.



The “status” command will display the program status screen.



The “mstatus” command shows how the machine is configured.



Home/Tool change/Set XY location/Zero XY



The “home” command is only applicable to machines that have limit switches installed. Selecting the command will move the cutting head to the position you have set as the home position. Z will move to a position you have set on the limits screen.

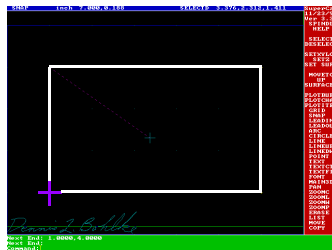
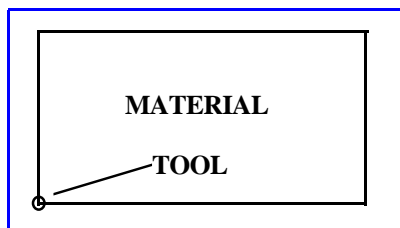


The “toolchng” command moves the cutting device to the front of the machine for convenient access.



The “setxyloc” is very useful in aligning the relationship of the cutter tool and the material to the item to be cut as it is displayed on the screen. First, the tool must be positioned over an identifiable point on the material surface, such as the lower left corner. Use the “move to” command and the jog keys to move the cutter tip over the point.

Now, click on the “setxyloc” command. Move the crosshair over the lower left corner of the drawing on the drawing on the screen and click. The tool position and material are matched to the crosshair and the drawing.



The “zeroxy” command will move the cutting tool to the zero point on the number line of the X and Y axis.

Find location/Origin/Motor control

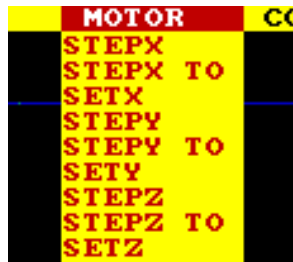


The “findloc” works in conjunction with the limit switches. It will refresh the machine position so that the screen and table coordinates match. This is useful if you suspect the machine may have lost steps and position.



“origin” is a very basic command that would not be used in common applications. It will zero all counters when applied. It has special uses mentioned in other areas of the manual.

Listed under the motor menu are a set of commands that are elemental. They can be used to test various stepper motor machines and set-ups.



SET X
SET Y
SET Z

ASSIGNS A NUMBER POSITION ON THE AXIS.

STEP X
STEP Y
STEP Z

MOVES FROM EXISTING LOCATION TO ONE ASSIGNED BY KEYBOARD ENTRY.

STEP X TO
STEP Y TO
STEP Z TO

MOUSE CONTROLLED MOVEMENT.
LEFT KEY INCREMENTS.
RIGHT KEY DECREMENTS.
ENTER/SPACE/ESCAPE EXITS THE COMMAND.

X
Y

USE MOUSE OR KEYBOARD TO MOVE.
‘\`’ KEY WILL ZERO THE AXIS.

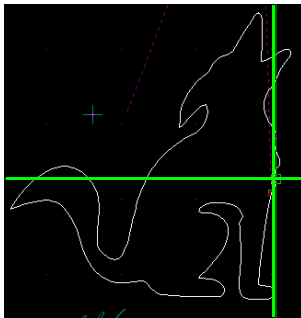
Lead in/Join

The “leadin” command is most used in plasma torch work. The torch begins cutting on a line that merges with the object being cut. This avoids over burn on the object line.

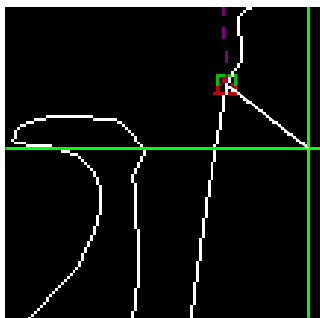
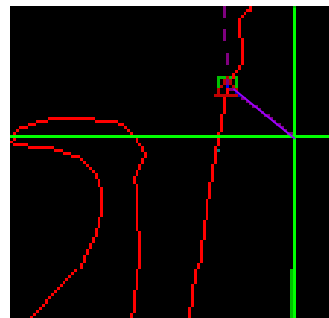


```
Select Item <nxtlend>:  
Command:leadin Select Graphic Item to Attach Lead In Too:
```

After selecting the “leadin” command, the command line will ask that you identify the graphic item the lead in line will attach to.



Select the graphic item with the crosshair. Item lines turn red. As the crosshair is moved away from the selection point, the purple connecting line will indicate the angle and length of the lead in line. When you have the angle and length desired, left click.

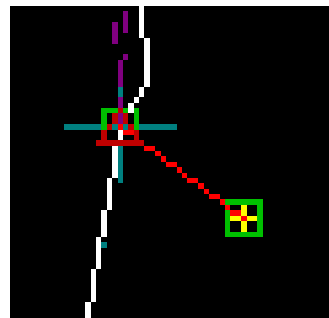


The lead in line and the graphic item lines turn white. Now, the lead in line must be joined to the object lines. Select the “join” command.

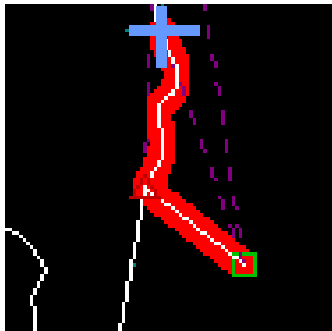


```
Length of Lead In <0.375>:0.4858  
Command:join, JOIN items, first line select:
```

The command line asks you to select the first of the two lines to be joined. Left click on the lead in line. It will turn red. Then left click on the object line and it will turn red. Command line asks if you want lines joined. Answer “yes”. (Continued next page)



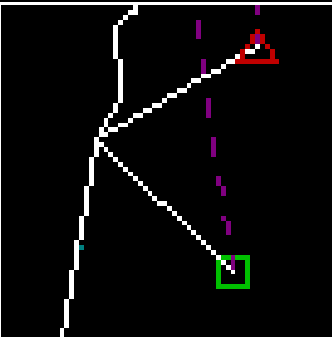
Leadout Line



You can test the lead in by selecting the “plot item” command and viewing the tool path.



The “leadout” command provides an exit path for the tool. The procedure is the same, except when joining the lead out, select the object first and then the leadout line.

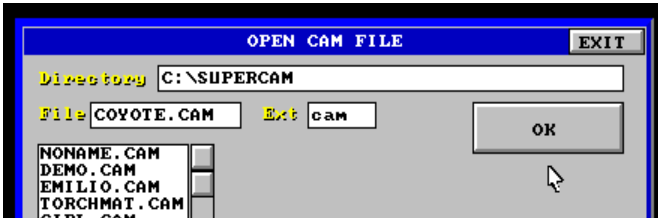
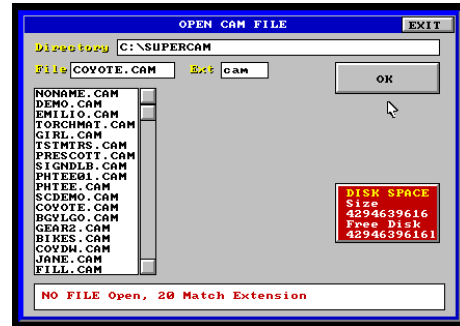


Notice the green square indicates the line start point for the lead in line and the red triangle shows the end of the lead out line.

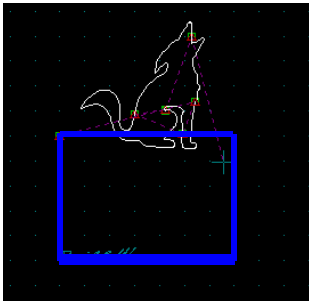
Files/Open/Close



The “open” command presents the .cam file screen. Files with the .cam extension were originated in SuperCam or, were imported files that were modified



The mouse moves the white pointer over the screen. Find the file you want in the scroll menu. Select with the pointer by left clicking. File will appear in the file window. Left click the “ok” button to bring the file



Remember that the contents of a file may exist beyond the view of the mechanical limits. Use the “zoom center” command to see the big picture. (Page 6) The “move” command will bring the object into the limits. (Page 7) and the “scale” command to size the object. (Page 8).

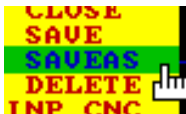
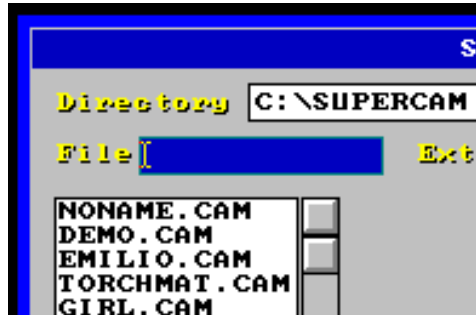


The “close” command will remove the currently displayed file. If the file has been modified, the command line will ask if you want to save the changes.

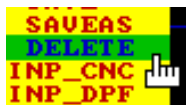
Files/Save/Save as/Delete



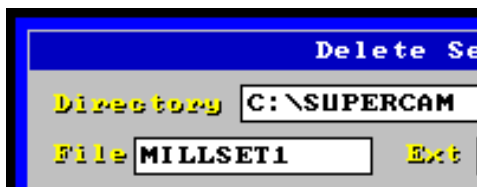
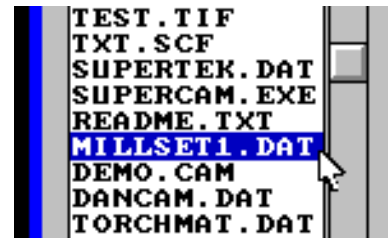
The “save” command saves a list of items in memory as a .cam file to the present sub-directory. It uses the default file name, or type the name of the file in the file window. Click on the file name window. Back space, or continue left clicking to clear the window. Type the name you want, press enter and left click on the “ok” button.



The “saveas” command saves the ram buffer as a user specified file name. This command allows you to duplicate the file with another name, should you want a second modified version.



The “delete” command will eliminate any file in the SuperCam sub-directory, regardless of the file extension. Select “delete” and the screen appears. Scroll to find the file and select with the pointer. It will appear in the file name window above.



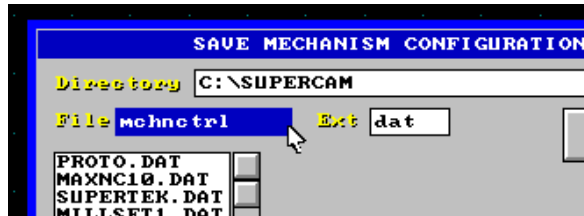
```
Command:close Buffer cleared!  
Command:delete  
Delete File C:\SUPERCAM\DBOX.CAM <Yes!No>
```

Left click on the “ok” button. The command line will ask if you are certain that you want the file deleted. Respond “yes” and the file is gone.

Files/Save mechanical config./Read mechanical config./Background/End/Quit



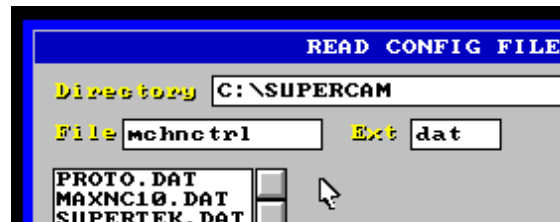
“savemcfg” makes a file of how your mechanism is currently configured.



The files will have a .dat extension. It can be used to build files for speed configurations for different materials, or different machines. Backspace to clear the selected window. Type name for the current machine setup. Press enter, click “ok”.



The “readmcfg” command presents a screen for selecting from the various machine configuration files you have created. The machine will be reconfig-



Scroll and select the setup file you want. Left click on the “ok” button.

The “backgrnd” command enables you to select any file for display as a reference. It is not a part of the drawing you are making. However, you can draw on top of it when making a new file.



To remove the background, bring up the screen again with the command, but select nothing. Left click on the exit button. Press the F1 key to clear the screen.



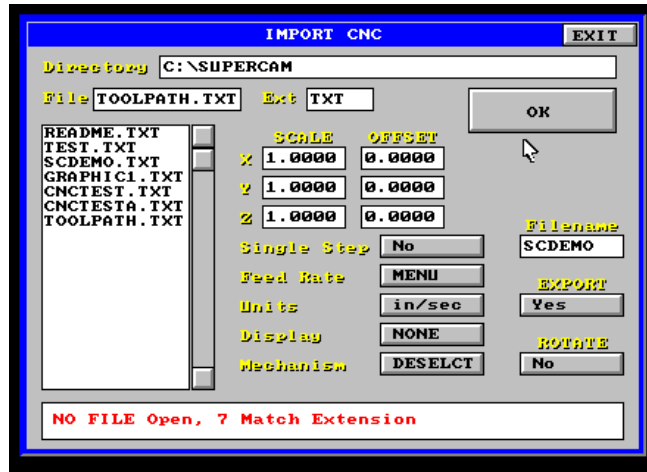
```
Command:quit  
Command:  
EXIT SuperCam? <N|Y> |
```

The “end” and “quit” commands are the same. Also, pressing the ESCAPE key has the same result.

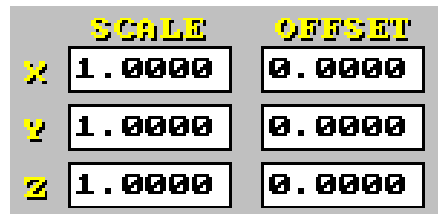
Importing CNC files/Scale/Offset position/Single stepping



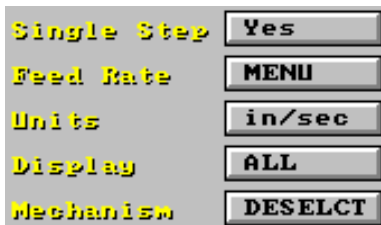
The “inp_cnc” command will bring in an existing CNC file which will have a .TXT extension. The import cnc screen has setting boxes that determine how the file will be displayed and executed.



The cnc file can be made to cut a larger or smaller pattern by scaling. For example, if the scale values were set at 2.0000 in X,Y, and Z, the part would be twice as large. Should you want to move the cutting position from that in the file, record the offset value away from the existing position in each axis.



```
X0.5000 Y0.5625
X2.6250 Y0.5625
X2.6250 Y2.6875
```



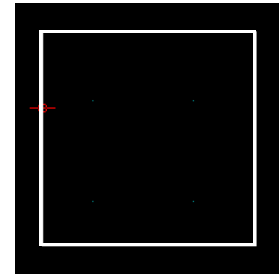
Single stepping means only one line of code will be executed at a time. Press the spacebar repeatedly to move from line to line. Table must be deselected to see file displayed. When the display is set to “all”, the coordinates are shown on the command line. The right hand screen shows the line number.



Importing CNC/File display/cutting action/Menu feed rate/Auto feed rate

This is the more typical setup for importing a file. Because the mechanism is deselected, the display is “all” and we are not single stepping, the file will be

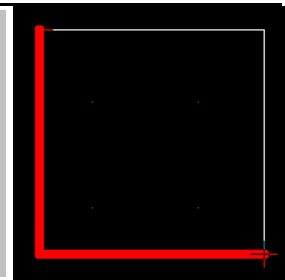
| | |
|-------------|---------|
| Single Step | No |
| Feed Rate | MENU |
| Units | in/sec |
| Display | ALL |
| Mechanism | DESELCT |



You can leave the display up, return to imp_cnc screen, “select” mechanism. Now you can execute machine motion while having the total display up.

This is an illustration of the machine at the half way point in cutting a displayed file. Note that the mechanism is selected.

| | |
|-------------|---------|
| Single Step | No |
| Feed Rate | MENU |
| Units | in/sec |
| Display | ALL |
| Mechanism | SELECTD |



Feed Rate

MENU

When “menu” is selected for the feed rate, the values you have set on the machine configuration screen determine the speed of machine moves.

| | |
|--------------|----------|
| TRAVEL Speed | 1.6680 |
| TRAVEL Slew | 2.0000 |
| TRAVEL Ramp | 0.001000 |
| Cut Speed | 0.3500 |
| Cut Slew | 0.7500 |
| Cut Ramp | 0.000100 |

Feed Rate

AUTO

When “auto” is selected, the values that were in place on the mechanical configuration screen at the time the CNC file was created, will be executed by the machine.

Importing CNC/Units/Export/Rotate/Red crosshair

Units

in/sec

This value must match the “units” value for the configuration setup you are using. The grid must also match. If the original file was a metric drawing, when the CNC file was created using “exp_cnc”, the configuration screen should have been configured in the same metric units. Both the grid and rate would be in millimeters or centimeters.

The “export” button is used in digitizing. See that section in the manual

Filename

TOOLPATH

EXPORT

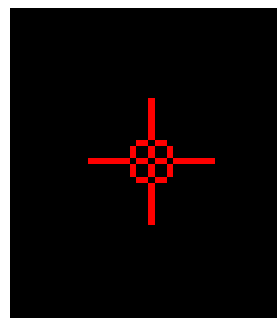
No

The “rotate” command can be an assist in positioning the pattern for cutting on a particular machine or for conserving material. It rotates the pattern 90 degrees.

ROTATE

No

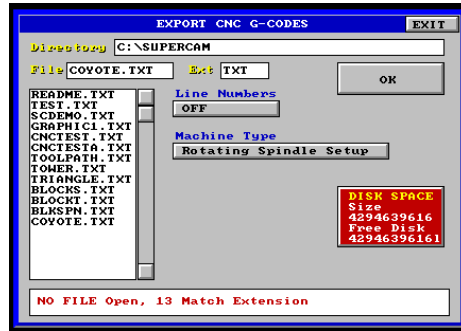
Just a side note. If you “escape” the program while the tool is cutting, this style of crosshair in red will remain displayed. This is because the tool was down when the cut was interrupted. Use the “up” command to lift the tool. The crosshair will turn light blue indicating the tool has returned to the travel position above the material. Use F1



Exporting CNC/Setup CNC/



The “exp_cnc” command is used to create a cnc file. Any file you have displayed on the SuperCam screen resides in the SuperCam sub-directory and can be made into a cnc code file.



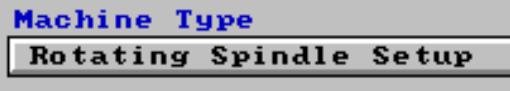
Remember that the travel and cut speeds you presently have set up on the configuration screen will be recorded into the cnc file. This may not be important since you will be able to select a configuration when importing the file for cutting using the “menu” button on the import screen. If the file is metric, it will be im-



The “setup_cnc” command brings up the cnc code configure file. You can determine what code you will assign to each function. Click on the “save”



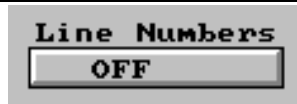
In the lower right corner of the code configure file is a button indicating the machine type. A file can be made for spindle, torch or laser cutting.



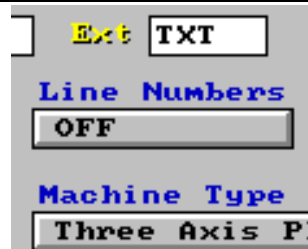
On the cnc export screen you will find the same button that can be used to toggle through and set the machine type. The buttons are connected. Changing one changes the other.

(Continued next page)

Exporting CNC/Line numbers/Creating CNC file/



Also in the lower right corner of the cnc code screen is a button for writing line numbers for code. It generally is not necessary and adds to the size of the file. It is an



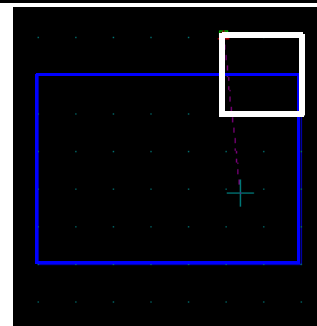
Line numbers can be selected on the “exp_cnc” screen also.

Example of Creating a CNC file

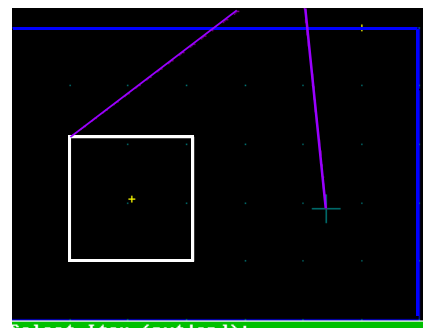


A source is required to create the cnc file. It can be a drawing made on the SuperCam screen, which would have a .cam extension after the file name. Or, it can be a file brought in from another program such as AutoCad, which would have a .dxf extension. If it was done in Coral Draw, it would have a .plt extension. In this demonstration, a .dxf is made in AutoCad and imported.

Notice that the pattern is outside the mechanical limits of the machine. It was drawn in AutoCad on a “inch” grid. It fits into the present machine configuration.

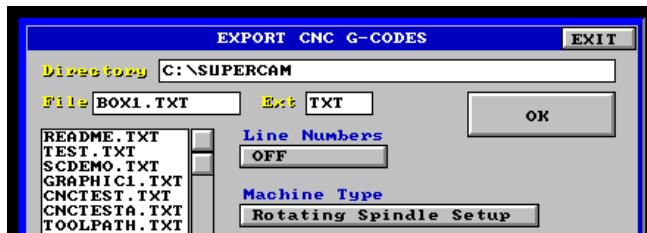
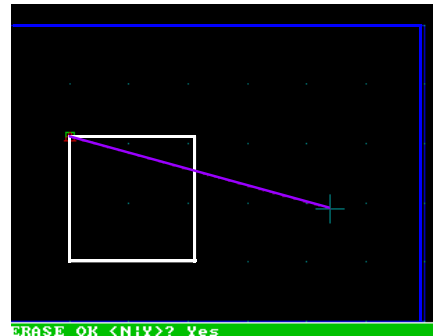


Use the “move” command and move the pattern into the mechanical limits. In this case the lower left corner of the pattern is placed on the 1,1, coordinate point. Notice that the machine travel path still goes beyond the limits. This means another object may exist in the file.



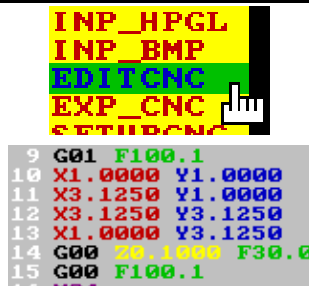
Creating CNC file/Example

Use “zoomc” to look at the big picture outside the limits. Find the origin of the travel line and use the “erase” command to get rid of it. This will assure that the header information in the file is for points within the mechanical limits.



Select the “exp_cnc” command. Type in the name you want the file called in the file window. Select the machine type. Click on the “ok” button. The cnc file you have just created will appear in the “imp_cnc” file menu.

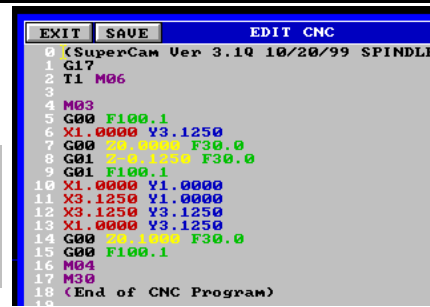
To view the code list that has been generated, select the “editcnc” command. Select the file in the scroll menu and click the “ok” button.



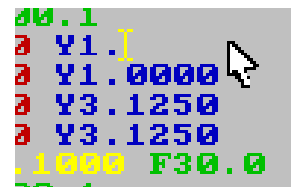
```

9 G01 F100.1
10 X1.0000 Y1.0000
11 X3.1250 Y1.0000
12 X3.1250 Y3.1250
13 X1.0000 Y3.1250
14 G00 Z0.1000 F30.0
15 G00 F100.1

```

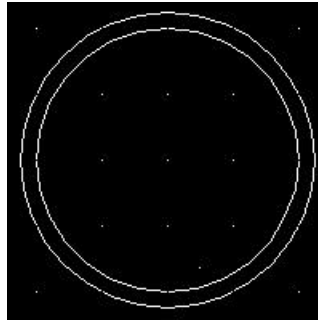


The “editcnc” command enables direct line revision of the cnc file. Use the mouse to activate the cursor. Backspace to erase and then type the new value. Click the “save” button to record the revision. You can create cnc files from scratch using the cnc editor screen if you want. Simply start with a blank screen in SuperCam, bring up the “exp_cnc” screen and give the file a name. Then you can write the list of machine moves the old fashioned way.



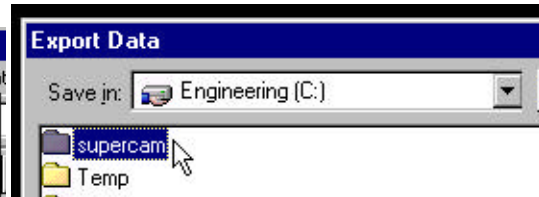
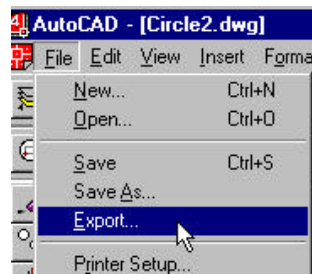
Creating DXF. files in AutoCad

Most users who are importing .dxf files will be making their original drawings in AutoCad.

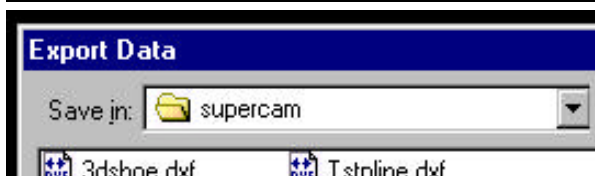


In this example, an offset tool path has been created in AutoCad. The file was saved in AutoCad as CIRCLE2.DWG.

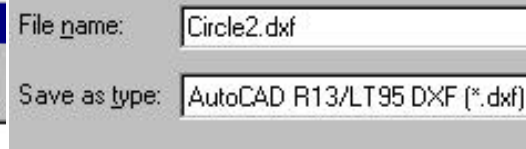
Open the Circle2.dwg file. Then, under file on the AutoCad screen, select “Export”.



Select the directory SuperCam resides in. Select SuperCam. Click “Open”.

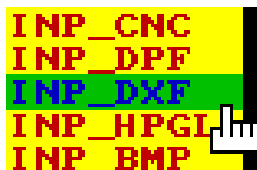


The file will be exported to the SuperCam sub-directory.



At the bottom of the export data screen is a scroll down menu. Select the AutoCad R13. Click “save”.

The file is exported



The dxf. file just created is ready to be viewed on the SuperCam screen by using the “imp_dxf” command.

