ANILAM

6000M
CNC Control
TRAINING GUIDE
Navigation Instructions

Follow the bookmarks at the left side of the page to navigate to desired topic.

Click plus and minus symbols to expand and compress menu display.
5000M CNC CONTROL
TRAINING GUIDE
**Turning the Control ON**

After the control has been turned ON press F10 to continue.

1. **Control Software**
   - This allows you enter CNC software

2. **Setup Utility**
   - Machine configuration

3. **Motion Setup/Testing**
   - Used to setup drive

Then press ENTER to select CNC mode.
First CNC Screen

Control Manual Screen

<table>
<thead>
<tr>
<th>MACHINE</th>
<th>PROGRAM</th>
<th>TARGET</th>
<th>DIST. TO GO</th>
</tr>
</thead>
<tbody>
<tr>
<td>X = 0.000</td>
<td>X = 0.000</td>
<td>X = 0.000</td>
<td>X = 0.000</td>
</tr>
<tr>
<td>Y = 0.000</td>
<td>Y = 0.000</td>
<td>Y = 0.000</td>
<td>Y = 0.000</td>
</tr>
<tr>
<td>Z = 0.000</td>
<td>Z = 0.000</td>
<td>Z = 0.000</td>
<td>Z = 0.000</td>
</tr>
<tr>
<td>U = 0.000</td>
<td>U = 0.000</td>
<td>U = 0.000</td>
<td>U = 0.000</td>
</tr>
<tr>
<td>V = 0.000</td>
<td>V = 0.000</td>
<td>V = 0.000</td>
<td>V = 0.000</td>
</tr>
</tbody>
</table>

SECTION #1

SECTION #2

SECTION #3

TOOL: 0  RPM: 0  %: 100  LOOP: 0  FIXTURE: 0
DIA: 0.0000  FEED: 0.8  %: 100  Dwell: 0.0
LENGTH: 0.0000
OVERRIDE: FEED, RAPID
G: G01 G17 G40 G70 G90
M: M05 M09

Parts: 0  Timer: 00:00:00 (00:00:00)
Section #1

- Program name
- Machine status
- Mode of operation
- Machine position status

Messages display in this area
Command line for **MANUAL** (MDI) instructions

Section #2

<table>
<thead>
<tr>
<th>MACHINE</th>
<th>PROGRAM</th>
<th>TARGET</th>
<th>DIST. TO GO</th>
</tr>
</thead>
<tbody>
<tr>
<td>X + 0.0000</td>
<td>X + 0.0000</td>
<td>X</td>
<td>X + 0.0000</td>
</tr>
<tr>
<td>Y + 0.0000</td>
<td>Y + 0.0000</td>
<td>Y</td>
<td>Y + 0.0000</td>
</tr>
<tr>
<td>Z + 0.0000</td>
<td>Z + 0.0000</td>
<td>Z</td>
<td>Z + 0.0000</td>
</tr>
<tr>
<td>U + 0.0000</td>
<td>U + 0.0000</td>
<td>U</td>
<td>U + 0.0000</td>
</tr>
<tr>
<td>W + 0.0000</td>
<td>W + 0.0000</td>
<td>W</td>
<td>W + 0.0000</td>
</tr>
</tbody>
</table>

Position relative to machine HOME
Target or position to reach
Position relative to part ZERO
Distance to go to reach target
### Section #3

<table>
<thead>
<tr>
<th>Active tool #</th>
<th>Tool diameter</th>
<th>Tool length offset</th>
<th>Spindle speed</th>
<th>Feedrate</th>
<th>Percentage of feed and RPM’s</th>
<th>Number of loops</th>
<th>Dwell in seconds</th>
<th>Active fixture offset</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOOL: 0</td>
<td>RPM: 0</td>
<td>DIA: 0.0000</td>
<td>FEED: 0.0</td>
<td>%: 100</td>
<td></td>
<td>LOOP: 0</td>
<td>DUELL: 0.0</td>
<td>FIXTURE: 0</td>
</tr>
<tr>
<td>DIA: 0.0000</td>
<td>RPM: 0</td>
<td>DIA: 0.0000</td>
<td>FEED: 0.0</td>
<td>%: 100</td>
<td></td>
<td>LOOP: 0</td>
<td>DUELL: 0.0</td>
<td>FIXTURE: 0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Active G codes</th>
<th>Active M functions</th>
<th>Override of feed and rapid</th>
<th>Parts counter and timer</th>
</tr>
</thead>
<tbody>
<tr>
<td>G: G01 G17 G10 G70 G90</td>
<td>M: M05 M09</td>
<td></td>
<td>PARTS: 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TIMER: 00:00:00 (00:00:00)</td>
</tr>
</tbody>
</table>
Softkey in Manual Mode

Access help menu → Help
Edit program in memory → Program
Single Step Mode → Edit
Delete from command line → Manual
Access tool page → S.Step

Access list of programs → Auto
Manual Mode → Delete
Automatic Mode → Insert
Insert in command line → Tool
Activate handwheel → HandWl

Note: When handwheel is active colors will be reversed in box as shown below.

F10

Softkey in Manual mode when shift key is pressed

Display last 8 messages → Message
Teach Mode → Teach
Sends machine home → Home
Exits back main manual screen → Exit
**Alfa - Numeric key board**

Note: Most used key are yellow. Most key also double functions, the shift key is use to use secondary functions.
Manual panel

There are two types of Manual panels.
1. As shown below.
2. The other without Handwheel on left side.
MOVING WITH JOG MOVEMENTS

1) Turn the jog mode selector switch on the MANUAL PANEL to 100, 10, 1, FEED or RAPID
2) Select the axis to move with the AXIS SELECTOR switch at the MANUAL PANEL
3) Press the − or the + key to move in the desired direction.

MOVING WITH COMMANDS

Type commands as needed and press the START button located on the MANUAL PANEL

EXAMPLE:
Type: G0 G90 Z-5. and press Start Z axis goes to Z-5. in ABS and RAPID

EXAMPLE:
Type G97 S1000 M3 and press START to start the spindle at a fixed 1000 rpm.
MOST COMMON G-CODE COMMANDS TO REMEMBER:

G00: Rapid move.
G01: Feed move.
G02: Arc clockwise
G03: Arc counter clockwise
G17: XY plane
G18: XZ plane
G19: Yz plane
G40: Cutter comp off
G41: Cutter comp left
G42: Cutter comp right
G53 Ox: Fixture offsets (absolute shift from Machine Home).
G70: Inch programming
G71: MM programming
G80: Used for Drilling Cycles.
G90: Absolute co-ordinate system
G91: Incremental co-ordinate system
G92: Incremental shift
G94: Feed Rate in Inch per minute (mm per minute).
G95: Feed Rate in Inch per revolution (mm per revolution)
M0: Stop program.
M2: End of program
M3: Spindle forward
M4: Spindle reverse
M5: Spindle OFF
M8: Coolant ON
M9: Coolant OFF
ADDRESS LETTERS USED IN PROGRAMMING

A: Used for angles around X axis.
B: Used for angles around Y axis.
C: Used for angles around Z axis.
D: Designates tool diameter in a program.
F: Feedrate.
I: Circle center X axis.
J: Circle center Y axis.
K: Circle center Z axis.
L: Designates tool length in a program.
M: Miscellaneous functions.
N: Line number prefix.
O: Program or subroutine number prefix.
P: Subroutine number call prefix.
S: R.P.M. prefix.
T: Tool call.
U: Rotary axis or spindle.
V: Rotary axis or spindle.
W: Rotary axis or spindle.
LOOP: Repete operation.
END: End of loop.

Note: Some of these letters are used inside canned cycles also some that are not listed above.
**RETREIVING PROGRAMS FROM DISK**

**Transferring a program from a disk into the control**

Programs can be restored into the control if they have been previously saved on a disk.

Insert the disk into the floppy disk drive (A:) on the machine.

From the MANUAL mode press **Program**, then press **SHIFT**

Next, press **Log** select “A:” and press **ENTER**

Using the arrow keys hi-lite the program to be restored and press

**Utility** then press twice to select **COPY** and then **C:**

**WARNING**
Remove the diskette when complete and save in a safe place. Do not leave the diskette in the machine, the will not start correctly with disk in the drive.
Keys to use when using Computer key board

<table>
<thead>
<tr>
<th>FUNCTION</th>
<th>CNC KEYBOARD</th>
<th>COMPUTER KEYBOARD</th>
</tr>
</thead>
<tbody>
<tr>
<td>X Axis Command</td>
<td>X</td>
<td>X Key</td>
</tr>
<tr>
<td>Y Axis Command</td>
<td>Y</td>
<td>Y Key</td>
</tr>
<tr>
<td>Z Axis Command</td>
<td>Z</td>
<td>Z Key</td>
</tr>
<tr>
<td>Preparity G Code</td>
<td>G</td>
<td>G Key</td>
</tr>
<tr>
<td>Machine Function Code</td>
<td>M</td>
<td>M Key</td>
</tr>
<tr>
<td>Spindle Function Code</td>
<td>S</td>
<td>S Key</td>
</tr>
<tr>
<td>Tool Command</td>
<td>T</td>
<td>T Key</td>
</tr>
<tr>
<td>ENTER</td>
<td></td>
<td>ENTER Key</td>
</tr>
<tr>
<td>SHIFT</td>
<td></td>
<td>SHIFT Key</td>
</tr>
<tr>
<td>Cycle HOLD Key</td>
<td></td>
<td>Alt + H Key</td>
</tr>
<tr>
<td>Cycle START Key</td>
<td></td>
<td>Alt + S Key</td>
</tr>
<tr>
<td>Cursor UP, DOWN, LEFT and RIGHT</td>
<td></td>
<td>ARROW Keys</td>
</tr>
<tr>
<td>CLEAR Key</td>
<td></td>
<td>Alt + C Key</td>
</tr>
</tbody>
</table>

FUNCTION Keys F1 - F10
Definitions of Function keys are given on screen
5000M CNC CONTROL
HELP MENU’S
The **HELP MENU’S** are accessed by pressing **F1**

This can be done from either Manual or Edit.

**Manual mode soft keys**

```
[Help]  [Program]  [Edit]  [Manual]  [S.Step]  [Auto]  [Delete]  [Insert]  [Tool]  [HandWl]
```

**Edit mode soft keys**

```
[Help]  [Del]  [Ins]  [DelBlk]  [PgUp]  [PgDn]  [Move]  [Editing]  [Misc]  [Exit]
```

**First Help screen**

**Note:** The center of the screen. There are two different displays, one has text about Help, the other shows the program as it is being entered.

Press **F4** will toggle these screens.
There are two different set of soft keys, as shown above.

F1: Arrow up and around to desired selection.
F2: Arrow down and around to desired selection.
F3: Not used.
F4: Text changes center of screen to display program.
F5: Selects required cycle. Enter will do this.
F6: ReEdits a cycle after it is already in a program.
F7: Goes back to edit without saving last set of inputs.
F8: Accepts inputs and stays in HELP.
F9: Goes back to previous page in HELP menu.
F10: Exits to EDIT page and saves inputs.
Section as shown below these are the major defaults. Highlight the required input and press either **Select** or **enter**.

Use up arrow key to move highlight to next selection.
Press the number 2 key or arrow up to number 2.

Press either [Select] or enter

Note:- the reverse color around Compensation.

Press either [Select] or enter

1. Puts a G40 into program comp off.
2. Enters G41 into program cutter comp left.
3. Enters G42 into program cutter comp right.
4. G68 rotates a shape around a center.
5. G72 scale program to required size.
Rotation G68

Note: The only entry that has to be program is C because it has 0 (Zero's) next to it.

This cycle can be programmed in main program or in a subroutine. If programmed in the main, the cycle is entered and then the dimension of the shape and turned off with a G68.

This example is show rotation just one time not using a subroutine, note the G68 to turn off rotation.

Example on left show rotation using subroutine, note there is no G68 turning off rotation, it’s not required when programming this way.
When using scaling if there are any the axis must be scaled the same on both of these axis. If part is required to be half size .5 would be factor. G72 alone will turn off scaling.
Lines

Press # 3 either [Select] or [Enter].

Screen will now appear as below.

Inputs will change according to which plane is active.

2. X axis input only.
3. Y axis input only.
4. X and Y axis.
5. Angle and X axis.
6. Angle and Y axis.
7. Angle and Radius
8. Radius and X axis
9. Radius and Y axis
Arc’s

Press # 4 either select or enter.

Screen will now appear as below.

Tool must be at start point before inputting arc’s
Centers of arcs X=I, Y=J and Z=K
Inputs will change according active plane.

2. Radius and End Point.

3. Center and End Point. This can be used for helical interpolation (thread mill.)

4. Center and X End Point.

5. Center and Y End Point.

6. Center and Angle. Angle is dependant Absolute or incremental.

7. Arc and Line. Inputs are Radius, Angle and End Point X and Y

8. Line and Arc. Inputs are Angle, Radius and End Point X and Y

9. Arc and Arc. Inputs Center X and Y first arc, Center X and Y second arc and End Point X / Y
Corner Rounding And Chamfering

Press # 5 either [Select] or enter.

Screen will now appear as below.

2. Radius  One shot corner rounding. Inputs mid point X / Y, radius and end point X/Y.
3. Chamfer  One shot chamfer. Inputs mid point X/Y, chamfer and end point X/Y.
4. Corner Radius  Modal command puts radius on all intersects.
5. Corner Chamfer  Modal command puts chamfer on all intersects.
6. Cancel  Cancels #4 and #5
Multiple line, arc and chamfer moves.

Press # 6 either Select or enter.

Screen will now appear as below.

---

Tool must be positioned at start point. 
Inputs will change according to active plane.

2. Definition Inputs first angle, second angle and end point.

3. Radius Inputs first angle, radius, second angle and end point.

4. Chamfer Inputs first angle, radius, second angle and end point.

5. Rad/Rad Inputs first angle, first radius, second angle, mid point, second radius and end point.

6. Chamf/Chamf Inputs first angle, first radius, second angle,

7. Rad/Chamf Inputs first angle, radius, second angle, mid point X/Y chamfer and end point.

8. Chamfer/Rad Inputs first angle, chamfer, second angle, mid point X/Y, radius and end point.
Pockets

Press # 7 either or enter.

Screen will now appear as below.

Cutter comp is built into all pocket except Mold Rotation.
X and Y centers are Optional but if not entered will assume it is positioned at center of pocket.

#2. Frame Milling. Leave island in middle of pocket.
#3. Hole Milling. Enlarges existing hole, used on smaller holes.
#5. Rectangular Pocket. Cuts flat bottom rectangular pocket.
#6. Area Clearance. Cuts irregular shape pocket and takes profile cut.
#7. Mold Rotation. Rotates a profile around an axis.
+/-. Plunge pockets. Rectangular and circular pocket plunging straight down.

Plunge Pockets
#2. Circular Plunge Pocket Plunges straight into material.
#3. Rectangular Plunge Pocket. Plunges straight into material.
Frame pocket G75

Only the input with in Zeros have to be entered the rest are optional. It will assume being at the center of pocket if no dimension are entered.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X Center</td>
<td>Center of pocket X axis. If not entered will assume tool is at center of pocket. Optional</td>
</tr>
<tr>
<td>Y Center</td>
<td>Center of pocket Y axis. If not entered will assume tool is at center of pocket. Optional</td>
</tr>
<tr>
<td>Length</td>
<td>Length of island (X).</td>
</tr>
<tr>
<td>Width</td>
<td>Width of island (Y).</td>
</tr>
<tr>
<td>Start Hgt.</td>
<td>Start height. .1 inch or 2mm above top surface of pocket.</td>
</tr>
<tr>
<td>Z Depth (abs)</td>
<td>Absolute depth to bottom of pocket.</td>
</tr>
<tr>
<td>Stepover</td>
<td>Cut per pass, not to exceed 70% of cutter dia. Negative value path will climb mill.</td>
</tr>
<tr>
<td>Max. Z cut</td>
<td>Depth per pass Z axis. Optional</td>
</tr>
<tr>
<td>Ramp Feed</td>
<td>Feedrate when feeding down into pocket. Optional</td>
</tr>
<tr>
<td>Rough Feed</td>
<td>Feedrate roughing pocket. Optional</td>
</tr>
<tr>
<td>Inside Rad.</td>
<td>Radius on corners of island. Optional</td>
</tr>
<tr>
<td>Outside Rad.</td>
<td>Radius on outside, will assume cutter radius if no entry. Optional</td>
</tr>
<tr>
<td>Frame Width</td>
<td>Dimension from island to outside.</td>
</tr>
<tr>
<td>Finish Stock</td>
<td>Amount of material left for finish pass. Optional</td>
</tr>
<tr>
<td>Finish Feed</td>
<td>Feedrate for finish pass. Optional</td>
</tr>
<tr>
<td>Retract Hgt.</td>
<td>High retract allows tool to be move above the surface part when finished. Optional</td>
</tr>
</tbody>
</table>

Hole Milling G76

A good use for this cycle to produce small counterbores.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>Diameter of pocket.</td>
</tr>
<tr>
<td>Rough Feed</td>
<td>Feedrate roughing.</td>
</tr>
<tr>
<td>Finish Stock</td>
<td>Finish Stock.</td>
</tr>
<tr>
<td>Finish Feed</td>
<td>Feedrate finish pass.</td>
</tr>
</tbody>
</table>
Circular pocket G77

X Center  X  Center X axis.  Optional
Y Center  Y  Center Y axis.  Optional
Z Start Hgt.  H  Start height .1 above surface to be cut into.
Z Depth (abs)  Z  Absolute Z depth.
Diameter  D  Diameter of pocket, comp built in.
Stepover  A  Move over per pass.  Optional
Z Max. cut  B  Max. Z depth per pass.  Optional
Rough Feed.  I  Feedrate roughing.  Optional
Finish Stock  S  Amount of material left for finish pass.  Optional
Finish Feed.  K  Finish pass feedrate.  Optional
Retract Hgt.  P  High return when finished.  Optional

Rectangular Pocket. G78

X Center  X  Center of pocket X
Y Center  Y  Center of pocket Y
Length  M  Actual length X axis
Width  W  Actual width Y axis
Z Start Hgt.  H .1 above surface to be cut
Z Depth (abs)  Z  Absolute depth of pocket
Corner Rad.  U  Radius in corners
Stepover  A  70% of cutter or less
Z Max. Cut  B  Max depth per pass
Ramp Feed  I  Feedrate on 3 axis first move
Rough Feed.  J  Feedrate for roughing
Finish Stock  S  Amount of stock for finish cut
Finish Feed  K  Finish feedrate
Retract Hgt.  P  Retract after finished.
Area Clearance  G169.

Input Sub #  W  Subroutine number.
X  X  X position tool will Z down into part
Y  Y  Y position tool will Z down into part
Z Start Hgt.  H  Start height .1 above top of pocket
Z Depth (abs)  Z  Total depth of pocket absolute
Cut Angle  C  Used if starting in middle of radius
X Start  D  Position of cut at start X axis.  Optional
Y Start  E  Position of cut at start Y axis.  Optional
Stepover  A  Cutter stepover each pass
Z Max. Cut  B  Max depth of cut per pass.
Ramp Feed  I  Ramp feedrate Z down
Rough Feed  J  Rough feedrate
Finish Stock  S  Stock left for finish pass
Finish Feed  K  Finish feedrate
Retract Htg.  P  Retract after finished.

Mold Rotation G45

Start Angle  A  Angle where rotation is going to start
End Angle  B  Angle where rotation is going to end
Num of Cycles  C  1 cycle equals 1 Fwd and 1 Rev. Sub.
Num of Fwd Sub  F  Sub. Profile forward direction
Num of Rev Sub  R  Sub. Profile Reverse direction
Axis of Rotation  X  Axis rotation is around X,Y or Z
Other axis CL  I  Center line X or Y if not Zero
Centerline  J  Center line Z Axis if not Zero
Rotation Angle  K  Angle rotation Z axis only
Elbow Milling G49

| Start Radius | B | Radius at start of Elbow |
| End Radius | K | Radius at end of Elbow |
| Included Angle | A | Included angle |
| Num of Cycles | C | Sub. Profile forward direction |
| X Center | I | Sub. Profile Reverse direction |
| Y Center | J | Axis rotation is around X,Y or Z |
| Direction CCW+ | D | Center line X or Y if not Zero |
| End Angle | F | Center line Z Axis if not Zero |
| Start Angle | E | Angle rotation Z axis only |
| Ctr. Line Radius | R | Radius at center of elbow |
| Rapid Height | Z | Starting height above surface |
| Start Height | H | Z height to start |
| Rough Feed | U | Rough feedrate |
| Finish Stock | S | Amount of stock for finish pass |
| Finish Feed | V | Feedrate for finish cut |

Note: Tool must be positioned at center of radius bottom left corner. Center of pocket must be cleared before using this cycle. When using flat endmill will go to programmed depth. If ball endmill uses will only go to depth minus cutter radius, .5 mill Absolute depth -1 actual depth it would go to is -.75.

Draft Pocket G73

| Length, bottom | X | Length at bottom of pocket. required |
| Width, bottom | Y | Width at bottom of pocket. required |
| Start Height | H | Height above part to rapid. required |
| Z Depth (abs) | Z | Absolute depth required |
| Lower Left Rad. | A | Lower left radius required |
| Lower Right Rad. | B | Lower right radius optional |
| Upper Left Rad. | C | Upper left radius optional |
| Upper Right Rad. | D | Upper right radius optional |
| Draft Angle | E | Draft angle Degrees required |
| Z step Rough | I | Depth per pass in Z axis required |
| Max XY Stepover | V | Maximum stepover XY optional |
| Finish Stock XY | S | Finish stock XY optional |
| Z step Finish | Q | Z step finish pass optional |
| Finish Feed | R | Finish feedrate optional |
| Flat 0, Ball 1 | W | Flat mill = 0 Ball mill = 1 optional |
Plunge Circular Pocket G177

**G177 PLUNGE CIRCULAR POCKET MILLING**

**NOTE:** Positions tool at center (or 3 o'clock) of pocket, Neg. diameter = CW.

- **X Center** X  Center X axis  Optional
- **Y Center** Y  Center Y axis. Same as above.  Optional
- **Z Start Hgt.** H  Start height .1 above surface to be cut.
- **Z Depth (abs)** Z  Absolute Z depth.
- **Diameter** D  Diameter of pocket, comp built in.
- **Stepover** A  Move over per pass.  Optional
- **Z Max. cut** B  Max. Z depth per pass.  Optional
- **Z Feedrate** I  Feedrate plunging  Optional
- **Rough Feed.** J  Feedrate roughing.  Optional
- **Finish Stock** S  Amount of material left for finish pass.  Optional
- **Finish Feed.** K  Finish pass feedrate.  Optional
- **Retract Hgt.** P  High return when finished.  Optional

Plunge Pocket Pocket G178

**G178 PLUNGE RECTANGULAR POCKET MILLING**

- **X Center** X  Center of pocket X
- **Y Center** Y  Center of pocket Y
- **Length** M  Actual length X axis
- **Width** W  Actual width Y axis
- **Z Start Hgt.** H  .1 above surface to be cut
- **Z Depth (abs)** Z  Absolute depth of pocket
- **Corner Rad.** U  Radius in corners
- **Stepover** A  70% of cutter or less
- **Z Max. Cut** B  Max depth per pass
- **Ramp Feed** I  Feedrate on 3 axis first move
- **Rough Feed** J  Feedrate for roughing
- **Finish Stock** S  Amount of stock for finish cut
- **Finish Feed** K  Finish feedrate
- **Retract Hgt.** P  Retract after finished.
Pockets with Islands (G162)

**Format:** G162 An Bn Cn Dn En

This cycle allows islands in irregular pockets. The main pocket must have the lowest subroutine number. Normally, this would be one (1). Pockets with Islands can be programmed using:

- DXF (see “Section 17, Using DXF for Pockets with Islands (G162)"
- CAM (see “Section 18, Example #12 Using CAM for Pockets with Islands (G162)"
- Subroutines

More than one G162 Island cycle can be programmed at a time. They may be strung together, but on separate lines. Islands can be programmed inside of islands. Five islands can be put on a line. The shape number subroutine number is used as inputs.

Refer to **Table 5-21**.

Activate a tool prior to programming G78, so cutter diameter is known.

**Table 5-21, G162 Address Words**

<table>
<thead>
<tr>
<th>Address Word</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>First island. Required.</td>
</tr>
<tr>
<td>B</td>
<td>Second island. Optional.</td>
</tr>
<tr>
<td>C</td>
<td>Third island. Optional.</td>
</tr>
<tr>
<td>D</td>
<td>Fourth island. Optional.</td>
</tr>
<tr>
<td>E</td>
<td>Fifth island. Optional.</td>
</tr>
</tbody>
</table>

**Using Subroutines for Pockets with Islands**

The program below is the same one used in the DXF portion with subroutines added for the letters. In the third G162 some of the numbers have negative sign (-) in front of them, this changes the side of the cutter comp for the islands in islands. See **Figure 5-13, Subroutines Pockets with Islands Example Workpiece** and **Table 5-22, Pockets with Islands Subroutines Programming Example**.
Figure 5-13, Subroutines Pockets with Islands Example Workpiece

Table 5-22, Pockets with Islands Subroutines Programming Example

| N1   | G00 G17 G70 G90 |
| N2   | T1D.0205 L-1 M6 |
| N3   | S1250M3         |
| N4   | M8              |
| N5   | G53O01          |
| N6   | G162 A2 B3 C4 D5 E6 |
| N7   | G162 A7 B8 C9 D10 |
| N8   | G162 A-11 B12 C-13 D-14 |
| N9   | G169 W1 H0.1 Z-0.0050 C299. A0.0080 I5.0 J12.0 S0.0010 K10.0 P1 |
| N10  | M2              |
| N11  |                |
| N12  | O11             |
| N13  | X1.1044 Y0.5   |
| N14  | Z0.1            |
| N15  | G01 Z-0.005     |
| N16  | G01 X1.159 Y0.65 |
| N17  | X1.2052         |
| N18 | X1.2598 Y0.5 |
| N19 | X1.2226 |
| N20 | X1.2135 Y0.525 |
| N21 | X1.1507 |
| N22 | X1.1416 Y0.5 |
| N23 | X1.1044 |
| N24 | G00 Z0.1 |
| N25 | M99 |
| N26 | |
| N27 | |
| N28 | O12 |
| N29 | X1.1634 Y0.56 |
| N30 | Z0.1 |
| N31 | G01 Z0.005 |
| N32 | G01 X1.1821 Y0.6112 |
| N33 | X1.2007 Y0.56 |
| N34 | X1.1634 |
| N35 | G00 Z0.1 |
| N36 | M99 |
| N37 | |
| N38 | O13 |
| N39 | X1.4007 Y0.55 |
| N40 | Z0.1 |
| N41 | G01 Z0.005 |
| N42 | G01 X1.3612 |
| N43 | G02 Y0.6 I-0.0312 J0.025 |
| N44 | G01 X1.4007 |
| N45 | G03 Y0.55 I-0.0707 J-0.025 |
| N46 | G00 Z0.1 |
| N47 | M99 |
| N48 | |
| N49 | O14 |
| N50 | G0 X1.42Y.5 |
| N51 | G1 X1.42Y.65 |
| N52 | G1 X1.460 Y.65 |
| N53 | G1 X1.460 Y.5 |
| N54 | G1 X1.42 Y.5 |
| N55 | M99 |
#2. Ellipse. Produces an ellipse, uses special cutter comp.
#5. Facing. Faces large surfaces.
#7. Circular Profile Cut circle either inside or outside.
#8. Rectangular Profile. Cut rectangle inside or outside.
**Ellipse G05**

*Note:* All dimensions are INCREMENTAL.

Cutter comp for ellipse uses M1040 X0 = off, X1 = outside and X2 = inside. Cutter must be positioned in compensated position before ellipse is programmed.

**Spiral G06**

*Note:* All dimensions are INCREMENTAL.

No compensation available for spiral.
Facing G170

G170 FACING

NOTE: Must specify A or B but not both.

<table>
<thead>
<tr>
<th>Length</th>
<th>X</th>
<th>Increamental length X axis.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width</td>
<td>Y</td>
<td>Increamental width Y axis.</td>
</tr>
<tr>
<td>X Stepover</td>
<td>A</td>
<td>Stepover X.</td>
</tr>
<tr>
<td>Y Stepover</td>
<td>B</td>
<td>Stepover Y.</td>
</tr>
<tr>
<td>Feedrate</td>
<td>F</td>
<td>Feedrate.</td>
</tr>
<tr>
<td>Z Start Hgt.</td>
<td>H</td>
<td>Start height .1 above surface.</td>
</tr>
<tr>
<td>Z Depth (also)</td>
<td>Z</td>
<td>Finish depth.</td>
</tr>
<tr>
<td>X Start</td>
<td>D</td>
<td>Start X axis.</td>
</tr>
<tr>
<td>Y Start</td>
<td>E</td>
<td>Start Y axis.</td>
</tr>
</tbody>
</table>

Note: Only A or B not both can be used.
Cutter will step away from start corner by half the cutter diameter.
Circular profile G171

Center X.  X  X Center (optional).
Center Y.  Y  Y Center (optional).
Start height.  H  Start height above surface to be cut.
Diameter.  D  Diameter of pocket (actual).
Z depth (absolute).  Z  0 = inside, 1 = outside of circle.
Ramp Distance.  R  Size of ramp on radius.
Z Maximum cut.  B  Maximum depth in Z per pass.
Finish Stock.  S  Amount of stock left for finish cut.
Z Feedrate.  I  Feedrate in Z axis.
Retract height.  P  High retract if higher than H value.

Rectangular Profile G172

Center X.  X  X Center (optional).
Center Y.  Y  Y Center (optional).
Start height.  H  Start height above surface to be cut.
Length  M  Length of pocket X axis (actual).
Width.  W  Width of pocket Y axis (actual)
Z depth (absolute).  Z  0 = inside, 1 = outside of circle.
Ramp Distance.  R  Size of ramp on radius.
Corner radius  U  Radius in corners.
Z Maximum cut.  B  Maximum depth in Z per pass.
Finish Stock.  S  Amount of stock left for finish cut.
Z Feedrate.  I  Feedrate in Z axis.
Retract height.  P  High retract if higher than H value.
Thread Mill Cycle (G181)

**Format:** G181 Xn Yn Zn Hn Pn Dn Cn Bn Rn Sn En Jn Kn Vn

**WARNING:** The first move in this cycle is a rapid move to the center of the thread before moving the Z axis. Make sure the tool is properly located before calling up this cycle.

Use the thread milling for cutting inside or outside threads. It will cut either Inch or MM, left or right hand, and Z movement up or down. A single tooth or multi-toothed tool may be used. Start can be at the top or bottom of the hole or boss. The tools are set, as you would normally set TLO.

**Programming the Thread Mill Cycle**

To program the Thread Mill Cycle:

1. In Edit mode, press **Help (F1)**, select **PATHS** from the menu, then **"THREAD MILL"** to display the G181 Thread Mill pop-up menu (refer to Figure 5-20).

2. Complete the entry fields (refer to **Table 5-28, G181 Address Words**), and press **EXIT (F10)**.

Depending on the parameters used and their values, when cutting a thread, the tool can ramp into the cut as illustrated in Figure 5-20. See **Table 5-28, G181 Address Words**.

---

**Figure 5-20, Thread Mill Pop-up Menu**
Table 5-28 describes the **Thread Mill Cycle** entry fields.

### Table 5-28, G181 Address Words

<table>
<thead>
<tr>
<th>Address Word</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>X</strong></td>
<td>Absolute X coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)</td>
</tr>
<tr>
<td><strong>Y</strong></td>
<td>Absolute Y coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)</td>
</tr>
<tr>
<td><strong>Z</strong></td>
<td>Absolute Z position where the thread cut will finish. This can be above or below the start position depending on the direction of the thread cut: up or down. (Required)</td>
</tr>
<tr>
<td><strong>H</strong></td>
<td>Absolute Z position where the thread cut starts. This can be above or below the finish position depending on the direction of the thread cut, up or down. If not set, cycle will use the current Z tool position. (Optional)</td>
</tr>
</tbody>
</table>
| **P**        | An Absolute safe Z position above the part for rapid moves in X and/or Y. (Required)  
**Warning:** P must be above the part to avoid a crash while positioning. |
| **D**        | Major thread Diameter. If this is a tapered thread, it is the major diameter at the Z start position. Hence, if you have a tapered hole and you start at the top and cut down, you would have a different major diameter than if you started at the bottom and cut up. A plus (+) value cuts in the CW direction and a minus (-) value cuts in the CCW direction. (Required) |
| **C**        | Depth of thread. The incremental depth of thread on one side. A plus (+) value is inside thread, a minus (-) value is outside thread. (Required) |
| **B**        | Threads per inch (TPI) or lead of thread in MM. Note: The minimum number of threads per inch is “1”. (Required) |
| **R**        | Size of radius arcing into start of thread. (Optional)  
**Note:** If R is a positive value or not set and the thread is “inside”, the cycle will always return to the center between passes.  
If R is a negative value, the cutter will move to the start or end point that is closest to the center if inside thread, and farthest away from center if outside thread.  
If R is not specified at all and the thread is outside, the cutter will back away from the largest diameter by an amount equal to the thread depth. |
| **S**        | Amount to leave for a finish pass after the roughing passes. (Optional) |
| **E**        | Number of roughing cuts to be taken. (Optional)  
**Note:** If “Stock” is not set or set to zero and E is 1 or 0, the cycle will make just one pass at the full depth.  
If “Stock” is set to greater than zero and E is 1 or 0, the cycle will make one pass at the stock depth and one pass at full thread depth.  
**Note:** If you would like all non-cutting positioning moves to be rapid, set E to a negative number. |

(Continued...)
Table 5-28, G181 Address Words

<table>
<thead>
<tr>
<th>Address Word</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>J</td>
<td>Feedrate for roughing. (If not set (blank), the cycle will use the current active feedrate) (Optional)</td>
</tr>
<tr>
<td>K</td>
<td>Feedrate for the finish pass. (If not set (blank), the cycle will use the current active feedrate) (Optional)</td>
</tr>
<tr>
<td>V</td>
<td>Angle on one side of the thread, (not the included angle). The angle is measured from the right side going counter clockwise with a positive number and clockwise with a negative number. A standard pipe tape with an inside cut would be -1.7833. If not set (blank), than the thread is straight. (Optional)</td>
</tr>
</tbody>
</table>

Tool Length Offset is set the same as with any other tool or operation. A tool diameter also has to be set in the tool table, as cutter compensation is built into this cycle (cutter compensation is not allowed during the use of this cycle).

If X and Y are not programmed, position tool center of the thread before the G181 line:

- X and Y will rapid to the starting position of the thread.
- Z will rapid to the safe height specified in “P”.
- The Z axis will feed down to the start cut position “H”. This could be above or below the Z position specified in the “Z” finish position.
- Depending on what is in the “R” parameter the tool will arc into the first cut position.
- Spiral up or down, depending on the difference between “Z” and “H” and go counterclockwise or clockwise depending if “D” is plus or minus.
- Then arc-out and feed to the thread center for inside threads or a safe distance away from the thread for outside threads depending on the value in “R”.
- Then feed back to the “H” height.
- Then feed X and Y to the next depth of cut. The depth of each roughing pass will be the thread depth specified in the “C” parameter minus the stock amount specified in the “S” parameter, divided by the number of roughing passes specified in the “E” parameter.
- The cycle repeats this process until the final finish pass.
- It will then cut the thread at the full thread major diameter.

When cutting a taper on an inside thread, care should be taken. An error will be generated if the diameter on the small end of the taper becomes too small for the tool to fit along with arc in and out moves. Not entering and arc-in value in the “R” parameter will allow the cycle to move to the center of the hole for maximum clearance.
Sample Thread Milling Cycle Program

1   G0 G90 G70 G17
2   T1 M6
3   S2000 M3
4   X0 Y0
5   G181 Z-1. H0.1 P.5 D1. C.0625 B8. R.1 S.002 E2 J20.0 K5.0
6   Z5
7   M5
8   M2

With a cutter diameter of 0.625, this program will cut a 1-8 inside thread at X0 Y0. The tool will spiral down the thread pitch of 8 threads per inch, finishing at a depth of –1. The starting height is 0.1, the safe rapid Z height is 0.5, the major thread diameter is 1 inch, and depth of thread is 0.0625. The arc-in radius is 0.1 and the stock amount for the finish pass is 0.002. The rough feedrate is 20.0 and the finish feedrate is 5.0.

Note: If you would like all non-cutting positioning moves to be rapid, set “E” to a negative number. The idea is to initially set “E” as a positive number and after proving out the program, change “E” to a negative number for faster production. If you only need one pass to size and you want the positioning moves to be rapid, set “E” to -1.
Drilling Cycles G80 Series

<table>
<thead>
<tr>
<th>COUNTERBORE</th>
<th>PECKING</th>
<th>TAPPING</th>
<th>BORE/BI</th>
<th>BORE/UNI</th>
<th>CHIP BREAK</th>
</tr>
</thead>
<tbody>
<tr>
<td>G02</td>
<td>G03</td>
<td>G04</td>
<td>G05</td>
<td>G06</td>
<td>G07</td>
</tr>
</tbody>
</table>

| CANCEL G80 | 1 |

<table>
<thead>
<tr>
<th>DRILLING</th>
<th>If they are not active, from group 1 Plane, inch/metric, and absolute/incremental can be selected.</th>
</tr>
</thead>
<tbody>
<tr>
<td>G81</td>
<td>Move the cursor around the screen with the arrow keys or by number required. Press SELECT key to pick the item. Combinations of items are not allowed in the same block and if selected, an error message will be displayed. Press ACCEPT to insert data into the program and remain in Help, press EXIT to insert data into the program and return to the Editor.</td>
</tr>
<tr>
<td>Incr.</td>
<td>G91</td>
</tr>
<tr>
<td>Absolute</td>
<td>G90</td>
</tr>
<tr>
<td>MM</td>
<td>G71</td>
</tr>
<tr>
<td>Inch</td>
<td>G70</td>
</tr>
<tr>
<td>YZ Plane</td>
<td>G19</td>
</tr>
<tr>
<td>XZ Plane</td>
<td>G18</td>
</tr>
<tr>
<td>XY Plane</td>
<td>G17</td>
</tr>
</tbody>
</table>

All of the G00 series cycles are modal and will activate the Z axis at each X,Y coordinate until canceled with a G00 code. 

**NOTE:** Must program drill cycle before bolt circle.

*Note:* That there is now a G80 in the first box on left. All G80 cycles must be turn OFF with a G80 as soon as drill operation is finished.
Basic Drilling Cycles G81

Finish Depth. Z Finish depth of hole.
Start Hgt. R Start height above surface to be drilled.
Feedrate. F Feedrate for drilling.
Return Hgt. P Return height if higher than R plane.

Counter Boring Cycles G82

Finish Depth. Z Finish depth of hole.
Start Hgt. R Start height above surface to be drilled.
Feedrate. F Feedrate for drilling.
Dwell time D Dwell time at bottom of hole.
Return Hgt. P Return height if higher than R plane.

Peck Drilling Cycles G83

Finish Depth. Z Finish depth of hole.
Start Hgt. R Start height .1 above surface to be drilled.
Feedrate. F Feedrate for drilling.
Maximun Peck I Maximum peck before retracting.
Return Hgt. P Return height if higher than R plane.
Tapping Cycles G84

Finish Depth.  Z  Finish depth of hole.
Start Hgt. R  Start height above surface to be drilled.
TPI/Lead. F  TPI if inch/Lead if MM.
Spindle. S  Spindle sync. 0 = OFF, 1 = ON
Return Hgt. P  Return height if higher than R plane.
Dwell time D  Dwell at bottom if necessary.

Boring Cycles G85

Finish Depth.  Z  Finish depth of hole.
Start Hgt. R  Start height above surface to be drilled.
Feedrate. F  Feedrate for drilling.
Return Hgt. P  Return height if higher than R plane.
Dwell time D  Dwell at bottom if necessary.

Boring Cycles One Direction G86

Finish Depth.  Z  Finish depth of hole.
Start Hgt. R  Start height above surface to be drilled.
Feedrate. F  Feedrate for drilling.
X Backoff I  Backoff before retracting from hole.
Dwell time D  Dwell to flat bottom hole.
Return Hgt. P  Return height if higher than R plane.
Index Angle C  Index angle to orient spindle to backoff.
Chip Breaking Cycle G87

Finish Depth. Z Finish depth of hole.
Start Hgt. R Start height .1 above surface to be drilled.
Feedrate. F Feedrate for drilling.
First Peck. I Amount of first peck.
Delta Peck. J Amount to decrease peck each peck.
Minimum Peck. K Smallest peck amount.
Retract Depth. U Depth full retract accures.
Return Hgt. P Return height if higher than R plane.

Flat Bottom Boring Cycle G89

Finish Depth. Z Finish depth of hole.
Start Hgt. R Start height above surface to be drilled.
Feedrate. F Feedrate for drilling.
Dwell. D Dwell in second at bottom of hole.
Return Hgt. P Return height if higher than R plane.
Bolt Hole Circle Drilling G79

- Center: X Center X axis.
- Center: Y Center Y axis.
- Index Angle: C Angle to rotate 0 angle from 3 o'clock.
- First Angle: A Angle of first hole from 0.
- Last Angle: B Angle of last hole, if full pattern not required.
- Number Holes: H Number of holes to drill.
- Diameter: D Diameter of pattern.
- Radial Path: R If 1 is entered will move radially around pattern.

Hole Pattern Drilling G179

- X Start: X Start point X axis.
- Y Start: Y Start point Y axis.
- Angle: C Angle if pattern is rotated.
- X Length: A Distance from first to last hole X axis.
- Y Width: B Distance from first to last hole Y axis.
- Num. Holes X: D Number of holes X axis.
- Num. Holes Y: E Number of holes Y axis.
- X Increment: U Distance between holes X axis.
- Y Increment: V Distance between holes Y axis.
- Pat.=0 Sqr.=1 W Pattern as shown or square around outside.

Use D & E or U & W not both.
**G-Code without Graphics**

**MISCELLANEOUS G CODES**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G04</td>
<td>Dwell In seconds with Tn</td>
</tr>
<tr>
<td>G09</td>
<td>Exact stop will stop exactly in position one shot.</td>
</tr>
<tr>
<td>G22</td>
<td>Stroke Limit.</td>
</tr>
<tr>
<td>G29</td>
<td>Reference Point Return.</td>
</tr>
<tr>
<td>G53</td>
<td>Fixture Offset(s) (Coord. Syst. Select)</td>
</tr>
<tr>
<td>G61</td>
<td>Exact Stop Mode (Contouring Mode OFF).</td>
</tr>
<tr>
<td>G64</td>
<td>Contouring Mode (Exact Stop Mode OFF).</td>
</tr>
<tr>
<td>G65</td>
<td>Macro Call, Single (Non-Modal).</td>
</tr>
<tr>
<td>G66</td>
<td>Macro Call, Modal.</td>
</tr>
<tr>
<td>G67</td>
<td>Cancel Modal Macro.</td>
</tr>
<tr>
<td>G92</td>
<td>Preset Zero.</td>
</tr>
<tr>
<td>G94</td>
<td>Per Minute Feed.</td>
</tr>
<tr>
<td>G95</td>
<td>Per Revolution Feed.</td>
</tr>
</tbody>
</table>

**Miscellaneous M-Codes**

**M-CODES**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0</td>
<td>Program Stop.</td>
</tr>
<tr>
<td>M2</td>
<td>End of Program.</td>
</tr>
<tr>
<td>M3</td>
<td>Spindle ON FWD.</td>
</tr>
<tr>
<td>M4</td>
<td>Spindle ON REV.</td>
</tr>
<tr>
<td>M5</td>
<td>Spindle OFF.</td>
</tr>
<tr>
<td>M6</td>
<td>Coolant ON.</td>
</tr>
<tr>
<td>M9</td>
<td>Coolant OFF.</td>
</tr>
<tr>
<td>M30</td>
<td>Jump to New Program.</td>
</tr>
<tr>
<td>M98</td>
<td>Call Subprogram.</td>
</tr>
<tr>
<td>M99</td>
<td>End Subprogram.</td>
</tr>
<tr>
<td>M100</td>
<td>Mirror Image.</td>
</tr>
<tr>
<td>M105</td>
<td>Dry-Run, All Axes.</td>
</tr>
<tr>
<td>M106</td>
<td>Dry-Run, NO Z Axis.</td>
</tr>
<tr>
<td>M107</td>
<td>Dry-Run OFF (Cancel M105, M106).</td>
</tr>
<tr>
<td>M100</td>
<td>Dry Run All Axis display shows motion but no table movement.</td>
</tr>
<tr>
<td>M107</td>
<td>Dry Run NO Z Axis X and Y move no Z axis movement.</td>
</tr>
<tr>
<td>M108</td>
<td>Dry Run OFF turns OFF dry run.</td>
</tr>
</tbody>
</table>
5000M CNC Control
Program Management
INTRODUCTION

The Program Directory provides access to all the program management and disk utilities. These functions include Creating, Selecting, Deleting, Undeleting and Copying programs. The Program Directory also provides access to the Floppy Drive utilities.

Accessing PROGRAM DIRECTORY page.

(1) From the MANUAL mode press: Program The Program files are listed in alphabetical order.

To access a PROGRAM file from the Program page.

There are 2 methods to access a particular Program File:

(1) Using the keys move the High Light to the desired program and Press: Edit

or

(2) Press the first letter of the desired program name, this will move the High Light to the first program name with that letter, then use the keys to move the High Light to the desired program and Press: Edit

To CREATE a new Program.

NOTE: There are 2 methods of creating a program file, both are shown below.

Method No.1 Creating a new program file. From the PROGRAM page:

(1) Press: Create At the prompt type in the new program name

NOTE: Program names can be up to 8 characters in length, but may not include spaces or periods.
Method No.2  Copying and using an existing program.

If a similar program to that required already exists, it may be copied and given a new name.
This allows similarly formatted programs to be used without having to re-type the information.

(1) Using the keys High Light the existing program to be copied

Press:  Utility  this brings up a menu, with Copy High Lighted

Press  Other  This will display a second menu High light Other and Press

This will bring up a third menu type in the new program name

“EXAMPLE” and Press  this will create a new program named EXAMPLE and also leave the

original program “SAMPLE1.G” intact. This copied program can now be edited to suit the new part.
It is not necessary to type the file extension (.G) as this is completed automatically.

To change or EDIT an existing program.

(1) High Light the program to be changed / edited using the keys.

(2) Press:  Edit  to enter the file editor.

To DELETE an existing program.

(1) High Light the program to be DELTED using the keys. Press  Delete

(2) To avoid accidental deletion of programs the system requests verification of deletion

To delete the program Press  Yes
To UNDELETE / RESTORE a deleted program.

Should a program become accidentally deleted the following method can be used to restore the program.

NOTE: The restore capability of the system is determined by the space available on the drive. If the space on the drive where the program was stored has been reused restore will not be an available function.

1. From the PROGRAM page Press: Utility

2. Using the keys, High Light Restore Press

3. The system will list any and all programs that are available to be restored. Using the keys High Light the program to be restored.

4. Type in the first letter of the program to be restored (the first letter of the program name is lost when a program is deleted).

5. Press Cont if the program can be restored the system will place the program file in its alphabetical location.

NOTE: ALWAYS CHECK THE VALIDITY OF A PROGRAM AFTER RESTORING. SOME OF THE INFORMATION WITHIN THE PROGRAM MAY HAVE CHANGED.

To SELECT a program to run in the AUTO mode.

Once a program has been created and verified to be correct the operator must SELECT the program to run when in the Auto or Single Step mode.

1. From the PROGRAM page using the keys High light the program name.

2. Press Select the selected program to be run will now be displayed in the lower right corner of the screen. Selected Program: EXAMPLE.G
To COPY a program to the A: Drive (Floppy Disk).

The 5000M control uses the C:\ drive and the USER directory to store programs within the system. When in the PROGRAM directory the drive and directory are displayed in the lower left corner of the screen.

The 5000M uses the A:/ drive for the Floppy Disk Drive. Programs may be copied to or copied from the Floppy Drive individually or in multiples.

1. From the PROGRAM directory use the keys to highlight the program to be copied to the A:\ (Floppy Disk) drive. (Place a floppy disk in the A:\ floppy disk drive unit).

2. If multiple programs are to be copied highlight the first program to be copied using the keys and Press . Then using the keys again highlight the next program to be copied and Press Continue until all programs to be copied are highlighted.

3. When all programs to be copied are highlighted Press Utility

4. With Copy highlighted Press Utility

5. This brings up a menu With A: High lighted Press

The lower left corner of the screen displays COPYING... while the program is copied. The procedure is complete when COPYING... is no longer displayed.
To COPY a program from the A: Drive (Floppy Disk) to the Program Directory C:\USER

The 5000M uses the A:\ drive for the Floppy Disk Drive. Programs may be copied to or copied from the Floppy Drive individually or in multiples. To view or copy the programs on a Floppy Disk, the operator must first LOG to the Floppy disk Drive (A:\)

1. From the PROGRAM directory Press (Shift) this will change the descriptions of the F keys, Press Log This shows a menu of the drives that the operator can view on screen.

2. High light A: and Press the programs on the Floppy Dive (A:\) will now be displayed.

3. Use the keys to High light the program(s) to be copied to the Program Directory (C:\)

If multiple programs are to be copied High light the first program to be copied using the keys and Press Then using the keys again High light the next program to be copied and Press Continue until all programs to be copied are High lighted.

4. When all programs to be copied are High lighted Press Utility

5. With Copy High lighted Press A menu is shown

With C: High lighted Press The lower left corner of the screen displays COPYING. while the program(s) are copied. The procedure is complete when COPYING…. is no longer displayed.
Continued…

To return to the Program Directory, LOG back to C:\USER

(1) Press (Shift) this will change the descriptions of the F keys,

(2) Press Log This shows a menu of the drives that the operator can view on screen.

(2) High light C: and Press the C:\USER (lower left corner of the screen)

will now be displayed.

To COPY program files to the A: (Floppy Disk) when using the CAM

When using the CAM Editor to write programs the 5000M system will create additional program files, and ALL these files MUST BE COPIED to ensure correct operation of the program. The program name to which the CAM is to be associated MUST be High lighted prior to entering CAM or (F4 from the Program Directory); this program will have extension of .G The correct program must be high light because upon entering the CAM, the system creates a file with the same name as the program but with an extension of .CAM and also a file with the extension of .GEO These files are used to record the CAM (.CAM file) settings inside the CAM and also to record the GEOMETRICAL ELEMENTS created (.GEO file).

Once a Shape has been created a 3rd file is also created, this file will have an extension of .1

NOTE: If more than one Shape is created the shape files will have extensions of .2 .3 etc. assigned in the order in which they were created.

Example:

<table>
<thead>
<tr>
<th>Original Program</th>
<th>PROGRAM.G</th>
</tr>
</thead>
<tbody>
<tr>
<td>File created by editor</td>
<td>PROGRAM.CAM</td>
</tr>
<tr>
<td>File created by editor</td>
<td>PROGRAM.GEO</td>
</tr>
<tr>
<td>First Shape file created</td>
<td>PROGRAM.1</td>
</tr>
<tr>
<td>Second Shape file created</td>
<td>PROGRAM.2</td>
</tr>
<tr>
<td>Third Shape file created</td>
<td>PROGRAM.3 etc.....</td>
</tr>
<tr>
<td>First Tool Path</td>
<td>PROGRAM.T1</td>
</tr>
<tr>
<td>Second Tool Path</td>
<td>PROGRAM.T2</td>
</tr>
<tr>
<td>Third Tool Path</td>
<td>PROGRAM.T3 etc.....</td>
</tr>
</tbody>
</table>
To COPY all the program files to the A: (Floppy Disk) when using the CAM continued…..

The 5000M uses the C:/USER drive in the Program Directory. Upon entering the Program Directory the programs displayed will all have an extension of .G.

To view all the program files described on the previous page the operator must first LOG to the root of C:/USER directory (this is where all program files with any extension can be viewed). This is achieved by using wildcard symbols. These symbols are * . * (any program name with any extension).

(1) From the PROGRAM directory Press (Shift) this will change the descriptions of the F keys,  
Press Log This shows a menu of the drives that the operator can view on screen. 

(2) Highlight Other: and Press

(3) At the next menu type *. Press

(4) Press The screen will now display all programs with all extensions.
To COPY all the program files to the A: (Floppy Disk) when using the CAM continued.....

An alternative method is also provided to obtain the screen showing all programs with all extensions Directory.

This method is completed using the up and Display keys.

This sequence of key strikes toggles the Program Directory display through the following screens. The key strikes are repeated to toggle through the screens shown below.
To COPY all the program files to the A: (Floppy Disk) when using the CAM continued……

To COPY all the EXAMPLE program files shown below complete the following:

1. Use the keys to highlight the first program to be copied (EXAMPLE.1) to the Floppy Drive

2. Using the key highlight the remaining EXAMPLE program files

3. Press and with Copy Hi-lited

A menu is displayed with A: Hi-lited Press

The lower left corner of the screen displays

COPYING…. will be displayed while the programs are copied. The procedure is complete when COPYING…. is no longer displayed.

To COPY program files (all extensions) from the A: (Floppy Disk) to the C:\USER directory.

1. From the Program Directory (C:\USER) use either method described previously to display the programs with all the extensions.

2. Insert the Floppy Disk into the Drive

3. LOG to the A: Drive

4. High light the program files to Copy to C:\USER Press ENTER

5. LOG back to the C: Drive
5000M CNC CONTROL
Simple Program
CREATING A PROGRAM

From Manual page press **F2**

The softkeys will change as shown below:

| Create | Delete | CAM | List | Select | Draw | Edit | Utility | Exit |

Press **F2**

Type the program name (Maximum 8 letters/numbers).

Press Enter put into program directory
High light will be on program just created.

Press **F8**  

EDIT
This is how screen will appear when entering **EDIT**

**Note:** Insert is turn ON, if not will not go past bottom of page.

**Note:** Softkeys have change.
This part needs to have 4 hole drilled .25 dia and .5 deep
Press **F1**

High light will be on **G0** press enter

Move high light **G17**, **G70** and **G90** press enter on each of these.

Press **M5** press

Press **F10** **Exit** it will go into program as shown below.

**G00 G17 G70 G90 M5**
Top line of program.

Program has been edited

Lines 2 and 3 are typed in manually.
The * allows commence into program control will read ignore after it.
Press F1 HELP press #9 for DRILL press

Screen will appear as shown below.

Press #2 for basic drilling, press enter.
Enter values Finish Depth -.5 press down arrow, enter Start Hgt .1 down arrow and Feedrate 12

Press **F10** **Exit** exits help menu and enters line into program.

---

**G00 G17 G70 G90 M5**  (*RAPID, XY PLANE, INCH, ABSOLUTE, POSITION, SPINDLE OFF*)

**G00 G17 G70 G90 M5**  (*RAPID, XY PLANE, INCH, ABSOLUTE, POSITION, SPINDLE OFF*)
**T1 M6**  (*LOAD TOOL, M6 ONLY REQUIRED IF TOOL CHANGER INSTALLED*)
**S2000 M3**  (*SET SPINDLE SPEED AND TURN ON SPINDLE*)
**G81 Z-0.5000 R0.1000 F12.0**  (*BASIC DRILLING CYCLE*)

The next enter hole positions.

Type **G0 X.75 Y.5**  [Diagram](#)  **X2.75**  [Diagram](#)  **Y1.5**  [Diagram](#)  **X.75**  [Diagram](#)

Type **G80**  [Diagram](#)

Cancel drill, this must always in program after last hole.
Type in last three lines.

Press **F10** [Exit] press **F7** [Draw]

Soft key will change as shown below.
Press **F5**  **DISPLAY** cursor will be on FIT press enter

This will fit drawing to screen.

Red lines are **RAPID** moves, blue circle are the holes and purple circle is the tool.

Press **F10**  **Exit** this brings control back to program page

Press **F6**  **Select** to select program, check that high light is on correct program.

Press **F10**  **Exit** go to **Manual**
SETTING PART ZERO.

If using an edge finder, touch edge of part using jog keys.

Go to tool page press **F9** Tool press **F1** OFFSETS

Cursor to required **OFFSET** using arrow up and down keys. Move X axis until part is located with edge finder.

Press **F4** CalibX is will enter value into table.

Because the edge find is .2 dia the position will be incorrect, press the letter **A** key, a box as shown below will appear. Type in **X.1** this will adjust offset to correct position press ENTER.

```
Enter axis and adjustment value: X.1
```

Repeat for **Y** axis

All offset are taken from machine Home position.
SETTING TOOL LENGTH OFFSETS

Put tool in spindle, bring tool tool down to top of part.

Press [F9] Tool high light correct tool number in tool page
press [F8] CalibZ

If there are more move Z axis up change to next tool and calibrate it as before, tool #0 must always be active when setting offsets.

Press [F10] Exit when all tools are calibrated.

RUNNING PART.

Press [F6] Auto turn down feedrate override knob to 10%

Press

If machine has a tool changer, it will put tool #1 in spindle if not already there.
If no tool changer install it will stop and allow you to change tool if necessary.

When program complete Z axis will go up to 5” above part.
5000M CNC CONTROL
GENERATING PROGRAMS USING CAM
STEP BY STEP CREATION OF A PROGRAM USING THE SHAPE EDITOR

The following presentation shows a step by step procedure of how to create a part program using the Anilam 5000 Control.
This procedure details the use of the interactive CAM that creates the part profile through simple entry of geometrical elements (points, lines and circles) without having to calculate intersections, tangency points etc. These elements are then connected or “chained” to complete the shape.

Creating the Part Program File in the PROGRAM page.

(1) From the MANUAL mode press: **Program**

**NOTE:** There are 2 methods of creating a program file, both are shown below.

**Method No.1** Creating a new program file.

(2) Press: **Create** At the prompt **NEW PROGRAM:** _ type in the program name

**NEW PROGRAM:** EXAMPLE_ and press

**NOTE:** Program names can be up to 8 characters in length, but may not include spaces or periods.

**Method No.2** Copying and using an existing program.

If a similar program to that required already exists, it may be copied and given a new name. This allows similarly formatted programs to be used without having to re-type the information.

(2a) Using the **Utility** keys hi-lite the existing program to be copied

Press: **Utility** this brings up a menu, **Copy** with Copy hi-lited

press This will display a second menu **Other...** Hi-lite Other and press

This will bring up a third menu **Copy to:** EXAMPLE_ type in the new program name

“EXAMPLE” and press this will create a new program named EXAMPLE and also leave the original program “SAMPLE1.G” intact. This copied program can now be edited to suit the new part. It is not necessary to type the file extension (.G) as this is completed automatically.
- Highlight the name the needs the program for, press **CAM**
- The machine program can be completely produced in CAM.

When **Shape** is press, soft keys will change as shown below.

**F4** - Back  
Moves cursor backwards on a shape.

**F5** - Forw  
Moves cursor forward on a shape.

**F6** - Prev-S  
Moves cursor to previous shape.

**F7** - Next-S  
Moves cursor to next shape.

**F8** - DelMove  
Deletes last move in shape.

**F9** - DelGeom  
Deletes geometry, an element number is required.

Press **Shape** again to turn OFF and return previous function keys.
Pressing **S-Edit** the following pop-up menu will appear:

- **Create** Create a start point for a shape.
- **Copy** Copy a shape to another location.
- **Move** Move a shape to a new location.
- **Delete** Delete a shape.
- **Rev Arc** Reverses direction of an arc in a shape.
- **Project** Replaces a radius and joins lines.
- **Join** Connects lines together.
- **Import** Imports shape from another CAM file.

Pressing **View** the following pop-up menu will appear:

- **XY plane**
- **XZ plane**
- **YZ plane**
- **Isometric**
Pressing **Display** the following pop-up menu will appear

- Fit to screen.
- Zoom in on windowed area.
- Redraw screen
- Half size of screen.
- Double size of screen.
- Scale screen.
- Pan move part around on screen.
- Erase screen.

Pressing **Misc** the following pop-up menu will appear

- List shapes by number.
- Lists all geometry. Showing coordinates.
- Recover a deleted shape.
- Recover a deleted tool path.

Pressing **Motion** the following pop-up menu will appear

- Generate a tool path around a shape.
- Generate tool path for pocketing a part.
- Drilling cycles and paths.
- Edit any of the above paths.
- Delete a tool path.
5. Pressing the following pop-up menu will appear

**Parameters for CAM.**
- Turn shapes ON/OFF.
- Turn geometry ON/OFF.
- Turn tool paths ON/OFF.

**Parameters for post.**

When high light is on setting press the pop-up menu will appear.

**Type of dimensioning ABS/INC.**
- Units INCH/MM.
- Turns Arrows ON/OFF.
- Turns ON/OFF element labels.
- Turns Axis markers ON/OFF.
- Turns grid ON/OFF.
- Size of grid.

When high light is on paths press the pop-up menu will appear.

**Out-put program name.**
- Over write existing program.
- Out-put type ABS/INC.
- Unit INCH/MM.
- Only out-put Axis if it moves.
- Text on while posting.
- Program number.
- Block number.
- Tool change requirements.
- Format Number of decimals.
The part below needs to be pocketed.
When CAM is accessed, the high light will be on the top left icon, for the following exercise, the third icon down is the one required.

Press down arrow key twice; icon on right will become point definitions.

Enter a point using X & Y coordinates.

Incrementally move from an existing point.

Move an existing point using an angle and a radius.

Center of a circle.

Intersect between two elements.

Using an existing point.
Press the icon on the right to change to line definitions.

- A line along X axis.
- A line along Y axis.
- A line between two points.
- A line through a point at an angle.
- A line parallel to an existing line.
- A line tangent to a circle through a point.

Press to get to circle icons.

- Circle between two existing elements.
- Circle knowing center point and radius.
- Circle tangent to a line through a point knowing radius.
- Circle tangent to a line knowing center point.
- Line between two circles, four options.
- Line tangent to a circle through a point.
High light third Icon down on left.

Press right and then up.

Geometery icon turn blue.

Circle icon high lighted

Press Enter R value: R value = 4

Press Select center definition

It will now ask for center definition and point definition will appear in right column of icons. The top icon will be high lighted.

Press it will ask for an value in X0 and Y0 twice.

When entry is 0 (zero) ,it is not required to press the 0 key.
Press `F5` **Display**. Fit and screen show the circle that was just entered.

Using same icon put in 2” radius circle.

Press ⏯️ enter 2” for radius

Press ⏯️ press ⏯️ to select point value X0, value Y 4.25 (6.25-2) press

Two circle will now show up on screen.
The next element required is the 1.8 radius arc.

In order to do this it is necessary put in some construction geometry. First a line has to be draw at -3.75 in the Y axis.

First high light geometry icon as shown below, press until line definition appear.

Use arrow keys to get to circled line definition.

Press enter a value of -3.75
The next geometry required are two points at the intersect of the 4” radius circle and the -3.75 line. To do this use icon circled below. These are the two points that needed to be found. They are the intersect of circle #1 and line #3.

Press Enter number of first element 1 press second element 3. There are now two selections, press 1 press. There is now an element #4 on left intersect. Do the same again only select #2.
It is now possible to drive an arc between points #4 & #5.

Use the Icon that allows a circle between two elements.

First question R value. Enter R value: 1.8_ Press

Second question Enter number of first element: 4_ Press

Third question Enter number of second element: 5_ Press
It is necessary to figure the best place to start. On this part the top will allow us to completely clean this shape.

Use icon circled line tangent to a circle at an angle.

First question Enter angle: 0 Press

Second question Enter number of circle: 2 Press

There are two chooses #1 or #2 we will take #2
All the necessary geometry has been established, the next thing is to make a shape, to pocket and contour.

Press **F3** **S-Edit**

A start point needs to be established for are shape, high light will be on **Create**. Create will set where the shape is going to start.

Press **select point definition,** use the icon circled.
Press 🡱 From point: 8_ Enter 8 when it says “From point”

Press 🡱

The white number one is the start point for shape #1.

The icon circle is the construction icon, highlight it press 🡱 it will ask for an element to be selected.

Select element: _

Select element: -2_ Enter -2 press 🡱

When selecting circles if cutting in clockwise the number is positive if counter clockwise the number is negative as shown above.

Selected element will show up in green.
The next selection is -1 and there are selection 1 and 2, in this case 2 is required.

Press 2

Notice that after 2 was selected element #2 when from green to white and element #1 became green.

The next element is #6, this is positive because the direction is clockwise.

Press 6
press -1
press -2
press 8

Notice the shape is now outlined in white.

The shape is now complete press F9 Cancel
The geometry and shape are complete, the next is to produce a tool path to pocket and contour.

Press **F7** Motion

The first tool path is **pocket** using arrow keys high light pocket press.
There are two methods of entering tool diameters, direct just type in value and select it from toll table.

A pop-up window will appear as above.

To enter a value in any of these parameters first press enter value press enter value.

When More is reached it will bring up anew pop-up window.
Second page of parameters.

---

**Pocket Parameters**

- **Comment:**
- **Interference check:** On
- **Angle of cut:** 200.0
- **Direction of cut:** Forward
- **Start point:** Default
- **Tool path color:** Green
- **Shape Reversed:** No
- **Entry Move:**
- **Exit Move:**

**Machine setup**

---

Angle of cut can be left at default or an angle entered, on this part an angle of 200 deg’s will be entered.

**Default Angle**

**Entry and Exit Move**

- **Linear**
- **Circular**

These are the three options.

Linear is straight line move on to start point, circular ram on move.

**Entry Move Setup**

<table>
<thead>
<tr>
<th>Move Type</th>
<th>None</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arc Length</td>
<td>0.0000</td>
</tr>
<tr>
<td>Arc Radius</td>
<td>0.0000</td>
</tr>
<tr>
<td>Origin Point</td>
<td>(0.0000, 0.0000)</td>
</tr>
</tbody>
</table>

Arc Length is the angle arc ramping on move.
Arc Radius size ram radius.

When **Machine Set** is hight lighted will go to a new pop-up window.
Coolant turn ON at start but not OFF at end as the same tool will be used for pocketing and contouring. The same applies with spindle.

On the right is how it will appear on screen.
- Red is geometry.
- White is the shape outline.
- Green is tool path.

Save toolpath?

Yes  No

F1  Yes
The Contour now need to be done as the edges are still rough.

Press F7 Motion

Contour Parameters

Shape number........ 1
Tool compensation..... CAM Left
Tool diameter......... 0.5000
XY stepover........... 0.0100
Number of XY passes.. 3
Z step.................. 0.0000
Approach height...... 0.1000
Top of contour........ 0.0000
Bottom of contour..... -0.5000
Stepover direction... Toward

It now needs to know which side to put tool comp.
Stepover this is around contour only.
Number of passes around contour.

Do this for both Entry and exit moves.
There is no need for a tool change because the same tool is being used for the contour as pocketing.

Note: Coolant and spindle at start are entered as None and turned Off at end, as they are still on from pocketing.
Arrow down to post

F9  Setup

If machine has a tool changer arrow down to Tool Change press enter arrow down to Tool Change Format if M06 is required it should read as follows T%02DM06.

Block Number are set to start at 10 and increment by 10 this can be change depending on your preference.
Format is set for 8 decimal place change these to 4.

Press F10 Cont press F9 Setup

Press F8 Post

When finish press F10 Exit it will now return to Program Page.
High light EXAMPLE.G press F7 Draw press F5 Display fit will be high lighted press

XY view

Isometric view

All that is left to do is set tool length offsets and fixture offsets, part is ready to run.
If the G-Code Configuration is now correct it can be save for future use.

Press F5 Display until top left corner shows ..\ high light EXAMPLE.CAM press F9 Utility high light
will be on Copy press Arrow down to Other Type in
C:\P5M\DEFAULTS.CAM
5000M CNC CONTROL
DXF Converter
DXF file can be converted into 5000 machine programs using the Offline software.

The DXF files are stored in the Program Page.

When going to Program Page only .G and/or .M file will be displayed, press shift twice, this will display all files on Program page.

If DXF is on disk it needs to be copied into C:\User directory.

Press shift select A:

High light required DXF program press Copy to C:

Log back to C:\User.

High light DXF program

Press Utility high light DXF Converter
**Select:** Used when selecting elements on drawing.
**Layers:** Allows layer on drawing to be turned Off or On.
**View:** XY,XZ,YZ or isometric.
**Display:** Fit, window, redraw, half or double.
**Save:** Saves program with .G once converted.
**Setup:** Allows setup of inputs and outputs.
**Exit:** Goes back to **Program** page
Zero on most drawings is usually not at a point that is convenient for programming, so there is a way it can be changed.

In the case of current drawing, the center of the hole in center of part is the best point X₀ Y₀.

To do this press the Ctrl key and hold it down put mouse point on to circle and press left mouse key, it will change to yellow, release keys.

At the bottom of screen X, Y, and Z coordinate will appear and also circle diameter.

Now press ALT key and letter T at the same time. This will input these coordinates into the SETUP and change X₀ Y₀ to the center of hole.

Press F9 to create a main program.
Press **F10** key and letter **F** key at the same time, this will mark the end of each element.

Press **F5** key high light **Window** press

A box will appear on screen move around screen using

press **F6** key **Compress** position box as shown below press
Press **F1** and press left mouse key. Line will turn green as above and put a number at low end of line, the position of the number is the start point. Now point to the line below it and press left mouse key, all off the line will be come green.

Press **ALT** key and letter **F** key at the same time the end of line markers will disappear.

Press **F5** and press **ENTER** Part will appear at full size on screen.
Press F8  Save

Successfully created XE2436-1.M.

Select  Layers  View  Display  |  Save  Setup  Exit

Press F10  Exit

Exit (Y/N)?
Yes  No  Enter  Enter  Enter  Enter  Enter  Enter  Cancel

Press F1 or Press Y  ENTER it will now return to Program page.

High light .G or .M file press

F4  Edit
High light .M file press F8  in conversational will appear as below

<table>
<thead>
<tr>
<th>1 Call 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 EndMain</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>4 Sub 1</td>
</tr>
<tr>
<td>5 Dim Abs</td>
</tr>
<tr>
<td>6 Rapid</td>
</tr>
<tr>
<td>7 Line</td>
</tr>
<tr>
<td>8 Line</td>
</tr>
<tr>
<td>9 Line</td>
</tr>
</tbody>
</table>

Start of program

719 Line X -14.40565 Y 0.16721
720 Line X -14.41445 Y 0.29290
721 Line X -14.42213 Y 0.41891
722 Line X -14.42368 Y 0.54589
723 Line X -14.43410 Y 0.67123
724 Line X -14.43339 Y 0.79760
725 Line X -14.44155 Y 0.92409
726 Line X -14.44357 Y 1.05070
727 EndSub

End of program

Program has to be Edited, to put in tool changes or cutter comp and Z moves.

High light .G file press F8  in G code format will appear as below

<table>
<thead>
<tr>
<th>G98 P1</th>
</tr>
</thead>
<tbody>
<tr>
<td>M2</td>
</tr>
<tr>
<td>01</td>
</tr>
<tr>
<td>G90 G0 X -14.44446 Y 1.17741</td>
</tr>
<tr>
<td>G1 X -14.44421 Y 1.30422</td>
</tr>
<tr>
<td>G1 X -14.44202 Y 1.43111</td>
</tr>
<tr>
<td>G1 X -14.44030 Y 1.55807</td>
</tr>
<tr>
<td>G1 X -14.43662 Y 1.68518</td>
</tr>
<tr>
<td>G1 X -14.43104 Y 1.81210</td>
</tr>
</tbody>
</table>

Program has to be Edited, to put in tool changes or cutter comp and Z moves.
This example will show multiple subroutines.

Press **F1** and pick all holes that are the same size, in this case 8. When going to second set of holes press right key on mouse.

The green circle is the last of previously selected holes. The yellow circle is the one selected with right mouse button and in the bottom left it is asking if this is a new shape, the answer is **Y**. It will put a number 2 next to this hole, meaning this is shape 2.
The print below shows the four shapes of the different size holes.

Press F8 Save
Press F10 Exit
Below is the output from the DXF converter. Some editing is required to put in drilling cycles.

<table>
<thead>
<tr>
<th>Subroutine calls</th>
<th>Subroutine for positions of the eight holes numbered in black.</th>
</tr>
</thead>
<tbody>
<tr>
<td>M98 P1</td>
<td></td>
</tr>
<tr>
<td>M98 P2</td>
<td></td>
</tr>
<tr>
<td>M98 P3</td>
<td></td>
</tr>
<tr>
<td>M98 P4</td>
<td></td>
</tr>
<tr>
<td>M2</td>
<td></td>
</tr>
<tr>
<td>01</td>
<td></td>
</tr>
<tr>
<td>G90 G0 X 0.15625 Y 0.84375</td>
<td></td>
</tr>
<tr>
<td>G0 X 0.15625 Y 0.15625</td>
<td></td>
</tr>
<tr>
<td>G0 X 1.34375 Y 0.15625</td>
<td></td>
</tr>
<tr>
<td>G0 X 1.34375 Y 0.84375</td>
<td></td>
</tr>
<tr>
<td>G0 X 1.34327 Y 2.66153</td>
<td></td>
</tr>
<tr>
<td>G0 X 6.38400 Y 3.50000</td>
<td></td>
</tr>
<tr>
<td>G0 X 14.30400 Y 3.00000</td>
<td></td>
</tr>
<tr>
<td>M99</td>
<td></td>
</tr>
<tr>
<td>02</td>
<td></td>
</tr>
<tr>
<td>G90 G0 X 15.10019 Y 3.12200</td>
<td></td>
</tr>
<tr>
<td>G0 X 15.10019 Y 4.12200</td>
<td></td>
</tr>
<tr>
<td>M99</td>
<td></td>
</tr>
<tr>
<td>03</td>
<td></td>
</tr>
<tr>
<td>G90 G0 X 7.42403 Y 4.75138</td>
<td></td>
</tr>
<tr>
<td>G0 X 6.45603 Y 4.75138</td>
<td></td>
</tr>
<tr>
<td>M99</td>
<td></td>
</tr>
<tr>
<td>04</td>
<td></td>
</tr>
<tr>
<td>G90 G0 X 14.21138 Y 4.51231</td>
<td></td>
</tr>
<tr>
<td>M99</td>
<td></td>
</tr>
</tbody>
</table>
The program below is Edited with the drill cycles in the program.

```
M1 G00 G70 G0 T0 20
M2 T1
M3 G81 Z-0.1500 R0.1000 F10.0
M4 M00 P1
M5 M98 P2
M6 M98 P3
M7 M98 P4
M8 G00
M9 G0 Z5
M10 T2
M11 G03 Z-0.5000 R0.1000 F12.0 I0.2500
M12 M98 P1
M13 G00
M14 G0 Z5
M15 T3
M16 G87 Z-1.0000 R0.1000 F12.0 I0.2500 J0.0200 K0.1500 W0.0100 U0.5000
M17 M98 P2
M18 G00
M19 G0 Z5
M20 T4
M21 G81 Z-0.2500 R0.1000 F15.0
M22 M98 P3
M23 G00
M24 G0 Z5
M25 T5
M26 G03 Z-1.0000 R0.1000 F8.0 I0.2500
M27 M98 P4
M28 G00
M29 G0 Z5
M30 M2
M31 01
M32 G90 G0 X 0.15625 Y 0.04375
M33 G0 X 0.15625 Y 0.15625
M34 G0 X 1.34375 Y 0.15625
M35 G0 X 1.34375 Y 0.04375
M36 G0 X 1.34375 Y 2.66153
M37 G0 X 6.38400 Y 3.50000
M38 G0 X 14.30400 Y 3.00000
M39 M99
M40 02
M41 G90 G0 X 15.10019 Y 3.12200
M42 G0 X 15.10019 Y 4.12200
M43 M99
M44 03
M45 G90 G0 X 7.42483 Y 4.75130
M46 G0 X 6.45603 Y 4.75130
M47 M99
M48 04
M49 G90 G0 X 14.21138 Y 4.51231
M50 M99
```
In this example of a full drawing and how to turn off unnecessary information, such as dimensions etc.
Press **F3** Layers

High light **Toggle Layers** press

Put high light on layers not required and press **ENTER** to turn OFF.

In the drawing shown the only layer required to be left on is #11
Only the part profiles and holes are left.

The circled area is blown up below. It shows an error in the drawing, where two lines are not connected. It will stop, select next element and the following message will appear:

**Entity not connected. Connect anyway (Y/N)?**

Press Y to continue.

It will then continue around part.