

# ShopCAM

## 2004

Copyright 2004 D. Broderick Software; Auburn Michigan

User Guide .....	1
Section 1 Getting started .....	5
File Types.....	5
Folders Or Directories .....	5
Learning To Use The System .....	6
The Screen Area .....	6
Layers .....	7
Coordinates .....	7
Zero Degrees.....	7
Five Steps of Part-Programming .....	8
Section 2 Operation Dialog .....	10
How to perform an Operation.....	10
Standard Operation Dialog.....	11
Operation Dialog .....	11
Standard parameter summary.....	11
Standard Lathe parameter summary .....	14
Determining The Tool Side .....	15
Section 3 Setup File, Tool & Material libraries .....	15
The Setup File .....	15
Section 4 Making the Machine Code file .....	18
Post Processors.....	18
Section 5 Picking geometry.....	19
Pick Modes .....	19
Picking with a Window .....	19

## Shopcam User Guide

Picking By Selecting .....	19
Picking via Chain.....	20
Pick lasso .....	21
Pick last layer.....	21
Select layer .....	21
Query Geom information.....	21
<b>Section 6 Geometry Creation.....</b>	<b>22</b>
CAD Files .....	22
Shopcam Geometry.....	22
Quick Keys Chart .....	22
Patterns .....	23
Lettering.....	23
<b>Section 7 Geometry Editing .....</b>	<b>24</b>
Trimming and Breaking.....	24
Trim Modal.....	25
Breaking.....	26
Copying and Rotating.....	26
<b>Section 8 Making Groups .....</b>	<b>27</b>
Cleaning up geometry.....	29
<b>Section 9 Approach methods .....</b>	<b>30</b>
NONE.....	30
OFFSET .....	31
LEAD-IN .....	31
PREDRILL .....	32
RAMP .....	32

## Shopcam User manual

START POSITION.....	33
Miscellaneous Commands.....	34
GRAPHIC CURSOR TYPES .....	34
Resequencing.....	35
Section 10 The N/C Editor .....	35
Section 11 The Operation Manager .....	36
Operation Manager Layout .....	37
Saving an Operation .....	37
Section 12 Tool Library.....	38
Saving and Deleting a Tool.....	38

## Section 1 Getting started

You should have the system installed and licensed prior to starting. This manual will try to demonstrate most of the features in the system. Shopcam is very flexible with several different ways to complete the same task. This Users Guide is provided as an example of ways to use the system. This manual is not a complete technical reference. Commands and Menus are covered in detail in the Technical Reference Manual. If you are new to CAD and CAM, read Section 6 (Geometry Creation) & Section 7 (Geometry Editing) before proceeding.

### File Types

The following file types are used in Shopcam:

PRT	Shopcam part-program that contains a drawing or tool path
TAP	The 'G-code' text file for the machine control
MCH	The post-processor "Post". Creates a tap (G-code) file from the part file
SET	Setup file; contains the post, tool & material libraries & default settings
TLB	Tool library. For storing tool information
MTL	Material library. For information about the material being machined

Most controls use their own extension for the G-code file. You may need to rename the .tap file for the control to recognize it.

### Folders Or Directories

The programs and files are organized in folders on the computer. Folders are also referred to 'Directories & subdirectories'. The system will be stored in a directory named "SHOPCAM". The files listed above, can be accessed as:

C:\PROGRAM FILES\DBS\SHOPCAM\PARTS\	.PRT
C:\PROGRAM FILES\DBS\SHOPCAM\TAPES\	.TAP
C:\PROGRAM FILES\DBS\SHOPCAM\POSTS\	.MCH
C:\PROGRAM FILES\DBS\SHOPCAM\MTRL\	.MTL
C:\PROGRAM FILES\DBS\SHOPCAM\TOOLS\	.TLB

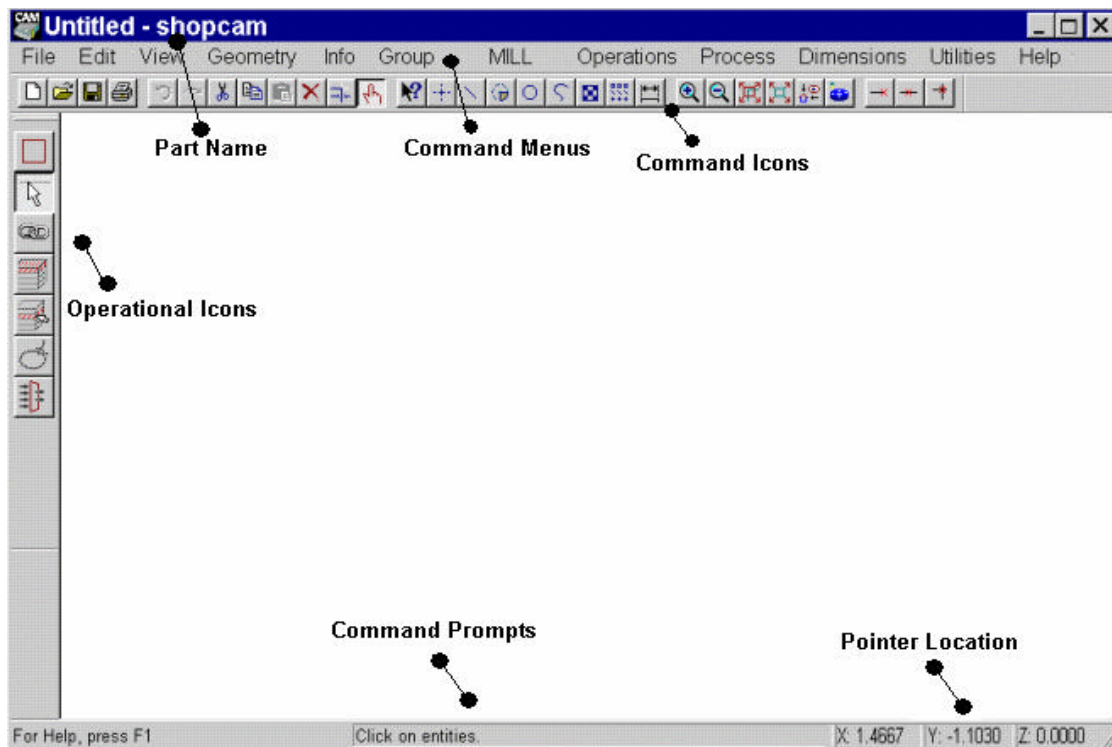
C:\PROGRAM FILES\DBS\SHOPCAM\FONTS\ .PRT

## Learning To Use The System

The best way to learn is to try different things and experiment. The on-line

HELP is always available in the system by pointing at the command in question and hitting the [F1] key or clicking the [?] icon and then the command in question.

## The Screen Area



**Command Menus** – The command menus are the heart of the SHOPCAM system. All functions are performed by selecting one of these items.

**Command Icons** – The command icons are Shortcuts located on the toolbar. The command icons perform identical functions found in the Command Menus.

**Operational Icons** – When a command icon is selected, this area displays the icons that show the individual choices within that command. For instance, the line icon will display all the commands for making lines.

**Command Prompts** – SHOPCAM displays messages on the Status Bar during each command. You should look at this area. If you are unsure what to do, refer

## Shopcam User Guide

to this prompt. If the display is empty, this is an indication that no function or command is active.

Pointer Location – The pointer location is the current X, Y, and Z position of your mouse.

### Layers

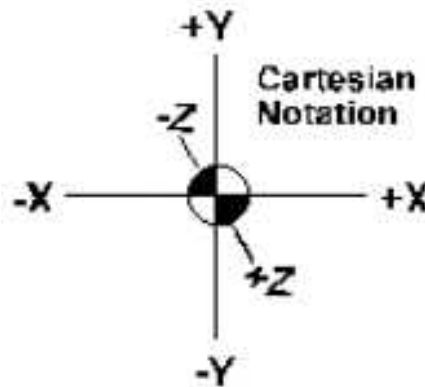
If you are familiar with using CAD, you should feel comfortable working with layers. Layers are a way to associate common geometry. For instance, when you group some geoms to create an Outline, that Outline geometry automatically goes on the next available layer. It will be labeled as Outline-X (where X is the next open layer number). The same rule holds true with Toolpaths. It doesn't change the base geoms into a group, it creates new geometry. When you want to delete a group, the system needs to know what layer that group is on. If you want to delete only a group or toolpath you should use 'Select Layer' or 'Last Layer' for your pick selection. If you use 'Window' to pick, you will also delete the base geometry underneath it.

NOTE: use the space bar to toggle between displaying the labels or hiding them.

### Coordinates

#### Cartesian Notation

The Cartesian coordinate system is a method of identifying any point in space. It uses three axis, called X, Y, and Z, to map a grid of cubes. The system identifies the three axis on the screen in the following manner:



X axis; The X axis is the horizontal axis. Positive X is to the right, negative to the left.

Y axis; The Y axis is the vertical axis. Positive Y is upward, negative is downward.

Z The Z axis is perpendicular to the screen. A positive Z is toward you, negative is away.

### Zero Degrees

The 3 o'clock position is always considered to be a zero degree angle. All angles are reference from 3:00 or 0 degrees. This means that a horizontal line is a zero degree or a 180-degree line, depending upon its direction.

## Two Dimensional Angles

A positive angle means a counter-clockwise rotation. For instance, a line going straight up on the screen is ninety degrees, but one going toward the lower right is a negative angle. All angles are normalized by the system (meaning a 270-degree angle is the same as a -90 degree angle. Angles are entered in decimal degrees. Enter the value as you would any other number to enter decimal degrees (to specify a 22.5-degree angle, enter: 22.5

## Units of measurement

Units of measurements refer to the intervals used to measure distances. Normally, coordinates are either in inches or millimeters, however centimeters are also supported. The part-program is not in any particular unit system. It consists of values that may represent inches or millimeters. In order for the system to generate a tapefile for the NC/CNC machine, it must know what units are to be used. The info table contains a selection for units. It is important that this is properly selected so that a tape can be generated correctly.

## EXPRESSIONS AND VARIABLES

Expressions are permitted in any numeric response. These expressions are evaluated immediately and the result is used in the answer. For complex problems, use the CALCULATOR.

### Basic Operations

In expressions, the following operations are permitted:

[+]	Addition	(e.g.: 2+2 is 4)
[-]	Subtraction	(e.g.: 5-3 is 2)
[*]	Multiplication	(e.g.: 3*2 is 6)
[/]	Division	(e.g.: 6/2 is 3)
[^]	Power	(e.g.: 2^3 is 8)

Notice that the asterisk is used for multiplication. The letter [X] is never used for multiplication on computers because that would cause confusion about a variable [X] and multiplication itself.

The slash [/] indicates division. Be careful not to use the backslash [\] by mistake.

## Five Steps of Part-Programming

Although it is not required to name your part until you save the file, it is good practice to name your file as soon as possible. You can give your part program any name but, it must follow the Windows standard for file naming (refer to Windows documentation for more information). Your file will be saved with the extension .PRT.

## Shopcam User Guide

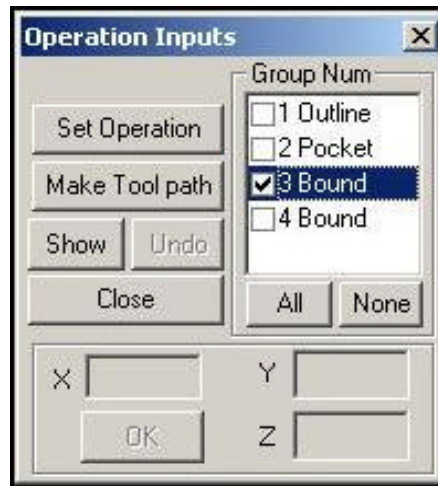
Making a part-program to run your machine is done in five steps:

Setup	Load a setup file.
Geometry	Either import a DXF or create geometry.
Groups	Group geometry to perform operations on.
Operations	Use the operations to make toolpaths
Processing	Translates the partfile into a tapefile G-code

These steps are listed below, with more detailed descriptions.

No matter how many part programs you create with SHOPCAM, you'll find that these five basic steps are repeated with each. As you gain experience with the system, you will find that these steps may be automated or revisited multiple times during a programming session.

Note: Some toolpaths generation does not require groups. Engraving, Threading, Facemill to name a few. These are covered in the Technical Reference Manual.



### Setup

The SETUP file presets the parameters for a given machine i.e. post processor, tool library, and various defaults. A part can be built without a SETUP file, but having a setup file loaded can make each session run faster. Most users will have a setup file saved for each unique machine. The system will load the last used setup file for each new part.

### Geometry (aka Geoms)

Geometry is the collection of lines and arcs that compose the part. Geometry can be created from within the system or imported from cad drawings via DXF files. See Section 6 (Creating and editing Geometry). This manual may call geometries "Geoms". There are basic geoms (the lines, points and arcs you create), group geoms and toolpath geoms.

### Groups

Groups are a collection of geometries that define pockets, contours, drill patterns, and any other machine-able features. Once the base geometry has been built, you must group together geometries to be machined. When you define a group, it is labeled and placed on the next open layer. The system then displays that layer number in the label. For instance, a group labeled [Outline-15] is a

group that will have the toolpath placed on the outside of the shape and the group is on layer 15. See Section ?? (Grouping tips)

### **Operations**

The operation is the heart of the CAM system. Machining operations act on Groups that have been built in the previous step. After selecting an operation type the system will allow setting the machining parameters. These parameters include selecting a tool, setting the step, depth, feed & speeds, CDC, coolant etc. When the operation is created, it is labeled and placed on the next open layer. The system then displays that layer number in the label. For instance, a finish path labeled [Toolpath-16] is a toolpath placed on layer 16. Some Shopcam operations do not need a group. If the system needs a group, the dialog shown above will appear. See Section ?? (How to perform an Operation)

### **Processing**

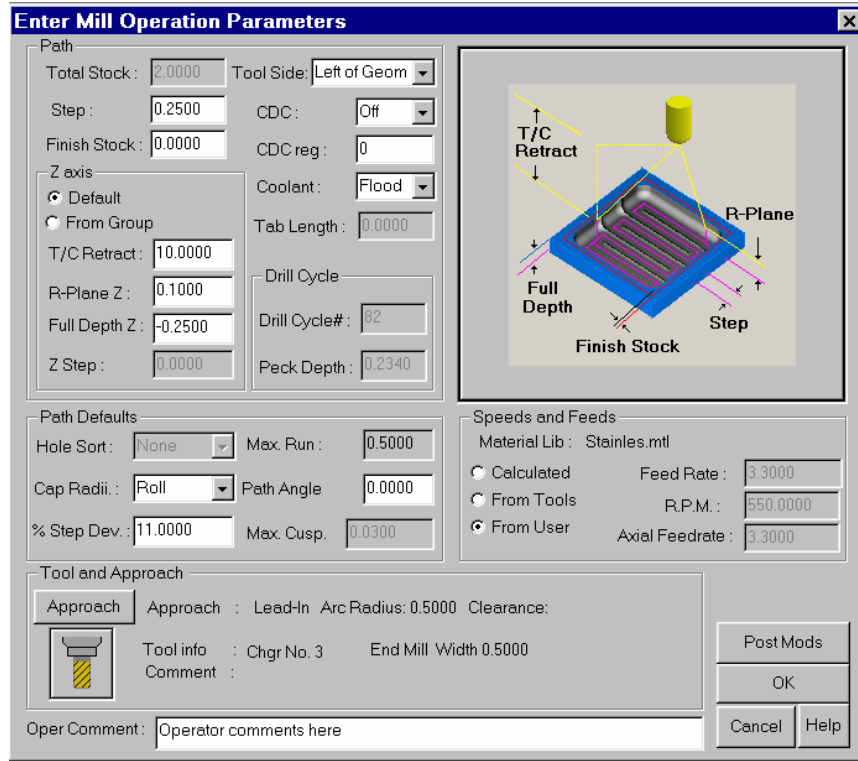
When the part-program has been completed and is correct on the screen, a tape can be made to run the CNC machine. This is called "post processing or posting" the part. Post-processing translates the operations into the numeric code that gets loaded into the CNC machine to make the part. Each machine has its own unique code; there are thousands of different kinds of NC/CNC machines! The resultant tape is saved in a file that has the same name as the part-program drawing. There are a couple ways to get the tap file to the machine. You can use a DNC program (like Dostek DNC) to send the file to the control. You may save the file to a floppy disk.

## **Section 2 Operation Dialog**

### **How to perform an Operation**

Since most people want to make a part right away, we put the operation section in the beginning. You will want to read the following Sections prior to making a part from scratch. Shopcam will either need groups or can generate a toolpath based on your parameters.

Once an operation is generated, you may choose to save all the information in the ‘Oper Library’ for use on another part. We will cover that in Section 11 ‘Operation Library’. For now, we will cover operations the regular way.



Regardless of the operation, you will have an ‘Operation Parameters’ dialog box to use. This dialog is presented when the [Set Operation] button is hit.

NOTE: Foam Cutting and some 2-axis users may not need to open the [Set Operation] dialog. If the parameters are the same for each shape, just define them in the setup file (First operation Default) button.

## Standard Operation Dialog

Here is the standard operation dialog used for mill and 2-axis.

### Operation Dialog

There are two ‘Operation Parameters’ dialog boxes, one for lathes and the standard one for all other modes. Only the information that effects the operation you are working on can be modified.

### Standard parameter summary

Here is a summary of the key parameters. Each parameter is described in the operation section of the Technical Manual.

Step	Used for the XY step in roughing cycles
------	---

Default or From Group	Determines how the Z-axis values are applied. Default will use the 'R-Plane Z' and 'Full Depth Z' from Z0.0. From Group will be the incremental distance from the group Z
R-Plane	The plane the Z axis rapids too. Usually .100 or .050
Tool Side	Which side to keep the cutter on.
CDC	Cutter diameter compensation; Usually causes a G41/G42 in the tape file
CDC Reg	CDC Register; Most posts use the tool number if set to 0.
Cap Radii	How the system treats sharp corners. Usually set to 'Roll'.
Drill Cycle #	For canned drilling cycles; cycle 1 is system generated
Path Angle	To change the path angle on Zig Zag rough.
%Step Dev.	Allow the step to deviate to equalize passes
Max Cusp	Adjusts the resolution of the steps of 3D operations
Calculated	If a material library is loaded, will figure RPM & feeds
From Tools	Loads the feeds and RPM from the tool library
From User	Allows you to set your own RPM & Feeds

### Approach Methods



Approach is how the tool gets on and off the part. A summary of the currently approach setting is displayed to the right of the button. All the approach methods are defined in the 'Approach methods' Section.

### Tool Button

The tool button will bring up the tool dialog. From there you can load



a tool from the library, use a tool that you already used in the current part or define a new tool. A summary of the current tool is displayed. The tool dialog is defined in detail in Section ??

### Oper Comment

You may add a description of the operation. This will show up in the “Operation Manager” dialog summary and in the report.

### Operation Image

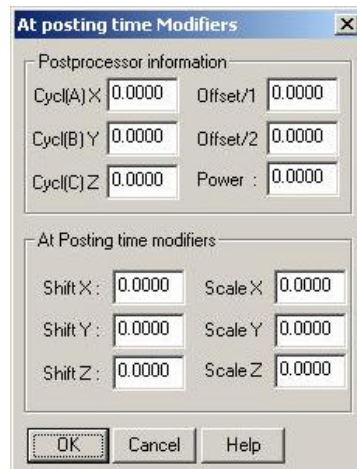
This is a image based on the operation you are using. It shows how the different parameters are used.

### Post mods (Modifications) and more



The post mod dialog contains parameters that may or may not be used by the post processor. If there are special instructions for a post, it will be in a txt file,

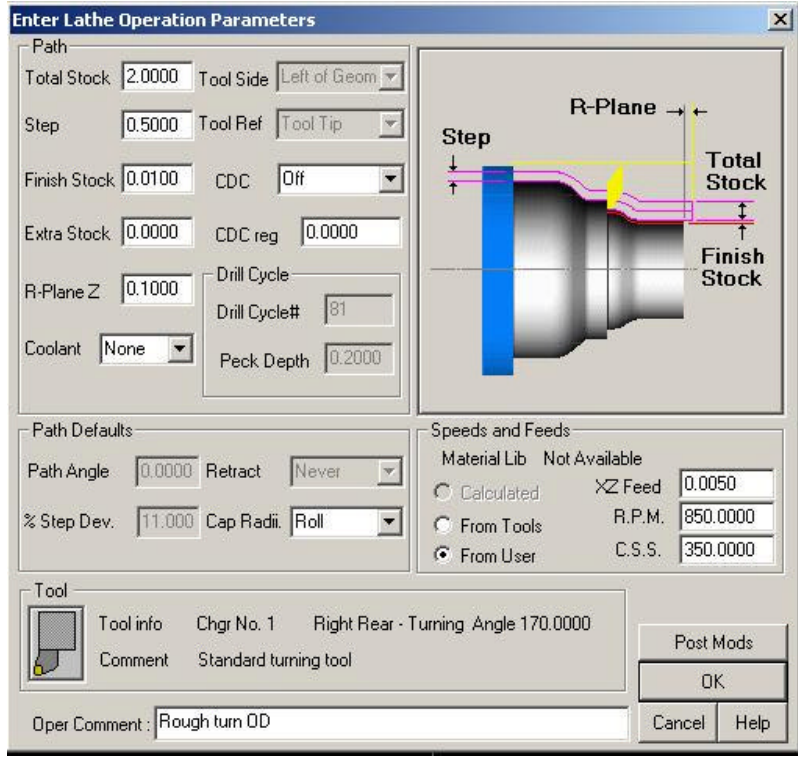
with the same filename as the post. For instance, as a general rule, if a post supports a 4th axis table, it will use Cycl(?) as the radius. Offset/1 and Offset/2 could be used on an old lathe or maybe something totally different. The ‘Power’ field is often used on Wire EDM and as a Fixture offset on mills.



The ‘At Posting time modifiers’ do just that, they change the G-Code output when you post process. These ‘At Posting time modifiers’ should be used with caution. It’s not obvious on the screen that the toolpath will be shifted or scaled. You are better off shifting with the ‘Move’ command.

## Standard Lathe parameter summary

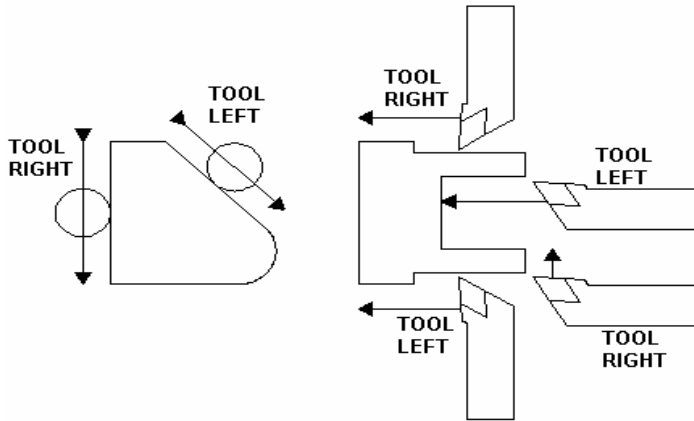
Here is a summary of the key turning parameters. Each parameter is described in the operation section of the Technical Manual. Each operation will gray out the boxes it does not need. A generic picture will show how the most important parameters will be used.



Step	Used for the XZ step in profile cycles
Extra Stock	Additional stock to leave on straight OD cuts
R-Plane	Where the Z axis positions for a pass; Absolute value.
Tool Side	Side to keep the cutter on. Usually right for OD left for ID
CDC	Not usually used on a lathe
Path Angle	The 'rough Turn' path angle usually=0 or 90 for facing.
%Step Dev.	Allow the step to deviate to equalize passes
Calculated	If a material library is loaded, will figure RPM & feed
From Tools	Loads the feed and RPM from the tool library
From User	Allows you to set your own RPM & Feed

## Determining The Tool Side

Determining the tool side is very easy. Imagine walking along the geometry you wish to cut. Is the tool to the right or left of that geometry? In the previous FINISH example, the Tool Side (Left of Geom) performed a Climb Cut. Though the OUTLINE was defined in the opposite direction, the computer knew on which side and in which direction to cut from the Group Type and the Tool Side.



Cutter Compensation is also a factor. It can be performed either by computer or machine tool. In determining the preferred method, consider the following:

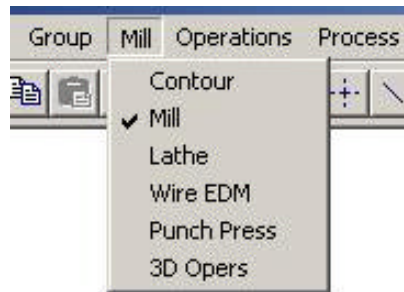
Allow the computer to compensate for all roughing cycles. Specify the tool side, tool width (and corner radius if any) and set CDC to OFF, which disables the machine compensation.

Allow the computer to compensate for most finishing cycles (Mill and Wire). Enable CDC on the machine and set the machine compensation to correct for variations due to wear and cutting conditions. You do not want to double compensate by having Shopcam offset the cutter and the machine do the same.

## Section 3 Setup File, Tool & Material libraries

### The Setup File

A Setup file allows you to set and save preferences. Normally you would have a setup file defined for each unique machine tool. Since the machine (Post-processor) is the most important part, You may want to 'save as' a filename that incorporates the machine and control. Change it from default.set so it doesn't get stepped on incase you ever reinstall.



## Shopcam User manual

Use the [Browse] buttons to change the file or the [Clear] button if you don't want a library file.

Prior to saving a setup file, set the machine mode on the 'Command Menu' at the top of the screen. The mode is located between the 'Group' and 'Operation' menus.

Contour mode is the same as 2-axis mode. If you use a Foam cutter, Waterjet, Plasma, Cutting torch or any two axis, select Contour

### Setup File parameter summary

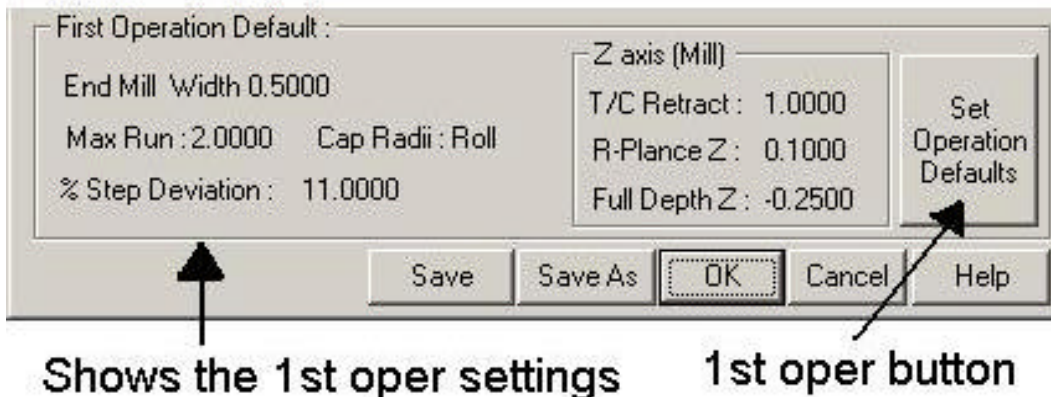
How you set the system defaults will depend on what you are writing programs for. Below is a list of the key parameters and suggested settings.

Setup File	The setup file being used.
Post Processor:	The post-processor to be used.
Material Library:	The material library to be loaded (optional)
Tool Library:	The tool library to be loaded (optional)
Inch or Metric:	This is for the post processor output. Most, but not all, posts support metric output. If you are in metric mode and the output is about 25 times too small, metric isn't supported. Contact your dealer to have metric output added. If you normally work in inch and receive a metric CAD file, use the [Scale] command to make your geometry inches.
Radius value or Diameter value	In Lathe mode, it determines whether the X axis values you enter are diameters or radial.
Decimal Display	How many places to the right of the decimal do you want to display on the screen. Most people set this to four. A WEDM user may prefer 5 while a router user may only need 2, This has no effect on accuracy.
Forgiveness	Normally this is set to the minimum move of your machine or .001 for a mill or lathe. It will also help with chaining. This has no effect on accuracy

<p>Toolchange X Toolchange Y and Auto 1st Toolchange</p>	<p>This serves two purposes. It is used to ensure compatibility with older posts and it makes sure the 1st move squares properly on a 3 axis machine. These values should be set to coordinates off the table. Check the 'Auto 1st Toolchange' and program a simple part. If the coordinates on the first couple moves are correct, leave it checked. If these coordinates are output at every Toolchange, uncheck it.</p>
--	--

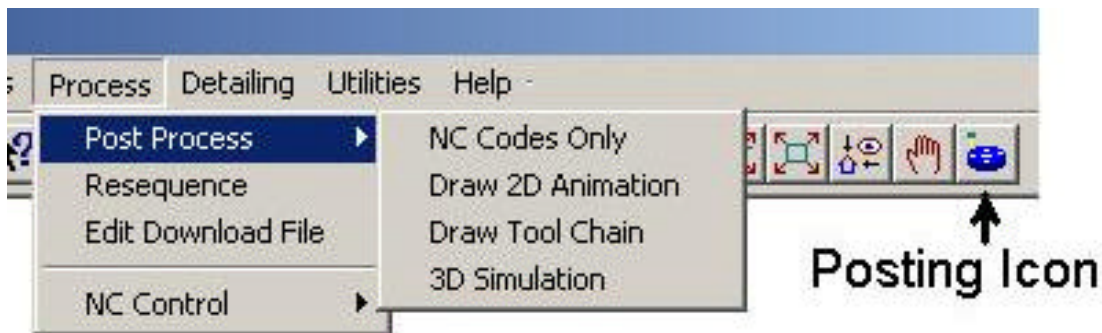
### First Operation Default

With each setup file you can and should set the default operation parameters. This is especially important if you don't use a tool library. 2-Axis users (foam cutters, water jet, and burning tables) should set these parameters as you do on all shapes. That is usually tool ID number and changer set to '1' and the tool width set to '0'. Also, set the tool to round and set a federate to something other than '0'.



## Section 4 Making the Machine Code file

After the toolpaths are defined, you are ready to translate the part program into



a file your machine can use. This is called 'post processing' or 'posting'. There are two ways to post process, with the menus or the icons.

With post processing you may chose a couple different ways to visualize the posting.

NC Codes only: Just makes the machine code file.

Draw 2D Animation: Fills the screen with different color then blackens where the tool visits.

Draw Tool Chain: Snakes an outline trail, of the tool, as it travels around the part.

3D Simulation: Runs the optional 3D modeling program. Mill & Turning only

### Post Processors

Every machine requires its own unique codes to cut a part. There is a standard 'RS-244', but for some reason, many Control Manufactures rely on their own modified version. Plus some Machinist just prefer to see the machine code a certain way. The program that writes the machine code file is called a 'post' or 'post processor' and has a .mch extension. The machine code file it creates is called a tapefile with a .tap extension. The system will save the part prior to posting. If an error message or warning appears during posting, it is usually generated by the post program. Refer to the postprocessor's instructions for an explanation of the message. The instructions will have the same name as the post with a .txt extension.

## Section 5 Picking geometry

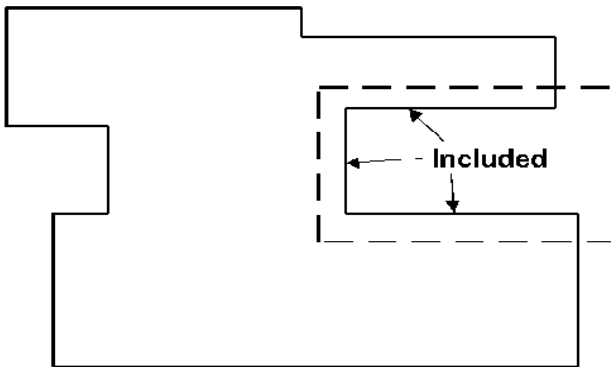
### Pick Modes

The pick methods are used to select single or multiple geometry items for any command that needs geometry. Almost everything you do, will need to pick something first.

You need to pick geometry to copy, delete, group, edit, trim, rotate, stretch and create geoms based on other geoms.

### Picking with a Window

To use a window to pick items, only the items that are ENTIRELY INSIDE of the window will be selected.



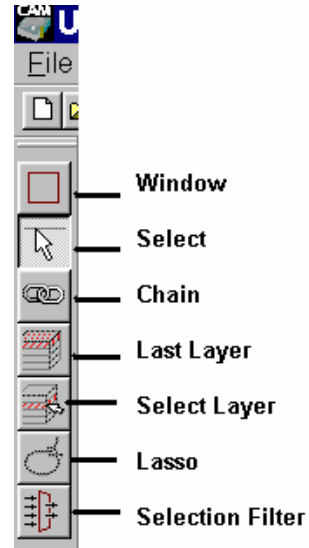
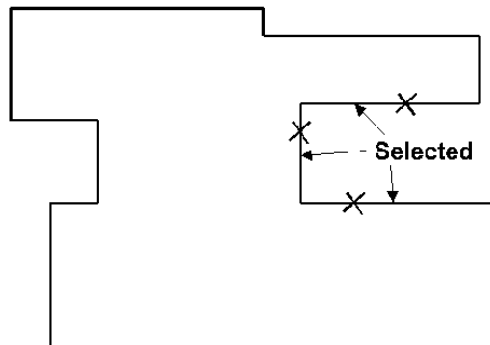
This means both ends of a line must be enclosed for the line to be picked. To use a window to select items, first indicate one corner. Any of the four corners of the window can be selected. When asked to indicate the other corner, it must

be the diagonally opposed corner. Here, the cursor changes to a WINDOW to indicate the area enclosed.

Exception: When using a window to STRETCH items (via the EDIT MENU) Lines lying entirely inside of the window will be moved.

### Picking By Selecting

When picking items with the SELECT method, continue picking single geometry items. When you have picked all you need, click on the [done] button in the dialog box.





### Pick lasso

The lasso is similar to window except you will create an irregular pick area by digitizing points around the selected items. Use this when you have to ‘snake’ around geometry you don’t want to include in the picked items. The start and end points must overlap to close the lasso. Once the lasso is complete, select the icon again to execute the pick. All other Pick Windows terms apply.

### Pick last layer

Pick last layer will select the last layer used that contains information. By clicking on the icon again, it will select the next to the last layer and so on. This makes it easy to delete completed layers.

### Select layer

Select layer icon will ask for the layer number to select.

### Query Geom information

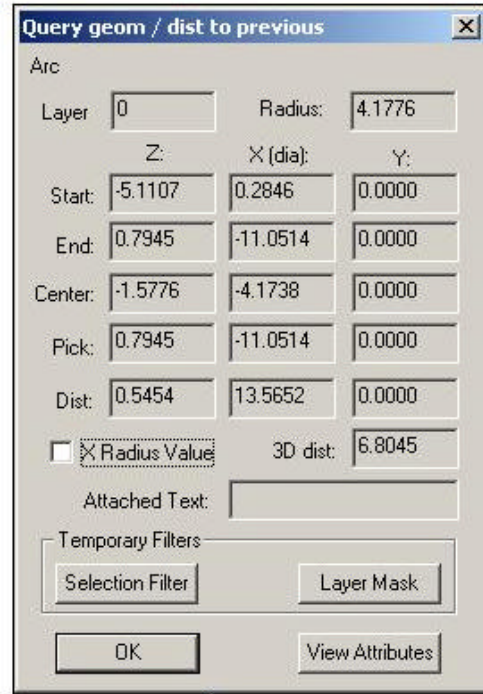
The Query command is used to gather information about geometry. The Query command can be selected from the info menu or with the [Q] key. After selecting a geometry item, the Query dialog box will appear showing the geometry data.

Two temporary filter buttons are located on the Query dialog box.

1. The [Selection Filter] for masking certain types of geometry
2. The [Layer Mask] for selecting specific layer.

The coordinates for the selected geometry will appear in the dialog box. If a line is selected, the angle will be displayed with the start and end coordinates. With an arc or circle, the radius, start and end are displayed.

To view attributes of the selected geom, click on the [View Attribute] button. The ‘Attributes for display only’ dialog box will appear. Refer Section 11 of the Technical Reference manual for Attribute Table information.



In lathe mode a check box for displaying the X axis as a radius or a diameter.

## Section 6 Geometry Creation

There are two ways to get geometry into the system. You can import/merge a CAD file from another package, or create geometry in Shopcam

### CAD Files

To merge a cad file, go to the [Files] menu then [Import]. The system recognizes two different CAD formats, DXF and IGES. We use the AutoCAD standard DXF format as a baseline. A couple notes about CAD files:

Avoid CAD files that consist of surfacing or a wire frame. Use basic geometry. If the DXF won't read, use the [Explode] feature of your CAD software.

### Shopcam Geometry

There are three ways to create geometry in the system.

Basic [Geometry] commands to create a point, line, arc or circle.  
 [Edit] commands (copying or mirroring) to create more geoms .  
 [Patterns] commands to create rectangles, ellipsis, gear, hole grids etc.

There are three ways to access the basic geometry commands.  
 With the command icons  
 Using the [Geometry] menu on the main menu.  
 With the shortcut keys listed below.

### Quick Keys Chart

key	Description	key	Description
A	Arc through 3 positions	S	Pan or Slide the display
B	Break two geoms at the intersection	U	Undo Last Command
C	Create a Circle	V	View All the geoms
F	Fillet on two geoms	W	View Window
I	Invert or reverse an Arc	X	Trim Both
J	View previous (jump back)	Z	Set Z Depth
L	Create a Line	F1	Help

## Shopcam User Guide

O	Set a temporary Origin	F2	Select an End Point
P	Create a Point	F3	Select a Mid or Center Point
Q	Query a geom for information	F4	Select an Intersect Point
R	Redraw or Refresh the screen	F7	Rotates Sprite CCW 5°
ESC	Abort/Cancel	F8	Rotates Sprite CW 5°
↑	Speeds Posting Graphic	←	Rotates Sprite CCW 1°
↓	Slows Posting Graphic	→	Rotates Sprite CW 1°

With the four basic geometry types (Point, Line, Arc and Circle), the ‘Operation Icons’ will appear on the left of the screen allowing you to select the command you need.

For a complete list of the basic geometry commands, see the ‘Technical Reference Manual’

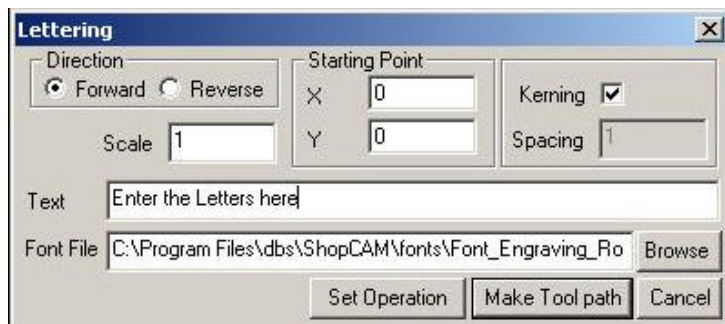
### Patterns

Shopcam has several patterns available. A pattern may be as simple as a square or rectangle or a complex pattern like a gear or circular cam. When a pattern is created, it will be a series of individual geometries.

For a complete list of pattern commands, see the ‘Technical Reference Manual’

### Lettering

This command creates the simple geometry that representing the alphabet. Do not confuse this with engraving, which will actually generate a tool path. Use lettering for raised or recessed shapes.



You need to group the shapes then perform a operation on the groups. Only select a font file that starts with 'Font\_Outline'

Fill out all fields indicated and click on the OK button to execute the command. All fields will be filled with system defaults except for the TEXT. Hit the [Make Tool path] button to create the geometry.

Some fonts may not include all of the upper and lower case letters of symbols found on a standard keyboard. If the item is not available in the font then it will not appear when the command is executed. The part files contain various fonts

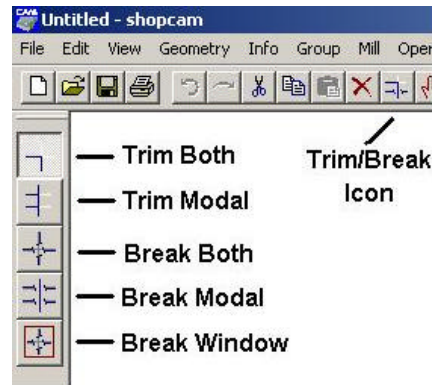
## Section 7 Geometry Editing

Geometry editing may be used to create more geometry via copy or mirror. You may also edit geom by trimming it or breaking it.

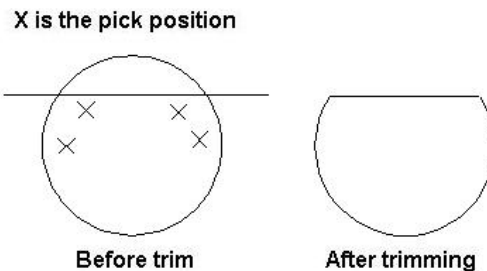
### Trimming and Breaking

After creating geometry, you may have to trim it to a different geom to form a sharp corner. You may have to break it to select a start point for a group. The most common trim command is [Trim Both]. The quick key for 'trim both' is the 'X' key

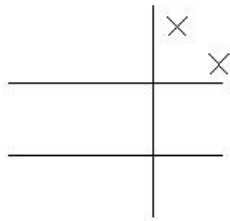
When trimming, the system will look at where you digitize to determine what you want to keep and what gets trimmed off. You will select the geometry to keep.



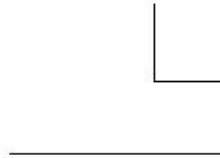
When breaking two geometries, it is not that critical where you pick because the all the geometry will remain. When you break two geometries, nothing appears to happen, but you will have four geometries instead of two.



X is the pick position



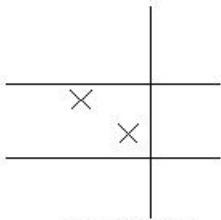
Before trim



After trimming

This circle example is often confusing. The circle doesn't appear to trim on the first trim.

Notice in the two examples on the left. Notice which part of the geometry gets trimmed off and what geometry remains.



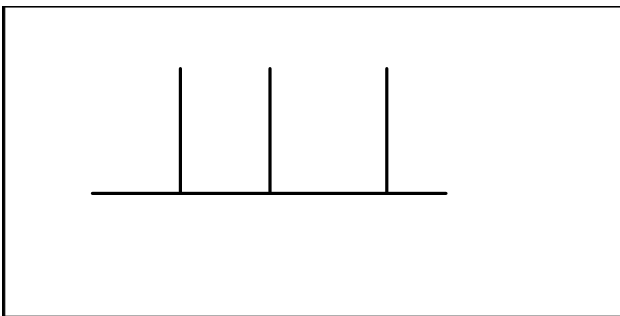
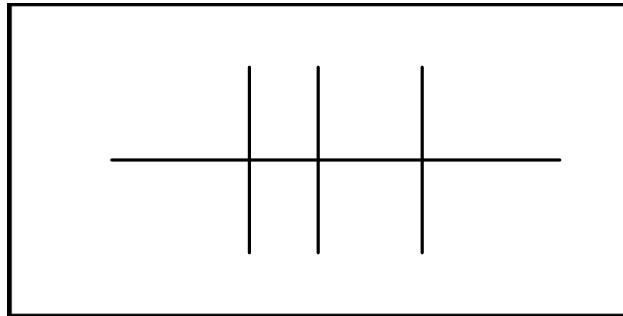
Before trim



After trimming

### Trim Modal

This instruction is used to trim off geometry items where they intersect another item. The system will ask for the trimming item first, then the items to be trimmed off. Unlike the TRIM BOTH instruction, this one requires that you pick the geometry items along the portion to be trimmed off, not the portion to be retained.



In this example, the vertical lines must be trimmed at the horizontal line as shown. Using the TRIM MODAL command, the horizontal line is selected as the trimming item. The vertical lines are selected as the items to be trimmed. They must be picked along their portion that lies below

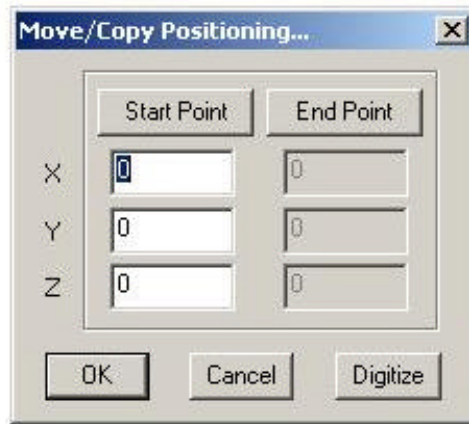
the horizontal line, as that is the portion to be discarded.

## Breaking

The most common use for breaking a geom is to specify a starting point for a group. Take a simple rectangle for instance. If you group a rectangle without breaking one of the four lines, the group will start on one of the corners. This may or may not be what you want. If you want to sweep onto the shape with a arc, the start and end geometries must form a 180 degree included angle. The easiest way to accomplish this is by breaking a line or circle. Another common use for breaking a geom is to specify a glue stop for a Wire EDM.

## Copying and Rotating

With the copy command, you must select the items to copy then specify the start point and the end point. You may think of the start point as the 'Reference Point or Anchor point' and the end point as the



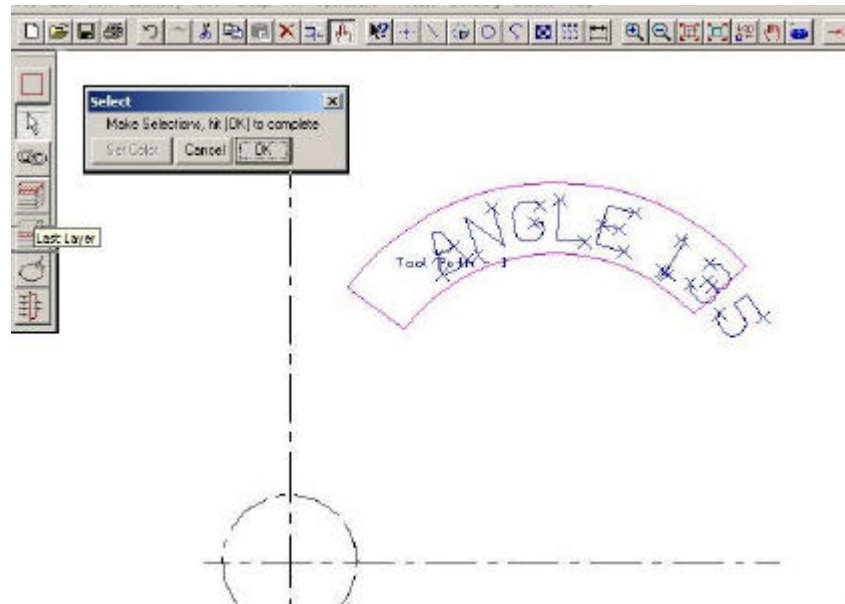
**COPY DIALOG**



**SNAP TO  
CENTER POINT**

'Destination Point'. If the start and end points are the same, the system assumes you want to rotate and will display a dialog box to enter degrees.

In this example, we have to rotate the engraved toolpath 15 degs counterclockwise. After invoking the copy command, we are present with the pick mode dialog. Since we know Toolpath-1 is the last layer, we can select 'Last Layer'. The Toolpath will turn



blue and we can then hit the [OK] button. At this point, a dialog box will appear with the start and end fields. In this example we simply want to rotate the toolpath around the center of the shape. Rather than entering the center point, we can snap to the center of the arc. After the [OK] is hit, the dialog for 'degrees' will appear.

If [Digitize] is selected, the picked geometry will turn into a sprite that can be moved with the mouse. The F7 and F8 keys will rotate the sprite five degrees, CW and CCW respectively. The right and left arrow will rotate it one degree.

## Section 8 Making Groups

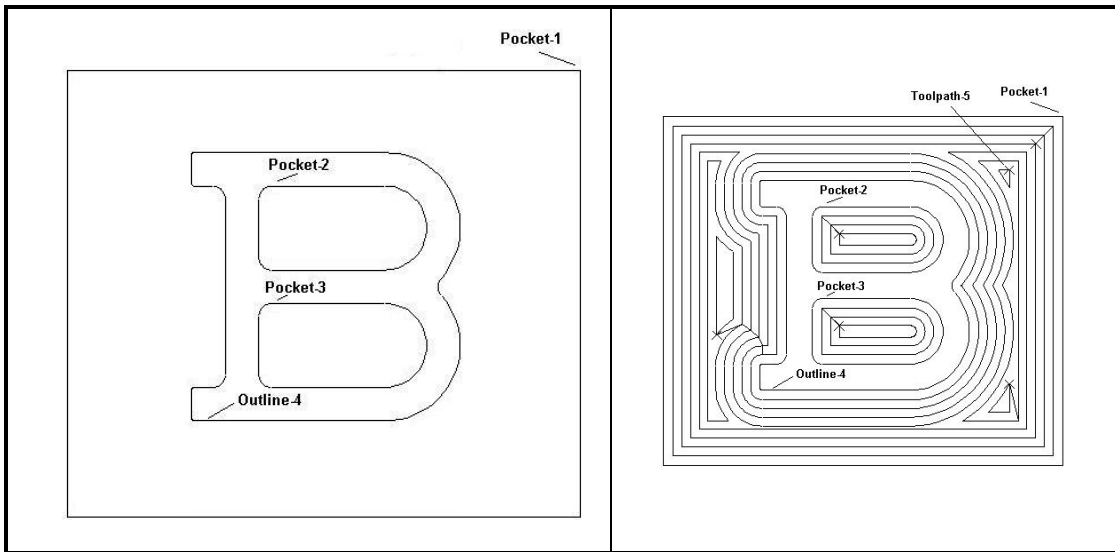
We have touched on groups earlier in this manual. Since groups are critical to generating operations and a bad group will cause an error message when trying to generate an operation, we will spend some time on them. The type of group is important when the system performs an operation on a group. If a group is a closed contour that defines the outside of a part, it should be called an OUTLINE. If a closed contour defines an internal form, it should be called a POCKET. If it is an open shape, it should be a BOUNDARY or FACE. Collections of circles and/or points that are to be drilled are called HOLES.

The group type determines on which side of the geometry the tool will cut. To profile rough the outside of a part, group it as an OUTLINE. To rough out the inside of a part, group it as a POCKET. With a BOUNDARY, the tool can cut on either side of the contour depending upon the value in the tool SIDE. With Pockets and Outlines, the group must start and end at the same place. It needs to be a closed shape.

Some operations can only be performed with certain groups:

MODE	OPERATION	GROUP
MILL	ZIG ZAG HOG	POCKET
MILL	PLUNGE ROUGH	POCKET
MILL	DRILL CYCLES	HOLES
LATHE	ROUGH TURN	BOUNDARY
LATHE	PROFILE	BOUNDARY
LATHE	GROOVE	BOUNDARY
LATHE	FINISH	BOUNDARY
WIRE EDM	MULTIPLE PASS	POCKET or OUTLINE
3D OPERS	MULT Z RUFF	POCKET
3D OPER	SWARF	BOUNDARY

In the examples below, we want to machine a raised letter 'B'. The 1st group we created is Pocket-1 around the box. Then we do the top inside of the 'B' (Pocket-2) and the bottom inside (Pocket-3). The outside of 'B' is grouped as an outline and is automatically placed on layer 4 (Outline-4).



When selecting one of the group types, the pick icons will appear on the left. Any of the pick modes may be used; it is good practice to always use chain to create a group. If you use chain to create a group, it guarantees the group to be mathematically correct prior to generating an operation with it. A group has to be correct for the path processor algorithms to calculate the passes and offsets. If a chain stops for no apparent reason, you shouldn't just click to get the chain going again.

## Shopcam User Guide

When a chain stops, and displays a bull's-eye, there is always a reason. There is something in the geometry causing it. This is especially true if you receive CAD files from outside sources.

There are several reasons for a chain to stop for no obvious reason:

Gap; The endpoint and start point of two geoms are not connected

Overlap; There are two geoms on top of each other

Mismatched layer; Part of the geometry is on a different layer

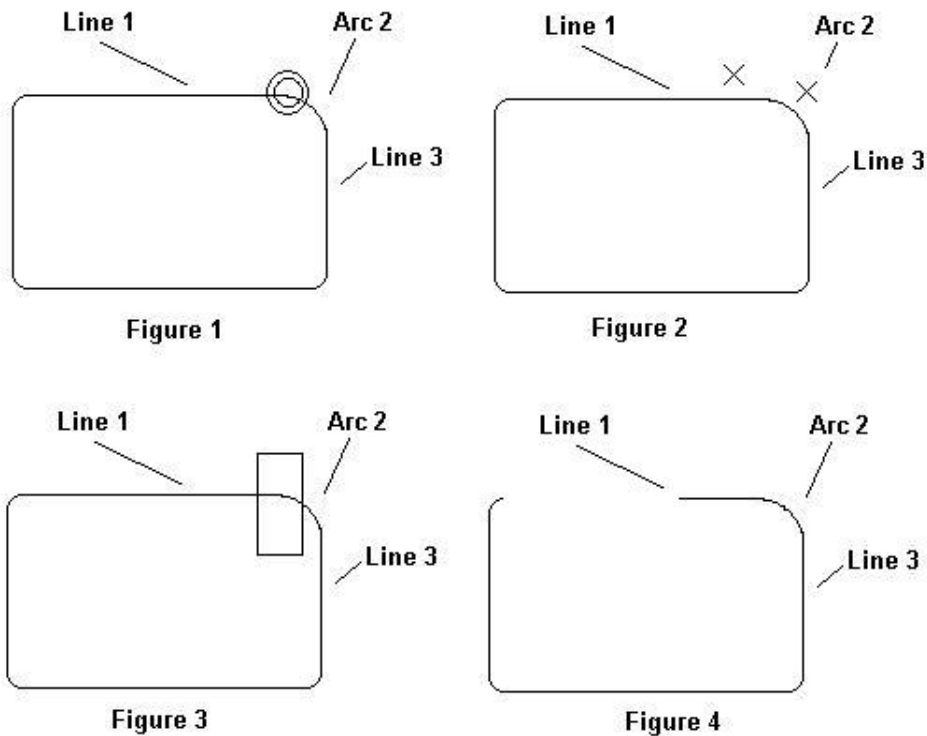
Miniscule geom; There is a geom that is very small

Z only line; A line has the same XY start/end but different Z depth

Since each one of the cases is not always obvious to the eye, there is a simple procedure to go through to clean up the geometry. If you are curious as to why it stopped, try to chain after each fix. If you don't care why it stopped, assume it could be any one and just do all the steps.

### Cleaning up geometry

In this example the chain stopped on the corner of this shape. (See Figure 1)



Let's eliminate each reason for the stoppage. First hit the [OK] button, then the

'U' key to undo the group. Over the years, we have found this procedure to work the best.

Gap; these tend to be rare and with good graphics are usually obvious. Since trimming is so easy, let's eliminate it first. Hit the 'X' key (quick key for trim both) and pick line 1 and arc 2 (see figure 2)

Miniscule geom and Z only line; The best thing to do with this is just delete with a window around the area it stopped. Hit the [Delete] key and select the [Box] pick method on the left icons. Make sure you don't include all of the arc (see figure 3). Since these two reasons cause about 70% of the problems, I would try a chain again. If it still stops at the same point, continue to C.

Overlapped Geoms; These are hard to catch and rare. The best way is to just delete the geom it stopped at and see if another geom or part of a geom remains. (Figure 4) If a line remains, try grouping again. If a partial geoms remains, trim the line to the next line and then group.

Mismatched Layers; If none of the above methods worked, it has to be this. Hit the 'Q' key and select the geom the chain stopped at. Note the layer and select the adjacent geom. When the oddball geom is determined, use the [Info] [Edit Geom Attributes] command, select the geom with the different layer, and change the layer number to the same as the rest.

This may seem confusing, but if you use CAD files from different customers, you will encounter each one of these conditions eventually.

## Section 9 Approach methods

The approach is how you get on and off the part. If starting on an arc, you can not apply CDC without a line move. It is a good idea to use an approach on finish operations where you would use a CDC offset. The approach and pull-off is not used by lathe, drill cycle, or 3D operations.

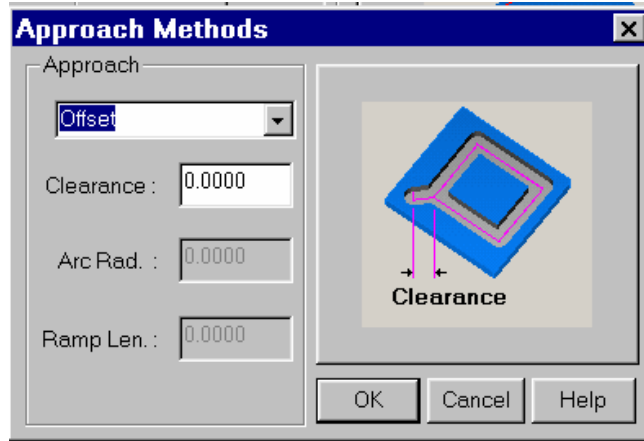
### APPROACH

#### NONE

None approach indicates that the tool should move directly into tangency with the finish work piece before beginning its machining path.

## OFFSET

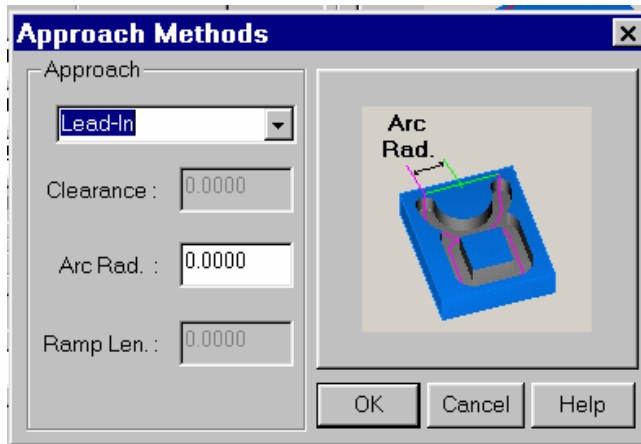
Offset approach method to cause the tool to move to a position that is the CLEARANCE distance away from the finish work piece, and then feed perpendicular to the 1<sup>st</sup> move. Upon completion, the tool will move away from the finish work piece until it is the CLEARANCE distance away before retracting. On an open group (boundary) the offset will extend the 1<sup>st</sup> move by the clearance amount.



## LEAD-IN

The Lead-In method causes one of three methods to be used, depending upon the location of the start point on the group to be machined:

If the start point is along a tangent move, then a ninety-degree arc will be constructed tangent to the start point, and used to "wipe onto" the work piece. Another ninety-degree arc will be added to the end of the tool path to "wipe off". These arcs will have a radius of the 'Arc Rad'.

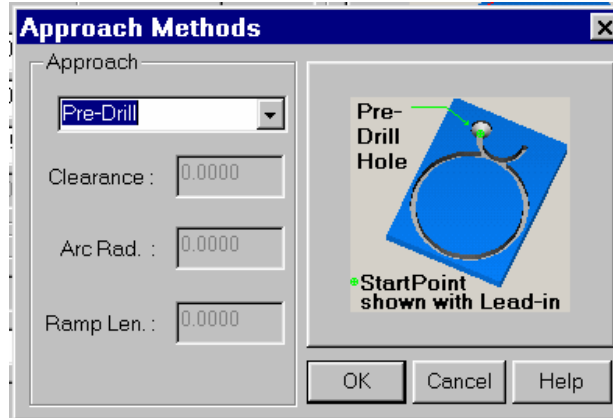


If the start point is at an inside corner, then the same approach is used as OFFSET. If methods two or three are expected but generate gouging (tool interference) then this method will be invoked instead.

If the start point is at an outside corner and CAP RADII are enabled then the tool path will begin off the part by extending the start point of the first geometry item back by the TAIL LENGTH. The tool path will also be extended past its end point by lengthening the last geometry item by the TAIL LENGTH. This allows the tool to "wipe on" and "wipe off" the work piece.

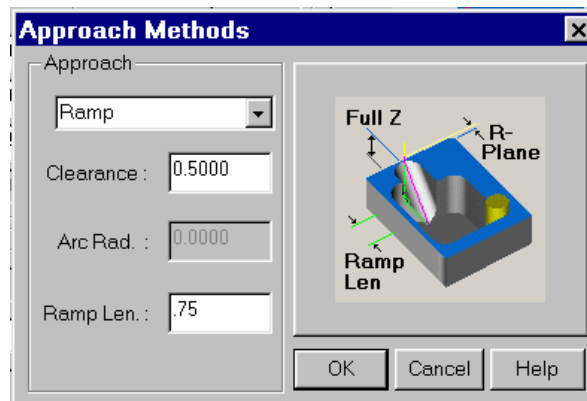
## PREDRILL

A group of circles will be constructed on the next open layer, and labeled "HOLES". A hole/circle will be generated at each plunge point where the approach is started. This is useful for selecting with a subsequent DRILL operation to pre-drill the plunge points. **If this is done, the DRILL operation needs to be resequenced to precede the machining operation.**



## RAMP

Ramp is commonly used for pocket roughing, when using PROFILE ROUGH, ZIGZAG HOG, or MULT-Z ROUGH. The system will attempt to create a three-axis ramp cut into the work from the R-PLANE Z. The 'XY length' of this plunge is specified as RAMP LENGTH. The system may need to shorten the RAMP LENGTH in order to fit into the removable material boundary.



## START POSITION

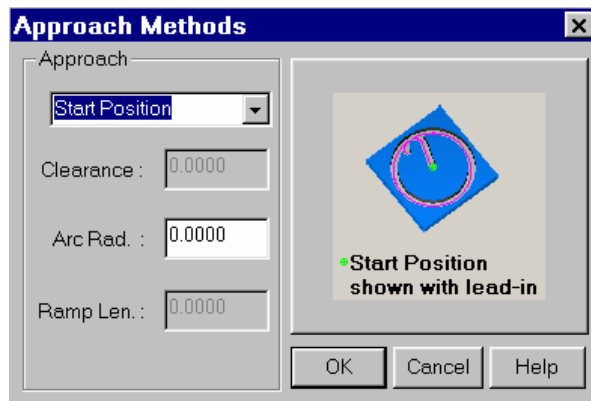
Start Position is used when a start hole is already drilled.

The system prompts for the start positions. Digitize or enter the coordinates of all of the permissible locations. The Z values will be ignored.

Whenever the system needs to plunge the tool into the work, it will select the nearest start point as the plunge position.

However, the system does

NOT validate the move from that plunge point to the pickup position on the contour. Thus, if not entered a start position that can be clear of the work near the start of a contour, the system may need to make a long cut from the plunge point to get to the contour. This move may interfere with the finish contour, so be sure to visually check for such conditions before post-processing.



The parameters listed below are used with the approach commands.

### **Clearance**

Clearance is used as a safe startup distance from the finish work piece. When approaching the work or retracting the tool, the system will attempt to move the tool to a position that is this distance away from the contour. This has to be entered with an OFFSET approach. This is usually at least half the tool width.

### **Arc Radius**

If LEAD-IN is chosen, this is used to specify the radius of the arcs.

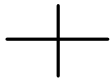
### **Ramp Length**

When the RAMP approach method is used, the system will make the three-axis moves a length (in the XY plane) of RAMP LENGTH. If this cannot be accomplished, the system will make the moves as long as it can without gouging the work piece.

## Miscellaneous Commands

### GRAPHIC CURSOR TYPES

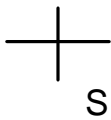
#### Crosshair Cursors



The plus shape indicates an XY or Z must be entered or digitized on the screen.



The **X** shape indicates that a “pick” of the geometry can be chosen.



The **+s** shape indicates that the filter mask is active

#### Round Cursor [F2]



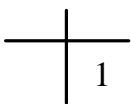
The round cursor indicates an ENDPOINT pick mode is enabled.

#### Box Cursor [F3]



The box cursor indicates a CENTER-POINT pick mode is enabled.

#### Intersection CURSOR [F4]



The intersection cursor indicates sequential selections of two geoms whose intersection will be digitized.

## Sprite Cursor

The sprite cursor is the actual shape of all the geometry that is digitize when using the move, copy, rotate, mirror, merge or scale commands

## Resequencing

The following operations have been performed:

PROFILE ROUGH with tool

- 1.
2. FINISH with tool 2.
3. DRILL with tool 3.
4. ENGRAVE with tool 4.

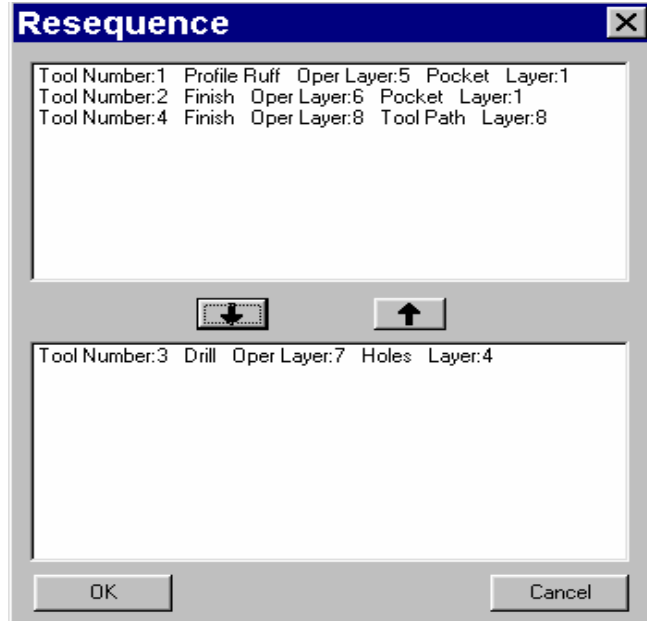
However, do not perform the operations in that order. The fourth item should be the first operation. The resequence command will allow the resequence the operations.

1. Select RESEQUENCE from the PROCESS Command Menu.
2. Select OPERATIONS by picking each in the following order:

- A. Drill Operation Layer 7.
- B. Profile Ruff Operation Layer 5.
- C. Finish Operation Layer 6.
- D. Finish Operation Layer 8.

3. Click on OK when finished.

If listed operations are not selected, then only those that were selected will be post processed. The others will be placed at the end of the table, but will be disabled until another RESEQUENCE command is performed.



## Section 10 The N/C Editor

A 3<sup>rd</sup> party editor is provided with Shopcam. It is the same editor supplied with the Dostek DNC package we recommend. For help on this editor, select the help menu from the editor.

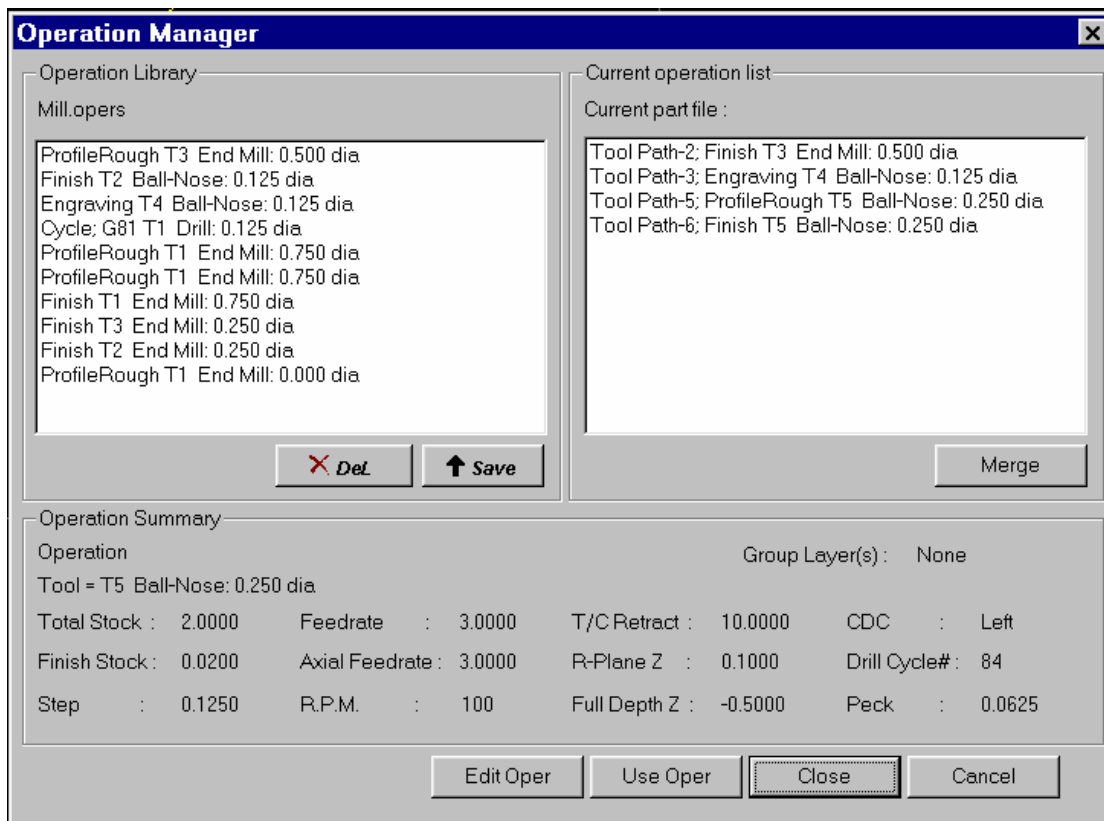
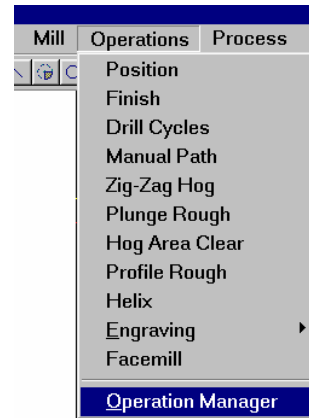
Almost any editor may be used. If you have a text editor you would prefer, you can change it. Select [Info] then [DNC Command]. Use the [Browse] and select your editor and hit [OK].

## Section 11 The Operation Manager

The Operation Manager is a dialog box with a list of all the operations in the current part and a list of operations from a library. This feature will allow you to save and recall all the information required to generate or regenerate a Toolpath.

There are two operation libraries. “Lathe.operators” and “Mill.operators”. These are located in the default install directory (C:\Program Files\DBS\Shopcam).

Lathe.operators is used for Turning/Lathe and Mill.operators is used for everything else.



The first time you save an operation, this file is created. This dialog is used for three purposes. Saving, retrieving or editing an operation. The Operation Manager selection is located at the bottom of the Operation menu

### Operation Manager Layout

The operation manager dialog consist of three main areas:

**Operation Library:** This is a list of the operations stored in the operation library file. Control buttons are located at the bottom of this list. This is the list on the left

**Current operation list:** This is a list of the operations in the current part. This is on the right.

**Operation Summary:** This is summary of the most important parameters of the selected operation. This is at the bottom.

### Saving an Operation

With the operation dialog open, one operation is always active. The operation summary area will tell you which operation is currently active. You may click on an operation from the library list or from the current part file list and make it active. Once active, you may edit it and save it to the library.



THIS BUTTON UPDATES  
THE OPER LIBRARY

Use the up arrow [Save] key to move the active operation into the library file. If the active operation was selected from the current part list, it will be appended to the end of the library list and the file saved. If the active operation was selected from the library list, you will be asked to replace or add to the end of the library list. The file is saved automatically.

Building your own custom library consist of either editing an exist operation from the library and adding to the end of the list or selecting a operation from a part and adding to the library.

### Using an operation from the library

To generate a operation in Shopcam, you can do it the traditional way by selecting a operation from the menu or from the operation manager. Select 'Operation Manager' from the bottom of the operation menu. When the operation dialog opens, select a operation from either the operation library list or from the current part file list. Once selected click on the 'use oper' button. If the operation requires a group the 'Operation Inputs' dialog will appear and allow you to select some groups and hit 'Make Toolpath' to create the toolpath. Operation that don't require groups (Engraving, facemill, threading etc), will go directly into their dialog box.

### Editing an operation

Editing a operation is pretty straight forward. After opening the operation manager, select an operation from either list, make the changes and save it. Part

programs generated with the new version of Shopcam will contain all the information needed to regenerate a toolpath or save to the library.

If you are editing a operation from the current part list, and want the toolpath to reflect the changes, you will need to hit the [Make Toolpath] button. If the operation requires a group the 'Operation Inputs' dialog will appear and allow you to select some groups and hit 'Make Toolpath' to create the toolpath. The layers that originally generated the toolpath will be checked. Operation that don't require groups (Engraving, facemill, threading etc), will go directly into their dialog box. You can see that it is important not to delete group layers if you might want to make changes to an existing toolpath. If you are editing a operation to save in the library, make sure to use the 'up arrow' to save your changes to the library.

## Section 12 Tool Library

Using the new tool library dialog is much like using the operation dialog. There are two list, the current library and the tools used in the part program. The new tool library dialog has been expanded to included a picture, default speeds and feeds, and a spot for tool comments.

### Saving and Deleting a Tool

Use the up arrow to save the active tool into the library. You may select a tool from the current tool list. This is a list of the tools in the opened part You may also select an existing tool from the tool library list. You may edit these values prior to saving them. Use the [Save as] button to save the existing tool library under a different name.

To delete a tool from the library, highlight the tool and hit the delete button. The ID number or 'Tool ID' is simply a numbered list of the tools in the library. Older revision called this the 'Tool No'. Since the 'Tool Changer No' is the actual number sent to the postprocessor and output as a T-word, it was confusing.

If a Tool ID number exist, you will be asked to replace or add. .

Tool library list: This is a list of the tools in t he currently loaded tool library

Partfile Tool list: This is a list of the tools in the current part program

Tooling parameters: This is where you enter information regarding the tool.

Tool Picture: This is a generic picture to help with tool types and parameters.

Speeds & Feeds: Theses are the default speeds and feeds for the tool.