ANILAM

3000M 3 Axis
Training Guide

Navigation Instructions

- 3000M Begin
- Training Guide
  - Power On
  - Main Display
  - Main Keyboard
  - Hot Keys
  - Function Keys
  - Messages
  - Program Page
  - Create
  - Edit
  - List
  - Select
  - Display
  - Editor
  - Tools 99
  - Offsets 9
  - Adjust / Tool / Offset
- Lines & Arcs
- Canned Cycles
- Rotate / Mirror / Scale
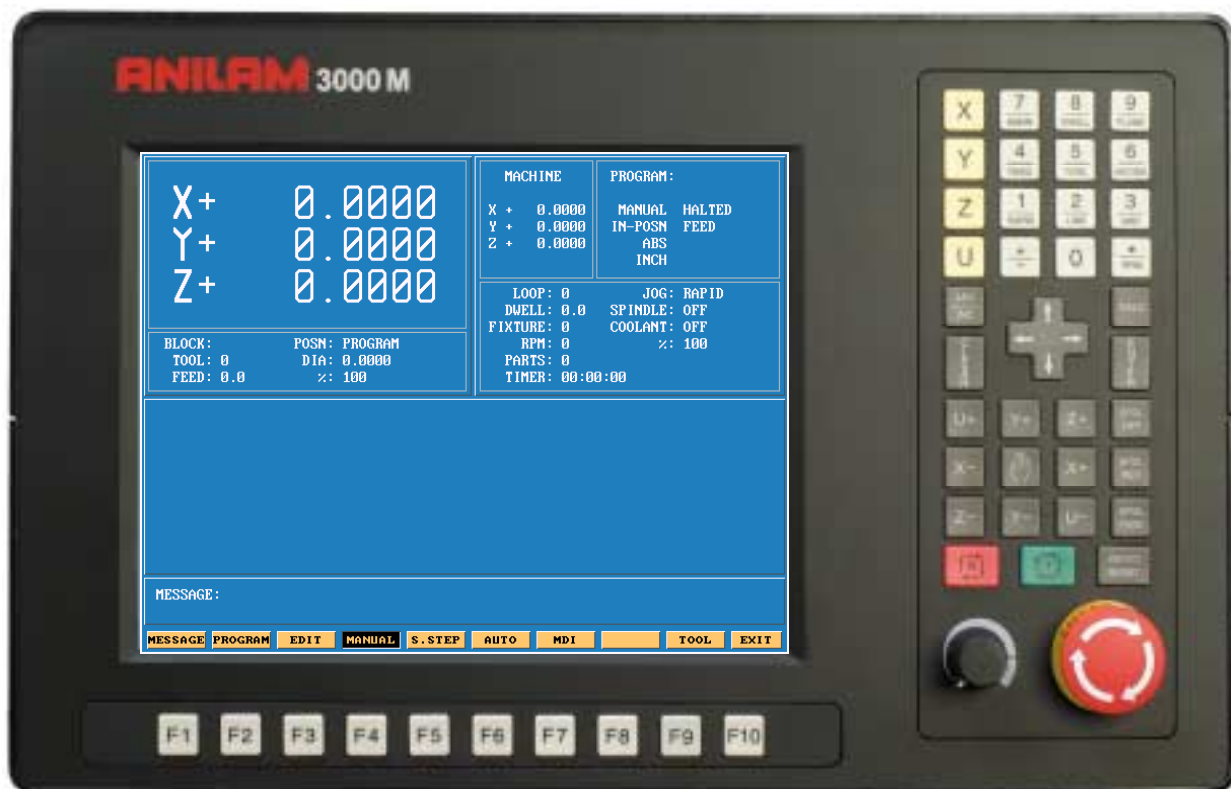- Cut / Copy / Paste
- Calculators
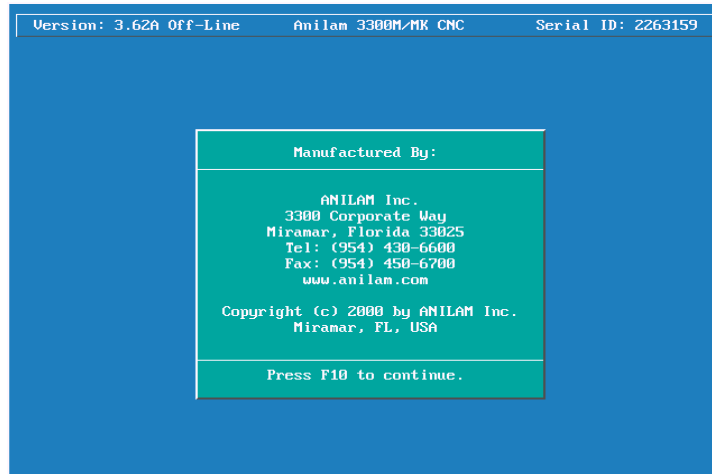- Program Example
- DXF Converter
- 3000M End
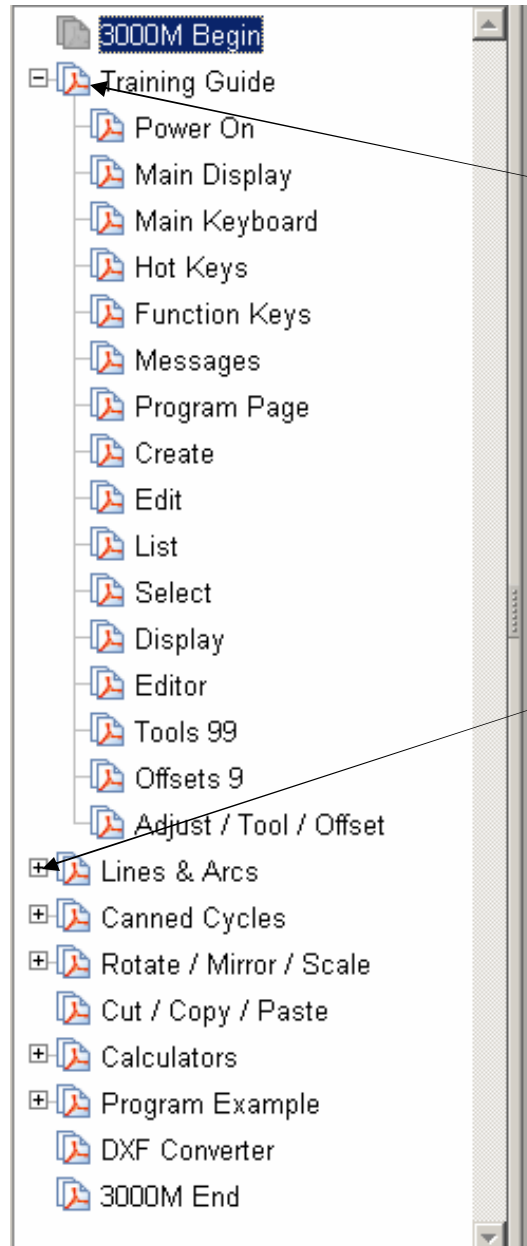
**Follow the bookmarks at the left side
of the page to navigate to desired topic**

**Click plus and minus symbols to expand
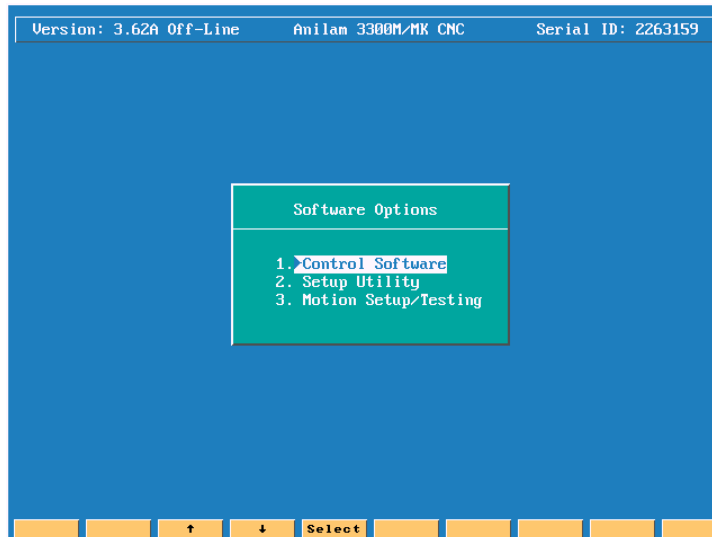and compress menu display**

ANILAM

3000M CNC Control
Training Guide

Turning the Control ON

```
Version: 3.62A Off-Line      Anilam 3300M/MK CNC       Serial ID: 2263159




                    Manufactured By:


                      ANILAM Inc.
                   3300 Corporate Way
                   Miramar, Florida 33025
                   Tel: (954) 430-6600
                   Fax: (954) 450-6700
                     www.anilam.com

                Copyright (c) 2000 by ANILAM Inc.
                     Miramar, FL, USA

                  Press F10 to continue.
```

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

```
Version: 3.62A Off-Line      Anilam 3300M/MK CNC       Serial ID: 2263159






                    Software Options

                 1. Control Software
                 2. Setup Utility
                 3. Motion Setup/Testing







              ↑      ↓    Select
```
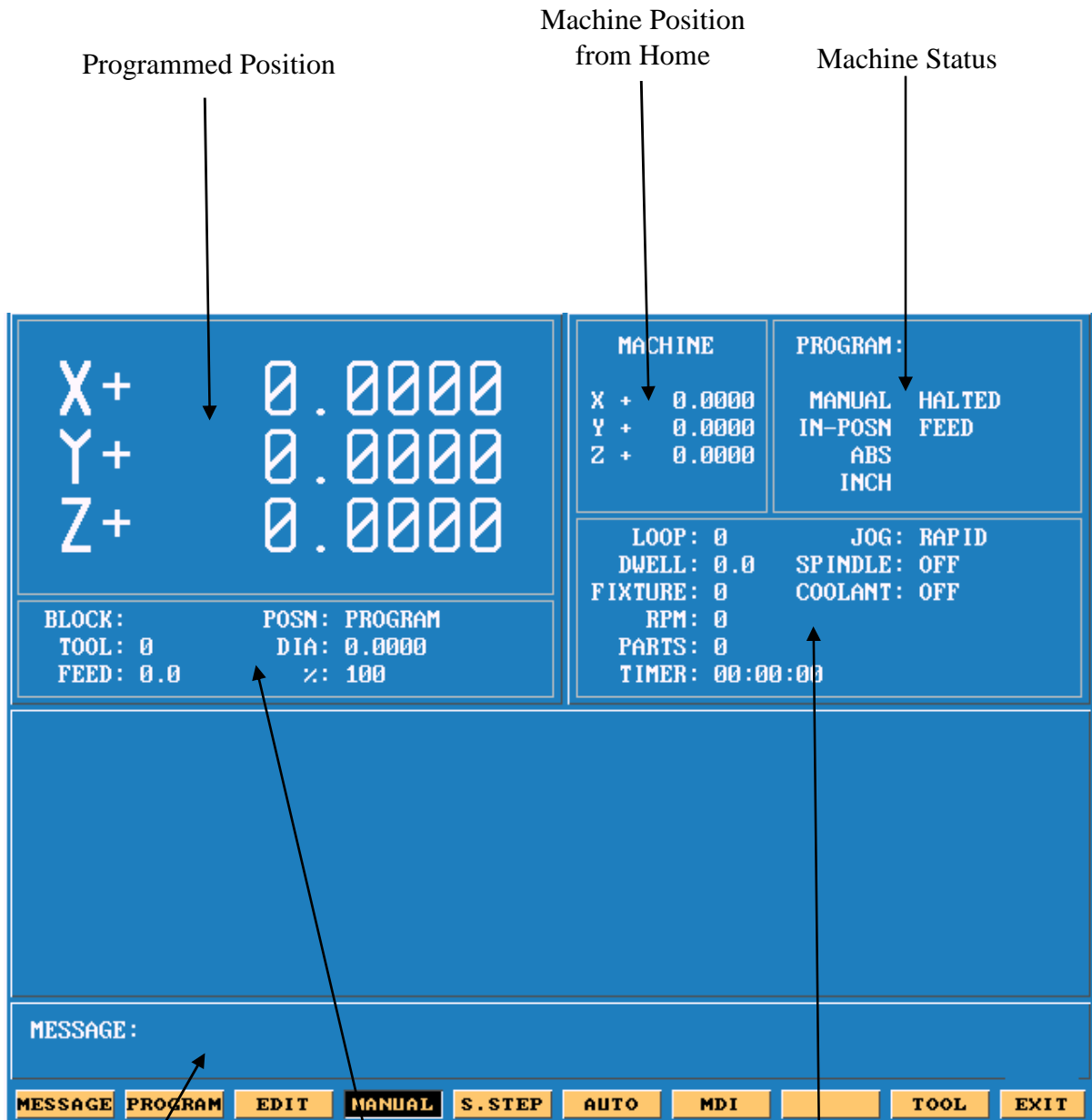
Then press **Select** to select next page

Main Areas of the Display

ANILAM

Programmed Position

Machine Position from Home

Machine Status

```
X+        0.0000        MACHINE          PROGRAM:
Y+        0.0000        X +    0.0000     MANUAL   HALTED
Z+        0.0000        Y +    0.0000     IN-POSN  FEED
                        Z +    0.0000        ABS
                                             INCH

BLOCK:         POSN: PROGRAM      LOOP: 0          JOG: RAPID
 TOOL: 0        DIA: 0.0000      DWELL: 0.0    SPINDLE: OFF
 FEED: 0.0        %: 100       FIXTURE: 0     COOLANT: OFF
                                   RPM: 0
                                 PARTS: 0
                                 TIMER: 00:00:00
```

MESSAGE:

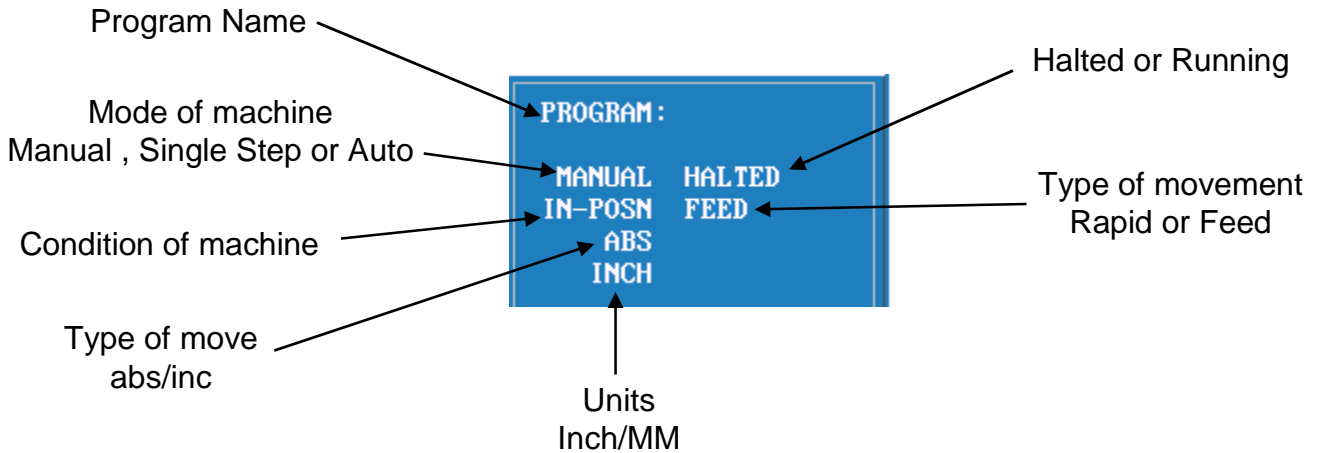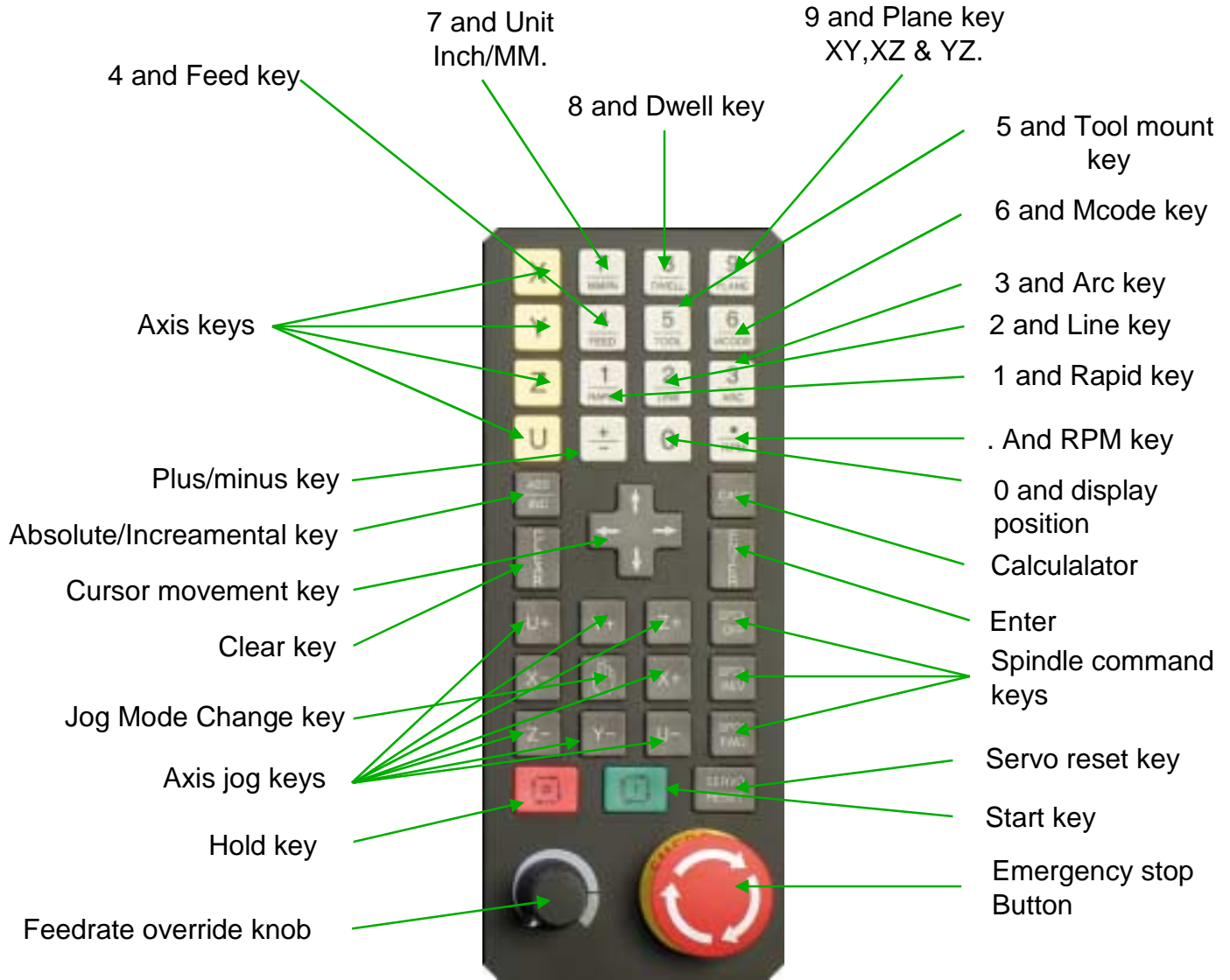| MESSAGE | PROGRAM | EDIT | MANUAL | S.STEP | AUTO | MDI | | TOOL | EXIT |

Message Area

Program & Tool information

Auxiliary Functions Information

<u>Areas</u> <u>of</u> <u>Main</u> <u>Screen</u>

Program Name

Mode of machine
Manual , Single Step or Auto

Condition of machine

Type of move
abs/inc

```
PROGRAM:

  MANUAL  HALTED
  IN-POSN FEED
    ABS
    INCH
```

Halted or Running

Type of movement
Rapid or Feed

Units
Inch/MM

Number of loops
remaining to be done

Time left
to Dwell

Active Fixture
Offset

Spindle Speed

Number or part
produced

Running Time

Type of Manual movement
Rapid,Feed,100,10 or 1

```
  LOOP: 0        JOG: RAPID
 DWELL: 0.0  SPINDLE: OFF
FIXTURE: 0   COOLANT: OFF
   RPM: 0
 PARTS: 0
 TIMER: 00:00:00
```

Spindle Condition
FWD,REV or OFF

Coolant condition
ON or OFF

Active line in Auto

Active Tool

Actual Feedrate

```
BLOCK:      POSN: PROGRAM
 TOOL: 0     DIA: 0.0000
 FEED: 0.0     %: 100
```

Posn:
Program or Dist to go

Tool Dia from
Tool Page

Percentage of programmed
Feedrate

Keyboard

7 and Unit Inch/MM.

4 and Feed key

9 and Plane key XY,XZ & YZ.

8 and Dwell key

5 and Tool mount key

6 and Mcode key

3 and Arc key

Axis keys

2 and Line key

1 and Rapid key

. And RPM key

Plus/minus key

0 and display position

Absolute/Increamental key

Calculalator

Cursor movement key

Enter

Clear key

Spindle command keys

Jog Mode Change key

Axis jog keys

Servo reset key

Start key

Hold key

Emergency stop Button

Feedrate override knob

All the number key and the decimal point have duel functions as show above.

Along with the above we have "Hot Keys"

| 1 Rapid | Accesses Rapid menus. |

| 2 Line | Accesses Line menus. |

| 3 Arc | Accesses Arc menus. |

| 4 Feed | Input feedrate. |

| 5 Tool | Tool mount. |

| 6 MCode | Mcode inputs. |

| 7 Unit | Inch/MM |

| 8 Dwell | Dwell in seconds |

| 9 Plane | Changes from XY,XZ or YZ. |

| 0 * | If * put in front of line in program ignores that line. |

| ● RPM | Input for spindle speed if available. |

Function Keys

ANILAM

```
  X+      0.0000     MACHINE        PROGRAM:
                     X +   0.0000   MANUAL   HALTED
  Y+      0.0000     Y +   0.0000   IN-POSN  FEED
                     Z +   0.0000      ABS
  Z+      0.0000                       INCH

                        LOOP: 0         JOG: RAPID
                       DWELL: 0.0   SPINDLE: OFF
 BLOCK:        POSN: PROGRAM  FIXTURE: 0   COOLANT: OFF
 TOOL: 0       DIA: 0.0000       RPM: 0
 FEED: 0.0       %: 100        PARTS: 0
                              TIMER: 00:00:00


 MESSAGE:
```

| MESSAGE | PROGRAM | EDIT | MANUAL | S.STEP | AUTO | MDI | | TOOL | EXIT |

| F1 | F2 | F3 | F4 | F5 | F6 | F7 | F8 | F9 | F10 |
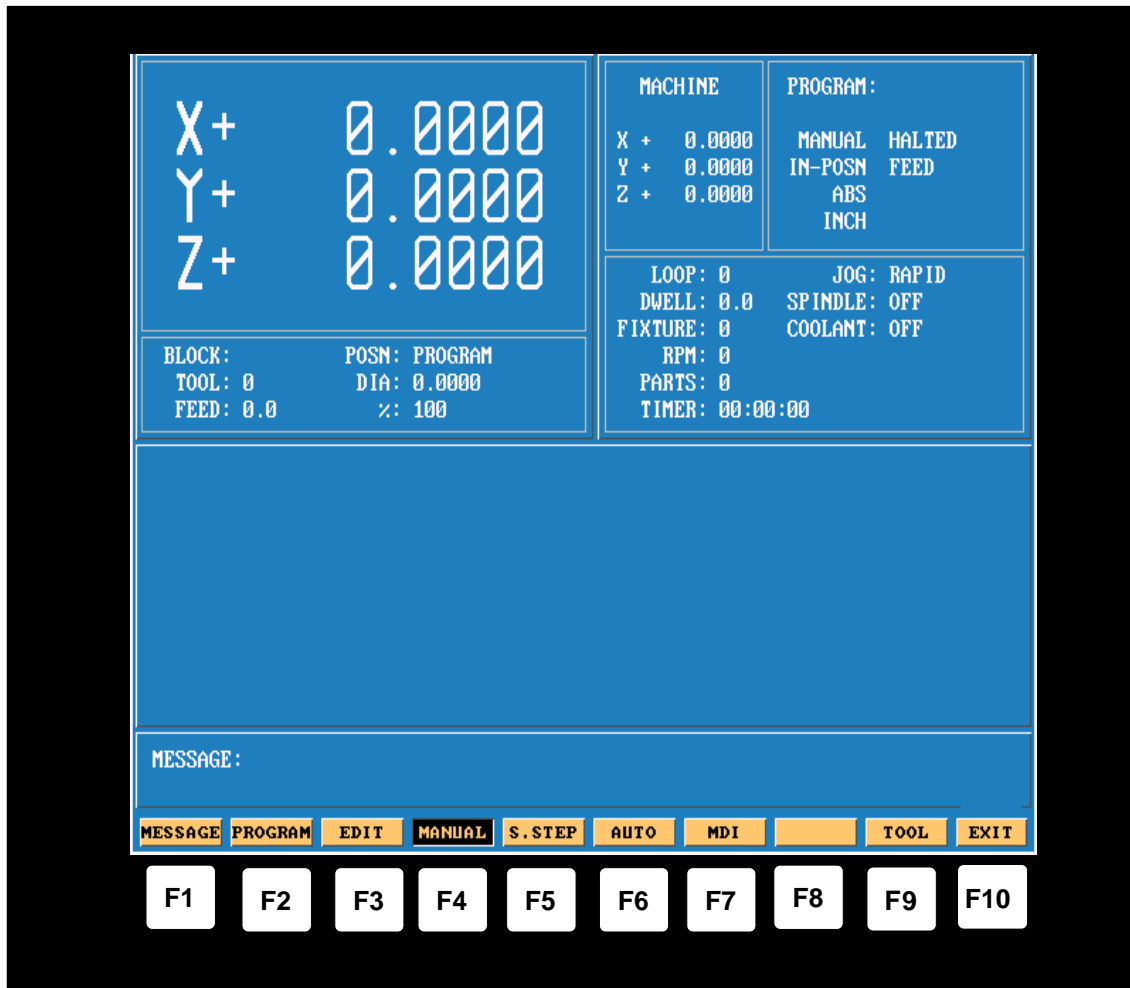
The Function (or F keys) activate the Mode shown directly above on the Display screen. The meaning of F keys change, depending upon what Mode of operation is selected.

| | PROGRAM | SEARCH | MANUAL | S.STEP | AUTO | | | | DRAW |

An example of how F keys is shown above,this is how it would change when going from Manual to in Single Step or Auto.

| MESSAGE | PROGRAM | EDIT | MANUAL | S.STEP | AUTO | MDI | | TOOL | EXIT |

F1 **Message** Display last 8 messages that have appeared

F2 **Program** Page at which program files are displayed

F3 **Edit** Go to EDIT from Manual

F4 **Manual** Manual mode of operation

F5 **S.Step** Single step mode of operation

F6 **Auto** Auto mode of operation

F7 **M D I** Manual data input,allow operation without writing a program

F8 Not used

F9 **Tool** Tool page and Fixture offsets can also be acessed from Edit

F10 **Exit** Exits control software

F1 Message Page

To enter message page press

F1

Message

This is how the messages will appear on the screen .
When control is turned off the messages will be erased.

This is useful to tech's when trouble shooting problems on the control

```
Illegal program name!
Illegal program name!
Illegal program name!
Press START to execute or MANUAL to Cancel.
WARNING [Block 35]: Circle adjusted beyond maximum adjustment!
```

Program Page

To enter page Press F2 **Program**

| | Create | Delete | Edit | List | Select | Log | Display | Utility | Exit |

F1 ☐ Not used

F2 Create This where new program names are type in.

F3 Delete Delete an existing program.

F4 Edit Programs are written and changed.

F5 List Allows you to look at program but no Editing .

F6 Select Picks program to be run.

F7 Log Changes disk drive.

F8 Display Changes how program page display program list

F9 Utility Contains such utility as Copy ,Rename etc.

F10 Exit Return to main screen in Manual Mode

```
MOLD.M          POKT.M          RICK4.M         ROTA.M          ST.M
MUM.M           PORTLAND.M      RICK5.M         ROTPOC.M        STANLEY.M
NEAL.M          POST.EXE        RICO.M          RPOCK.M         STEVE.M
NEAL1.M         PRO.M           ROB.M           RYAN.M          STEVE1.M
NEIL.M          PROGRAM.G       ROBIN.DXF       SAL.M           STEVE2.M
NICE.G          PROGRAM.M       ROBIN.M         SAM.M           STEVE3.M
NICE.M          PROGRAM.S       ROBIN1.M        SAMPLE.M        STOPLINK.M
OF.M            PUMPPLAT.M      ROBOT3.M        SANJOSE.M       SUGAR.M
OPBEZ1.M        Q1.M            ROBOT31.M       SCHTPAAA.M      TANDEMO.M
PALLET.M        Q1.S            ROBOTG.M        SCHTPAAB.M      TAP.M
PART.M          QWE.M           ROCKER.M        SCOTT.M         TERRY.M
PASS_1.M        R320T.M         ROD.M           SCOTT1.M        TEST-123.M
PATT.M          RANDY.M         RODNEY.M        SEAN.M          TEST-124.M
PAUL.M          README.M        RODNEY1.M       SGRO.M          TEST.M
PAUL2.M         RICH.M          RODNEY2.M       SHELDON.M       TEST1.M
PCBC0V20.M      RICH1.M         RON.M           SHEPHERD.M      THERESA.M
PETER.M         RICH3.M         RON1.M          SHOW.M          THOMAS.M
PHIL.M          RICHARD.M       RON2.M          SHOW1.M         THOMAS1.M
PHIL1.M         RICHARD1.M      RON3.M          SHOW98L3.M      THOMAS2.M
PIVOT.M         RICK.M          RON4.M          SIEMENS.M       THREAD.M
PLUG.M          RICK1.M         RON5.M          SKYLAND.M       TIM.M
POCKET.M        RICK10.M        RON6.M          SNAWMAN1.M      TIM1.M
POCKET.S        RICK2.M         RON7.M          SNOWMAN.M       TIMBOB.M
POKT-0.M        RICK3.M         ROT.M           SPRO.M          TIMTOS97.M
```

C:\3300USER   SELECTED PROGRAM: TEST.M

| | Create | Delete | Edit | List | Select | Log | Display | Utility | Exit |

Create

| | ASCII | Ins | Del | ← | → | BkSpace | | Cancel | |

When ASCII is pressed, this how the soft keys will look and a table will appear as show below Using the arrow keys pick the letters then press the enter key.When complete press ASCII again and press enter, this will put program into the list of programs with .M extension and highlight will be on program that was just created.
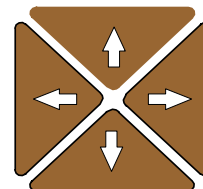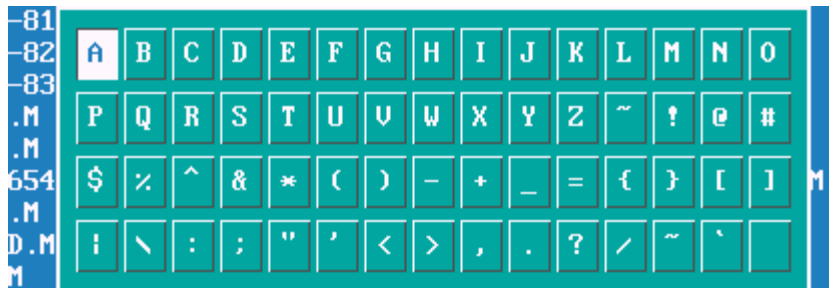
Edit

When [**Edit**] is press, the screen appear as shown below.

```
Program: TRAINING.M          Blocks: 0          Free: 3,895 KB   Inch   Abs   XY




















     1 <End Of Program>




   Teach   Draw   Drill   Pocket   Mill   Tool   Calc   Sub   Misc   Exit
```
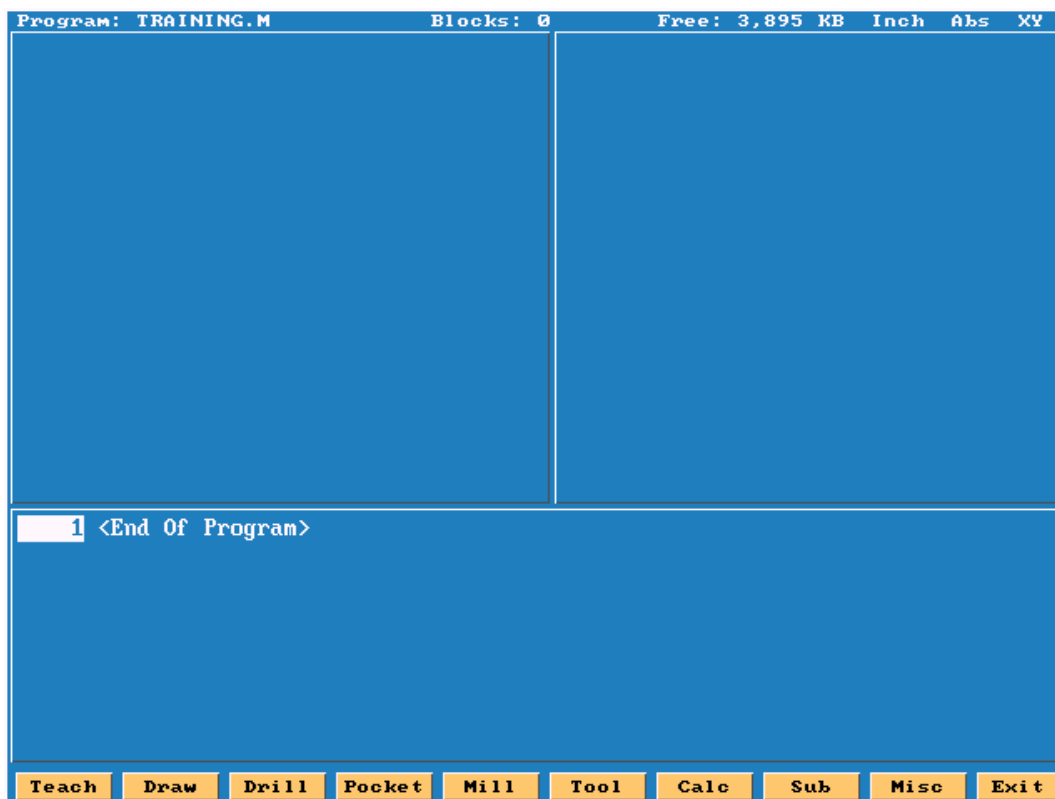
Pressing [**List**] allows user to look you to look at program only.

Select

```
MOLD.M          POKT.M          RICK4.M         ROTA.M          ST.M
MUM.M           PORTLAND.M      RICK5.M         ROTPOC.M        STANLEY.M
NEAL.M          POST.EXE        RICO.M          RPOCK.M         STEVE.M
NEAL1.M         PRO.M           ROB.M           RYAN.M          STEVE1.M
NEIL.M          PROGRAM.G       ROBIN.DXF       SAL.M           STEVE2.M
NICE.G          PROGRAM.M       ROBIN.M         SAM.M           STEVE3.M
NICE.M          PROGRAM.S       ROBIN1.M        SAMPLE.M        STOPLINK.M
OF.M            PUMPPLAT.M      ROBOT3.M        SANJOSE.M       SUGAR.M
OPBEZ1.M        Q1.M            ROBOT31.M       SCHTPAAA.M      TANDEMO.M
PALLET.M        Q1.S            ROBOTG.M        SCHTPAAB.M      TAP.M
PART.M          QWE.M           ROCKER.M        SCOTT.M         TERRY.M
PASS_1.M        R320T.M         ROD.M           SCOTT1.M        TEST-123.M
PATT.M          RANDY.M         RODNEY.M        SEAN.M          TEST-124.M
PAUL.M          README.M        RODNEY1.M       SGRO.M          [TEST.M]
PAUL2.M         RICH.M          RODNEY2.M       SHELDON.M       TEST1.M
PCBCOV20.M      RICH1.M         RON.M           SHEPHERD.M      THERESA.M
PETER.M         RICH3.M         RON1.M          SHOW.M          THOMAS.M
PHIL.M          RICHARD.M       RON2.M          SHOW1.M         THOMAS1.M
PHIL1.M         RICHARD1.M      RON3.M          SHOW98L3.M      THOMAS2.M
PIVOT.M         RICK.M          RON4.M          SIEMENS.M       THREAD.M
PLUG.M          RICK1.M         RON5.M          SKYLAND.M       TIM.M
POCKET.M        RICK10.M        RON6.M          SNAWMAN1.M      TIM1.M
POCKET.S        RICK2.M         RON7.M          SNOWMAN.M       TIMBOB.M
POKT-O.M        RICK3.M         ROT.M           SPRO.M          TIMTOS97.M
```

```
  C:\3300USER                              SELECTED PROGRAM: TEST.M
```

```
  Create  Delete   Edit    List   Select    Log   Display Utility  Exit
```

Program that is highlighted and then selected will shows up in this area.
A program may be edited without being selected , but not run in Single
step or auto.

Display

```
054FIN.M     ARIEL.M     DALE.M      DELTA.M     FIL.M
054_CH~1.M   BACKPUT.M   DALE1.M     DELTA1.M    FIX.M
123.M        BANTUM.M    DALE2.M     DELTA2.M    FORGE.M
221061A.M    BARNES.M    DAN.M       DENIS.M     FP.M
22PRGM1.M    BEN.M       DAN1.M      DIM.M       FRANK.M
2PRGM1.M     BERN.M      DANIEL.M    DON.M       FRANK1.M
321.M        BILL.M      DAREN.M     DOUG.M      FRED.M
3300LET.M    BOB.M       DAREN1.M    DOUG1.M     FRED1.M
442-8.M      BRIAN.M     DARREL.M    DOUG2.M     FRED2.M
442-81.M     BUTISL.M    DARRELL.M   DOWNING.M   FTR.M
442-82.M     CHEVSHOW.M  DARRELL1.M  EARNST.M    FX303.M
442-83.M     CHIP.M      DAVE.M      ED.M        GARY.M
456.M        CHRIS.M     DAVE1.M     ED2.M       GENE.M
543.M        CHUCK.M     DAVE10.M    ED3.M       GEORGE.M
789654.M     CHUCK1.M    DAVE11.M    ED4.M       GEORGE1.M
963.M        CHUCK2.M    DAVE2.M     EDDIE.M     GEORGE2.M
ABCD.M       CIR.M       DAVE5.M     ELBOW.M     GEORGE3.M
ACCURACY.M   CIRPOCK.M   DAVE6.M     ELL.M       GEROTECH.M
AL.M         CONCORD.M   DAVE7.M     ERV.M       GLORIA.M
ALAN.M       CP.M        DAVE8.M     ERV1.M      GLORIA1.M
AMT.M        CPV1.M      DAVE9.M     EXAM7.M     GOOFY.M
AMT2.M       CRAIG.M     DAVIS.M     FACE.M      GORDON.M
ANILAM.M     CRP.M       DD150CV.M   FCP.M       GREG.M
APM42.M      CURT.M      DEAN.M      FEATHER.M   GREG1.M

C:\3300USER                    SELECTED PROGRAM:

   Create  Delete  Edit  List  Select  Log  Display Utility  Exit
```

Normal screen only show .m files

```
054FIN.M       730,530    04/09/2001  02:38 pm
054_CH~1.M   1,643,679    04/09/2001  01:13 pm
123.M            1,049    04/18/2001  03:38 pm
221061A.M        5,506    05/15/2001  02:18 pm
22PRGM1.M       60,626    05/15/2001  02:03 pm
2PRGM1.M        60,626    05/15/2001  02:03 pm
321.M              550    04/20/2001  03:07 pm
3300LET.M        8,991    11/18/1993  04:16 pm
442-8.M        699,105    12/13/2000  05:21 pm
442-81.M       666,091    12/13/2000  05:21 pm
442-82.M       697,377    12/11/2000  07:45 am
442-83.M       664,450    12/13/2000  05:21 pm
456.M              360    02/08/2001  09:45 am
543.M              121    11/29/2000  10:15 am
789654.M           257    01/16/2001  03:15 pm
963.M              279    10/25/2000  02:02 pm
ABCD.M             592    01/19/1999  03:14 pm
ACCURACY.M         772    05/15/2001  08:52 am
AL.M               540    11/25/1998  10:03 am
ALAN.M             374    07/07/2000  10:05 am
AMT.M              720    05/08/1997  09:15 am
AMT2.M             671    05/08/1997  01:56 pm
ANILAM.M         3,339    11/17/2000  09:50 am
APM42.M            702    04/03/1998  04:28 pm

C:\3300USER                    SELECTED PROGRAM:

   Create  Delete  Edit  List  Select  Log  Display Utility  Exit
```

Show all in program files but show size of page program,date and time written

```
..             <DIR>    05/28/1998  11:06 am
11002AX\       <DIR>    05/28/1998  11:06 am
HYBCO\         <DIR>    01/04/1999  02:29 pm
RICHARD\       <DIR>    09/19/2000  09:50 am
054FIN.M       730,530    04/09/2001  02:38 pm
054FIN.S     1,287,086    04/20/2001  09:27 am
054_CH~1.M   1,643,679    04/09/2001  01:13 pm
054_CH~1.S   2,874,788    04/20/2001  08:00 am
123.M            1,049    04/18/2001  03:38 pm
123.S            1,440    04/18/2001  03:38 pm
221061A.M        5,506    05/15/2001  02:18 pm
221061A.S        8,512    05/15/2001  02:30 pm
22PRGM1.M       60,626    05/15/2001  02:03 pm
2PRGM1.M        60,626    05/15/2001  02:03 pm
321.M              550    04/20/2001  03:07 pm
3300LET.M        8,991    11/18/1993  04:16 pm
485AXIS.DOC      1,508    04/20/2001  02:31 pm
442-8.M        699,105    12/13/2000  05:21 pm
442-8.S      1,268,388    04/12/2001  07:48 am
442-81.M       666,091    12/13/2000  05:21 pm
442-82.M       697,377    12/11/2000  07:45 am
442-83.M       664,450    12/13/2000  05:21 pm
456.M              360    02/08/2001  09:45 am
543.M              121    11/29/2000  10:15 am

C:\3300USER                    SELECTED PROGRAM:

   Create  Delete  Edit  List  Select  Log  Display Utility  Exit
```

Show only .m files but show size of program, date and time written

```
..          789654.M     CHIP.M       DAREN1.M    DOUG.M
11002AX\    963.M        CHRIS.M      DARREL.M    DOUG1.M
HYBCO\      ABCD.M       CHUCK.M      DARRELL.M   DOUG2.M
RICHARD\    ACCURACY.M   CHUCK1.M     DARRELL1.M  DOWNING.M
054FIN.M    ACCURACY.S   CHUCK2.M     DAVE.M      EARNST.M
054FIN.S    AL.M         CIR.M        DAVE1.M     ED.M
054_CH~1.M  ALAN.M       CIRCLES.DXF  DAVE10.M    ED2.M
054_CH~1.S  AMT.M        CIRPOCK.M    DAVE11.M    ED3.M
123.M       AMT2.M       CLOVER.DXF   DAVE2.M     ED4.M
123.S       ANILAM.DXF   COMPLEX.DXF  DAVE5.M     EDDIE.M
221061A.M   ANILAM.M     CONCORD.M    DAVE6.M     ELBOW.M
221061A.S   APM42.M      CP.M         DAVE7.M     ELL.M
22PRGM1.M   ARIEL.M      CPV1.M       DAVE8.M     ERV.M
2PRGM1.M    BACKPUT.M    CRAIG.M      DAVE9.M     ERV1.M
321.M       BANTUM.M     CROSS.DXF    DAVIS.M     EXAM7.M
3300LET.M   BARNES.M     CRP.M        DD150CV.M   FACE.G
485AXIS.DOC BEN.M        CURT.M       DEAN.M      FACE.M
442-8.M     BERN.M       DALE.M       DELTA.M     FACE.S
442-8.S     BILL.M       DALE1.M      DELTA1.M    FCP.M
442-81.M    BOB.M        DALE2.M      DELTA2.M    FEATHER.M
442-82.M    BRIAN.M      DAN.M        DENIS.M     FIL.M
442-83.M    BUTISL.DXF   DAN1.M       DIM.M       FIX.M
456.M       BUTISL.M     DANIEL.M     DOC1.DOC    FORGE.M
543.M       CHEVSHOW.M   DAREN.M      DON.M       FP.M

C:\3300USER                    SELECTED PROGRAM:

   Create  Delete  Edit  List  Select  Log  Display Utility  Exit
```

Press display one time it will change to display all files in directory
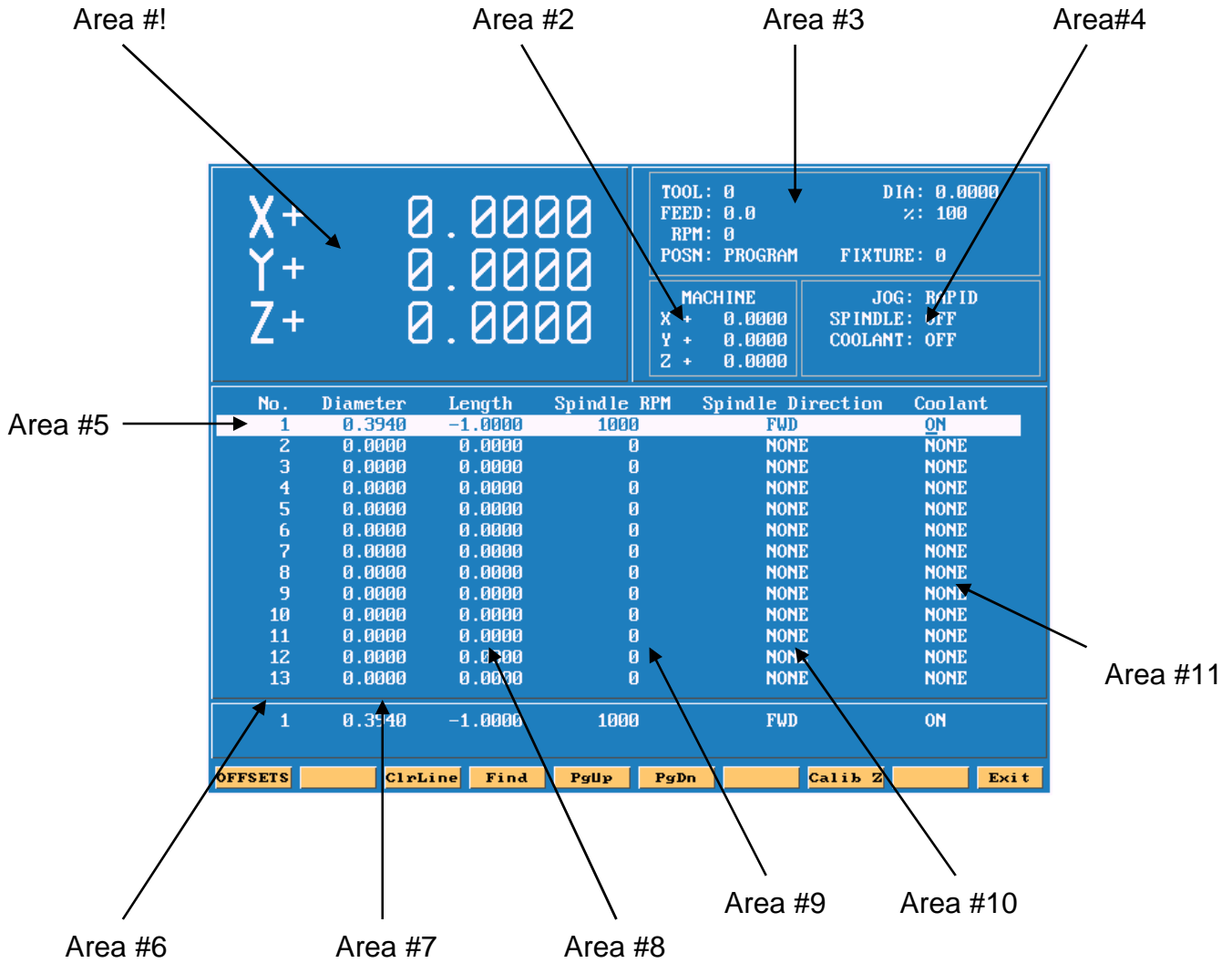
ANILAM

F keys there different functions.

First press **Create** type in a program name or number.

High light will be on program just created press **Edit**

| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

F1 **Teach** Let user move machine around and record positions.

F2 **Draw** Draws program be running to see that it is correct.

F3 **Drill** Access to drilling canned cycles.

F4 **Pocket** Access to pocketing canned cycles.

F5 **Mill** Access to milling features.

F6 **Tool** Go to tool Page.

F7 **Calc** Go to calculators.

F8 **Sub** Sub more editing features.

F9 **Misc** More editing features.

F10 **Exit** Exits editing.

Press **F6** **Tool** This will bring up the **Tool Page.**

Area #! Area #2 Area #3 Area#4

```
TOOL: 0              DIA: 0.0000
FEED: 0.0            %: 100
 RPM: 0
POSN: PROGRAM     FIXTURE: 0
```

```
      MACHINE          JOG: RAPID
X +    0.0000     SPINDLE: OFF
Y +    0.0000     COOLANT: OFF
Z +    0.0000
```

X+ 0.0000
Y+ 0.0000
Z+ 0.0000

No.	Diameter	Length	Spindle RPM	Spindle Direction	Coolant
1	0.3940	-1.0000	1000	FWD	ON
2	0.0000	0.0000	0	NONE	NONE
3	0.0000	0.0000	0	NONE	NONE
4	0.0000	0.0000	0	NONE	NONE
5	0.0000	0.0000	0	NONE	NONE
6	0.0000	0.0000	0	NONE	NONE
7	0.0000	0.0000	0	NONE	NONE
8	0.0000	0.0000	0	NONE	NONE
9	0.0000	0.0000	0	NONE	NONE
10	0.0000	0.0000	0	NONE	NONE
11	0.0000	0.0000	0	NONE	NONE
12	0.0000	0.0000	0	NONE	NONE
13	0.0000	0.0000	0	NONE	NONE
1	0.3940	-1.0000	1000	FWD	ON

| OFFSETS | | ClrLine | Find | PgUp | PgDn | | Calib Z | | Exit |

Area #5

Area #6 Area #7 Area #8 Area #9 Area #10 Area #11

Area #1 Machine position relative to part Zero.
Area #2 Machine position relative to machine zero.
Area #3 Tool information , Feed , RPM and Fixture offset.
Area #4 Jog , spindle and Coolant condition.
Area #5 Active line.
Area #6 Tool number.
Area #7 Tool diameter.
Area #8 Tool length offset.
Area #9. RPM's only input if M-Functions or an inverter is installed.
Area #10 Spindle forward or reverse same conditions as above apply.
Area #11 Coolant on or off must have M-Functions.

OFFSETS		ClrLine	Find	PgUp	PgDn		Calib Z		Exit

F1   **OFFSET**   Go to FIXTURE OFFSET page

F2   Not used

F3   **ClrLine**   Clear highlighted line info.

F4   **Find**   Search for a tool.

F5   **PgUp**   Page up

F6   **PgDn**   Page down

F7   Not used

F8   **Calib Z**   Calibrate Z axis offset.

F9   Not used

F10   **Exit**   Exit to edit page when entered from edit page.

If entered from **MANUAL** page   **F10**   **Exit**   will exit to **MANUAL** page .

Press **F1** **OFFSET** this will take you to offset page.

All entries are taken from **Machine Home.**

Entries may be entered manually or by using **CalibX** or **CalibY.**

When doing manual input , select axis you wise to enter a value , by pressing that axis key and input number require press **ENTER**.

Using the **Calib** key move to required position and press desired calib axis key.

Press **F10** **Exit** to return to **Tool Page**

Press **F10** **Exit** to return to **Manual Page**

17

Enter adjustment value is an added feature that allows altering of existing values in the tool page or fixture offsets display. Use the **ABS/INC** Key to activate this feature.

No.	Diameter	Length	Spindle RPM	Spindle Direction	Coolant
1	* 2.0000	-2.0000	0	NONE	NONE
2	0.2500	-2.0000	0	NONE	NONE
3	0.1600	-2.0000	0	NONE	NONE
4	0.1400	-2.0000	0	NONE	NONE
5	0.1200	-2.0000	0	NONE	NONE
6	0.1000	-2.0000	0	NONE	NONE
7	0.0800	-2.0000	0	NONE	NONE
8	0.0600	-2.0000	0	NONE	NONE
9	0.0400	-2.0000	0	NONE	NONE
10	1.2500	-2.0000	0	NONE	NONE
11	1.0000	-2.0000	0	NONE	NONE
12	0.0000	0.0000	0	NONE	NONE
13	0.0000	0.0000	0	NONE	NONE

1	2.0000	-2.0000	0	NONE	NONE

Enter adjustment value: -.125_

| ASCII | Ins | Del | ← | → | BkSpace | | Cancel | |

Example: Altering tool diameter using **Enter adjustment value.**

* Asterisk highlights the selected value.

No.	Diameter	Length	Spindle RPM	Spindle Direction	Coolant
1					ONE
2			Fixture Offsets		ONE
3					ONE

	X	Y	Z	U	
1. ▶	0.8000	1.3000	-19.0000	0.0000	ONE
2.	2.6000	1.1750	0.0000	0.0000	ONE
3.	0.8000	0.9000	0.0000	0.0000	ONE
4.	0.0000	0.0000	0.0000	0.0000	ONE
5.	2.8750	0.9250	0.0000	0.0000	ONE
6.	0.0000	0.0000	0.0000	0.0000	ONE
7.	0.0000	0.0000	0.0000	0.0000	ONE
8.	0.0000	0.0000	0.0000	0.0000	ONE

▶Enter axis and adjustment value: X-1.250_

| 1 | | | | | ONE |

| ASCII | Ins | Del | ← | → | BkSpace | | Cancel | |

Example: Altering fixture offsets using **Enter axis and adjustment value**
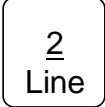
3000M CNC Control
Lines and Arcs
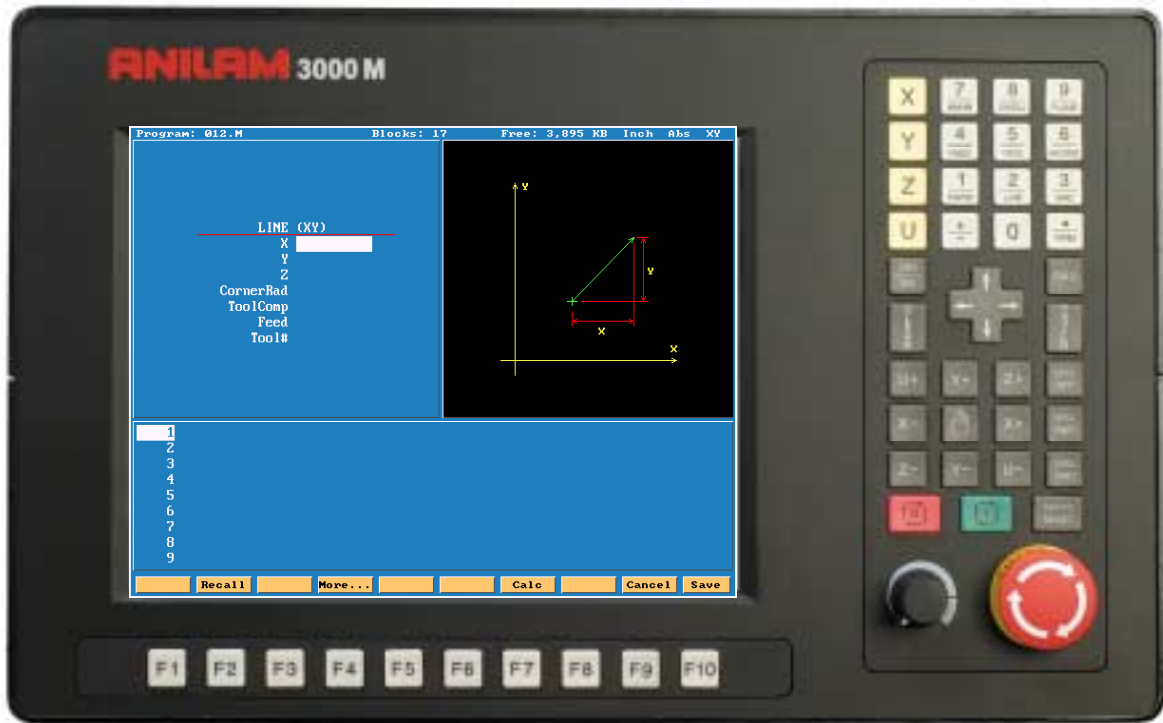
Lines and Arcs

Lines and arcs can be access in two ways .
1. Using hot keys.

1 Rapid	Accesses Rapid menus.
2 Line	Accesses Line menus.
3 Arc	Accesses Arc menus.

2.Using soft keys

| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

Press **F5** **Mill**

| | Rapid | Line | Arc | | | More | | Prev | |

Press **F2** **Rapid**

| | Recall | | More... | | | Calc | | Cancel | Save |

Press **F4** **More...**

Screen will now show 6 icons , these apply to both rapid and line moves.
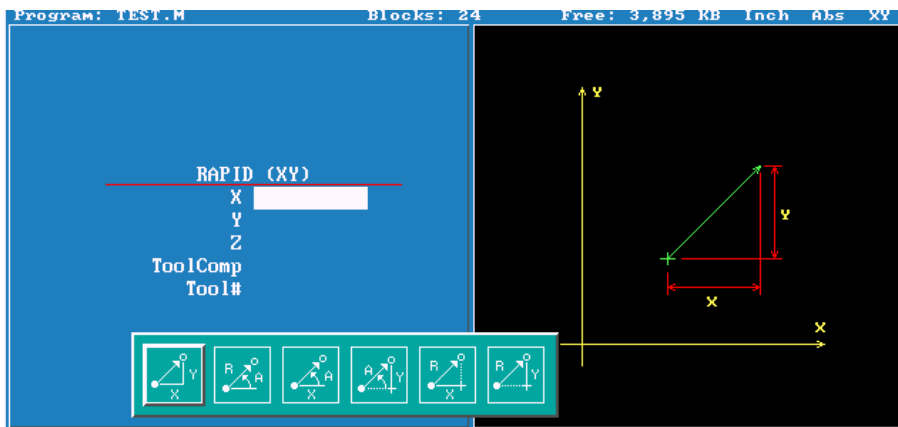
Line and **Rapid** are the same with one exception CornerRad and feed are not in rapid.
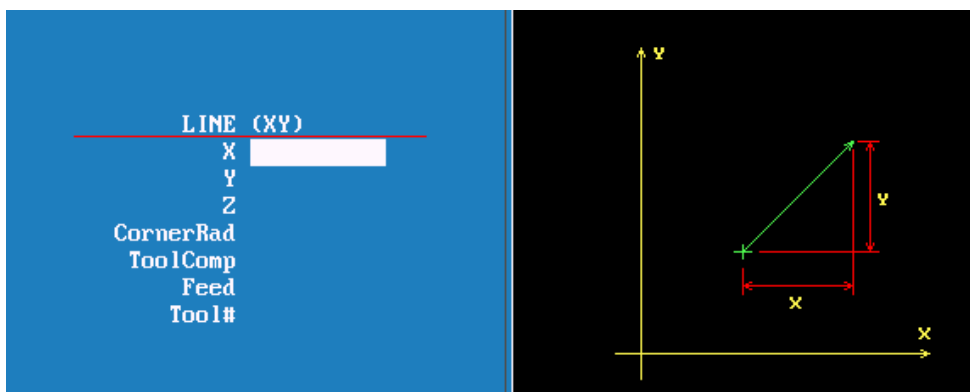
1. Enter coordinates in any or all axis.

2. Using Radius and Angle from current position.

```
LINE (ANGLE - RADIUS)
     Radius    0.0000
     Angle     0.0000
         Z
  CornerRad
   ToolComp
      Feed
     Tool#
```
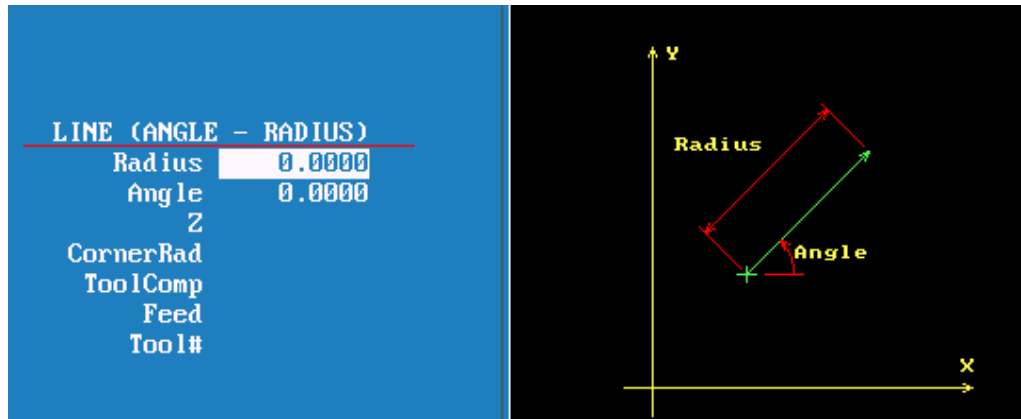
3. X axis move is an Absolute dimension, the Angle relative to Zero (three O'clock). Below it is Y axis and Angle.

```
LINE (X - ANGLE)
         X    0.0000
     Angle    0.0000
         Z
  CornerRad
   ToolComp
      Feed
     Tool#
```

```
LINE (Y - ANGLE)
         Y    0.0000
     Angle    0.0000
         Z
  CornerRad
   ToolComp
      Feed
     Tool#
```
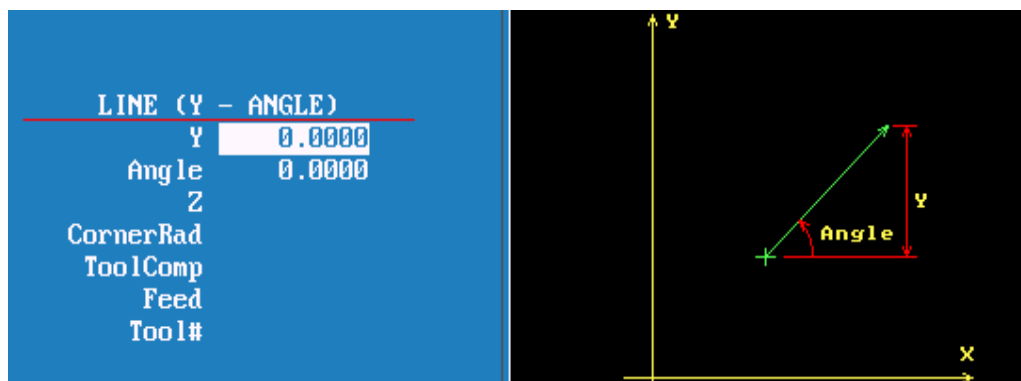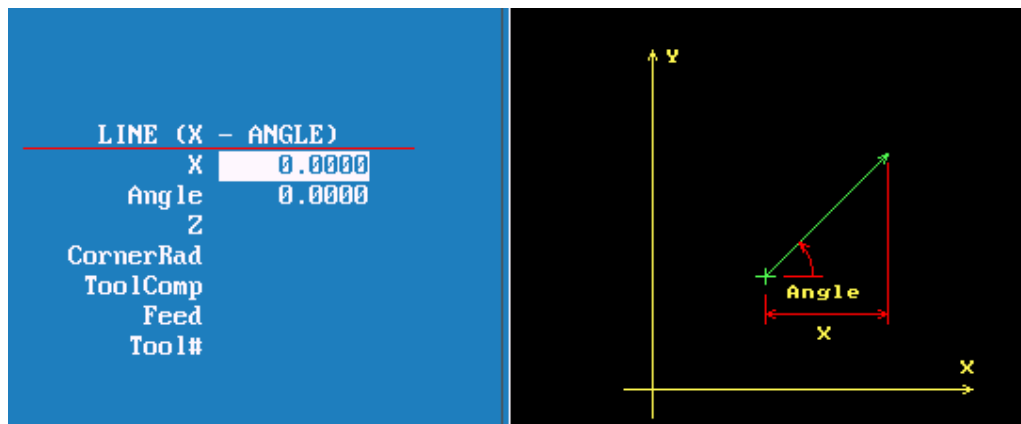
4.Axis and Angle

If programming absolute X or Y dimension is from Part 0 and radius is from current position.

If programming incremental X or Y dimension and radius are from current position.

Teach	Draw	Drill	Pocket	Mill	Tool	Calc	Sub	Misc	Exit

F5 **Mill**

	Rapid	Line	Arc			More		Prev	

F4 **Arc**

	Recall		More...			Calc		Cancel	Save
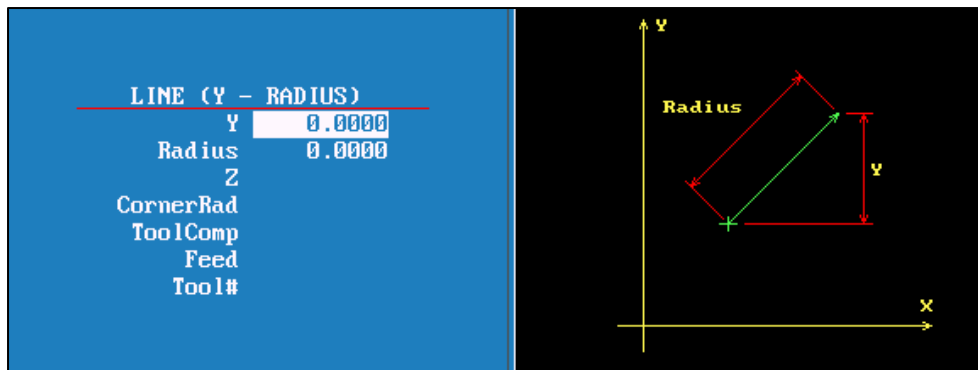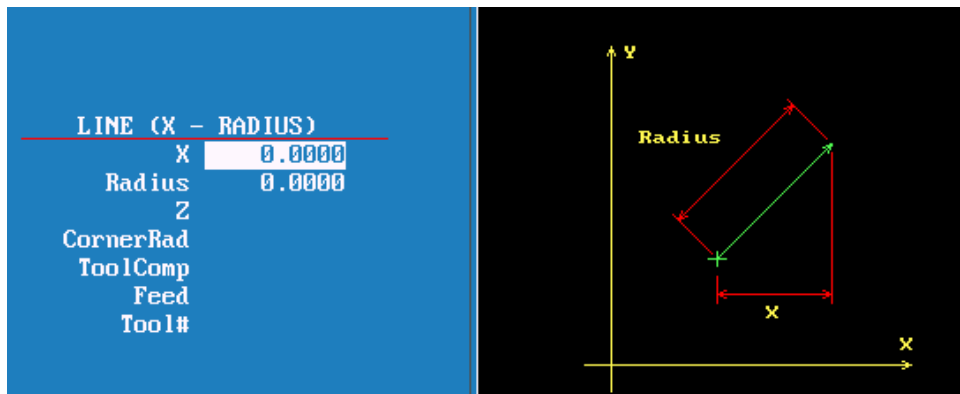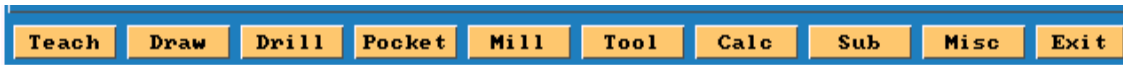
F4 **More...**

The icon below are for arcs.
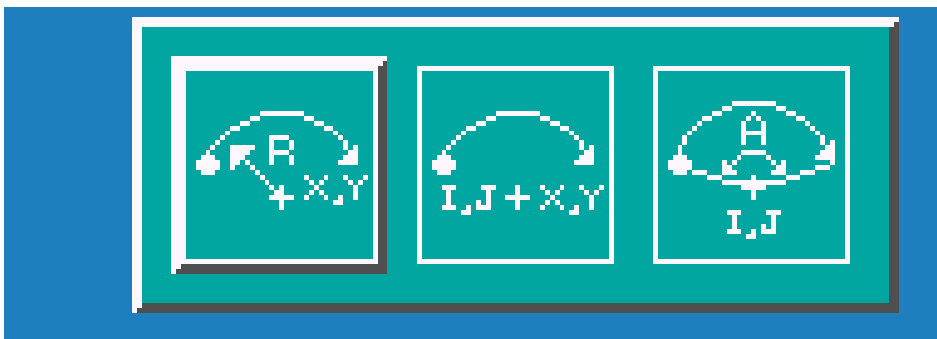
1   2   3

Tool must be at start point of arc before it is programmed.
Arc may be cut in any plane but must selected, default is XY.

```
ARC (END POINT - RADIUS)
   Direction        Cw
          X       0.0000
          Y       0.0000
          Z
     Radius       0.0000
  CornerRad
       Feed
      Tool#
```
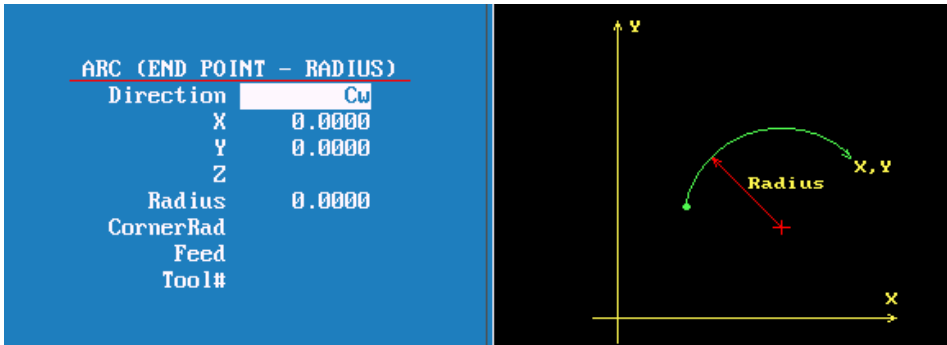
1. This is the default arc , End points for 2 axis and radius.
   Direction is changed by pressing +/- key.
   If programming an arc over 180 deg the radius is entered
   as a minus (-) value.


```
ARC (CENTER - END POINT)
   Direction        Cw
          X
          Y
          Z
    XCenter       0.0000
    YCenter       0.0000
       Revs
  CornerRad
       Feed
      Tool#
```
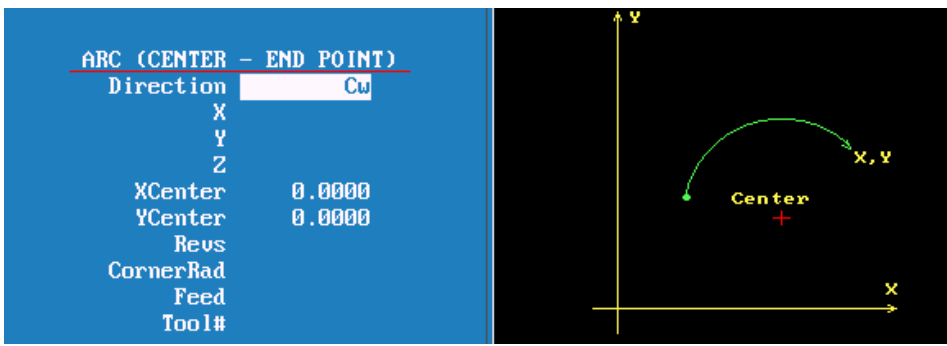
2. With this arc it is possible to mill a thread .
   Enter X Y and Z end point , X Y center point and Number of Revolutions.
   The pitch of thread is controlled by the Z movement and the number of
   revolutions.

```
ARC (CENTER - ANGLE)
Direction        Cw
   XCenter    0.0000
   YCenter    0.0000
     Angle    0.0000
         Z
CornerRad
      Feed
     Tool#
```
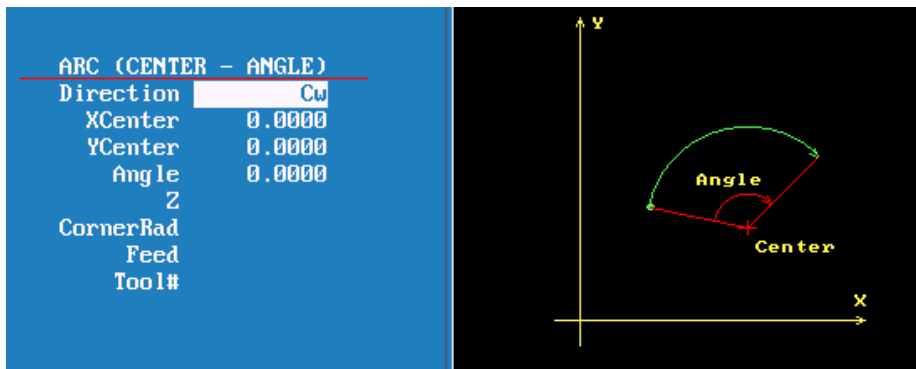
3. Center and angle

  Programming absolute angle is from 3 o'clock position direction is also critical.

```
1 Rapid      X 0.7071 Y 0.7071
2 Arc Ccw     XCenter 0.0000 YCenter 0.0000 Angle 90.0000
3 <End Of Program>
```

The above lines of program starts at 45 deg's the center is X0, Y0 and go to 90 deg's in a counter clockwise direction, it will go to 12 o'clock, actually only moving 45 deg's. If direction was clockwise it would move 315 deg's.

```
1 Rapid      X 0.7071 Y 0.7071
2 Dim Incr
3 Arc Ccw     XCenter -0.7071 YCenter -0.7071 Angle 90.0000
4 Dim Abs
```

With the above program in incremental Counter clockwise will go from 45 deg's ,were it starting to 125 deg's, so it moves the actual number of degrees programmed. If clockwise is used it will go to -45 deg's , so it still only moves 90 deg's.

6.

| Recall | | More... | | | Calc | | Cancel | Save |

F1   [ ]   Not used.

F2   **Recall**   Used to recall enters from calculator , will cover later.

F3   [ ]   Not used.

F4   **More . . .**   Used with Rapid , Line and Arc as shown previously .

F5   [ ]   Not used.

F6   [ ]   Not used.

F7   **Calc**   Calculator this will be cover later.

F8   [ ]   Not used.

F9   **Cancel**   Cancels what is being done .

F10   **Save**   Saves information in to Program.

3000M CNC Control
Canned cycles
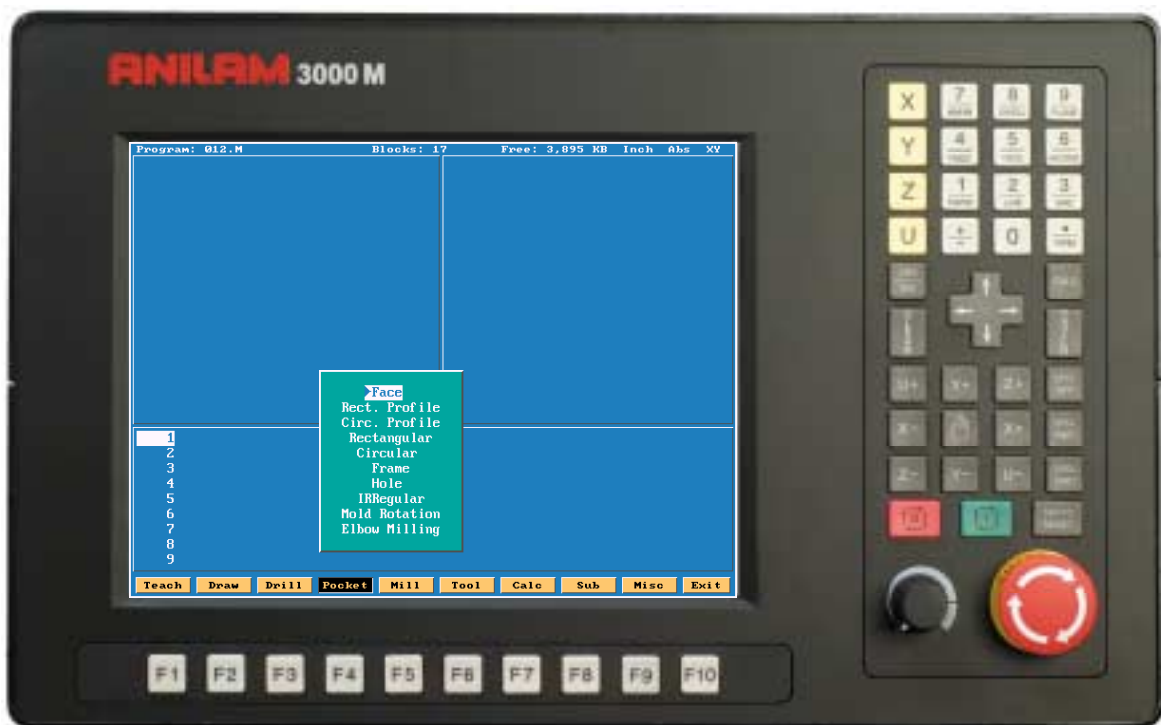
Drill Canned Cycles & Patterns

1. Basic   Rapid to Z start, Feed to Z depth, Rapid to Z return.

2. Pecking   Rapid to Z start, Feed to peck, Rapid to Z start, Rapid to last peck, Repeat peck cycle to Z depth,
                  Rapid to Z return.

3. Boring   Rapid to Z start, Feed to Z depth, Dwell, Feed to Z start, Rapid to Z return.

4. Chip Break   Rapid to Z start, Feed to first peck, Retract chip break Inc, Feed first peck minus peck decr,
                  Retract chip break Inc, Repeat cycle until min peck is reached and continue to Z depth, Rapid to Z return.

5. Tapping   Rapid to Z start, Feed to Z depth, Dwell, Reverse spindle, Feed to Z start, Rapid to Z return.

6. Pattern   Drilling locations that can be defined into a symmetrical Pattern.

7. Bolt Hole   Drilling locations that can be defined into a symmetrical Hole Pattern.

Basic Drilling
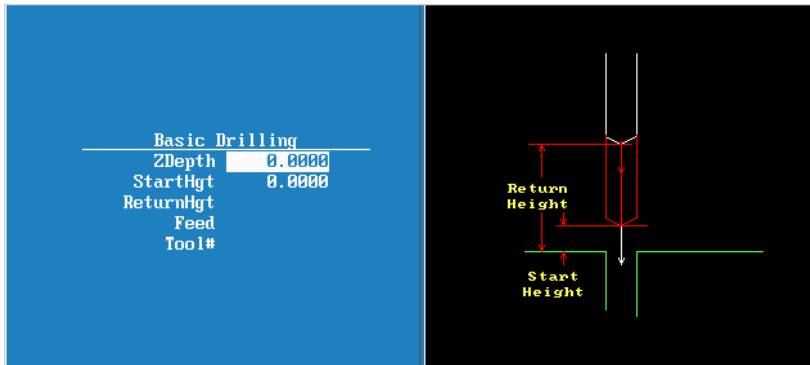
ZDepth   Feed Z axis to depth.

StartHgt   Rapid start of Z axis before feed to depth.

ReturnHgt   Rapid return of Z axis from Z depth (Optional).

Feed   Feed rate " Inches per minute" (Optional).

Tool#   Activate new tool (Optional).

Peck Drilling
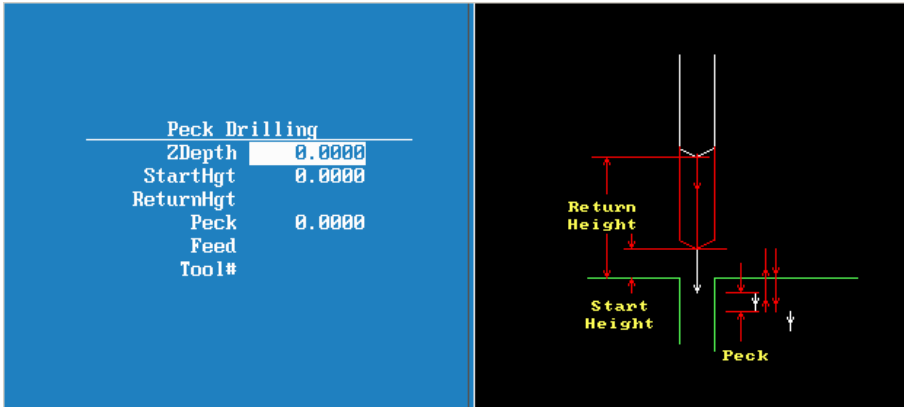

```
        Peck Drilling
    ZDepth        0.0000
   StartHgt       0.0000
  ReturnHgt
     Peck         0.0000
     Feed
    Tool#
```

ZDepth   Feed Z axis to depth.

StartHgt   Rapid start of Z axis before feed to depth.

ReturnHgt   Rapid return of Z axis from Z depth (Optional).

Peck   Peck increment " + Value"

Feed   Feed rate " Inches per minute" (Optional).

Tool#   Activate new tool (Optional).

Boring
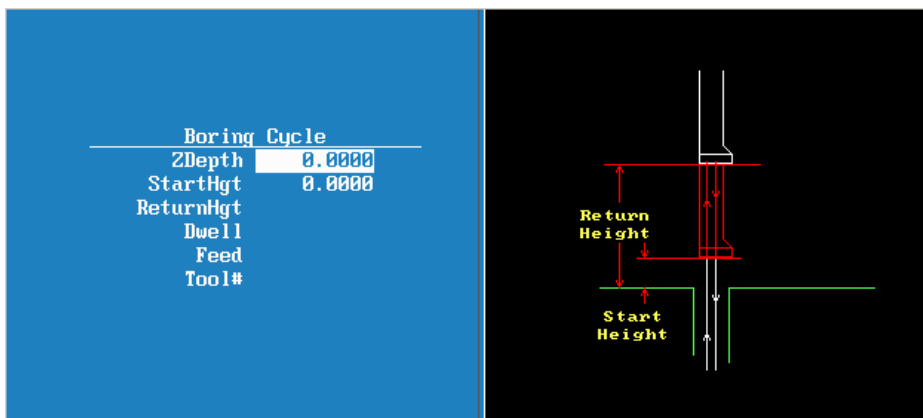

```
       Boring Cycle
    ZDepth        0.0000
   StartHgt       0.0000
  ReturnHgt
    Dwell
    Feed
    Tool#
```

ZDepth   Feed Z axis to depth.

StartHgt   Rapid start of Z axis before feed to depth.

ReturnHgt   Rapid return of Z axis from Z depth (Optional).

Dwell   Dwell time at Z depth "Seconds" (optional)

Feed   Feed rate " Inches per minute" (Optional).

Tool#   Activate new tool (Optional).

Chip Break Cycle

ZDepth   Feed Z axis to depth.

StartHgt   Rapid start of Z axis before feed to depth.

ReturnHgt   Rapid return of Z axis from Z depth (Optional).

FirstPeck   First unaltered pecking increment " + Value".

PeckDecr   Decrement first peck each cycle " + Value".

MinPeck   First peck altered to minimum value " + Value".

ChipBrkInc   Retraction increment after peck " + Value" (Optional).

RetractDep   Retract to Z start and return to last peck at Retract Depth (Optional).

Feed   Feed rate " Inches per minute" (Optional).

Tool#   Activate new tool (Optional).

Tapping Cycle

ZDepth   Feed Z axis to depth.

StartHgt   Rapid start of Z axis before feed to depth.

ReturnHgt   Rapid return of Z axis from Z depth (Optional).

TPIorLead   Threads per inch or Lead to establish feed rate

SyncSpin   Synchronize spindle with Z axis if so Equipped

Dwell   Dwell at Z depth (Optional)

Tool#   Activate new tool (Optional).

Pattern Drill

X	X start location (Optional).
Y	Y start location (Optional).
#XHoles	Number of holes on X axis including starting hole.
#YHoles	Number of holes on Y axis including starting hole.
XInc	Incremental distance between all holes on X axis "Use – or + values for direction".
YInc	Incremental distance between all holes on Y axis "Use – or + values for direction".
Angle	Rotate entire pattern about X and Y starting hole (Optional).

Bolthole Drill

XCenter	X center of circle.
YCenter	Y center of circle.
#Holes	Number of holes in circle including starting hole.
Diameter	Diameter of circle.
StartAngle	First hole angle from 3 o'clock zero (0 to 360 degrees counter clockwise) or (+ counter clockwise degree increment / - clockwise degree increment).
EndAngle	End hole angle from 3 o'clock zero (0 to 360 degrees counter clockwise) or (+ CCW or – CW degree increment)_____(EndAngle Optional).
IndexAngle	Rotate entire pattern about circle center (Optional).

Thread Milling Cycle

> **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.

Thread Milling Cycle simplifies the programming required to mill a thread. 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. 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). The tool must be positioned at center of hole or boss, or the parameters for **XCenter** and **YCenter** must be specified in the cycle.

XCenter, **YCenter, ArcInRad, StockAmt, Passes, RoughFeed, FinFeed**, and **TaperAng** are all optional Input; all other parameters must be programmed. If the feed rates are not programmed, the CNC will use last feed rate used.

In a tapered thread the Major Diameter is always the major diameter of thread where the **ZStart** is set. Inside diameter is at finished depth and outside diameter is the diameter of boss.

To program a Thread Milling Cycle:

1. In Edit Mode, press **Drill** (**F3**) to display a pop-up menu.

2. Select **Thread Mill** from the pop-up menu, and press **ENTER** to display the **Thread Mill** Graphic Menu. Refer to **Figure 5-8**.
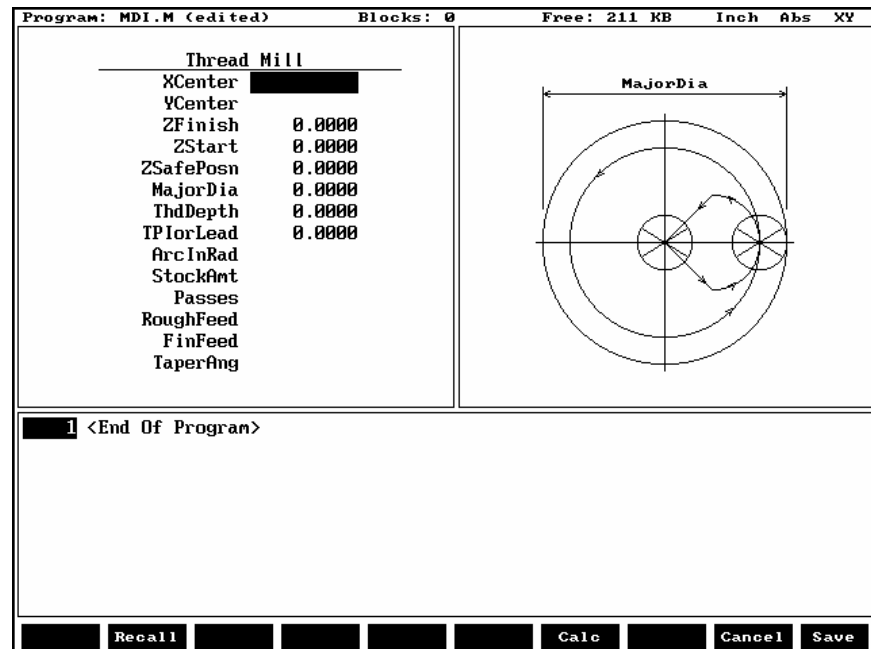
Figure 5-8, Thread Mill Graphic Menu

3. Enter the following required values and settings in the **Thread Mill** entry fields:

XCenter	Absolute X coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread as the current tool position. (Optional)
YCenter	Absolute Y coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of the thread as the current tool position. (Optional)
ZFinish	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)
ZStart	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, the cycle will use the current Z tool position. (Optional)
ZSafePosn	An Absolute safe Z position above the part for rapid moves in X and/or Y. (Required)

> **WARNING: ZSafePosn** must be above the part to avoid a crash while positioning.

MajorDia	Major thread diameter. For a tapered thread, it will be the major diameter at the Z start position. 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. (Required)

  +   Value cuts in the CW direction

  −   Value cuts in the CCW direction

ThdDepth	Depth of thread. The incremental depth of thread on one side: (Required)

  +   Value is inside thread

  −   Value is outside thread

TPIorLead	Threads per inch (TPI) or lead of thread. (Required)

> **NOTE:** The minimum number of threads per inch is "1"

	ArcInRad	Size of radius arcing into start of thread. (Optional)

> **NOTE:** If **ArcInRad** is a positive value or not set and the thread is "inside," the cycle will always return to the center between passes.
>
> If **ArcInRad** 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 **ArcInRad** is not specified and the thread is "outside," the cutter will back away from the largest diameter by an amount equal to the thread depth.

	StockAmt	Amount to leave for a finish pass after the roughing passes. (Optional)
	Passes	Number of roughing cuts to be taken. (Optional)

> **NOTE:** If **StockAmt** is not set or set to zero and **Passes** is 1 or 0, the cycle will make just one pass at the full depth..
>
> If **StockAmt** is set to greater than zero and **Passes** is 1 or 0, the cycle will make just one pass at the stock depth and one pass at full thread depth.
>
> If **Passes** is set to a negative number, all non-cutting positioning moves will be rapid.

	RoughFeed	Feedrate for roughing. (If not set, the cycle will use the current active feedrate.) (Optional)
	FinFeed	Feedrate for the finish pass. (If not set, the cycle will use the current active feedrate.) (Optional)
	TaperAng	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 straight pipe tape with an inside cut would be -1.7833. If not set, then the thread is straight. (Optional)

If X and Y are not programmed, position tool center of the thread before calling the "Thread Mill" cycle:

- X and Y will rapid to the starting position of the thread.

- Z will rapid to the safe height specified in **ZSafePosn**.

- The Z axis will feed down to the start cut position **ZStart**. This could be above or below the Z position specified in the **ZFinish** position.

- Depending on what is in the **ArcInRad** parameter, the tool will arc into the first cut position.

- Spiral up or down, depending on the difference between **ZFinish** and **ZStart** and go counterclockwise or clockwise depending if **MajorDia** 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 **ArcInRad**.

- Then feed back to the **ZStart** 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 **ThdDepth** parameter minus the stock amount specified in the **StockAmt** parameter, divided by the number of roughing passes specified in the **Passes** 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 tapered 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 an arc-in value in the **ArcInRad** parameter will allow the cycle to move to the center of the hole for maximum clearance.

Sample Thread Program

This program will cut an 8 TPI thread starting at 0.1 above the hole. The major diameter of the thread is 1 inch. It will take 2 roughing cuts and 1 finish cut with a 0.625 cutter.

```
Dim Abs
Tool# 1
Rapid       X 1.0000 Y 1.0000
Rapid       Z 0.1000
ThreadMill  Finish -1.0   ZStart 0.1    ZSafePosn .5   MajorDia
            1.0     ThdDepth .0625   TPIorLead 8   ArcInRad
            0.1    StockAmt 0.002   Passes 2   RoughFeed
            20.0   FinFeed 5.0
Rapid       Z 5.0000
EndMain
```

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 **Passes** to a negative number. The idea is to initially set **Passes** as a positive number and after proving out the program, change it 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 **Passes** to -1.

Pocketing Canned Cycles
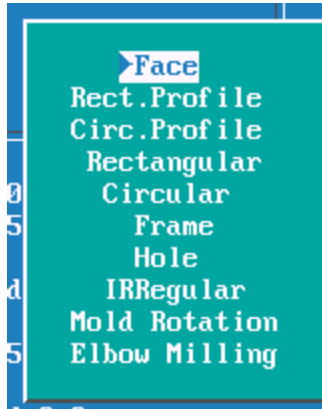
Note
The pockets marked with * all have cutter compensation built into them, so all dimension are as show on print.

1.Face.	Cleans large area with one line of information.
2.Rectangular profile.*	Cleans inside or outside of a rectangle.
3.Circular profile.*	Clean inside or outside of a circle.
4.Rectangular pocket.*	Cuts a rectangular pocket to a specified depth.
5.Circular pocket.*	Cuts a circular pocket to specified depth.
6.Frame pocket.*	Cuts rectangular pocket with an island in the middle.
7.Hole.*	Opens up existing holes.
8.Irregular pocket.*	Cleans the inside of a closed contour.
9.Mold rotation.	Cuts three axis shape but only program 2 axis.
10.Elbow milling	Cuts a radial grove around a radius.

Face Pocket
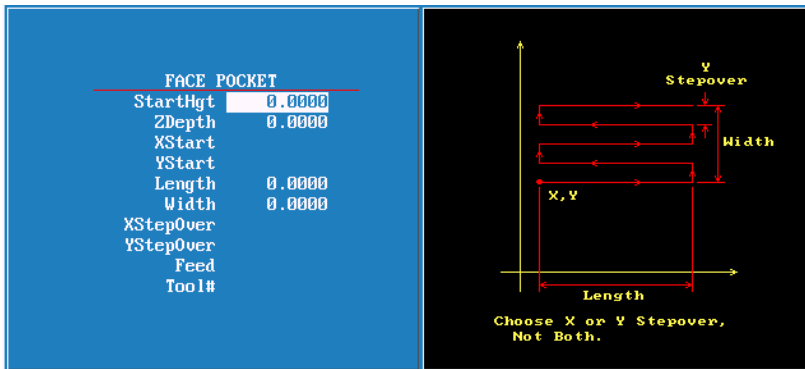
StartHgt.	Distance above surface to be cut.	
ZDepth.	Depth of Z axis.	
XStart.	X axis start position.	Optional
YStart.	Y axis start position.	Optional
Length.	Length of surface to be cut.	
Width.	Width of surface to be cut.	
XStepOver.	Distance X axis steps over between passes.	Only one step over to be programmed.
YStepOver.	Distance Y axis steps over between passes.	Only one step over to be programmed.
Feed.	Feedrate in inches per minute.	Optional
Tool#.	Tool number.	Optional

Only input with 0.0000 have to be programmed , this will apply to all canned cycles.Tools do not have to be programmed in cycles , in most cases it is not a good idea.

Rectangular profile
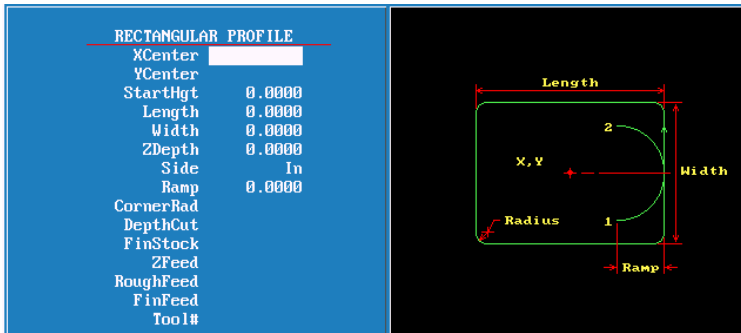
XCenter.	Center of profile along X axis.	Optional
YCenter.	Center of profile along Y axis	Optional
StartHgt.	Height Above surface to be cut.	
Length.	Length of pocket.	
Width.	Width of pocket.	
ZDepth.	Absolute depth of pocket.	
Side.	Inside or Outside.	
Ramp.	Size of ramp radius.	
CornerRad.	Radius on the corners. On the inside must be larger than cutter radius.	Optional
DepthCut.	How deep per pass.	Optional
FinStock.	Material left for finish pass.	Optional
ZFeed.	Z axis down feed.	Optional
RoughFeed.	Feedrate used for roughing passes.	Optional
FinFeed.	Feedrate for finish pass.	Optional
Tool#.	Tool to be used.	Optional

Circular profile
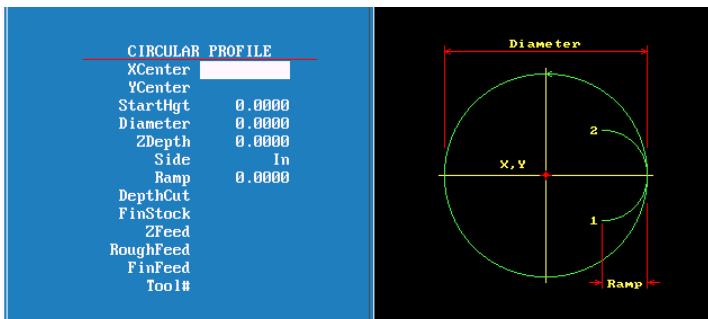
XCenter.	Center of profile along X axis.	Optional
Ycenter.	Center of profile along Y axis	Optional
StartHgt.	Height Above surface to be cut.	
Diameter.	Diameter of pocket.	
ZDepth.	Absolute depth of pocket.	
Side.	Inside or Outside.	
Ramp.	Size of ramp radius.	
DepthCut.	How deep per pass.	Optional
FinStock.	Material left for finish pass.	Optional
Zfeed.	Z axis down feed.	Optional
RoughFeed.	Feedrate used for roughing passes.	Optional
FinFeed.	Feedrate for finish pass.	Optional
Tool#.	Tool to be used.	Optional

Rectangular Pocket
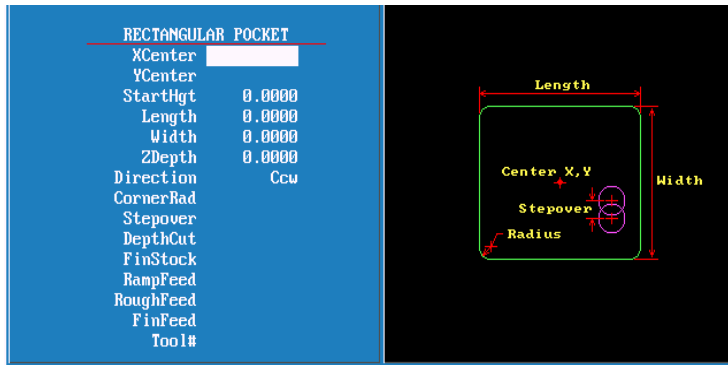
Xcenter.	Center of pocket X axis.	Optional
Ycenter.	Center of pocket Y axis.	Optional
StartHgt.	Must be .1" or 2mm above surface to be cut.	
Length.	Actual length of pocket.	
Width.	Actual width of pocket.	
Zdepth.	Absolute distance to bottom of pocket.	
Direction.	Defaulted to climb mill.	
CornerRad.	Radius in corners must be larger than cutter radius.	Optional
StepOver.	Step over between passes , cannot exceed 70% of cutter diameter.	Optional
DepthCut.	Depth of Z per pass.	Optional
FinStock.	Amount of material left for finish cut , material left on side and bottom.	Optional
RampFeed.	Feedrate for initial # axis move.	Optional
RoughFeed.	Rough feedrate.	Optional
FinFeed.	Finish feedrate.	Optional
Tool #.	Tool number normally not input here.	Optional

Circular Pocket
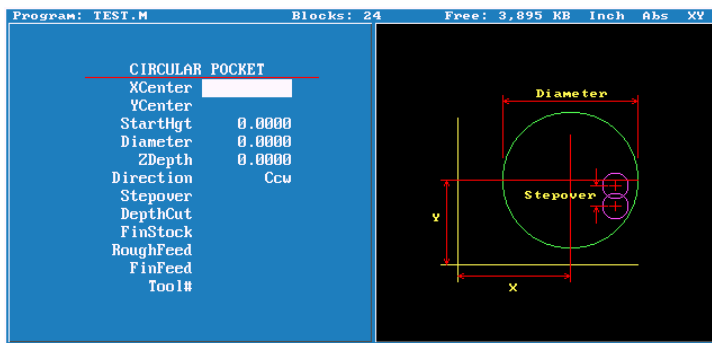
Xcenter.	Center of pocket X axis.	Optional
Ycenter.	Center of pocket Y axis.	Optional
StartHgt.	Must be .1" or 2mm above surface to be cut.	
Diameter.	Actual diameter of pocket.	
Zdepth.	Absolute distance to bottom of pocket.	
Direction.	Defaulted to climb mill.	
StepOver.	Step over between passes , cannot exceed 70% of cutter diameter.	Optional
DepthCut.	Depth of Z per pass.	Optional
FinStock.	Amount of material left for finish cut , material left on side and bottom.	Optional
RoughFeed.	Rough feedrate.	Optional
FinFeed.	Finish feedrate.	Optional
Tool #.	Tool number normally not input here.	Optional

Frame Pocket
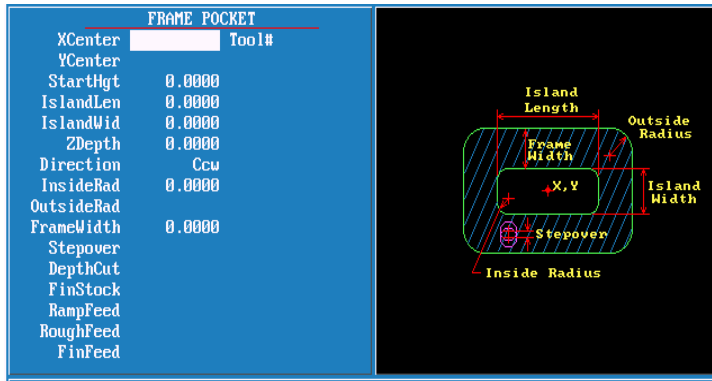

```
         FRAME POCKET
   XCenter  _____   Tool#
   YCenter
   StartHgt   0.0000
  IslandLen   0.0000
  IslandWid   0.0000
     ZDepth   0.0000
  Direction      Ccw
  InsideRad   0.0000
 OutsideRad
 FrameWidth   0.0000
   Stepover
   DepthCut
   FinStock
   RampFeed
  RoughFeed
    FinFeed
```

Xcenter	Center of pocket X axis.	Optional
Ycenter	Center of pocket Y axis.	Optional
StartHgt	Must be .1" or 2mm above surface to be cut.	
IslandLen	Actual length of center island.	
IslandWid	Actual width of center island.	
Zdepth	Absolute distance to bottom of pocket.	
Direction	Defaulted to climb mill.	
InsideRad	Radius in corners must be larger than cutter radius.	
OutsideRad	Radius outside corners , must be larger than radius of cutter.	Optional
FrameWidth	Distance from island to outside.	
StepOver	Step over between passes , cannot exceed 70% of cutter diameter.	Optional
DepthCut	Depth of Z per pass.	Optional
FinStock	Amount of material left for finish cut , material left on side and bottom.	Optional
RampFeed	Feedrate for initial # axis move.	Optional
RoughFeed	Rough feedrate.	Optional
FinFeed	Finish feedrate.	Optional
Tool #	Tool number normally not input here.	Optional

Hole-Mill Pocket


```
      HOLE-MILL POCKET
     XCenter  _____
     YCenter
     Diameter   0.0000
      ZDepth
    Direction      Ccw
     DepthCut
     StartHgt
     FinStock
    RoughFeed
      FinFeed
        Tool#
```
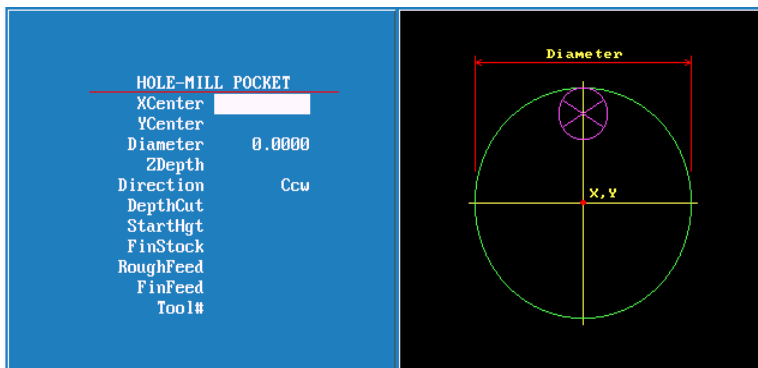
Xcenter	Center of pocket X axis.	Optional
Ycenter	Center of pocket Y axis.	Optional
Diameter	Actual diameter of pocket.	
Zdepth	Absolute distance to bottom of pocket.	Optional
Direction	Defaulted to climb mill.	
DepthCut	Depth of Z per pass.	Optional
StartHgt	Start height above surface to be cut.	Optional
FinStock	Amount of material left for finish cut , material left on side and bottom.	Optional
RoughFeed	Rough feedrate.	Optional
FinFeed	Finish feedrate.	Optional
Tool #	Tool number normally not input here.	Optional

Irregular Pocket
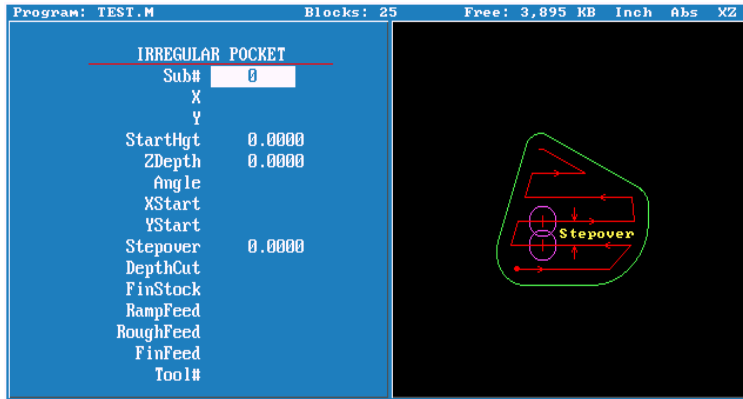
Sub#	**#** of profile subroutine.	
X	Start position of profile X axis.	Optional
Y	Start position of profile Y axis.	Optional
StartHgt	Start height .1" or 2mm above surface to be cut.	
Zdepth	Z depth of pocket absolute.	
Angle	Angle of first cut.	Optional
Xstart	Position of X axis before moving to start of profile.	Optional
Ystart	Position of Y axis before moving to start of profile.	Optional
Stepover	Distance cut will move over between passes.	
DepthCut	Depth of cut per pass.	Optional
FinStock	Amount of material left for finish pass. Leave stock on side and bottom of pocket.	Optional
RampFeed	Feedrate into material.Normally Z axis into material.	Optional
RoughFeed	Feedrate for roughing passes.	Optional
FinFeed	Feedrate for finish pass	Optional
Tool#	Tool #	Optional

Note

A subroutine has to be programmed for this cycle . The subroutine must start and end at the same coordinates. The first move can be a Rapid , put both X and Y axis in this block also the last block should have both X and Y axis coordinates.

Angle would normally only be used when starting point of profile is on a radius.

Mold Rotation


```
Program: TEST.M              Blocks: 25        Free: 3,895 KB  Inch  Abs  XZ

              MOLD ROTATION
       StartAngle    0.0000
        EndAngle     0.0000
          Cycles     0
          FwdSub     0
          RevSub     0
         AxisRot          X
         BAxisCL
         CAxisCL
          ZAngle
            Feed
           Tool#
```
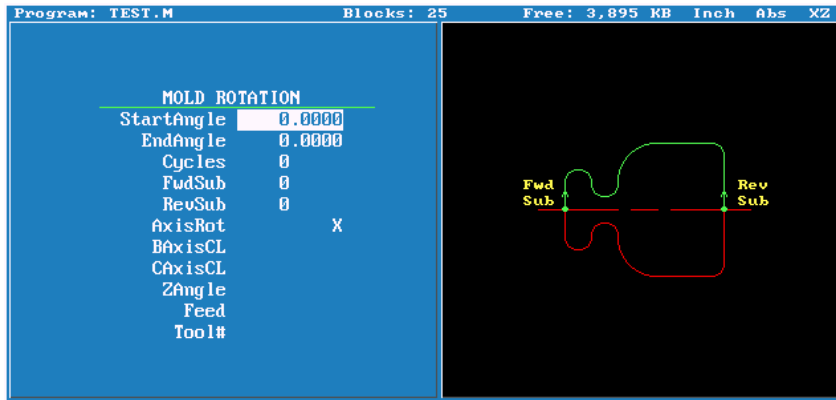
StartAngle	Angle at which rotation is going to start.
EndAngle	Angle at which rotation is going to end.
Cycles	Number of passes , one cycle is a pass in each direction.
FwdSub	Number of first subroutine.
RevSub	Number of second subroutine.
AxisRot	Axis around which rotation is going take place.
BAxisCL	Position of rotated axis if not zero.
CAxisCL	Position of second axis if not zero.
ZAngle	If rotating X or Y rotation around Z.
Feed	Feedrate
Tool#	Tool # .

Elbow Milling


```
Program: TEST.M              Blocks: 25        Free: 3,895 KB  Inch  Abs  XZ

            ELBOW MILLING
       Direction       Cw
        StartRad    0.0000
         EndRad     0.0000
       InclAngle    0.0000
         Cycles     0
        XCenter
        YCenter
           Feed
          Tool#
```
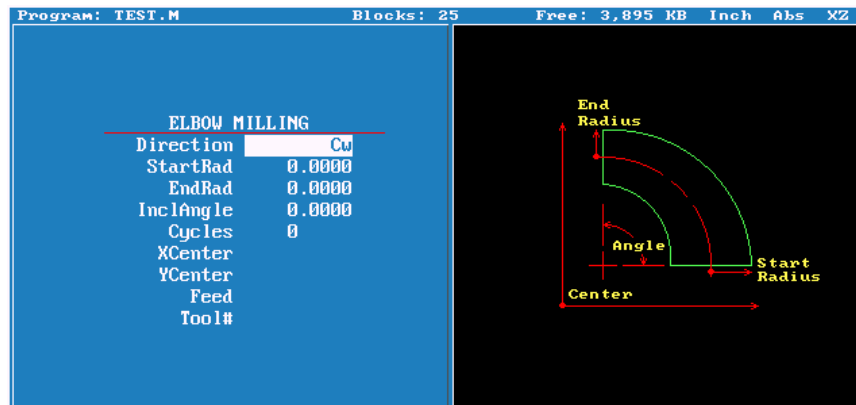
Direction	Cut direction of first pass.	
StartRad	Radius at start end.	
EndRad	Radius at opposite end.	
InclAngle	Included angle of cavity.	
Cycles	Number of passes , one cycle equals a pass in each direction.	
XCenter	Center of arc X axis.	Optional
YCenter	Center of arc Y axis.	Optional
Feed	Feedrate.	Optional
Tool#	Tool #.	Optional

Pockets with Islands

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

- DXF (see "Section 15, Using DXF for Pockets with Islands (G162)")
- Subroutines

More than one 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.

Using Subroutines for Pockets with Islands

This example using subroutinges for Pockets with Islands uses the following illustration. See **Figure 5-24** and **Table 5-1 Pockets with Islands Subroutines Programming Example**.

The numbers are the subroutine numbers.
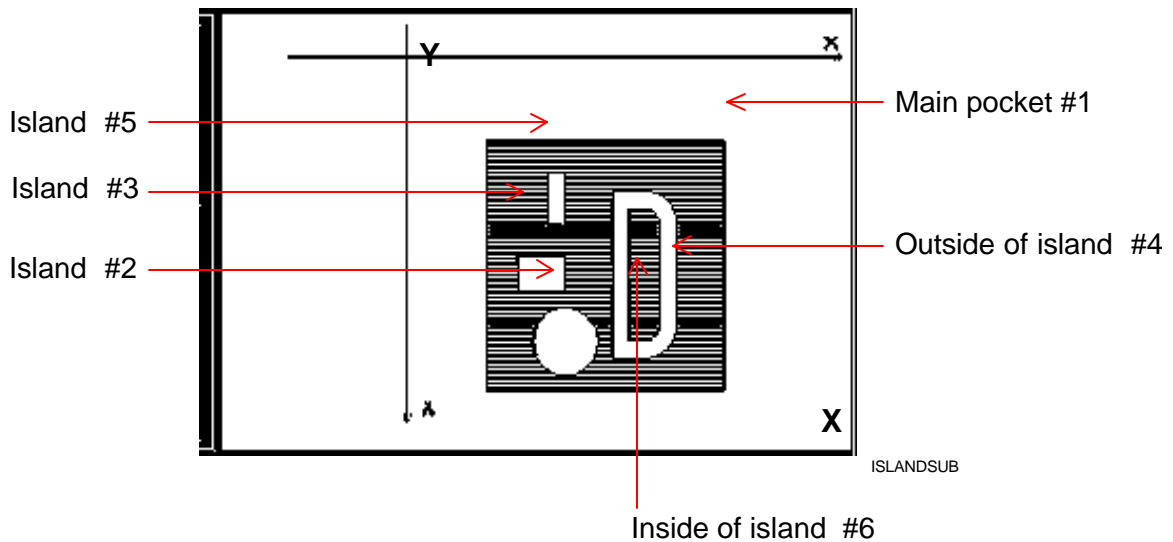
Figure 5-24, Subroutines Pockets with Islands Example Workpiece

In **Table 5-1** Island # 4 (FourthIsl) has a – (minus) in front of it, this is because the comp needs to be on the inside, as it is a pocket inside of an island.

Table 5-1 Pockets with Islands Subroutines Programming Example

Unit Inch	
Dim Abs	
Tool# 0	
Rapid	X 0.0000 Y 0.0000 Z 0.0000 Feed 50.0000
Islands	FirstIsl 4.0000 SecondIsl 3.0000 ThirdIsl 2.0000 FourthIsl -6.0000 FifthIsl 5.0000
Pocket	Sub# 1.0000 StartHgt 0.2000 ZDepth -0.9000 Stepover 0.1900 DepthCut 0.2500 RoughFeed 50.0000 FinStock 0.0100 FinFeed 50.0000
RampFeed 50.0000 Tool# 1.0000	
MCode 5 Z 5.0000	
EndMain	
Sub 1	
Rapid	X 5.0000 Y 5.0000 Feed 50.0000
Line	X 20.0000
Line	Y 20.0000
Line	X 5.0000
Line	Y 5.0000
EndSub	
Sub 2	
Rapid	X 9.0000 Y 7.0000 Feed 50.0000
Line	X 10.0000
Line	Y 10.0000
Line	X 9.0000
Line	Y 7.0000
EndSub	
Sub 3	
Rapid	X 7.0000 Y 12.0000 Feed 50.0000
Line	X 10.0000
Line	Y 14.0000
Line	X 7.0000
Line	Y 12.0000
EndSub	
Sub 4	
Rapid	X 13.0000 Y 8.0000 Feed 50.0000
Line	X 15.0000
Arc Ccw	X 17.0000 Y 10.0000 Radius 2.0000
Line	Y 16.0000
Arc Ccw	X 15.0000 Y 18.0000 Radius 2.0000

Line	X 13.0000
Line	Y 8.0000
EndSub	
Sub 5	
Rapid	X 8.0000 Y 17.0000 Feed 50.0000
Arc Ccw	X 12.0000 Y 17.0000 Radius 2.0000
Arc Ccw	X 8.0000 Y 17.0000 Radius 2.0000
EndSub	
Sub 6	
Rapid	X 14.0000 Y 9.0000
Line	X 15.0000
Arc Ccw	X 16.0000 Y 10.0000 Radius 1.0000
Line	Y 16.0000
Arc Ccw	X 15.0000 Y 17.0000 Radius 1.0000
Line	X 14.0000
Line	Y 9.0000
EndSub	

Teach	Draw	Drill	Pocket	Mill	Tool	Calc	Sub	Misc	Exit

There are two more canned cycles , to get to these press **F5** **Mill** **soft key.**

	Rapid	Line	Arc			More		Prev	

Soft key will change as above press **F7** **More** **a pop-up will appear as below.**

```
►Feed
 RPM
 Plane
 Unit
 Offset
 SetZero
 Home
 Ellipse
 Spiral
```
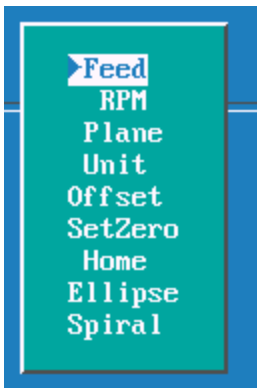
Ellipse and spiral both must be programmed incrementally .

Put height light on Ellipse press **ENTER** **screen will change as below.**

```
Program: TEST.M (edited)        Blocks: 25      Free: 3,895 KB  Inch  Abs  XY

                                                        Half Length
              ELLIPSE
          Direction         Cw                                      ___
                 X     0.0000                                       Half
                 Y     0.0000                                       Width
                 Z
           XCenter     0.0000                          Y Center
           YCenter     0.0000                         Y
         HalfLength    0.0000
         HalfWidth     0.0000
              Feed                               X
          CompSide                        X Center
             Tool#
```
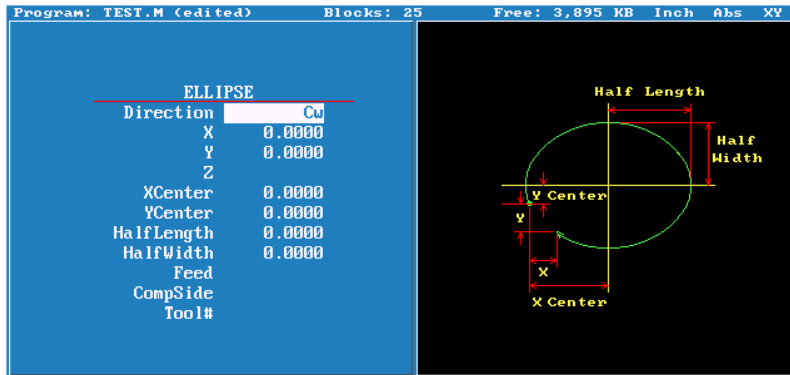
Direction   Direction of cut CW or CCW.
X   Distance from Start to End X axis of Ellipse.
Y   Distance from Start to End Y axis of Ellipse.
Z   Distance from Start to End Z axis of Ellipse.
Xcenter   Distance from Start to Center X axis of Ellipse.
Ycenter   Distance from Start to Center Y axis of Ellipse.
HalfLength   Half the length of Ellipse X axis.
HalfWidth   Half the width of Ellipse Y axis.
Feed   Feedrate.
CompSide   Tool compensation none , inside or outside.
Tool#   Tool number.

If plane is changed to XZ plane Ycenter would change to ZCenter and half width is Z axis .
If plane is changed to YZ plane Xcenter would change to YCenter and half length would be Y axis .
Using in side or outside tool compensation the the cutter must be placed in the correct compensated
position , before programming Ellipse. All dimension **MUST** be Incremental when programming this
cycle.

```
Program: TEST.M          Blocks: 25        Free: 3,895 KB  Inch  Abs  XZ

                    SPIRAL
            Direction        Cw
                   X
                   Y
                   Z
            XCenter     0.0000
            ZCenter     0.0000
              Revs      0
              Feed
             Tool#
```
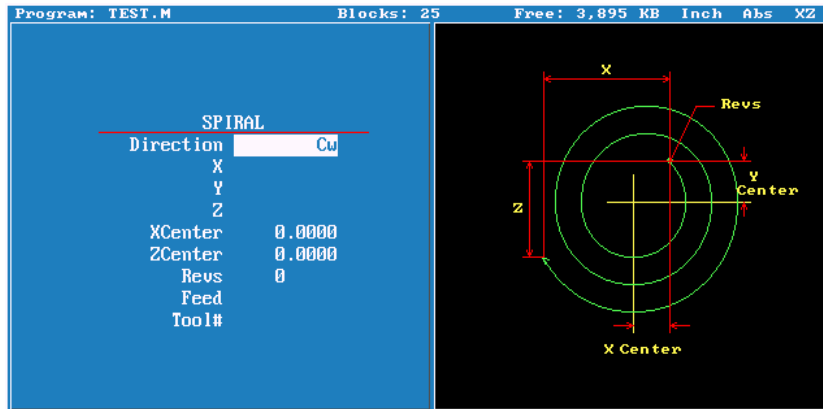
Direction	Direction of Spiral Clockwise or Counter Clockwise.
X	Distance from Start to End X axis .
Y	Distance from Start to End Y axis .
Z	Distance from Start to End Z axis.
Xcenter	Distance from Start to Center X.
Ycenter	Distance from Start to Center Y.
Revs	Number of Revolutions.
Feed	Feedrate.
Tool#	Tool number.

This can be programmed in XY , XZ or YZ planes , the center designations with change accord selected plane .
All dimension **MUST** be Incremental when programming this cycle
Cutter compensation no allowed with this cycle.
If cutting a thread using this cycle the distance moved in Z into number
of revolutions will equal lead of thread .

Engraving, Repeat, and Mill Cycles

This section describes operation of three new cycles:

- ❑ Engraving Cycle
- ❑ Repeat Cycle
- ❑ Mill Cycle

Engraving Cycle

The Engraving cycle provides a quick and easy way to engrave part numbers, legends, or any alpha/numeric inscription. The usual type of cutter is a sharp point or center-drill type tool. Options are given for engraving on an angle and mirror is supported for engraving molds. When executed, the CNC rapids to the start point, then to the StartHgt. It then feeds to the Zdepth specified and begins cutting the Text selected.

Programming the Engraving Cycle

To program the Engraving Cycle:

1. In Edit mode, press **Mill** (**F5**) and **More** (**F7**) to display the More pop-up menu, **Figure 5-48**. Highlight **Engrave** and press **ENTER** to display the Engraving Cycle screen, **Figure 5-49, Engraving Cycle Screen**.

2. Complete the entry fields (refer to **Table 5-2, Engraving Cycle Entry Fields**), and press **ENTER**.
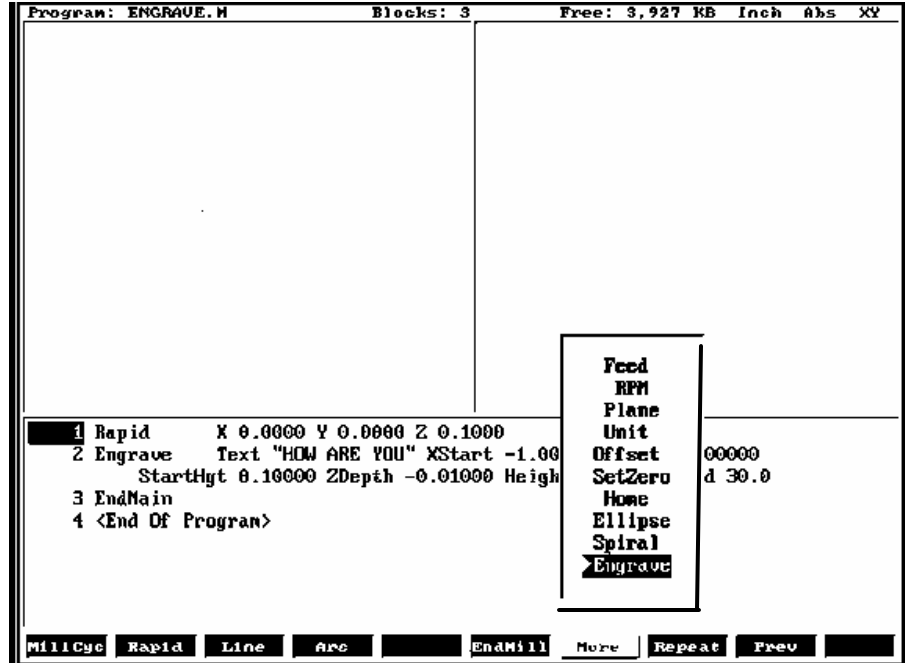
Figure 5-48, More Pop-up Menu
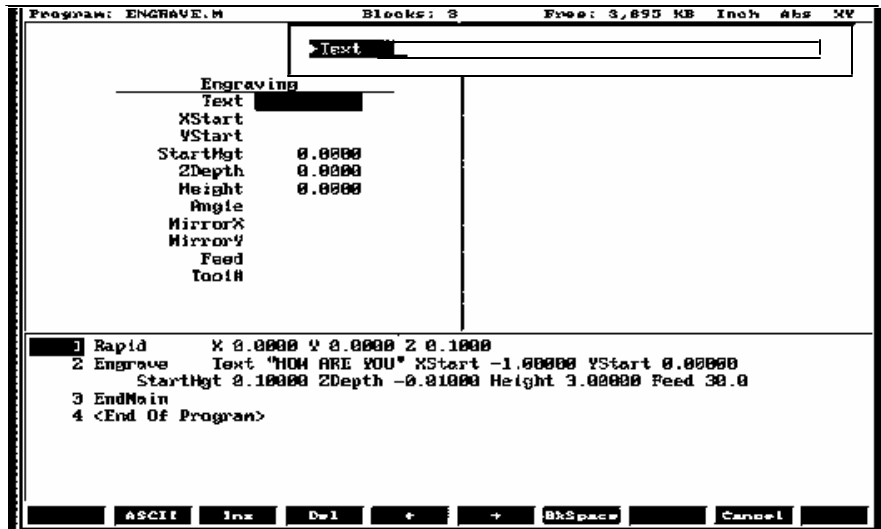
Figure 5-49, Engraving Cycle Screen

Table 5-2, Engraving Cycle Entry Fields

Entry Fields	Description
Text	When the cursor is on Text, it displays an entry field for the letters to be engraved. Letters A – Z, numbers 0 – 9, and: space, ampersand, plus, minus, comma, period, and slash right are supported. No lower case letters are allowed. If you have no external keyboard, use the ASCII Chart to enter text (see **Figure 2-4, ASCII Chart Pop-up**). Press **ENTER** to accept the text. [Required]
XStart	X coordinate for lower-left corner of the text. Defaults to current position if not given. [Optional]
YStart	Y coordinate for lower-left corner of the text. Defaults to current position if not given. [Optional]
StartHgt	Z absolute start height. Must be higher than Zdepth. [Required]
Zdepth	Z absolute depth of engraving. Must be below StartHgt. [Required]
Height	Letter height. Width will be proportional to height. Height is measured at the centerline of the cutter. [Required]
Angle	Angle in degrees. Default is 0 degrees. [Optional]
MirrorX	Mirrors all X moves. Set by using minus key (-) while in this field. [Optional]
MirrorY	Mirrors all Y moves. Set by using minus key (-) while in this field. [Optional]
Feed	Feedrate used while engraving. Default is current feedrate. [Optional]
Tool#	Active Tool [Optional] Used only on 3000M controls – not on 6000M

▶Text _

Engraving

A	B	C	D	E	F	G	H	I	J	K		7	8	9
L	M	N	O	P	Q	R	S	T	U			4	5	6
V	W	X	Y	Z								1	2	3
&			,				–	0	.
							+		/

1 <End Of Program>

Caps   ASCII   Ins   Del   ←   →   BkSpace   Cancel

Sample Engraving Cycle Program

1 Dim Abs
2 Unit Inch
3 Rapid   X 0.00000 Y 0.00000
4 Tool# 1
5 Rapid   X 1.00000 Y 1.00000
6 Rapid   Z 0.10000
7 Engrave   Text "ABCD" StartHgt 0.0100 ZDepth -0.0100 Height 0.5000
8 Rapid   Z 1.00000
9 Rapid   X 0.00000 Y 0.00000
10 EndMain

> This program will rapid to X1.0 Y1.0. Z will rapid to 0.1 and the letters ABCD will be engraved 0.0100" deep and 0.500" high.

Repeat Cycle

The Repeat cycle allows a series of previously programmed blocks to be repeated. Some examples are going over the same contour while lowering the Z-axis, or drilling over a series of holes with a different drill cycle, or moving an operation to a different location using fixture offsets. Wherever it is used, the repeated blocks will be processed, just as if they were written in the program at that point.

Programming the Repeat Cycle

To program the Repeat Cycle:

1. In Edit mode, press **Mill** (**F5**) and **Repeat** (**F8**) to display the Repeat Cycle screen, **Figure 5-50**.
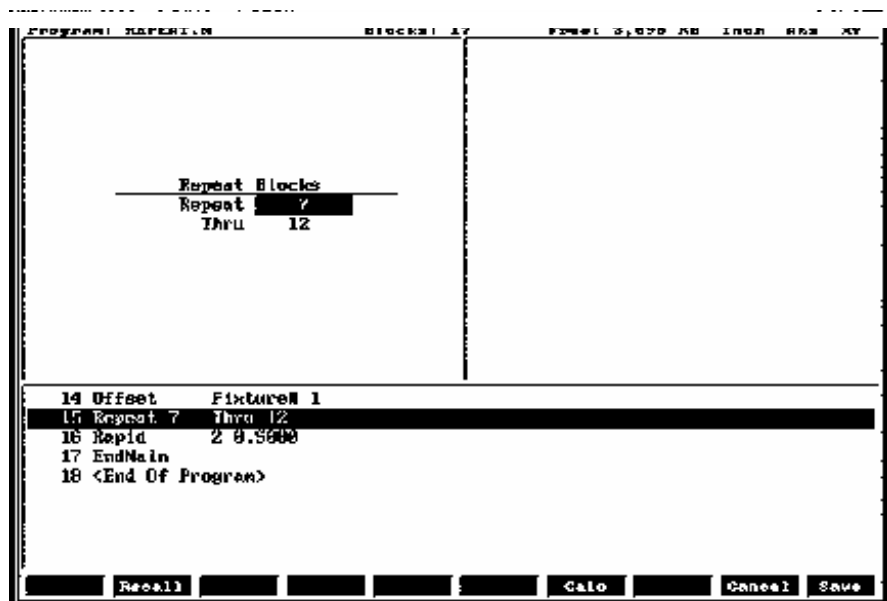
Figure 5-50, Repeat Cycle Screen

2. Complete the entry fields (refer to **Table 5-3**), and press
    ENTER.

Table 5-3, Repeat Cycle Entry Fields

Entry Field	Description
Repeat	Type the block number you want to begin repeating. [Required]
Thru	Type the block number you want to end the repeat. [Required]

3. When using a Modal Drilling Cycle with the Repeat feature, a
    DrillOff or non-move command must be included as the final
    block. For example, see "Sample Repeat Cycle Program"
    block 7–12 and
    block 15.

Sample Repeat Cycle Program

```
1       Dim Abs
2       Unit Inch
3       Offset     Fixture# 0
4       Rapid      X 0.0000 Y 0.0000
5       Tool# 1
6       Rapid      Z 0.1000
7       BasicDrill ZDepth –0.50000 StartHgt 0.10000 Feed 15.0
8       Rapid      X 1.00000
9                  Y 1.0000
10                 X 0.0000
11                 Y 0.0000
12      DrillOff
13      Offset     Fixture# 1 X 3.0000 Y 0.0000
14      Offset     Fixture# 1
15      Repeat 7   Thru 12
16      Rapid      Z 0.5000
17      EndMain
```

This program will drill four holes. A Fixture Offset is used to
relocate X Y zero. When the Repeat Cycle is encountered, it will
drill four more holes at the offset location.

Mill Cycle

The Mill cycle is intended for contour milling operations. Cutter compensation, Z pecking, Z finish stock, RoughFeed, and FinishFeed are supported. The cycle will rapid to the XY start point (compensated, if comp is on) rapid to the start height and then feed to the Zdepth or DepthCut using the Zfeed. Subsequent milling blocks are then executed using the ToolComp parameter and Feed specified. The feedrate can be changed in the blocks that are being milled, but not Cutter Comp. The cycle is terminated with the EndMill block at which point it rapids up to the StartHgt and returns to the un-comped XStart YStart location.

Programming the Mill Cycle

To program the Mill Cycle:

1. In Edit mode, press **Mill** (**F5**) and **MillCyc** (**F1**) to display the Mill Cycle screen, **Figure 5-51**.

2. Complete the entry fields (refer to **Table 5-4, Mill Cycle Entry Fields**), and press **ENTER**.

Programming the EndMill Block

To program the EndMill Block:

1. In Edit mode, press **EndMill** (**F6**) to end the cycle.
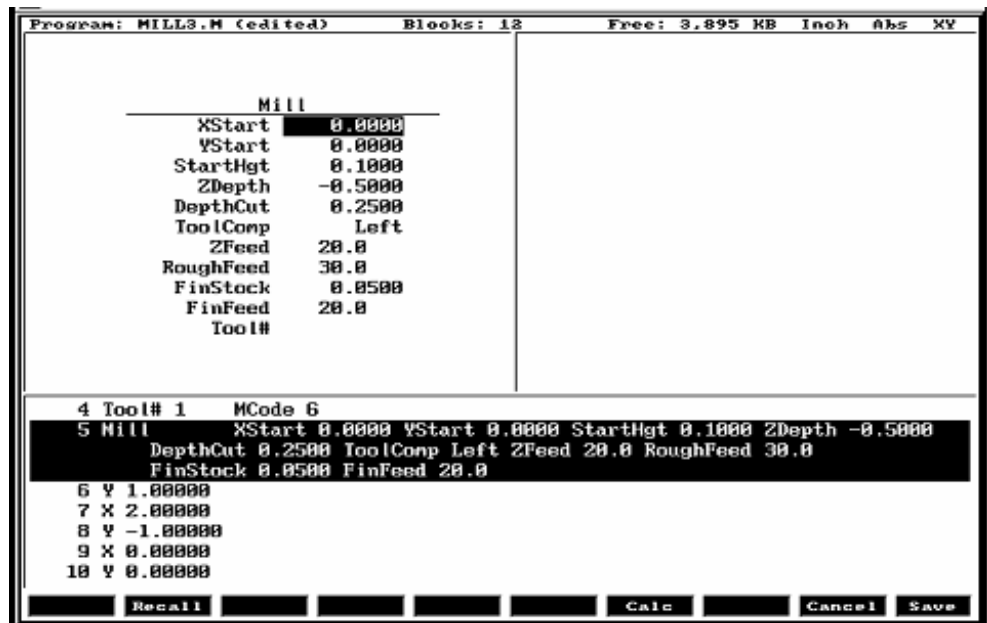
Figure 5-51, Mill Cycle Screen

Table 5-4, Mill Cycle Entry Fields

Entry Field	Description
XStart	X coordinate for start of Mill cycle. Defaults to current position if not given. [Optional]
YStart	Y coordinate for start of Mill cycle. Defaults to current position if not given. [Optional]
StartHgt	Z absolute start height. Must be 0.100" above work surface (0.2mm). [Required]
ZDepth	Absolute depth of finished contour. [Required]
DepthCut	Depth of cut taken in a single pass. Cuts will be adjusted so that all are equal pecks. [Optional]
ToolComp	Tool radius compensation Left or Right of programmed path. Set by using minus key (-) while in this field. [Optional]
ZFeed	Feedrate for Z-axis. Defaults to current feedrate. [Optional]
RoughFeed	Feedrate for X and Y-axis. Defaults to current feedrate. [Optional]
FinStock	Amount of stock to take for last Z peck. [Optional]
FinFeed	Feedrate used for FinStock. [Optional]
Tool #	Active Tool. [Optional] Used only on 3000M controls – not on 6000M

Sample Mill Cycle Program

```
1    Dim Abs
2    Unit Inch
3    Rapid     X 0.00000 Y 0.00000
4    Tool# 1    MCode 6
5    Mill      XStart 0.00000 YStart 0.00000 StartHgt 0.10000
     ZDepth -0.50000 DepthCut 0.25000 ToolComp Left ZFeed 20.0
     Feed 30.0
6    Y 1.00000
7    X 2.00000
8    Y -1.00000
9    X 0.00000
10   Y 0.00000
11   EndMill
12   EndMain
```

This program will contour a square, in two Z pecks of 0.250" each. The blocks 6 thru 10 are the contour moves that will be comped to the left of tool path direction. Block 11, EndMill is required to show the end of the contour. The cutter will be returned to the start point, X0 Y0 at the start height of 0.100".

3000M Rotate / Mirror / Scale

ANILAM
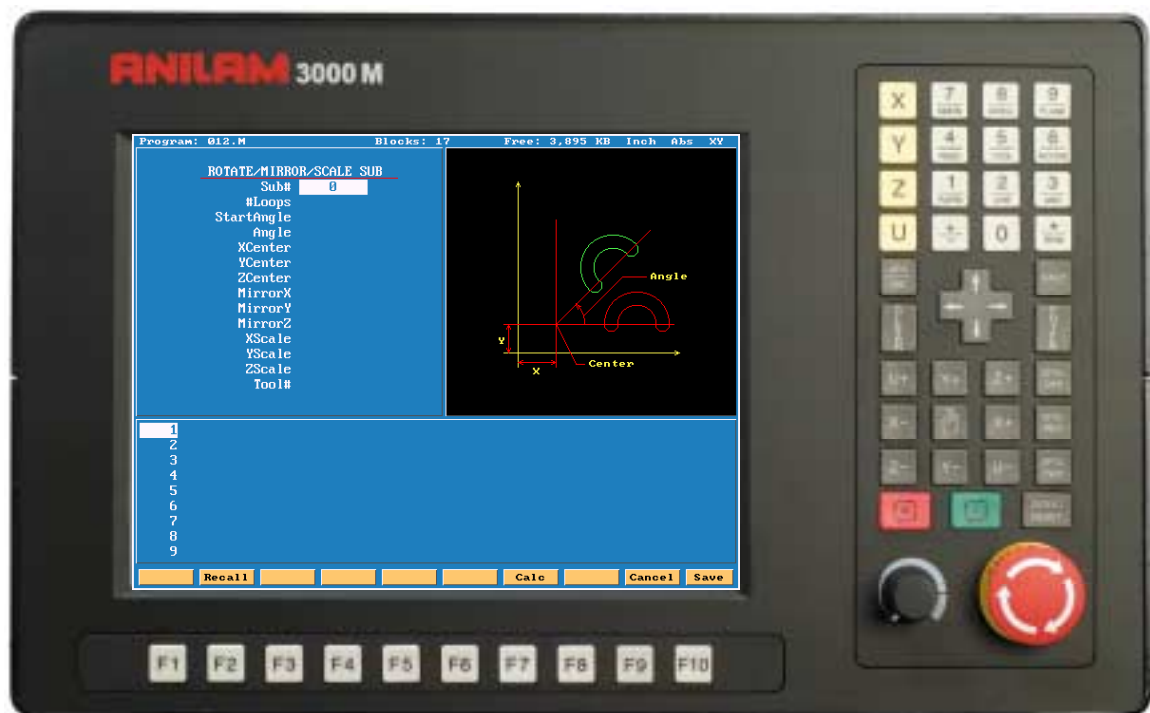
To get to RMS , when in edit press **F8** Sub

Press **F6** RMS

```
Program: TEST.M (edited)        Blocks: 3        Free: 3,895 KB   Inch   Abs   XY

        ROTATE/MIRROR/SCALE SUB
              Sub#    0
            #Loops
        StartAngle
            Angle
          XCenter
          YCenter
          ZCenter
          MirrorX
          MirrorY
          MirrorZ
           XScale
           YScale
           ZScale
            Tool#
```
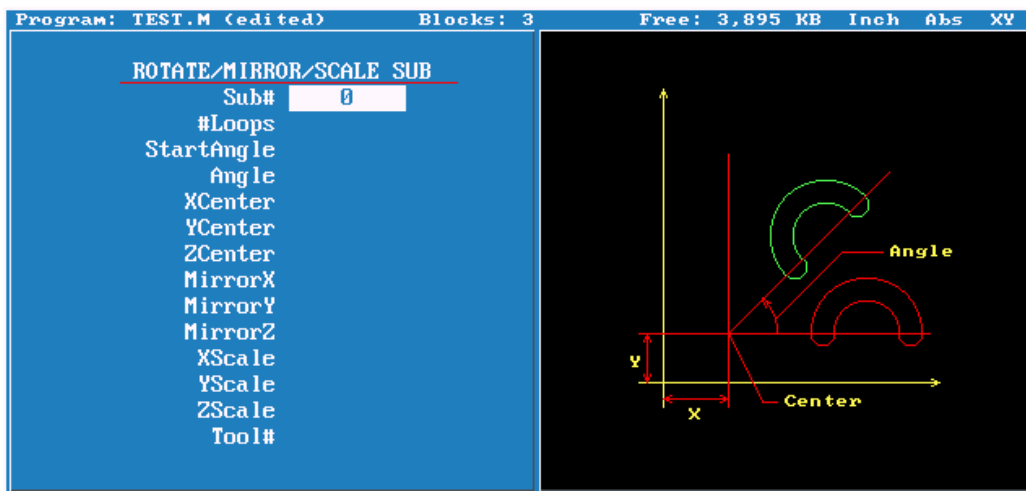
Sub#	Number of subroutine to Rotated, Mirrored or Scaled
#Loops	Number of time to repeat Rotation.
StartAngle	Start angle of rotation.
Angle	Angle between Rotations.
Xcenter	Center of Rotation X axis.
Ycenter	Center of Rotation Y axis.
Zcenter	Center of Rotation Z axis.
MirrorX	Mirror X axis.
MirrorY	Mirror Y axis.
MirrorZ	Mirroe z axis.
Xscale	Scale X axis.
Yscale	Scale Y axis.
Zscale	Scale Z axis.
Tool#	Tool number.

1.

Rotation

When using RMS a subroutine must be writen.

```
Program: TEST.M (edited)          Blocks:

            ROTATE/MIRROR/SCALE SUB
                Sub#      1
               #Loops     4
            StartAngle      0.0000
                Angle     90.0000
             XCenter      3.0000
             YCenter      3.0000
```

Above are the entries in canned cycle.

Press **F10**   Save

```
 1 Dim Abs
 2 Rapid       Z 0.0000 Tool# 0
 3 Tool# 1
 4 RMS          Sub# 1 #Loops 8 StartAngle 0.0000 Angle 45.0000
        XCenter 3.0000 YCenter 3.0000
 5 Rapid       Z 0.0000 Tool# 0
 6 EndMain
 7 Sub 1
 8 Rapid       X 4.0000 Y 3.0000
 9 Rapid       Z 0.1000
10 Line        Z -0.1000 Feed 5.0
11 Line        X 5.0000 Feed 10.0
12 Line        Z 0.1000
13 EndSub
```

This how the program will look, line #4 is the rotation cycle.
Note all moves including Z's are in the subroutine.
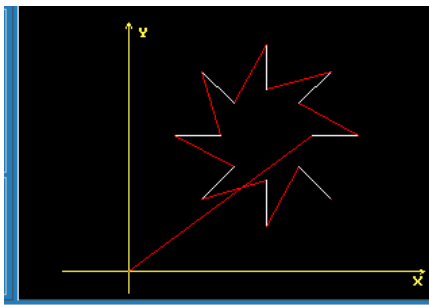
This is how it look in graphics.

In the case only one rotation is required the entry would be as below.
Note only 4 entries Sub , StartAngle , X & Y centers.

Program line would look as below.

Graphics of the single rotation appears below.
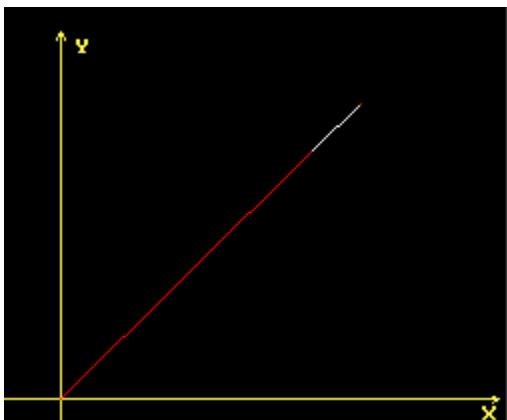
Mirror

Mirrror also requires a Subroutine to be writen.
Put hight light on axis to be mirrored press +/- key to turn on.

```
ROTATE/MIRROR/SCALE SUB
      Sub#        1
    #Loops
 StartAngle
      Angle
    XCenter
    YCenter
    ZCenter
    MirrorX        Yes
    MirrorY        Yes
    MirrorZ
```

Line #4 as programmed
Line #6 mirror imaged

```
 1 Dim Abs
 2 Rapid        Z 0.0000 Tool# 0
 3 Tool# 1
 4 Call 1
 5 RMS          Sub# 1 MirrorX Yes MirrorY Yes
 6 Rapid        Z 0.0000 Tool# 0
 7 EndMain
 8 Sub 1
 9 Rapid        X 1.0000 Y 1.0000
10 Rapid        Z 0.1000
11 Line         Z -0.1000 Feed 5.0
12 Line         X 3.0000 Feed 10.0
13 Line         Y 3.0000
14 Line         X 1.0000
15 Line         Y 1.0000
16 Line         Z 0.1000
17 EndSub
```


As programmed

Mirrored X & Y
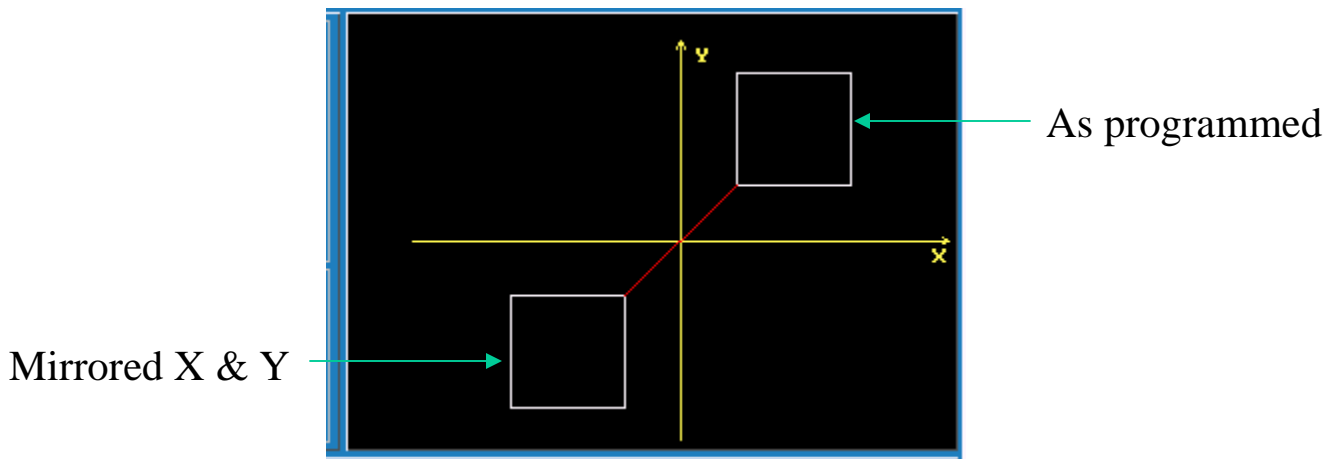
The following program shows the part cut in all four Quadrants. The one thing to keep in mind when using mirror image is that when using cutter compensation the cut direction will change in the diagonal quadrants.

```
 1 Dim Abs
 2 Rapid      Z 0.0000 Tool# 0
 3 Tool# 1
 4 Call 1
 5 RMS        Sub# 1 MirrorX Yes
 6 RMS        Sub# 1 MirrorX Yes MirrorY Yes
 7 RMS        Sub# 1 MirrorY Yes
 8 Rapid      Z 0.0000 Tool# 0
 9 EndMain
10 Sub 1
11 Rapid      X 1.0000 Y 1.0000
12 Rapid      Z 0.1000
13 Line       Z -0.1000 Feed 5.0
14 Line       X 3.0000 Feed 10.0
15 Line       Y 3.0000
16 Line       X 1.0000
17 Line       Y 1.0000
18 Line       Z 0.1000
19 EndSub
```


Mirror X — As programmed
Mirror X&Y — Mirror Y
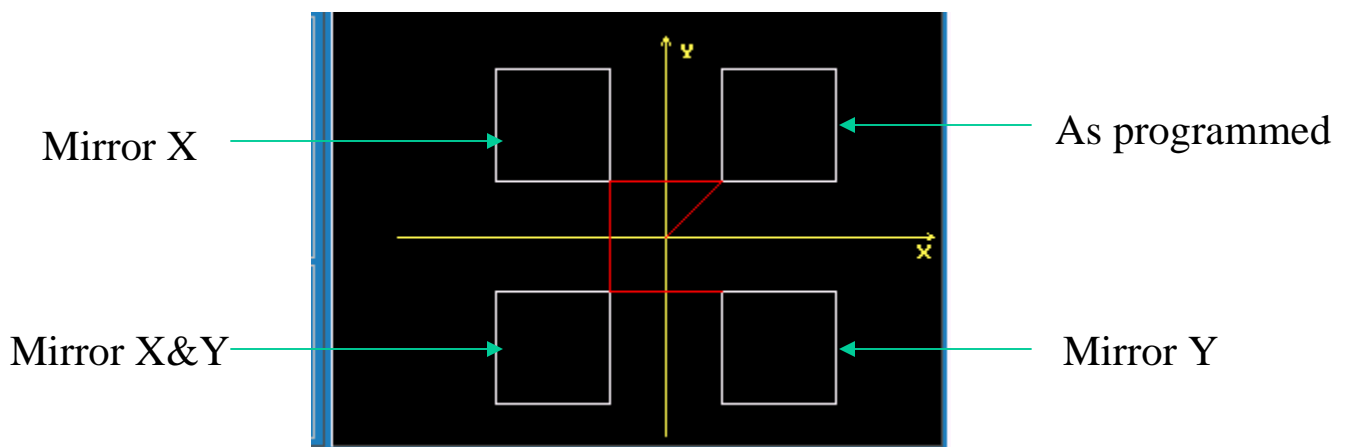
Scale

Scale allow programmer to change the size of the part.
One thing to remember is that if radii are involved both
axis must be scale the same amount.

```
 1 Dim Abs
 2 Rapid        Z 0.0000 Tool# 0
 3 Tool# 1
 4 Rapid        X 0.0000 Y 0.0000
 5 Call 1
 6 Rapid        X 0.0000 Y 0.0000
 7 RMS          Sub# 1 XScale 2.0000 YScale 2.0000
 8 Rapid        Z 0.0000 Tool# 0
 9 EndMain
10 Sub 1
11 Rapid        Z 0.1000
12 Line         Z -0.1000 Feed 5.0
13 Line         X 3.0000 Feed 10.0
14 Line         Y 3.0000
15 Line         X 0.0000
16 Line         Y 0.0000
17 Line         Z 0.1000
18 EndSub
```

Line #5 original line #6 scaled x2

Scaled x2

Original size
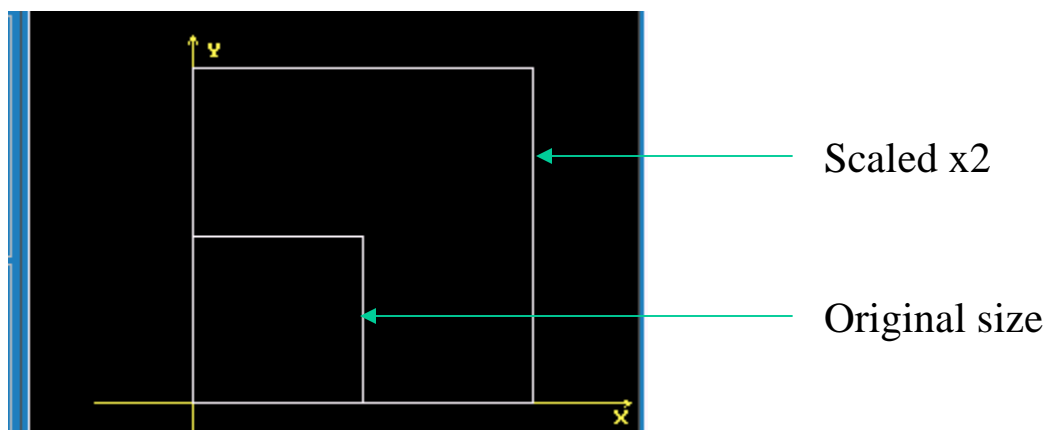
```
 1 Dim Abs
 2 Rapid      Z 0.0000 Tool# 0
 3 Tool# 1
 4 Rapid      X 0.0000 Y 0.0000
 5 Call 1
 6 Call 2
 7 Call 3
 8 Dim Abs
 9 Rapid      X 0.0000 Y 0.0000
10 RMS        Sub# 100 StartAngle 20.0000 XCenter 0.0000 YCenter 0.0000
      MirrorX Yes MirrorY Yes XScale 2.0000 YScale 2.0000
11 Rapid      Z 0.0000 Tool# 0
12 EndMain
13 Sub 100
14 Call 1
15 Call 2
16 Call 3
17 EndSub
18 Sub 1     *"A"
19 Dim Abs
20 Line Z-.005
21 Dim Incr
22 X0.0739 Y0.2300
23 X0.0164
24 X0.0739 Y-0.2300
25 Dim Abs
26 Rapid
27 Z.05
28 Dim Incr
29 X-0.1432 Y0.0657
30 Dim Abs
31 Line Z-.005
32 Dim Incr
33 X0.1221
34 Dim Abs
35 Rapid Z.05
36 Dim Incr
37 X0.0211 Y-0.0657
38 X.06
39 EndSub
40
41 Sub 02  * "B"
42 Dim Abs
43 Line Z-.005
44 Dim Incr
45 Y0.2300
46 X0.0780
47 Arc Cw X0.0000 Y-0.1068 XCenter0.0000 YCenter-0.0534
48 Line X-0.0780
49 X0.0863
50 Arc Cw X0.0000 Y-0.1232 XCenter0.0000 YCenter-0.0616
51 Line X-0.0863
52 Dim Abs
53 Rapid Z.05
54 Dim Incr
55 X0.1479
56 X.06
57 EndSub
58
59 Sub 03  * "C"
60 Dim Incr
61 Rapid X0.1508 Y0.0522
62 Dim Abs
64 Dim Incr
65 Arc Cw X-0.1413 Y0.0000 XCenter-0.0707 YCenter0.0217
66 Arc Cw X0.0000 Y0.1256 XCenter0.2041 YCenter0.0628
67 Arc Cw X0.1413 Y0.0000 XCenter0.0707 YCenter-0.0217
68 Dim Abs
69 Rapid Z.05
70 Dim Incr
71 X.06 Y-0.1778
72 EndSub
```

7.

8.
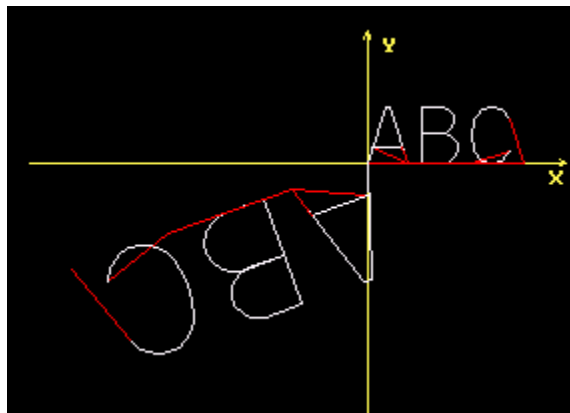
ANILAM

3000M Cut / Copy / Paste

ANILAM
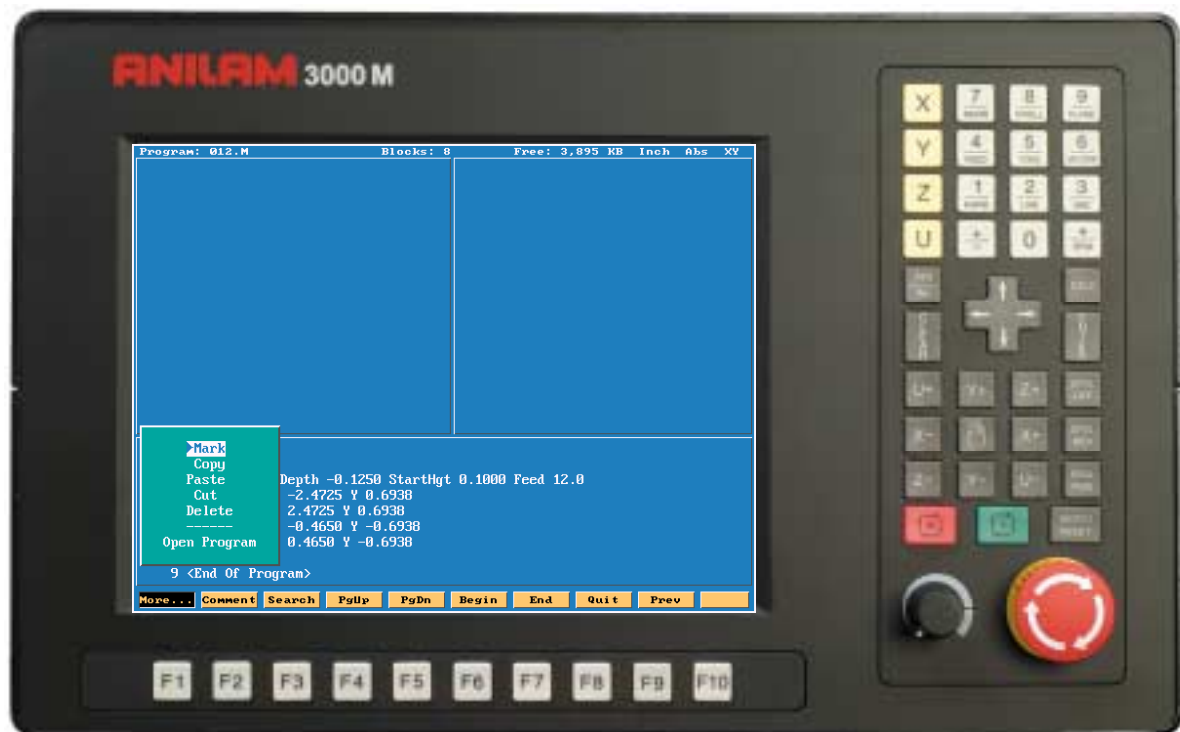
This section will deal will copying, pasting and other editting features.
Copying portions of programs and inserting then into another program.

Press   **F2**   **Program**

Press   **F2**   **Create**

Type program name 012.

Press   **F4**   **Edit**

```
1 Dim Abs
2 Tool# 1
3 BasicDrill ZDepth -0.1250 StartHgt 0.1000 Feed 12.0
4 <End Of Program>
```

Start writing program as above.

1.

| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

Softkeys will appear as above press **F9** **Misc**

| More... | Comment | Search | PgUp | PgDn | Begin | End | Quit | Prev | |

Press **F1** **More….**

```
Mark
Copy
Paste
Cut
Delete
----------
>Open Program
```

This pop-up window will come up curser down to **Open Program.**

Press **ENTER**

```
1 Dim Abs
2 Tool# 1
3 BasicDrill ZDepth -0.1250 Sta
4 <End Of Program>




Enter program name: 0123_
```

| | ASCII | Ins | Del | |
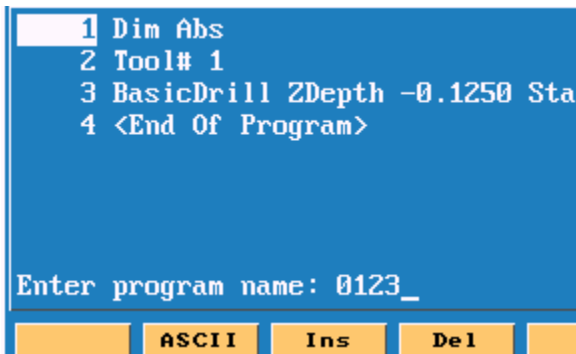
Enter name of program number (0123)

that you need to copy from press **ENTER**

New program will now be on screen.

```
3 PeckDrill   ZDepth -1.5000 StartHgt 0.
4 Rapid       X -2.4725 Y 0.6938
5 Rapid       X 2.4725 Y 0.6938
6 Rapid       X -0.4650 Y -0.6938
7 Rapid       X 0.4650 Y -0.6938
8 DrillOff
9 Rapid       X 0.0000 Y 0.0000 Z 0.0000
```

Press **F9**   **Misc**   Press **F1**   **More….**

```
►Mark
 Copy
 Paste        D
 Cut          r
 Delete
 ------
 Open Program
```

Press **ENTER** on mark.

```
3 PeckDrill   ZDepth -1.5000 StartHgt 0.1000 Pec
4 Rapid       X -2.4725 Y 0.6938
5 Rapid       X 2.4725 Y 0.6938
6 Rapid       X -0.4650 Y -0.6938
7 Rapid       X 0.4650 Y -0.6938
8 DrillOff
9 Rapid       X 0.0000 Y 0.0000 Z 0.0000 Tool# 0
```

Put high light on first block be to be copied.Press **ENTER**
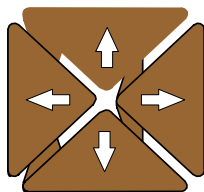
Use arrow keys to mark all blocks required.

ANILAM

Press **F1**   More….

Mark
►Copy
Paste
Cut
Delete

Open Program

Press ENTER

Mark
Copy
Paste
Cut
Delete

►Open Program

Enter original program number.

Enter program name: 012_

```
3 BasicDrill ZDepth -0.1250 StartHgt 0.1000 Feed 12.0
4 <End Of Program>
```

High light block #4 (End of Program.)

Press **F1**   More….

```
        Mark        D
        Copy        r
       ►Paste
        Cut
        Delete
        ──────
       Open Program
```
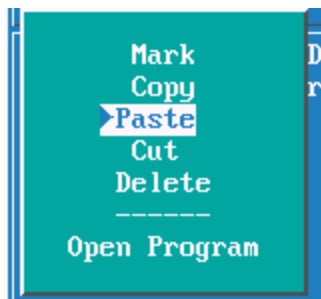
High light **Paste** press

```
ENTER
```

```
1 Dim Abs
2 Tool# 1
3 BasicDrill ZDepth -0.1250 StartHgt 0.1000 Feed 12.0
4 Rapid      X -2.4725 Y 0.6938
5 Rapid      X 2.4725 Y 0.6938
6 Rapid      X -0.4650 Y -0.6938
7 Rapid      X 0.4650 Y -0.6938
8 DrillOff
```

It inserted lines 4 - 8 into proggram #012.

Cut is used remove a section of program once it's marked.

Delete will cut pieces of marked program out.

3000M Calculators
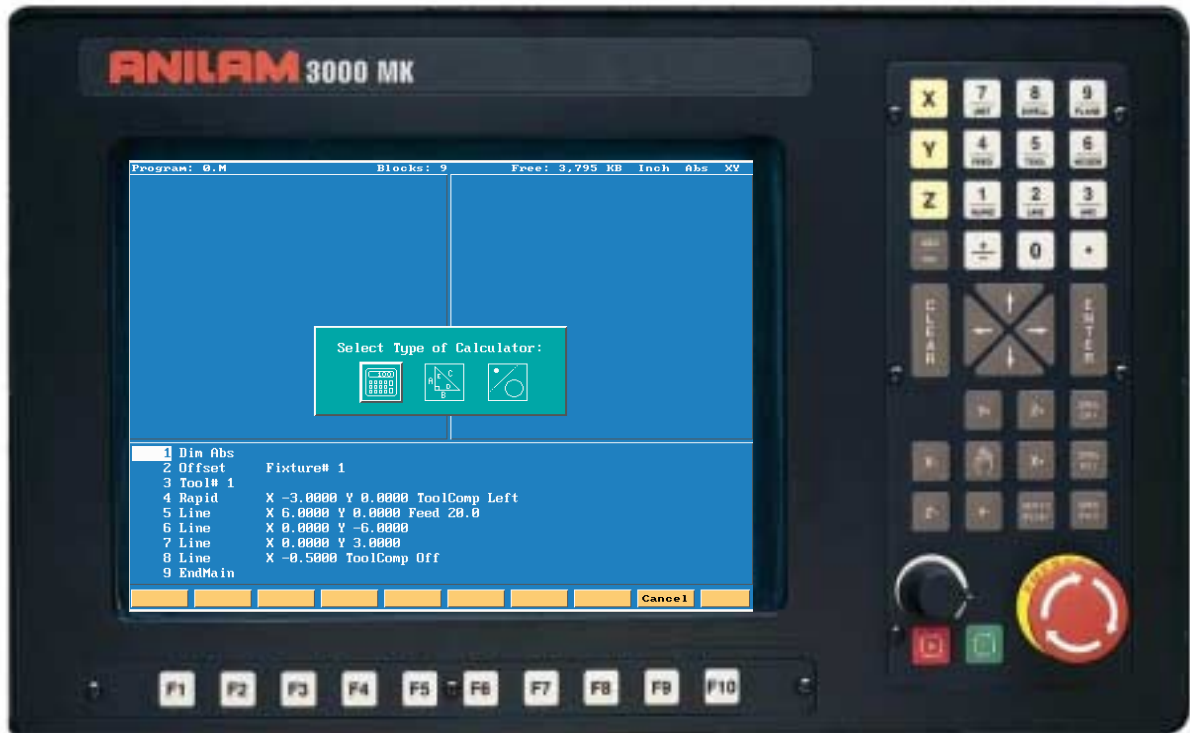
To get to calculator Press   **F4**   Edit

| | Create | Delete | Edit | List | Select | Log | Display | Utility | Exit |

Press   **F7**   Calc

| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

Box will appear as below.

1.Pocket.
2. Rightangled triangle.
3. Geometery

Select Type of Calculator:
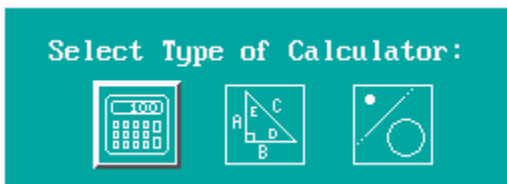
The box with the hight light around it is the active one. In this case
it is the left hand box.

1.

Press **ENTER** screen will appear as below.

Program: CALCUL.M   Blocks: 0   Free: 3,895 KB   Inch   Abs   XY

1 <End Of Program>

0.00000000

| + | − | * | / | (|) | Func | CE | Store | Exit |

Plus   Minus Multiply   Divide   Brackets   Functions Clear   Store   Exit

2.
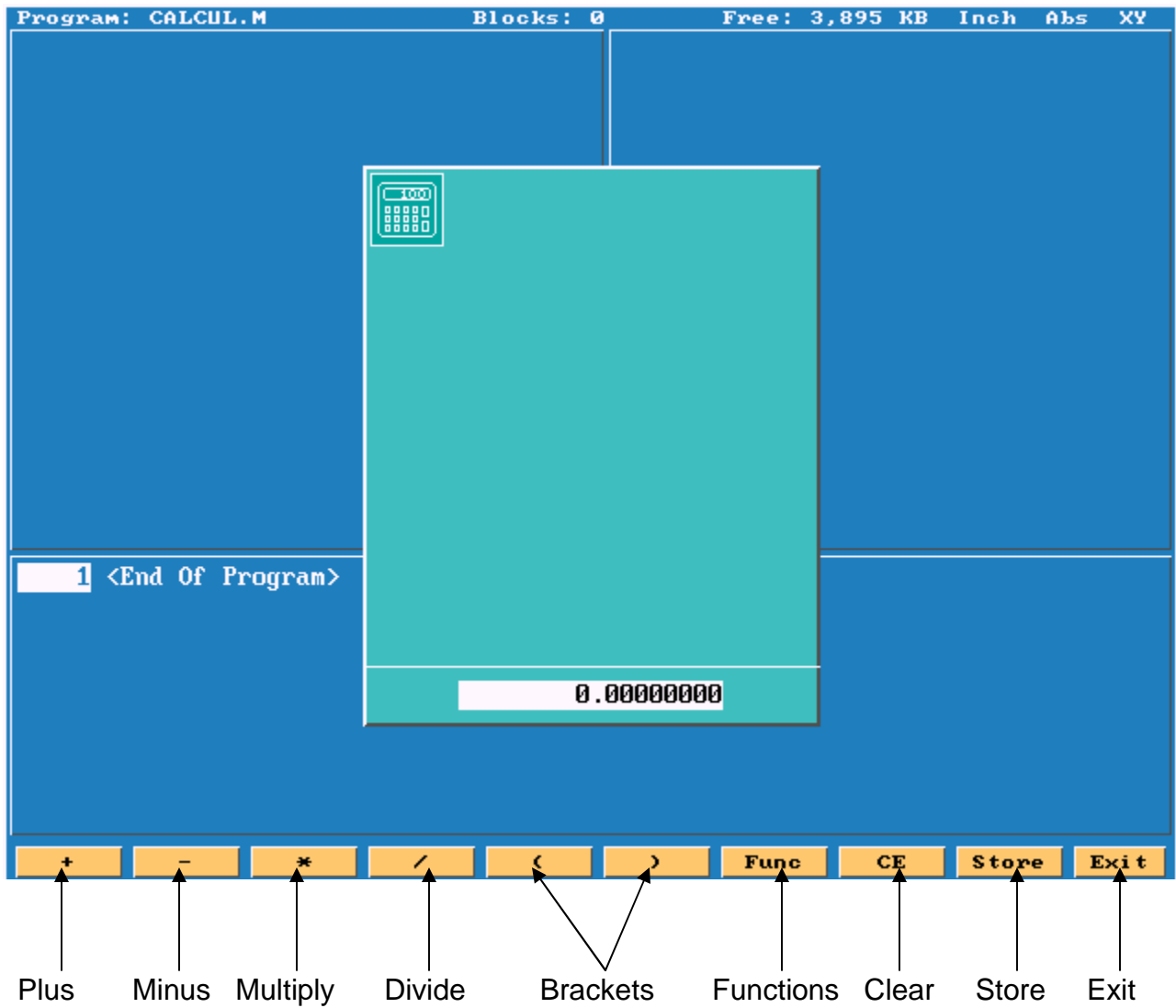
When **F7** **Func** is press the listed functions are available.

```
▶Trig - Sine
 Trig - Cosine
 Trig - Tangent
 Trig - ASine
 Trig - ACosine
 Trig - ATangent
 Math - SQRT
 Math - SQR
 Math - LN
 Math - Log
 Math - Exp
 Misc - ToMetric
 Misc - ToInch
 Misc - ToDegs
 Misc - ToRads
▶Misc - Inverse
```
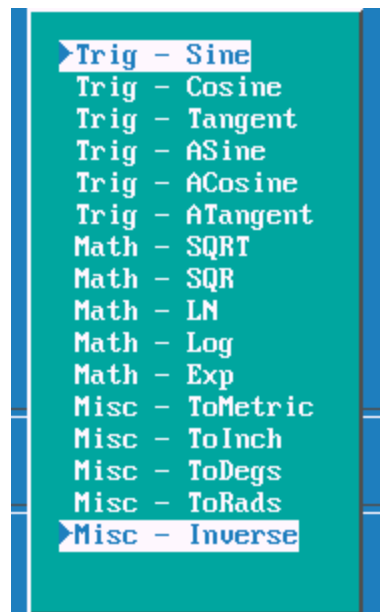
These funtions allow you to do trig and math problems.

Hight light center icon , this is rightangled triangle calculator.
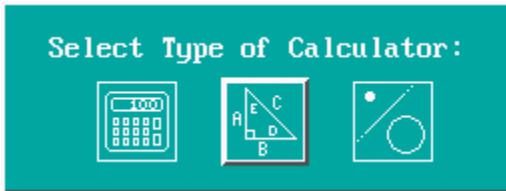
Press **ENTER** screen will appear as below.
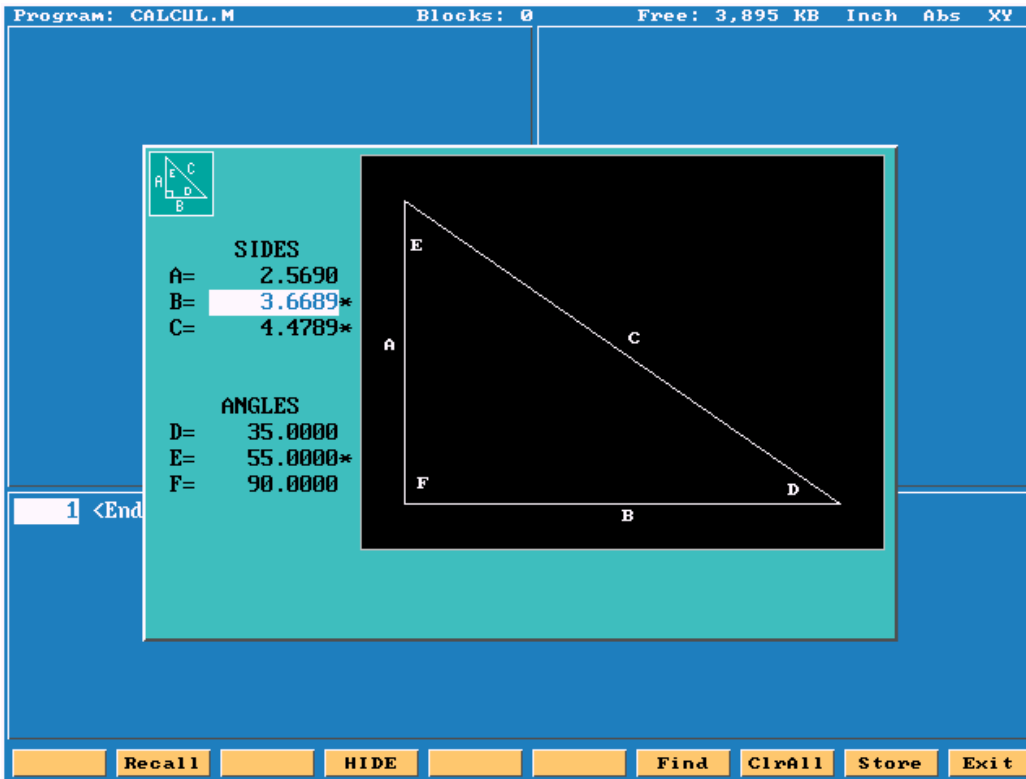
Enter any 2 sides or a side and an angle press **F7** **Find**
all of the blanks will be filled in , the calculated
dimensions will have an asterisk behind them.
They can be stored and recalled later into a program.

Hight light right icon , this is the geometry calculator.
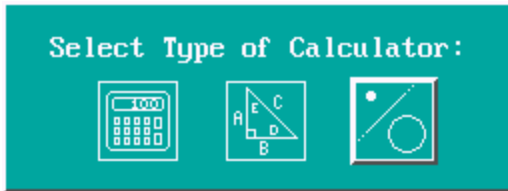
Press **ENTER** screen will appear as below.
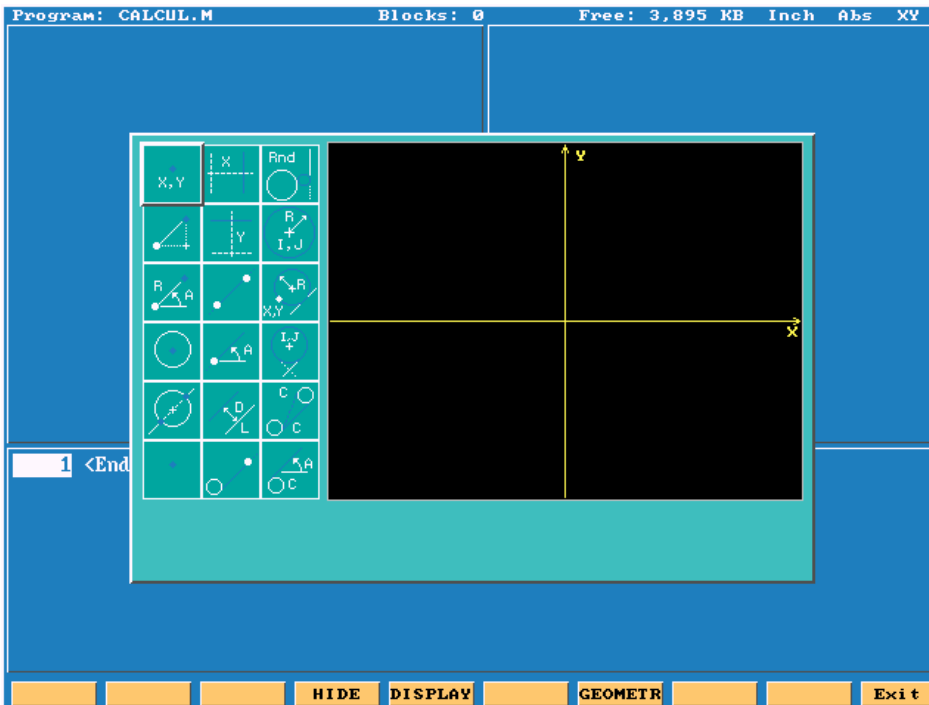
This calculator allows us to generate lines,points and circles.
We will need to gererate poins at all intersections , as points
are the only items we can recall.

The soft keys will appear as below.

F4   **HIDE**   Allow you change from calculator to program mode.

F5   **DISPLAY**   See below

F7   **GEOMETR**   See below

F10   **EXIT**   Exit Geometry calculator.

Fit to screen.
Zoom in to an area on the screen.
Redraw at current size.
Halfs the screen size.
Doubles screen size.
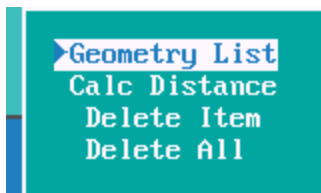
Lists all geometry.
Calculates distance between two points.
Deletes an item, need to give item number.
Deletes all geometry.

ANILAM

Point Definitions
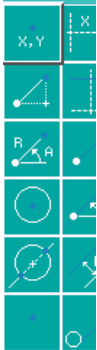
Point defined by co-ordinates **X** and Y

Point at a position **X & Y** from a previously defined point

Point at a distance **R** and an angle **A** from a previously defined point

Point at the centre of a circle

Point at an **INTERSECTION** between 2 elements

Point previously defined

Line Definitions

Vertical Line at a distance X from datum

Horizontal Line at a distance Y from part centreline

Line passing through 2 points

Line passing through a point at an angle **A**

Line parallel to another line **L** at a distance **D**

Line tangent to a circle passing through a point

Arc Definitions

Circle tangent to 2 geometry elements

Circle defined by a Centre **I & K** with a radius **R**

Circle passing through a point **X & Y** with a radius **R**

Circle tangent to a line with a centre **X & Y**

Line Tangent to 2 circles

Line tangent to a circle at an angle **A**

From the drawing below we are going to get all points required to program an irregular pocket.

6.25"

R 2.00

R 4.00

X0 Y0

3.75

R 1.8

8.
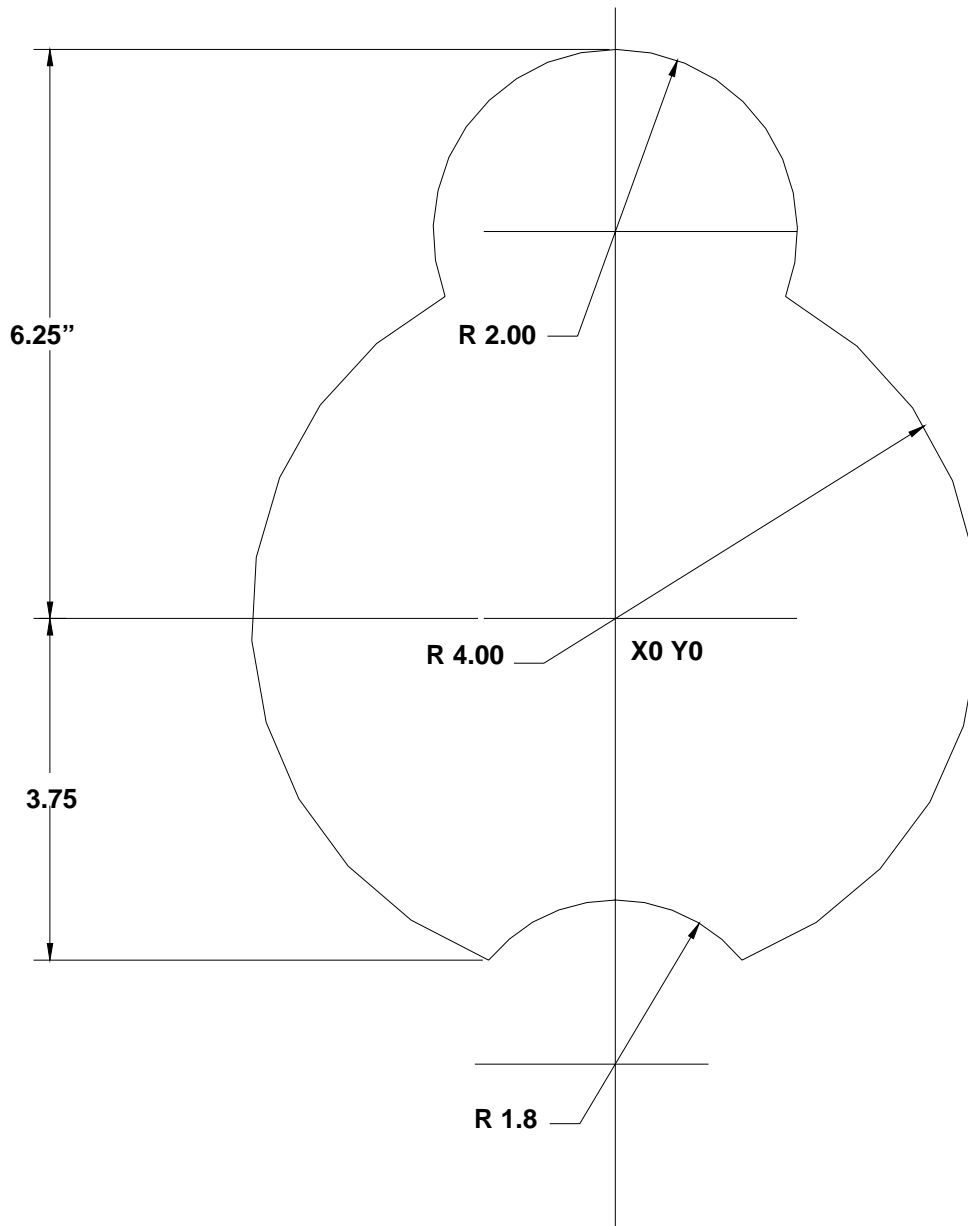
The first element to find is the 4.00 circle

Press **F4** | Edit | press **F7** | Calc

Select Type of Calculator:
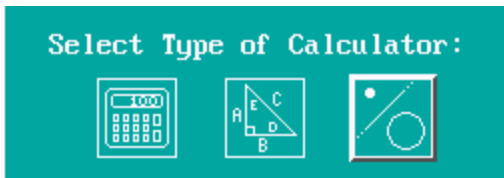
The third icon is hight lighted press **ENTER**

Press **ENTER**

Enter R value: 4
Select center definition ...
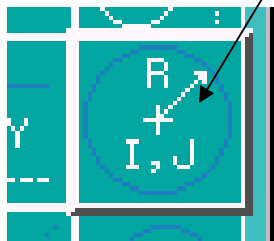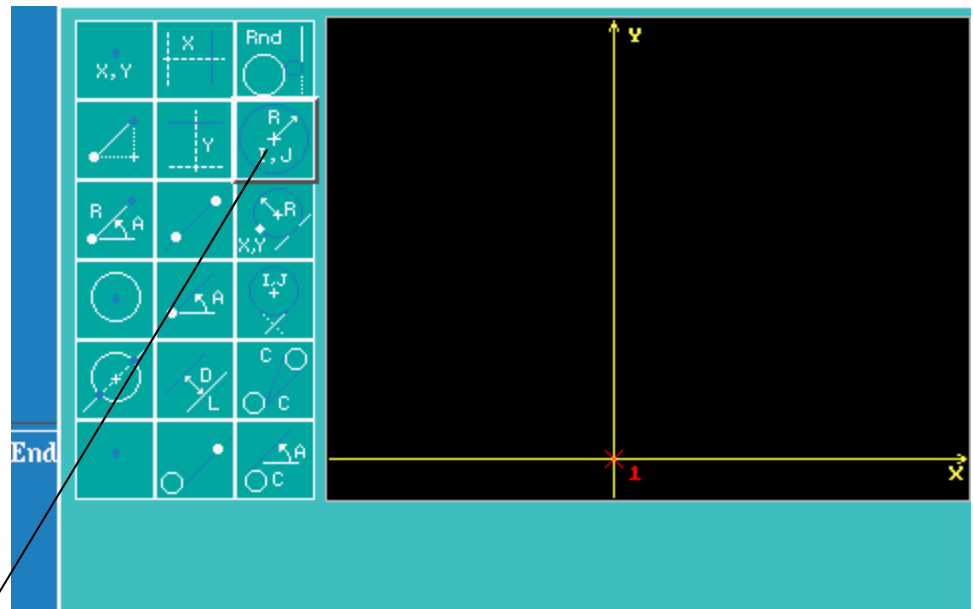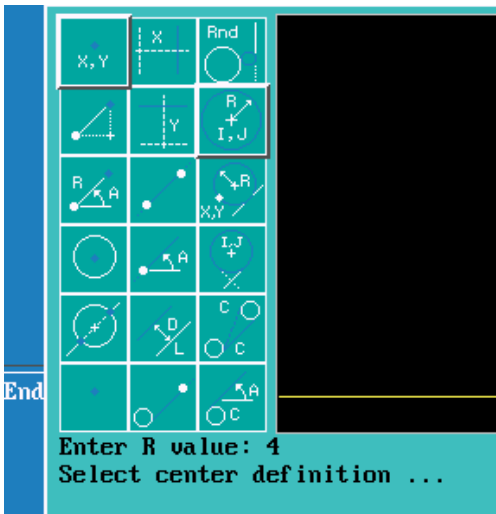
When **ENTER** is pressed screen will ask for Radius value, in this case 4., zero's not required.

Press **ENTER** It will now ask for a center definition and top left icon is hight lighted.
This is one to use in this case.

Enter R value: 4
Select center definition ...
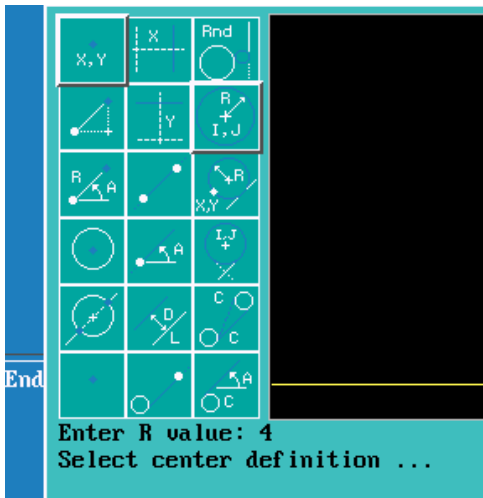
Press **ENTER**

It will now ask for an X value, in this case it 0 ,so press

```
E
N
T
E
R
```

It will now ask for an Y value, in this case it 0 ,so press

```
E
N
T
E
R
```



```
        ○      ○c
Enter X value:
Enter Y value: _
```
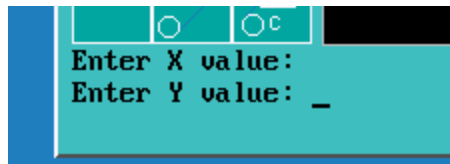
The circle will now appear on the screen, if it show as a dot on screen

press **F5** **DISPLAY** select **FIT** press

```
E
N
T
E
R
```

ANILAM
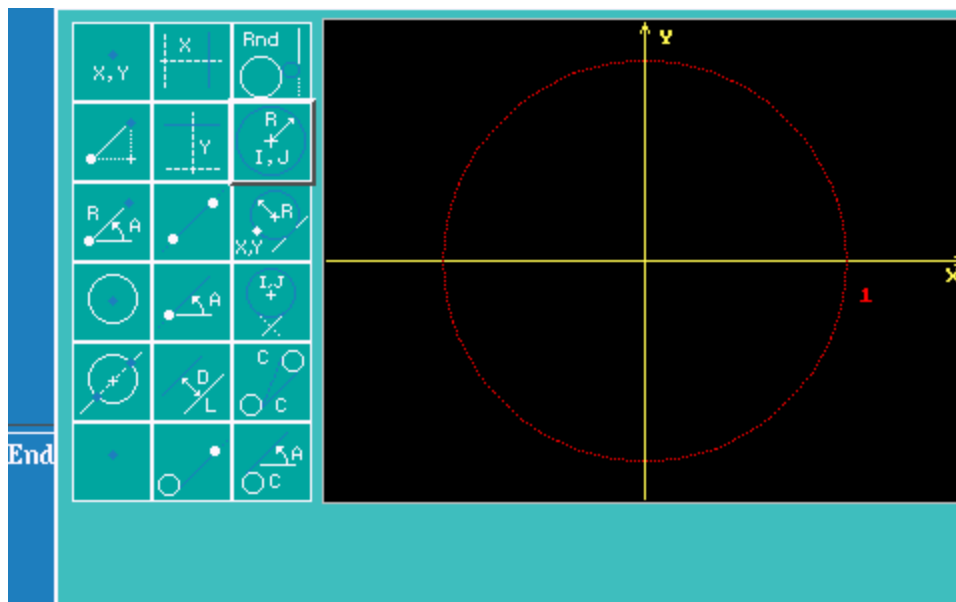
The 2" circle will now be entered. Use the same icon as before.
Radius 2" use same center icon **X 0** and **Y4.25** press **ENTER.**
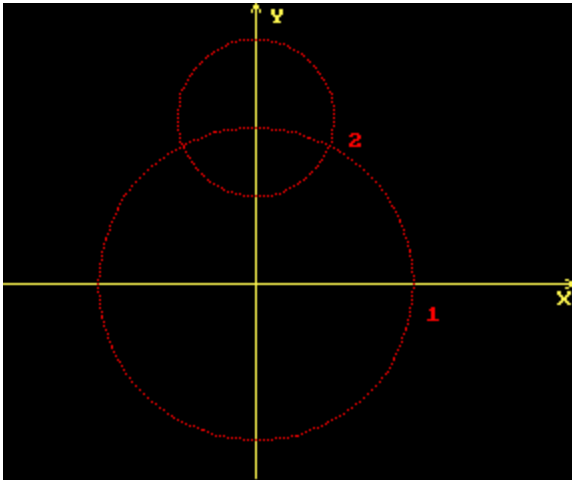
Second circle will now appear on screen, it may be necessary to fit again.

Note: The first circle has a 1 beside it and second has a 2. Each element
will be numbered.

The next element we need is a line at **Y**-3.75.

     Select **Y** Line icon and enter -3.75.   E N T E R
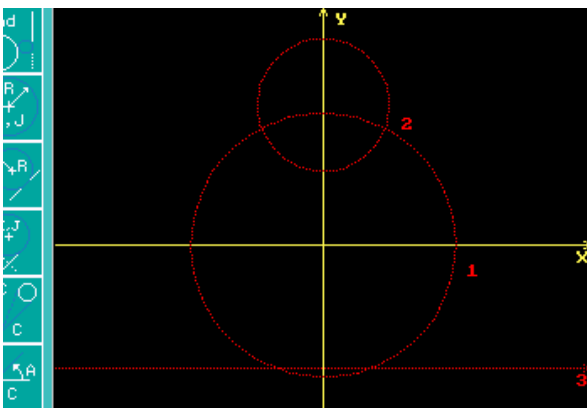
     There is now an element 3.

There is now enough geometry to find all the intersect required to program this part.

It is only possible to recall points into a program, so the thing that has to be done is find the points at all intersects using the icon below.

Required
intersects
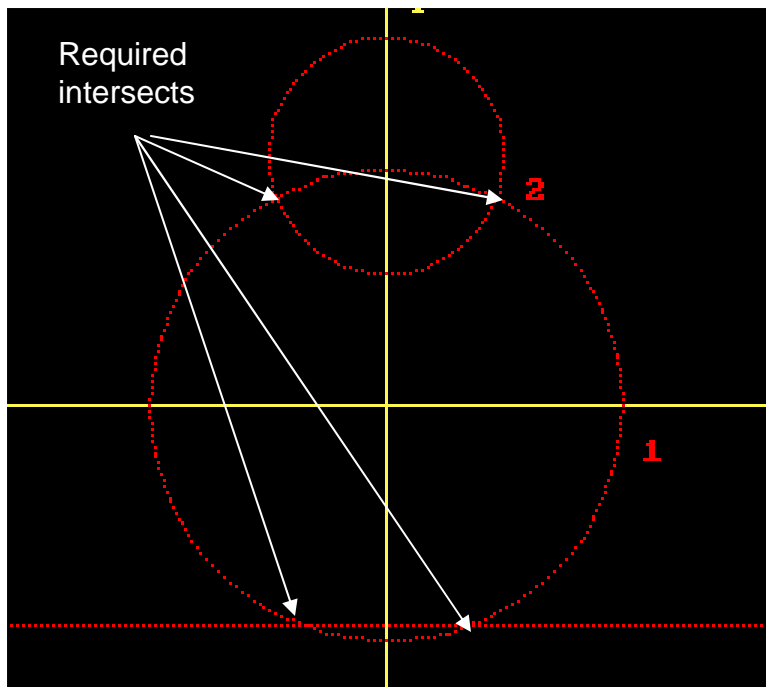
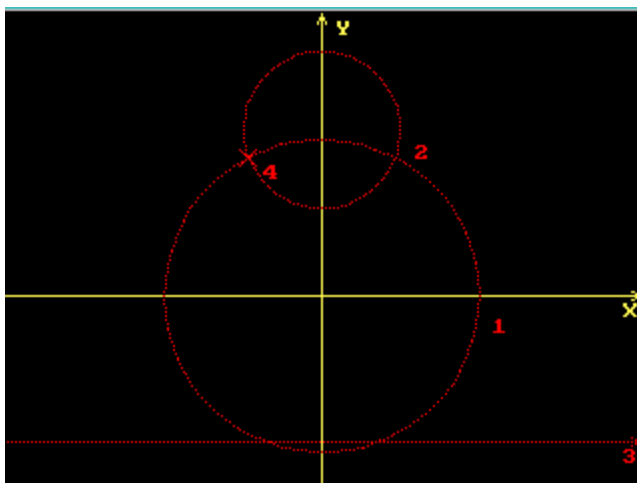The first intersect is between element 1 & 2 on the left side.

Select the icon with a line going through a circle.

Enter number of first element: 1
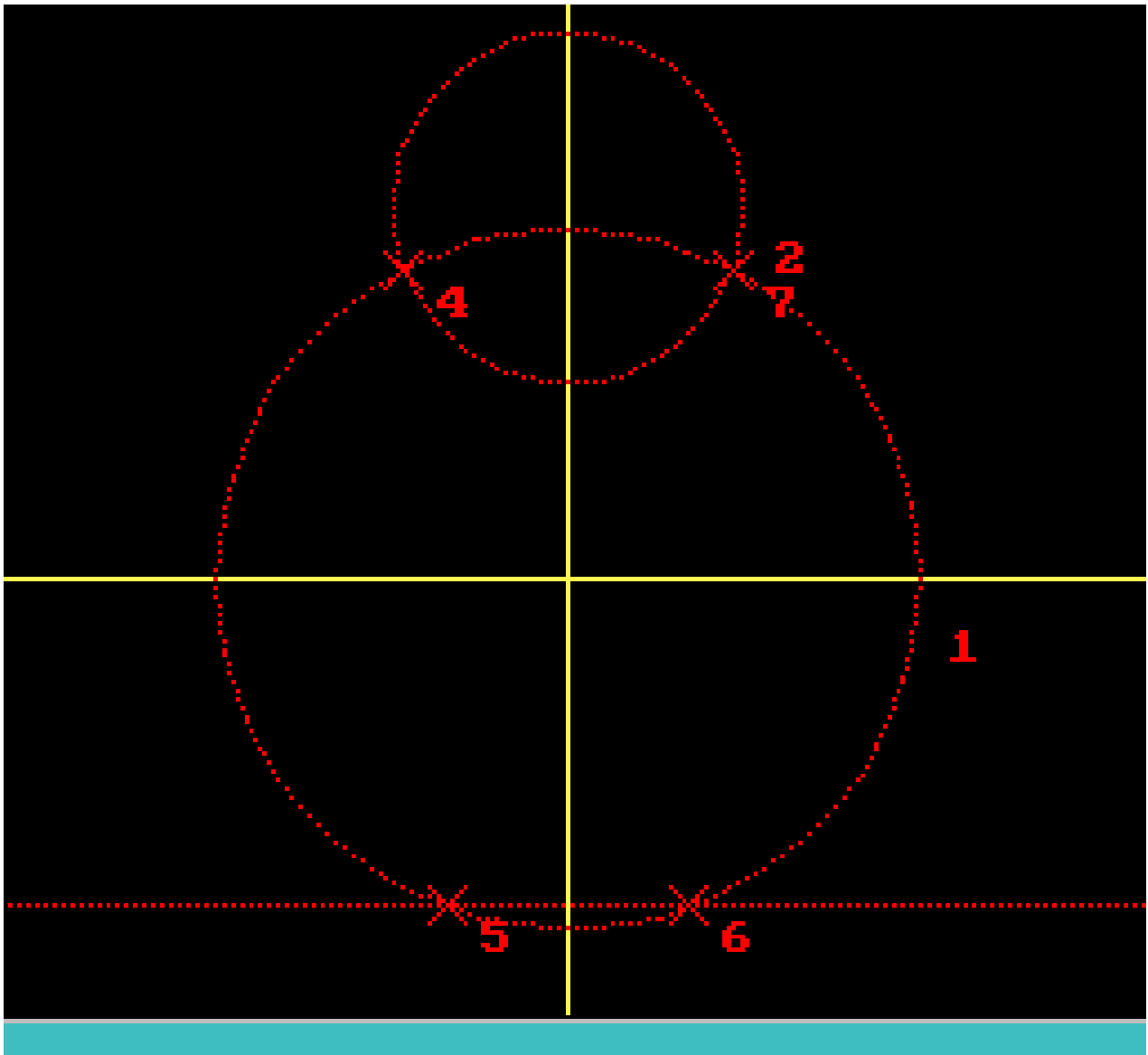Enter number of second element: 2
Select 1-2: _

Press **ENTER** select first element #1 select second #2, as there are two intersects there is a choice of 1 or 2 in this case the desired one is #1.

There is now an element #4 which is the intersect between #! And #2.

Using same icon,find intersects between ! & 3, two places and 1 & 2 right side.

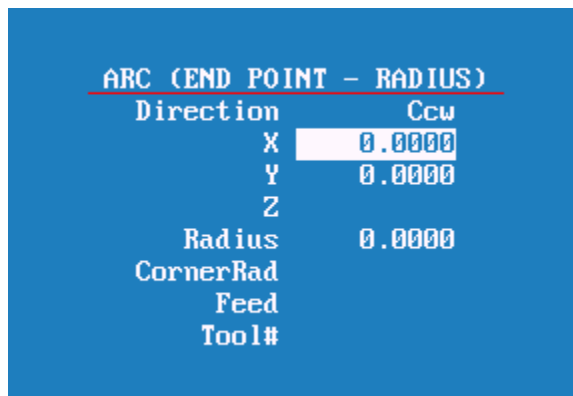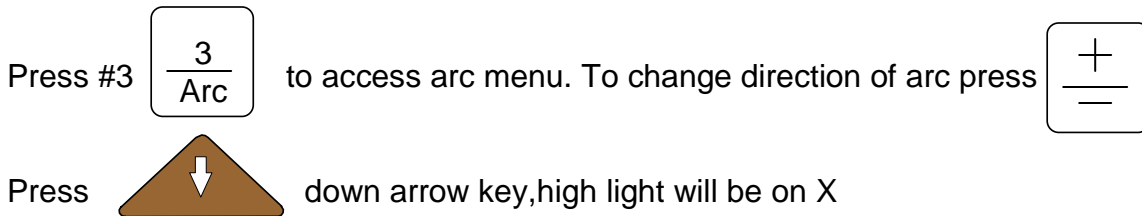Above is completed geometry, with all intersections marked with a point.

Writing program to pocket this shape.

Below is how main program would look, the next thing to do is write the profile of part to be pocketed.

```
Dim Abs                              * Absolute dimensioning
Rapid        Z 0.0000  Tool# 0        *Clear all offsets
Tool# 1                               * Call tool #1
Pocket                 Sub#1 StartHgt 0.1000 Zdepth -0.2500 Angle200 Stepover 0.3500
                       FinStock 0.0100 RampFeed 2.0 RoughFeed 15.0 FinFeed 15.0
                                        *Pocket cycle requires a subroutine of profile.
                                        *Cutter compensation is built into cycle.
                        *Angle is because of starting on a radius.
Rapid        Z0.0000 Tool# 0
EndMain
Sub 1                                 * Subroutine #1
Rapid        X 0.0000  Y6.2500         * This is the top 2" circle and the start point.
```

The next point needed is intersect between 2"radius and 4" radius circles.

Press #3 | 3 / Arc | to access arc menu. To change direction of arc press | +/− |

Press ▼ down arrow key, high light will be on X

```
ARC (END POINT - RADIUS)
    Direction          Ccw
            X       0.0000
            Y       0.0000
            Z
       Radius       0.0000
    CornerRad
         Feed
        Tool#
```

16.

Press **F2** Recall

```
ARC (END POINT - RADIUS)
     Direction          Ccw
            X     0.0000
            Y     0.0000
            Z

       Select point:

  1. X     0.0000 Y     0.0000 [C]
  2. X     0.0000 Y     4.2500 [C]
  4. X    -1.8685 Y     3.5368 [P]
  5. X    -1.3919 Y    -3.7500 [P]
  6. X     1.3919 Y    -3.7500 [P]
  7. X     1.8685 Y     3.5368 [P]

  9 <End Of Program>
```
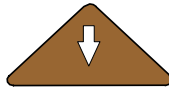
Point Required #4

The letter C at end of line means circle center and P means point.

Move high light down to #4. Using ↓ key press **ENTER** press **ENTER**

```
       Select point:

  1. X     0.0000 Y     0.0000 [C]
  2. X     0.0000 Y     4.2500 [C]
  4. X    -1.8685 Y     3.5368 [P]
  5. X    -1.3919 Y    -3.7500 [P]
  6. X     1.3919 Y    -3.7500 [P]
  7. X     1.8685 Y     3.5368 [P]
```

```
        Select term:
.00
.25  Both X and Y
.53  X only
.75  Y only
.75
```

```
ARC (END POINT - RADIUS)
    Direction          Ccw
           X    -1.8685
           Y     3.5368
           Z
      Radius     0.0000
   CornerRad
        Feed
       Tool#
```

Dimensions for X & Y will be input into program, move cruser down to Radius

enter 2" .

```
ARC (END POINT - RADIUS)
   Direction          Ccw
          X    -1.8685
          Y     3.5368
          Z
     Radius     2.0000
  CornerRad
```
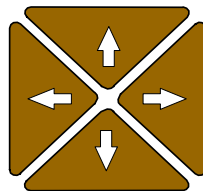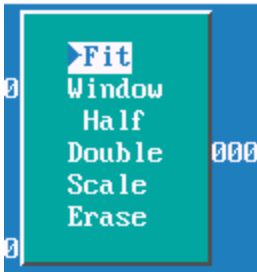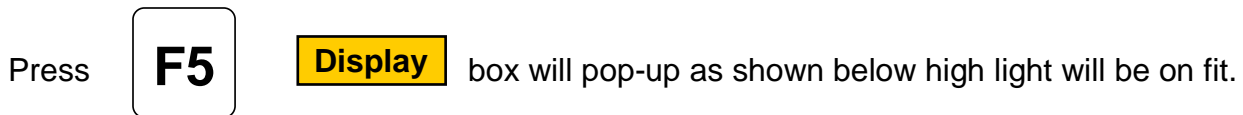
F10 Save

```
9 Arc Ccw    X -1.8685 Y 3.5368 Radius 2.0000
```

The same can now be done with points 5,6 & 7, the last point will be the same as the first X0 Y6.26 end point of 2" arc.
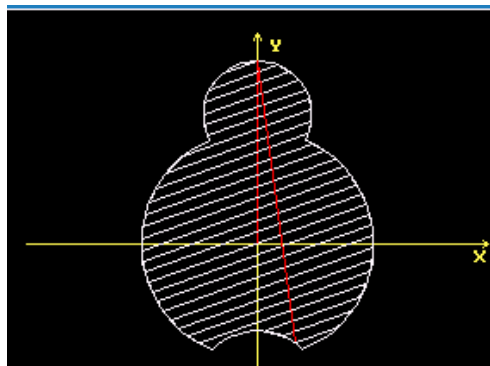
The following is how the program should look when complete.

```
Dim Abs
Rapid       Z 0.0000 Tool# 0
Tool# 1
Pocket      Sub# 1 StartHgt 0.1000 ZDepth -0.2500 Angle 200.0000 Stepover 0.3500
FinStock 0.0100 RampFeed 2.0 RoughFeed 15.0 FinFeed 15.0
Rapid       Z 0.0000 Tool# 0
EndMain
Sub 1
Rapid       X 0.0000 Y 6.2500
Arc Ccw     X -1.8685 Y 3.5368 Radius 2.0000
Arc Ccw     X -1.3919 Y -3.7500 Radius 4.0000
Arc Cw      X 1.3919 Y -3.7500 Radius 1.8000
Arc Ccw     X 1.8685 Y 3.5368 Radius 4.0000
Arc Ccw     X 0.0000 Y 6.2500 Radius 2.0000
EndSub
```

It is now time to check the program using the draw funtion.

Press **F2** Draw

Press **F5** Display box will pop-up as shown below high light will be on fit.

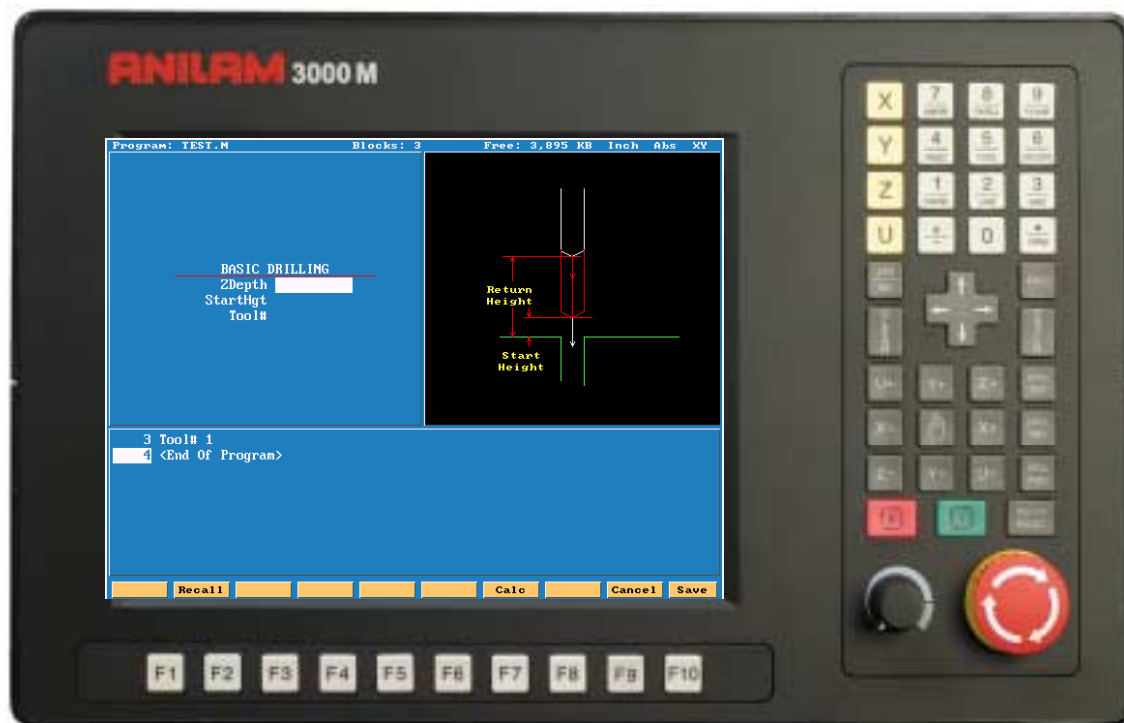Use arrow keys to move high light up and down.

Press E N T E R

18.

If all looks good in draw part is ready to cut.

Press   **F10**   Exit
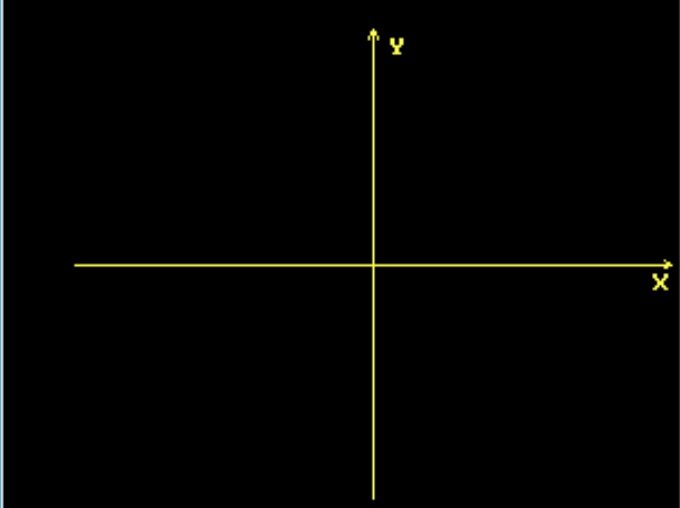
Press   **F10**   Exit

3000M Program Example

Typical starting program

X   0.0000
Y   0.0000
Z   0.0000

Block: 5
   Tool: Dia: 0.0000
Feedrate: 0.0   Feed
ToolComp: Off   Abs
   Loop: 0   XY

1 Dim Abs
2 Plane XY
3 Unit Inch
4 Offset   Fixture# 0
5 Tool# 0
6 Rapid   Z 0.0000
7 Offset   Fixture# 1
8 Tool# 1
9 Rapid   X 0.0000 Y 0.0000

| Auto | S.Step | Motion | Text | Tool | Rapid | Start | Hold | Cancel | |

ANILAM

Press **F2** | **Edit** to enter editor.

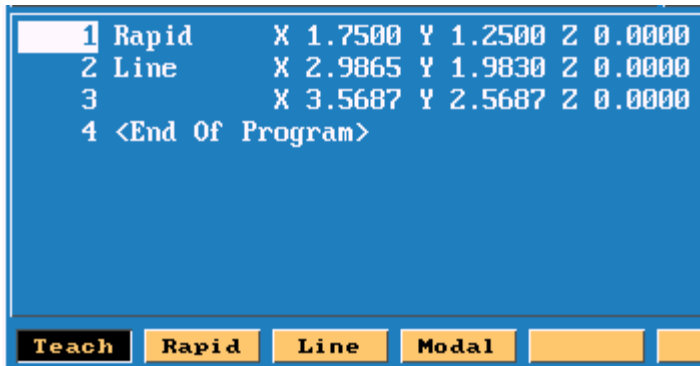| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

F1 — **Teach** — Dimension can be entered into by F2 Rapid , F3 feed or Modal.

F2 — **Draw** — Simulation draw allows checking program before run in auto.

F3 — **Drill** — Drilling canned cycles.

F4 — **Pocket** — Pocketing canned cycles.

F5 — **Mill** — Mill operations.

F6 — **Tool** — Tool Page.

F7 — **Calc** — Calculators Pocket , Right-angled triangle & Geometry.

F8 — **Sub** — Various auxiliary functions.

F9 — **Misc** — Miscellaneous functions

F10 — **Exit** — Exit to program page

Before entering **Teach Mode** you must create a program.
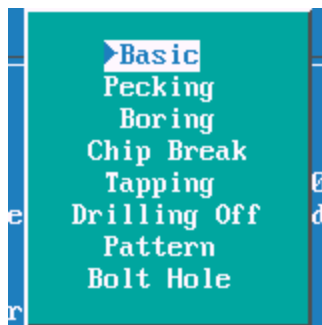
Press **F1** | **Teach** to enter teach mode.

Teach	Rapid	Line	Modal						

F1 **Teach** Exit from **Teach Mode.**

F2 **Rapid** Inputs a **Rapid** move.

F3 **Line** Inputs a **Line** move.

F4 **Modal** Inputs a **Modal** move.

```
1 Rapid      X 1.7500 Y 1.2500 Z 0.0000
2 Line       X 2.9865 Y 1.9830 Z 0.0000
3            X 3.5687 Y 2.5687 Z 0.0000
4 <End Of Program>
```

Teach	Rapid	Line	Modal		

1. Rapid input.
2. Line input
3. Modal input

Modal meaning it will do this move the same as previous move , in this case Line.

To exit **Teach** press **F1** | **Teach**

2.

F3   **Drill**

```
▶Basic
 Pecking
  Boring
Chip Break
 Tapping          0
Drilling Off      d
 Pattern
Bolt Hole
```

Drilling Cycles

Basic :-   Drills a hole one shot.

Pecking :-   Drills in steps depending on the amount of peck entered.

Boring :-   Feeds in And out of hole.

Chip Break::-   Used for deep holes , peck and then at specified depth retract all the way out of hole.

Tapping:-   Taps hole feeds and speed must be calculated correctly.

Drilling Off:-   Drilling must be turn **OFF** when done.

Pattern:-   Program a regular pattern of holes giving Number of holes, Distance between holes.

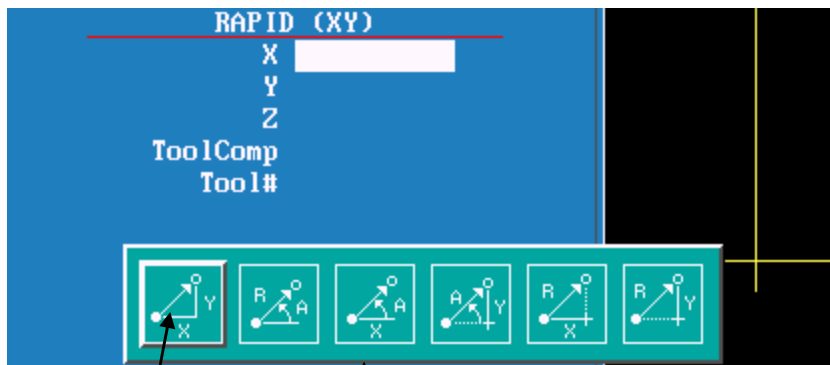**Bolt Hole:-**   Full or partial bolt hole may be programmed.

Mill

| F5 | | Mill |

| | Rapid | Line | Arc | | | More | | Prev | |

| F2 | | Rapid |

```
RAPID (XY)
        X  [          ]
        Y
        Z
ToolComp
   Tool#
```

Active one has a border

These are various way of entering a line or rapid move.

F4   Arc

```
ARC (END POINT - RADIUS)
    Direction            Cw
            X        0.0000
            Y        0.0000
            Z
    Radius           0.0000
  CornerRad
       Feed
      Tool#
```
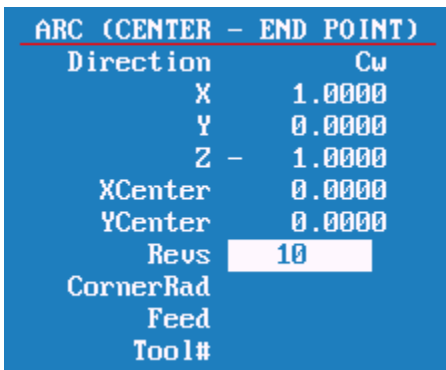
This is the default for arc's and will always come up looking this way.
There two other chooses , end point and center or center and angle.

Note the center icon is high lighted.

```
ARC (CENTER - END POINT)
    Direction            Cw
            X        1.0000
            Y        0.0000
            Z  -     1.0000
     XCenter        0.0000
     YCenter        0.0000
        Revs      10
  CornerRad
       Feed
      Tool#
```
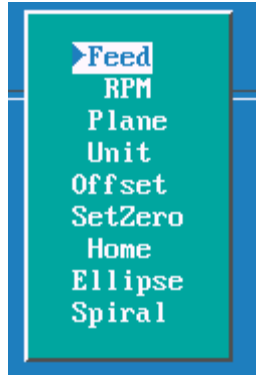
With this arc the machine is capable
of milling a thread.
It needs an X , Y and Z end point
X , Y center point and Rev's.
With Z starting at zero the inputs shown
on left would cut a 10 TPI thread.

5.

ANILAM

Pressing **F4**   **More**   this key will bring up following box menu.

```
►Feed
 RPM
 Plane
 Unit
 Offset
 SetZero
 Home
 Ellipse
 Spiral
```

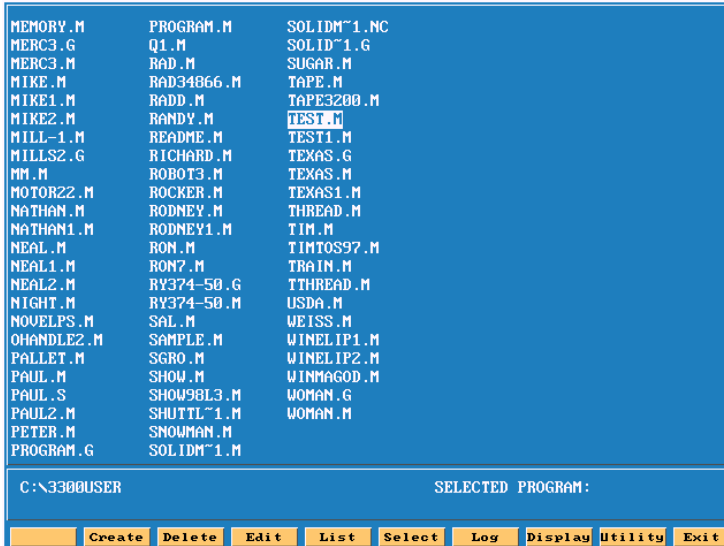|	
Feed	Enter a feedrate on line by itself.
RPM	Put spindle speed on of it's own.
Plane	Change planes XY,XZ or YZ.
Unit	Inch or MM.
Offset	Enter fixture offset , this is an <u>absolute</u> shift relative to Machine Zero.
SetZero	<u>Incremental</u> Zero shift.
Home	Returns machine to home.
Ellipse	Programs an ellipse with comp inside or outside.
Spiral	Spiral gives the ability to program tapered threads.

Press **F9**   **Prev**   to return to previous screen.

6.

X,Y zero

5."

1."

1."

3."

1.5"

5.25"
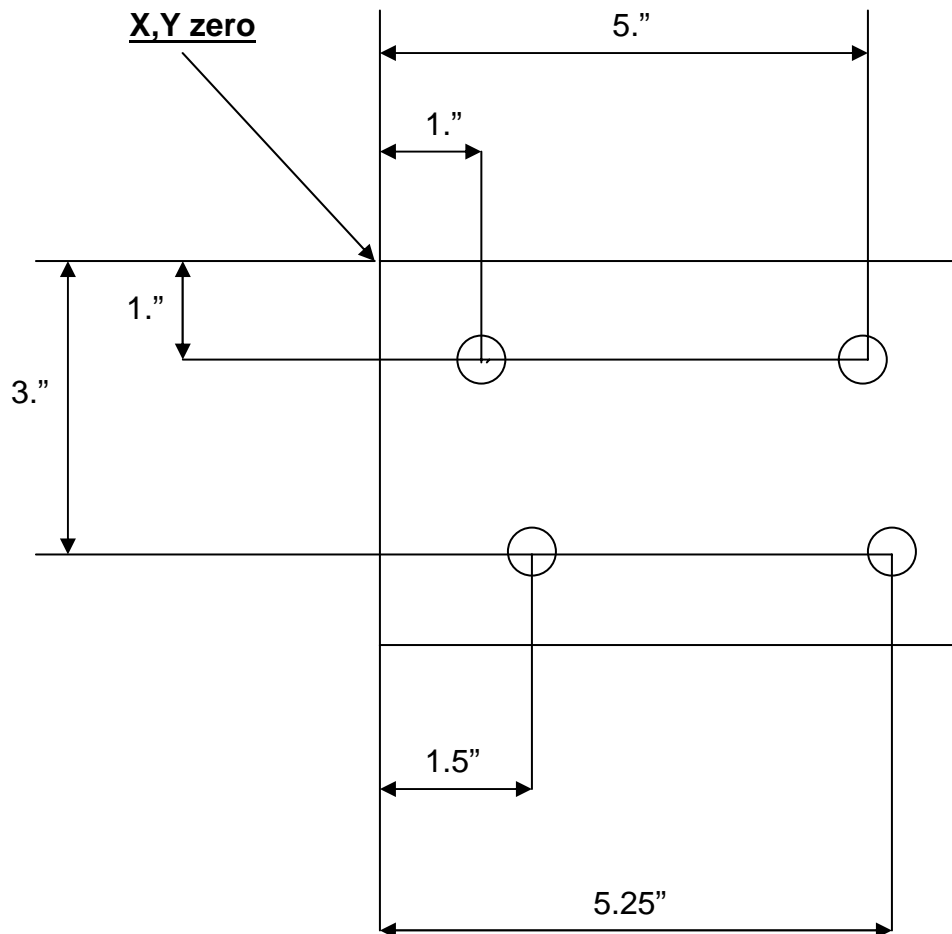
We will now write a program to center drill and drill this part, we will use a subroutine in this program because we are going to use the same dimensions twice.
A subroutine is a mini program <u>outside</u> of the Main program that will be Called into the Main program.
Program lines are in **bold** print.

7.

The first thing to do is **Create** a program.

```
MEMORY.M       PROGRAM.M      SOLIDM~1.NC
MERC3.G        Q1.M           SOLID~1.G
MERC3.M        RAD.M          SUGAR.M
MIKE.M         RAD34866.M     TAPE.M
MIKE1.M        RADD.M         TAPE3200.M
MIKE2.M        RANDY.M        TEST.M
MILL-1.M       README.M       TEST1.M
MILLS2.G       RICHARD.M      TEXAS.G
MM.M           ROBOT3.M       TEXAS.M
MOTOR22.M      ROCKER.M       TEXAS1.M
NATHAN.M       RODNEY.M       THREAD.M
NATHAN1.M      RODNEY1.M      TIM.M
NEAL.M         RON.M          TIMTOS97.M
NEAL1.M        RON7.M         TRAIN.M
NEAL2.M        RY374-50.G     TTHREAD.M
NIGHT.M        RY374-50.M     USDA.M
NOVELPS.M      SAL.M          WEISS.M
OHANDLE2.M     SAMPLE.M       WINELIP1.M
PALLET.M       SGRO.M         WINELIP2.M
PAUL.M         SHOW.M         WINMAGOD.M
PAUL.S         SHOW98L3.M     WOMAN.G
PAUL2.M        SHUTTL~1.M     WOMAN.M
PETER.M        SNOWMAN.M
PROGRAM.G      SOLIDM~1.M

C:\3300USER                       SELECTED PROGRAM:

    Create  Delete   Edit   List   Select   Log   Display Utility  Exit
```

Press **F2**   **Create**

```
C:\3300USER
NEW PROGRAM: TEST_

        ASCII    Ins
```

Type in a program name 8 letters, numbers or acombinationof both.
To type in letter use the ASCII (**F2** key) will bring up the chart.

When the name is typed in press E N T E R

The program name will be entered into program page and an **.M** will
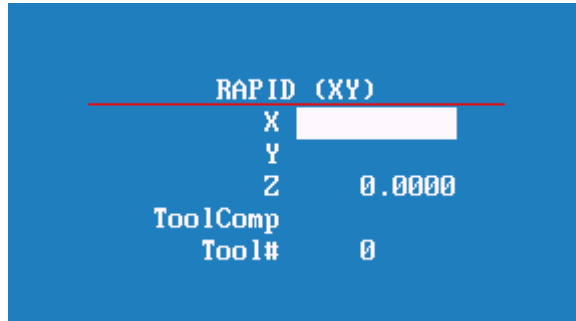be added to it.

Press [ABS INC] Press [ENTER] Set the control in absolute

1. **Dim Abs**   Use [+ −] to toggle between Abs/Inc.

[1 Rapid]

```
          RAPID (XY)
             X  [        ]
             Y
             Z     0.0000
         ToolComp
            Tool#    0
```

When !/Rapid key is pressed a box as show on left will appear.

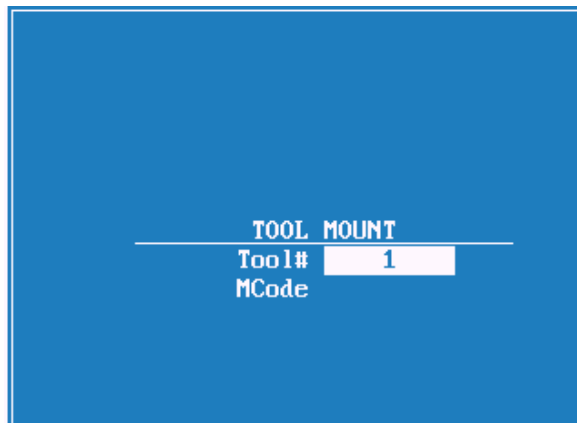2. **Rapid   Z0.000   Tool# 0**   **F10**   [Save]   Rapid to Z home.

Press [1 Rapid]

3. **Rapid   X -4.000 Y 2.0   000**   **F10**   [Save]   Rapid to tool change position.

[5 Tool]

```
         TOOL MOUNT
         Tool#  [    1    ]
         MCode
```

5/Tool key is press screen will look as shown on left. The Mcode would only be necessary if a tool changer is installed.

F10   [Save]

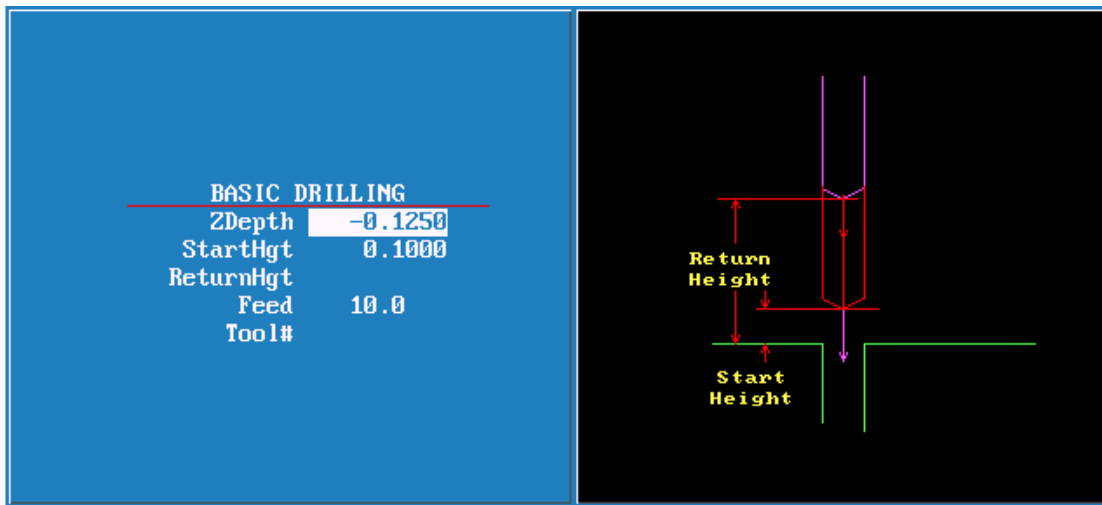4. **Tool# 1**   Call tool #1

9.

F3 **Drill** **E N T E R**

```
▶Basic
 Pecking
 Boring
 Chip Break        0
 Tapping
 Drilling Off      a
 Pattern
 Bolt Hole
                   0
```
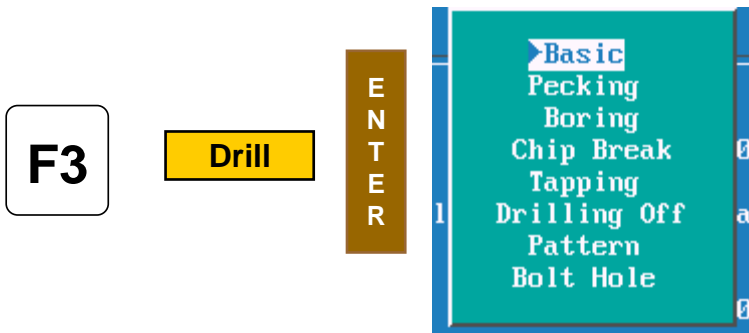
F3 Drill is selected this box will appear. These are your chose of how you are going to drill the hole , the first time we will use Basic for the center drill , the second time we will use pecking for the drill.

```
BASIC DRILLING
  ZDepth    -0.1250
 StartHgt    0.1000
 ReturnHgt
     Feed    10.0
    Tool#
```

Return Height

Start Height

E N T E R

When **E N T E R** is pressed the screen will appear as above.

Zdepth = depth of hole.
StartHgh = Distance above Surface you are drilling into.
ReturnHgt = Distance above to retract to before moving to next hole.
Feed = Feedrate
Tool# = Tool# may be entered here.

F10 **Save**

5. **BasicDrill Zdepth -0.125 StartHgt 0.1000 Feed 10.0**

10.

| Teach | Draw | Drill | Pocket | Mill | Tool | Calc | Sub | Misc | Exit |

Press **F8** | **Sub** | **Sub** Function key will change as below.

| Sub | EndSub | Call | EndMain | Loop | RMS | Dwell | MCode | Prev | |

F1 | **Sub** | Enter subroutine number. A subroutine is a program entered after main program and call into main program using a **Call.**

F2 | **EndSub** | Entered at the end of a subroutine.

F3 | **Call** | Bring a subroutine into main program.

F4 | **EndMain** | Ends main program.

F5 | **Loop** | Repeats operation desired number of times.

F6 | **RMS** | Allow subroutines to be **Rotated , Mirrored** or **Scaled.**

F7 | **Dwell** | Enters **Dwell** into program , this is also available on a hot key.

F8 | **MCode** | Enters an **Mcode** into program , also available as a hot key.

F9 | **Prev** | Return to previous screen.

F10 | | Not used.

11.

Press **F3**   **Call**

```
              SUB CALL
              Call      1
```
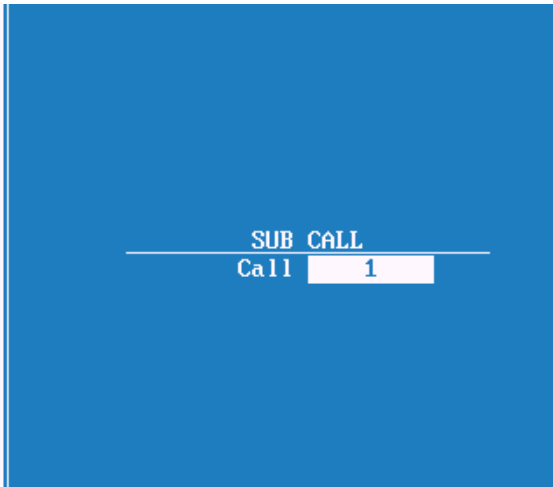
Press Key number 1   **E N T E R**

6. Call 1

We have now finished the with the first tool.

Press **1 Rapid**

F10   **Save**

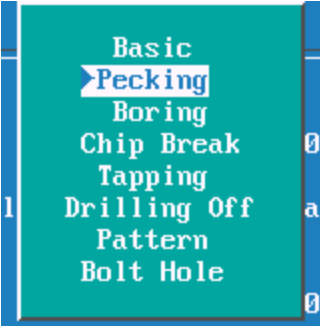### 7. Rapid Z0.000 Tool# 0   Rapid to Z home.

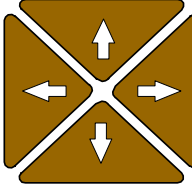Press **1 Rapid**

F10   **Save**

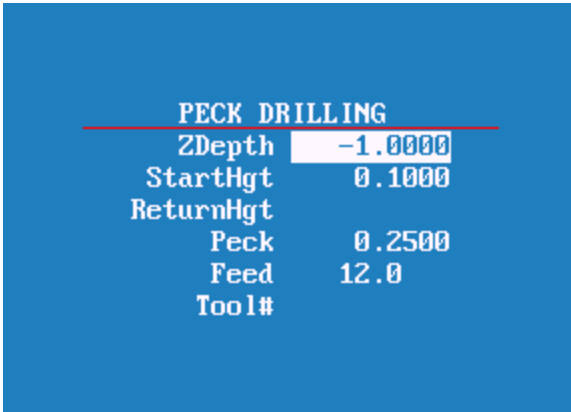### 8. Rapid x -4.000 Y 2.0000   Rapid tool change position.

Press $\boxed{\dfrac{5}{\text{Tool}}}$ $\boxed{\textbf{F10}}$ **Save**

9. **Tool# 2**   Activate tool #2

Press $\boxed{\textbf{F3}}$ **Drill**

```
        Basic
       ►Pecking
        Boring
        Chip Break      0
        Tapping
      l Drilling Off   a
        Pattern
        Bolt Hole
                        0
```

ENTER

Press down arrow key to high light **Pecking ,** press

```
        PECK DRILLING
        ZDepth    -1.0000
       StartHgt    0.1000
      ReturnHgt
           Peck    0.2500
           Feed    12.0
          Tool#
```

Input values $\boxed{\textbf{F10}}$ **Save**

10.   **PeckDrill Zdepth -1.0000 StarHgt 0.1000 Peck 0.2500 Feed 12.0**

Press **F3**   Call

Press Key number 1   ENTER

11. Call 1

We have now finished the with the second tool.

Press   1
Rapid

12. Rapid Z0.000 Tool# 0   Rapid to Z home.

F10   Save

Press   1
Rapid

13. Rapid x -4.000 Y 2.0000   Rapid tool change position.

F10   Save

Press **F4**   EndMain

14. EndMain

Press **F1** [Sub] press #1 key Press **F10** [Save]

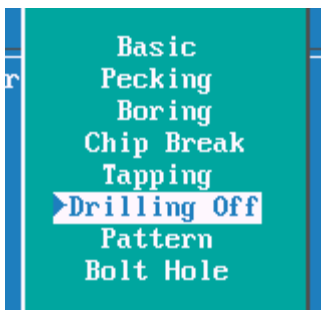**Sub 1**

Press [1 / Rapid]

Rapid   X 1.0000   Y -1.0000 press F10 save
Rapid   X 5.0000   press F10 save
Rapid   X 5.2500   Y -3.0000 press F10 save
Rapid   X 1.5000   press F10 save

Drilling must now be turn off as soon as last hole is drilled

Press **F9** [Prev]

Press **F3** [Drill]

```
    Basic
    Pecking
    Boring
    Chip Break
    Tapping
  ▶Drilling Off
    Pattern
    Bolt Hole
```

High light Drilling Off press **E N T E R**

Press **F2** [EndSub]

EndSub

Program for this part is now complete.

This is above program will look in control.

1. Dim Abs
2. Rapid   Z 0.0000 Tool# 0   (See note 2 below)
3. Rapid   X -4.0000 Y 2.0000
4. Tool# 1
5. BasicDrill ZDepth -0.1250 StartHgt 0.1000 Feed 10.0
6. Call 1
7. Rapid   Z 0.0000 Tool# 0 (See note 3 below)
8. Rapid   X -4.0000 Y 2.0000
9. Tool# 2
10. PeckDrill ZDepth -1.0000 StartHgt 0.1000 Peck 0.2500 Feed 12.0
11. Call 1
12. Rapid   Z 0.0000 Tool# 0   (See note 3 below)
13. Rapid   X -4.0000 Y 2.0000
14. EndMain
15. Sub 1
16. Rapid   X 1.0000 Y -1.0000
17. Rapid   X 5.0000
18. Rapid   X 5.2500 Y -3.0000
19. Rapid   X 1.5000
20. Drilling Off
21. EndSub

Note
If running parts on a machine with **Homing** a fixture offset may be added to program
at Line #2 or #3 to get to part zero.
With Bed Mill Z0 Tool#0 not required , just move **Z** axis up plus to a convenient height
 to change Tools.(I.e. Z5.0000.)

Now that the part is programmed , we need to verify that it is correct.
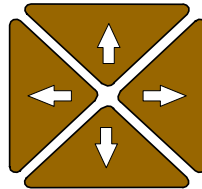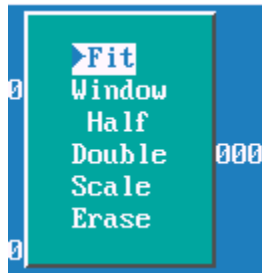To do this we use **Draw** .

Press **F9** | Prev | to return to main edit screen.

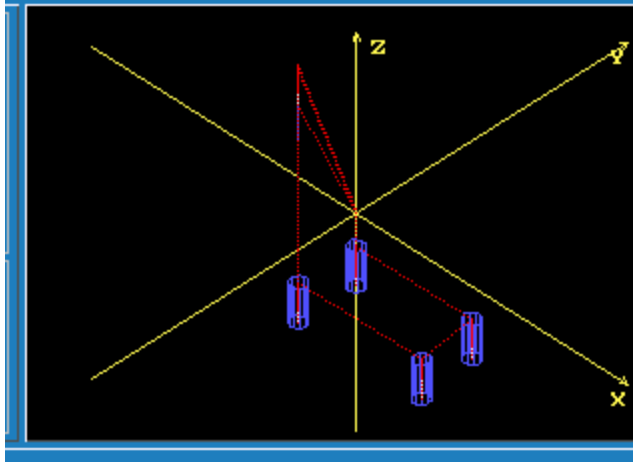Press **F2** | Draw | to enter **Draw Mode.**

| Draw | Run | VIEW | DISPLAY | | | | Parms | Exit |

F1 | | Not used

F2 | Draw | Exit draw mode.

F3 | Run | Runs program in simulation draw.

F4 | View | There are four views available XY,XZ,YZ and Iso.

F5 | Display | Calculates the window size to show complete part.

F6 | | Not used

F7 | | Not used

F8 | | Not used

F9 | Parms | Draw parameters.

F10 | Exit | Exit to main edit screen.

Press **F5** Display box will pop-up as shown below high light will be on fit.

```
    ┌──────────────┐
    │ ►Fit         │
  0 │  Window      │
    │  Half        │
    │  Double  000 │
    │  Scale       │
    │  Erase       │
  0 │              │
    └──────────────┘
```
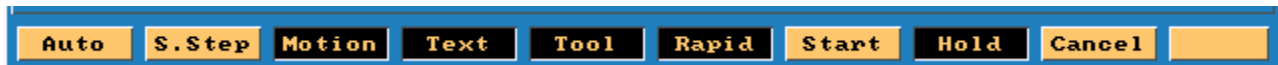
Use arrow keys to move high light up and down.

Press E N T E R

Red lines are **Rapid** moves.
White Lines are **Feed** moves.
Yellow are axis lines.
Blue represent the tools sizes ,In this case tool #1 is center drill with .0000 Diameter and tool #2 is .5000 drill.

When **F3** [**Run**] or **F5** [**Display**] is press soft key will change as below.

| Auto | S.Step | Motion | Text | Tool | Rapid | Start | Hold | Cancel | |

F1 — **Auto** — Will run program all the way through.

F2 — **S.Step** — Ever time **Start** is pressed runs one program line.

F3 — **Motion** — Runs a move ever time **Start** is pressed.

F4 — **Text** — If high lighted will scroll text.

F5 — **Tool** — Will show tools if high lighted.

F6 — **Rapid** — Displays **Rapid** moves when high lighted.

F7 — **Start** — Starts draw moves one line in S Step or Motion.

F8 — **Hold** — Stop draw until Start is pressed.

F9 — **Cancel** — Cancel current drawing.

F10 — — Not used

Program has been written and checked on simulation graphics , it is now time to set **Part zero** and **Tool offsets.**

Press **F10** | Exit | twice to return to **Manual** page .

Using **Jog** keys find edge of part or center of hole where you want **X0** and **Y0** are to be
located .

If machine does not have homing press **X0** E N T E R   display will change to zero

on X axis , use same procedure on **Y** axis .

If machine has **Home** Jog to **X0 , Y0** as above then press **F9** | Tool |

Softkeys will change as shown below.

OFFSETS		ClrLine	Find	PgUp	PgDn		Calib Z		Exit

Press **F1** | OFFSET |

No.	Diameter	Length	Spindle RPM	Spindle Direction	Coolant
1	0.1000				NONE
2	0.5000		Fixture Offsets		NONE
3	0.2200				NONE
4	0.2180		X	Y	Z	NONE
5	0.2500	1.	0.0000	0.0000	0.0000	NONE
6	0.4000	2.	-20.0000	-6.0000	0.0000	NONE
7	0.1540	3.	-9.0000	-6.0000	0.0000	NONE
8	1.0000	4.	-6.0000	-6.0000	0.0000	NONE
9	1.0000	5.	-3.0000	-6.0000	0.0000	NONE
10	0.7500	6.	0.0000	-6.0000	0.0000	NONE
11	1.0000	7.	0.0000	0.0000	0.0000	NONE
12	0.2180	8.	0.0000	0.0000	0.0000	NONE
13	0.0000	9.	0.0000	0.0000	0.0000	NONE
1	0.1000				NONE

				CalibX	CalibY	CalibZ			Exit

If machine is at **X0** press **F5** | CalibX |

Do the same with **Y** **F6** | CalibY | Press **F10** | Exit |

20.

You are now back at the tool Page .
The thing to do is set tool length Offsets.
Check to see that Tool #0 is active .
Put tool #1 into spindle jog down to top of part .
Check to see that high light is on Tool #1 .

Now press **F5** **Calib Z**

Move spindle up , put in tool #2 and repeat above process , until all tools offsets

are set and press . **F10** **Exit**

Control is now back at **Manual** and ready to cut part.

Press **F6** **AUTO**

Put **Tool #1** in spindle press

Machine will stop on **Tool Change** press

Machine will rapid **X** and **Y** position of first hole and then **Z** rapid to .1000
above part . Next it will then feed to give depth and rapid back out of the hole
and rapid to next hole and repeat process until all holes are drilled.

It will now on **Tool Change** and repeat process for tools #2 and #3.

3000M DXF Converter
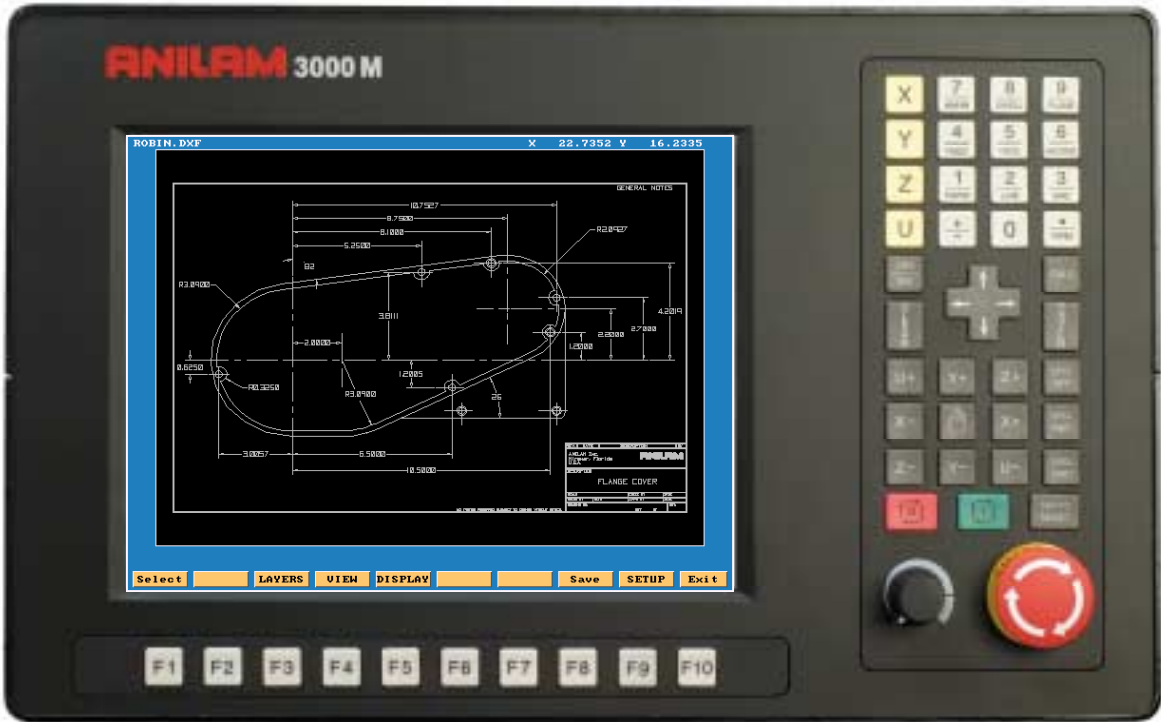
DXF file can be converted into 3000 machine programs using the Offline software.

The DXF files are stored in the Program Page.

When going to Program Page only .M file will be displayed,

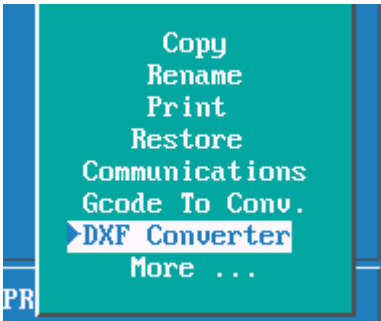press **F8** **Display** twice, this will display all files.

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

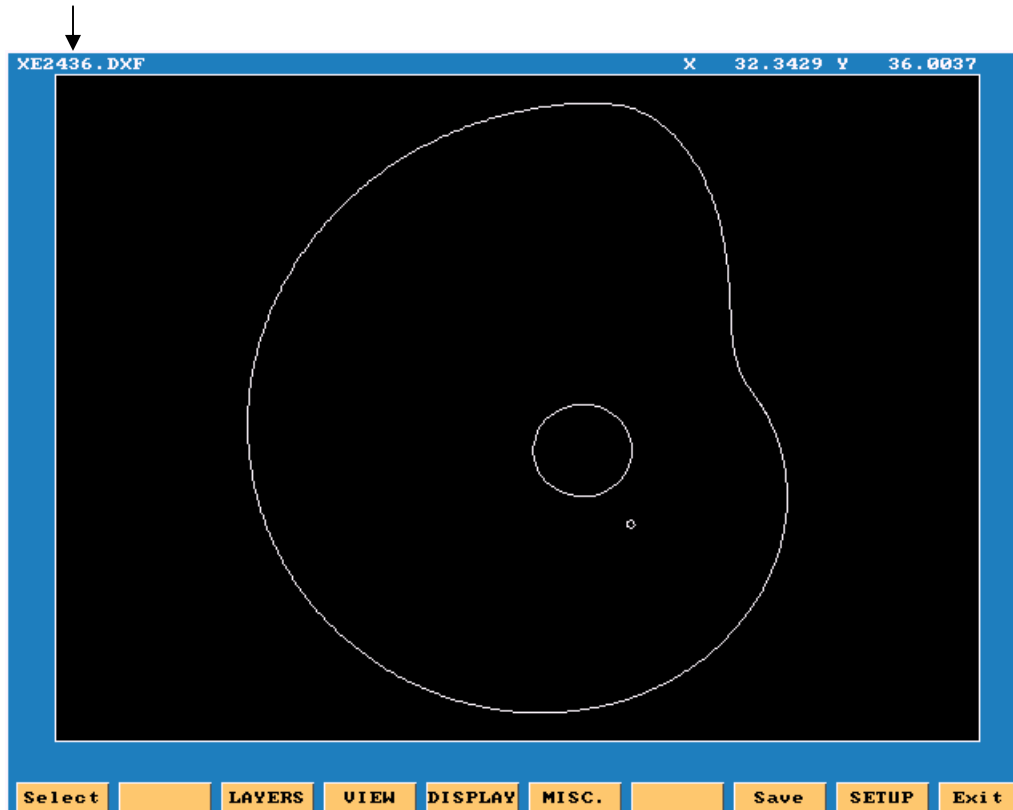press **F7** **Log** select **A:**

High light required DXF program press **F9** **Utility** Copy to **C:**

High light DXF program

```
    TOOL2.DXT
F   UPARM2.DXF
    XE2436-1.DXF
    XE2436-1.FXD
    XE2436-1.M
```

Press **F9** **Utility** high light DXF Converter

```
    Copy
    Rename
    Print
    Restore
    Communications
    Gcode To Conv.
  ►DXF Converter
    More ...
PR
```

Press **E N T E R**

1.

File name

ANILAM

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.
Misc. See below.
Save: Saves program with .G once converted.
Setup: Allows set of inputs and outputs.

Information on a particcular entity.
Set shifted zero in the **set-up.**
Toggle end points of entities **on** or **off.**
When turn **ON** will ask question link or new shape.

Zero on most drawings is usually not at a point that is covenient for programing, so there is a way itcan be changed.
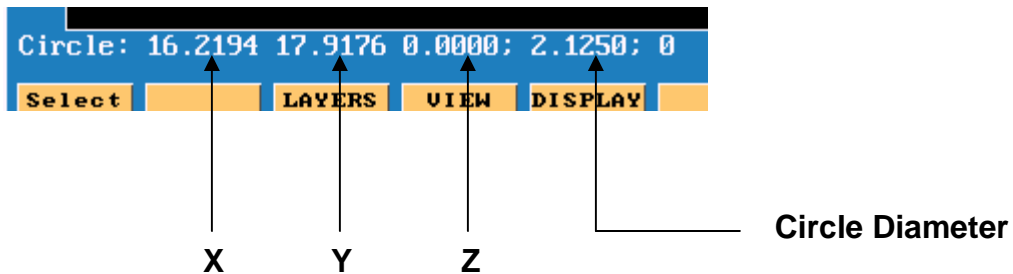
2.

In the case of current drawing, the center of the hole in center of part is the best
point X0 Y0. There are two methods find new zero's from an entity on drawing.
1. To do this Press the Ctlr key and hold it down put mouse point on to circle and
press left mouse key, it will change to yellow, release keys.
2. Press [MISC] key select [▶Entity Info] press **ENTER** select entity using mouse

it will turn yellow
At the bottom of screen X, Y, and Z cordinate will appear and also circle diameter.

```
Circle: 16.2194 17.9176 0.0000; 2.1250; 0
Select          LAYERS    VIEW   DISPLAY
```

Circle Diameter

X   **Y**   **Z**

Now press **ALT** key and letter **T** at the same time first method or [MISC]
[▶Set Shift] press **ENTER.**
This will in put these coordinates in to the **SETUP** and change **X0 Y0** to the center of hole.
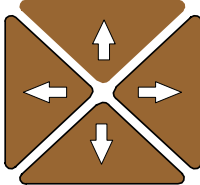
Press [F9]   [SETUP]   [▶Output... Display.. ▶]   Press [E N T E R]

```
▶Output program name....    XE2436-1
 Shift X...............      16.21939
 Shift Y...............      17.91758
 Output dimensions......      Absolute
 Create Mode...........         Smart
 Starting sub number....            1
 Output warnings........          Yes
 Re-calc Intersections..          Yes
 Output resolution......            5
 Create main in new.....          Yes
 Convert values........          None
 Convert polyline.......          None
```

Program name
Imported X shift
Imported Y shift
Absolule/increamental
Smart/Overwrite
Starting subroutine number
Warns if elements done meet
Re-calculates intesects
Decimal out put
Create a main program
Inch / Metric / None
Convert polyline to arc/prompt/none

3.
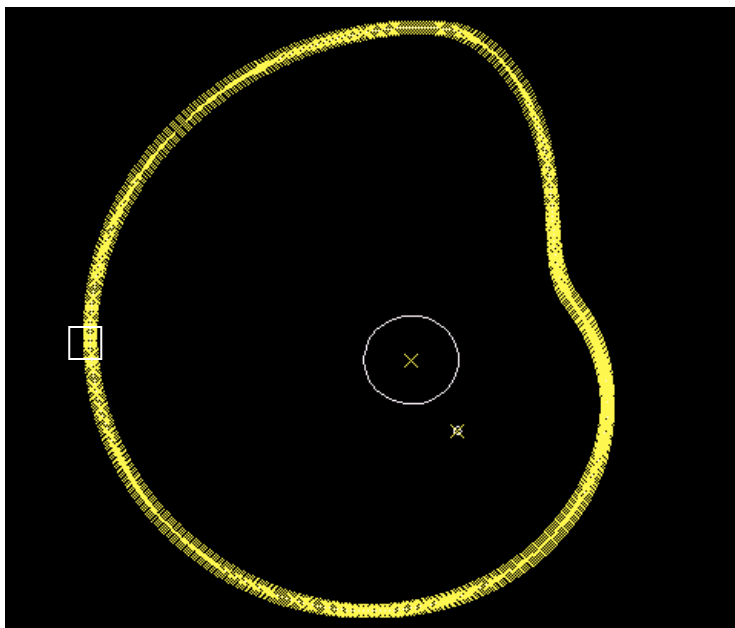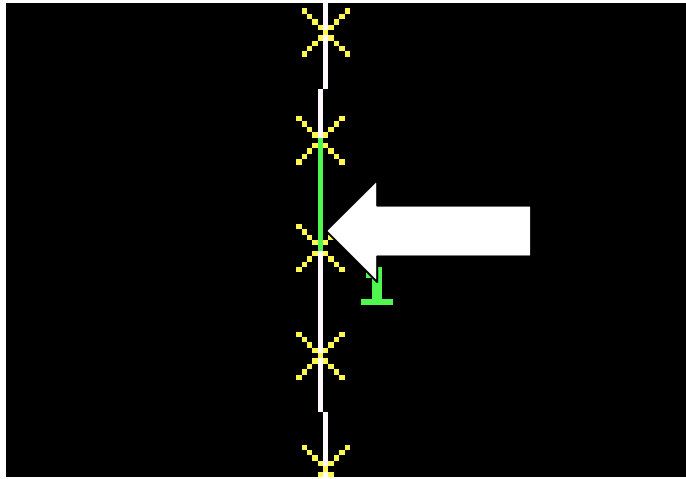
Press F10 Exit

Press **ALT**key and letter **F**key at the same time, this will mark the end of each element.

Press F5 DISPLAY high light **Window**
```
  Fit
►Window
  Redraw
  Half
  Double
```
press ENTER

A box will appear on screen move over using

press F6 Compress position box as shown below press ENTER

4.

Press **F1**   Select   point mouse arrow to lower end of a line as shown

 and pess 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**   DISPLAY   press E N T E R   Part will appear at full size on screen

5.

Press **F8** Save

Succesfully created XE2436-1.M.

| Select | | LAYERS | VIEW | DISPLAY | | | Save | SETUP | Exit |

Press **F10** Exit

Exit (Y/N)?

| Yes | No | | | | | | | Cancel | |

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

XE2436-1.FXD
XE2436-1.M
XE2436.DXF
XE2436.FXD

High light **.M** file press **F4** Edit

6.

High light .**M** file press F4 Edit

```
  1 Call 1
  2 EndMain
  3
  4 Sub 1
  5 Dim Abs
  6 Rapid      X -14.44357 Y 1.05070
  7 Line       X -14.44446 Y 1.17741
  8 Line       X -14.44421 Y 1.30422
  9 Line       X -14.44282 Y 1.43111
```

Start of program

```
719 Line       X -14.40565 Y 0.16721
720 Line       X -14.41445 Y 0.29298
721 Line       X -14.42213 Y 0.41891
722 Line       X -14.42868 Y 0.54500
723 Line       X -14.43410 Y 0.67123
724 Line       X -14.43839 Y 0.79760
725 Line       X -14.44155 Y 0.92409
726 Line       X -14.44357 Y 1.05070
727 EndSub
```
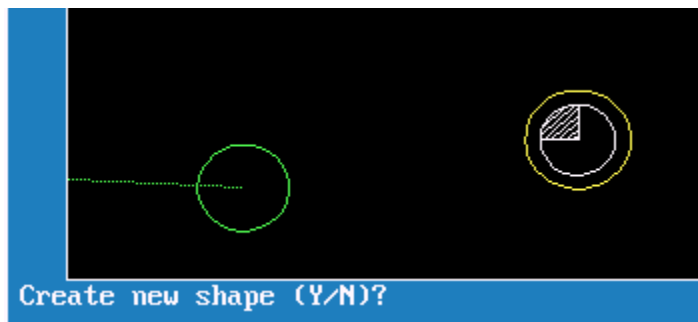
End of program

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

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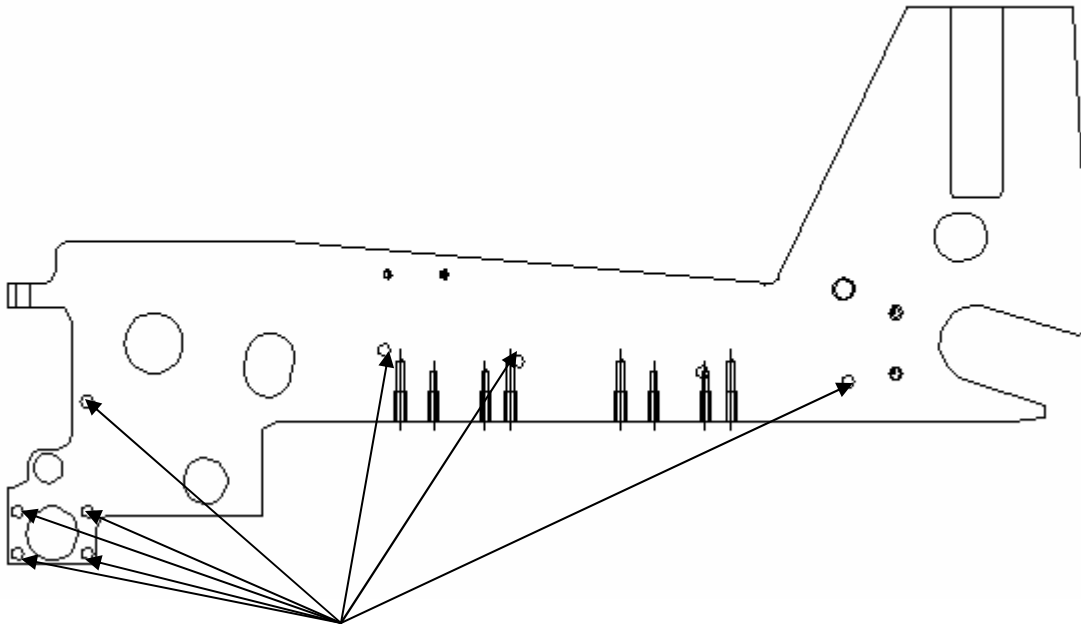
Press **F1** Select and pick all holes that are the same size , in this case 8. When going to second set of holes press right key on mouse.

Create new shape (Y/N)?

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.

8.

The print below shows the four shapes of the different sizes holes.
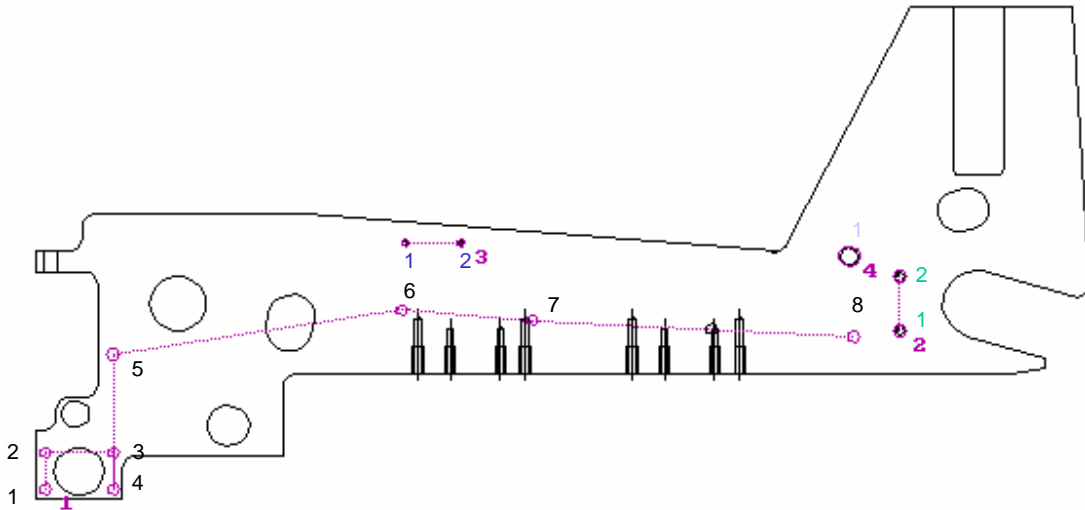
Press **F8** [Save]

Press **F10** [Exit]

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

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

Below show program as it comes from DXF converter.
Some work will have to be done in main program to
center drill and drill and these holes.

```
 1 Call 1
 2 Call 2
 3 Call 3                                          ← Subroutine calls
 4 Call 4
 5 EndMain
 6
 7 Sub 1                                           ← Subroutine for positions of the eight
 8 Dim Abs                                            holes numbered in black.
 9 Rapid      X 1.34375 Y 0.15625
10 Rapid      X 0.15625 Y 0.15625
11 Rapid      X 0.15625 Y 0.84375
12 Rapid      X 1.34375 Y 0.84375
13 Rapid      X 1.34327 Y 2.66153
14 Rapid      X 6.38400 Y 3.50000
15 Rapid      X 8.67900 Y 3.31400
16 Rapid      X 14.30400 Y 3.00000
17 EndSub
18
19 Sub 2                                           ← Subroutine for positions of the two
20 Dim Abs                                            holes numbered in green.
21 Rapid      X 15.10019 Y 3.12200
22 Rapid      X 15.10019 Y 4.12200
23 EndSub
24
25 Sub 3                                           ← Subroutine for positions of the two
26 Dim Abs                                            holes numbered in blue.
27 Rapid      X 6.45603 Y 4.75138
28 Rapid      X 7.42403 Y 4.75138
29 EndSub
30
31 Sub 4                                           ← Subroutine for positions of the one
32 Dim Abs                                            hole numbered in light blue.
33 Rapid      X 14.21138 Y 4.51231
34 EndSub
35
36 <End Of Program>
```

10.

The following program has been edited to put in drilling cycles
and tool changes

Dims Abs	Dimensions Absolute.
Rapid Z0.0000 Tool#0	Rapid to Z0 and Tool 0.
Rapid X-2.0000 Y-2.0000	Rapid Tool change position.
Tool#1	Call tool #1.
BasicDrill Zdepth -0.2000 StartHgt 0.1000 Feed 12.0	Drill cycle for Center Drill.
Call 1	Calling subroutines to center drill
Call 2	all four sets of holes.
Call 3	
Call 4	
DrillOff	Turn off drilling.
Rapid Z0.0000 Tool#0	
Rapid X-2.0000 Y-2.0000	
Tool#2	Call tool #2.
PeckDrill Zdepth -.5000 StartHgt 0.1000 Peck 0.125 Feed 12.0	Peck drilling Cycle for eight holes.
Call 1	Call eight hole pattern.
DrillOff	Turn off drilling.
Rapid Z0.0000 Tool#0	
Rapid X-2.0000 Y-2.0000	
Tool#3	
PeckDrill Zdepth -.5000 StartHgt 0.!000 Peck 0.1250 Feed 11.0	
Call 2	
DrillOff	
Rapid Z0.0000 Tool#0	
Rapid X-2.0000 Y-2.0000	
Tool#4	
PeckDrill Zdepth -0.5000 StartHgt 0.1000 Peck 0.1250 Feed 10.0	
Call 3	
DrillOff	
Rapid Z0.0000 Tool#0	
Rapid X-2.0000 Y-2.0000	
Tool#5	
PeckDrill Zdepth -0.5000 StartHgt 0.1000 Peck 0.1250 Feed 9.0	
Call 5	
DrillOff	
Rapid Z0.0000 Tool#0	
Rapid X-2.0000 Y-2.0000	
EndMain	

In this example of a full drawing and how to turn off unnecessary information ,such as dimensions etc.

The first thing to do is turn off some of the layers so as to leave only the part.
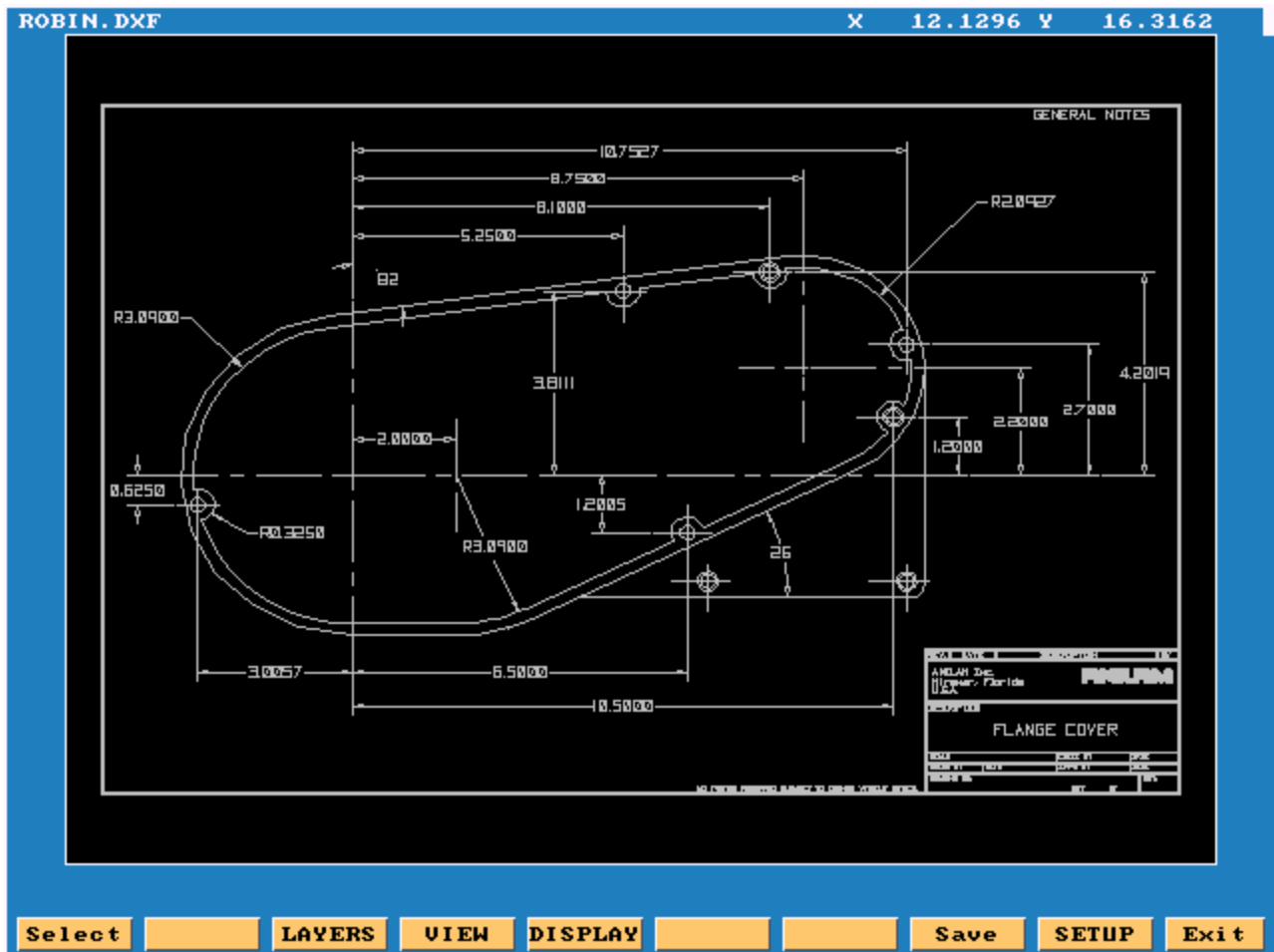
Press F3   Layers

```
All Layers on
Invert Layers
▶Toggle Layers
```

High light **Toggle Layers** press

E
N
T
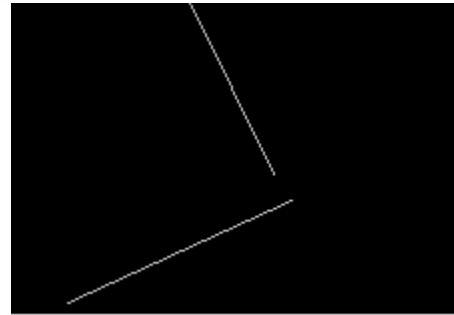E
R

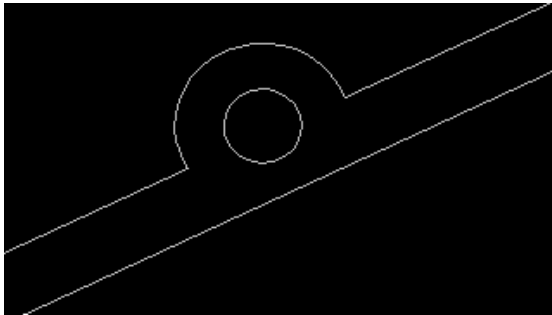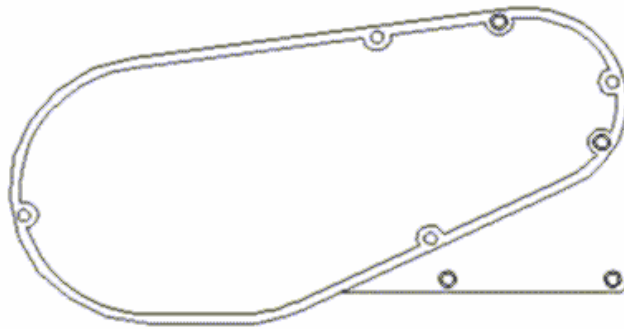Put high light on layers not required and press

E
N
T
E
R

to turn OFF.

```
            Layers

  1.▶0.......... Off
  2. AM_TR...... Off
  3. AM_BM...... Off
  4. AM_BL...... Off
  5. AM_VIEWS... Off
  6. AM_PARDIM.. Off
  7. AM_REFDIM.. Off
  8. AM_VIS..... Off
  9. AM_HID..... Off
 10. AM_SUPPR... Off
 11. CONT.......  On
 12. CEN........ Off
 13. HID........ Off
```

In the drawing shown the only layer left on is #11

13.

Only the part profiles and holes are left.

The first picture shows the area with the problem and shows the problem that the line do not meet. The software will take care of this and join the lines together.
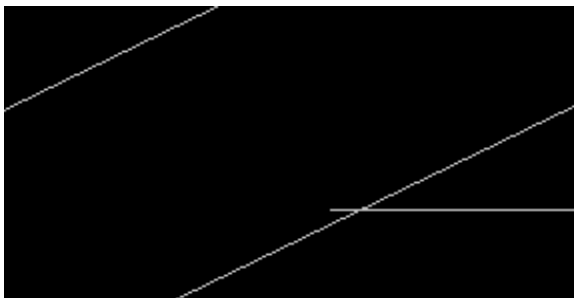
This is another type of problem you see from CAD drawings.

14.