

PSM-00-G2UMS-N-UG

GRAFX[®] II

Version 1.16

User Manual Supplement

Datacut

GRAFX II User Reference Manual Supplement

This document is a supplement to the GRAFX II User Manual (PSM-00-G2UM*-UG) supplied with the GRAFX II CAD/CAM/NC Programming System. It includes new features and changes to the system made after the publication of the latest User Manual. Prior versions of this document may be discarded.

The layout of this document is the same as the User Manual. It is suggested that when referencing a section of the User Manual, you look for the same chapter heading in the supplement for updated information.

For descriptions on changes to the system which occur in revisions within versions (noted by revision letters), you should examine the GRAFX2.REV file which is distributed on the GRAFX II installation disks.

GRAFX II User Reference Manual Supplement

Table of Contents

<u>Installation</u>	5
<u>Mill Tutorial</u>	11
<u>Lathe Tutorial</u>	11
<u>General</u>	12
<u>ROOT menu</u>	14
<u>GEOMETRY > POINT</u>	17
<u>GEOMETRY > LINE</u>	18
<u>GEOMETRY > CONNECTED</u>	19
<u>GEOMETRY > ARC</u>	22
<u>GEOMETRY > ELLIPSE</u>	23
<u>GEOMETRY > FILLET</u>	24
<u>GEOMETRY > MODIFY</u>	25
<u>GEOMETRY > TRANSFORM</u>	26
<u>GEOMETRY > EXAMINE</u>	28
<u>OBJECTS > BOUNDARY</u>	29
<u>OBJECTS > BOUNDARY > ATTRIBUTES</u>	30
<u>OBJECTS > SET</u>	37
<u>OBJECTS > SET > ATTRIBUTES</u>	39
<u>OBJECTS > PATTERN</u>	41
<u>OBJECTS > HATCH</u>	44
<u>DELETE Menu</u>	45

GRAFX II User Reference Manual Supplement

<u>LAYERS Menu</u>	46
<u>DIMENSION Menu</u>	47
<u>TEXT Menu</u>	48
<u>IMPORT Menu</u>	49
<u>EXPORT MENU</u>	51
<u>ENGRAVE Menu</u>	52
<u>ENGRAVE - Corner Cut</u>	55
<u>NC menu</u>	56
<u>NC > CUT PART</u>	57
<u>NC > CUT PART > MILL</u>	58
<u>NC > CUT PART > TURN</u>	86
<u>NC > CUT PART > WIRE</u>	113
<u>NC > CUT PART > STEP</u>	118
<u>NC > POST</u>	119
<u>NC > VIEW/PLOT</u>	120
<u>NC > TOOLING</u>	122
<u>NC > ATTRIBUTES</u>	125
<u>FILE Menu</u>	127
<u>SCREEN Menu</u>	129
<u>MODES Menu</u>	130
<u>ACCESS Menu</u>	131
<u>INFO</u>	145

GRAFX II User Reference Manual Supplement

UNDO 146

GPOST - Using the GPOST System 147

GPOST - Explanation of Queries 149

GPOST - Picture File Symbols 152

Appendix C - Z Surface Projections 154

Appendix F - Communications Protocols 155

Appendix G - File Retrieval 156

Appendix H - Utility Programs 158

Appendix I - Data Paths 163

GRAFX II User Reference Manual Supplement

Reference - Installation

Change the following on Page 1-2:

Supported computers are IBM PC AT, 100% true compatible 80x86 and PENTIUM based processors.

Add the following to the before item 1) on Page 1-5

This version of GRAFX II may have been supplied to operate without the need of a SIM. In which case, item 1), which follows may be skipped.

Change the following at the bottom of Page 1-5:

Change:

A:INSTALL2

To:

A:INSTALL or B:INSTALL

Add to item 5) on Page 1-7:

When digitizing, GRAFX II uses the Summagraphics Bit Pad I emulation with stream mode and binary parameters. You must setup your digitizer to these parameters. You may also need to select a sampling rate and resolution. Fast sampling and .005" resolution is a good start.

Add to "Select a printer:" on Page 1-9:

After selecting the printer, you are then asked to select the Printer Port for the printer output. The *Default Printer Port* as set in the operating system may be selected or you may specifically direct the output to a particular parallel port *LPT1*, *LPT2* or *LPT3*. You may also select *File* and enter a path and file name. When this is done, **PRINT** commands will send the printer image to this file which may later be dumped to the printer with the DOS command:

copy /b (filename) lpt(n):

Add to "Select a plotter:" on Page 1-9:

If the plotter selected is the optional *FPLOT Pen Plotter Emulator*, you will be asked if you wish to run the FPLOT Configuration Program in order to select a

GRAFX II User Reference Manual Supplement

printer and its port assignment. After doing this you then select the port to which GRAFX II will do the output. You may have more than one printer configured in FLOT to different ports (common when using network printers). The *Default Printer Port* as set in the operating system may be selected or you may specifically direct the output to a particular parallel port *LPT1*, *LPT2* or *LPT3*. You may also select *File* and enter a path and file name. When this is done, **XYLOT** commands will send the printer image to this file which may later be dumped to the printer with the DOS command:

copy /b (filename) lpt(n):

Add the following after Page 1-10:

Select Default Units:

You may set GRAFX II to default to either INCH or METRIC data. You will however, have the ability to override the default on a job by job basis when running the GRAFX II system.

OPTIONAL CONFIGURATIONS

Install extended file lists:

This selection is for pre-building expanded directory lists as described in Appendix G of this manual.

Select User defined Data Paths:

You may organize data into separate sub directories. See Appendix I in this supplement for details on how data paths work.

Select Graphic Cursor Blink Rate:

This allows you to specify the rate at which the cursor blinks in the Graphic User Interface. 250 milliseconds is a good rate to select.

GRAFX II User Reference Manual Supplement

Select screen saver timeout:

Enter the number of minutes of non-activity before the screen saver is invoked in GRAFX2. You may also instruct GRAFX2 to automatically execute a POINT PURGE (see DELETE menu) when the screen saver is invoked.

Test Printer:

This selection prints a graphical test file to the last selected printer. This verifies the graphical printing operations of the system and the position alignment of the printer.

Adjust Printing Position:

This selection provides for the adjustment of the printing position in order to center the printer plots correctly. The amount of adjustment needed, if any, can be determined by first doing a TEST PRINTER.

Select User Defined Editor:

If you prefer to use a different Editor program from the one supplied with GRAFX II, you may specify the program you wish to use and the system will load it for file edits.

*Add the following to the **Using Extended Memory** section at the end of Page 1-13:*

With the availability of high performance disk caching software it will usually be more desirable to dedicate extended memory to disk caching rather than a RAM drive. The reason is that a large disk caching buffer will be just as effective as a RAM drive for scratch files and screen saves and in addition will greatly improve the performance of frequent overlay swapping and backup file creation.

The SMARTDRV utility which comes with DOS 5 and higher or WINDOWS 3.X can be used. Refer to your operating system manuals for installation.

One situation in which a RAM drive might be desirable in addition to disk caching is for handling the virtual memory point swap file. A point swap file is created when the number of GRAFX2 points exceeds 2500 and its size requirement grows by 60,000 bytes for each 2500 points up to 25,000 points or

GRAFX II User Reference Manual Supplement

600,000 bytes. GRAFX2 will run faster when more than 2500 points are used by creating a sufficiently large RAM drive and setting the *point swap file* path to it in the configuration program.

Add the following:

Using an Extended Memory Manager in GRAFX II

GRAFX II will automatically use UMB (Upper Memory Block) extra memory. On 386, 486 or PENTIUM® and compatible computers, EMM386 style memory managers may be used to map any extended memory you may have into this region.

If you have such a memory manager installed on your system and it is providing extra memory in the form of UMBs (Upper Memory Blocks), GRAFX II will automatically detect and use this extra memory for its internal operations. This means extra memory space for entity storage or for Zig Zag pocketing for example. The amount of extra memory provided depends on your configuration, but is often in the 30K to 100K range.

NOTE: There is no performance penalty if a memory manager is not installed on your system.

As a concrete example, MS-DOS Versions 5.0 and higher provide an EMM386.EXE memory manager which is installed in the config.sys file in a manner resembling this:

```

      .
device=HIMEM.SYS
      .
device=EMM386.EXE NOEMS
      .
DOS=UMB,HIGH
      .

```

You will need to consult your computer system manual for exact installation.

Starting with MS-DOS Version 6.0, the utility program MEMMAKER is supplied. This utility can be run to have the system automatically configure the DOS startup files to efficiently utilize upper memory blocks.

When running GRAFX II as a DOS application under a WIN9x operating system such as Windows 95, Windows 98 or Windows ME, the memory manager files are distributed in the \windows directory, so that the entries in the CONFIG.SYS file should look like:

GRAFX II User Reference Manual Supplement

device=c:\windows\himem.sys
device=c:\windows\emm386.exe NOEMS
dos=umb,high

The utility program MEMDOS.EXE is installed in the \GX2 directory. Running this program will automatically create or update the CONFIG.SYS file to include the entries listed here.

GRAFX II User Reference Manual Supplement

Update Fig 1 on Page 1-15:

UNDO	INFO	FILE	ACCESS	MODES	SCREEN	ROOT
[graphics display window]						Geometry
						Objects
						Delete
						Layers
						dimenSion
						Text
						Import
						Export
						engraVe
						NC
						Communic
						digitize
						editoR
						Quit
[current menu path]						[coordinates]
[data input / message line]						
[talk window]						F1=HELP F8=REPEAT

Fig. 1: The Main Display Screen

GRAFX II User Reference Manual Supplement

Mill Tutorial

*Move ===== **automatic switch to next screen** ===== before the "Spindle speed (RPM)" parameter on the **Non-Turning Parameters** screens for all tools.*

*Add new parameter to **Non-Turning Parameters** screens for all tools:*

*Tool display orientation: **0 CW (No mirror)***

Change first parameter on Page 2-51 to:

*Final "Z" depth of drill stroke: **-1.05** <ENTER>*

Add new parameter:

*Direction to machine point set: **Forward***

Lathe Tutorial

*Change the order of the first three parameters on the second **Turning Parameters** screen for all tools to:*

Maximum spindle speed (RPM):

Constant surface speed (FPM):

Spindle speed (RPM):

*Add new parameters to **Turning Parameters** screens for all tools:*

Post Processor Option M-code #1:

Post Processor Option M-code #2:

Post Processor Option M-code #3:

*Tool display orientation: **0 CW (No mirror)***

*Add new parameter to **Thread Cutting Parameters** screen on Page 2-115:*

*Direction of thread cutting passes: **Forward***

GRAFX II User Reference Manual Supplement

Reference - General

Insert at the beginning of Page 3-1

Entering Data

When inputting data through the keyboard, the line editor is usually available to make it easy to enter the data. The HELP screen lists the editing control keys. When in an edit line, text may be copied to the clipboard or pasted from the clipboard to facilitate repetitive entries or to copy data from one entity to another.

Operators

Add to Page 3-1

<u>operator</u>	<u>description</u>
>	angular fraction
	tolerance

When inputting angles, angular fractions may be input using the ">" operator. For example:

$50>45>30$ will be evaluated as 50 degrees, 45 minutes, 30 seconds or $50 + 45/60 + 30/3600$ degrees.

Tolerance calculations may automatically be accomplished by using the "|" operator. For example:

$n1/n2$ - RANGE: returns the value $(n1+n2)/2$. -> $1.5|1.51 = 1.505$

$n/t1/t2$ - TOLERANCE: returns $n+(t1-t2)/2$ -> $2|.005|0 = 2.0025$
 $5|+.002|-.005=4.9985$

Functions

Add the following paragraph on Page 3-2:

Angles referenced in the functions sin, cos, tan, asin, acos and atan are interpreted as either DEGREES or RADIANS depending on the units set in the ACCESS > UNITS pulldown menu.

GRAFX II User Reference Manual Supplement

Variables

Change on pp. 3-2 and 3-3:

To define a variable:

- 1) Select *VARIABLE* from the *ACCESS* pull down menu
- 2) Enter the alphabetical character (*a-z*), which will be the name of the variable.
- 3) Edit or enter the value to be assigned to the variable.

Reference - ROOT menu

- EDITOR** - Load the system text editor to edit a specified text file. The system editor may be specified in G2CONFIG.
- USER** - Invoke the user defined applications menu(s) for calling other application programs from inside of GRAFX2. This menu selection appears only if the file *USER.MNU* exists in the GRAFX II home directory. The format for *USER.MNU* is as follows:

```
#menu number
title
item 1*item 1 description@item 1 program
item 2*item 2 description@item 2 program
.....
item n*item n description@item n program
```

The menu must start with a menu number followed by a title. The number of the first menu must be 0. The title will be displayed at the top of the menu in yellow. It is suggested that the title be capitalized to be consistent with the other GRAFX2 menu titles and that leading spaces be used to center it.

The first text in each item is what will appear in the menu. Therefore this text should be no more than 10 characters in length. The first capital letter of these entries will be used as the hot key when selecting menu items with the keyboard.

The second text which must start with the * is the menu item description which will be displayed in the info window when the item is pointed to in the menu.

The third text which must start with the @ is the program which will be run when the item is selected. These may be .exe, .com or .bat files. If programs of the same name with different extensions exist, it is best to specify the extension also. If the program call requires any parameters it should be included in this text the same way you would run the program from the DOS prompt. If you need to run a program from within the directory it resides, you can use a batch file to first change the current directory with a "cd" and then call the program. It is not necessary to change the directory back when

GRAFX II User Reference Manual Supplement

done because GRAFX2 will do this automatically. You may specify a symbolic reference within the program section by including *%symbol* in this section. The symbols supported are:

- jobfile** -The current job file specification including path
- jobname** -The current job name only
- da** - The extension for post processor output files (da is the default)
- cl** - The extension for APT cutter line input files (cl is the default)

In the example that follows, the menu entry **DOWNLOAD** in the *USER.MNU* file calls the communications program *COMMOUT.EXE* to output the current post processed file:

*Download*Download G-code file@\comm\commout %jobfile .%da*

You may specify a maximum of 12 entries in a menu. If you need to include more than 12 application programs you can create more menus by making your menu look something like this:

```
#0
title1
.....
item n*menu 1 item n description@item n program
More*Next menu of programs@+1
#1
title2
item 1*menu 2 item 1 description@item 1 program
item 2*menu 2 item 2 description@item 2 program
.....
```

Here a second menu has been created by including an entry which starts with a + followed by the menu number to load in its program field. The second menu starts with the *#1* and *title2* entries. This procedure may be used to create as many menus as you wish up to a limit of 32 menus.

GRAFX II User Reference Manual Supplement

Note that it is very important to follow the rules exactly when creating the *USER.MNU* file. If a key character or sequence is not done correctly, unpredictable results may occur.

If by chance an application program called from the GRAFX2 user menu hangs up, you can escape without having to reboot the computer by pressing *CTRL-ALT-INS*.

Reference - GEOMETRY > POINT

Change the following definitions:

ENDPOINT - One of the two points at the end of a line or arc or an existing point entity

To get an endpoint, position the cursor directly over a line, arc or point close to the desired endpoint and click the mouse.

INTERSECT - intersection of 2 entities (lines and/or arcs)

If multiple intersect points are desire, the 'F9' key may be pressed during the selection of the first entity to create a window. Points will be created at the intersections of all crossing line entities falling inside the window.

SHIFTED - shifted relative to the previous point defined

When you select SHIFTED, you are prompted to enter X and Y values which represent the translation relative to the last point defined in the POINT menu.

This definition is most useful when geometry dimensions are given in relative rather than absolute terms such as a line with a known horizontal or vertical length.

POLAR - in polar coordinates relative to the previous point defined

When you select SHIFTED, you are prompted to enter X and Y values which represent the translation relative to the last point defined in the POINT menu.

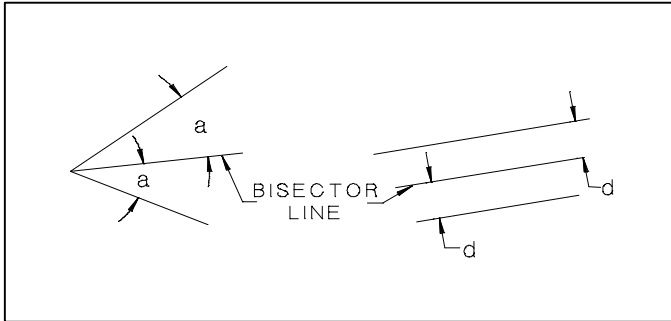
When you select POLAR, you are prompted to enter radius and angle which represent the translation relative to the last point defined in the POINT menu.

Reference - GEOMETRY > LINE

*Change the Line menu selection **ANGLES** to **ANGLE***

*Add to the **LINE** menu:*

BISECT - Create a line which bisects two other lines.



Select the first line, then select the second line. If the lines are parallel, the bisector line will be another parallel line equidistant between the two object lines. The length of the bisector line will be the average length of the two lines measured from their intersection point except in the case of parallel lines which will be bounded within the parallelogram formed by the two object lines.

*Change the last line of the descriptions **LINE > ANGLE** and **LINE > TANGENT > ANGLE** to:*

Then move the mouse to extend the line to the desired length and **CLICK** the **RIGHT** button or **CLICK** the **LEFT** button and select a line or arc with the cross hairs to **SNAP** the line to.

Reference - GEOMETRY > CONNECTED

POINT	- Create a line segment to a point using the point menu
DELTAX	- Create a line segment incrementally in "X"
DELTAY	- Create a line segment incrementally in "Y"
*DELTAZ	- Create a line segment incrementally in "Z"
DELTAXY	- Create a line segment incrementally in "X" and "Y"
*DELTAXZ	- *Create a line segment incrementally in "X" and "Z"
XC	- Create a line segment to an "X" coordinate
YC	- Create a line segment to a "Y" coordinate
*ZC	- Create a line segment to a "Z" coordinate
XCYC	- Create a line segment to an "X,Y" coordinate
*XCZC	- Create a line segment to an "X,Z" coordinate
ANGLE	- Create an angled line segment to a dragged or snapped endpoint
FILLET	- Create a fillet between the last and next line segments
CHAMFER	- Create a chamfer between the last and next line segments
REVERSE	- UNDO the last element in the connected geometry
BOUNDARY	- Link the connected geometry into a boundary

*NOTE: If X and Z coordinates are enabled in the ACCESS > LATHE pulldown menu, these selections replace the ones listed above them, and if the *diameters* parameter is also set, the "X" values will be input as diameters.

Connected Geometry Usage Notes:

Connected Geometry is an alternative method of inputting part geometry. Its advantages are most apparent for drawings using *ordinate* instead of *baseline* geometry. That is, where geometry elements are dimensioned relative to each other instead of from a common reference point. Connected geometry can significantly speed up the geometry input process by enabling geometry to be defined from a single menu and utilizing other time saving features.

GRAFX II User Reference Manual Supplement

Some parts may have their dimensions depart at perpendicular angles to each other, in which case the *DELTA*X and *DELTA*Y(Z) or *XC* and *Y(Z)*C selections enable these dimensions to be input with a single value. To further speed up the process, after a single coordinate type dimension has been entered, the menu automatically shifts to the corresponding selection for the other axis. This eliminates the need to move the menu selection, thereby saving a step. When inputting the value to any single coordinate type dimension, the `F5' key or LEFT MOUSE button may be pressed to enter CURSOR mode whereby the MOUSE or ARROW keys may be used to lay down the end point of the line segment or select an existing line or arc to SNAP the end of the line to. If SNAP mode is entered from the **ANGLE** definition, the following popup menu appears to control how the endpoint of the line will be terminated:

SNAP menu

GEOMETRY	- Snap line segment to an existing line or arc
DELTA X	- Create line segment an incremental "X" distance
DELTA Y	- Create line segment an incremental "Y" distance
*DELTA Z	- Create line segment an incremental "Z" distance
XC	- Terminate line segment at an "X" coordinate
YC	- Terminate line segment at an "Y" coordinate
*ZC	- Terminate line segment at an "Z" coordinate

*NOTE: If X and Z coordinates are enabled in the ACCESS > LATHE pulldown menu, these selections replace the ones listed above them, and if the *diameters* parameter is also set, the "X" values will be input as diameters.

FILLET or *CHAMFER* is selected in the logical order in which they appear on the part. If a fillet or chamfer falls between line A and line B, after line A has been defined, *FILLET* or *CHAMFER* is then selected and when line B is defined, the fillet or chamfer then appears between the two lines. When a fillet or chamfer is pending, the *FILLET* or *CHAMFER* menu selection will be displayed in RED. Attempting to connect a fillet or chamfer to a last segment which happens to be a fillet (can only be attempted after a *REVERSE*), will result in the message "*Cannot connect fillet (chamfer) at this point*".

After a complete part surface has been defined, *BOUNDARY* may be selected to create a boundary from the connected geometry in the order in which it was defined. After this is done, the *CONNECTED* menu is automatically exited and

GRAFX II User Reference Manual Supplement

the *GEOMETRY* menu restored.

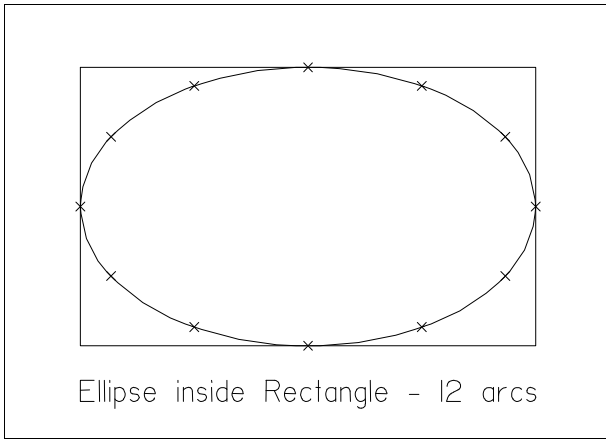
During the process of defining connected geometry, *REVERSE* or *UNDO* will continuously delete segments all the way back as far as the start of the sequence. The coordinates of the last endpoint in the sequence of connected geometry are displayed in the coordinate window just below the menus on the right hand side of the screen.

Reference - GEOMETRY > ARC

*Change the text below the description of **ANGLES** on Page 3-23:*

...Enter the start and finish angles. After entering the second angle, the assumed arc is displayed in green and the complement arc is displayed in light white. Toggle between the two solution arcs with either the LEFT mouse button or the SPACE bar. Accept the solution arc (displayed in green) with either the RIGHT mouse button or the ENTER key.

Reference - GEOMETRY > ELLIPSE



An ellipse is defined as a string of connected arc entities; the ellipse does not exist as a single entity. The following options are available in the ELLIPSE sub-menu:

RECTANGLE - inscribe the ellipse inside a rectangle

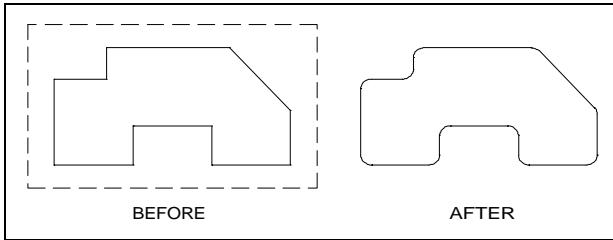
CENTER - describe the ellipse as center point, horizontal radius and vertical radius

ARCS=n - set the number of arcs created to approximate the ellipse (the number of arcs can be between 4 and 99)

Reference - GEOMETRY > FILLET

Insert before the NOTE on Page 3-39:

While selecting the first geometric entity in a FILLET operation, the 'F9' key may be pressed to invoke a windowing operation as indicated by the change in color of the cross hairs. All pairs of lines which intersect each other at a common point falling inside the window will have fillets with the last specified fillet radius created between them and the lines will be trimmed if the TRIM flag is ON. In this type of filleting operation, the smoothest fillet will automatically be created without the toggle being invoked.



Fillet radius options:

Selecting a fillet radius may be accomplished in three different modes as selected in the FILLET menu.

- 1-MODAL** - Use the currently store modal radius.
- 1-ENTER** - Query for the radius value before creating each fillet.
- 1-SELECT** - Use the cross hairs to select another arc whose radius will be used to create the fillet. If the **Esc** key is pressed during arc selection, the last fillet radius will be used.

Reference - GEOMETRY > MODIFY

Add to the MODIFY menu:

ARC-CIRCLE - Convert the selected arc into a full circle.

VALUES - Edit the values which define the point, line or arc selected with the cursor. The following data values are editable for the entity:

POINT - X,Y(*Z) & Z (if 3-D point)

LINE - X,Y(*Z) of endpoint selected

ARC - X,Y(*Z) of center, radius, angle of endpoint selected

Lines and arcs may not be edited if they are part of a boundary.

*NOTE: If X and Z coordinates are enabled in the ACCESS > LATHE pulldown menu, these selections replace the selections referencing the Y axis. If the *diameters* parameter is also set, the "X" values will be displayed and input as diameters.

Expand definition in the TRIM/EXTEND sub menu:

1 ENTITY - Trim or extend 1 element of geometry to an intersecting element or to an endpoint in the case of a parallel line.

Reference - GEOMETRY > TRANSFORM

Add to MIRROR menu:

***Z AXIS** - mirror over the Z axis.

POINT > - mirror about a point.

HORIZONTAL - over a horizontal line through the point

VERTICAL - over a vertical line through the point

ANGLE - over a line at an angle through the point

Add to TRANSLATE menu:

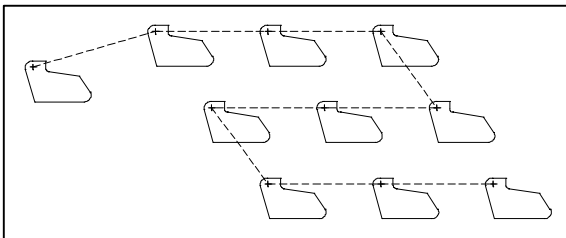
***XZ** - Directly enter the X and Z displacement

HORIZONTAL - By the horizontal component between 2 points

VERTICAL - By the vertical component between 2 points

SET - Onto a predefined point set (excluding positioning points). The point set may be defined using the PATTERN menu.

DRAG - Select a point, line or arc with the cursor and drag it to a new position. If selected line or arc is part of a boundary, the entire boundary is dragged. If a point within a boundary or point set is selected, only the selected point is dragged.



TRANSLATE > SET

GRAFX II User Reference Manual Supplement

*NOTE: If X and Z coordinates are enabled in the ACCESS > LATHE pulldown menu, this selection replaces the selection referencing the Y axis. If the *diameters* parameter is also set, the "X" value will be input as diameters.

Add to ROTATE menu:

INCREMENT -Toggle between incremental rotation angle or angle defined as
ABSOLUTE absolute resultant angle of a selected base line after the transform

.

Add to the TRANSFORM > LATHE description on Page 3-46:

The 'connecting' lines will be stored as hatch entities.

Modify the beginning of the third paragraph on Page 3-55 describing SHIFT:

After selecting SHIFT, enter the shift distance in response to the prompt in the input window, then move the cursor into the graphics area.

Reference - GEOMETRY > EXAMINE

- DIMENSIONS** - Report statistics on picked dimension entity
- TEXT** - Report statistics on picked text entity
- REPORT >** - Write a file to disk reporting the data on the geometry within all boundaries and point sets contained in the current job. The default file name is the current job name with a .txt extension. After the file is written, a popup menu appears:
- DONE** - Nothing further, dismiss the menu.
- VIEW** - Display the report file on the screen.
- PRINT** - Print the report file on the printer.

After picking the entity or object to examine, a popup screen appears containing the statistics of the object. The cursor may be moved to any decimal number displayed on the screen and selected with the mouse button. When this is done, a query appears asking which variable letter the value is to be stored in. This enables parameters to be easily fetched from defined geometry and input into expressions.

Reference - OBJECTS > BOUNDARY

- LINK** - 2) Click again on the first element or strike the SPACE BAR...
- POSITION** - Invoke the POINT menu to define tool positioning points relative to a boundary.
- When doing DRILL or THREAD operations in turning work, if the line picked on which the operation to be performed is in a boundary, any positioning points included in the boundary will be positioned to during the operation.
- DELETE** - Delete an accepted boundary.
- MERGE** - Merge two boundaries into one. This provides an easy method of appending approach and retract segments, including positioning points, to an already existing boundary. The endpoint of the starting boundary must match the start point of the ending boundary, otherwise the merge will not proceed. All retract elements of the starting boundary and approach elements of the ending boundary will automatically be untagged.
- ATTRIBUTES** - Invoke menu to create or modify machining attributes associated with boundary elements.

Link options:

The LINK selection can be toggled to perform the linking operation in three different ways. To change the mode of operation, position over the menu selection which is shown below and either CLICK the mouse or strike the <ENTER> key. The following are the descriptions of the available LINK modes:

- 1 - PICK** - Upon reaching a branch point in the boundary, the system asks the user to pick the next element to link into the boundary.
- 1 - CW** - The most clockwise element will automatically be linked when a branch point is reached.
- 1 - CCW** - The most counter clockwise element will automatically be linked when a branch point is reached.

Reference - OBJECTS > BOUNDARY > ATTRIBUTES

The ATTRIBUTES menu is used to store features or machining information directly into elements in part boundaries. Most simply it provides a way of providing much more control of the toolpaths generated from GRAFX2 and allows access to custom features of many Post Processors.

APPEND - Add an attribute to a boundary element or entire boundary. On GRAFX II/LE systems, only the **MACRO** sub-menu is accessible and is automatically invoked.

MODIFY - Examine, Modify or Delete attributes of a boundary element or entire boundary. This selection brings up a menu of the attributes which are stored in the picked element or global boundary and enables the user to change the attributes' values, delete individual attributes or insert new attributes in front of, or after, any other attribute associated with the element.

COPY - Copy the attributes of a boundary element to another element or global attributes from one boundary to another.

MOVE - Move the attributes of a boundary element to another element or global attributes from one boundary to another.

DELETE - Remove all the attributes from a boundary element or the global attributes from a boundary.

SHOW - Draw elements with attributes in highlighted color. Boundaries or sets with GLOBAL attributes will display a highlighted box at the start of the boundary or set.

REPORT > - Write a file to disk reporting the data on the geometry and attributes within all boundaries and point sets contained in the current job. The default file name is the current job name with a .txt extension. After the file is written, a popup menu appears:

DONE - Nothing further, dismiss the menu.

VIEW - Display the report file on the screen.

PRINT - Print the report file on the printer.

GRAFX II User Reference Manual Supplement

ELEMENT - Toggle between attributes associated with individual elements
GLOBAL and those associated with the entire boundary.

Usage Notes:

When attributes are attached to any ELEMENT in a BOUNDARY or SET, any toolpath generated, which tracks the boundary or point set, will execute the attribute prior to the motion associated with that element. When attributes are attached GLOBALLY to a BOUNDARY or SET, any type of toolpath generated will execute the attributes according to the GLOBAL attributes specified.

This allows the programmer to do things such as change a feedrate, control cutter comp., insert an ASCII block, send a control code to a Post Processor, jump a clamp, etc. in the middle of a machining operation. This technology permits the permanent storage of these types of features inside the geometric model, whereby the machine codes are then automatically output when the toolpath is generated.

When boundaries and sets are copied or moved through LAYER or TRANSFORM commands, their attributes are preserved in the new boundaries and sets.

Attributes are supported in the reading and writing of .VEC files. This is useful in controlling the machining of fonts and creating machining methods which can be embedded in template boundaries.

The attributes available closely resemble STEP MODE operations in GRAFX2, with several additions. The ATTRIBUTE pop up menu appears when APPEND is selected or an insert attribute operation is executed within a MODIFY selection. The attributes are:

FEED - Change the feed rate (ONE-SHOT or MODAL).

SPEED - Change the spindle speed.

Z LEVEL - Manually move to a new "Z" position (STEP or RAMP at a FEEDRATE).

G-CODE - Insert a control code.

M-CODE - Insert a machine code.

DATA - Insert a data value.

GRAFX II User Reference Manual Supplement

ASCII	- Insert an ASCII block directly (with or without "N" word).
SYNCH	- Set a synchronization point for 4 axis lathe or wire programs.
BREAK	- Set a breakpoint whereby toolpath generation enters STEP mode.
ID TEXT	- Store ASCII text to identify the boundary element.
GFX	- Invoke menu for GFX machine control syntax.
MACRO	- Invoke a menu for commonly used machine control attributes. This menu is automatically invoked when ATTRIBUTES > APPEND is selected in GRAFX II/LE systems.
R LEVEL	- Change the current R-Level.
PART RT	- Retract drill to R-Level.
FULL RT	- Retract drill to initial Z-Level.
LEFT	- Cutter Comp. Left.
RIGHT	- Cutter Comp. Right.
CANCEL	- Cutter Comp. Cancel.

When the GFX selection is made, another popup menu appears with syntax found in the GFX part programming language. These types of commands often control customized features of Datacut Post Processors.

ABSC	- Control the output of absolute or incremental data.
ADOV	- Turn on/off acceleration deceleration override.
BLKD	- Turn on/off output of block deletes in the post processed output.
CLMP	- Output a clamp on or clamp off "M" code.
COOL	- Turn the coolant on or off.

GRAFX II User Reference Manual Supplement

DC	- Execute a GFX "DC n" type command.
DWEL	- Output a programmed dwell.
HOLD	- Output a program stop "M" code.
INDX	- Output an index "M" code.
KLOC	- Output a keylock "M" code.
NOUT	- Control the block sequence "N" words.
OPTN	- Turn on/off option shift for post processors.
OVBP	- Output an override bypass on or override bypass off "M" code.
SPDL	- Turn the spindle on or off.
SPRE	- Output the "M" code assigned to the SPRE ON & OFF functions.
TSTP	- Execute a "TSTP n" command.

When GRAFX II generates toolpaths, there are usually approach and retract blocks of motion generated in addition to the toolpath associated with elements of the boundaries or point sets. These blocks of motion may also be tagged with attributes by use of the GLOBAL mode when APPENDING or MODIFYING attributes. Access to these attributes is accomplished by toggling the ELEMENT/GLOBAL menu selection to GLOBAL and then when an attribute operation is selected, any element on the boundary/set may be picked with the cross hairs and a popup menu appears in order to select the particular leg of the approach or retract motion desired. This menu is as follows:

1-INIT APPROACH - Attribute on position move to first element. This applies only if there are no approach positioning points.

2-INIT APPROACH - Attribute on position move to "Z approach" clearance. (NON-TURNING)

"Z" PLUNGE - Attribute on plunge to the "Z depth". (NON-TURNING)

"XY" APPROACH - Attribute on XY motion from last positioning point, if it exists, to first element of the boundary. (NON-TURNING)

GRAFX II User Reference Manual Supplement

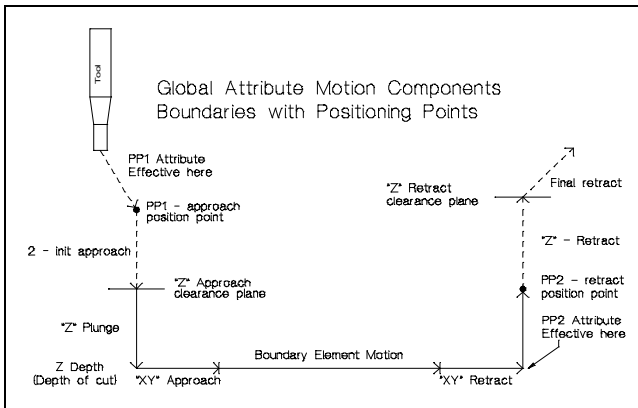
"XY" RETRACT - Attribute on XY motion to retract positioning point (if it exists). (NON-TURNING)

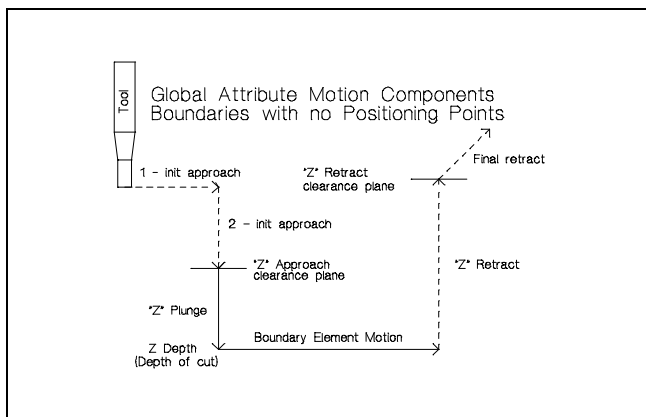
"Z" RETRACT - Attribute on vertical move to "Z retract clearance". (NON-TURNING)

FINAL RETRACT - Attribute on final motion away from operation. (This is the only active menu selection when a point set is selected.)

When this menu appears, selections which are not accessible either because they do not apply to the type of operation when APPENDING or they do not exist when MODIFYING, will be locked out from selection as designated by their light cyan color in the menu.

The diagrams that follow show the elements of motion that correspond to the menu selections listed above. Note that the positioning point attributes take effect on the motion block *TO* (i.e. "before") the positioning point. There is no element of CONTOUR or DRILL motion that is not accessible to attribute control.





ATTRIBUTE SPECIAL CASES

Positioning Points:

Positioning point attributes affect nearly all motion that involves positioning points. This includes approach and retract CONTOUR style motion, DRILL motion (approach, internal and retract), and entry motion to the rough passes of SPIRAL POCKETing motion. The exception is entry motion to the rough passes of ZIG ZAG POCKETing motion.

The Z-Level Attribute is not implemented for positioning points. Generally, you should use the Z-coordinate of the positioning point instead.

Clamp Jumping:

To jump a clamp in a CONTOUR operation, use the Z-Level Attribute of the associated boundary elements. To jump a clamp in a DRILL operation, use the Z-Coordinate of internal positioning points.

Global Attribute Notes:

1. A "Global Motion Element" is defined as a leg of motion automatically generated without any direct correspondence to predefined geometry. The "2-Init Approach" and "Z-Retract" selections of the "GLOBAL" menu are two examples of Global Motion Elements.
2. Attributes of Global Motion Elements pertain to CONTOUR operations and the finish pass of POCKETing (SPIRAL and ZIG ZAG) operations

GRAFX II User Reference Manual Supplement

only. More specifically, they have no effect on the entry/retract motion of POCKETing rough passes.

3. The Z-Level Attribute is implemented only for the Z-Retract Global Motion Element. Setting this attribute on any other Global Motion Element will have no effect on any tool motion. You should control Z-axis motion using the appropriate Machining Parameter.
4. The One-Shot/Modal selection of the Feed Attribute is essentially irrelevant on all of the Global Motion Elements. In effect, it is always "One-Shot". This is because each leg of motion is generated more-or-less independently from the others using its own non-attribute defaults. As a matter of practice, it is suggested that you always choose "One-Shot".
5. Tool motion associated with point sets (e.g. DRILLing) contains only one Global Motion Element - the Final Retract. All other components of motion are directly associated with geometric elements of the point set either the points themselves or positioning points.

Z-Level Attribute Implementation:

The Z-Level Attribute is implemented for the elements of a boundary and the Z-Retract Global Motion Element only. It is not implemented on any of the other Global Motion Elements. It is also not implemented on positioning points or on the points of a point set.

When does an attribute take effect?

An attribute always takes effect on the motion *ALONG* the line or arc element or *TO* the point element that has the attribute. Motion *AWAY* from the element is not affected by any attribute on the element.

Reference - OBJECTS > SET

POSITION - Invoke the POINT menu to define tool positioning points relative to a set.

When positioning points are defined in between non positioning points in the point set, drilling operations, upon encountering an interior positioning point, will position to the point, and the drilling cycle will be reinstated on the next non positioning point member of the point set. This is commonly done for clamp avoidance.

CONNECTED - Invoke the POINT menu to define points and simultaneously include them in a point set in the order they are defined. While defining a point set in this manner, UNDO or REVERSE will remove the last point from the set and delete the point at the same time. CANCEL will delete all points so far included in the set. The point set is closed by the ACCEPT menu selection or by pressing the space bar on the keyboard.

ACCEPT - ...The length of the point set path is displayed upon acceptance. Any point in the set which has a "Z" value associated with it will be displayed in brown instead of red when MODES > DRAW is disabled.

MERGE - Merge two point sets into one. This provides an easy method of appending approach and retract positioning points, to an already existing point set.

DELETE - Delete an accepted point set.

DEPTH - Used to attach a drilling depth to points within the point set. When a point set is drilled, those points tagged with a depth will be drilled to that depth while points which are not tagged will be drilled to the depth specified in the parameter screen. DEPTH may be applied to either pending or already accepted point sets. When editing an existing "Z" value, delete the displayed value to remove the tagged depth from a point. When entering the DEPTH value, the 'F5' key or LEFT MOUSE button may be used to bring up the Drilling Depth Calculator (see Page 69).

If no "Z" depth parameter is entered in the current drilling operation, the DEPTH attached to a point remains modal in the

drill set.

When removing a point in a set which has a depth attached to it and the next point in the set has no depth specification (either by an attached DEPTH or by the "Z" depth parameter), a dialog box will appear which will allow the depth of the removed point to be automatically transferred to the next point in the set.

INVERT

- Invert (reverse) the direction of an accepted point set.

Reference - OBJECTS > SET > ATTRIBUTES

The ATTRIBUTES menu is used to store features or machining information directly into elements in point sets. Most simply it provides a way of providing much more control of the toolpaths generated from GRAFX2 and allows access to custom features of many Post Processors.

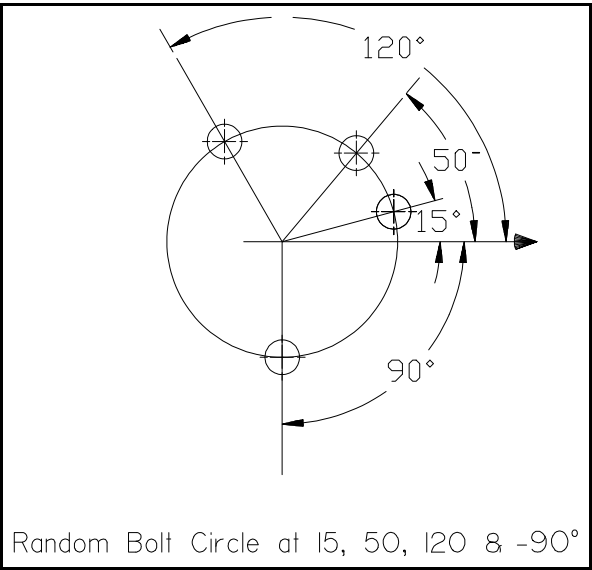
- | | |
|--------------------|--|
| APPEND | - Add an attribute to a set element or entire set. |
| MODIFY | - Examine, Modify or Delete attributes of a set element or entire set. This selection brings up a menu of the attributes which are stored in the picked element or global set and enables the user to change the attributes' values, delete individual attributes or insert new attributes in front of, or after, any other attribute associated with the element. |
| COPY | - Copy the attributes of a set element to another element or global attributes from one set to another. |
| MOVE | - Move the attributes of a set element to another element or global attributes from one set to another. |
| DELETE | - Remove all the attributes from a set element or the global attributes from a set. |
| SHOW | - Draw elements with attributes in highlighted color. Boundaries or sets with GLOBAL attributes will display a highlighted box at the start of the boundary/set. |
| REPORT > | - Write a file to disk reporting the data on the geometry and attributes within all boundaries and point sets contained in the current job. The default file name is the current job name with a .txt extension. After the file is written, a popup menu appears: |
| DONE | - Nothing further, dismiss the menu. |
| VIEW | - Display the report file on the screen. |
| PRINT | - Print the report file on the printer. |

GRAFX II User Reference Manual Supplement

ELEMENT - Toggle between attributes associated with individual elements
GLOBAL and those associated with the entire set.

For a detailed explanation on ATTRIBUTES, refer to the usage notes in the
OBJECTS > BOUNDARY > ATTRIBUTE section on Page 30 of this manual.

Reference - OBJECTS > PATTERN



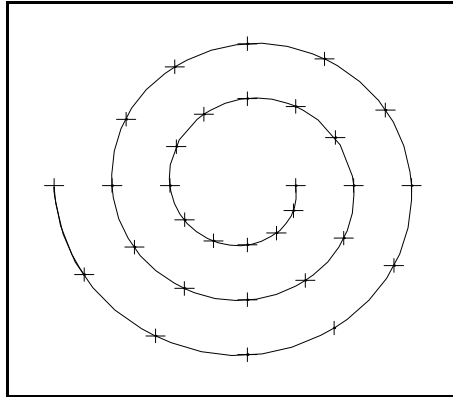
Add to the **Bolt Circle Parameters**:

Angular placement of points: **UNIFORM RANDOM**

The "Angular placement of points" parameter allows bolt circle points to be generated in either a uniform pattern where the angular spacing is constant throughout the pattern, or in a random pattern where each angular position is specified.

Add the following note to the "Angle of 1st point in bolt circle", "Number of equally spaced points in the bolt circle" and "Direction in which points will be generated" parameters:

This parameter applies to UNIFORM bolt circles only.



SPIRAL - Generate points along an Archimedean spiral

Use the **POINT** menu to indicate the position of the center of the Archimedean spiral, the parameter screen shown below then appears:

Archimedean Spiral Parameters

Starting radius:

Ending radius:

Starting angle:

Total angular translation:

Points per quadrant:

Direction in which points will be generated:

Automatically link the pattern into a point set?

The "*Starting radius*" is a positive value or 0 which measures the distance from the center point of the spiral to the first point generated.

The "*Ending radius*" is a positive value or 0 which measures the distance from the center point of the spiral to the last point generated.

The "*Total angular translation*" measures the length of the spiral. For example if the spiral makes $2\frac{1}{2}$ revolutions, this value = $2.5 * 360 = 900$.

The "*Points per quadrant*" is the number of points that will be generated every 90 degrees. If an arc fit is used to produce a curve, generating 4 points per quadrant yields an extremely accurate and smooth result.

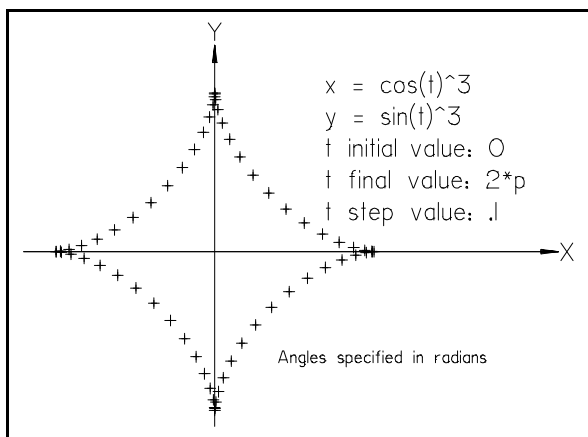
The "*Direction in which points will be generated*" parameter indicates which circular direction to use, CW or CCW, as the points are generated from start to end.

GRAFX II User Reference Manual Supplement

The pattern of points generated into a point set may be arc fit into a curve to create an Archimedean spiral curve to be machined.

Change the second NOTE on Page 3-85 to:

NOTE: Arguments to the trigonometric functions, i.e. $\sin(a)$, $\cos(a)$, and $\tan(a)$, are angles specified in either degrees or radians according to the UNITS selected in the ACCESS pulldown menu.



Hypocycloid from PARAMETRIC

PARAMETRIC - Generate points from two input parametric equations.

This is a point generating function similar to *FUNCTION*, except that the x and y coordinates of the points are derived from two input equations instead of one. One equation expresses x as a function of an auxiliary variable t and the other expresses y as a function of t . The range and step value of the parameter t is specified.

Parametric Equation Parameters

$x =$ <text>

$y =$ <text>

t initial value:

t step value:

Automatically link the pattern into a point set? **Yes No**

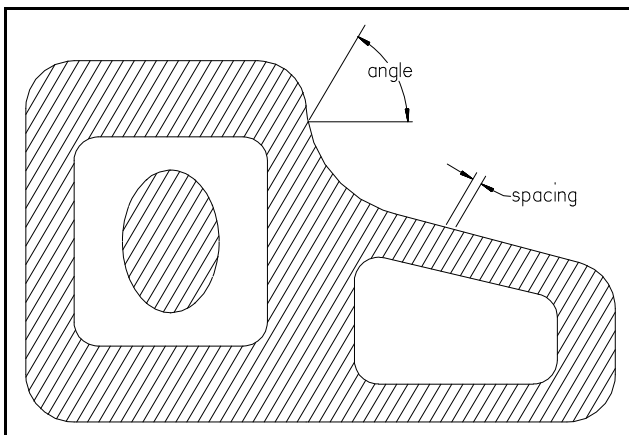
Reference - OBJECTS > HATCH

HATCH

-Fill a closed boundary with a crosshatching pattern. You are asked to input the angle of the hatch lines and the spacing between them, after which the crosshairs appear in order to select the boundary or boundaries to be filled with the pattern. If more than one boundary is selected, boundaries nested inside others will alternately fill and not fill and those not nested inside others will individually fill with the pattern.

When selecting the boundaries with the crosshairs, the 'F9' key may be used to invoke a window selection of multiple boundaries. After selecting all boundaries either individually, with windows, or combinations of the two, the crosshatching operation will commence upon pressing the ESC key. The lines generated by this operation can be deleted with an UNDO.

Crosshatch lines may be looked upon as special line entities. Once created, they are treated just like any other line entities except that they cannot be linked into boundaries and will not have TICKS drawn at their endpoints.



Crosshatch with nesting

Reference - DELETE Menu

- HATCH** - Enable the crosshairs to form a window around all crosshatching lines to be deleted. This operation is similar to DELETE > WINDOW except that only crosshatch lines are affected.
- DUPLICATES** > - Remove redundant points and/or geometric entities not part of boundaries or point sets. After selection, a popup menu appears:
- POINTS** - Remove duplicate POINT components from entire database only.
- GEOMETRY** - Remove point, line and arc entities from current layer only.
- BOTH** - Remove both POINT components and GEOMETRY.

Reference - LAYERS Menu

CHANGE - Turn layer display on and off

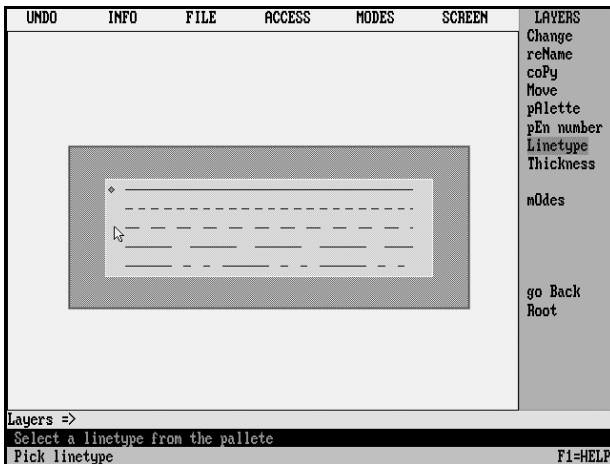
NOTE: Selecting "Turn all layers on" or "Turn all layers off" will toggle all layers at once.

The current layer is displayed in bold black.

REMOVE - Remove the currently active layer. If the layer contains any entities, a dialog box will query whether to proceed and delete all entities contained in the layer. After removing a layer, the active layer is set to the first layer in the list. UNDO may be used to restore the layer and its entities immediately after executing REMOVE.

LINETYPE - Pop up a menu to select a linetype for the currently active layer.

Any of 5 different linetypes may be selected by positioning the cursor over the linetype pattern as it appears. The selected linetype will be plotted by the XYPLOT command and displayed on the screen when drawing LINE entities.



Change the next from last paragraph on Page 3-93:

The COPY sub-menu is identical to the ENTITIES sub-menu (Page 3-57) with the addition of DIMENSION and HATCH entity types...

Reference - DIMENSION Menu

- DIAMETER** - Draw a diameter dimension on an arc or circle. Select the arc or circle where the diameter dimension is to be drawn.
- POINT** - Draw a point dimension of the form (x,y) or (x,z) if lathe dimensions are enabled. The POINT menu is invoked to define the point to be dimensioned.
- FLIP** - Flip the direction of the arrow leaders generated when creating a LINEAR, RADIUS or ANGLE dimension with AUTO/ON.

GRAFX II User Reference Manual Supplement

Reference - TEXT Menu

When entering text entities, symbols may be inserted into the text string that will be output on the display or plot according to the table below:

Symbol	Output
&[job]	The current job name
&[file]	The current job file and path
&[descr]	The current job description
&[layer]	The current active layer
&[date]	The current date
&[time]	The current time
&[scale]	The current drawing scale

GRAFX II User Reference Manual Supplement

Reference - IMPORT Menu

ASCII

add to format menu

3 - RT - Polar point data (Radius, Angle)

4 - TR - Polar point data (Angle, Radius)

GRAFX DMB

1 - P - Import GRAFX point data only

2 - LC - Import GRAFX line and arc data only

3 - PLC - Import GRAFX point, line and arc data

VEC - Import a Datacut VEC vector/font file

CL - Import data from an APT CL format cutter line file

MODES >

BOUNDARY - Enable or disable importation of geometry into boundaries or point sets. When enabled, GRAFX-DMB data will link continuous toolpath contours into boundaries and continuous drill cycle sequences into point sets. This mode also places the IMPORT of DXF POLYLINES into boundaries. All other IMPORT file types are unaffected by this mode.

FILTER - Enable or disable the filtering of imported data by layers when importing DXF, IGES, CADL, HPGL, DMB(Grafx) and VEC files. In this mode, only data which resides in layer names which have corresponding layer names which are ON, will be imported.

File Importation Notes:

- 4) When importing **GRAFX-DMB** or **CL** files with **LAYERS** mode on, the imported geometry is assigned to layers according to tool number. Part draw data will be stored in a layer named "PART DRAW" (GRAFX-DMB only), and toolpath data will be stored in layers named "TOOL n" where n is the tool number. When Importing **HPGL** files with **LAYERS** mode on, the imported geometry is assigned to layers according to the pen number (1, 2, 3, etc.).

GRAFX II User Reference Manual Supplement

- 5) Several formats will IMPORT points as 3-D point entities. If these 3-D points are subsequently linked into a point set, they may be machined as 3 axis surfaces with the *PT to PT* toolpath operation. The formats supporting 3-D points are DXF POINTS and POLYGON MESH, IGES POINTS, CADL POINTS, ASCII XYZ and VEC POINTS.
- 6) When importing **CL** files produced by the *STRATA[®]* "ncoutput clfile" command and some other toolpath generators, the file *cl.sys* should exist with the numeral 1 in it to instruct GRAFX2 how to handle arc data.

STRATA[®] is a trademark of Spatial Technology Inc.

Reference - EXPORT MENU

HPGL	- Export HPGL format graphics (.PLT file)
VEC	- Export Datacut vector/font file (.VEC file)
GFX	- Export GFX geometry definitions (.PT file).
APT	- Export APT geometry definitions (.APT file).
COMPACT	- Export COMPACT II [®] geometry definitions (.C2 file).
PRECIS=n	- Set number of digits output in DXF, IGES, VEC, GFX, APT & COMPACT Export operations. The default is 6 places when PRECIS=0.
MODES >	- Pull down menu for modal settings pertaining to file exportation.
BOUNDARY	- Enable or disable exportation of geometry included in boundaries or point sets only. When enabled, the geometric entities are exported in their linked sequences instead of in groups of points, lines and arcs.

File Exportation Notes:

Delete the first paragraph referring to HPGL files

Add to DXF export notes:

GRAFX2 entity colors exported to a DXF file will be mapped to the AutoCad[®] 8-color palette.

Reference - ENGRAVE Menu

SCALE

-...This selection queries for the character height scale factor, the character width scale factor and the slant angle. When the height and width scale factors are not equal, the resultant characters are stretched whereby arcs are transformed into elliptical data. The aspect ratio of the axes scales are maintained through resizing done when ALIGN justification is done. Character spacing is automatically adjusted by the width scale factor. When a slant angle is specified, the resultant characters are slanted and arc elements are transformed into elliptical data.

JUSTIFY >

-Pop up menu for modal justification settings for engraving on GEOMETRY.

NONE

- Engraving done on GEOMETRY will prompt for a start point for the text.

LEFT

- Left justify the text onto the GEOMETRY at the current scale and spacing.

RIGHT

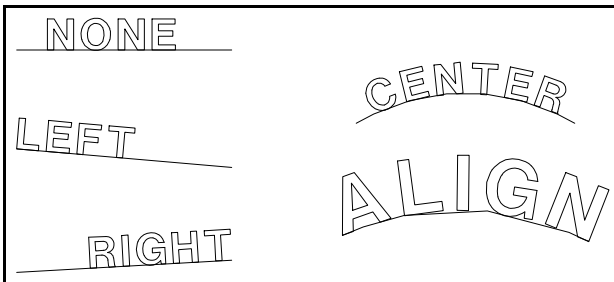
- Right justify the text onto the GEOMETRY at the current scale and spacing.

CENTER

- Center the text onto the GEOMETRY at the current scale and spacing.

ALIGN

- Automatically scale the text so that it aligns exactly to the start and end of the GEOMETRY. The computed scale will remain the character base scale after the operation.



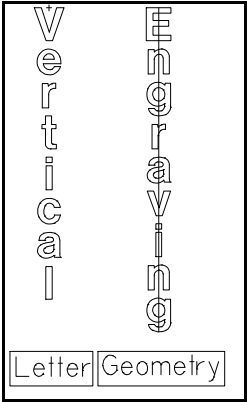
GRAFX II User Reference Manual Supplement

FONT SIZE - Input the font size in terms of pointsize at 72 points per inch.

MODES > - Pull down menu for modal settings pertaining to engraving.

BOUNDARY - Enable or disable the creation of machinable boundaries on engraving operations. This mode defaults to ON and would probably be turned OFF only when using engraving for the creation of text in drawings.

VERTICAL - When this mode is active, LETTER engraving or GEOMETRY engraving on a vertical line creates the characters in a vertical orientation with each character centered on its directional vector. This mode will automatically be turned off when GEOMETRY engraving on an arc or non vertical line.



GENERAL RULES:

Delete paragraphs 1 and 2 on Page 3-107; dynamic entity resizing removes the necessity for this.

Delete paragraph 4 on Page 3-107; the ability to import VEC files removes the absolute need for this.

FONT CREATION:

Changes to the Font Creation Process descriptions

- 2) Create a boundary for each character. Determine how the character is to be machined and create the boundary accordingly. The machinable boundary may be marked with approach and retract elements. Geometry which is not linked into boundaries will become part of the character and will produce the corresponding geometry entities when the character is accessed.

- 4) MOVE each character to the appropriate layer whose name corresponds to

GRAFX II User Reference Manual Supplement

the character being moved. Layer names may be either the one corresponding character or of the form *k_text* where *k* is the corresponding character and *text* is any descriptive text of up to 30 more characters, i.e. *AI* is not acceptable but *A* or *A_I* is.

Examples:

The boundary for the character *A* (capital A) would be moved to a layer named *A* or possibly *A_Symbol* for *A*.

The boundary for the character *v* (small V) would be moved to a layer named *v* or possibly *v_Vise*.

The boundary for the character *@* would be moved to a layer named *@* or possibly *@_special character #1*.

- 6) Review each character's associated layer and check that the boundaries and/or geometry have been correctly created. Layer names longer than one character where the second character is not an UNDERSCORE are ignored (see step #4).
- 7) Select ENGRAVE and CREATE. Type the name of the font you wish to create then enter the description of the font with up to 79 characters.
- 9) *Change to:*
It is suggested that the original geometry (.GEO) file be retained for the purpose of making changes to a font at a later date. However, the .GEO file can be reproduced from the font file by an IMPORT > VEC with the LAYERS and BOUNDARY modes enabled.
- 10) *Delete this step; dynamic entity resizing removes the necessity for this.*

Reference - ENGRAVE - Corner Cut

Add the following to the Corner Cut parameter screen:

Effective tool diameter at "Depth of cut":

Enter or report the effective cutting diameter of the tapered tool at full depth. The displayed value is automatically updated whenever a change made to another parameter causes a change in the effective cutting diameter. If a value is entered directly to this parameter, the value displayed by the "*Depth of cut.*" parameter will automatically adjust correspondingly.

NEXT SCREEN:

This selection has been added to the bottom of the parameter screen so that the screen is not automatically dismissed after entering the "*Effective tool diameter.*" parameter. The screen is dismissed by pressing *Enter* or the *RIGHT* mouse button at this selection.

Usage Notes:

Page 3-117: Change the warning message description and NOTE to:

After a warning message is displayed, the current cursor pointer is returned to the erroneous parameter. If desired, the error may then be bypassed by using the *DOWN* arrow.

Reference - NC menu

*Change menu selection **PLOT** to **VIEW/PLOT***

- FAMILY** - Create a part family toolpath parameter file. Part family files designated with .DMF extensions are similar to DMB toolpath files and may be stored on the system to be used as default templates when family of parts are programmed by using the same or similar machining operations on different geometric part shapes.

When selected, the user is asked for the name of the DMB file to be transferred to the part family file. The default is the DMB file corresponding to the current JOB name, but any existing DMB file may be specified. Next, the user is asked for the name of the part family file which will get a .DMF extension attached to it. The default path for GEO and DMB files are assumed. After specifying the family name, a dialog box appears asking if PART DRAW operations are to be stripped out when creating the file. This usually will be YES since part family files normally apply the same machining operations to different part geometries.

Accessing the part family files is done when entering the name of the toolpath file during toolpath creation. After entering the DMB file name, which is usually the same as the JOB name, instead of terminating with the ENTER key, the user terminates with the 'F5' key. This brings up a menu of all the DMF files found in the GEO/DMB default path. If one of these files is selected from the menu, the operations parameters of the selected file will be used as the defaults in creating the new toolpath file. If a part family file is selected in this manner, and the DMB file for the job already exists, a warning dialog box will appear, asking the user if the existing DMB file is to be overwritten.

- SETUP** - Load the text editor to enter or modify the part setup information that is stored in the file *jobname.SUP*. When this file exists, the operations REPORT will include the text of this file in the setup section of the report.

Reference - NC > CUT PART

Add the following parameters to the initial parameter query screen:

Machine orientation: **Vertical Horizontal Unspecified**

For MILL only, select machine orientation. This parameter will affect the orientation of the PLOT and, other than with some custom Post Processors, has no profound effect on the NC program.

Tool library name:

Select a tooling library to be used for menu selection of tooling files on the current job. The 'F5' key or LEFT MOUSE button may be used to menu select the available tooling libraries. See *NC > TOOLING > LIBRARY* for information on creating tooling libraries.

Part load/unload time (mins):

Enter the total time for loading and unloading the workpiece on the machine for inclusion in the time study report generated by POST. This value may include fractional data.

Reference - NC > CUT PART > MILL

Modify paragraph 2 on Page 3-123

(the initial default value is tool number 1 and it is automatically bumped up by 1 each time you select PARAMETERS except if a next toolpath operation already exists in the file in which case the tool number of that operation will be displayed in the prompt).

Add to Page 3-123 between paragraphs 2 and 3

If the previous tool number is entered to the tool number prompt, the next operation of that tool is opened for toolpath creation or modification in the case of an existing toolpath file.

If toolpath operations already exist, by entering the 'F5' key or the LEFT MOUSE button to the tool number prompt, a menu of the existing operations in the toolpath file is displayed, allowing the user to immediately access any operation by selecting it off the menu. The menu selections are of the form *Tn Description*, where *n* is the tool number and *description* is the operation description as entered in the parameters or the generic name of the operation if none was entered. *Draw Part* operations are not displayed with a tool number. The last operation passed to the output file is displayed in red.

If a *Draw Part* operation is selected off the menu of operations, the boundary corresponding to the selected Draw Part operation is highlighted and the following dialog menu appears:

FOUND PART DRAW - WHAT DO YOU WANT TO DO?

- | | |
|---------------|---|
| REDO | - Reselect the part boundary and reenter the top and bottom surface "Z" levels for the Draw Part operation. |
| DELETE | - Remove the Draw Part operation from the toolpath file. |
| KEEP | - Pass through the Draw Part operation to the output file. |
| FILE | - Move the current Draw Part operation to a file for later retrieval. This operation initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name in which to store the operation and appends a "DMC" extension to it. The default path for GEO |

GRAFX II User Reference Manual Supplement

and DMB files is assumed. After storing the operation in the "DMC" file, the Draw Part operation may be fetched back into a toolpath file at any point.

CANCEL - Do not take any action.

When DELETING the first operation of a tool or inserting a new operation before the first operation of a tool, the next sequential operation should be re-executed in order to create or delete a tool change as required. A tool change is automatically supplied when executing an operation where the tool number differs from the previous operation. When bringing up the menu of operations, any inconsistencies in missing or redundant tool changes are noted by a lower case *t* in the operation which should be re-executed.

Modify paragraph 4 on Page 3-123

After completing the parameters the following dialog menu appears:

EXECUTE THIS OPERATION?

YES - Proceed with the operation. The cross hairs come up and you are prompted to select the boundary(s) to be machined or the point set(s) to be drilled.

NO - Do not proceed with the operation.

BACK - Go back to the last displayed parameter screen.

FILE - Move the current operation parameters to a file for later retrieval. This operation initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name in which to store the operation and appends a "DMC" extension to it. The default path for GEO and DMB files is assumed. After storing the operation in the "DMC" file, the operation parameters may be fetched back into a toolpath file at any point. This is useful for moving operations around in a toolpath file or quickly loading in operations which are frequently done.

KEEP - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the existing parameters and toolpath of the operation will be passed to the output file. Even if changes were made to the parameters, the preexisting values will be passed along except for the

GRAFX II User Reference Manual Supplement

following which can be changed:

Tool number, Tool length offset number, Tool diameter offset number, Fixture offset number

DELETE - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the current operation will be removed from the toolpath file.

Add to note at bottom of Page 3-123

* Entering `i' or `I' will insert an operation before the next preexisting operation in the input file. Entering `a' or `A' will append a new operation to the end of the file. Entering a `g' or `G' will prompt for the name of a "DMC" file whose parameters are to be inserted. The `F5' key or LEFT MOUSE button may be pressed to bring up a menu of existing "DMC" files to select from.

On CONTOUR and ZIG ZAG POCKETING operations the following popup menus appear immediately after selecting the boundaries to be machined:

CUT SIDE menu

LEFT - The cutter travels on the left side of the boundary.

RIGHT - The cutter travels on the right side of the boundary.

OVER - The center of the cutter travels over the boundary.

INSIDE - The cutter travels on the inside of the boundary.

OUTSIDE - The cutter travels on the outside of the boundary.

CUT DIRECTION menu

FORWARD - Cut in the direction the boundary is defined.

REVERSE - Cut in the reverse direction the boundary is defined.

CLMB - Maintain a climb cut around the boundary.

CONV - Maintain a conventional cut around the boundary.

The **CLMB** and **CONV** selections of the CUT DIRECTION menu are not

GRAFX II User Reference Manual Supplement

available if **OVER** is selected in the CUT SIDE menu or if the spindle has not been programmed to come on in the operation.

The **CUT SIDE** and **DIRECTION** selections made from the popup menus are retained in the toolpath file. When modifying existing operations, these menus will come up with the retained selection highlighted. Therefore, the same selection may easily be made by simply pressing the <ENTER> key or the RIGHT mouse button. Note however, that if more than one pick is made for any single operation, only the first side and direction selections for the operation are recalled.

Non Turning Parameters:

Tool number:

Add: Entering an asterisk (*) before the tool number will force the output of a tool change sequence even if the tool number is the same as the prior operation.

Machining operation: **Contour Drill(type) Pocket(type) Pt to pt**

 If DRILL or POCKET is selected, a popup menu appears to select the type of operation. The selected type is then displayed in parentheses after the operation name.

Active work view:

 Select the active work view to be used with this machining operation. To select from a list of existing work views, press the `F5' key or CLICK the LEFT MOUSE button. When a work view is made active by entering a selection, certain parameters are automatically set from parameters stored in the work view. The following table describes this action:

<u>Machining Parameter</u> <u>Work View Parameter</u>	
Rotate index table to..	Index angle
Work offset number	Work view offset number
Active layer	Layers (current)

GRAFX II User Reference Manual Supplement

Work offset number:

Enter an index value from 0-62. This parameter is used to associate a machining operation with a work offset, usually for the purpose of using work coordinates in Post Processing. When a non zero value is entered, a popup menu queries whether or not to shift the output data by the work view local origin when outputting work coordinate blocks in the Post Processor.

X tool change position:

Add: Entering `h' or `H' uses the home position X value from the initial parameter screen.

Y tool change position:

Add: Entering `h' or `H' uses the home position Y value from the initial parameter screen.

Z tool change position:

Add: Entering `h' or `H' uses the home position Z value from the initial parameter screen.

Spindle Speed (RPM):

Feedrate (IPM/MMPM):

Add: The RPM and IPM/MMPM values can be derived using the Speed and Feed calculator by pressing `F5' key or the LEFT MOUSE button. If this is done, a secondary parameter screen appears with the following:

Cutting speed (FPM/MPM):

The cutting speed from which the RPM value is derived.

Feed per flute:

This parameter appears on all but drilling operations. It is the distance the cutter moves while a flute or tooth makes one pass

through the material.

Number of flutes:

This parameter appears on all but drilling operations. It is the number of flutes or teeth on the cutter.

Speed calculation OD:

This parameter appears on all turning operations except drilling. It defines the tool "X" position which will be used to calculate the spindle RPM. When entering this value, the 'F5' key or LEFT MOUSE button may be used to invoke the point menu in order to define the value.

Effective tool diameter at "Depth of cut":

This parameter appears on all non-turning operations as well as drilling operations in turning. It defines the diameter of the tool at the surface of the part material. This is usually equal to the tool or drill diameter but may be another value when countersinking or using tapered cutters. In the case of countersinking, this parameter will usually equal the *chamfer diameter* parameter in the drill depth calculator. In corner cutting, it will usually equal the operation's *Effective tool diameter* parameter. When first creating a new countersinking or corner cutting operation, you can define or calculate those parameters and then go back to the speed and feed calculator and the *Effective tool diameter* parameter will be carried over as the default value.

Feedrate (IPR/MMPR):

This parameter appears on all drilling operations except tapping. It defines the distance the drill moves for each revolution of the spindle.

Threads per inch/Thread lead:

This parameter appears on tapping operations only. It defines the thread parameter for the tapped hole.

After entering the calculator parameters, the prior parameter screen will be restored and the RPM and IPM/MPM parameters will be calculated.

GRAFX II User Reference Manual Supplement

If the *tool diameter* parameter is zero, the RPM parameter will not be calculated. In milling operations, if the *number of flutes* parameter is 0, the IPM/MMPM parameter will not be calculated. If you exit the calculator with the **ESC** key, neither parameter will be calculated. The calculated RPM value may be manually modified by simply reentering a value. After doing this, you can recalculate the IPM/MMPM based on the new RPM by pressing the `F5' key or LEFT MOUSE button while in the IPM/MPM parameter.

Coolant:

Add: **On through tool** toggle selection

Tool display orientation: **0 90 180 270 (mirror state)**

Select transformation angle and mirror for display of tool geometry such as drills for non-turning and turning applications.

"Z" depth per pass:

...However, if a "FULL RT" attribute (see Page 32) is attached to any element of the boundary, the tool will be raised to the last "Z" position prior to the first "Z" approach move instead.

Change all:

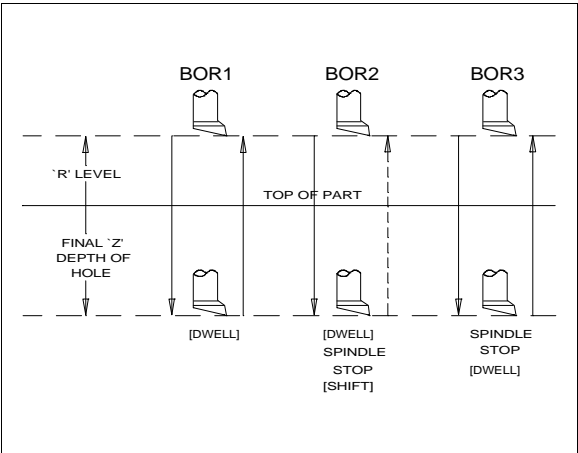
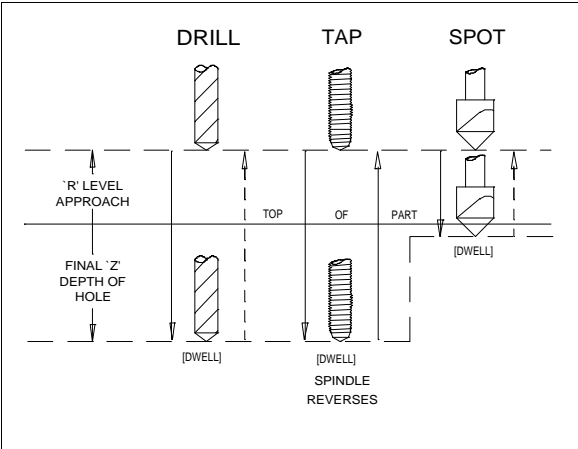
Rotary axis angle:

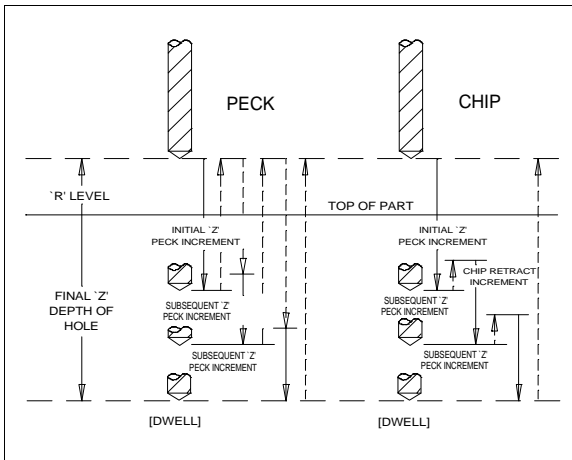
To:

Rotate index table to (angle in degrees):

Drill:

Drilling cycles diagrams are modified to reflect changes in cycles:





Delete the "Drilling operation" parameter from the DRILLING PARAMETERS screen. The popup menu to select the type of drilling operation appears automatically after selecting Drill to the "Machining operation" parameter in the first operations screen.

Change:

Final "Z" depth of hole:

To:

Final "Z" depth of drill stroke:

..A null value may be entered for this parameter in which case the previous "Z" depth value will be used. The previous value is defined as the last DEPTH defined to a point in the point set being drilled. This method is used when it is most convenient to set modal drill depths in the point set rather than in the drilling parameters.

When in this parameter, the drill stroke "Z" depth may be derived by pressing the `F5' key or LEFT MOUSE button to bring up the depth calculator. When this is done, a secondary parameter screen appears with the following:

Mode: **Depth Thru hole Chamfer**

DEPTH will calculate the depth from the "Z increment". "THRU HOLE" will calculate the depth required to clear the "Thru hole depth" based on the

GRAFX II User Reference Manual Supplement

Drill point angle of the drill and the "Thru clearance". "CHAMFER" will calculate the depth required to produce the specified "Chamfer diameter" using the "Drill point angle" or "Chamfer angle". (See diagram on Page 69)

"Z" level for drill plane:

Drilling depth will be calculated relative to this value

"Z" increment from drill plane:

This parameter appears in DEPTH mode only. It defines the final "Z" distance relative to the drill plane surface. The direction is always assumed to be into the work no matter which sign is entered.

Drill point angle:

This parameter which appears in THRU HOLE and CHAMFER modes only defines the angle of the drill point. This may be automatically loaded from the tool parameters if a tooling file is used. Otherwise a default value of 118 degrees is supplied on new operations. This value must be between 0 and 180 degrees for Thru holes and between 1 and 179 degrees for chamfers.

Thru hole depth:

This parameter appears in THRU HOLE mode only. It defines the depth of material to be cleared on thru holes. The direction is always assumed to be into the work no matter which sign is entered.

Thru clearance:

This parameter appears in THRU HOLE mode only. It defines the depth of material to be cleared on thru holes. The direction is always assumed to be into the work no matter which sign is entered.

Center Drill body diameter:

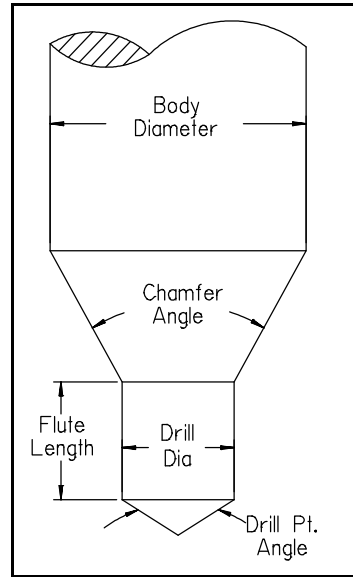
This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the shank diameter of the center drill (see diagram).

Center Drill flute length:

This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the length of the *drill diameter* section of the center drill (see diagram).

Center Drill chamfer angle:

This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the inclusive angle of the chamfer between the *drill diameter* section and the *body diameter* section (see diagram). This parameter must be between 1 and 179 degrees in order to create chamfers.



Center Drill Parameters

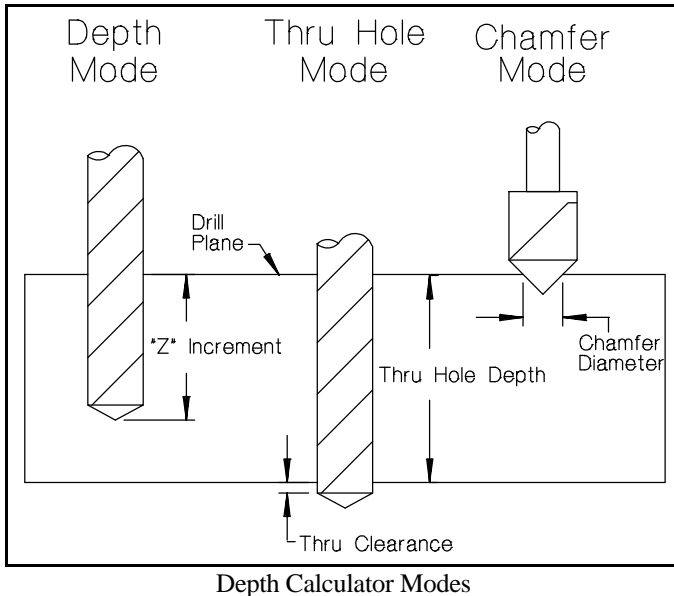
Chamfer diameter:

This parameter which appears in CHAMFER mode only defines the diameter of the chamfer to be created. The drill depth will be calculated based on the drill point angle of the drill. If the specified diameter exceeds the shank diameter of the tool, a warning message will be displayed. This parameter also appears in DEPTH mode for display purposes only. i.e. the value cannot be changed and is displayed in gray. When plotting tool circles in GRAFX2 and PLOT, the chamfer diameter will be plotted instead of the tool diameter.

In the case of center drills, chamfer diameters between 0 and the body diameter of the center drill can be calculated. Calculated depths will automatically be shifted to the chamfer angle section of the center drill when the chamfer diameter exceeds the *drill diameter* value.

GRAFX II User Reference Manual Supplement

After the *Final "Z" depth* value is entered, either by calculation or manually, the *Chamfer diameter* is recalculated based on the value of the *Final "Z" depth* parameter. This means that after using the calculator to determine a depth required to achieve a chamfer diameter and the depth is then changed, GRAFX2 will then recalculate the chamfer diameter. You can always check the chamfer value which will be plotted by bringing up the depth calculator and reading the *Chamfer diameter* value.



Add to DRILL parameters:

Direction to machine point set: **Forward Reverse Query**

Select the direction in which the point set is to be drilled. If **FORWARD** is selected, the path taken will be in the same direction as that which the point set was defined whereas **REVERSE** will take the reverse direction. If **QUERY** is selected, a popup menu will appear after selecting the point set at which time the forward or reverse direction may be specified.

Bore cycle shift distance:

GRAFX II User Reference Manual Supplement

Enter the distance that the tool shifts away from the wall of the bore before retracting in the "Z". This parameter corresponds to what is sometimes referred to as the *Fine Boring Cycle* in some machines which is used to prevent scoring of the bore wall on retraction of the boring bar. Post Processors setup to support this cycle will output this parameter.

Add the new set of parameters:

DRILL (macro):

In addition to the canned drill cycles which are available, users may create their own drill cycles by writing a macro and selecting the DRILL (MACRO) operation. When this is specified, the MACRO DRILLING CYCLE PARAMETER screen will appear.

Macro file name:

Select the macro file to be used in the operation. The 'F5' key or LEFT MOUSE button may be used to menu select the available files.

Final "Z" depth of drill stroke:

This value is symbolically referenced when writing a drilling macro.

"R" level approach:

This value is symbolically referenced when writing a drilling macro.

Direction to machine point set: **Forward Reverse Query**

Operates in the same way as with the canned drilling cycles (see above).

Writing a drilling macro:

A drilling macro is a file containing a set of text commands describing the elements of the drilling cycle being created. To create the macro, you would enter text with a text editor into a file with a .mac extension. This may be done in GRAFX II by selecting EDITOR off the ROOT menu and specifying a file name

GRAFX II User Reference Manual Supplement

of the form *fname*.MAC. The following commands may be used in creating a macro:

ASCII <i>text</i>	Output the <i>text</i> as is without a sequence number <i>N</i>
CYCLE <i>text</i>	Output the <i>text</i> as is with a sequence number <i>N</i>
Xxxx	Make an incremental move of <i>xxx</i> in the X axis
Yyyy	Make an incremental move of <i>yyy</i> in the Y axis
Zzzz	Make an incremental move of <i>zzz</i> in the Z axis
DEPTH	Move in Z to the <i>Depth of cut</i> parameter value
RLEVEL	Move in Z to the <i>R level</i> parameter value
DWELL <i>n</i>	Program dwell for <i>n</i> seconds
FEED <i>n</i>	Change the feedrate to <i>n</i> ; 0 results in rapid
SPDL OFF	Turn the spindle off
SPDL CW	Turn the spindle on in the clockwise direction
SPDL CCW	Turn the spindle on in the counter clockwise direction
COOL OFF	Turn the coolant off
COOL FLOOD	Turn the flood coolant on (M8)
COOL MIST	Turn the mist coolant on (M7)
COOL THRU	Turn the "through the tool" coolant on
GOUT <i>n</i>	Output a <i>GOUT</i> control code block
MOUT <i>n</i>	Output an <i>MOUT</i> machine code block

GRAFX II User Reference Manual Supplement

Notes on macros:

X, Y & Z moves may be combined on a single line. Commands may be in lower case. Comments may be programmed with a \$ (dollar sign) causing the macro interpreter to ignore all text after the \$ unless they appear on a line starting with the *ASCII* or *CYCLE* command. The @ (at sign) may be used to identify the macro file when installing the macro in the MACLIST.TXT menu file.

Following is an example of a simple drill macro which might be used for a fine bore cycle where the "R" level=.1 and depth of cut=-.1:

MACRO COMMAND	DESCRIPTION
@Fine bore cycle macro	Macro identification comment
FEED 2	change feedrate to 2
Z-.5	feed down from .1 to -.4
DWELL 1	dwel 1 second
DEPTH	feed down from -.4 to depth (-.1)
SPDL OFF	turn the spindle off
CYCLE M19	orient the spindle
X.020	move off the wall .020
FEED 0	rapid feed
RLEVEL	bring tool back to "R" level (.1)
SPDL CW	turn spindle back on CW

Pocket:

When **Pocket** is selected from the "*Machining operation*" parameter query, another popup menu now appears with **Zig Zag** and **Spiral**. **Zig Zag** pocketing invokes the former "Pocket" parameter screen and spiral pocketing invokes the "Spiral pocket" parameter screens.

Zig Zag pocket:

Depth of cut:

"Z" approach clearance plane:

"Z" retract clearance plane:

"Z" clearance plane between plunges:

"Z" level for top of part:

"Z" depth per pass:

"Z" axis in feedrate:

1st pass feedrate:

GRAFX II User Reference Manual Supplement

Uniform stock left:

Cutter comp: **Right Left None**

Generate profile with stock circles? **No On Sharp Corners On All Corners**

Avoid undercutting profile? **No Yes**

Roughing pass cutting angle:

Stepover factor (fraction of tool dia., .8 e.g.):

Roughing cut style: **Stay down Always jump**

Generate a finish pass around the profiles? **Yes No Islands only**

"Z" surface projection: **None Cylinder Plane Cone Sphere**

Pocket Finish Pass (Optional)

Stock left on the finish pass:

Finish pass feedrate (IPM or mm/min):

Finish pass spindle speed (RPM):

Corner cut profiles with taper tool? **No Yes**

Zig Zag pocketing now incorporates the following additions or changes to the parameters:

Parameter descriptions (additions):

"Z" clearance plane between plunges:

Enter the retract height from the "Z" zero surface of the part to which the tool retracts if it must pick up and move to another area of the pocket. This value defaults to the "Z" retract clearance plane but allows for a secondary retract plane within the pocket.

1st pass feedrate:

Enter the feedrate at which the tool cuts from the initial plunge into the pocket to the end of the first roughing pass. This feedrate is used to cut the portion of the pocket where the tool is cutting through material on both sides and where a slower feedrate may be desired. All other XY cutting will be done at the feedrate entered in the first parameter screen.

Roughing cut style: **Stay down Always jump**

Select the style of roughing passes in respect to the "Z" motion of the tool when it encounters islands or convolutions. The "Stay down" selection causes the tool to take the longest continuous path inside the pocket before retracting and moving to another area of the pocket. The "Always jump" selection causes the tool to always rough completely across the pocket from end to end and jump over the islands and convolutions. The "Always jump" style was always used in the prior versions of Zig Zag pocketing.

When the "Stay down" style is selected, if the tool must move to another area of the pocket, wherever possible it will plunge into a point where the material has previously been removed by the pocketing operation.

Parameter descriptions (changes):

Change:

Stepover distance between passes:

To:

Stepover factor (fraction of tool dia., .8 e.g.):

Enter the fraction of the current tool diameter to be used to compute the stepover distance between roughing passes. This value must be greater than 0 and is typically less than 1.

Zig Zag pocketing formerly input the absolute distance of the stepover.

Change:

Direction of pocketing passes: **Parallel X (Y) axis**

To:

Roughing pass cutting angle:

Enter the angle of the rough passes as measured from the horizontal. Any angle may be entered. For example, 0 degrees produces rough cuts that are parallel to the X axis. (Entering 10 degrees would turn them slightly counter-clockwise.) Entering 90 degrees produces vertical toolpath.

An angle greater than 180 degrees reverses the usual start position of the toolpath. For example, 30 degrees starts the first rough pass at the top

GRAFX II User Reference Manual Supplement

left while $180+30 (=210)$ starts down and to the right. The rough cuts will be at 30 degrees from the horizontal in both cases.

Change:

Automatically generate a pass around the profiles?

To:

Generate a finish pass around the profiles?

This parameter has been changed in wording only to be consistent with the "Pocket Finish Pass" parameter screen. This finish pass is done with the same tool as the roughing passes and may be utilized to smooth out the scallops generated by Zig Zag pocketing or finish to size. Finishing may also be accomplished by changing to another tool and performing contour operations around the profiles.

Pocket Finish Pass (Optional) - This parameter screen only appears if a response other than *No* is given to the *"Generate a finish pass around the profiles?"* parameter.

Stock left on finish pass:

This parameter has been moved from the "Zig Zag Pocketing" parameter screen to the "Pocket Finish Pass" screen and has been reworded to indicate "finish" instead of "profile".

Finish pass feedrate (IPM or mm/min):

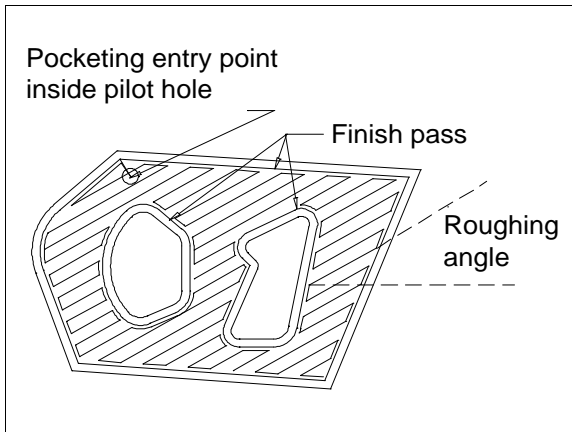
Enter the feedrate used to cut the profile on the finish pass.

Finish pass spindle speed (RPM):

Enter the spindle speed used to cut the profile on the finish pass.

Corner cut profiles with taper tool?

This parameter has been moved from the "Zig Zag Pocketing" parameter screen to the "Pocket Finish Pass" screen.



Zig Zag Pocket with Islands

Zig Zag Pocketing Notes:

The approach to the start point of the pocket is controlled by the presence of positioning points in the boundary. If no positioning points are defined, the tool steps to the start point and plunges straight down into the pocket before starting the roughing passes. If a positioning point is defined, the tool positions in "XY" over the positioning point and plunges straight down into the pocket, then cuts at feedrate to the pocket start point before starting the roughing passes.

Spiral Pocket:

Depth of cut:

"Z" approach clearance plane:

"Z" retract clearance plane:

"Z" level for top of part:

"Z" depth per pass:

"Z" axis in feedrate:

1st pass / side feedrate:

Uniform stock left:

Cutter comp: **Right Left None**

Stepover factor (fraction of tool dia., .65 e.g.):

Innermost stepover factor (.5 e.g.):

Cut direction of roughing passes: **Forward Reverse Clmb Conv**

Position of last roughing pass: **Inside Over**

Generate a finish pass around the profiles? **No Yes**

"Z" surface projection: **None Cylinder Plane Cone Sphere**

Spiral Pocket Finish Pass (Optional)

Generate profile with stock circles? **No On Sharp Corners On All Corners**

Avoid undercutting profile? **No Yes**

Stock left on the finish pass:

Cut direction of finish pass: **Forward Reverse Clmb Conv**

Transition to finish pass: **Jump Stay down**

Transition feedrate:

Finish pass feedrate:

Finish pass spindle speed (RPM):

Corner cut profiles with taper tool? **No Yes**

Parameter descriptions:

Depth of cut:

Enter the absolute "Z" depth of cut from the "Z" zero surface of the part.
This is the final "Z" depth to be cut.

"Z" approach clearance plane:

Enter the clearance height from the "Z" zero surface. The tool positions to this height prior to cutting each pocket unless an approach positioning point was used in which case the tool will position to the "Z" height of the positioning point.

"Z" retract clearance plane:

Enter the retract height from the "Z" zero surface of the part. Upon completion of the pocketing sequence, the tool is retracted to this height.

"Z" level for top of part:

Enter the value from the "Z" zero surface to the top of the part. This value is used in conjunction with the previous parameters to determine the proper "Z" display for verification with PLOT.

GRAFX II User Reference Manual Supplement

"Z" depth per pass:

Enter the incremental depth per pass to generate multiple passes at different depths. The tool automatically steps down by this amount and repeats the cut until the final depth is reached. On each pass the tool is raised to a clearance level which starts at the "Z" approach clearance plane and then floats downward with each pass by the depth per pass increment.

"Z" axis in feedrate:

Enter the feedrate at which the tool feeds down from the "Z" clearance level or the current level.

1st pass / side feedrate:

Enter the feedrate at which the tool cuts from the start of spiral until the end of the first pass. This feedrate is also used to cut the connecting element between passes. All other "XY" cutting will be done at the feedrate entered in the first parameter screen.

Uniform stock left:

Enter the amount of material to be left on the side walls of the boundary for a subsequent finish pass.

Cutter comp: **Right Left None**

Select the cutter compensation mode to be used at the machine tool. (Typically G41 or G42 is output to the G-code file depending on the post processor).

Stepover factor (fraction of tool dia., .65 e.g.):

Enter a value that is greater than 0 and less than or equal to 1 representing the percentage of the tool diameter used to compute the distance between passes.

GRAFX II User Reference Manual Supplement

Innermost stepover factor (.5 e.g.):

Enter the percentage of the tool diameter use to compute the stepover distance of the first pass. This is typically less than the general stepover factor.

Cut direction of roughing passes: **Forward Reverse Clmb Conv**

Enter the direction in which the roughing passes will be cut. Forward cuts in the same direction as the part boundary is defined while reverse cuts in the opposite direction. Typically in spiral pocketing, it is good machining practice to climb cut the roughing passes while conventional cutting the finish pass.

Position of last roughing pass: **Inside Over**

Select the position of the cutter relative to the part boundary when it makes the last roughing pass.

Generate a finish pass around the profiles? **No Yes**

Specify whether or not to take a finish pass around the part profile with the same tool upon completion of the spiral roughing passes. Selecting YES brings up another parameter screen for this operation allowing new machining parameters to be entered.

"Z" surface projection: **None Cylinder Plane Cone Sphere**

See Appendix C, "Z Surface Projections", for information concerning toolpath projection.

Spiral Pocket Finish Pass (Optional) - This parameter screen only appears if **Yes** is selected to the *"Generate a finish pass around the profiles?"* parameter.

Generate profile with stock circles? **No On Sharp Corners On All Corners**

Select the mode to be used when contouring around outside corners formed by the intersection between two elements.

GRAFX II User Reference Manual Supplement

A stock circle is defined as a circular cut around the outside corner formed by two intersecting elements of a boundary with the radius equal to the radius of the tool being used. A zero radius (sharp) corner is always created on the part.

Stock circles are normally used to prevent wasted tool motion by cutting air on outside corners.

Avoid undercutting profile? **No Yes**

See Appendix B, "Undercut Avoidance", for more information.

Stock left on the finish pass:

Enter the amount of material to be left on the side walls of the boundary for a subsequent finish pass.

Cut direction of finish pass: **Forward Reverse Clmb Conv**

Enter the direction in which this will be cut. Forward cuts in the same direction as the part boundary is defined while reverse cuts in the opposite direction.

Transition to finish pass: **Jump Stay down**

Specify how the tool moves from the end of the roughing passes to the start of the finish pass.

If **Jump** is selected, the tool positions upward to the "Z" retract plane, then positions to the XY start position of the finish pass and finally positions to the "Z" approach plane before feeding to "Z" depth and cutting around the profile.

If **Stay down** is selected, no "Z" motion is made as the tool transverses to the start of the finish pass at the feedrate specified by the next parameter. Lead-in and lead-out elements will be cut while approach positioning points are ignored.

GRAFX II User Reference Manual Supplement

Transition feedrate:

Enter the feedrate used to move from the end position of the roughing passes to the start position of the finish pass if **Stay down** was specified. A value of 0 results in a rapid traverse move. If **Jump** was specified, this parameter is ignored.

Finish pass feedrate:

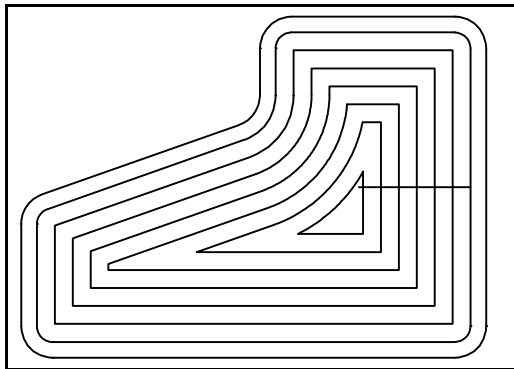
Enter the feedrate used to cut the profile on this pass.

Finish pass spindle speed (RPM):

Enter the spindle speed used to cut the profile on this pass.

Corner cut profiles with taper tool? **No Yes**

See ENGRAVE - Corner Cut, *Page 3-114*, for more information.



Spiral Pocket

Spiral Pocketing Notes:

Boundaries which are to be spiral pocketed must have at least three elements. For example, if a circle is to be spiral pocketed, it must be broken up into 3 arcs.

The boundaries must be closed, that is the start and end points must be the same.

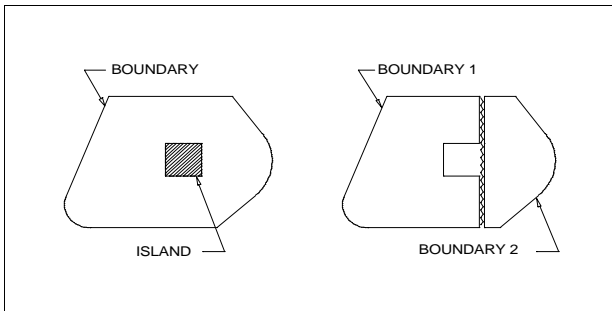
It does not matter where you start a boundary. Spiral pocketing always finds its own start point when generating the roughing toolpath.

GRAFX II User Reference Manual Supplement

The approach to the start point of the pocket is controlled by the presence of positioning points in the boundary. If no positioning points are defined the tool steps to the start point. A flat rapid move is made to over the spiral start center point. Then a vertical move at rapid is made to the approach clearance plane followed by a vertical plunge to depth at the "Z" axis in feedrate.

If a positioning point is defined, there is a ramp rapid approach to the 3 axis position of the point followed by a ramp move to depth at the "Z" axis in feedrate. If more than one positioning point is used, rapid ramp motion will be made to each.

Spiral pocketing does not avoid islands inside the pocket. If island avoidance is required, simply break the pocket up into separate boundaries thereby removing the islands as illustrated below. You may also opt to use Zig Zag pocketing instead which supports complete nested island avoidance.



In spiral pocket processing, the optimum center is first computed. The advantage to this is that the most efficient toolpath is generated, however this process can be time consuming on slower computers. While the center is being searched for, a graphic showing the current best candidate is displayed and an early acceptance for the center may be made by pressing the **F10** key. The optimum center would be the center of the largest circle which fits completely within the boundary and by viewing the graphic, one can often visualize when this solution has been reached before all possibilities have been exhausted. There may be a time delay after pressing the **F10** but multiple keystrokes should be avoided since they will cancel subsequent toolpath generation.

Pt to Pt:

Depth of cut:

"Z" approach clearance plane:

"Z" retract clearance plane:

"Z" level for top of part:

"Z" depth per pass:

"Z" axis in feedrate:

Direction to machine point set: **Forward Reverse Query**

Parameter descriptions:

Depth of cut:

Enter a value which will be added to the "Z" values of the points to determine the "Z" position of the tool as it moves from point to point.

"Z" approach clearance plane:

Enter the "Z" clearance height from the "Z" zero surface of the part. The tool is positioned to this height prior to cutting each point set. This is also the "Z" level at which the tool is rapid traversed between point sets or positioned to when performing multiple passes (point set selected more than once) on the same point set.

"Z" retract clearance plane:

Enter the "Z" retract height from the "Z" zero surface of the part. Upon completion of the point to point machining, the tool is retracted to this height. When multiple passes ("Z" depth per pass is smaller than "Z" depth of cut) are being performed, the tool is not retracted to this height until completion of the final pass.

"Z" level for top of part:

Enter the "Z" value for the top of the part from the "Z" zero surface. This value is used in conjunction with the previous parameters to determine the proper "Z" display when verifying the toolpath with PLOT.

"Z" depth per pass:

Enter the incremental depth per pass for multiple passes at different depths. The tool automatically steps down by this amount and repeats the point set machining until the final depth is reached. The approach and retract motions are not repeated if the toolpath which machines the point set forms a closed loop. If the toolpath does not form a loop, the approach and retract sequences are performed on each pass; in between passes, the tool is raised to the "Z approach clearance plane" prior to positioning for the next pass.

"Z" axis in feedrate:

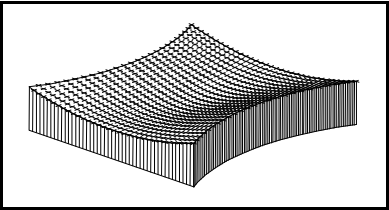
Enter the feedrate at which the tool feeds down from the "Z" clearance level or the current "Z" level if multiple passes are being performed. If a "retract positioning point" is used, the "Z axis in feedrate" is used as the "Z" retract feedrate of the positioning point (see OBJECTS > SET > POSITION, Page 3-69, for more information on positioning points).

Direction to machine point set: **Forward Reverse Query**

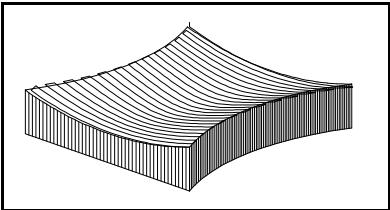
Select the direction in which the point set is to be machined. If FORWARD is selected, the path taken will be in the same direction as that which the point set was defined whereas REVERSE will take the reverse direction. If QUERY is selected, a popup menu will appear after selecting the point set at which time the forward or reverse direction may be specified.

Pt To Pt Machining notes:

In point to point machining, the center of the tool travels in linear segments through a point set. If the points contained in the point set have "Z" axis positions associated with them, the tool will travel to their positions in three space. The "Z" depth parameter is added to the defined "Z" values of the points for the "Z" position of the tool. Performing point to point machining on 3 dimensional point data, which may be IMPORTED from several CAD formats, provides a means of generating 3 axis surface toolpath.



3-D Point Mesh



Pt to Pt Toolpath

GRAFX II User Reference Manual Supplement

Reference - NC > CUT PART > TURN

Add to Page 3-145 between paragraphs 2 and 3

If the previous tool number is entered to the tool number prompt, the next operation of that tool is opened for toolpath creation or modification in the case of an existing toolpath file.

Modify paragraph 2 on Page 3-145

(the initial default value is tool number 1 and it is automatically bumped up by 1 each time you select PARAMETERS except if a next toolpath operation already exists in the file in which case the tool number of that operation will be displayed in the prompt).

If toolpath operations already exist, by entering the 'F5' key or the LEFT MOUSE button to the tool number prompt, a menu of the existing operations in the toolpath file is displayed allowing the user to immediately access any operation by selecting it off the menu. The menu selections are of the form *Tn/o Description*, where *n* is the tool number, *o* is the offset number and *description* is the operation description as entered in the parameters or the generic name of the operation if none was entered. *Draw Part* operations are not displayed with a tool and offset number. The last operation passed to the output file is displayed in red.

If a *Draw Part* operation is selected from the menu of operations, the boundary corresponding to the selected Draw Part operation is highlighted and the following dialog menu appears:

FOUND PART DRAW - WHAT DO YOU WANT TO DO?

- | | |
|---------------|--|
| REDO | - Reselect the part boundary and reenter the top and bottom surface "Z" levels for the Draw Part operation. |
| DELETE | - Remove the Draw Part operation from the toolpath file. |
| KEEP | - Pass through the Draw Part operation to the output file. |
| FILE | - Move the current Draw Part operation to a file for later retrieval. This operation initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name in which to store the operation and appends a "DMC" extension to it. The default path for GEO and DMB files is assumed. After storing the operation in the |

GRAFX II User Reference Manual Supplement

"DMC" file, the Draw Part operation may be fetched back into a toolpath file at any point.

CANCEL - Do not take any action.

When DELETING the first operation of a tool or offset, or inserting a new operation before the first operation of a tool or offset, the next sequential operation should be re-executed in order to create or delete a tool change as required. A tool change is automatically supplied when executing an operation where the tool number or offset number differs from the previous operation. When bringing up the menu of operations, any inconsistencies in missing or redundant tool changes are noted by a lower case *t* in the operation which should be re-executed.

Modify paragraph 4 on Page 3-145

After completing the parameters the following dialog menu appears:

EXECUTE THIS OPERATION?

YES - Proceed with the operation. The cross hairs come up and you are prompted to select the boundary(s) to be machined or the line(s) to be drilled or threaded.

NO - Do not proceed with the operation.

BACK - Go back to the last displayed parameter screen.

FILE - Move the current operation parameters to a file for later retrieval. This initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name to store the operation and appends a "DMC" extension to it. The default path for GEO and DMB files is assumed. After storing the operation in the "DMC" file, the operation parameters may be fetched back into a toolpath file at any point. This is useful for moving operations around in a toolpath file or quickly loading in operations which are used frequently.

KEEP - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the existing parameters and toolpath of the operation will be passed to the output file. Even if changes were made to the parameters, the preexisting values will be passed along except for the

GRAFX II User Reference Manual Supplement

following which can be changed:

Tool number, Tool offset number, Tool diameter offset number

DELETE - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the current operation will be removed from the toolpath file.

Add to note at bottom of Page 3-145

- * Entering `i' or `I' will insert an operation before the next preexisting operation in the input file. Entering `a' or `A' will append a new operation to the end of the file. Entering a `g' or `G' will prompt for the name of a "DMC" file whose parameters are to be inserted. The `F5' key or LEFT MOUSE button may be pressed to bring up a menu of existing "DMC" files to select from.

Change the order of the Spindle Speed parameters on Page 3-147 to:

Maximum spindle speed (RPM):

Constant surface speed (FPM/MPM):

Spindle Speed (RPM):

Change the gauge length descriptions on Page 3-150 to reflect the reversal in sign convention:

X gauge length:

Enter the radial distance from the center of the tip nose radius to the turret home position measured parallel to the lathe's "X" axis.

Z gauge length:

Enter the radial distance from the center of the tip nose radius to the turret home position measured parallel to the lathe's "Z" axis.

GRAFX II User Reference Manual Supplement

Add to "Spindle Speed (RPM)" parameter description on Page 3-152:

The RPM value can be derived using the Speed calculator by pressing `F5' key or the LEFT MOUSE button. If this is done, a secondary parameter screen appears with the following:

Cutting speed (FPM/MPM):

The cutting speed from which the RPM value is derived. This value will default to the "Constant surface speed" parameter value.

Speed calculation OD:

This parameter appears on all but drilling operations. The value represents the tool "X" position which will be used to calculate the RPM. When inputting this value, the `F5' key or LEFT MOUSE button may be used to invoke the POINT menu in order to define the value.

Feedrate (IPR/MMPR):

This parameter appears on drilling operations only. It defines the distance the drill moves for each revolution of the spindle.

After entering the calculator parameters, the prior parameter screen will be restored and the RPM parameter will be calculated with its value limited by the *Maximum spindle speed* parameter value if non zero. If the *Speed calculation OD* parameter is zero or you exit the calculator with the **ESC** key, the RPM parameter will not be calculated. The calculated RPM value may be manually modified by simply reentering a value.

Change:

Tool length offset number:

To:

Tool offset number:

Change all:

Cutter comp: **Right Left None**

Create toolpath with cutter offset: **Yes No**

GRAFX II User Reference Manual Supplement

To:

Tool nose radius comp: **Right Left None**

Create toolpath with tool nose radius offset: **Yes No**

Tool number:

Add: Entering an asterisk (*) before the tool number will force the output of a tool change sequence even if the tool number is the same as the prior operation.

Active work view:

Select the active work view to be used with this machining operation. To select from a list of existing work views, press the 'F5' key or CLICK the LEFT MOUSE button. When a work view is made active by entering a selection, certain parameters are automatically set from parameters stored in the work view. The following table describes this action:

Machining Parameter Work View Parameter

Work offset number	Work view offset number
Lathe type	Turret
Active layer	Layers (current)

Work offset number:

Enter an index value from 0-62. This parameter is used to associate a machining operation with a work offset, usually for the purpose of using work coordinates in Post Processing. When a non zero value is entered, a popup menu queries whether or not to shift the output data by the work view local origin when outputting work coordinate blocks in the Post Processor.

X tool change position:

Add: Entering 'h' or 'H' uses the home position "X" value from the initial parameter screen.

GRAFX II User Reference Manual Supplement

Z tool change position:

Add: Entering `h' or `H' uses the home position "Z" value from the initial parameter screen.

Tool Point angle (integer degrees):

Change:

Enter the tool point angle...

To:

Select the tool point angle off a menu...

Coolant:

Add: **On through tool** toggle selection

Post Processor option M-code #1:

Post Processor option M-code #2:

Post Processor option M-code #3:

Enter the optional 'M' code values. These code are primarily for custom post processors which require additional data codes, therefore, in most cases, you can skip over these parameters.

Tool display orientation: **0 90 180 270 (mirror state)**

Select transformation angle and mirror for display of tool geometry such as drills for non-turning and turning applications or turning tools used in front or rear lathe applications.

Add to **Roughing parameters:**

Back turn part? **No Yes**

Selecting YES results in the tool making its roughing passes in the reverse direction (usually left to right). This is typically done with a right hand tool

GRAFX II User Reference Manual Supplement

where the back angle of a left hand tool cannot clear the profile of the part.

Delete the words "and striking the ESC key" from the NOTES on Pages 3-155 and 3-161.

Add the following note for ROUGHING and BORE operations:

Positioning points may be used to control approach and retract motion in roughing. When no positioning points are used, the tool will retract to the prior tool change position. Tool retraction is done in a way to avoid collision with the part but it is imperative to verify the toolpath with PLOT before running the part program at the machine.

*Add to **Finishing parameters:***

Part surface clearance plane:

Enter a value by which the part surface will be cleared by the tool on the approach and retract ends of the part boundary. On the finish operation, the boundary is automatically extended by the tool radius plus this value.

Back turn part? **No Yes**

Selecting YES results in the tool tracking the boundary in the reverse direction in which it was defined. This is typically done with a right hand tool where the back angle of a left hand tool cannot clear the profile of the part.

Delete the words "and striking the ESC key" from the NOTE on Page 3-167.

*Add to **Groove:***

Tool nose radius comp: **None Left Right**

Select the cutter compensation mode to use at the machine tool. (Typically G41/G42 is output to the tape depending on the post processor)

Create toolpath with tool nose radius offset? **Yes No**

GRAFX II User Reference Manual Supplement

Select whether or not to offset the tool from the workpiece.

YES - Use the tool nose radius and angle plus stock currently in effect.

NO - Output as though "0" tool nose radius and stock is in effect.

Plunge retract feed factor (0 = rapid): *

Enter a value to determine at what feedrate the tool retracts from inside the groove on roughing operations. The factor is the multiple of the plunge feedrate. For example, if the roughing feedrate is *.012 IPR* and *3* is entered for this parameter, the tool will retract from the groove at *.036 IPR*. Entering *0* will result in the retract as a rapid move.

Finish from both ends to the middle? **Yes No**

Select whether to finish the groove in two steps or in a single unidirectional pass.

Change the first sentence in the last paragraph on Page 3-173 to:

If a two step finish pass is selected;...

*Add to **Thread:***

Direction of thread cutting passes: **Forward Reverse**

The default FORWARD direction cuts towards the headstock or towards the spindle axis on face threads while REVERSE cuts away from the headstock or away from the spindle axis on face threads .

GRAFX II User Reference Manual Supplement

Delete the words "and striking the ESC key" from the NOTE on Page 3-177.

Delete "DRILLING OPERATION" from the DRILL parameters. This is now selected with a popup menu when selecting the DRILL machining operation in the first parameter screen.

Change Drill cycle parameters:

Final "Z" depth of drill stroke:

The drilling location is established from the 'drill line' selected from the graphics display window. If the null response is entered for this parameter, the "Z" depth will be set to the endpoint of the drill line. If a non null response (including 0) is entered, the "Z" depth will be set to that value in which case the 'F5' key or the LEFT MOUSE button may be pressed to bring up the Drill Depth Calculator to derive the value. When this is done, a secondary parameter screen appears with the following:

Mode: **Depth Thru hole Chamfer**

DEPTH will calculate the depth from the "Z increment". "THRU HOLE" will calculate the depth required to clear the "Thru hole depth" based on the Drill point angle of the drill and the "Thru clearance". "CHAMFER" will calculate the depth required to produce the specified "Chamfer diameter" using the "Drill point angle" or "Chamfer angle".

"Z" level for drill plane:

Drilling depth will be calculated relative to this value

"Z" increment from drill plane:

This parameter appears in DEPTH mode only. It defines the final "Z" distance relative to the drill plane surface. The direction is always assumed to be into the work no matter which sign is entered.

Drill point angle:

GRAFX II User Reference Manual Supplement

This parameter which appears in THRU HOLE and CHAMFER modes only defines the angle of the drill point. This may be automatically loaded from the tool parameters if a tooling file is used. Otherwise a default value of 118 degrees is supplied on new operations. This value must be between 0 and 180 degrees for Thru holes and between 1 and 179 degrees for chamfers.

Thru hole depth:

This parameter appears in THRU HOLE mode only. It defines the depth of material to be cleared on thru holes. The direction is always assumed to be into the work no matter which sign is entered.

Thru clearance:

This parameter appears in THRU HOLE mode only. It defines the depth of material to be cleared on thru holes. The direction is always assumed to be into the work no matter which sign is entered.

Body diameter:

This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the shank diameter of the center drill (see diagram on Page 68).

Flute length:

This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the length of the *drill diameter* section of the center drill (see diagram on Page 68).

Chamfer angle:

This parameter appears in CHAMFER mode only and is only applicable to center drills. It defines the inclusive angle of the chamfer between the *drill diameter* section and the *body diameter* section (see diagram on Page 68).

Chamfer diameter:

GRAFX II User Reference Manual Supplement

This parameter which appears in CHAMFER mode only defines the diameter of the chamfer to be created. The drill depth will be calculated based on the drill point angle of the drill. If the specified diameter exceeds the shank diameter of the tool, a warning message will be displayed. This parameter also appears in DEPTH mode for display purposes only. i.e. the value cannot be changed and is displayed in gray.

In the case of center drills, chamfer diameters between 0 and the body diameter of the center drill can be calculated. Calculated depths will automatically be shifted to the chamfer angle section of the center drill when the chamfer diameter exceeds the *drill diameter* value.

After the *Final "Z" depth* value is entered, either by calculation or manually, the *Chamfer diameter* is recalculated based on the value of the *Final "Z" depth* parameter. This means that after using the calculator to determine a depth required to achieve a chamfer diameter and the depth is then changed, GRAFX2 will then recalculate the chamfer diameter. You can always check the chamfer value which will be plotted by bringing up the depth calculator and reading the *Chamfer diameter* value.

Add to Drill cycle parameters:

Threads per inch (chase in):

Enter the "chase in" tapping lead to be used on a TAP cycle. If this value is non-zero, chase tapping at the specified lead using the threading cycle will be output by Post Processors with the *Chase in/out TAP "G" code* parameter set. If this parameter is 0 or if the *Chase* parameter is not set in the Post Processor, POST will output a feed in and out (G01).

Threads per inch (chase out):

Enter the "chase out" tapping lead if it differs from the "chase in" lead. If chase tapping is programmed and this parameter is 0, the "chase in" lead will be used on the "chase out" retracts.

Delete the words "and striking the ESC key" from the second NOTE on Page 3-180.

GRAFX II User Reference Manual Supplement

Add the new set of parameters:

DRILL (macro):

In addition to the canned drill cycles which are available, users may create their own drill cycles by writing a macro and selecting the DRILL (MACRO) operation. When this is specified, the MACRO DRILLING CYCLE PARAMETER screen will appear.

Macro file name:

Select the macro file to be used in the operation. The `F5' key or LEFT MOUSE button may be used to menu select the available files.

Final "Z" depth of drill stroke:

This value is symbolically referenced when writing a drilling macro.

"R" level approach:

This value is symbolically referenced when writing a drilling macro.

Writing a drilling macro:

A drilling macro is a file containing a set of text commands describing the elements of the drilling cycle being created. To create the macro, you would enter text with a text editor into a file with a .mac extension. This may be done in GRAFX II by selecting EDITOR from the ROOT menu and specifying a file name of the form *fname*.MAC. The following commands may be used in creating a macro:

ASCII <i>text</i>	Output the <i>text</i> as is without a sequence number <i>N</i>
CYCLE <i>text</i>	Output the <i>text</i> as is with a sequence number <i>N</i>
Xxxx or Yyyy	Make an incremental move of xxx in the X axis
Zxxx	Make an incremental move of zzz in the Z axis
DEPTH	Move in Z to the <i>Depth of cut</i> parameter value

GRAFX II User Reference Manual Supplement

RLEVEL	Move in Z to the <i>R level</i> parameter value
DWELL <i>n</i>	Program dwell for <i>n</i> seconds
FEED <i>n</i>	Change the feedrate to <i>n</i> ; 0 results in rapid
SPDL OFF	Turn the spindle off
SPDL CW	Turn the spindle on in the clockwise direction
SPDL CCW	Turn the spindle on in the counter clockwise direction
COOL OFF	Turn the coolant off
COOL FLOOD	Turn the flood coolant on (M8)
COOL MIST	Turn the mist coolant on (M7)
COOL THRU	Turn the "through the tool" coolant on
GOUT <i>n</i>	Output a <i>GOUT</i> control code block
MOUT <i>n</i>	Output an <i>MOUT</i> machine code block

Notes on macros:

X & Z moves may be combined on a single line. Commands may be in lower case. Comments may be programmed with a \$ (dollar sign) causing the macro interpreter to ignore all text after the \$ unless they appear on a line starting with the *ASCII* or *CYCLE* command. The @ (at sign) may be used to identify the macro file when installing the macro in the MACLIST.TXT menu file.

GRAFX II User Reference Manual Supplement

Following is an example of a simple drill macro which might be used for a basic drilling cycle with a chip break and bottom dwell where the "R" level=.1 and depth of cut=-2:

MACRO COMMAND	DESCRIPTION
@Drilling cycle macro	Macro identification comment
FEED .01	change feedrate to .010 IPR
Z-1	feed in from .1 to -1
DWELL 1	dwell 1 second to break chip
DEPTH	feed in from -1 to depth (-2)
DWELL 1	dwell 1 second
FEED 0	rapid feed
RLEVEL	bring tool back to "R" level (.1)

Facing:

Type of facing cycle: **Finish only Rough only Rough and Finish**

Thickness from face line to material line:

Part surface clearance plane:

Uniform stock left:

Face cut direction: **Face in Face out**

Face OD (override face line):

Face ID or center (override face line):

Tool nose radius comp: **None Left Right**

Create toolpath with tool nose radius offset? **Yes No**

Maximum depth per pass:

Feed to "Z" depth? **No Yes**

Finish pass spindle speed (RPM): §

Finish pass spindle speed (*¹): §

Finish pass feedrate (*²): §

Finish pass feedrate (*³): §

Stock left on the finish pass: §

Cancel constant surface speed when positioning? **No Yes ***

Dwell in secs. before each facing cut: *

* These parameters are displayed only when **Rough** passes are to be generated

§ These parameters are displayed only when both **Rough** and **Finish** passes are to be generated

*1 Enter the constant surface speed specified as FPM (feet per minute) or, if in metric mode, M/min (meters per minute).

GRAFX II User Reference Manual Supplement

^{*2} Enter the feedrate per minute specified as IPM (inches per minute) or, if in metric mode, mm/min (millimeters per minute).

^{*3} Enter the feedrate per revolution specified as IPR (inches per revolution) or, if in metric mode, mm/rev (millimeters per revolution).

Parameter descriptions:

Type of Facing cycle: **Finish only Rough only Rough and Finish**

Select the type of facing motion to be performed. The options include the ability to rough and finish using the same tool or different tools.

Thickness from face line to material line:

Enter the incremental amount of material to be removed off the face of the part. On face-out operations the tool will clear this amount before feeding into the work.

Part surface clearance plane:

Enter the incremental clearance distance used on all surfaces of the part. This determines where feedrate motion begins after positioning to the part.

Uniform stock left:

Enter the amount of stock to be left on the part face for a finish pass.

Face cut direction: **Face in Face out**

Select the direction of cut on the facing passes. Face in means that the tool starts at the OD and cuts inward towards the spindle axis. Face out cuts from the ID to the OD.

GRAFX II User Reference Manual Supplement

Face OD (override face line):

Enter a value for the OD of the facing operation. This value overrides the endpoint of the drawn face line and is often used if facing the part before the material has been turned down. Instead of entering a value, the 'F5' key or the left mouse button may be pressed, thereby bringing up the POINT menu to define a point whose vertical component will be used to define this value. If no value is entered for this parameter, the top endpoint of the face line will be used. An existing value may be removed by deleting the value displayed in the parameter. Do not enter 0 to eliminate the override value as 0 will be used for the override. This value is measured as the diameter if "diameter" is active in the "Lathe Units" ACCESS parameters otherwise it is measured as the vertical distance from the centerline. If programming a front lathe this value should be negative as defined below the centerline.

Face ID or center (override face line):

Enter a value for the ID of the facing operation. The data entry rules for this parameter is identical to those of the "Face OD" parameter.

Tool nose radius comp: **None Left Right**

Select the cutter compensation mode to use at the machine tool. (Typically G41/G42 is output to the tape depending on the post processor)

Create toolpath with tool nose radius offset? **Yes No**

Select whether or not to offset the tool from the workpiece.

YES - Use the tool nose radius and angle plus stock currently in effect.

NO - Output as though "0" tool nose radius and stock is in effect.

Maximum depth per pass:

Enter the maximum depth per pass to be used on the roughing passes. The actual depth per pass, when cutting, is never greater than this although it can be somewhat smaller. If 0 is entered, 1 pass to full depth will be made.

GRAFX II User Reference Manual Supplement

Feed to "Z" depth? **No Yes**

Select whether or not to feed in to the cutting "Z" depth prior to each facing pass. Typically this will be *No* if the tool clears the material when it positions to each pass.

Finish pass spindle speed (RPM):

Enter the constant spindle speed for the finish pass as RPM (revolutions per minute) when roughing and finishing with the same tool.

Finish pass spindle speed (FPM or M/min):

Enter the constant surface speed per minute for the finish pass when roughing and finishing with the same tool.

Finish pass feedrate (IPM or mm/min):

Enter the feedrate per minute for the finish pass when roughing and finishing with the same tool.

Finish pass feedrate (IPR or mm/rev):

Enter the feedrate per revolution for the finish pass when roughing and finishing with the same tool.

Stock left on the finish pass:

Enter the amount of stock to leave on the face of the part for another finish pass or tool.

Cancel constant surface speed when positioning? **No Yes**

Select whether or not to cancel the active constant surface speed when positioning between roughing passes. If *Yes* is selected, constant RPM will be invoked on the first positioning move after the facing pass with the RPM computed to be equal to the spindle speed when beginning the next cut. Constant surface speed will then be reinstated prior to the next feed move. If constant surface speed was not specified this parameter will have no effect.

Dwell in secs. before each facing cut:

Enter the number of seconds of dwell prior to each rough facing cut when reinstating constant surface speed. The dwell is used to allow the spindle speed to settle if constant surface speed is in effect. A zero value will result in no dwell output. If constant surface speed was not specified or if it is not to be canceled when positioning, this parameter will have no effect.

Facing Usage Notes:

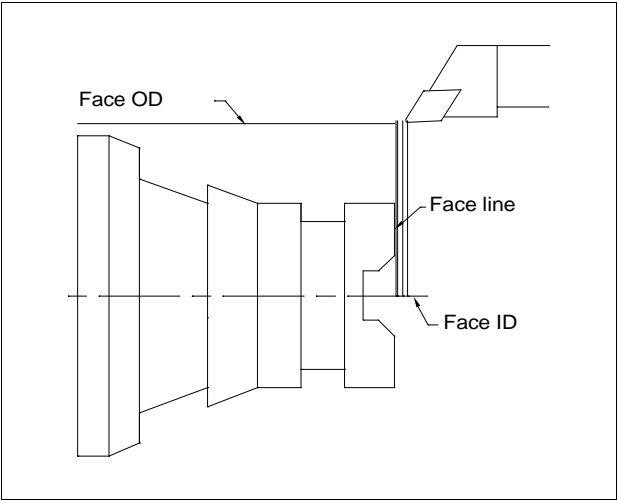
FACE can be used to rough and finish a part face with either different machining parameters or different tools. The part face to be machined is selected as the line entity representing the part face in the part geometry. By entering the amount of stock on the material blank and indicating the OD and ID to face, the facing operation can be performed directly from the part drawing without having to define a separate boundary to be machined. However, if positioning points are required in the approach and/or retract motion, the facing line should be defined within a boundary which has positioning points associated with it.

The facing cuts may be specified as face in (toward the spindle axis) or face out (away from the spindle axis). When facing in, the tool clears the OD and cuts to the ID, then retracts at feed to the clearance value away from the part. When facing out, the tool positions to the ID and cuts past the OD, then retracts the clearance distance. If so specified, on each pass the tool will position to the clearance distance away from the remaining face then feed to depth before making the facing pass. If the ID is exactly at the spindle axis (X0), the center of the tool is brought to the centerline of the part. This presumes that cutting past the centerline may be undesirable. Otherwise, the edge of the tool is brought past the ID by the clearance amount. If cutting past the centerline is desired, simply specify a nonzero ID override, for instance -.001.

GRAFX II User Reference Manual Supplement

Facing lines are presumed to be vertical, however they may be angled somewhat. If the facing line selected is too horizontal, the following message is displayed and the operation is not performed:

Error...face line is too horizontal



GRAFX II User Reference Manual Supplement

Cutoff:

Width of cutoff/grooving tool:

Part surface clearance plane:

"Z" retract clearance increment:

Stock left on back of part:

Cutoff OD (override cutoff line):

Cutoff ID or center (override cutoff line):

Tool nose radius comp: **None Left Right**

Create toolpath with tool nose radius offset: **Yes No**

Plunge increment:

No. of revolutions dwell at end of each plunge:

Back chamfer size:

Back chamfer feedrate:

Retract tool straight up to clearance plane? **Yes No**

Enable part catcher during cutoff? **Yes No**

Advance part catcher diameter:

Dwell in secs. for part drop:

Parameter Descriptions:

Width of cutoff/grooving tool:

Enter the width of the tool. Cutoff operations generally use grooving tools. The width of the tool is measured from edge to edge, this is not the width of the flat on the bottom.

Part surface clearance plane:

Enter the incremental clearance distance used on all surfaces of the part. This determines where feedrate motion begins after positioning to the part.

"Z" retract clearance increment:

Enter the incremental distance the tool will retract away from the stock after the part is cut off. If 0 is entered, this motion will not take place.

Stock left on back of part:

Enter the amount of stock to be left on the back of the part. A positive value entered here will increase the total length of the cutoff part.

Cutoff OD (override cutoff line):

Enter a value for the OD of the cutoff operation. This value overrides the endpoint of the cutoff line to be picked. Instead of entering a value, the `F5' key or the LEFT MOUSE button may be pressed, thereby bringing up the POINT menu to define a point whose vertical component will be used to define this value. If no value is entered for this parameter, the top endpoint of the back line will be used. An existing value may be removed by deleting the value displayed in the parameter. Do not enter 0 to eliminate the override value as 0 will be used for the override. This value is measured as the diameter if "diameter" is active in the "Lathe Units" access parameters otherwise it is measured as the vertical distance from the centerline. If programming a front lathe this value should be negative as defined below the centerline.

Cutoff ID or center (override cutoff line):

Enter a value for the ID of the cutoff operation. The data entry rules for this parameter are identical to those of the "Cutoff OD" parameter.

Tool nose radius comp: **None Left Right**

Select the cutter compensation mode to use at the machine tool. (Typically G41/G42 is output to the tape depending on the post processor)

Create toolpath with tool nose radius offset? **Yes No**

Select whether or not to offset the tool from the workpiece.

YES - Use the tool nose radius and angle plus stock currently in effect.

NO - Output as though "0" tool nose radius and stock is in effect.

Plunge increment:

Enter the incremental depth at which the tool pauses to break the chip. Entering zero (0) or a number larger than the total diameter of the cutoff line

GRAFX II User Reference Manual Supplement

will cause the tool to plunge to full depth in one motion.

No. of revolutions dwell at end of each plunge:

Enter the number of spindle revolutions to pause (dwell) at the completion of each plunge step in order to break the chip.

Back chamfer size:

Enter a non zero value to automatically machine a 45 degree chamfer of this size on the back of the part before cutting off the part.

Back chamfer feedrate:

This parameter appears only if a non zero value is entered to the *Back chamfer size* parameter. Entering a non zero value to this parameter will result in the machining of the chamfer at this feedrate, while the rest of the machining is done at the feedrate specified in the second TURNING PARAMETERS screen. The feed type for this feedrate will always be the same as the overall feedrate (i.e. IPR or IPM).

Retract tool straight up to clearance plane? **No Yes**

Specifying **Yes** will cause the tool to retract straight up to the clearance plane following the last motion of the part cutoff cycle. This provides a safe retract in case the part does not drop (broken tool or misprogrammed ID).

Specifying **No** will cause the tool to take a direct path to the tool change position which would usually travel through the area that the part used to occupy prior to dropping.

Enable part catcher during cutoff? **No Yes**

Selecting **Yes** will bring up the PART CATCHER PARAMETERS screen with the parameters that follow. The system will generate part catcher codes which are then output by Post Processors which support part catchers.

GRAFX II User Reference Manual Supplement

Advance part catcher diameter:

Enter a diameter to which the cutoff tool will cut into the part before the part catcher will be advanced. This allows a delay in advancing the part catcher in order to minimize the amount of chips that will fall into it. If *0* or the *NULL* response is entered or if the diameter is greater than the cutoff *OD* or less than the cutoff *ID*, the part catcher will advance prior to the plunge into the part (but after the back chamfer, if one has been specified).

Dwell in secs. for part drop:

Entering a non zero value will result in a programmed dwell just prior to retracting the part catcher, thereby allowing time for the part to drop into it.

Cutoff Usage Notes:

CUTOFF can be used to cut off the finished part as when machining parts from bar stock. The location to be machined is selected as the line entity representing the back of the part in the part geometry. By indicating the OD and ID to cut, the cutoff operation can be performed directly from the part drawing without having to define a separate boundary to be machined. However, if positioning points are required in the approach and/or retract motion, the back line should be defined within a boundary which has positioning points associated with it. If using positioning points, it should be noted that the left side of the tool will position to the points and that these points should be located so that the entire tool clears any obstructions when positioning.

The machining sequence for the cutoff operation is as follows:

- 1) The tool positions to clear the OD at the cutoff point.
- 2) If a back chamfer has been specified, the tool will:
 - a) Machine a groove by the clearance amount deeper than the chamfer end point, and away from the back face by the clearance amount.
 - b) Retract and machine the chamfer.
 - c) Reposition from just above the point where the chamfer meets

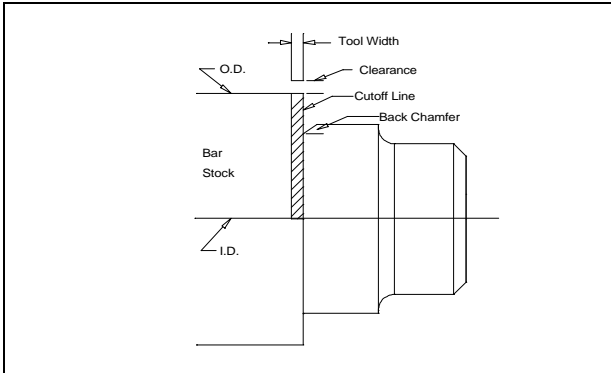
the back face.

- 3) If a part catcher is enabled, it will advance at this point unless deferred by the *Advance part catcher diameter* parameter. The tool then proceeds to plunge towards the center. If a plunge increment has been specified, the tool will travel in by that amount and then pause if a dwell has been specified in the parameters and continue to repeat this sequence until it has reached the ID point. If an *Advance part catcher diameter* has been specified, the tool will travel in to that diameter, advance the part catcher, and then complete the cut to the final ID.

If the ID is exactly at the spindle axis (X0), the center of the tool is brought to the centerline of the part. This presumes that cutting past the centerline may be undesirable. Otherwise, the edge of the tool is brought past the ID by the clearance amount. If cutting past the centerline is desired, simply specify a nonzero ID override, for instance -.001.

The cutoff line is required to be vertical. If the line selected is not vertical, the following message is displayed and the operation is not performed:

Error...Cutoff line is not vertical



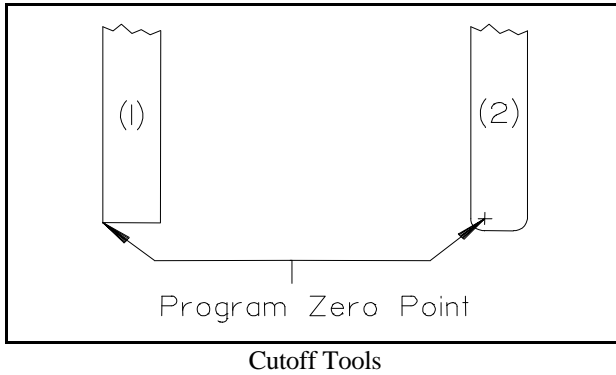
Tooling Notes:

Tooling files for part cutoff tools are the same as vertical grooving tools. Program the zero point of the tool as follows:

- 1) For a square or rectangular tool shape, and no corner radii, use the left hand corner of the tool.
- 2) For a tool shape with corner radii, use the center of the left corner radius of

the tool.

The cutoff tools are defined with their zero points on the left side and they cut at the cutoff line with their right side. The cutoff operation automatically shifts the center of the toolpath from the right to left side to accommodate this. This also allows the setup of the tool to be done by touching the part face.



Copy Turning:

OD or ID cutting: **OD ID**

"X" depth per pass:

"Z" depth per pass:

No. of roughing passes:

Stock left on "X" axis surface:

Stock left on "Z" axis surface:

Back turn part? **No Yes**

Parameter descriptions:

OD or ID cutting: **OD ID**

Select the type of turning operation to be performed.

"X" depth per pass:

Enter the amount shifted toward the spindle axis on each pass. This value may be zero if the "Z" depth per pass parameter is non zero.

"Z" depth per pass:

Enter the amount shifted toward the part face on each pass. This value may be zero if the "X" depth per pass parameter is non zero.

No. of roughing passes:

Enter the number of times the pattern is repeated. Each pass will take an equal amount in each axis direction.

Stock left on "X" axis surface:

Enter the amount of material to be left on the part diameters. This value may be zero.

GRAFX II User Reference Manual Supplement

Stock left on "Z" axis surface:

Enter the amount of material to be left on the part shoulders. This value may be zero.

Back turn part? **No Yes**

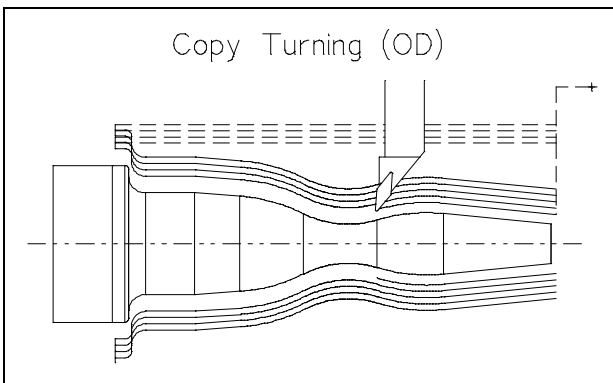
Selecting YES results in the tool making its passes in the reverse direction (usually left to right). This is typically done with a right hand tool where the back angle of a left hand tool cannot clear the profile of the part.

Copy Turning Usage Notes:

COPY TURNING is used to make multiple cuts over the OD or ID contour of a part. It is similar to "tracing" but has additional capability of independently controlling the amount of "X" and "Z" shift on successive passes. The programmer also independently specifies the final "X" and "Z" stock value.

This feature is most useful where the rough material outline of a casting, forging or semi finished part approximates the contour of the finished part and the depth of cut remains fairly constant throughout the cut.

A special advantage of the COPY TURNING cycle is that the start and end "X-Z" clearance positions of the tool remain constant throughout the cycle. Give special consideration to the machining of shoulders and steep tapers so as to not overload the cutting tool during these operations.



Reference - NC > CUT PART > WIRE

Add to Page 3-183 between paragraphs 2 and 3

If the previous tool number is entered to the tool number prompt, the next operation of that tool is opened for toolpath creation or modification in the case of an existing toolpath file.

If toolpath operations already exist, by entering the 'F5' key or the LEFT MOUSE button to the tool number prompt, a menu of the existing operations in the toolpath file is displayed allowing the user to immediately access any operation by selecting it off the menu. The menu selections are of the form *Tn Description*, where *n* is the tool number and *description* is the operation description as entered in the parameters or the generic name of the operation if none was entered. *Draw Part* operations are not displayed with a tool number. The last operation passed to the output file is displayed in red.

If a *Draw Part* operation is selected off the menu of operations, the boundary corresponding to the selected Draw Part operation is highlighted and the following dialog menu appears:

FOUND PART DRAW - WHAT DO YOU WANT TO DO?

- | | |
|---------------|---|
| REDO | - Reselect the part boundary and reenter the top and bottom surface "Z" levels for the Draw Part operation. |
| DELETE | - Remove the Draw Part operation from the toolpath file. |
| KEEP | - Pass through the Draw Part operation to the output file. |
| FILE | - Move the current Draw Part operation to a file for later retrieval. This operation initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name in which to store the operation and appends a "DMC" extension to it. The default path for GEO and DMB files is assumed. After storing the operation in the "DMC" file, the Draw Part operation may be fetched back into a toolpath file at any point. |
| CANCEL | - Do not take any action. |

When DELETING the first operation of a tool or inserting a new operation before the first operation of a tool, the next sequential operation should be re-

GRAFX II User Reference Manual Supplement

executed in order to create or delete a tool change as required. A tool change is automatically supplied when executing an operation where the tool number differs from the previous operation. When bringing up the menu of operations, any inconsistencies in missing or redundant tool changes are noted by a lower case *t* in the operation which should be re-executed.

Modify paragraph 3 on Page 3-183

After completing the parameters the following dialog menu appears:

EXECUTE THIS OPERATION?

- | | |
|---------------|--|
| YES | - Proceed with the operation. The cross hairs come up and you are prompted to select the boundary(s) to be cut. |
| NO | - Do not proceed with the operation. |
| BACK | - Go back to the last displayed parameter screen. |
| FILE | - Move the current operation parameters to a file for later retrieval. This operation initiates what is commonly referred to as a "Cut and Paste". After making the selection, the system asks for the file name in which to store the operation and appends a "DMC" extension to it. The default path for GEO and DMB files is assumed. After storing the operation in the "DMC" file, the operation parameters may be fetched back into a toolpath file at any point. This is useful for moving operations around in a toolpath file or quickly loading in operations which are used frequently. |
| KEEP | - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the existing parameters and toolpath of the operation will be passed to the output file. Even if changes were made to the parameters, the preexisting values will be passed along except for the following which can be changed:

<i>Wire diameter offset number</i> |
| DELETE | - This selection appears only if the operation existed in the toolpath file prior to accessing it. When selected, the current operation will be removed from the toolpath file. |

GRAFX II User Reference Manual Supplement

Add to note at bottom of Page 3-183

- * Entering `i' or `I' will insert an operation before the next preexisting operation in the input file. Entering `a' or `A' will append a new operation to the end of the file. Entering a `g' or `G' will prompt for the name of a "DMC" file whose parameters are to be inserted. The `F5' key or LEFT MOUSE button may be pressed to bring up a menu of existing "DMC" files to select from.

Change all:

Cutter comp: **Right Left None**

To:

Wire comp: **Right Left None**

Add the following parameters:

Tool file name:

Enter the name of the tooling file to be used (the .TLL extension is automatically assumed). To select from a list of tooling files, press the `F5' key or click the LEFT MOUSE button. The associated data, description, diameter, length, etc., is automatically used as the parameter defaults. (see NC > TOOLING, Page 3-174, for more information.) The displaying of the tool geometry can be toggled on and off by pressing the `F8' key.

Wire Description:

Enter text, a description of the wire being used, which is to be output to the G-code file (see GPOST section, the picture file `at' command and REPORT). If a tool file name was entered, the description associated with that file is automatically displayed.

Operation description:

Enter a description of the operation to be performed for output to the machining REPORT.

Active work view:

Select the active work view to be used with this machining operation. To select from a list of existing work views, press the `F5' key or CLICK the LEFT MOUSE button. When a work view is made active by

GRAFX II User Reference Manual Supplement

entering a selection, certain parameters are automatically set from parameters stored in the work view. The following table describes this action:

Machining Parameter Work View Parameter

Work offset number	Work view offset number
Active layer	Layers (current)

Work offset number:

Enter an index value from 0-62. This parameter is used to associate a machining operation with a work offset, usually for the purpose of using work coordinates in Post Processing. When a non zero value is entered, a popup menu queries whether or not to shift the output data by the work view local origin when outputting work coordinate blocks in the Post Processor.

Active layer:

Select the active layer to be used with this machining operation. When an active layer is specified, it is automatically made the current layer for this operation. To select from a list of existing layers, press the 'F5' key or CLICK the LEFT MOUSE button.

Wire length:

Enter the length of the wire for viewing purposes. This value does not affect the program, it is used strictly by the VIEW/PLOT software when displaying the tool motion in other than the "X-Y" view.

Power: **On Off None**

Select the state of the discharge for those machines with programmable power.

GRAFX II User Reference Manual Supplement

Machining conditions code:

Enter a Machining conditions code used by some Post Processors.

Material removed per pass:

A null response results in a single pass around the selected boundaries (the default condition). A positive, non zero value results in multiple progressive passes. A "0" value results in a multiple spring pass operation.

Starting stock:

This parameter appears only if a positive non zero value was entered to the "Material removed per pass" parameter. The first pass will leave the "Starting stock" minus the "..per pass" amount and each subsequent pass will cut deeper by the "..per pass" amount until the "Uniform stock left" amount is reached.

No. of spring passes:

This parameter appears only if zero was entered to "Material removed per pass" parameter. These number of passes will be made at the "Uniform stock.." amount.

Reverse direction on alternate passes?

This parameter appears only when a value has been entered to the "Material removed per pass" parameter thereby implying multiple passes. After completing a pass, the pass that follows is done in the reverse direction. This is usually only done on open boundaries where back and forth cutting can be accomplished in a continuous motion path.

Delete the following parameter:

Create toolpath with wire offset?

Delete the words "and striking the ESC key" from the NOTE on Page 3-184.

Reference - NC > CUT PART > STEP

GFX	- Insert codes to control Post Processor output. The following describes the implied action of the codes however in most cases this may be customized when creating a Post Processor with GPOST.
ADOV on/off	- Turn on/off acceleration deceleration override.
BLKD on/off	- Turn on/off output of block deletes in the post processed output.
CLMP on/off	- Output a clamp on or clamp off "M" code.
DWEL	- Output a programmed dwell.
HOLD	- Output a program stop "M" code.
INDX	- Output an index "M" code.
KLOC	- Output a keylock "M" code.
OPTN on/off	- Turn on/off option shift for post processors.
OVBP on/off	- Output an override bypass on or override bypass off "M" code.
SPRE on/off	- Output the "M" code assigned to the SPRE ON & OFF functions.
TSTP	- Execute a "TSTP n" command.
Esc	- Exit this menu and return to the STEP menu.

GRAFX II User Reference Manual Supplement

Reference - NC > POST

*The **FINISH** menu is eliminated. All selections in this menu have been moved to the **UTILITIES** menu.*

SELECT >

- | | |
|------------------------------|---|
| Sample Post Processor | - Sample a library Post Processor directly from the distribution floppy disks. |
| Back to main menu | - <i>This menu selection has been omitted, use the ESC key to cancel the SELECT menu.</i> |

UTILITIES >

- | | |
|--------------------|---|
| WRITE | - Write the post processor output file to a device such as a floppy disk. |
| PRINT | - Output the post processor output file to the printer. |
| FILE > - | Redirect the input and/or output of the Post Processor. |
| INPUT | - Respecify the input file to be Post Processed. |
| OUTPUT | - Respecify the output file from the Post Processor. |

Reference - NC > VIEW/PLOT

DISPLAY>

- SCALE** - Display or plot the current view using a specified scale factor with the part centered.
- CSCALE** - Display or plot the current view using a specified scale factor while maintaining the current part reference point.

VIEWS >

*For LATHE programs, change the **XY** selection to **XZ** and specify the **YZ**, **ZX**, **YZRH** and **ZXBK** selections are not available.*

MODES >

*Indent **DRAW TOOL >** and its sub menus one level.*

*Modify the **DRAW TOOL > SPEED > SLOW** description to:
Slow display of tool and set the starting display speed.*

PALETTE >

- TOOLS** - Set the color for tools 1 through 32, or if **FILE** is selected, the toolpath colors will be the same as the colors selected in the GRAFX2 operations parameter screens.

WORKVIEWS > - Select whether or not to sort the PLOT by work view numbers. When enabled, the graphics screen is cleared before each work view number found and all operations associated with that work view are plotted together. If ALL views is active, the sorted work view plots will be done at the currently selected view sequentially in the four viewport windows in a clockwise direction until all workviews have been plotted. If no work views have been established in the toolpath file, this menu selection is disabled.

- YES** - Enable PLOT sort by work view number and bring up menu to turn on or off up to 32 individual work views.

GRAFX II User Reference Manual Supplement

NO - Disable PLOT sort by work view number.

ROTARY > - The ROTARY mode, when enabled, will transform the toolpath data by the index angle programmed for the operation, thereby simulating the toolpath at the workpiece.

YES - Simulate rotary axis indexing.

NO - Do not simulate rotary axis indexing.

BORDER > - The BORDER mode allows you elect whether or not to plot the outside border and JOB info. Suppressing the BORDER may be useful in speeding up plots done on devices such as pen plotter emulators.

YES - Plot the border and JOB info.

NO - Suppress the border and JOB info.

*Change the note at the bottom of Page 3-197 to * Not supported in GRAFX II*

OS - Temporarily shell out to the operating system (type EXIT at the operating system prompt to return to PLOT).

Reference - NC > TOOLING

Type of Tooling: **Turning Non-Turning Turning and Non-Turning Wire**
..Some tools, such as drills, taps and reamers may be defined as both Turning and Non-Turning tools. These tools will then have both turning and milling parameters defined and the tools will be installed in both the milling and turning tool lists.

Type of Tool:

Taper Cutter - Enter parameters for a taper cutter. (Non-turning)

Machining Parameters for All Tooling:

Tool number:

Enter the default tool number (Turning and non-turning). When a value is entered for this parameter, it will be loaded used as the default when loading the tooling file, otherwise no default will be assumed.

Feedrate (IPM):

Enter default feedrate for the tool (Turning and non-turning)

Feedrate (IPR):

Enter default feedrate for the tool (Turning)

Spindle speed (RPM):

Enter default speed for the tool (Turning and non-turning)

Constant Surface Speed (FPM):

Enter default speed for the tool (Turning except Taps)

X Gauge Length:

Enter default "X" gauge length for the tool (Turning)

Z Gauge Length:

Enter default "Z" gauge length for the tool (Turning)

Maximum depth per pass:

Enter the maximum depth of cut per pass (Turning - Single Point and Grooving tools, non turning - End Mills and Face Mills)

GRAFX II User Reference Manual Supplement

Drilling depth:

Enter the default "Z" depth of the hole (Turning and non-turning center drills). If a non zero value is stored, loading the tooling file will force "Depth" mode in the drill depth calculator.

Threads per inch:

Enter the default thread lead (Taps and threading tools)

The following three parameters apply to center drills only. Please refer to the diagram on 68

Body diameter:

Enter the diameter of the tool shank.

Flute length:

Enter the length of the "Drill diameter" section.

Chamfer angle:

Enter the inclusive angle of the chamfer between the "Drill diameter" section and the "Body diameter" section.

For drilling tools change "Tool Diameter" to "Drill Diameter"

If defining a tool for use in both turning and non-turning applications, the following parameters appear:

Non Turning Tool display orientation: **0 90 180 270 ((No) Mirror)**

Enter the default rotation angle and mirror state for displaying the tool in Non Turning applications.

Turning Tool display orientation: **0 90 180 270 ((No) Mirror)**

Enter the default rotation angle and mirror state for displaying the tool in Turning applications.

LIBRARY

-Install a tooling file into a tooling file library. Tooling file libraries group together tooling files associated with common functions. For example, non-turning tools, turning tools and wire tools are grouped into MTOOLING.TXT, LTOOLING.TXT and WTOOLING.TXT respectively, and the appropriate one of these libraries is the default when selecting a tool for an operation. Another common usage of

tooling libraries is to group together tools associated with a particular machine. When selecting this menu item, you are first queried for the name of the tooling file to be installed in the library after which you are queried for the name of the library in which to install it. The `F5' key or LEFT MOUSE button may be used to menu select either of these files.

Reference - NC > ATTRIBUTES

The ATTRIBUTES menu is used to store features or machining information directly into elements in boundaries and point sets. Most simply it provides a way of providing much more control of the toolpaths generated from GRAFX2 and allows access to custom features of many Post Processors.

- | | |
|--------------------|---|
| APPEND | - Add an attribute to a boundary/set element or entire boundary/set. |
| MODIFY | - Examine, Modify or Delete attributes of a boundary/set element or entire boundary/set. This selection brings up a menu of the attributes which are stored in the picked element or global boundary/set and enables the user to change the attributes' values, delete individual attributes or insert new attributes in front of or after any other attribute associated with the element. |
| COPY | - Copy the attributes of a boundary/set element to another element or global attributes from one boundary/set to another. |
| MOVE | - Move the attributes of a boundary/set element to another element or global attributes from one boundary/set to another. |
| DELETE | - Remove all the attributes from a boundary/set element or the global attributes from a boundary/set. |
| SHOW | - Draw elements with attributes in highlighted color. Boundaries or sets with GLOBAL attributes will display a highlighted box at the start of the boundary/set. |
| REPORT > | - Write a file to disk reporting the data on the geometry and attributes within all boundaries and point sets contained in the current job. The default file name is the current job name with a .txt extension. After the file is written, a popup menu appears: |
| DONE | - Nothing further, dismiss the menu. |
| VIEW | - Display the report file on the screen. |
| PRINT | - Print the report file on the printer. |

GRAFX II User Reference Manual Supplement

ELEMENT - Toggle between attributes associated with individual elements
GLOBAL and those associated with the entire boundary.

For a detailed explanation on ATTRIBUTES, refer to the usage notes in the
OBJECTS > BOUNDARY > ATTRIBUTES section on Page 30 of this manual.

Reference - FILE Menu

Change the description:

SAVE - Save the current geometry file. It is recommended that the file be saved whenever a stable point is reached. If a change was detected since the last SAVE operation, a dialog box will query you before you QUIT GRAFX II. If you SAVE the file under a name which differs from the current JOB name, a dialog box will ask if you want to change the JOB name and if there is also a .DMB toolpath file which exists under the original name, another dialog box will ask if you want to duplicate it under the new name.

After the file name is entered, you are asked to enter or edit a job description which will be displayed on the expanded directory menu when retrieving geometry files.

While working in GRAFX II you may find it convenient to SAVE files by typing the CTRL-F instead of using the FILE menu. If a file name has already been established, this is a quick way to save the file since no dialog boxes or other user input events occur with this method.

Change the first line of the description:

RESET - Discard all data on the current job and restore the default modes.

Add the menu selections:

JOBLIST - Rebuild the extended job file list (JOBLIST.DIR) in the GEO path to reflect the current .GEO files. This might be done after using DOS to copy or delete .GEO files whereas the list is not automatically updated.

RESTORE - Restore the most current geometry backup file. The file GEOMETRY.BAK will be accessed from the backup path. A JOB name will be asked for and if a JOB name was in effect at the time the backup file was created, that name will be displayed as the default for restoring the JOB name.

GRAFX II User Reference Manual Supplement

- RESIZE** - Immediately execute a reallocation of entity memory based on the current entity usage. After selecting, a dialog menu will ask for confirmation. Note that this process will automatically occur whenever any entity demand exceeds the current allocation, therefore this selection is not commonly required.
- SETUP** - Leave GRAFX2 temporarily and enter G2CONFIG to change the system setup configuration. GRAFX2 will be reentered with the new setups in effect upon exiting G2CONFIG.

GRAFX II User Reference Manual Supplement

Reference - SCREEN Menu

SCALE - Enter a new graphics display scale factor and redraw with the part centered on the screen.

CSCALE - Enter a new graphics display scale factor and redraw with the current part reference point.

FOCUS - Redisplay the current view centered on the last focus point. This point is continuously redefined to the last point used while creating entities.

PRINT >

YES - Proceed with the printing of the graphics window.

PREVIEW - Preview the printing operation on the screen.

CANCEL - Cancel the printing operation.

Note: Following the execution of REPAINT, ZOOM, UNZOOM, TRANSLATE, EXTENTS, SCALE, CSCALE or PRIOR from the SCREEN menu, the 'F8' hot key may be used to repeat the operation without having to re-access the menu.

Reference - MODES Menu

- GRAPH** - Toggle the display of a line grid ON/OFF at the current GRAPH spacing. When toggling to the ON mode, the modal graph spacing is asked for. If the spacing is set to 0, the current GRID spacing will be used.
- OSNAP** - Toggle the POINT menu default ON/OFF to ENDPOINT instead of CURSOR except in some tasks where cursor point creation is generally more appropriate.
- TOOLPATH >** Invoke this sub-menu to control the display of toolpath when doing a screen redraw with an open toolpath file:
- ALL** - Display all toolpaths up to the current operation.
 - TOOL** - Display only toolpaths associated with the current tool number.
 - OPERATION** - Display only the toolpath in the current operation.
 - LAYER** - Display only toolpaths whose operation's *Active layer* is currently an *ON* layer. This includes the current layer which is automatically set to the *Active layer* when in an operation.
- AUTOSV** - Toggle the automatic backup of the current geometry (GEO) file ON/OFF and set the threshold at which AUTO SAVE takes place. The threshold level is the incremental number of POINTS defined before the backup is triggered. Virtually all entities created in GRAFX2 result in the creation of POINTS. During an AUTO SAVE, the current geometry file is saved into the file GEOMETRY.BAK. If necessary, this file may be recovered by doing a FILE > RESTORE. AUTO SAVE will temporarily be disabled during mass geometry creation tasks such as TRANSFORM, IMPORT, PATTERN and ENGRAVE operations.

GRAFX II User Reference Manual Supplement

Reference - ACCESS Menu

LATHE -Pop up parameter screen to set coordinate system for turning work.

add parameters:

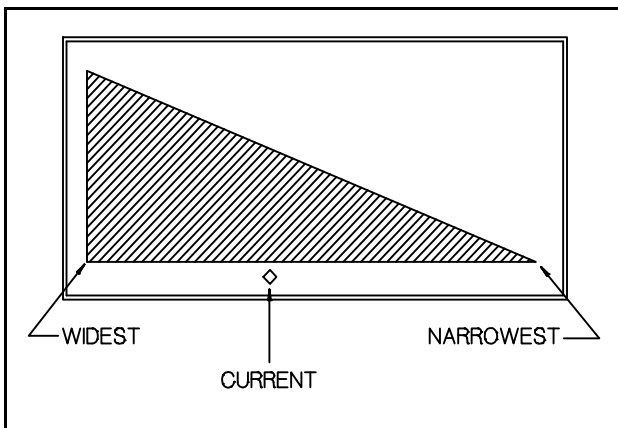
Is this a turning job? **Yes No**

Select YES to set system defaults for turning work. Select NO for non turning work.

Input "X" tool change position as diameters? **Yes No**

This parameter appears only when the parameter *"Input values parallel to the spindle axis as diameters?"* is set to YES. When NO is selected for this parameter, HOME and TOOLCHANGE positions in the "X" axis will be input as their nominal (radial) values.

PICK -Pop up a graphic to widen or narrow the picking tolerance factor when selecting geometric elements with the cross hairs.



WORKVIEW -Change current work view or create or modify a work view.

WORKVIEW Usage notes:

Work Views provide in-depth control over many GRAFX II programming tasks by giving you greater control over the programming environment and allowing better organization of the graphics display.

CREATING AND ACTIVATING A WORK VIEW:

A Work View is created by selecting *WORKVIEW* from the *ACCESS* pull down menu. When *WORKVIEW* is selected, GRAFX II will prompt you for a Work View name. You may type in the name of a new or existing Work View, or press the 'F5' key or the LEFT mouse button to select from a pop up menu of existing names.

Entering a new Work View name automatically creates a new Work View and displays a default parameter screen; selecting an existing Work View name will make that Work View active and will also access its parameter screen if the name is typed or the LEFT mouse button or 'Enter' key is used after selecting the name off the menu.

A Work View may be deleted by bringing up the menu of existing Work Views with the 'F5' key or the LEFT mouse button, highlighting the Work View with the cursor and pressing the 'DEL' key.

A number of associated conditions are defined for each Work View. These conditions are instantly enabled when a particular Work View is activated.

Once a Work View is entered or selected, the current Work View is changed and the Non Turning Work View parameter screen appears as follows:

NON TURNING (includes WIRE)

Work view name:

Local X origin:

Local Y origin:

Local Z origin:

Index angle:

Work view offset number:

Freeze layers?

Layers:

If the "turning job" parameter is enabled in the *LATHE* parameter screen of the *ACCESS* menu, this Turning Work View parameter screen will appear instead:

TURNING

Work view name:

Local X origin:

Local Z origin:

Turret: (rear or front)

Freeze layers?

Layers:

Work view name:

This parameter indicates the name used to access the Work View. The name may be edited in order to rename the Work View. A maximum of 16 characters, including spaces, is allowed.

Local XYZ origin:

These parameters establish the location of a local coordinate system associated with a Work View. A local coordinate system is in effect only if it is different (X,Y or Z do not = 0) from the absolute origin. The local coordinate system is represented by a second set of axes (drawn in cyan) when *AXES* is enabled in the *MODES* pulldown menu. The Coordinate Position display (*MODES* > *COORDS*) is relative to the local system. The *EXAMINE* function reports local coordinate values.

Index angle:

This parameter specifies an angle for indexing tables on horizontal and vertical machining centers. This value will be automatically entered into the "*Rotate index table to (Angle in degrees):*" machining parameter.

Work view offset number:

This parameter associates a work coordinate offset number to the Work View. When set to a non zero value, the post processor, if so prepared, will output a work offset block (e.g. G54).

See section "WORK VIEWS APPLICATIONS" below for more information.

GRAFX II User Reference Manual Supplement

Freeze layers? **No Yes**

When set to **Yes**, the *Layers* parameter is omitted from the parameter screen on subsequent accesses to the WORK VIEW, thereby preventing modification of the layer list for that Work View. Subsequent changes to the layers may be accomplished by resetting the query to **No** and then reentering the WORK VIEW to select the *Layers* parameter.

Layers:

This parameter associates a group of layers with a Work View. Choosing this parameter displays a menu of the existing layers. The *On* and *Active* layers for the Work View are then selected as usual.

Work view parameters are retained by the system; therefore, when you change to an already existing Work View, data does not have to be reentered.

If you only want to change to a new active Work View, select the desired Work View name and press the **Esc** key immediately. This dismisses the parameter screen and leaves the system in the Work View you selected.

The Work View name "0" is automatically created on a new job. It is the default view and can easily be modified or restored by the user. The *G2SIZE* program may be used to increase or decrease the default number of Work Views allowed.

USING A WORK VIEW FOR TOOLPATH GENERATION:

The "*Active work view:*" parameter is found on the first parameter screen of the *TURNING*, *NON-TURNING* and *WIRE* PARAMETERS. Like the "*Active layer*" parameter, specifying an "*Active work view*" in a machining operation makes that Work View active when executing the operation. In addition, if an "*Active work view*" is specified, the following machining parameters are automatically set by the associated Work View parameters:

GRAFX II User Reference Manual Supplement

<u>Machining Parameter</u>	<u>Work View Parameter</u>	<u>Application</u>
Rotate index table to	Index angle	Non-turning only
Work offset number	Work view offset number	All
Lathe type	Turret	Turning only
Active layer	Layers (current)	All

The "*Work offset number*" machining parameter associates a machining operation with a work offset, usually for the purpose of using work coordinates in Post Processing. This parameter is automatically loaded when an "*Active work view*" is specified and the work view has a "*Work view offset number*" assigned to it. (see OUTPUT CONTROL below.)

All Z coordinates in the machining parameter screens are entered relative to the active local Z origin. The system automatically adjusts these values so that toolpath is correctly displayed in three dimensions. Coordinate output is also automatically adjusted according to the active "*Work view offset number*" parameter "*Shift*" or "*No shift*" selection.

For example, if the local "Z" origin is +2.0 the table below shows "*No shift*" output relative to input.

MACHINING PARAMETERS

INPUT (local)	CALCULATION (local+relative)	OUTPUT (absolute)
Depth of cut: -1	(2-1)	Z1
"Z" approach clearance plane: .1	(2+.1)	Z2.1
"Z" retract clearance plane: 2	(2+2)	Z4
"Z" clearance plane between plunges: .06(2+.06)	Z2.06	
"Z" level for top of part: 0	(2+0)	Z2
Final "Z" depth of drill stroke: -2.5	(2-2.5)	Z-.5

The "Z" level specified by positioning points is also relocated. On *DRAW PART* operations, the top and bottom surface planes will be modified by the local "Z" origin of the currently active work view.

WORK VIEWS - APPLICATIONS:

DISPLAY CONTROL

At a basic level, you can use work views to display a customized selection of drawing layers when generating toolpath for a particular operation. This technique permits you to display only those boundaries that should be selected for that operation along with any other drawing information desired.

The boundary(s) required for an operation can be placed in one layer and related text or drawing information placed in other layers. A Work View controlling the display of these layers is then created and specified in the machining parameter screen. When executing this operation, only the pre-designated layers are displayed. The extents of the display are determined by the active layer specified in the Work View.

To organize the display screen to best suit your programming needs, additional Work Views can be created or existing ones recalled as required when operations are added.

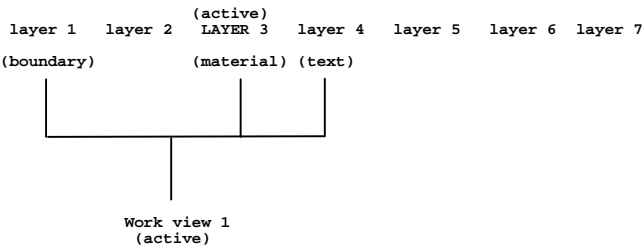
Examples:

The following diagram illustrates the organization of a GRAFX II CAD/CAM program. The CAD portion consists of seven layers of drawing, dimension and text information. Each layer contains information necessary for the development of the drawing or programming process. You may however not want to display all of the CAD information when generating toolpath. Different operations usually require totally different information; displaying that information selectively keeps the screen uncluttered.

Operation #1 - Work view #1

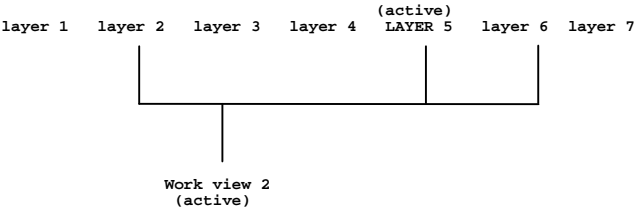
Work View #1 consists of Layer 1, Layer 3, and Layer 4. When Operation #1 is executed, only these layers will be displayed. Layer 3 is the active layer and controls the extents of the display. Layer 1 for example might contain the boundary to be machined while Layer 3 would contain the drawing of the material block. Layer 4 might contain text instructions.

GRAFX II User Reference Manual Supplement



Operation #2 - Work view #2

Work view 2 designates Layer 2, Layer 5, and Layer 6. When Operation #2 is executed, only these layers will be displayed. Layer 5 is the active layer and controls the extents of the display.



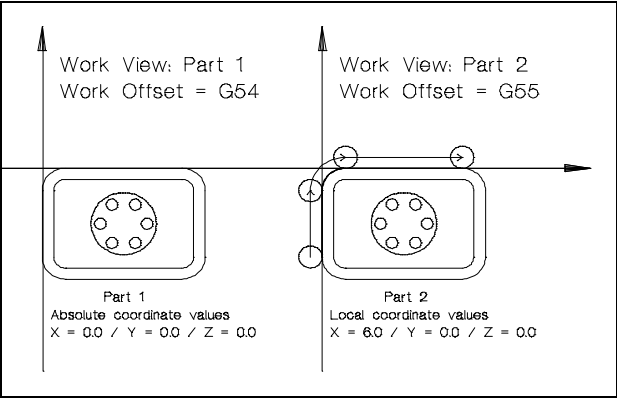
Work Views may also be used for display control in the VIEW/PLOT utility by enabling the WORKVIEWS mode while in VIEW/PLOT. This will sort the viewing or plotting of operations by the Work Views. PART DRAW operations may be assigned to particular Work Views by making the Work View active before executing the PART DRAW operation.

OUTPUT CONTROL

There are two general methods of using Work Views to output G-code and related coordinate offsets for multiple part machining. Both methods are illustrated in the example below which uses a .5" diameter end mill to cut a partial contour on identical parts that are spaced 6.0" apart.

GRAFX II User Reference Manual Supplement

Both methods use the same geometry which is shown below:



GRAFX II User Reference Manual Supplement

A Work View is created for the machining of each part; both methods use identical Work View parameters. The parameter screens follow.

Work View for Part 1...

Work View

Work view name: Part 1

Local X origin: 0

Local Y origin: 0

Local Z origin: 0

Index angle:

Work view offset number: 1

Layers:

part arrangement

MILL - Part 1

text annotation #1

⇐ active layer

Work View for Part 2...

Work View

Work view name: Part 2

Local X origin: 6

Local Y origin: 0

Local Z origin: 0

Index angle:

Work view offset number: 2

Layers:

part arrangement

MILL - Part 2

text annotation #2

⇐ active layer

Method #1:

The first method uses the "*SHIFT*" selection at the "*Work view offset number:*" parameter to output axis coordinates for Part 2 in terms of its local origin system. A machine work coordinate system is then used with work offsets to provide "BIG" offset information.

Note: The term "BIG" and "LITTLE" as used here mean an offset value that positions parts using the machine tool control's internal local coordinate system. The local origin of Part 2

GRAFX II User Reference Manual Supplement

is 6.0" to the right of the local origin of Part 1. This is a "BIG OFFSET". "LITTLE" offsets are used to compensate for slight machine positional or fixture errors.

A "BIG" offset is often compensated by a "LITTLE" offset.

The G-code for Part 1 uses work offset code G54 while the G-code for Part 2 uses G55. All machining is relative to the local coordinates in effect for each operation.

When entering the *"Work view offset number:"* in the machining operation parameters you are prompted to select *"SHIFT"* or *"NO SHIFT"*. Coordinate output is adjusted based on this selection. The example below illustrates this.

The parameter screen for Part 1 follows:

Non-Turning Parameters

Tool file name: em20500

Tool description: .500" DIA END MILL

Tool Number: 1

Tool length offset number: 1

Fixture offset number:

Machining operation: Contour

Operation description: Contour Mill / Part 1

Work view: Part 1 *

Work view offset number: 1 (shift) **U** "1" = G54; "SHIFT" = local coords. *

Active layer: MILL - Part 1 *

X tool change position: 0

Y tool change position: 0

Z tool change position: 0

Tool diameter: 0.5

Tool length: 3

Spindle speed (RPM): 1000

Feedrate (IPM): 5

* Note: These parameters are automatically entered from the Work View when Part 1 is selected in the Work view parameter.

The parameter screen for Part 2 is identical to Part 1 except for the following:

GRAFX II User Reference Manual Supplement

Operation description: Contour Mill / Part 2

Active work view: Part 2

Work view offset number: 2 (shift) **Ů "2" = G55; "SHIFT" = local coords.**

Active layer: MILL - PART 2

It is important to note that the *"SHIFT"* selection in the above operations only affects the G-code file output; it has no effect on geometry or toolpath display.

When post processed, the output is shifted by the amount of the local origin coordinates specified in the *"Active work view:"*.

Contour Mill / Part 1

```
N1 G54
N2 G90 G0 X-.25 Y-2.0
N3 Z.1
N4 G1 Z-.125 F2.
N5 Y-.5 F5.
N6 G2 X.5 Y.25 I.75 J0
N7 G1 X3.0
N8 G0 Z.1
```

← Work offset

Contour Mill / Part 2

```
N9 G55 X-.25 Y-2.0 Z.1
N10 X-.25 Y-2.0
N11 G1 Z-.125 F2.
N12 Y-.5 F5.
N13 G2 X.5 Y.25 I.75 J0
N14 G1 X3.0
N15 G0 Z.1
```

← Work offset

Because of "SHIFT", these output coordinates are relative to Work View "Part 2" local origin

Prior to machining the parts, the machine tool operator enters the corresponding "BIG" offset local coordinate values of $X=0.0$ $Y=0.0$ and $Z=0.0$ into the machine offset register for *G54*, and $X=6.0$, $Y=0.0$, $Z=0.0$ into the offset register for *G55*.

Method #2:

This second method illustrates how to use a local coordinate origin in conjunction with the *"NO SHIFT"* option. In this case a machine work coordinate system is used with work offsets to provide "LITTLE" offset information.

The output created here differs from Method #1 in that the coordinates for Part 2 are relative to the absolute origin rather than the Work View local origin.

GRAFX II User Reference Manual Supplement

The Work View parameters themselves are exactly the same as in the first method.

The "*NO SHIFT*" selection is made when the "*Work view offset number:*" parameter is entered. This is the sole mechanism that produces absolute, not local, output coordinates.

The parameter screen for Part 1 follows. Notice that the only difference between this screen and the screen at Method #1 is the "*NO SHIFT*" selection.

Non Turning Parameters

Tool file name: em20500

Tool description: .500" DIA END MILL

Tool Number: 1

Tool length offset number: 1

Fixture offset number:

Machining operation: Contour

Operation description: Contour Mill / Part 1

Work view: Part 1

Work view offset number: 1 (no shift) **U** "1" = G54; "NO SHIFT" = abs. coords.

Active layer: MILL - Part 1

X tool change position: 0

Y tool change position: 0

Z tool change position: 0

Tool diameter: 0.5

Tool length: 3

Spindle speed (RPM): 1000

Feedrate (IPM): 5

The parameter screen for Part 2 is identical to Part 1 except for the following:

Non Turning Parameters:

Operation description: Contour Mill / Part 2

Active work view: Part 2

Work view offset number: 2 (no shift) **U** "1" = G54; "NO SHIFT" = abs. coords.

Active layer: MILL - Part 2

When post processed, the output is Not SHIFTEd by the amount of the local origin coordinates of the specified "*Active work view:*".

GRAFX II User Reference Manual Supplement

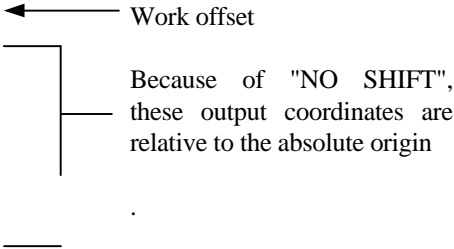
Contour Mill / Part 1

```
N1 G54
N2 G90 G0 X-.25 Y-2.0
N3 Z.1
N4 G1 Z-.125 F2.
N5 Y-.5 F5.
N6 G2 X.5 Y.25 I.75 J0
N7 G1 X3.0
N8 G0 Z.1
```



Contour Mill / Part 2

```
N9 G55 Z.1
N10 X5.75 Y-2.0
N11 G1 Z-.125 F2.
N12 Y-.5 F5.
N13 G2 X6.5 Y.25 I.75 J0
N14 G1 X9.0
N15 G0 Z.1
```



Summary:

To have the machine tool's control work coordinate system locate the part select *"SHIFT"*. GRAFX II will use local coordinate output and supply the required work coordinate offset number.

To locate the part with G-code, select *"NO SHIFT"*. GRAFX II will use absolute coordinate output with the required work coordinate offset number.

UNITS

- ...

Angle units for expressions: **Degrees Radians**

VARIABLE

-Examine or set values for variables which may be used in entering mathematical expressions when inputting data in the system. When selected, you are prompted for a variable letter which must be a single alphabetical character *a-z*. Upper case characters are accepted but converted to lower case. When the letter is input, you are then queried for the value to be assigned to the variable. The current value of the variable is displayed as the default which you may keep by pressing ENTER or change by typing in a new value. The new value may be entered with an expression, which means any variable may be used to enter new variables.

Once a variable has been assigned a value, you may then use the letter to represent the value when entering expressions. Initially, several variables are predefined by the system (see Assigned Symbols on Page 3-2 in Reference Manual). User defined variables are stored in the GEO file and restored when the file is loaded in.

MACHINE

-Set the default machine type to MILL, LATHE or WIRE. If set prior to toolpath generation, the machine type may be used to make a selection of an extended directory list which has been sorted by type (such as tooling files).

Note: Following the execution of EXAMINE, LAYERS, WORKVIEW, VARIABLE or PICK from the ACCESS menu, the 'F8' hot key may be used to repeat the operation without having to re-access the menu.

Reference - INFO

Change the description to:

Selecting INFO reports the current JOB name, display SCALE and currently active LAYER and WORK VIEW in a pop up window along with statistics on the entity database. The entity statistics report the number of entities defined and available based on the current adjusted size parameters. The sizes allocated to the entity types will adjust according to usage as soon as any requirement exceeds the allocation. The default maximum size parameters may be adjusted with the G2SIZE utility (see Appendix H).

If a toolpath file is open at the time, the INFO screen will also display the operation description if it exists for the current operation.

Reference - UNDO

UNDO gives the ability to:

- delete the last entity that was created
- restore the last entity that was deleted
- restore geometry prior to a TRIM/EXTEND operation
- restore geometry prior to a FILLET or CHAMFER operation
- restore geometry prior to a BREAK or UNBREAK operation
- restore geometry prior to an ARC-CIRCLE modify operation
- restore geometry prior to a VALUES modify operation
- delete the geometry created from the last IMPORT operation
- delete the geometry created from the last LETTER/GEOMETRY operation
- delete the last generated toolpath and/or convert it to a boundary
- delete the geometry created from the last PATTERN operation
- delete the last CONNECTED GEOMETRY element
- delete the last positioning point created
- delete the last RECTANGLE created
- delete the last ELLIPSE created
- restore the last LAYER REMOVED
- restore the last WORKVIEW REMOVED
- delete the last CROSSHATCHING created
- restore the entities prior to the last WINDOW DELETE
- restore the entities prior to the last HATCH DELETE
- restore the entities prior to the last DELETE ALL
- restore the last deleted boundary or point set
- restore the entities prior to the last TRANSFORM
- restore the entities prior to the last LAYERS COPY/MOVE

When the cursor is moved into the UNDO box at the top of the screen, the menu info window will display what operation the UNDO will perform.

If the current UNDO operation is to delete or convert the last generated toolpath, a pulldown menu appears with the following selections.

DELETE - Delete the last generated operation from the toolpath file.

CONVERT - Convert the last generated toolpath operation into a boundary. This boundary which represents the path of the center of the tool may be edited and a toolpath regenerated by using a CONTOUR operation OVER the boundary. In turning, a FINISH with no offset may also be executed. After the toolpath is converted, a dialog box queries whether or not to delete the original toolpath.

GRAFX II User Reference Manual Supplement

GPOST - Using the GPOST System

Modify from line 5 in paragraph 2 on Page 4-11:

A null can be recorded by pressing the **DEL** key followed by **ENTER**. If there was no response originally entered, you can simply skip over the query by pressing the Cursor Down (↓) key, the **ENTER** key only, or the right MOUSE button.

Add to the end of the first sentence in the third paragraph on Page 4-15:

or press 'F5' or the LEFT MOUSE button and select **MYPOST** off the menu list.

Modify paragraph 2 on Page 4-17:

In Datacut's GPOST System, a provision is made to create pictures of six different conditions. They are:

- 1) the start of a tape sequence,
- 2) the end of a tape sequence,
- 3) the start of a tool sequence,
- 4) the end of a tool sequence,
- 5) the start of an operation sequence,
- 6) the end of an operation sequence.

Add the conditionals to Page 4-18:

<os Start of a machining operation

<oe End of a machining operation

GRAFX II User Reference Manual Supplement

Add before the last paragraph on Page 4-21:

The post processor directory may also be established by setting a PMB data path in the PATHS.SYS file (see Appendix I)

Delete the last sentence in the last paragraph on Page 4-21 referring to the DOS program SORT.EXE. It is no longer needed.

Add the following section to Page 4-22:

Restoring a Post processor

When a Post processor is saved, the former version is backed up to the file **POST.BAK**. If you wish to get back the old file, go to DOS and type the following:

copy post.bak ppname.pmb [Enter]

GRAFX II User Reference Manual Supplement

GPOST - Explanation of Queries

Add the following queries:

Enter program number word address: Often "o" or ":"

Enter program number placement:

Use the right or left arrow keys to select one of the three choices: **Before RWS code**, **After RWS code** or **Suppress program numbers**

Enter search block word address:

Applies to controls with block searching capability. See "nb" in *Picture File Symbols*.

Enter fixture offset code word address:

NON-TURNING only. Sometimes this is an "E".

Enter fixture offset code field width:

NON-TURNING only. Use the HELP screen for an explanation of field widths. In this case, usually either a 0 or a 2 will apply.

Is the spindle direct reversing?

TURNING and NON-TURNING only. If "N" is selected, the Post Processor will output a spindle off code prior to reversing the spindle in simulated TAP cycles. If "Y" is selected, spindle off codes will not be output in these cases.

Enter initial PECK increment word address:

NON-TURNING only. Some machines provide for output of an initial PECK increment different from the subsequent PECK increments. Setting this parameter will output those values programmed in the toolpath file.

GRAFX II User Reference Manual Supplement

Enter BOR2 shift increment word address:

NON-TURNING only. Specifying this parameter will cause the Post Processor to output shift increment values contained in the toolpath file. This applies to machines with fine boring cycles where the boring bar shifts away from the wall of the bore before retracting.

Convert .DA file informative text to comments? "Y" or "N"

Specifying "Yes" for this parameter will convert informative text in the .DA file into tape comments if the "Start of comment character" parameter has been set. Informative text lines start with the ¶ character and are normally stripped out by NCTALK before the file is sent to the machine control. By setting this parameter, these lines may appear as comments which are displayed but not processed at the control.

Chase in/out TAP "G" code:

TURNING only. If a value is entered, TAP operations which have a lead (threads per inch) specified, will output the in and out stroke using this "G" code and the thread lead. Usually this will be the same as the threading "G" code.

Work coordinate "G" code (lowest number):

Establishes the base code for a sequence of work coordinate "G" codes, for example G54-G59. This parameter is used when work offsets are programmed in the toolpath (.DMB) file and the picture file. A work offset is specified in GRAFX2 by the work offset number.

Fixed cycle full retraction "G" code: NON-TURNING only. Often 98

Fixed cycle partial retraction "G" code: NON-TURNING only. Often 99

Change:

Are the axes measured from a fixed point on the turret?

To:

Adjust axes output by Gauge Lengths?

TURNING only. Toggle selecting **Yes: adjust all output** results in the adjustment of all axes output by the gauge lengths input in the toolpath parameters. Selecting **Yes: adjust PRESET block only** results in the adjustment only to the PRESET block (typically G50 or G92). Selecting **Yes: except on tool change position** results in the adjustment on all blocks except on the moves to the tool change position.

GRAFX II User Reference Manual Supplement

Through the tool coolant on "M" code: TURNING and NON-TURNING only.

Part catcher advance "M" code: TURNING only.

Part catcher retract "M" code: TURNING only.

Note: These changes also affect the NON-TURNING, TURNING and WIRE printouts which begins on Page 4-55.

GRAFX II User Reference Manual Supplement

GPOST - Picture File Symbols

Add the following symbols:

sym	M	L	W	Description
gq		x		Output maximum RPM "G" code
gr		x		Output constant RPM block ("G" & "S") if RPM has been programmed
nb	x	x	x	Replace the sequence number word address with the search block word address on this block.
ss	x	x		Output constant RPM "S" code.
tq	x			Output fixture offset code.
tx		x		Cancel tool point angle before the output of the move to the home position on a tool change and the end of program.
wo	x	x	x	<p>Output work coordinate block (Gwv [Xo Yo Zo]) if a work offset number has been programmed and has changed from that of the previous operation. Gwv is the "G" code equal to the base work coordinate "G" code plus the work offset number minus 1 and Xo Yo Zo are the next x, y, and z positions which only come out on the second or subsequent work coordinate blocks output in a particular tool change.</p> <p>Unless the "No shift" option has been specified in the "Work view offset number" parameter when generating the toolpath in GRAFX2, subsequent output from POST will be translated by minus the values of the local origin of the work view when this symbol is executed.</p>

Utilities Menu

*All **PRINT** utilities have the option of outputting to a file. When **FILE** is selected from the **OUTPUT** menu, the user is asked to enter the path and name of the file which will be created.*

add the menu selection:

PRINT POST PROCESSOR DOCUMENT

Output the Post Processor document (if it exists), on the active Post Processor to the screen, printer or to a file. The document exists if there is text in the picture file following a line starting with #T.

GRAFX II User Reference Manual Supplement

*Change the references to **NPOST.PMB** on Page 4-52 to **NPOST.PMI**.*

Appendix C - Z Surface Projections

Add the parameter for all projections:

Coordinates on tape: Tip Center

"Tip" means that the coordinates output to the toolpath file will be relative to the bottom tip of the ball end mill. "Center" outputs coordinates relative to the center of the ball and is the default parameter.

GRAFX II User Reference Manual Supplement

Appendix F - Communications Protocols

Correction to: Numonics 2200 digitizing tablets

Change Data Bits from 7 to 8

Change Switch B-5 to ON and Switch B-7 to OFF

Appendix G - File Retrieval

GRAFX II offers a convenient way of loading existing files on the system. Whenever a file needs to be loaded, such as loading in a geometry file, importing a file, loading a tooling file etc., a menu of candidate files in the default data path (see Appendix I) may be displayed by pressing the `F5' key and the desired file may be picked off the menu. In addition to loading files off the menu, files may also be deleted by highlighting a file and then striking the **Del** key. When this is done, a dialog box confirms the deletion of the file and if it is a .GEO file that is being deleted and a .DMB file of the same name exists, another dialog box asks whether or not to delete the .DMB file. If an .OPS file of the same name exists, another dialog box asks whether or not to delete the .OPS file.

Another option is to enter a file specification into the "Enter File" input window and then press the `SHFT/F5' key. This specification may be a path, that is a device name and directory path followed by a partial file name which is used to mask file names conforming to the name. The following examples illustrate input strings to the "Enter File" with the resulting menu created by the `SHFT/F5' key:

Enter File

a:	all target extension files in the current path on the a: drive
\acad\dx\ca	all target extension files starting with "ca" in the directory \acad\dx
\gx2\tooling	all target extension files in the directory \gx2\tooling
d:\data*.dat	all files with a ".dat" extension in the d:\data path

If the number of candidate files exceeds the number which can be displayed on one screen, **PgDn** and **PgUp** selections will appear on the menu allowing the user to quickly bring up the next or previous menus of candidate files.

When entering the file name to load .GEO files, the `CTRL/F5' key may be pressed to display a menu of recent .GEO files accessed. This menu will display up to 25 file names in the order they were last accessed. Appearing next to the file name is the date and time the file was last accessed. The history list may be modified by editing the file HISTORY.SYS.

Some file types support expanded directory lists. Expanded directory lists not only generate menus of file names, but attach descriptions to the file names. The

GRAFX II User Reference Manual Supplement

file name plus description may be up to 78 characters long. If no description was entered when a file was created, the file creation date and time will appear in the description field. Besides giving detailed descriptions of the files, expanded lists may be sorted by usage, for example, Tooling Files and Post Processors may be put into separate lists sorted by Mill, Lathe or Wire usage. The short form menus are built on the fly from the existing files at the time of menu generation. The expanded lists are pre-built by several utility programs and then updated when files of the supported types are created or modified. These utility programs are described in Appendix H. When a file menu is created, if the expanded list file exists, it will be displayed, otherwise the short form list will be displayed. When an expanded list menu is up on the screen, the **F10** key may be pressed and the short form list will replace it. The short form lists may be desirable in some cases because they are built dynamically and are guaranteed to be accurate or occasionally total unsorted lists may be needed.

The following table lists the file types supported by the expanded directory lists:

EXTENSION	FILE TYPE	LIST FILENAME	PREBUILD UTILITY
.GEO	geometry job file	JOBLIST.DIR	EXTLIST.EXE
.TLL	lathe tooling file	LTOOLING.TXT	TOOLIST.EXE
.TLL	mill tooling file	MTOOLING.TXT	TOOLIST.EXE
.TLL	wire tooling file	WTOOLING.TXT	TOOLIST.EXE
.PMB	lathe post processor	TURN.SYS	INSPPOST.EXE
.PMB	mill post processor	NTURN.SYS	INSPPOST.EXE
.PMB	wire post processor	WIRE.SYS	INSPPOST.EXE
.DMC	toolpath operation	DMCLIST.TXT	DMCLIST.EXE
.DMF	part family template	DMFLIST.TXT	none
.VEC	engraving font	VECLIST.TXT	EXTLIST.EXE
.MAC	drilling macro	MACLIST.TXT	EXTLIST.EXE

Since expanded directory lists are not accessed unless they've been initially created (except for .DMC & .DMF files), you must build the lists outside of GRAFX2. The pre-build utilities are all automatically accessible transparently through the G2CONFIG configuration program. You may run G2CONFIG from DOS or from GRAFX2 as described in Appendix H, and select *Install extended file lists* to pre-build the lists. This may also be done at any time to create up to date lists. For example, if you copied one or more files onto your system, you would need to run the configuration to get the files into the lists. In the case of geometry job files, JOBLIST.DIR may also be built at any time without leaving GRAFX2 by selecting JOBLIST off the FILE pulldown menu.

Appendix H - Utility Programs

Distributed with the GRAFX II software are a number of utility programs which are not accessible from the GRAFX II menu. These programs may be run from DOS and may be of value in using the GRAFX II system.

G2CONFIG

This program is used to configure the GRAFX II system. By running G2CONFIG you may configure your hardware by selecting such items as graphics modes, pointing devices, printers, plotters and digitizers. You may also set up some software configurations such as a user specified editor program, default paths to the data files used in the GRAFX II system and to create expanded directory lists for tooling files, Post Processors and toolpath operations files. The expanded directory list files will be created in any default data path previously set for the particular file type. Use of this program is documented in the **installation** section of the User Reference Manual. G2CONFIG may be run either from the DOS prompt or loaded from within GRAFX2 by selecting SETUP off the FILE pulldown menu.

G2CONFIG runs in conjunction with the selection menus in the file G2CONFIG.MNU.

INSPPOST

This program builds the Post Processor selection menus used when selecting a Post Processor while running POST or Backplotting in GRAFX2. INSPPOST is automatically run when doing an **INSTALL** from GPOST but it may also be accessed from DOS. A common usage may be to delete unneeded Post Processors from the GX2 directory and rebuild the menus so that only the Post Processors you need to access come up. This may be done as follows:

First delete the .PMB files you don't use. Next enter the following command:

INSPPOST

This will build the menu files NTURN.SYS, TURN.SYS and WIRE.SYS from which Post Processors are selected.

You may also specify the **-TXT** option to INSPPOST to create the files MILLPOST.TXT, TURNPOST.TXT and WIREPOST.TXT which list the Post Processors with their descriptions.

GRAFX II User Reference Manual Supplement

In addition to running **INSPOST** from DOS, you may also run it by running **G2CONFIG** and selecting *Install extended file lists* followed by *Post Processors (.PMB)*. When this is done, **INSPOST** will look for .PMB files existing in the default path to *.PMB Post Processors* if the path has been previously set.

G2SIZE

This program is used to resize the default entity sizes used by GRAFX2. This may be desired for a number of reasons. In some programs you may be creating a large amount of certain entities which exceed the default number. You can use G2SIZE to decrease the number for entity types you don't need and increase for those you need more of although dynamic entity resizing done by GRAFX2 should take care of this automatically.

From the DOS prompt in the GX2 directory you type the following:

G2SIZE

This brings up the sizing utility which first loads GRAFX2 to find out how much memory is available for entities.

When the G2SIZE screen appears, pick **SIZES** off the top menu in the usual way and a screen listing all the entity types with a corresponding number appears. You may step through the entities and change the sizes. Notice that when you change an entity size the window at the bottom titled "Memory Left" displays a new value for the number of bytes of available memory. If you select a value which will result in more than the available memory, the system will display a message and adjust the value to the largest number attainable given the memory availability. Remember that the **F1** key is always available to give help while using this program.

After you have adjusted the entity size you may return to the main menu by falling through the last parameter or hitting the **Esc** key. You then select **QUIT** and a popup menu will ask if you want to save your changes. If you select save, a file named **G2SIZE.SYS** is created or modified with the numbers you have entered. This file is examined by GRAFX2.EXE when it first starts up in order to set the storage for the entities. If you set new entity sizes for GRAFX2 and later wish to run GRAFX2 with the default sizes, you should first rerun G2SIZE and select **DEFAULTS** off of the **FILE** menu. This renames the **G2SIZE.SYS** file so GRAFX2 doesn't find it. You can subsequently restore the size file you created by again running G2SIZE and selecting **RESTORE** off of the **FILE** menu.

TOOLIST

This program builds files which list the names and descriptions of the tooling files (.TLL) found on your system. These list files are used to create expanded directory lists when accessing tooling files in GRAFX2. After running TOOLIST, the files LTOOLING.TXT for lathe tooling and MTOOLING.TXT for mill tooling are created. You may use the **VIEW** command from the **FILE** pulldown menu in GRAFX II to look into these files.

In addition to running **TOOLIST** from DOS, you may also run it by running **G2CONFIG** and selecting *Install extended file lists* followed by *Tooling files (.TLL)*. When this is done, **TOOLIST** will look for .TLL files existing in the default path to .TLL Tooling files if the path has been previously set.

SPDIAG

This program diagnoses your Software Interface Module (SIM). This is the green device (Activator) or black device ("U" type) you need to plug into a parallel port on your computer in order for GRAFX II to operate. In systems supplied without a hardware SIM, SPDIAG diagnoses the software SIM ("F" type) installed on the hard disk during software installation.

After entering SPDIAG from the DOS prompt a report is generated on the screen. First the assigned Port addresses for each of 3 possible parallel ports (LPT1, LPT2 and LPT3) are given. Next the Port address where the SIM is found is given (Activators only), and finally the letter A, U or F followed by the access code of the SIM is given.

If the message "No SIM...exiting program" is displayed after the port assignments or if the access code is a negative number, check to see that the hardware SIM, if supplied, is plugged firmly into an operable parallel port and that if a printer is plugged into the SIM that the printer's power switch is on.

If the access code is 0, you will not be able to run GRAFX II and you should call Datacut Customer Service.

GRAFX II User Reference Manual Supplement

DXFSIZE

This utility is used to scan a DXF file to determine the number of entities which would be translated into GRAFX2 on a DXF IMPORT. From the DOS prompt in the GX2 directory, you type the following:

DXFSIZE fname [-b]

Where "fname" is the path and name of the DXF file and "-b" is optionally specified if you want BLOCK searches printed.

The program will scan the specified file and display a report as follows:

DXFSIZE Version 1.xx

n points

n point entities

n line entities

n arc entities

The values reported may be used as a guide for entity resizing using **G2SIZE** to accommodate large DXF files.

GPPATH

This utility is used to alter an individual Post Processor's default output path. This may be useful in situations where the user has organized his computer to store his G-code files in a different directory for each machine. From the DOS prompt in the GX2 directory, you type the following:

GPPATH ipost [opost] -opath

Where "ipost" is the path and name of the input post processor
"opost" is optionally the path and name of the output post processor (to create a new Post in addition to the original)
"-opath" is the default path for the Post Processors output (.DA) file

for example:

gppath \gx2\post\matmx2 -\nct\matsuura

gppath reports on the screen:

Replacing xxxxxx

Storing \nct\matsuura

GRAFX II User Reference Manual Supplement

In this example, the default output path of the MATMX2 Post Processor, residing in the \GX2\POST\ path is sent to the \NCT\MATSUURA\ path. The program reports any previously stored path which may be overwritten. **GPPATH** overrides the default output path for Post Processing setup by the da parameter in the *PATHS.SYS* file (see Appendix I).

DMCLIST

This program builds a list file named DMCLIST.TXT which lists the toolpath operation parameter files (.DMC) which exist on the system. The .DMC files are created by selecting FILE on the dialog menu which after exiting operations parameter screens or after searching a PART DRAW operation. The format for each record in the list file is as follows:

DMC file name - (DMB file name) Operation description

The *(DMB file name)* refers to the name of the original toolpath file from which the DMC file was cut.

In addition to running **DMCLIST** from DOS, you may also run it by running **G2CONFIG** and selecting *Install extended file lists* followed by *Toolpath operations (.DMC)*. When this is done, **DMCLIST** will look for .DMC files existing in the default path to *Geometry & Toolpath files* if the path has been previously set.

Appendix I - Data Paths

GRAFX II provides for the organization of data into separate sub directories in order to make data more manageable, and to easily link up data with other applications. This is accomplished by setting up default data paths to the different type of data files used in the GRAFX II system. The record of these paths is kept in a file named **PATHS.SYS** which resides in the \gx2 directory.

Many of the system programs in the GRAFX II system look for the **PATHS.SYS** file to determine where they look for data. If the **PATHS.SYS** file is not found, all data files are assumed to exist in the \gx2 directory except in some instances where a DOS environmental path can also direct a system program to where data files reside. The **PATHS.SYS** file may have one or more of the following specifications:

Specification	Description
bak=path	Default path to <i>GEOMETRY.BAK</i> and <i>TOOLPATH.BAK</i> backup files
geo=path	Default path to the <i>.GEO</i> and <i>.DMB</i> files
pmb=path	Default path to the <i>.PMB</i> Post Processors
tll=path	Default path to the <i>.TLL</i> tooling and import files
vec=path	Default path to the <i>.VEC</i> engraving font files
mac=path	Default path to the <i>.MAC</i> drilling macro files
swp=path	Default path to the <i>PTSWAP.TMP</i> point swap file
dxf=path	Default path to the <i>.DXF</i> import/export files
igs=path	Default path to the <i>.IGS</i> import/export file
cdl=path	Default path to the <i>.CDL</i> import files
plt=path	Default path to the <i>.PLT</i> import/export files
gfx=path	Default path to the <i>.PT</i> GFX export files
apt=path	Default path to the <i>.APT</i> export files
c2=path	Default path to the <i>.C2 COMPACT II[®]</i> export files
cl=path.ext	Default path to the <i>.CL</i> import files
ncw=path.ext	Default path for <i>WRITE</i> output in <i>NCTALK</i>
da=path.ext	Default path for output files in <i>POST</i> and input files in <i>NCTALK</i> and <i>BACKPLOT</i>
cpl=path.ext	Default path for communication protocols in <i>NCTALK</i>
ncr=path	Default path for files created with <i>UPLOAD/READ</i> in <i>NCTALK</i>

path in the above specifications refers to the DOS path where the files reside. *path* may include a drive letter followed by the directory structure. If *path.ext* is indicated, this means that a user defined default extension may be specified in the path which will override the system default extension.

GRAFX II User Reference Manual Supplement

The following is an example of a possible path specification for DXF files:

dxf=d:\acad\dx

Here the default path for DXF files is the subdirectory **dx**f under the directory **acad** found on the **d** drive. If the drive letter of the path is the same as your **gx2** directory, you need not specify it.

Here are some more examples of path specifications:

geo=data

tll=tll

ncw=d:\nc\data.nc

The first example sets the default path for **.GEO** and **.DMB** files to the subdirectory **data** under your **gx2** directory or \gx2\data. The second example sets the default path to **.TLL** tooling files to the **tll** subdirectory under your **gx2** directory or \gx2\tll. The third example sets the default output of NCTALK WRITE operations to be written to the \nc\data subdirectory on the "D" drive and the output file will default to a **.nc** extension.

Setting a path for Post Processors with the **pmb=path** specification in the **PATHS.SYS** file will override a path to Post Processors set with the **set post=path** specification in the **AUTOEXEC.BAT** file.

When fetching a file of a type for which a default data path has been defined, the path will automatically be displayed in the file name entry window. If the `F5' key or LEFT MOUSE button is used to create a menu of candidate files, the file name menu will be built from the defined data path. If you wish to fetch a file from a path other than the defined default data path, you can edit the displayed path name when typing in the file name or use the `Shift/F5' key feature to specify the path to the candidate files.

The **PATHS.SYS** file may be created or modified in the **G2CONFIG** program.