M 03 Spd Fw

M 05

SERIES 1100 3-AXIS MILL

PROGRAMMING / OPERATING MANUAL

P/N 70000289
Revision (3) date:
March 30, 1994

CONTENTS

1.0 GENERAL

1.1 INTRODUCTION TO THE ANILAM SERIES 1100M CONTROL

Congratulations! You have selected the most powerful and easy-to-use 3-axis CNC control. The Series 1100 utilizes a "machinist's language" programming format. It is a high accuracy, high performance control designed specifically for 3 (and 2) axis milling applications.

The control circuit incorporates the use of a PC (personal computer) system. It utilizes high-speed 32-bit microprocessors, with practical features such as "canned cycles", tool diameter compensation, use of subprograms, and many other outstanding features.

The 1100 is a closed loop CNC control, and can utilize a variety of drive systems and feedback devices.

This manual will describe the programming method and operational features of the 1100 control. For maintenance and specific details concerning your particular machine/control combination, consult the manual provided by the machine tool builder.

1.2 NOTES TO USERS

(1) If you are upgrading from two (2) to three (3) axis, please consider the following characteristics:
   a) Your new 3-AXIS system is designed for continuous operation. **NO AUTOMATIC DWELL**(s) will occur between RAPID and FEED moves, as was the case with 2-axis. Nor will dwells occur with "drill cycles", since the Z-axis is now under power. Your 3-AXIS machine can now be fully automatic, with the exception of Tool Changes.
   b) More "canned cycles" are present in the 3-AXIS system, such as peck drilling, boring, chip-breaker drilling, mold rotation, and more. These functions are explained in appropriate sections of this manual.
   c) Your machine may still be used in DRO mode, as it was in 2-axis.

(2) The notes in explaining procedures or features may sometimes refer to a subject covered later in this manual. If this happens, skip the particular note and refer to it later.

(3) Important notes may be included at the end of each section of each chapter. These notes highlight important points covered in the section.

(4) Words that appear **BOLD**, UNDERLINED or in CAPITAL letters remind the reader of important points. Particular attention should be given to the print contained.

(5) The contents of this manual are subject to change without notice.

(6) If metric examples are included with any combination drawing/ program example, millimeter values are shown in parenthesis, and are usually on the right side of the page.

1.3 TURNING THE SYSTEM ON AND OFF

1.3.1 Turning On Power

Power On/Off is provided by a two (2) position switch on the rear of the computer cabinet. Emergency-Stop button may be either IN or OUT during power on.

Make sure all doors on the equipment are closed prior to turning on the power.

"0" : Power Off position
"1" : Power On position

When the switch is rotated to the ON position, the computer is completely reset.

Something is displayed within several seconds of turning the power on. If the CRT screen remains blank, turn power off and repeat the procedure.

The first screen to appear will be an information screen, with Serial Number, Software Version Number, and Distributor information. As instructed by the screen, please press soft-key F10 to continue. The soft-keys are located directly under the CRT screen, and are labeled F1 through F10.

The next step is to make a choice between the three (3) selections available on the new screen:

1. CNC Control
2. Diagnostics
3. Set-Up Utility

#1 will enter into the CNC mode, where parts are programmed and cut on the machine tool. To select it, press F5 (Select) or press ENTER.

#2 is a special Diagnostic mode, to be used only by a qualified factory-trained technician.

#3 are menus where the CNC is configured to the machine tool. Parameter settings such as rapid rates for the axes, backlash compensation, leadscrew pitch-error compensation, and more are set up through this menu. Consult your Anilam distributor or Anilam Electronics if you wish to change any settings.

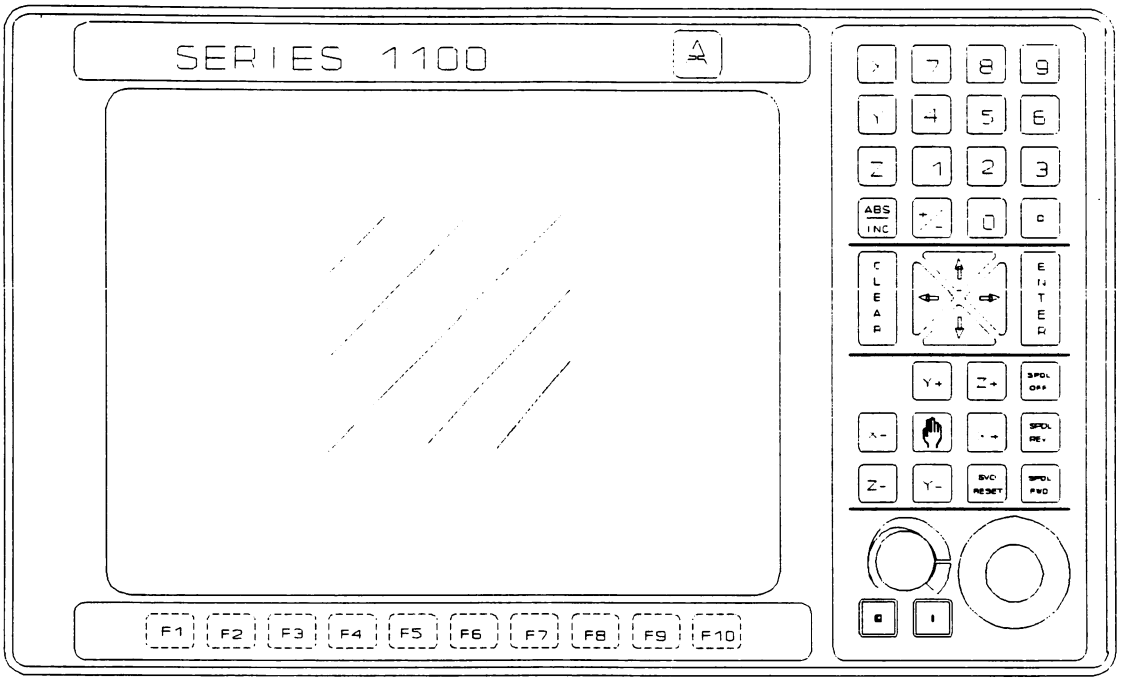
A special "Installation Manual" explains both #2 and #3 selections above, and is not necessary to operate the 1100 control. This manual will cover only the CNC control.

1.3.2 Turning Off Power

Power On/Off switch is operated as described above in section 1.3.

Before turning power off, make sure the control is in MANUAL mode, by pushing the Emergency-Stop button in. Turn the power On/Off switch to the off position as described above.

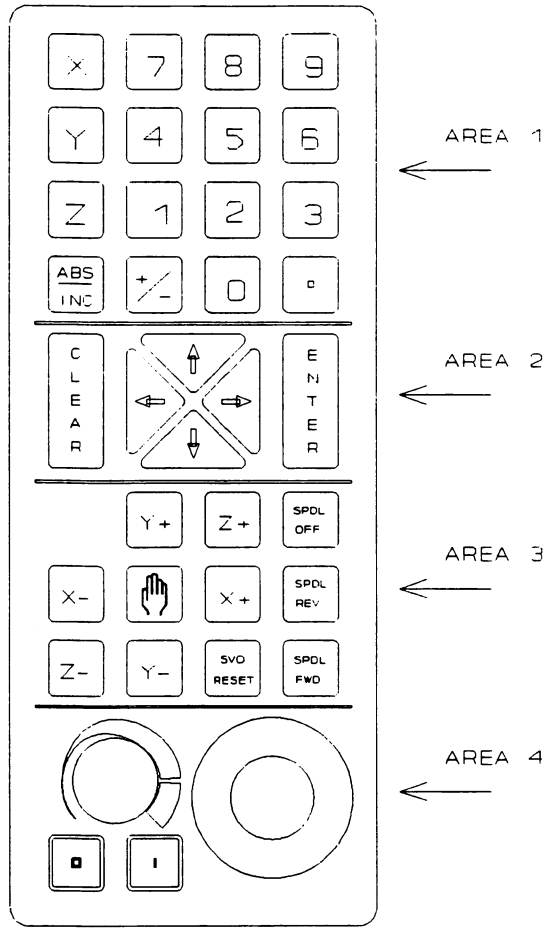
2.0 DESCRIPTION



SERIES 1100 CONSOLE

2.1 KEYBOARD

The keyboard of the Series 1100 console is shown below. Please refer to the notes for key descriptions.



AREA 1:

X,Y,Z: selects axis for subsequent commands.

ABS/INC: switches dimensions from absolute to incremental, or from incremental to absolute.

0-9, +/-, . : For numeric input. Whole numbers do not require a decimal point. Data is assumed + unless +/- key is pressed. While in EDIT mode, +/- key also toggles (switches) Tool Comp to Left, Right or Off, and ABS to INC. Number keys 1 through 9 may also be used as "Hot-Keys", see sections 4.5 and 6.17 for more details on "Hot-Key" functions.

AREA 2:

CLEAR: Clears messages from the message line in Draw, Manual, Single Step and Auto Modes. Also clears values while programming.

ARROWS: Used to move the "cursor" while programming. When there may be more than one selection to choose from, the ARROW keys allow the cursor (or "pointer") to be moved to the desired selection. When the selection is reached, **ENTER** must be pressed to select that function.

ENTER: Used to select functions that the **ARROW** key has been moved to. Also used to "send" data to the CNC, such as program names.

AREA 3:

HAND SYMBOL: Used to toggle (switch) the JOG movement from RAPID to FEED to 100 to 10 to 1 (then back to RAPID). This is how the machine table/quill will move when the X+, X-, Y+, Y-, Z+, Z- keys are pressed (see below). The RAPID and FEED rate is determined by the machine builder, and is usually between 100 and 200 inches per minute for RAPID, and 40 I.P.M. for FEED. These rates are used only for jogging with the X+, X-, Y+, Y-, Z+, Z- keys described below.

RAPID and FEED are "continuous jog" functions, in other words, the axes will continue to move as long as the key is HELD IN. When the key is released, the axis will stop. Travel limit switches are provided to prevent machine damage.

JOG 100, JOG 10, and JOG 1 are "incremental jog" functions, in other words, only one (1) increment will be moved for EACH press of the X+, X-, Y+, Y-, Z+ or Z- keys. JOG 100 = resolution x one-hundred (usually .01"); JOG 10 = resolution x ten (usually .001"); JOG 1 = resol. x one (usually .0001").

NOTE: The current selection of the HAND SYMBOL is shown on the CRT screen in the upper-right area after the word **JOG:**

X+, X-, Y+, Y-, Z+, Z- : These keys are used to "JOG" (move) the machine table or quill (Z) while the CNC is in MANUAL Mode, with the SERVOS ON. This can help the operator move the machine to find the edge of the workpiece, indicate a bore, do a Tool Length Offset, even make a cut.

Depending on what is selected by the HAND SYMBOL (see above), the machine will move in either RAPID, FEED, JOG 100, JOG 10, or JOG 1. When RAPID or FEED is selected, the rate may be overridden by the rotary switch in Area 4 of the keyboard.

NOTE: The machine can be used manually, (without servo motor power) by pressing the E-Stop (emergency stop) described below. The CNC acts like a DRO (digital read-out) when the E-Stop is IN. The operator can use the handles of the machine to move the axes.

SPDL OFF, SPDL FWD, SPDL REV: These keys are used to turn the machine's spindle OFF, ON FORWARD, or ON REVERSE. These keys may or may not function, depending on whether this function was installed with the system. If not, they allow the system to be upgraded in the future, if so desired.

SVO RESET: This key, when pressed, will energize the SERVO MOTORS that drive the axes of the machine. The motors must be on in order to move the machine under power in the Manual mode, and to enter into the Auto and Single-Step modes. The E-STOP (emergency stop) must be reset (push in, and twist CW out) in order for this key to function.

AREA 4:

ROTARY SWITCH (FEEDRATE OVERRIDE): This rotary switch allows the RAPID and FEED movements of the machine to be "slowed down", or "sped up". The switch has increments of 10 % (percent), and the range is from 0% to 120%. As an example, if 12.0 I.P.M. was programmed, the moves could be overridden by 1.2 I.P.M. per increment of the rotary switch.

This switch applies to both RAPID and FEED moves.

E-STOP (emergency stop): The LARGE RED button is the E-STOP. When pressed, it will de-energize the SERVO MOTORS. While in the IN position, the machine can be moved manually by the handles provided at the front and side of the machine table. The CNC can be used as a DRO (digital read-out) while the E-Stop is IN.

To energize the Servo Motors, Press the E-Stop IN, rotate it CW (clockwise) and it will pop out. Then press the SVO RESET key described above.

START KEY: This key is used to start running a program in the Single Step and Auto modes, and to execute a manually programmed move while in Manual mode. It is GREEN in color.

HOLD KEY: This key is used to TEMPORARILY hold the program (or Manual move) after it is started by the Start key. If hold is pressed, the program will resume if the Start key is pressed again.

In order to "ABORT" or cancel a program or manually programmed move, the MANUAL (F4) key must be pressed.

The HOLD key is RED in color.

2.2 SOFT-KEYS

Located directly below the CRT (screen) are the "Soft-Keys", otherwise knows as Function Keys. They are labeled F1 through F10, and their functions change slightly, depending on which mode the CNC is in.

The function of each soft-key is found on the CRT-screen directly above each soft-key.

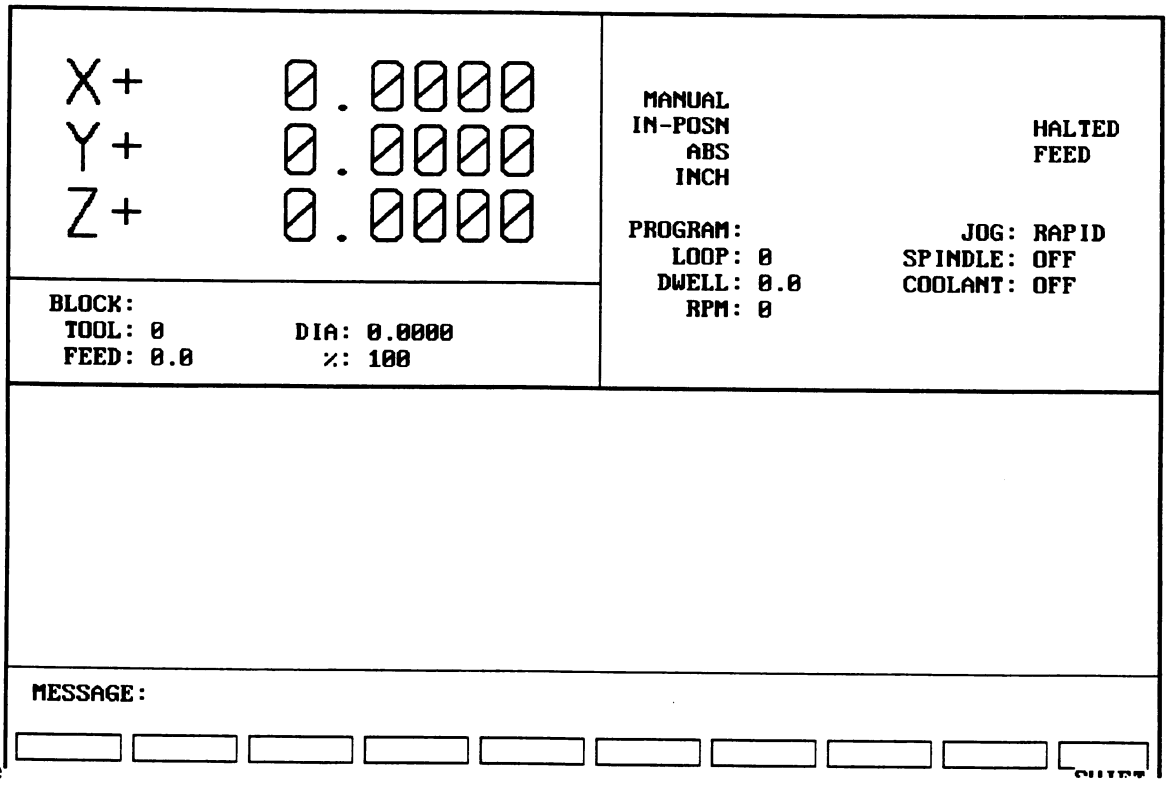
The Soft-keys will be explained in detail in appropriate places throughout this manual.

2.3 CRT - SCREEN

The main screen (CRT) of the Series 1100 console is shown in the figure below. This screen will be seen in any of the "operating" modes;

This is the screen while in Manual, Auto or Single Step modes.

See below for complete details of the CRT-screen.



The screen is divided into two main parts, TOP and BOTTOM, by a heavy white line.

TOP:

The top half of the screen contains the STATUS areas:

- Top left: X, Y, and Z position display in large numerals.
  Active block (line or event) of the program.
  Active Tool number, and its diameter (as set in Tool Page)
  Current feedrate and override percentage (rotary switch position).

- Top Right: Mode (Manual, Auto or Single Step (S.Step).
  In-Position Indicator.
  Absolute or Incremental active.
  Inch or Millimeter active.
  Halted or Running.
  Feed, Rapid or Arc active (for program moves).
  Program name, active.
  Loop counter (# of loops remaining).
  Dwell counter (seconds remaining).
  RPM display, if installed.
  JOG indicator: RAPID, FEED, 100, 10, or 1.

SPINDLE On/Off indicator (if applicable).
COOLANT On/Off indicator (if applicable).

BOTTOM:

The bottom half of the screen contains the program TEXT as it is running, and the MESSAGE / SOFT-KEY area:

- TEXT: The text area will remain blank while in the MANUAL mode, unless a "manual move" is programmed by pressing X, Y or Z key (see section 4.5).
  While in AUTO or S.STEP modes, the TEXT area will show the program blocks as they are being run. A double arrow POINTER shows the active / current program block.
  The operator can view the blocks as they run, and can also see the program for block searching.

- MESSAGE: The message area just above the Soft-Key labels is where messages to the operator will appear, such as "Tool Change", and other requests by the CNC to the operator, or any programming ERROR messages.

- SOFT-KEY: The Soft-Key area at the far bottom of the screen contains the soft-key "labels". They are related to the soft-key (F1, F2, F3, F4, etc.) directly below each one, and when a soft-key is PRESSED, that function or mode will be initiated.
  Soft-keys will be explained in detail for each Mode of the CNC, beginning with the Manual Mode in section 4.0 below.

3.0 PROGRAMMING CONCEPTS and DEFINITIONS

3.1 WHAT IS PROGRAMMING ?

Programming is the method in which instructions are given to the CNC control. Each "Program" must contain "blocks" of information to direct the movement of the machine table. Each block is separate from the others, so that the program can be "stepped" along, one block at a time, to check the moves before cutting material.

Programs can also be called "files". The programs are stored in memory in an area knows as a "directory". The directory contains the programs that the operator creates and edits. Programs can be created, deleted, copied or renamed, as explained in section 5.0 below.

TERMS:

- Program: A file where the individual instructions to the CNC are stored, so they may be used to mill and/or drill a workpiece. Each workpiece should have its own, separate program. Programs are identified by NAMES. These program names appear in the directory.

- Block: A single line of a program. Programs are made up of blocks.

3.2 AXIS DESCRIPTIONS

Each of the machine travels are known as an AXIS, and each axis has a direction, either POSITIVE or NEGATIVE:

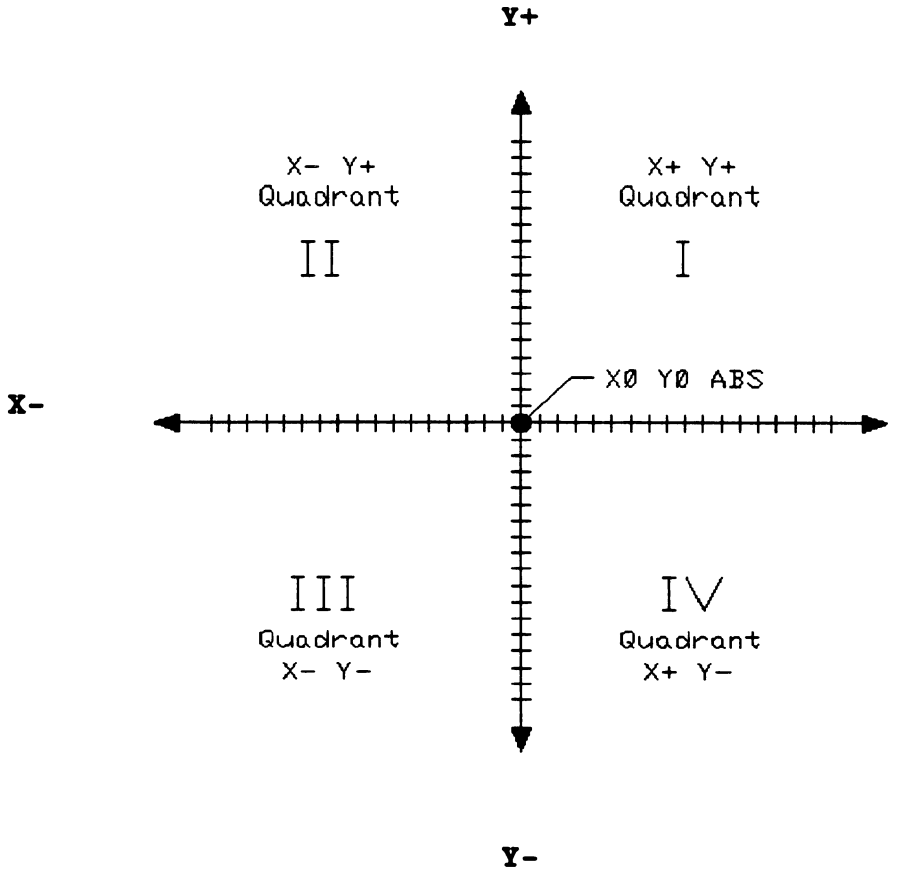
NOTE: You should always visualize the TOOL as moving, rather than the TABLE as moving. This will give a better understanding of CNC programming, no matter which model CNC you use.

X AXIS: Table travel left and right.
POSITIVE motion is defined as the table moving to the left, therefore measurement to the right is POSITIVE on the workpiece. See below figure.

Y AXIS: Table travel in and out.
POSITIVE motion is defined as the table moving toward the operator, therefore measurement toward the machine is POSITIVE on the workpiece. See below figure.

Z AXIS: Quill travel up and down.
POSITIVE motion is defined as the quill moving up, therefore measurement into the workpiece is NEGATIVE. See below figure.

Each measurement or dimension can be referenced from a fixed point, known as ABSOLUTE ZERO (part zero). Absolute zero is usually set at a corner of the workpiece, or at the center of a round part or existing bore. ABSOLUTE ZERO is where the dimensions in the program originate from.



Each dimension or X-Y point is known as a coordinate. If a position 2 inches to the right, and 2 inches down (toward you) were programmed, the X coordinate would be X2.0 and the Y coordinate would be Y-2.0.

X2.0 Y-2.0 could be a hole location, an arc center, or the end of a line. Regardless, it is known as a coordinate.

The above illustrates a "top view" of the coordinate system. Z-axis is up (+) and down (-), and is perpendicular to the flat X-Y plane.

3.3 ABSOLUTE and INCREMENTAL REFERENCE

Absolute and Incremental are methods of dimensioning:

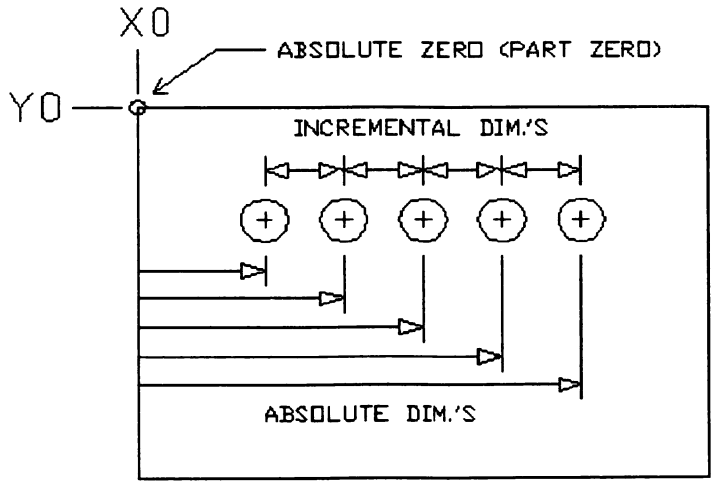
ABSOLUTE: Absolute dimensions are referenced from a known point on the workpiece, and can be any point the operator chooses, such as the upper-left corner, the center of a round

part, or an existing bore.

The key to understanding ABSOLUTE dimensions is that they are ALWAYS in reference to the ABSOLUTE ZERO (part zero, see below figure). This part zero is set by the operator in the manual mode, and can be switched to a new position during the program (in case 2 or more identical parts need machining at separate locations on the machine table).

INCREMENTAL: Incremental dimensions are referenced from one point to another (see below figure). This can be a convenient way to input dimensions into the program, depending on the blueprint.

The key to understanding INCREMENTAL dimensions is that the next point to be programmed (whether it is a hole location, end of line or arc, or center of arc) is ALWAYS in reference to the CURRENT TOOL POSITION.



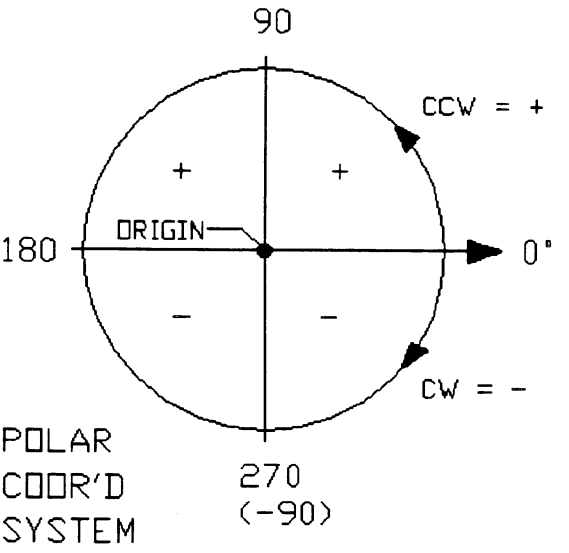
NOTE: ABSOLUTE or INCREMENTAL can be selected by pressing the ABS/INC key on the keyboard, any time while in MANUAL or EDIT modes. While in MANUAL mode, the ABS/INC status indicator will reflect the current setting. While in EDIT mode if ABS/INC key pressed, then the +/- key will toggle (switch) from ABS to INC or from INC to ABS.

3.4 POLAR COORDINATES

A coordinate point may be given in another format besides the X-Y (Cartesian) coordinate system. Another method is the POLAR coordinate system, where a point is described by RADIUS and ANGLE from an ORIGIN. See below figure.

If a position 39 degrees at a radius of 1.5 is programmed, the CNC will automatically calculate the X-Y equivalent, and move to that position.

The ORIGIN of the coordinate is where it is referenced from, usually ABSOLUTE ZERO while in ABSOLUTE mode, or the CURRENT TOOL POSITION while in INCREMENTAL mode.



3.5 TOOL DIAMETER / LENGTH OFFSETS: The TOOL PAGE (F9)

Tool Diameter and Length Offsets tell the CNC each tool's DIAMETER, and "Z" location in reference to the top of the workpiece. The CNC uses this information as the program is run, to ensure the part will come out to the correct dimensions, both XY dimensions, and Z-depths.

DIAMETER OFFSETS:

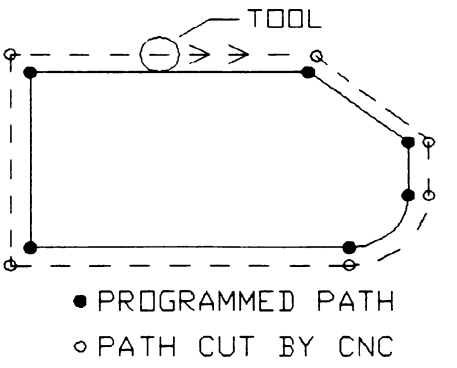
Diameter offsets allow the actual EDGE of the part to be programmed, rather than the operator allowing for the tool's radius in the dimensions he or she programs. This is known as TOOL COMP (tool diameter compensation). It will be explained in detail below.

The figure below shows how TOOL COMP affects the path the CNC takes.

The operator programs the MACHINED EDGE of the part, and TOOL COMP automatically offsets the tool the correct distance.

This way, if a repeat job comes up, a different size tool may be used, and the operator simply inputs the actual diameter into the TOOL PAGE.

Also, if the part comes out too big or too small, the operator simple changes the DIAMETER value in the TOOL PAGE. The next time the part is run, the dimensions will reflect the change made.

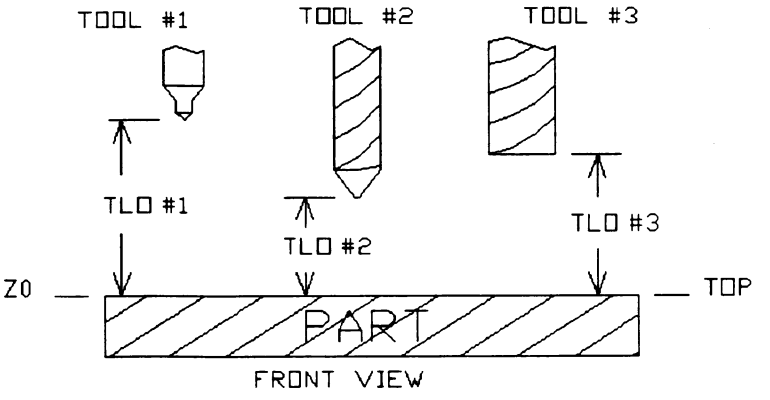


TOOL LENGTH OFFSETS:

Tool Length Offsets (Z offsets) are an important feature to understand, in any CNC. The Tool Length Offsets allow the operator to make the top of the workpiece Z0 for every tool used in the program, regardless of the tools' difference in length, or how many tools there are. Otherwise, the operator would have to "Set Zero" on the Z axis for every tool, every time the program was run, which is inconvenient.

The figure to the right shows what a TOOL LENGTH OFFSET (TLO) is. It is defined as the distance from Z-HOME to the TOP of the part (or any other convenient surface).

Since it is unlikely that each tool will have the same TLO, the TOOL PAGE can be used to make up for the differences.



DOING A TLO (Tool Length Offset):

1. First, Z-HOME must be set: To set "Z-HOME", the operator must ensure that there is no tool offset active, by checking the CRT-screen in the upper-left status area. (If a tool # is active, it can be de-activated through the MDI mode. See section 4.6, 4.7 below for MDI mode.)
   Then, the quill should be raised to the top of travel, and

while in MANUAL MODE, press Z0 ENTER. When ENTER is pressed, the Z display will zero-out.

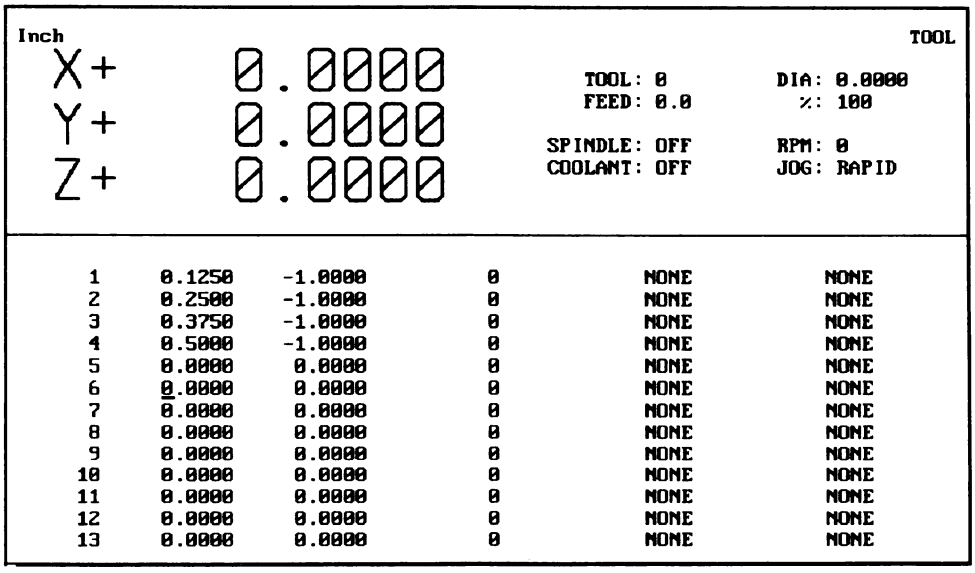
2. Place the tool in its holder and tighten it, then put the holder into the spindle. Carefully lower the tool onto the top of the workpiece, enter the TOOL PAGE (F9), place the cursor on the desired tool number's line (1,2,3, etc.) and press CALIB-Z (F9). The offset is automatically entered into the Tool Page, and the cursor advances down to the next tool number. Repeat step 2 for all remaining tools.

3. The DIAMETER offsets should be entered by measuring the tool with a micrometer, and entering the value in the DIAMETER column for each tool. Use the ARROW keys to move the CURSOR to the desired tool diameter column in the tool page.

NOTE: TLO's are not mandatory, but are useful anytime there is 2 or more tools needed for a program. If only 1 tool is used, the operator can lower the tool onto the surface of the part, and press Z0 ENTER. This, in effect, sets Z-HOME at the top of the workpiece.

3.5.1 Tool Page: F9

Each offset for each tool is stored in a special TOOL PAGE. The Tool Page is accessed by pressing F9 while in the MANUAL mode. See below figure.



1	0.1250	-1.0000	0	NONE	NONE
2	0.2500	-1.0000	0	NONE	NONE
3	0.3750	-1.0000	0	NONE	NONE
4	0.5000	-1.0000	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

When the tool page is first entered into, the cursor's vertical position is indicated by a bright-thick horizontal line, and the cursor's horizontal position is indicated by a flashing underscore. The upper area of the screen displays the machine table's position relative to ABSOLUTE ZERO. The next area down displays several tool offsets.

The CNC arrow keys are active, and should be used to move the cursor up, down, left, and right.

The Tool Page screen is divided into 3 basic areas:

TOP: The top area shows position in reference to ABSOLUTE ZERO, and whether entries are in INCH or MM. Top right denotes the JOG status (jog explained in section 4.3 below), as the table and quill may be JOGGED while in the tool page.

CENTER: The middle area of the screen is where the numeric data is stored for each tool. It is in this area that the cursor appears and data can be entered.

This section is divided into columns:

Col. 1: This column contains the TOOL NUMBERS, which are fixed and cannot be changed.

Col. 2: This column is where tool DIAMETERS are entered. When a diameter is keyed in, the ENTER key need not be pressed. The value will remain.

Col. 3: This column is where tool LENGTH OFFSETS (Z OFFSETS) are entered. Values may be keyed in, or the CALIB Z key may be used to automatically enter the Z axis value in reference to Z home position (usually top of quill travel).

Col. 4: This column is where the SPINDLE RPM may be entered. It will only affect the machine if that function is installed. Do not use it if it is not installed.

Col. 5: This column is where SPINDLE DIRECTION is entered. It will only affect the machine if that function is installed. Toggle with +/- key (None, Off, Fwd, Rev).

Col. 6: TYPE: This column is where COOLANT is entered. It will only affect the machine if that function is installed. Toggle with +/- key (None, Off, On).

BOTTOM: The single-wide strip just above the soft-key labels shows whichever tool the cursor is on, with that tool's offsets.
The bottom strip shows each soft-key's function. See below for description of soft-keys while in TOOL PAGE.

3.5.2 Tool Page Soft-key Functions

Explanation of Tool Page Soft-Keys:

F3 ClrLine : clears the entire single line (across).

F5 Page Up : moves cursor one page up.

F6 Page Down : moves cursor one page down.

F7 Find : allows "search" of a tool number (col. 1). After entering the tool number, ENTER must be pressed.

F9 Calib-Z : inputs Z offset: the dimension from Z-HOME (top of quill travel) to the TOP of the workpiece is input into the active (highlighted) Length Offset space when F9 is pressed. This is also the number shown in the Z display at the TOP of the CRT-screen.

F10 Exit : EXITs the Tool Page and SAVES any changes made.

3.5.3 Using TOOL COMP in the program:

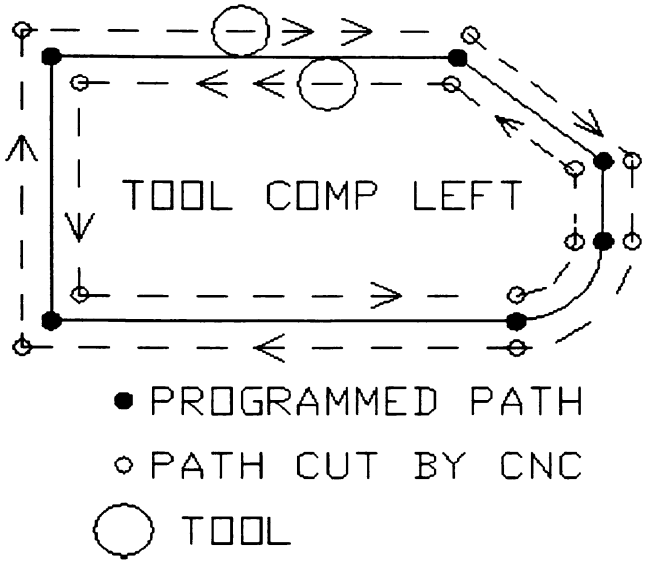
While editing the program (see section 6.0 for EDIT), the operator may choose to use TOOL COMP. It is recommended anytime a CONTOUR needs to be milled. It is NOT necessary for any built-in POCKET cycles, because the CNC automatically compensates for the tool size in pocket cycles.

In order to use TOOL COMP, the operator must tell the CNC whether he/she wants TOOL COMP LEFT, or TOOL COMP RIGHT. The third and final choice is TOOL COMP OFF. These settings may be set while programming a RAPID or LINE move. TOOL COMP LEFT or RIGHT should be put with the RAPID or LINE move that positions the tool near the workpiece (see section 6.16 for program examples), so the first cut can then be made with a LINE or ARC.

To decide whether TOOL COMP LEFT or TOOL COMP RIGHT is needed, the operator must look at the machined part edge (the path he/she will program) and decide which side of that path the tool must be on. As a GENERAL RULE, whether milling an inside or outside contour:
CLIMB MILLING = TOOL COMP LEFT, and,
CONVENTIONAL MILLING= TOOL COMP RIGHT. See below figure:
(CENTERLINE of cutter moves to programmed points = NO TOOL COMP.)

See below figures:

TOOL COMP LEFT : CLIMB MILLING.



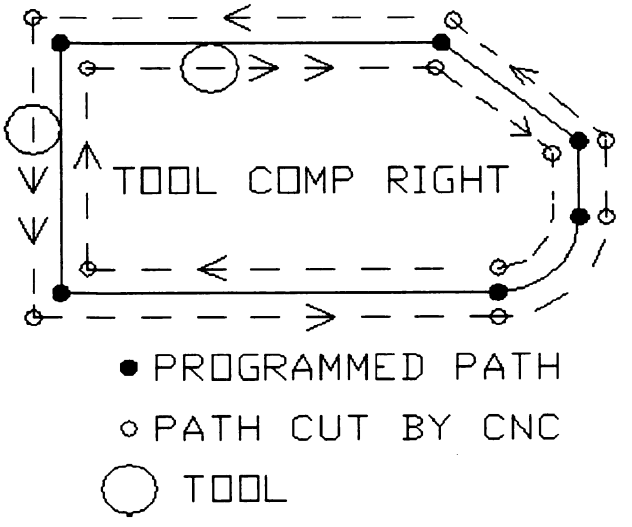
In either case, COMP-LEFT or COMP-RIGHT, the path must be viewed in the direction of tool travel.

TOOL COMP RIGHT : CONVENTIONAL MILLING.

Notice that no matter which way you look at it, the tool ALWAYS remains either LEFT of path (figure above),

or,

RIGHT of path (figure to right).



Generally, CLIMB milling is preferred in CNC work, because of better chip relief, and tool deflection away from the cut. Unlike a manual machine, the SERVO motors will hold the other axis in

position while the cut is being made.

For more information on activating and deactivating TOOL COMP, see section 6.7 below.

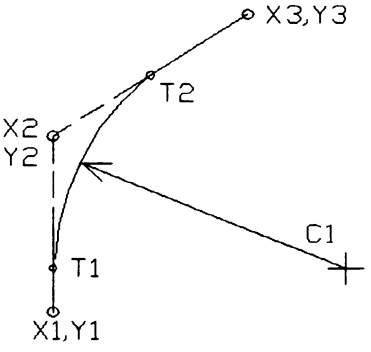
3.6 CORNER ROUNDING

"Corner Rounding" is a convenient function that allows the operator to put a radius on any corner of a workpiece, without programming an ARC. This saves time and effort. The operator must know the X-Y point where the two LINES intersect. Intersections between 2 LINES, LINE and ARC, ARC and LINE, even intersections between 2 ARCS can be "corner rounded".

On the "input screen" for LINE and ARC moves, an item appears called **CornerRad**. The operator simply fills in the desired RADIUS while programming the LINE or ARC, and the radius will blend the end of the move being input, with the next move. See below examples.

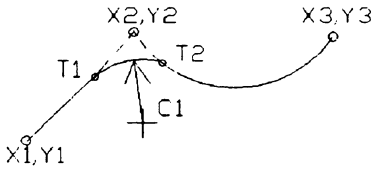
LINE to LINE:

The figure to the right shows what happens when a "CornerRad" is used on the move from X1,Y1 to X2,Y2. The CNC figures out tangency points T1,T2 and center point C1 automatically. The LINE move to X3,Y3 (programmed after the CornerRad move) will happen as expected, through tangent points T1 and T2.



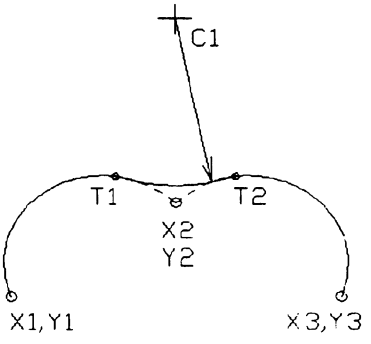
LINE to ARC:

The figure to the right shows what happens when a "CornerRad" is used on the move from X1,Y1 to X2,Y2. The CNC figures out tangency points T1,T2 and center point C1 automatically. The ARC move to X3,Y3 (programmed after the CornerRad move) will happen as expected, through tangent points T1 and T2.



ARC to ARC:

The figure to the right shows what happens when a "CornerRad" is used on the move from X1,Y1 to X2,Y2. The CNC figures out tangency points T1,T2 and center point C1 automatically. The ARC move to X3,Y3 (programmed after the CornerRad move) will happen as expected, through tangent points T1 and T2.



4.0 MANUAL MODE

4.1 WHAT IS MANUAL MODE ?

Manual Mode is the first of the 3 "operating" modes including Manual, S.Step, and Auto. Manual mode is necessary to be able to JOG the machine (move the axes under power), set ABSOLUTE ZERO (part-zero), do TOOL LENGTH OFFSETS (TLO's), and to get to the PROGRAM screen and the EDIT mode. Manual is also the mode that the CNC can be used as a DRO (digital read-out).

Manual mode is the mode the CNC will be in when the power is turned on. The screen is shown and explained in section 2.3 above.

In this section of the programming manual, the SOFT-KEYS of Manual Mode will be explained, as well as how to set ABSOLUTE ZERO, use the helpful M.D.I. mode, and use FEEDRTAE OVERRIDE. Please refer to the screen details in section 2.3 above while reviewing this section (4).

4.2 DRO - MOVING the MACHINE WITHOUT SERVO POWER

The term "DRO" refers to Digital Read-Out. With the E-STOP in the IN position, the handles provided can be used to move the X, Y, and Z axes.

DRO mode is a very useful way to position the machine in order to use an edge finder, sweep indicate a bore, set ABSOLUTE ZERO (Part-zero), even make cuts without creating a program, or without using the power of the servo motors.

NOTE: A Manual/Auto toggle switch is provided at the rear of the servo cabinet. When the switch is in AUTO, the machine can be used under servo power. When the switch is in MANUAL, the machine can be used in DRO-mode (servo-off), with spindle on/off capability.

See section 4.4 for setting ABSOLUTE ZERO (Part-Zero).

Many other things can be done in DRO mode (without servo power). Things such as creating and editing programs, using DRAW mode, using the Floppy Disk Drive, and more.

4.3 JOG - MOVING the MACHINE UNDER SERVO POWER

The term "JOG" refers to moving the machine with the X+, X-, Y+ and Y-, Z+, Z- keys explained in section 2.1 above. The SERVO motors must be ON in order to JOG the machine. While the servo motors are OFF, the machine can still be moved by hand with the handles provided, thus using the CNC like a DRO (see section 4.2 above).

Referring to the axis description in section 3.2, the machine can be jogged in the X+, X-, Y+, Y-, Z+, or Z- directions, in one of several ways.

First, the SERVO motors MUST BE ON. To do this, press the E-STOP IN, and TWIST CW OUT, then press the SVO RESET key. If the servos do not come on, the machine may be on a LIMIT SWITCH. The limit switches are provided to ensure the machine is not jogged past the end of its travel (while under power), resulting in mechanical damage. The limit switches also work in AUTO and S.STEP modes.

After energizing the servos, the machine may be JOGGED. The steps required to JOG the machine are as follows:

1. Select the method of JOG (Rapid, Feed, 100, 10, or 1) by using the "HAND" symbol in the center of the X+,X-,Y+,Y-,Z+,Z- keys. The current selection of the HAND SYMBOL is shown on the CRT screen in the upper-right area after the word **JOG:**

Rapid	= set by the machine builder,	usually 100-200 I.P.M.
Feed	= set by the machine builder,	usually 40 I.P.M.
100	= system resolution times 100,	usually .0100"
10	= system resolution times 10,	usually .0010"
1	= system resolution times 1,	usually .0001"

This step will determine how the tool (machine table) will move when step #2 below is done.

2. Press the desired key:

X+ = Moves the tool to the right (machine table left).
X- = Moves the tool to the left (machine table right).
Y+ = Moves the tool toward the machine column (machine table toward you).
Y- = Moves the tool toward you (machine table towards machine column).
Z+ = Moves the tool up (away from table surface).
Z- = Moves the tool down (toward table surface).

You should always visualize the TOOL as moving, rather than the TABLE moving. This will give a better understanding of CNC programming, no matter which model CNC you use.

NOTES: RAPID and FEED are "continuous jog" functions, in other words, the axes will continue to move as long as the key is HELD IN. When the key is released, the axis will stop. Travel limit switches are provided to prevent machine damage.

JOG 100, JOG 10, and JOG 1 are "incremental jog" functions, in other words, only one (1) increment will be moved for EACH

press of the X+, X-, Y+, Y-, Z+ or Z- keys. See above (step 1) for amount of movement per key-strike.

JOG is a very useful way to position the machine in order to use an edge finder, sweep indicate a bore, set ABSOLUTE ZERO (Part-zero), even make skim cuts without creating a program.

4.4 SETTING ABSOLUTE ZERO (PART ZERO)

Absolute zero is THE reference point where the dimensions of a blueprint are taken from. It is also known as PART-ZERO. On most quality blueprints, the dimensions are referenced from a corner of the part, or a centrally located bore, or at the center of a round workpiece. The operator must decide where the most convenient point is, and set ABSOLUTE ZERO there.

For workpieces held in a VISE, the best place to set PART-ZERO is on the SOLID JAW for Y ZERO, and at ONE END for X ZERO.

A ZERO point must be set for all 3 axes; X, Y, and Z. For Z axis, TLO's (Tool Length Offsets) are generally used for each tool.

In order to set ABSOLUTE ZERO, the CNC must be in MANUAL mode (servos on), or in DRO mode (servos off). In either mode, the operator must move each axis to "find" the desired edge or bore.

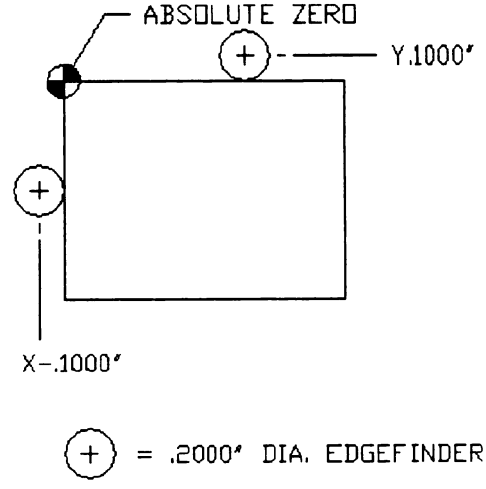
In the figure and example below, a rectangular workpiece is held in a vise. Most dimensions on the blueprint originate from the upper-left corner of the part. ABSOLUTE ZERO will be set at that corner for X and Y axes. Z-HOME position will be set at the top of the quill's travel. Use the below figure with the example steps, to set ABSOLUTE ZERO.

EXAMPLE:

1. Place the .2000" dia. edge-finder into a tool holder, tighten, and place in spindle.

2. Using DRO or MANUAL mode, move the tool beyond the left edge of the part, and lower the Z axis.

3. Move the X axis until the edge-finder wobbles. No matter what is showing in the X display, we know that the current X position must be set to X-.1000" (1/2 the dia.



of the edge-finder).

4. Press X-.1 ENTER The display will change to X-.1000. You have just set ABSOLUTE ZERO for the X AXIS.

5. Repeat steps 3 and 4, except for the Y AXIS. The Y dimension, however is POSITIVE, not negative.

6. To do a visual check, raise the (Z-axis) edge-finder clear of all obstructions, and press X0 Y0 START. The servo motors must be ON for this step. The machine will move to the new ABSOLUTE ZERO (Part-Zero). How the machine moves there (rapid or feed) depends on what is showing behind the word **JOG:** in the upper-right status area of the CRT-screen. See section 4.5 below for MANUAL MOVES UNDER POWER.
   If in DRO mode, the visual check can be done by cranking each axis to ZERO by hand.

7. To set ABSOLUTE ZERO for the Z AXIS (Z-HOME), move the quill to the top of its travel, and press Z0 ENTER. TLO's (Tool Length Offsets) can now be done from this position. See section 3.5 above for TLO's.

NOTES:

1. As seen above, ANY number can be set into the axis displays, not only 0 (zero). Pressing X5 ENTER will set the current X position to 5.0000"
2. **BE SURE** to press **ENTER**, not START, when setting ABSOLUTE ZERO. If START is pressed, the CNC will move to the position programmed (like X.1000 or X5.0000), NOT change the display as desired. This is known as a MANUAL MOVE, described in section 4.5 below.

4.5 EXECUTING MANUAL (ONE-SHOT) MOVES UNDER POWER

While in MANUAL mode (servo motors ON), X-Y-Z positioning commands may be given to the CNC. These are known as MANUAL MOVES, since they are done in the MANUAL mode. This allows the operator to quickly position the table to any coordinate. It can be used as a "positioner" to drill holes, mill the end of a part, or any other simple operation that requires only RAPID and/or FEED moves.

4.5.1 STATUS of the MANUAL MOVE

First, the operator should look at the **JOG:** status item in the upper-right status area of the CRT - screen. This will determine whether the move he/she programs will be a RAPID move, or a FEED move. Also, the operator must consider whether an **ABSOLUTE** or **INCREMENTAL** move is needed. This can also be found in the upper-

right status area.

If the **JOG:** status shows:

RAPID	=	Rapid move (at machine's rapid rate)
FEED	=	Feed move (at last programmed feedrate)
100	=	Feed move (at last programmed feedrate)
10	=	Feed move (at last programmed feedrate)
1	=	Feed move (at last programmed feedrate)

The actual feedrate at which the machine will travel when the JOG: status is in FEED, 100, 10, or 1 will be the last programmed feedrate. So if a program with a FEED 15.4 was previously run, 15.4 will be the feedrate for the MANUAL MOVE.

The FEEDRATE OVERRIDE switch may be used to adjust any of the above feedrates, from 0% to 120%. If the desired feedrate cannot be reached through the above choices and OVERRIDE, an M.D.I. program with a feedrate programmed in it can be done. See section 4.6 and 4.7 below for M.D.I. mode.

If the move required should be **ABSOLUTE** (in reference to PART-ZERO), the status item must be ABSOLUTE. If the required move should be **INCREMENTAL** (in reference to the CURRENT X-Y POSITION), the status item must be INCREMENTAL.

This can be "toggled" back and forth by the **ABS/INC** key on the keyboard.

4.5.2 MAKING the MANUAL MOVE

After the operator is satisfied the correct FEEDRATE will be used, and the move is ABSOLUTE or INCREMENTAL, the move may now be programmed.

EXAMPLE MANUAL MOVES:

1. Press X1 Y1 START. — The CNC will move to X1.0000 Y1.0000 if in ABSOLUTE, or move at a 45 degree angle from its current position if in INCREMENTAL. It will either move in RAPID or at a FEEDRATE, depending on the status of JOG:

2. Press Y-3.75 START. — The Y axis will move to Y-3.7500 if in ABS, or Y-3.7500 from its current position if in INCR. A move like this could be used to mill the end of a part off.

Notice how the **START** key must be pressed to "execute" the MANUAL MOVE. If ENTER had been pressed, the axis display would be set to that position.

NOTES:

1. The MANUAL MOVE can be used to position the CNC to a known coordinate quickly and accurately.
2. The START key will activate the move immediately.
3. A move can be HALTED by pressing the HOLD key. If HOLD is pressed, the move can be re-started by pressing START again.
4. A move can be ABORTED (canceled all-together) by pressing soft-key F4 (MANUAL). If aborted, arrow-up will still recall the move.
5. The CNC assumes "0.0000" if no coordinate is given.
6. The "Hot-Keys" may be used directly from the MANUAL mode, to select RAPID, LINE, ARC moves, as well as FEEDRATE, TOOL#, MCODE, and UNITS. If selected and saved, they will be executed as a one-shot move after pressing the START key. See section 6.17 below for more detatil on "Hot-Key" functions.

4.6 WHAT IS M.D.I. MODE ?

M.D.I. (Manual Data Input) Mode is a quick way to execute a move, or SERIES of moves without creating and loading a program. Under normal programming circumstances, an operator will
1.) go to the PROGRAM screen,
2.) CREATE a program,
3.) DRAW the program to check for errors,
4.) LOAD the program, then
5.) RUN the program in AUTO to produce piece-parts.
With M.D.I. mode, steps 1,2 and 4 can be eliminated. M.D.I. mode is intended to shorten the programming process. The M.D.I. program, however, can be made into a full-fledged program later.

4.7 ENTERING and USING M.D.I. MODE

To enter the M.D.I. mode, the servo motors must be ON, and the CNC in MANUAL MODE. Pressing Soft-Key F7 (MDI) will activate the MDI mode.

When MDI is pressed, the CRT - screen and Soft-Keys will change to the program EDIT screen. All EDIT functions are available including DRAW, DRILL, POCKET, MILL, CALC and MISC. All of these are explained in detail in section 6 below, and apply to MDI mode also. See section 6 for EDIT.

After using the edit functions to build the MDI program, the operator can check the programmed moves using DRAW (F2). See section 9 below for details on DRAW mode.

4.7.1 RUNNING the MDI PROGRAM

Once the MDI program moves are entered by the operator, and checked out using DRAW, the program is ready to be RUN.

To RUN the MDI program, simply press F10 (EXIT), and the program will begin to run, after a message appears, and the START key is pressed. Messages may appear, such as "Tool Change", or any message that might apply to the MDI program. Follow the message prompts, press start, and the MDI program will execute.

NOTES:

1. DRAW (F2) can be used to see the programmed moves before cutting material. See section 9 for DRAW mode details.
2. The MDI program will be run completely, and when finished, the CNC will return to MANUAL mode.
3. The MDI program is run automatically when F10 (EXIT) is pressed. Messages will appear as normal.
4. All normal programming functions (DRILL, POCKET, MILL, etc.) may be used in MDI mode.
5. If the MDI mode is entered into again, the programmed blocks are shown, and the operator may edit a block(s) and when EXIT (F10) is pressed, the MDI program will run again.
6. If the operator wishes to save the MDI program as a permanent program file, he/she should press F2 (PROGRAM), place the cursor on MDI.M, and press F10 (UTILITY), then select RENAME. This will allow the MDI program to be given a name and saved in the program directory. Now, a new and different MDI program may be created. The renamed program may be run like any other program in the directory, in AUTO or S.STEP modes. See section 8 for AUTO and S.STEP modes.

EXAMPLE:

1. Ensure the servo motors are ON, and the CNC is in MANUAL mode.
2. Press MDI (F7).
3. Press TOOL# (F6).
4. Press 0 ENTER (or F10 SAVE).
5. Press MILL (F5).
6. Press RAPID (F2).
7. Key in 0.0000 for Z-axis, then press SAVE (F10).
8. Press PREVIOUS (F9).
9. Press EXIT (F10) to begin running the MDI program.
10. The CNC will cancel any TOOL OFFSET that may have been active, then RAPID the Z-axis to its home position (after the message appears and the START key is pressed).

4.8 MESSAGE SCREEN

While in MANUAL mode or DRO mode, Soft-Key F1 is MESSAGE. When pressed, the last eight (8) messages given on the "Message Line" are displayed, with the most recent one at the bottom.

This screen can be used to check error messages if they occur,

and keeps a record of the last 8 messages. As mentioned, the most recent of the eight is at the bottom, and the oldest of the eight is at the top. The message screen is cleared when power is turned OFF to the CNC.

4.9 FEEDRATE OVERRIDE

The FEEDRATE OVERRIDE is controlled by a rotary switch located near the E-STOP on the front of the CNC console. The switch has thirteen positions (0% through 120%), and can be rotated while the CNC is idle or while it is cutting. It allows the CNC's MOTION to be "slowed down" or "sped up".

The switch applies to both RAPID and FEED moves: The switch will override feed moves (lines, arcs, pockets) from 0% to 120%. It will override rapid moves from 0% to 100%

If a feedrate of 10.0 I.P.M. is being used and the rotary switch is rotated to 30%, a feedrate of 3.0 I.P.M. will be in effect. The effective feedrate and percentage of the programmed feedrate are shown in the upper-left status area of the CRT - screen.

The machine WILL NOT MOVE under servo power if the switch is at zero % (0%).

4.10 SOFT-KEYS of MANUAL MODE

The Soft-Key strip of the MANUAL mode allows the operator to access different modes of the CNC. Each Soft-Key is briefly explained below, and may refer you to the section of this manual that explains the function of the key in full detail.

MESSAGE

F1: A special message screen where the last eight (8) messages may be viewed. See section 4.8 above.

PROGRAM

F2: Brings the "PROGRAM" screen to the CRT, where programs are created, deleted, edited, copied, etc. See section 5 below.

EDIT

F3: Allows the "work" (active) program to be edited. See section 6 below.

F4: This key is active now, and is shown as highlighted

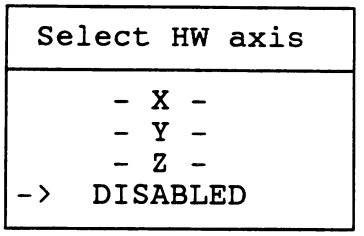
MANUAL	**F4:** This key is active now, and is shown as highlighted in reverse video (bright background, dark letters). See section 4 above.
S.STEP	**F5:** Puts the CNC into Single-Step mode, where the WORK program may be "stepped" through, one step at a time. See section 8.2 below.
AUTO	**F6:** Puts the CNC into Auto mode, where the WORK program may be run continuously. See section 8.3 below.
MDI	**F7:** Allows the operator to program and execute a series of moves without creating and loading a program. See sections 4.6 and 4.7 above.
	F8: Blank key. Has no use in Manual Mode.
TOOL	**F9:** Brings the Tool Page to the CRT, where diameter and length offsets are stored for the tools used in a program. See section 3.5 above.
EXIT	**F10:** Exits the CNC mode, and brings the "Software Options" screen to the CRT. This is where the Diagnostic and Set-Up utilities are located. See the Service Manual. The E-STOP (emergency-stop) button must be IN (servo motors off) in order to exit to the software options screen.

4.11 HANDWHEEL MODE

"Handwheel Mode" is a special part of the MANUAL mode, where the machine may be "jogged" by using a remote handwheel. Soft-key F8 while in Manual Mode will activate/deactivate handwheel mode.

4.11.1 ACTIVATING and USING the HANDWHEEL:

The Handwheel (hereby referred to as HW) must be plugged into its receptacle (DB-25 pin) in the rear of the CNC's console. Once the HW is plugged in, it may be activated by pressing F8 (HANDWHL) while in Manual mode. When F8 is pressed, a window appears with the available axes shown (X,Y,Z) and also shown is DISABLED.



The cursor (arrow) will appear next to the active selection (DISABLED). To Activate any axis, simply move the cursor to the desired axis, and press ENTER. The window will 'close' and the HW will be active for that axis.

SPEED and DIRECTION:

When the HW is rotated clockwise (CW) the move is POSITIVE.
When the HW is rotated (CCW) the move is NEGATIVE.

The SPEED of the movement is dependant on the JOG selection, and how fast the operator turns the handle:

JOG 100 = fast.
JOG 10 = medium.
JOG 1 = slow.

The HW will NOT operate if the JOG selection is RAPID or FEED.

Note that the HANDWHL soft-key (F8) is highlighted, identifying that the HW mode is active. The active axis is denoted after the word **HANDWH:** in the upper-right status area.

4.11.2 DE-ACTIVATING the HANDWHEEL:

The HW is deactivated by selecting F8, then selecting DISABLED. After selecting DISABLED, the window will 'close', and the handwheel will no longer be active. The status area will change accordingly.

5.0 PROGRAM HANDLING

5.1 WHAT IS PROGRAM MODE ?

Program mode is a screen where new programs may be created, old programs edited or deleted. It is through this screen where a program must be LOADED in order to be run in Auto or S.Step modes.

The PROGRAM screen shows the "directory" of programs. A directory is a group of programs in the CNC's memory. These programs will stay in memory until deleted by the operator. Turning power On-Off does not affect the programs in the directory.

Other functions may be done through the PROGRAM screen, such as RS-232 communications, sending or getting programs to - from a floppy disk, and more. Each Soft-Key of PROGRAM mode will be explained in section 5.3 through 5.11 below, and may refer you to the section of this manual that explains the function of the key in full detail.

5.2 ENTERING the PROGRAM MODE

PROGRAM mode is entered into by pressing F2 (PROGRAM) while in the Manual Mode. The CRT - screen will change, showing all programs in the CNC's "user" memory (other programs exist in the CNC, but are special programs than run the CNC, and cannot be edited or deleted).

The ARROW keys are active, and may be used to move the highlighted block (cursor) from one program to another by going up, down, left, or right.

Before a function is performed on a program (such as load, copy, edit, delete, etc.), the CURSOR must first be placed ON that program name.

5.3 CREATE - F2

F2 (Create) allows a new program to be created in the directory. When F2 is pressed, a special cursor will appear near the lower-left corner of the screen asking for the NEW PROGRAM's name.

NEW PROGRAM: ___

At this point, a name must be keyed in, and ENTER pressed. Program names may contain a MAXIMUM of 8 characters (with the exception of the decimal point - it cannot be used in a program

name). Any combination of LETTERS or NUMBERS may be used to make up the program name (see below for access to letters). The suffix of (.M) is given automatically.

Once a program is CREATEd, it will appear in the directory of the PROGRAM page, and the cursor will automatically move to the new program name. Editing the new program is now possible, by pressing F4 (Edit).

USING LETTERS for program names:

When CREATE is pressed, the operator may use any numbers or letters present on the CNC keyboard (0-9, and X,Y,Z). This however, may not be enough to give a proper name:

Softkey **F2 (ASCII)** allows the operator to pick any letter of the alphabet, as well as other characters, to make up the program name.

EXAMPLE: To name a program **DWG-1010**

1. Press F2 (CREATE).
2. Press F2 (ASCII).
3. Use the arrow keys to move the cursor to the letters D, W and G, pressing ENTER for each letter.
4. Press F2 (ASCII), to turn off the ASCII table.
5. Continue, using the CNC keyboard to key in the - and the 1010.
6. Press ENTER, and the program will appear in the directory.

The process of creating a new program is now complete.

5.4 DELETE - F3

DELETE

F3 (Delete) allows the program that the CURSOR IS ON to be deleted from memory. When F3 is pressed, a message will appear in the center of the CRT-screen asking the operator to verify if the program is to be deleted, or not. Pressing F1 (Yes) will delete the program, pressing F2 (No) will abort the process. This ensures a program is not accidentally deleted.

5.5 EDIT - F4

EDIT

F4 (Edit) allows the program that the CURSOR IS ON to be edited. Edit mode is where the "blocks" of the program are input into a NEW program, or blocks edited in an existing program.

This is where the X-Y movement instructions are given to machine a workpiece.

NOTE: A program does not need to be "LOADed (F6)" to be edited from the PROGRAM page (see LOAD below). Simply move the cursor onto the desired program and press F4 (Edit), and the CNC will be in Edit mode. See section 6.0 for EDIT mode details.

5.6 LIST - F5

LIST

F5 (List) allows the program the CURSOR IS ON to be reviewed. LIST simply shows the program to the user with the appearance and some functions of EDIT mode.

The ARROW keys allow movement of the cursor so the entire program can be viewed.

The soft-keys also provide other cursor movement functions, like the SEARCH (F3) function, which may be used to search a program block number (16,17,27,55, etc.). Page-Up, Page-Down, Begin (of program) and End (of program) functions are also available on Soft-Keys F4 - F7. Draw Mode is available through F2 (see section 6.4 below for Draw mode details).

EXIT (F10) will change the CRT - screen back to the PROGRAM directory screen.

In LIST mode the program cannot be changed (edited), only viewed.

5.7 LOAD - F6

LOAD

F6 (Load) takes a copy of the program the CURSOR IS ON, and puts into the "work area" of the CNC's memory. In order to RUN a program in AUTO or S.STEP modes, it MUST first be LOADED.

To LOAD a program, move the cursor to the desired program, and press **F6** (LOAD). The program name will appear at the lower-right area of the CRT - screen after the word **WORK PROGRAM:**

Example: If the cursor is moved down so it is resting on a program named 12345678.M, and F6 is pressed, that program is loaded into the working memory of the CNC and can now be RUN. The "work program" name is displayed near the lower-right corner of the CRT:

WORK PROGRAM: 12345678.M

This indicates the "working" program.

Once a program is LOADed, it remains loaded until another is loaded. If another program is loaded, the program that was previously loaded is moved out of the working memory, but remains in the program storage directory.

NOTES:

1. A program must be loaded to perform the following tasks with that program:
   a) Run the program in AUTO or S.STEP modes.
   b) Edit the program from the MANUAL Mode.

2. Once the PROGRAM page is exited, all functions that have to do with a program will be performed on the LOADed program.

3. When the program page is exited, the LOADed (or working) program is displayed after the word **PROGRAM:** in the upper-right status area of the CRT - screen.

5.8 LOG - F7

LOG

F7 (Log) allows the operator to "log onto" a floppy disk in the floppy drive. The floppy disk must be placed into the floppy drive before LOG is used.

Pressing F7 will show the available drives in a box at the center of the CRT - screen. Usually only 2 drives will be available:

A: The floppy disk drive, and
C: The CNC'C memory (where system files are stored), and
D: The USER'S memory (where the part programs are stored).

NOTE: A: and D: are the only drives the operator should work with. C: contains system files, and should not be accessed.

The ARROW keys can then be used to move the cursor onto the desired drive. Once a drive is "Logged" onto, the CRT - screen will show the programs on that disk. Always log back to D:, this is the normal operating drive for part-programs.

The main function of LOG (F7) is so the operator can copy programs to and from the floppy disk and the CNC's memory.

NOTES:

1. See section 10 below "Using the Floppy Disk Drive, and RS-232 Communications" for copying to/from floppy disk.
2. If the floppy disk is logged onto (drive A:), you MUST log back to drive D: to operate the CNC.

5.9 DISPLAY - F8

DISPLAY

F8 (DISPLAY) allows the operator to see more information about the programs in memory. F8 is a toggle key, with 4 screen possibilities. The first is already present on the screen:

PART PROGRAM **NAME** and SUFFIX:
CRT - screen shows only the part program name and extension (suffix) on the screen.

Press F8 again .....

PART PROGRAM **NAME**, SUFFIX and **SIZE** (bytes), **DATE** and **TIME** last edited:
CRT - screen shows the part program name, size in number of bytes, and the date and time the program was last edited.

Press F8 again .....

ALL PROGRAM **NAMES** and SUFFIX:
CRT - screen shows all the programs' names and extensions on the screen (including directories). Only the programs ending with the suffix (.M) can be used. Do not attempt to edit any other programs that appear on the screen, except the programs ending with (.M).

Press F8 again .....

ALL PROGRAM **NAMES** and SUFFIX, **SIZE**, **DATE** and **TIME** last edited:
CRT - screen shows all the programs' names and extensions on the screen (including directories), as well as the size in bytes, and the date and time last edited. Only the programs ending with the suffix (.M) can be used. Do not attempt to edit any other programs that appear on the screen, except the programs ending with (.M).

Press F8 again .....

and you are back to the first and most common PROGRAM screen display, where only the program name and suffix are shown.

You can set the CNC to show any one of the above displays as a default condition (in other words, it will show whatever screen you want when you turn the power on. You have a choice. See the Service manual for default settings).

5.10 UTILITY - F9

UTILITY

F9 (Utility), when pressed, will bring a "Pull-Up window" to the screen where several different "program handling" functions are available to the operator. If the operator does not wish to use any of the functions in the pull-up window, pressing CLEAR or F9 again will "put the window away".

As with most pull-up windows in the Series 1100, the cursor will initially be positioned on the 1st function inside the window. To "select" a function, use the up/down ARROW keys to move the cursor to the desired function, then press ENTER. The function will begin, on the program the CURSOR IS ON prior to selecting F9.

Copying programs, renaming programs, printing programs, and many more functions can be done in the Utility window of the PROGRAM page.

Utility functions:

5.10.1 COPY

COPY allows the program that the CURSOR IS ON to be copied, either to another drive (such as floppy drive A:), or a NEW program name. When COPY is selected, a message will appear at the center of the CRT - screen asking the operator to specify where the program is to be copied to (Drive A:). If A: is selected, the program will be copied onto the floppy disk automatically. All that is necessary is to select **A:** and the program will be copied to the floppy disk. The original program will remain in the CNC memory.

Copy is generally used to copy programs to the floppy disk for future use, or to make a duplicate program with the same data in it.

EXAMPLES:

a) Copy 12345678.M to A: will copy the program onto floppy disk. The program 12345678.M will now be present on both CNC memory and floppy disk. The original program remains untouched.

b) Copy 12345678.M to "Other...". When OTHER is selected, a prompt will appear in the center of the CRT-screen, asking "Copy to:" The operator must key in a name (such as 55-6), and the CNC will make a copy of the original program with the new name. The contents of the two programs will be identical.

5.10.2 RENAME

RENAME allows the program that the CURSOR IS ON to be given a new program NAME. When Rename is selected, a message will appear at the bottom-left of the CRT - screen, asking the operator to key in the new name, otherwise called the DESTINATION PROGRAM.

EXAMPLE:

Rename 12345678.M to 55-6.M will change the name of the program. 12345678.M will no longer exist under its original name.

5.10.3 PRINT

PRINT allows the program that the CURSOR IS ON to be down-loaded to a printer. The printer must be of the parallel type, and be connected to the printer port, which may be located in the rear of the computer cabinet.

When YES (F1) is selected, the program will be downloaded to the printer, and a message will appear on the CRT indicating when the process is complete, how many lines and pages were printed, and to press F10 to continue. If the printer connection is bad, a message to that effect will appear, and the operator may cancel the process or try again.

5.10.4 UNDELETE

UNDELETE allows a program that was previously deleted to be "undeleted". This can be used to recover a program that was accidentally deleted using the Delete (F3) selection of the PROGRAM Page (see 5.4 above).

EXAMPLE:

Select "Undelete" from the Utility window, and a window will appear with all the programs that are able to be 'UnDeleted'. Move the cursor up or down onto the desired program (ONLY programs with the SUFFIX (.M) should be undeleted), then press ENTER. At this point, the prompt will ask for the 1st character of the program-name. Key it in, then press ENTER. If possible, the program will be recovered into the PROGRAM directory, and a message will appear on the CRT - screen, asking you to check the program carefully.

5.10.5 FLOPPY FORMAT

FLOPPY FORMAT (Format Floppy Disk) allows the operator to format a 3-1/2" 1.44MB 2S-HD floppy disk in the floppy disk drive located on the side of the computer cabinet. All floppy disks must be "formatted" before programs can be placed onto them (some floppy disks are pre-formatted before you buy them). Simply select this choice from the Utility window AFTER placing a disk in the floppy drive. If a disk is not present, or if an error occurs, a message

to that effect will appear. Press F10 to continue after format is complete. **WARNING:** Any data present on the floppy disk will be deleted when formatted.

5.10.6 COMMUNICATIONS

COMMUNICATIONS, is **RS-232**, and when selected, the screen will change to the Communications page. Communications is used to transfer programs into or out of the CNC to an offline system.

See section 10 below for COMMUNICATIONS.

5.10.7 G-CODE CONVERTER (G-code to S1100)

The G-CODE CONVERTER, when selected, TRANSLATE a G-coded program into the Series 1100's machinist language.

See section 10.3 below for G-CODE CONVERTER.

5.10.8 MORE

MORE, when selected, will "pull-up" another window with MORE utility functions. The functions in this window are less common, therefore put into this secondary "window". The selections under MORE are:

Check Disk allows a disk-check function to be performed on a floppy disk or the CNC's computer. When Check Disk is selected, the prompt will ask which disk to check. After selecting which disk, the function will begin. If any errors on the disk are found, they can be corrected by answering YES to the question that will appear at the end of the disk check function. If NO or CANCEL is answered, no damage to the memory will occur.

System Info (System Information), when selected, will display information on the CNC hardware and software in a window that will appear in the center of the screen. Press F10 to continue after reviewing the information.

Copy ? allows the operator to copy programs that DO NOT appear in the PROGRAM screen. This function could be used to copy a program from a floppy disk to the CNC, without "Logging" onto the floppy disk. The **?** denotes this function is to work on an operator-specified program, not the program the cursor rests on.

Rename ? allows the operator to rename programs that DO NOT appear in the PROGRAM screen. This function could be used to rename a program on a floppy disk, without "Logging" onto the floppy disk. The **?** denotes this function is to work on an operator-specified program, not the program the cursor rests on.

Print ? allows the operator to print programs that DO NOT

appear in the PROGRAM screen. This function could be used to print a program on a floppy disk, without "Logging" onto the floppy disk The **?** denotes this function is to work on a operator-specified program, not the program the cursor rests on.

Mark All allows the operator to "MARK" all the programs in the PROGRAM screen. This function could be used to quickly MARK, then copy ALL programs from CNC memory to the Floppy disk. See section 5.10.8 below for description of MARK.

UnMark All allows the operator unmark all the programs in the PROGRAM screen. This function could be used to unmark all the programs.

5.10.8 MARK

MARK is a way to do an operation on several programs, instead of just one program. As noted, when COPY or DELETE is selected, the CNC assumes that you want to copy or delete the program that the CURSOR IS ON. By "Marking" several programs, the CNC will assume you want to copy or delete only the MARKED programs. This can save time when several programs need to be copied to the floppy disk, or deleted from memory.

While in the PROGRAM screen, pressing **Enter** while the cursor rests on a program will **Mark** the program. This allows programs to be individually "flagged", so a function can be performed on more than one program. Any number of programs may be marked, in random selection. If one or more programs are marked, the COPY or DELETE keys will act ONLY on the MARKed program(s), and will override the cursor position.

EXAMPLE:

The cursor is moved onto 5 random programs, and ENTER is pressed when the cursor was on each program. These programs will now be MARKed. The operator may now copy or delete the marked programs by performing the copy or delete function only once.

Pressing **Enter** while on a "Marked" program will **UnMark** the program. Also, exiting the PROGRAM page will automatically UNMARK any Marked programs.

Programs that are marked, then deleted need not be UNMARKED, since they no longer exist.

5.11 EXIT - F10

EXIT

F10 (Exit) will exits the PROGRAM mode, and brings the MANUAL screen back to the CRT. This is where the AUTO and S.STEP modes are accessed from.

6.0 EDIT MODE - PROGRAMMING THE PART

6.1 WHAT IS EDIT MODE ?

Edit mode is the mode where a part-program is "built", block - by - block. As described in section 3.1 above, a PROGRAM is a series of instructions that the CNC uses to machine a workpiece. The program consists of blocks (individual lines in the program). It is in the EDIT mode where the blocks are programmed and stored in memory.

Edit mode is used to build a new program, or to change data already present in an existing program. Soft-Keys in the EDIT mode allow the programming of LINES, ARCS, POCKETS, DRILL holes, and RAPID moves. Other features such as CORNER-ROUND, MIRROR IMAGE, ROTATION, SCALING and SUBPROGRAMS are also done using the edit mode. The CALCULATOR(S) may also be used in EDIT mode, to do addition, subtraction, multiply, divide, and do trig functions. CALC also allows the operator to find sides or angles of a right-triangle, and build "geometry" (points, lines and circles) in order to find tough-to-find answers.

TYPICAL STEPS INVOLVED IN THE PROGRAMMING PROCESS:

1.) go to the PROGRAM screen,
2.) CREATE a program,
3.) EDIT the program,
4.) DRAW the program to check for errors,
5.) LOAD the program, then
6.) RUN the program in AUTO (or Single-Step) to produce piece-parts.

It is also through edit mode that DRAW graphics can be used to show the programmed moves in a part-program, so the operator may verify the program before cutting material.

All aspects of EDIT mode will be detailed throughout section 6, or refer you to the appropriate section of this manual.

6.2 ENTERING EDIT MODE

Edit mode is entered into (accessed) in one of two ways, both from the MANUAL screen display:

a) Press F2 (PROGRAM), place the cursor on the program you wish to edit (or create a new one), and press F4 (EDIT).

OR

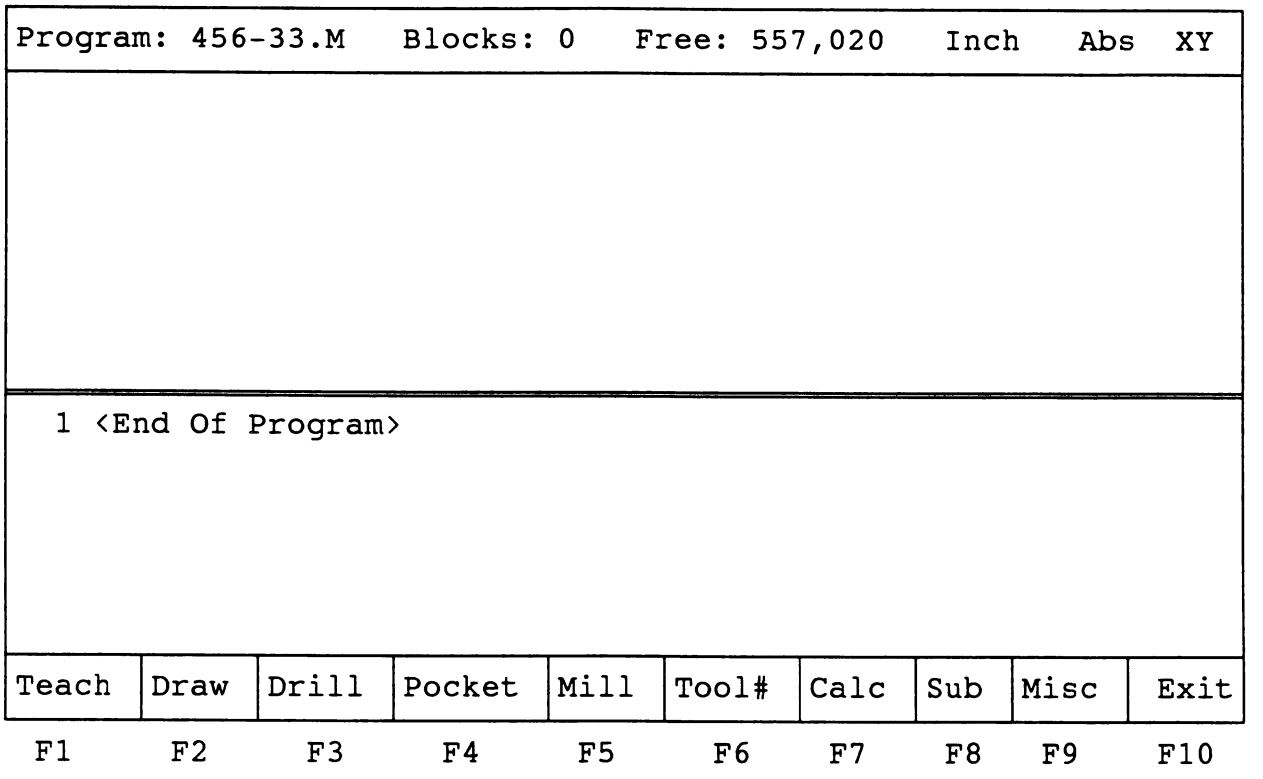
b. Press F3 (EDIT). A program must first be "LOADED" for edit to be reached this way. See section 5.7 above for LOAD.

In either case, the CNC will be in EDIT mode, ready to input

new blocks, or edit existing blocks.

NOTE: During the entry of data such as pockets, lines and arcs, you will notice that some entries are initially filled with zeroes --- this means that you **MUST** fill in that piece of data. Any entry that is blank is optional, in other words, you do NOT have to fill in that data, only if you deem necessary.

The following figure will show the EDIT mode screen when a NEW program is created, a program called "456-33".



The screen is divided into 2 main areas:

TOP AREA:

1. The narrow strip at the very top contains some status items: Program name, # of blocks in program, free memory (available bytes), Inch (or MM), Abs (or Incr) - for absolute or incremental dimensions active, and XY (or XZ, or YZ...for active plane).

2. The large blank area below the status strip will show the "entry graphics" and functions such as pocket, line, and arc are selected. The entry graphics are fixed (cannot be

changed), and are intended to give a general idea to the operator as to what kind of block he/she is about to enter into the program. This is where the "conversation" takes place, where the operator fills in the blanks.

BOTTOM AREA:

1. The blank area will show the program blocks as they are entered into the program. An <End Of Program> block will automatically show, and tells the CNC that this block is the end-of-file.

2. At the very bottom are the Soft-Key labels for the main EDIT screen. These Soft-Keys will change, depending on which Soft-Key is chosen. For EXAMPLE: If MILL (F5) is pressed, the Soft-Key strip will now show the choices the operator has in reference to milling. This is called the MILL MENU. If DRILL (F3) is pressed, the DRILL WINDOW will be shown. If POCKET (F4) is pressed, the POCKET WINDOW will be shown, and so on.

3. The MAIN MENU of the EDIT mode is shown in the above figure, and will always show when EDIT mode is entered into.

The sections to follow (6.3 through 6.15) will explain in detail the functions of EDIT, or refer you to the appropriate section in this manual related to that Soft-Key.

6.3 TEACH - F1

TEACH

F1 (Teach) allows the machine's current position to be "Taught into" or "digitized" into the program. This can be beneficial if, when working in DRO mode, the operator wants to input the current XYZ position into a program.

See section 12 below for complete details on the TEACH mode.

6.4 DRAW - F2

DRAW

F2 (Draw) allows the program being edited to be "drawn" on the screen. This allows the operator to visually check the program before cutting material. Programming errors may be found, and the mistake can be edited, and the program DRAWN again.

See section 9 below for complete details on the DRAW mode.

6.5 DRILL - F3

DRILL

F3 (Drill) brings the DRILL WINDOW to the CRT-SCREEN. The DRILL window includes the BASIC, PECK, BORE, and CHIP-BREAK cycles. These cycles are designed to perform drilling with the Z-axis. The cursor will appear on the first item in the "window". To select an item, move the cursor up-down, and press ENTER. It is here in the DRILL window that the operator must first SELECT a type of drilling cycle, then he/she can give X-Y hole locations (by pressing X or Y, and keying in the coordinates, then press SAVE. See section 6.5.2 below). Also, the 2 hole patterns (PATTERN and BOLT-HOLE) are accessed from the DRILL window.

When the last hole location is programmed, the operator must "turn-off" the drill cycle. This is necessary so you don't have to program the BASIC drill cycle (or PECK, etc.) for every-single hole location. You program it once, BEFORE the 1st hole location, then program DRILL-OFF after the last hole location. See section 6.5.3 below.

NOTE: During the entry of data such as pockets, lines and arcs, you will notice that some entries are initially filled with zeroes --- this means that you **MUST** fill in that piece of data. Any entry that is blank is optional, in other words, you do NOT have to fill in that data, only if you deem necessary.

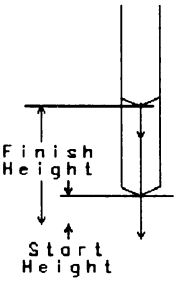
6.5.1 BASIC

While in the DRILL WINDOW, the first selection is BASIC. When selected, this function allows the operator to program a BASIC drill cycle, which feeds to depth, then rapids up-out of the hole.

Inputs:

Z-Depth: The ABSOLUTE Z-depth of the hole.

StartHgt: The ABSOLUTE Z position to begin feeding into the workpiece. The Z axis will rapid to this position before feeding in.



ReturnHgt: The ABSOLUTE Z position to rapid up-out of the hole, after it is drilled. (Optional, if not programmed, the Z will rapid out to the StartHgt dimension.)

Feed: The feedrate at which the drill will feed into the workpiece.

Tool #: The tool number from the Tool Page. (optional).

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.5.2 PROGRAMMING VARIOUS HOLE LOCATIONS

After setting the BASIC drill cycle described above (or any other DRILL cycle), the operator must tell the CNC WHERE to drill the holes. Various X-Y locations may be given by pressing X (or Y) from the current screen (the main edit screen). When X or Y is pressed, the input screen will appear on the top-half of the CRT-screen. The operator keys-in the hole location, then press F10 (SAVE, or ENTER). The location will go into the program. As many locations as necessary can be keyed in, one after another.

NOTES:

1. The two available "patterns" may also be used after setting any DRILL cycle. See sections 6.5.7 and 6.5.8 below for patterns.
2. Additional X-Y point formats (such as radius-angle) may be used for hole locations. These may be entered through the MILL, RAPID selection, by the softkey MORE...

6.5.3 PECKING

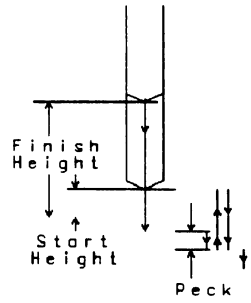
While in the DRILL WINDOW, the second selection is PECKING. When selected, this function allows the operator to program a PECK drill cycle, which feeds to the peck-depth, then rapids up-out of the hole, rapids back in (less the chip break increment), feeds to the next peck-depth, rapids out ... and continues this process until the full depth is reached.

Inputs:

Z-Depth: The ABSOLUTE Z-depth of the hole.

StartHgt: The ABSOLUTE Z position to begin feeding into the

workpiece. The Z axis will rapid to this position before feeding in.



ReturnHgt: The ABSOLUTE Z position to rapid up-out of the hole, after it is drilled. (Optional, if not programmed, the Z will rapid out to the StartHgt dimension.)

Peck: The maximum peck amount. All pecks will be equal.

Feed: The feedrate at which the drill will feed into the workpiece.

Tool #: The tool number from the Tool Page. (optional).

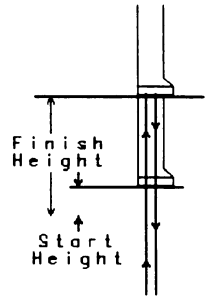
After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.5.4 BORING

While in the DRILL WINDOW, the third selection is BORING. When selected, this function allows the operator to program a BORE cycle, which feeds to the Z-depth, then feeds up-out of the hole.

Inputs:

Z-Depth: The ABSOLUTE Z-depth of the hole.



StartHgt: The ABSOLUTE Z position to begin feeding into the workpiece. The Z axis will rapid to this position before feeding in.

ReturnHgt: The ABSOLUTE Z position to feed up-out of the hole, after it is bored. (Optional, if not programmed, the Z will feed out to the StartHgt dimension.)

Dwell: The amount of time to dwell at the bottom of the hole. Time is in seconds.

Feed: The feedrate at which the drill will feed into the

workpiece.

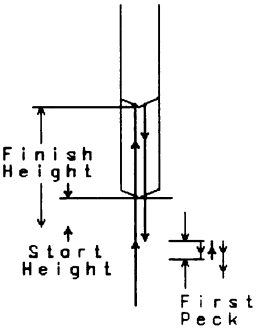
Tool #: The tool number from the Tool Page. (optional).

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.5.5 CHIP-BREAKER

While in the DRILL WINDOW, the fourth selection is CHIP-BREAK. When selected, this function allows the operator to program a CHIP-BREAKER drill cycle, which feeds to the 1st peck-depth, then breaks the chip with a standard chip-break of .0200", feeds to the next peck-depth, breaks the chip, ... and continues this process until either the full depth is reached, or the RetractDep depth is reached. The RetractDep makes the drill retract ALL THE WAY out of the hole. In this way, the drill may be "full-retracted" every-so-often. Between each full-retract, the drill will break the chip with the .0200" Chip Break Increment.

Inputs:



Z-Depth: The ABSOLUTE Z-depth of the hole.

StartHgt: The ABSOLUTE Z position to begin feeding into the workpiece. The Z axis will rapid to this position before feeding in.

ReturnHgt: The ABSOLUTE Z position to rapid up-out of the hole, after it is drilled. (Optional, if not programmed, the Z will rapid out to the StartHgt dimension.)

1st Peck: The amount of the FIRST peck.

PeckDecr: The amount to decrement from the 1st peck (the second peck will be this much less than the 1st peck... the 3rd peck this much less than the 2nd peck, etc.).

MinPeck: The minimum peck distance. No peck will be less than this amount.

ChipBrkInc: The amount to retract for breaking the chip. If not programmed, will be .0200". This may be adjusted in

light of drill size.

RetractDep: The drill will retract ALL THE WAY out of the hole each time it travels this amount into the workpiece. i.e; if total depth is -2.0000, and the RetractDep is .5000, the drill will full-retract 4 times.

Feed: The feedrate at which the drill will feed into the workpiece.

Tool #: The tool number from the Tool Page. (optional).

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.5.6 DRILL-OFF

While in the DRILL WINDOW, the fifth selection is DRILL-OFF. This MUST BE programmed AFTER the last hole location. This "turns-off" the BASIC drill cycle. In any CNC, this is necessary so you don't have to program a drill cycle for every-single hole location. You program the drill cycle once, BEFORE the 1st hole location, then program DRILLOFF after the last hole location.

After DRILLOFF is selected, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

In addition, if the system was upgraded to 3-axis, the 2-axis programs will be compatible.

6.5.7 PATTERN

While in the DRILL MENU, the 6th selection is PATTERN. When pressed, this function allows the operator to program a square or rectangular hole pattern, even a single line of holes:

Inputs:

X and or Y START: The X-Y coordinate of one of the CORNER holes.

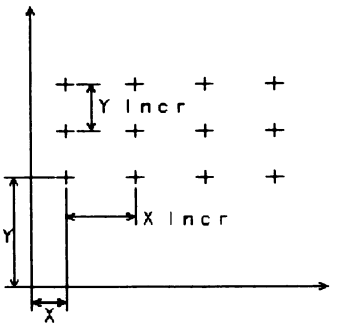
#X Holes: The number of holes running along the X-axis.

#Y Holes: The number of holes running along the Y-axis.

X Incr: The incremental distance between the holes

along the X-axis.

Y Incr: The incremental distance between the holes along the Y-axis.



Angle: The angle of the hole pattern. Straight would be zero. (default is 0 degrees).

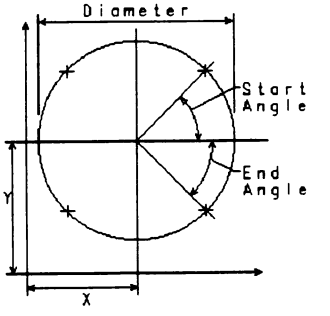
Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.5.8 BOLT-HOLE

While in the DRILL MENU, the last selection is BOLT-HOLE. When selected, this function allows the operator to program a bolt-hole circle:

Inputs:



X and Y CENTER: The X-Y coordinate of the CENTER of the bolt-hole pattern.

Holes: The number of holes in the pattern.

Diameter: The diameter of the bolt-hole pattern.

Start Angle: The angle of the first hole drilled, in reference to the polar coor'd system. 0 degrees is 3-o'clock position.

End Angle: The angle of the last hole drilled. Only necessary when drilling a "partial" bolt-hole pattern.

Index Angle: Used to rotate the whole bolt-pattern a specified amount of degrees. For example, if an 8 hole pattern was to be rotated +5 degrees, the start angle could be zero (rather than 5), the end angle left blank (rather than +320) and the index angle set to 5.

Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6 POCKET - F4

POCKET

F4 (Pocket) brings the POCKET WINDOW to the CRT-screen. The cursor will appear on the first item in the "window". To select an item, move the cursor up-down, and press ENTER. It is here in the POCKET window that the operator picks what type of pocket is needed, whether a rectangular, circular, frame, hole or irregular pocket. For "mold" cycles, see section 6.6.7 and 6.6.8 below.

Each pocket "canned cycle" requires certain inputs, such as length, width, depth, corner radius, etc.. No matter what the size of the pocket, the CNC will automatically do the moves necessary to mill out the pocket. As mentioned above;

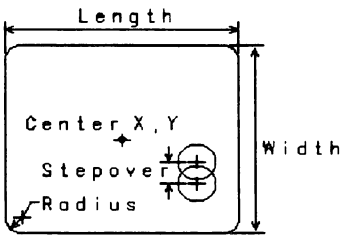
NOTE: During the entry of data such as pockets, lines and arcs, you will notice that some entries are initially filled with zeroes --- this means that you **MUST** fill in that piece of data. Any entry that is blank is optional, in other words, you do NOT have to fill in that data, only if you deem necessary.

See section 6.6.6 below for NOTES regarding pocket cycles.

6.6.1 RECTANGULAR POCKET

While in the POCKET window, the first selection is RECTANGULAR. When pressed, this function allows the operator to program a rectangular pocket.

Inputs:



X-Center: The X coordinate of the centerline of the pocket.

Y-Center: The Y coordinate of the centerline of the pocket.

StartHgt: The ABSOLUTE position of the Z-axis before beginning to mill the pocket. This MUST BE .1000" (or 2 mm) above the surface to be cut

into.

Length: The total length of the pocket (along the X axis).

Width: The total width of the pocket (along the Y axis).

Z-Depth: The ABSOLUTE depth of the pocket.

Direction: CCW = climb milling, CW = conventional milling.

Corner Rad: The actual radius of the corners of the pocket (must be at least the tool radius).

Stepover: The distance the tool will step-over (width of cut) as it mills out the pocket.

DepthCut: The depth per pass: If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth-Cut can be programmed to allow two or more passes. EXAMPLE: If a 2" deep pocket is needed, Z-Depth would be -2.0000", and Depth-Cut could be -.5000". This would result in 4 depth cuts (plus a finish pass, if programmed).

Fin Stock: Amount of stock to leave for a finish pass. The finish pass will be done automatically. Stock is left on all sides and bottom.

RampFeed: The feedrate at which the tool will "ramp" into the pocket with all 3 axes.

Rough Feed: The feedrate at which the pocket will be "roughed-out".

Fin Feed: The feedrate at which the pocket will be "finished".

Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6.2 CIRCULAR POCKET

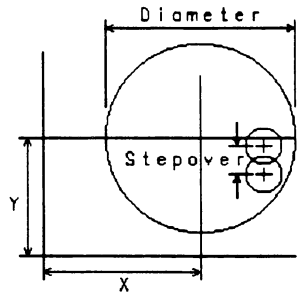
While in the POCKET window, the second selection CIRCULAR. When pressed, this function allows the operator to program a circular pocket.

Inputs:

X-Center: The X coordinate of the centerline of the pocket.

Y-Center: The Y coordinate of the centerline of the pocket.

StartHgt: The ABSOLUTE position of the Z-axis before beginning to mill the pocket. This MUST BE .1000" (or 2 mm) above the surface to be cut into.



Diameter: The diameter of the pocket.

Z-Depth: The ABSOLUTE depth of the pocket.

Direction: CCW = climb milling, CW = conventional milling.

Stepover: The distance the tool will step-over (width of cut) as it mills out the pocket.

Depth Cut: The depth per pass: If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth-Cut can be programmed to allow two or more passes.

Fin Stock: Amount of stock to leave for a finish pass. The finish pass will be done automatically.

RampFeed: The feedrate at which the tool will "ramp" into the pocket with all 3 axes.

Rough Feed: The feedrate at which the pocket will be "roughed-out".

Fin Feed: The feedrate at which the pocket will be "finished".

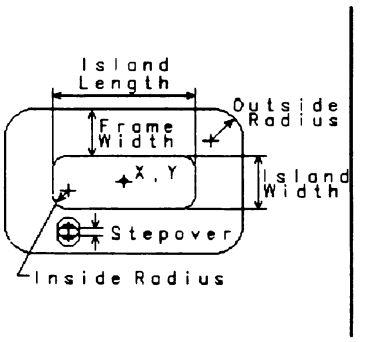
Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6.3 FRAME POCKET

While in the POCKET window, the third selection is FRAME. When pressed, this function allows the operator to program a "frame" pocket. This is a rectangular pocket with an island in the center.

Inputs:



X-Center: The X coordinate of the centerline of the pocket or island.

Y-Center: The Y coordinate of the centerline of the pocket or island.

StartHgt: The ABSOLUTE position of the Z-axis before beginning to mill the pocket. This MUST BE .1000" (or 2 mm) above the surface to be cut into.

Island Len: The length of the "island" (along the X axis).

Island Wid: The width of the "island" (along the Y axis).

Z-Depth: The ABSOLUTE depth of the pocket.

Direction: CCW = climb milling, CW = conventional milling.

Inside Rad: The actual radius of the corners of the island.

Outside Rad: The actual radius of the corners of the pocket (must be at least tool radius).

Frame Width: The width of the area to be milled out.

Stepover: The distance the tool will step-over (width of cut) as it mills out the pocket.

Depth Cut: The depth per pass: If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth-Cut can be programmed to allow two or more passes.

Fin Stock: Amount of stock to leave for a finish pass. The finish pass will be done automatically.

RampFeed: The feedrate at which the tool will "ramp" into the pocket with all 3 axes.

Rough Feed: The feedrate at which the pocket will be "roughed-out".

Fin Feed: The feedrate at which the pocket will be "finished".

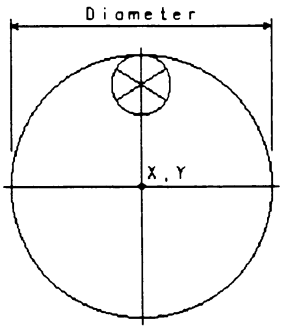
Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block.

When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6.4 HOLE

While in the POCKET window, the fourth selection is HOLE. When pressed, this function allows the operator to program a "hole-mill" cycle. HOLE cycles are ideal for counterbores on existing holes, or for larger through-holes.



Inputs:

X-Center: The X coordinate of the centerline of the hole.

Y-Center: The Y coordinate of the centerline of the hole.

Diameter: The diameter of the hole.

Z-Depth: The ABSOLUTE depth of the hole.

Direction: CCW = climb milling, CW = conventional milling.

Fin Stock: Amount of stock to leave for a finish pass. The finish pass will be done automatically.

Rough Feed: The feedrate at which the hole will be "roughed-out".

Fin Feed: The feedrate at which the hole will be "finished".

Tool #: The tool number from the Tool Page.

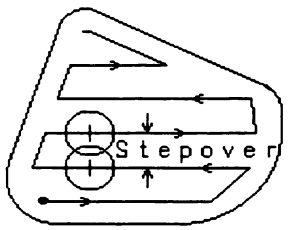
After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6.5 IRREGULAR POCKET

While in the POCKET window, the 5th selection is IRREGULAR. When pressed, this key allows the operator to program an "irregular" pocket, of nearly any shape. This pocket, however, requires a "subprogram" to describe the PERIMETER of the irregular pocket shape. See section 6.6.5.1 below for IRREGULAR pocket examples, and see section 6.10 below for structure of subprograms.

Inputs:

Sub #: The number of the subprogram that contains the perimeter of the irregular pocket. See note 5 below



X: The X position from where the step-over cuts will begin from. MUST be inside the pocket, including the tool radius. NOTE: If left blank, the CNC will pick its own point....It is recommended to leave blank.

Y: The Y position from where the step-over cuts will begin from. MUST be inside the pocket, including the tool radius. NOTE: If left blank, the CNC will pick its own point....It is recommended to leave blank.

StartHgt: The ABSOLUTE position of the Z-axis before beginning to mill the pocket. This MUST BE .1000" (or 2 mm) above the surface to be cut into.

Z-Depth: The ABSOLUTE depth of the pocket.

Angle: The angle of the cut, in reference to 0 degrees (3-o'clock position). This is only necessary if the first element of the profile subroutine is an ARC, or if the operator wishes to change the original path automatically output by the CNC.

X Start: The X starting coordinate of the tool. The tool will move from this point to the X-Y point described above, in a 3-axis ramp move. This is a "ramp-from" point.

Y Start: The Y starting coordinate of the tool. The tool will move from this point to the X-Y point described above, in a 3-axis ramp move. This is a "ramp-from" point.

Stepover: The distance the tool will step-over (width of cut) as it mills out the pocket.

Depth Cut: The depth per pass: If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth-Cut can be programmed to allow two or more passes.

Fin Stock: Amount of stock to leave for a finish pass. The finish pass will be done automatically. In the I-POCK cycle, a perimeter pass is automatically done after the pocket is roughed out. This finish stock amount applies to sides and bottom.

RampFeed: The feedrate at which the tool will "ramp" into the pocket with all 3 axes.

Rough Feed: The feedrate at which the pocket will be "roughed-out".

Fin Feed: The feedrate at which the pocket will be "finished".

Tool #: The tool number from the Tool Page.

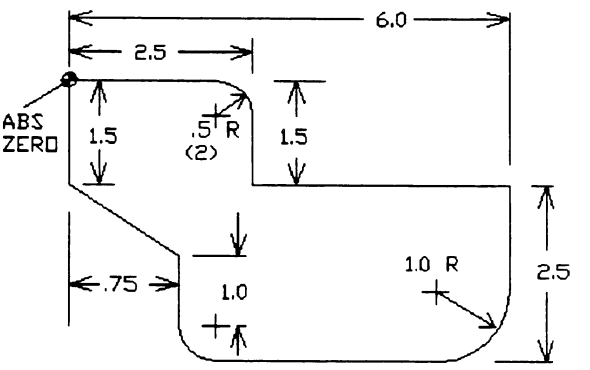
After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main EDIT menu returns to the Soft-Key strip.

6.6.5.1 IRREGULAR POCKET EXAMPLES:

EXAMPLE #1:

An IRREGULAR pocket cycle will be used to cut the pocket shape shown below-right. As mentioned, the "perimeter" of the pocket must be put into a subprogram, so the CNC can automatically calculate the moves necessary to mill out the pocket. Use a 3/8" dia. tool in the Tool Page, for this example. This one part program, will consist of a MAIN program, and a SUBPROGRAM:

```
1     Dim Abs
2     Pocket  Sub#1  StartHgt .1
      ZDepth -.125   XStart 1.0
      YStart -1.0  Stepover .15
      FinStock .01  RampFeed 7.5
      RoughFeed 12.5
      FinFeed 9.5  Tool# 1
3     Rapid  Z0.0  Tool# 0
4     EndMain
5     Sub 1
6     Rapid  X0.0  Y0.0
7     Line  X2.5  CornerRad .5
8     Y-1.5
9     X6.0
10    Dim Incr
11    Line Y-2.5  CornerRad 1.0
12    Dim Abs
13    Line  X.75  CornerRad .5
14    Y-2.5
15    X0.0 Y-1.5
16    Y0.0
17    EndSub
18    <End Of Program>
```

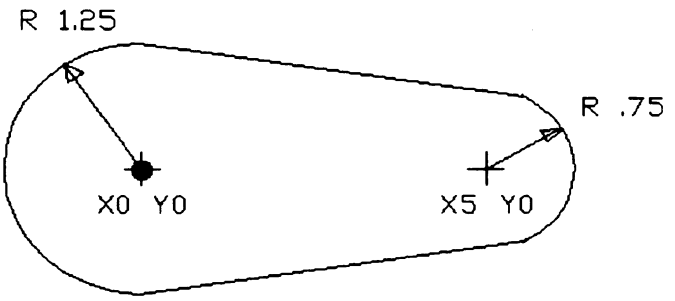


NOTES: 1. If the TOOL DIAMETER is changed in the tool page, the MAIN PROGRAM must be edited to affect the change.
2. Be sure to program a .3750" dia. tool in the tool page for Tool #1, for this and the next example.

EXAMPLE #2:

An IRREGULAR pocket cycle will be used to cut the pocket shape shown below-right. As mentioned, the "perimeter" of the pocket must be put into a subprogram, so the CNC can automatically calculate the moves necessary to mill out the pocket. Use a 3/8" dia. tool in the Tool Page, for this example. This one part program, will consist of a MAIN program, and a SUBPROGRAM:

```
1     Dim Abs
2     Pocket     Sub#  99
      StartHgt .1
      ZDepth -.25
      Angle 89.9
      XStart 0.0
      YStart     0.0
      Stepover .16
      DepthCut .125
      FinStock    .01
      RampFeed    7.5
      RoughFeed 12.5
      FinFeed 9.5
      Tool# 1
3     Rapid  Z0.0  Tool# 0
4     EndMain
5     Sub 99
6     Rapid  X-1.25  Y0.0
7     Arc Cw  X .125  Y 1.2437  Radius 1.25
8     Line  X 5.075  Y .7462
9     Arc Cw  X5.075  Y-.7462  Radius .75
10    Line  X .125  Y -1.2437
11    Arc Cw  X -1.25  Y0.0  Radius 1.25
12    EndSub
13    <End Of Program>
```



NOTE: Be sure to program a .3750" dia. tool in the tool page for Tool #1, for this and the previous example. The CALC mode was used to find the intersections of the lines and arcs shown above. See section 7 for CALCULATOR mode.

6.6.6 NOTES ON POCKET CYCLES

1. As some entries are OPTIONAL, the following shows the "default" (what is set if the operator DOES NOT program the optional entry):

X-Y Center or Start:		The current tool position is used.
Corner Rad	:	The tool radius is used.
Stepover	:	50% of the tool diameter is used.
Depth-Cut	:	The tool diameter is used.
Fin Stock	:	None.
Ramp Feed	:	The previously used feedrate.
Rough Feed	:	The previously used feedrate.
Fin Feed	:	The previously used feedrate.
Tool#	:	The previously used (active) tool #.

1B. Irregular pocket defaults:

X-Y	:	The CNC will pick the best point.
Angle	:	The angle of the first move in the Sub.
XStart-YStart	:	Not used if not programmed.
Finish Stock	:	If none given, no "perimeter" pass will occur.

Depth-Cut, Fin Stock, Ramp-Rough-Fin Feed, Tool #...see above.

2. If a negative (-) finish stock is programmed, NO finish pass will be done. BUT, the finish stock is left in the pocket / hole.
3. The depth per pass entry: If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth-Cut can be programmed to allow two or more passes. EXAMPLE: If a 2" deep pocket is needed, Z-Depth would be -2.0000", and Depth-Cut could be -.5000". This would result in 4 depth cuts (plus a finish pass, if programmed).
4. If finish stock is programmed, the CNC will automatically make the finish cut at the finish feedrate.
5. The SUBPROGRAM for the IRREGULAR pocket cycle must contain the perimeter of the pocket ONLY. ToolComp and Z-moves should NOT be used to define the perimeter, only the ACTUAL perimeter without ToolComp or Z-moves. The first move in the subprogram MUST BE a rapid move to the pocket start point (corner, end, or the easiest place to start).

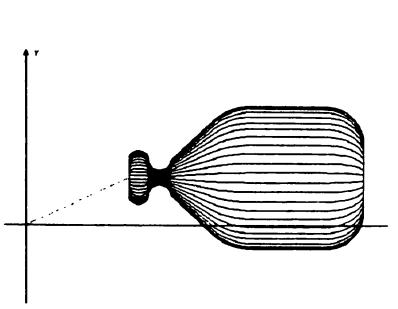
   The perimeter subprogram must be a CLOSED shape... in other words, the START point of the FIRST (rapid) move and the END point of the LAST move (line or arc) must be exactly the SAME.
6. If the TOOL diameter is changed in the TOOL page after running the IRREGULAR pocket, the MAIN program must be edited to affect the change.

6.6.7 MOLD ROTATION

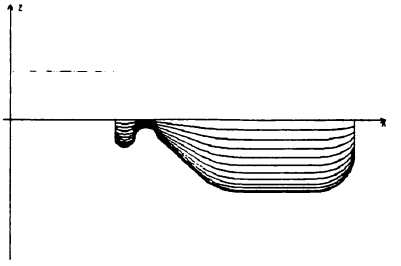
Mold Rotation is one of the advanced functions, or, "canned cycles" in the CNC control. It takes a 2D tool path, and rotates the path into 3D. It can be used to mill concave or convex shapes,

such as mold cavities and mold cores. The mold shape must be symmetrical around its centerline to utilize this feature of the CNC. NOTE: All other aspects of the CNC should be understood (such as Lines, Arcs, CornerRad, and Subprograms) before attempting Mold Rotation.

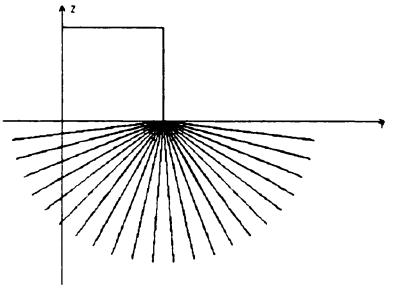
The example used here will be a female (concave/cavity) bottle shape, rotated about the X axis.



XY View



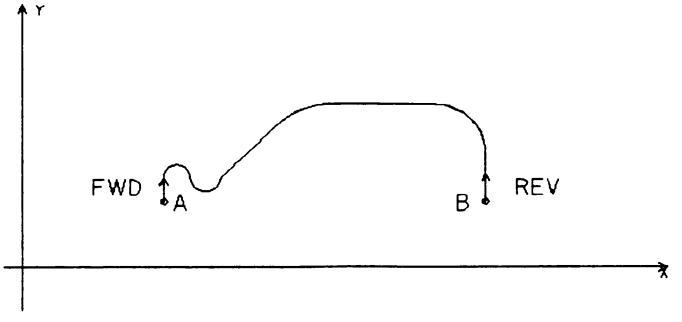
XZ View



YZ View

As with any canned cycle, the variables must be programmed in order for the system to perform the function. With the "Mold Rotation" cycle, additional information must be known, and placed into two (2) separate SUBPROGRAMS:

a) Tool path of 1/2 the perimeter of the shape, from center-line to center-line. See below drawing. Explanation: The tool path from point A to point B must be known. The tool paths must be programmed in incremental (Dim Incr). ToolComp (Left or Right) IS permitted. If tool comp. is NOT used, the tool radius must be added (male/core), or subtracted (female/cavity) from the part edge. Note: The tool path from A to B is put into a subprogram (forward sub). The tool path from B to A is put into a second subprogram (reverse sub). These tool paths are rotated by the Mold Rotation canned cycle around the described centerline/axis (X,Y or Z). If ToolComp is used, it **MUST** be turned on and off in BOTH subprograms.



Tool paths needed

The drawings in this section represent actual tool path. Cutter diameters have been taken into consideration, and no tool comp. is used.

The FORWARD and REVERSE subprograms are mandatory for the MOLD ROTATION canned cycle, as they will be rotated through the included angle programmed.

The tool must be at the XY starting position, and at Z depth. Generally, a ball-end mill is used for mold work. The Tool Length Offset (TLO) should be set so Z0 is at the center of the radius of the ball-end mill.

Inputs:

StartAngle: Usually 0 degrees. This start angle is in reference to the "end view". See above drawing.

EndAngle: Usually 180 or -180 degrees. This end angle is in reference to the "end view". See above drawing.

Cycles: A cycle is equal to a pass in both directions (one FWD cut, one REV cut). This will determine the finish on the piece-part. Usually set to a low number for roughing, then re-programmed to a higher number for finishing.

FwdSub: The number of the FORWARD subprogram. If the path from A to B is programmed in Subprogram #1, then FwdSub is programmed as 1.

RevSub: The number of the REVERSE subprogram. If the path from B to A is programmed in Subprogram 2, then RevSub is programmed as 2.

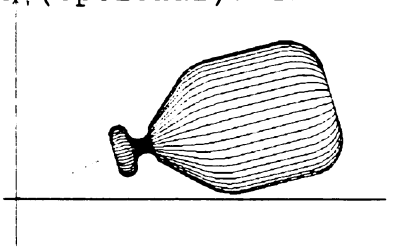
AxisRot: X or Y or Z = The axis of rotation. The above example is rotated about the X axis. If the paths from A to B, B to A were programmed along the Y axis, this variable would be set to Y. No number should accompany this variable.

OtherCL: The center-line of rotation in the "other" axis. Other axis = the axis of X/Y which is NOT the rotation axis described in the above variable (X,Y). Absolute dimension.
EXAMPLE: The XY start position of the above drawing is X2.0 Y1.0. The "OtherCL" value in the example must be set to 1.0. This is the centerline of the shape in the Y (other) axis.

ZaxisCL: Center-line of rotation in the Z axis. Normally 0. If a rough pass is required, the programmer will command Z above zero before initiating Mold Rotation, and then set

ZaxisCL equal to that value.

ZAngle: Angle of rotation around the Z axis (optional). In the above example, this was not used. However, if ZAngle 15.0 was programmed, the shape would be rotated 15 degrees CCW. This angle is in reference to the Polar coordinate system in the (XY) plane, and the origin is the XY start point (X2.0 Y1.0). See drawing at right.



As always, press F10 to SAVE>

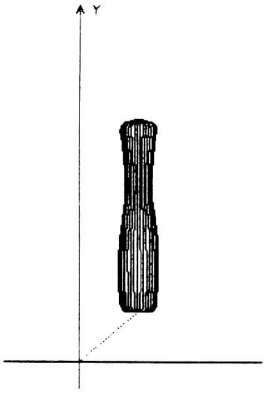
EXAMPLE 1:

The blocks required to cut the shape on the previous pages are:

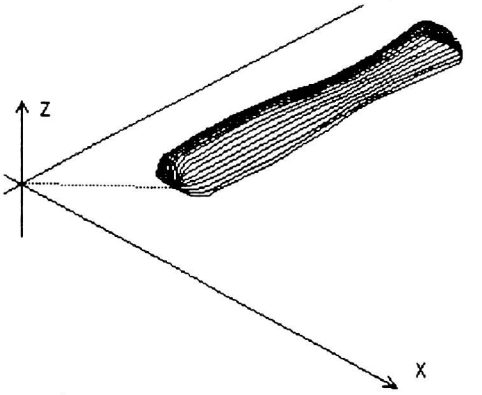
```
1     Dim Abs
2     Rapid  X 2.0  Y 1.0  Tool# 1
3     Z.50
4     Line  Z 0.0  Feed 10.0
5     Mold Rot  StartAngle 0.0  EndAngle -180.0  Cycles 12
      FwdSub 1  RevSub 2  AxisRot X  OtherCL 1.0
6     Rapid  Z0.0  Tool# 0
7     EndMain
8     Sub 1
9     Dim Incr
10    Line  Y .375
11    Arc Cw  X .375  Y 0.0  Radius .1875
12    Arc Ccw  X .4375  Y 0.0  Radius .2188
13    Line  X .75  Y .75
14    Arc Cw  X .75  Y .375  Radius 1.125
15    Line  X 2.25  CornerRad .75
16    Line  Y -1.5
17    EndSub
18    Sub 2
19    Line  Y 1.5  CornerRad .75
20    Line  X -2.25
21    Arc Ccw  X -.75  Y -.375  Radius 1.125
22    Line  X -.75  Y -.75
23    Arc Cw  X -.4375  Y0.0  Radius .2188
24    Arc Ccw  X -.375  Y0.0  Radius .1875
25    Line  Y -.375
26    EndSub
27    <End Of Program>
```

The blocks required to cut the shape immediately above are the same, except (ZAngle 15) is added to the Mold Rot block.

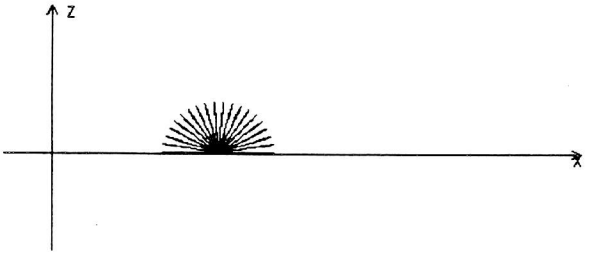
EXAMPLE 2: "Handle" mold core (male), using Mold Rotation, around Y axis.



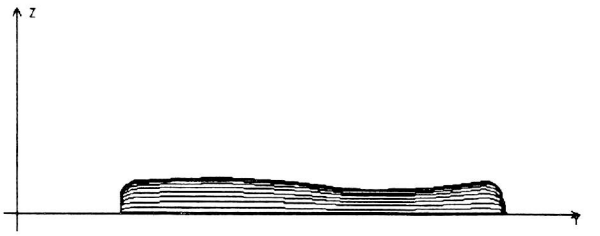
XY View



Iso View



XZ View



YZ View

Main and Sub Programs, for above HANDLE mold core:

```
1     Dim Abs
2     Rapid  X 1.625  Y 1.5  Tool# 1
3     Z.50
4     Line  Z 0.0  Feed 10.0
5     Mold Rot  StartAngle 0.0  EndAngle 180.0  Cycles 10
      FwdSub 15 RevSub 16  AxisRot Y  OtherCL 1.625
6     Rapid  Z0.0  Tool# 0
7     EndMain
8     Sub 15
9     Dim Incr
10    Line  X -.25
11    Arc Cw  X -.25  Y .27  Radius .25
12    Arc Cw  X .125  Y 3.0  Radius 12.0
13    Arc Ccw  X -.125  Y 2.0  Radius 9.0
14    Arc Cw  X .25  Y .25  Radius .25
15    Arc Cw  X .25  Y .0625  Radius 3.0
```

```
16    EndSub
17    Sub 16
18    Arc Ccw  X -.25  Y -.0625  Radius 3.0
19    Arc Ccw  X -.25  Y -.25  Radius .25
20    Arc Cw  X .125  Y -2.0  Radius 9.0
21    Arc Ccw  X -.125  Y -3.0  Radius 12.0
22    Arc Ccw  X .25  Y -.27  Radius .25
23    Line  X .25
24    EndSub
25    <End Of Program>
```

Notes on Mold Rotation cycle:

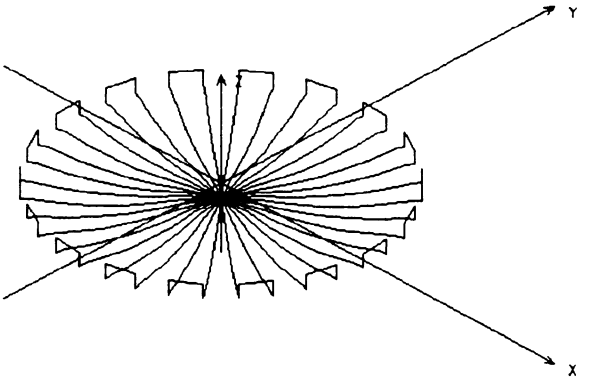
1) The MOLD ROTATION cycle requires two (2) subprograms: One of a "forward" tool path, and a second subprogram of the "reverse" path. The subprograms MUST BE INCREMENTAL programs.
2) ToolComp IS permitted. If NOT USED, the tool path must be calculated: ADD tool radius for MALE (core) ; SUBTRACT tool radius for FEMALE (cavity). IF USED, both the ToolComp-on and ToolComp-off codes must be included in each subprogram.
3) By altering (start and finish angles), a section of the core or cavity may be cut.
4) Generally, the shapes will be roughed out first, then finished. To accomplish this, leave the tool above Z0 for the rough pass (i.e; Z.015), and program a small number of cycles (i.e; 10), and program ZAxisCL to equal the amount above Z zero that Z is positioned (i.e; ZAxisCL .015). THEN, to finish, lower Z axis to Z0, and reprogram the MOLD-ROT cycle with a larger (cycle) number, and set ZAxisCL to 0.
5) ZAxisCL = centerline of rotation (Z dimension of centerline) for the Z axis. Generally, for female shapes is set to 0, and for male shapes is set down to the Z centerline of the shape.
6) If optional ZAngle (angle about Z axis) is used, the origin of rotation will be the XY start point of the MOLD-ROT cycle.
7) The tool path programmed (inside or outside) from the part edge, combined with the finish angle (+ or -), will determine if the shape is a core or cavity.

EXAMPLE 3: Mold Rotation around Z axis.

In this example, a dish shape will be cut. The FWD and REV tool paths are programmed in the XZ plane, with ToolComp.

MAIN and SUBPROGRAMS:

```
1     Dim Abs
2     Rapid  X 0.0  Y 0.0  Tool# 1
3     Z .50
4     Line  Z 0.0  Feed 5.0
5     MoldRot  StartAngle 0.0
      EndAngle 350.0  Cycles 9
      FwdSub 1  RevSub 2
      AxisRot Z  Feed 5.7
6     Rapid  Z0.0  Tool# 0
7     Plane  XY
8     EndMain
9     Sub 1
10    Plane XZ (see section 6.7.5)
11    Dim Incr
12    Line  Z -.25  ToolComp Right
13    Line  X .50
14    Arc Cw  X 2.0  Z .125  Rad 5.0
15    Line  X .125
16    Line  Z .25  ToolComp Off
17    EndSub
18    Sub 2
19    Line  Z -.25  ToolComp Left
20    Line  X -.125
21    Arc Ccw  X -2.0  Z -.125  Radius 5.0
22    Line  X -.50
23    Line  Z .25  ToolComp Off
24    EndSub
25    <End Of Program>
```



6.6.8 ELBOW MILLING

The Elbow canned cycle allows an "elbow" shape to be programmed. The shape can be conical (different radius at each end of the elbow). It can be male (concave), or female (convex). (If no "bend" of the elbow shape is required, i.e; straight cylinder, the operator should use Mold Rotation-described above).

The ELBOW cycle can be used to cut a complex radial conical shape using only one block of program. See below drawing.

EXAMPLE: The following information is known about the shape in the drawing below:

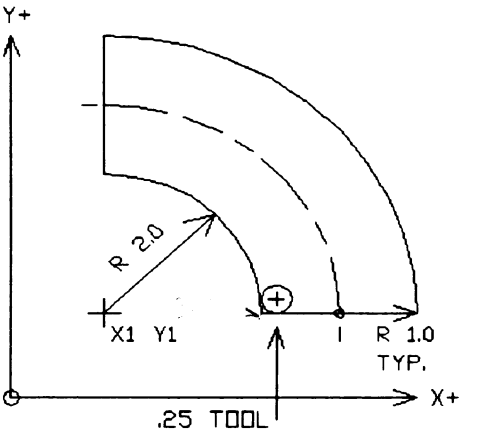
```
Radius of ends                          = 1.0
Radius of inside dia. of elbow          = 2.0
Origin of elbow radius                  = X1.0 Y1.0
Included angle (end-to-end) of shape  = 90 degrees
Ball-end mill diameter to be  used = .2500"
```

With the above information (shown in drawing below), the below

"elbow" cavity can be milled.

EXAMPLE:

```
1     Dim Abs
2     Rapid  X 3.125  Y 1.125
      Tool# 1
3     Z .50
4     Line  Z 0.0  Feed 6.5
5     Elbow  Ccw  SrartRad 1.0
      EndRad 1.0  InclAngle  90.0
      Cycles 10.0  XCenter 1.0
      YCenter 1.0  Feed 25.5
6     Rapid  Z 0.0  Tool# 0
7     <End Of Program>
```



Format:

Direction: The direction of the included angle described below. Either Cw or Ccw (toggle with +/- key).

StartRad: The Radius of the inside (cross-section) of the cavity itself (shown as 1.0" Typ. in above drawing). This is for the tool-start end of the elbow.

EndRad: The Radius of the inside (cross-section) of the cavity itself (shown as 1.0" Typ. in above drawing). This is for the OTHER end of the elbow. NOTE: If the StartRad and EndRad are different, a CONICAL elbow will occur.

InclAngle: The total included angle of the elbow cavity. Tool must be taken into consideration.

Cycles: The number of passes in each direction (back and forth, lengthwise along the elbow cavity). Ten (10) cycles will result in 20 passes.

XCenter: The origin of the major radius of the Elbow, for X axis (Shown as 1.0 in above example).

YCenter: The origin of the major radius of the Elbow, for Y axis (Shown as 1.0 in above example).

Feed: The feedrate at which the tool will cut.

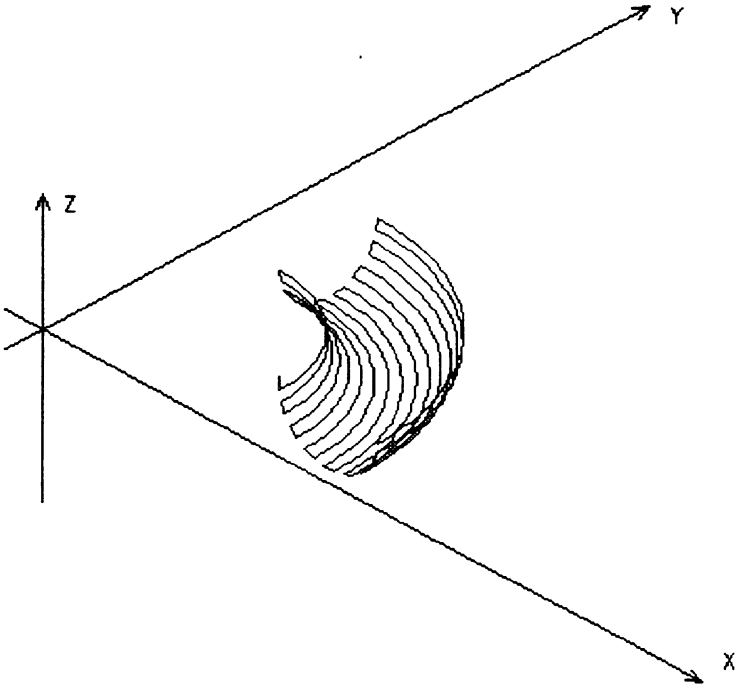
Tool#: The tool # to activate prior to cutting the cavity.

As always, press F10 to SAVE.

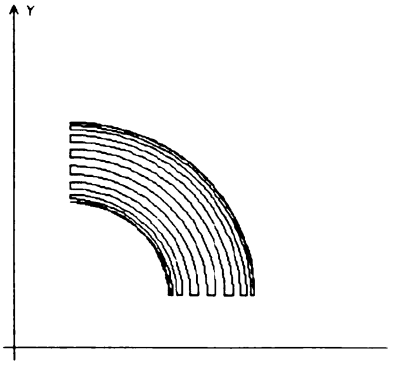
NOTE: The XY tool starting position must be calculated. The Geometry Calculator will come in handy here (see section 7). In the above case, no tool radius was taken into consideration.

The tool must be positioned at the XY start point, and be at the required depth. A ball-end mill should be used, and the TLO should be to the center of the ball-end mill radius.

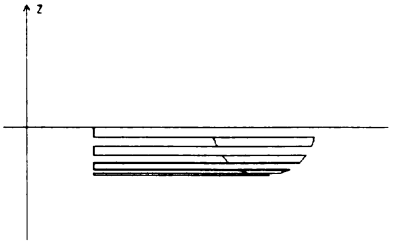
The drawings below show the 4 views the example above will create:



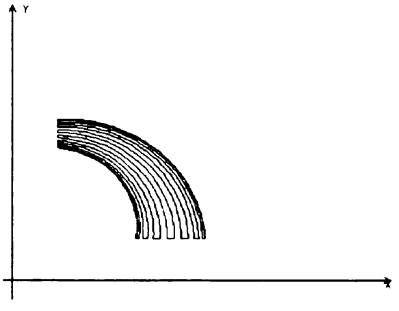
Iso View

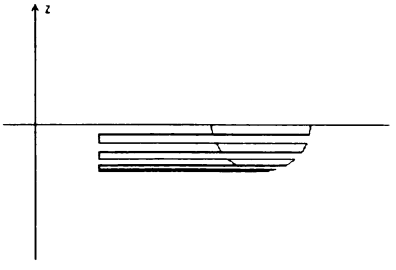


XY View



XZ View





YZ View

Conical Elbow

The EndRad for the elbow can be different than the StartRad. This would produce the "conical" elbow shown above-left. The above program, with EndRad programmed to be .375 will produce an elbow with a cone shape.

MORE Notes on Elbow canned cycle:

1) The Elbow cycle allows an "elbow" cavity or core to be cut.
2) ToolComp is not permitted. Tool start position must be calculated.
3) By altering StartRad and EndRad, a conical elbow core or cavity may be cut.
4) Generally, the shapes will be roughed out first, then finished. To accomplish this, leave the tool ABOVE Z0 for the rough pass (i.e; Z.015), and program a small number of cycles. THEN, to finish, lower Z axis to Z0, and reprogram the Elbow cycle with a larger cycle number.
5) A positive CYCLE # word will produce a female cavity. Negative CYCLE # will produce male core.
6) InclAngle can be positive (CCW) or negative (CW).
7) If optional (elbow origin) is NOT used, the origin of the elbow radius will be Absolute XY zero.

6.7 MILL - F5

MILL

F5 (Mill) brings the MILL menu to the **Soft-Key Strip**. It is here that the operator picks what type of move is needed, whether a rapid positioning move, a line, or an arc.

Other functions such as Corner-Round and Calculator with Recall are present while programming Rapids, Lines or Arcs. Additional line and arc formats are also found, under the MORE Soft-Key. See section 6.9.2 below for MORE Rapid, Line and Arc formats such as radius-angle, end-point/center-point, etc..

NOTE:

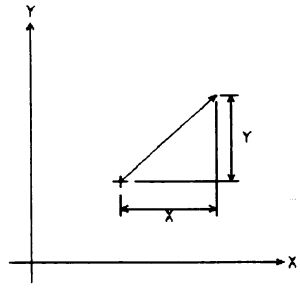
During the entry of data such as pockets, lines and arcs, you will notice that some entries are initially filled with zeroes --- this means that you **MUST** fill in that piece of data. Any entry that is blank is optional, in other words, you do NOT have to fill in that data, only if you deem necessary.

6.7.1 RAPID : F2

While in the MILL MENU, Soft-Key F2 is RAPID. When pressed,

this key allows the operator to program a rapid positioning move. Rapid moves are usually used to quickly get the machine to a starting position, so Z can then be moved down, and a LINE or ARC can be programmed to start a contouring cut. See section 6.9.2 for MORE rapid formats.

Inputs:



X: The X coordinate end-point.

Y: The Y coordinate end-point.

Z: The Z coordinate end-point (above X/Y/Z end-point may be ABS or INC).

Tool Comp: Tool Compensation (Left, Right, Off or None). See section 3.5 above for Tool Comp.

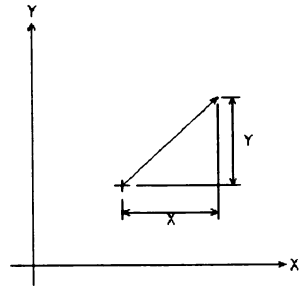
Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the MILL MENU returns to the Soft-Key strip. Soft-Key F9 (Prev) will bring the PREVIOUS Soft-Key strip to the screen, the MAIN edit MENU.

6.7.2 LINE : F3

While in the MILL MENU, Soft-Key F3 is LINE. When pressed, this key allows the operator to program a linear feed move. Line moves are usually used to mill-cut a straight line or angle, and to get to the starting position of an ARC, during a contouring cut. See section 6.9.2 for MORE line formats.

Inputs:



X: The X coordinate end-point.

Y: The Y coordinate end-point.

Z: The Z coordinate end-point (above X/Y/Z end-point may be ABS or INC).

Corner Rad: The blend radius between this line and the next line or arc. See section 3.6 above.

Tool Comp: Tool Compensation (Left, Right, Off or None). See section 3.5 above for Tool Comp.

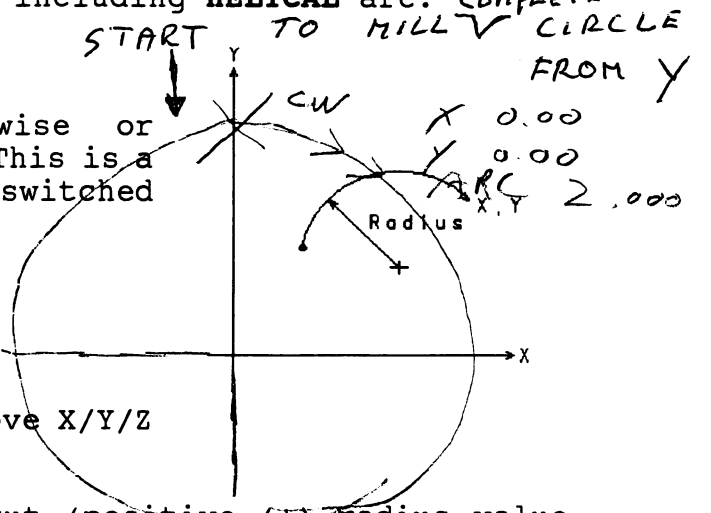
Feed: The feedrate at which the tool will move along the line.

Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the MILL MENU returns to the Soft-Key strip. Soft-Key F9 (Prev) will bring the PREVIOUS Soft-Key strip to the screen, the MAIN edit MENU.

6.7.3 ARC : F4

While in the MILL MENU, Soft-Key F4 is ARC. When pressed, this key allows the operator to program a circular feed move, either a full circle or a partial circle (arc). Arc moves are used to mill-cut radii on a contour, holes, or anything requiring circular movement (remember the circular pocket and hole pocket canned cycles can shorten programming time and effort in some cases). See section 6.9.2 for MORE arc formats, including **HELICAL** arc.



Inputs:

Direction: CW or CCW (clockwise or counter-clockwise). This is a "toggle", and is switched with the +/- key.

X: The X coordinate end-point.

Y: The Y coordinate end-point.

Z: The Z coordinate end-point (above X/Y/Z end-point may be ABS or INC).

Radius: The actual radius to be cut (positive (+) radius value produces an arc of LESS THAN 180 degrees; negative (-) radius value produces an arc of MORE THAN 180 degrees).

Corner Rad: The blend radius between this arc and the next arc or line. See section 3.6 above.

Feedrate: The feedrate at which the tool will move along the arc.

Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the MILL MENU returns to

the Soft-Key strip. Soft-Key F9 (Prev) will bring the PREVIOUS Soft-Key strip to the screen, the MAIN edit MENU.

6.7.4 MORE milling functions : F6

While in the MILL MENU, Soft-Key F7 is MORE. When pressed, a window appears on the CRT-screen with more milling functions. Included here are FEED, PLANE, UNIT, SET-ZERO, HOME, ELLIPSE and SPIRAL functions.

ELLIPSE and SPIRAL are cutting functions, where-as SET-ZERO and UNIT are functions designed to make programming easier.

6.7.4.1 FEED

While in the MORE window of the MILL menu, the first selection is FEED. This function allows a feedrate to be set on a block by itself. It is a secondary way to set feedrates...remember, in the LINE and ARC menus, there is a FEEDRATE entry in each.

6.7.4.2 PLANE

While in the MORE window of the MILL menu, the second selection is PLANE. This is used when it is necessary to program an arc in one of the "other" planes. The plane selection is toggled with the +/- key on the CNC keyboard. A PLANE consists of any two axes: X-Y, X-Z, and Y-Z. The Series 1100 is capable of doing arcs in any of those planes. Typically, with flat work, only arcs and lines in the X-Y plane are necessary.

Before programming an ARC in the X-Z or Y-Z plane, the "PLANE" must be set, otherwise, the control will give an error message. After switching to another plane, the ARC commands will prompt the operator with the appropriate axes (such as X and Z, or Y and Z).

REMEMBER... be sure to switch back to the X-Y plane at the end of the program.

6.7.4.3 UNIT

While in the MORE window of the MILL menu, the third selection is UNIT. This is used to switch the programming format from INCH to MILLIMETER (metric), or vice-versa. If a blueprint is dimensioned in millimeter, this function can be used to avoid the programmer / operator from manually converting the dimensions. After use, the INCH mode should be set, in the NEXT program.

The selection is toggled with the +/- key on the CNC keyboard.

6.7.4.4 SET-ZERO

While in the MORE window of the MILL menu, the fourth selection is SET-ZERO. This function allows the PART-ZERO reference point (Absolute Zero) to be shifted or switched from one point to another on the machine table.

This function could be used if more than one part is to be

machined with the same program.

Inputs:

X: The coordinate to set the current X axis position to.

Y: The coordinate to set the current Y axis position to.

Z: The coordinate to set the current Z axis position to.

See section 4.4 above for the concept of ABSOLUTE ZERO.

EXAMPLE: Two (2) vises are mounted on the machine table, each holding an identical part. All the data for Tool #1 should be put in a subprogram, say, SUB 1. All the data for Tool #2 should be put into SUB 2.

Then, in the main program, the operator CALLS SUB 1, does an "set-zero" (to the location of the second vise), then CALLS SUB 1 again. The operator **MUST** then re-SET-ZERO back to the original workpiece. The same process will apply for Tool #2, 3, 4, etc.. See section 6.10 below for SUBPROGRAMS.

6.7.4.5 HOME

While in the MORE window of the MILL menu, the next selection is HOME. This function allows the machine to be referenced with what is known as a "Machine-Home" function.

This function is used to move the machine to a FIXED, KNOWN reference point, then zero-out the axes. It is commonly used on larger machines to "reference" the axes. From this fixed, known point, SET-ZERO commands may be given to switch the zero point from HOME to a new PART-ZERO location. In this way, if a part-zero is accidentally lost, the operator can perform a HOME function, and return to the PART-ZERO without re-indicating the workpiece.

Inputs:

X: The coordinate to rapid-move the X axis to, before the HOME function begins. The HOMING will occur from this point (if no coordinate is given, toggle the +/- key to highlight the X axis for homing...this way, the homing will take place from the current position).

Y: The coordinate to rapid-move the Y axis to, before the HOME function begins. The HOMING will occur from this point (if no coordinate is given, toggle the +/- key to highlight the Y axis for homing...this way, the homing will take place from the current position).

Z: The coordinate to rapid-move the Z axis to, before the HOME function begins. The HOMING will occur from this point (if no coordinate is given, toggle the +/- key to highlight the Z axis for homing...this way, the homing will take place from the current position).

HOME is a quick way to send the machine to the "home" position (a position where the machine axes are referenced from). This position is usually at the extreme positive end-of-travel for the X, Y and Z axes.

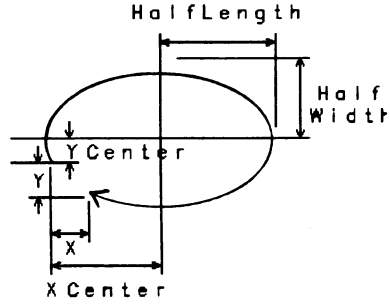
NOTE: The machine should be equipped with "vector limit switches" for this function to be available. DO NOT use the HOME function without the proper type of limit switches. No damage can occur, however.

The HOME function, in combination with the SET-ZERO function described in section 6.8.1 above, can be useful on larger machine tool applications.

6.7.4.6 ELLIPSE

While in the MORE window of the MILL menu, the next selection is ELLIPSE. This "canned-cycle" function allows the operator to cut an elliptical shape. An ellipse is a symmetrically out-of-round circle. NOTE that some dimensions may be ABS or INC, depending on the ABS/INC status of the CNC.

Inputs:



Direction: CCW or CW.

X: The X end-point coordinate (ABS, or distance from start to end if in INCR).

Y: The Y end-point coordinate (ABS, or distance from start to end if in INCR).

Z: The Z coordinate end-point (ABS, or distance from start to end if in INCR).

X-Center: The coordinate of the X axis center of the ellipse (if in ABS), or, the INCREMENTAL distance from the X START to the X CENTER point of the ellipse (if in INC).

Y-Center: The coordinate of the Y axis center of the ellipse (if in ABS), or, the INCREMENTAL distance from the Y START to the Y CENTER point of the ellipse (if in INC).

Half-Length: The dimension of 1/2 of the full LENGTH of the

ellipse.

Half-Width: The dimension of 1/2 of the full WIDTH of the ellipse.

Feed: The feedrate at which the ellipse will be milled.

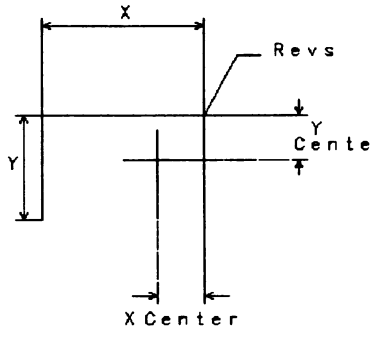
Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main MILL menu returns to the Soft-Key strip.

6.7.4.7 SPIRAL

While in the MORE window of the MILL menu, the last selection is SPIRAL. This "canned-cycle" function allows the operator to cut a spiral. A spiral is a arc with an ever-changing radius. A spiral could be used to rough-out a hole, and in fact is combined with an arc to produce the "circular pocket" canned cycle described in section 6.6.2 above. NOTE that some dimensions may be ABS or INC, depending on the ABS/INC status of the CNC.

Inputs:



Direction: CCW or CW.

X: The X end-point coordinate (ABS, or distance from start to end if in INCR).

Y: The Y end-point coordinate (ABS, or distance from start to end if in INCR).

Z: The Z coordinate end-point (ABS, or distance from start to end if in INCR).

X-Center: The coordinate of the X axis center of the spiral (if in ABS), or, the INCREMENTAL distance from the X START to the X CENTER point of the spiral (if in INC).

Y-Center: The coordinate of the Y axis center of the spiral (if in ABS), or, the INCREMENTAL distance from the Y START to the Y CENTER point of the spiral (if in INC).

Revs: The number of complete revolutions (a line drawn from the center-point through the start point is considered the "line to cross" when counting revolutions).

Feed: The feedrate at which the ellipse will be milled.

Tool #: The tool number from the Tool Page.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the main MILL menu returns to the Soft-Key strip.

6.7.5 PREV

While in the MILL MENU, Soft-Key F9 is PREV. When pressed, this key will switch the CRT-screen to its previous screen, which is the MAIN EDIT soft-key strip.

6.8 TOOL# - F6

While in the MAIN EDIT menu soft-key strip, F6 is TOOL#. This key allows the operator to activate or de-activate a TOOL OFFSET by simply programming the tool number (#).

The information on the tool# programmed is taken from the TOOL PAGE. Note that tool# also appears in many of the other menus, such as Rapid, Line, Pockets, etc.. The function is the same...to activate or de-activate a tool offset.

When pressed, F6 (Tool#) will prompt the operator with a TOOL#. The operator simply enters the number (0 through 99), then press F10 (or ENTER). The block is put in the program, the cursor advances, and the MAIN EDIT menu returns to the soft-key strip.

6.9 RECALL, MORE, CALC, CANCEL and SAVE SOFT-KEYS

The Soft-Key strip of the RAPID, LINE, ARC, HOME, SET-ZERO, ELLIPSE and SPIRAL screens allow the operator to access the CALCULATOR modes, as well as RECALL values previously calculated.

The Soft-Key "MORE..." also appears on the screen, as does SAVE. Each Soft-Key is briefly explained below, and may refer you to the section of this manual that explains the function of the key in full detail.

6.9.1 RECALL

F2 (Recall) allows values previously calculated and STORED in any CALC mode to be "recalled" into area(s) of the input screens for RAPID, LINE, ARC, HOME and SET-ZERO.

See section 7.5 below for RECALL function.

6.9.2 MORE

MORE...

F4 (More) allows additional RAPID, LINE and ARC formats to be selected from. For example, the standard format for RAPID and LINE is a simple "X-Y end point" format. If the X-Y end point is not known, the operator has two choices: (a) go to CALC mode and calculate it .... or (b) select another format.

Depending on the information given on the blueprint, the operator may wish to define the end-point in an ANGLE-RADIUS format, rather than an X-Y format. the Soft-Key MORE allows this. Whether selecting MORE from Rapid, Line or Arc screens, the MORE selection will automatically do a TRIG function during the program.

Described below are the additional rapid/line and arc formats available to the operator. They are accessed by pressing F4 (More) while in the RAPID, LINE or ARC menus. The left-right ARROW keys may be used to move to the desired selection, then ENTER must be pressed to select that format.

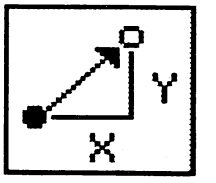
NOTES:

1. When another Rapid/Line/Arc format (other than the original one) is selected, it will only be used once. The next time Rapid/Line/Arc is selected, it will automatically use the original (default) format. The FIRST format shown under MORE is the default format.
2. ABS and INC programming has an effect on the following formats. Remember... ABSOLUTE dimensions are in reference to the Part-Zero. INCREMENTAL dimensions are in reference to the Current tool position.

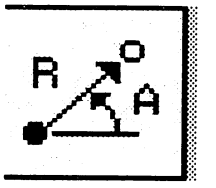
see next page.....

RAPID and LINE ADDITIONAL FORMATS:

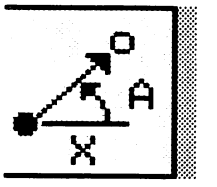
6.9.2.1 X and Y
Default format. End-Point given as X and Y dimension.



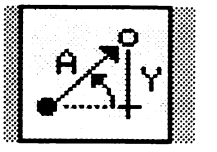
6.9.2.2 RADIUS and ANGLE
End-Point given as radius and angle. Angle is in reference to 0 degrees (3-o'clock position), and "radius" is the length of the line (hypotenuse).



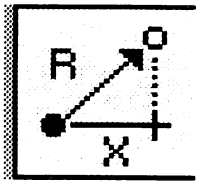
6.9.2.3 X and ANGLE
End-Point is given as X value and angle. This will automatically calculate the Y dimension (side opposite).



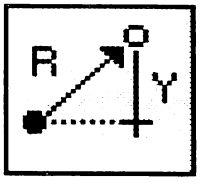
6.9.2.4 Y and ANGLE
End-Point given as Y value and angle. This will automatically calculate the X dimension (side adjacent).



6.9.2.5 X and RADIUS
End-Point given as X value and radius (hypotenuse). This will automatically calculate the Y dimension (side opposite).



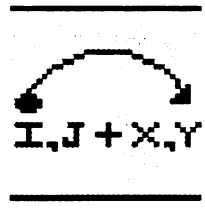
6.9.2.6 Y and RADIUS
End-Point given as Y value and radius (hypotenuse). This will automatically calculate the X dimension (side adjacent).



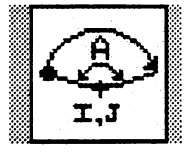
ARC ADDITIONAL FORMATS:

6.9.2.7 X, Y and RADIUS
Default format. Arc given as Direction, X-Y End-Point, and Radius values.

6.9.2.8 X, Y and CENTER (XY)
Arc given as Direction, X-Y End-Point, and X-Y Center-Point. **NOTE:** This format also allows a **HELICAL ARC** to be programmed (use for thread milling, etc.). For Helical, use the "Revs" variable for number of complete revolutions. The X-Y axes will continue to arc around the # of Revs, while the Z axis achieves position simultaneously.



6.9.2.9 CENTER (XY) and ANGLE
Arc given as Direction, X-Y Center-Point, and Angle (from 0 degrees if ABS, from previous angle if INCR).



6.9.3 CALC

CALC

F7 (Calc) allows the operator to access any one of the three (3) CALC modes. When pressed, a window will appear on the screen with the prompt "Select Type of Calculator:"

1. Standard Calculator (+,-,*,/, and TRIG functions).
2. Right-Triangle Calculator (find sides or angles).
3. Geometry Calculator (build points, lines, circles).

The left-right ARROW keys may be used to move the cursor (dark background) to the desired CALCULATOR. When ENTER is pressed, the chosen calculator will become active.

See section 7 below for complete details on CALC modes.

6.9.4 CANCEL

CANCEL

F9 (Cancel), ANYTIME it is shown, allows the operator to CANCEL the programming step he/she is working on. F9 will simply discard any partial data input (on the top half of the screen) so far, and return the CRT to the previous screen. No other data will be lost.

6.9.5 SAVE

SAVE

F10 (Save) will SAVE (put) the block currently being worked on, into the program. When SAVE is pressed, either the MAIN or MILL Soft-Key menu will re-appear on the CRT - screen, depending on which one was being used.

SAVE is how you store "blocks" (program steps) into the part program. Blocks are numbered automatically as they are input.
Note: Pressing ENTER with the cursor on the LAST item of a screen will do the same as SAVE does.

See section 6.15 below for Inserting, Deleting, and Re-Editing blocks in a program.

6.10 SUBPROGRAMS

SUB

F8 (Sub) while in the MAIN edit Soft-Key strip allows the programming of SUBPROGRAMS (SUB), the CALL of subs, the END of subs, the LOOP function, and the ROTATE-MIRROR-SCALE functions. It also allows a DWELL and M-CODE to be programmed.
When F8 is pressed, the SUB Soft-Key strip appears.

6.10.1 WHAT IS A SUBPROGRAM ?

A SUBPROGRAM is a "program within another program". Subprograms (referred to as SUB) are time-saving tools for programming. They allow a sequence of blocks to be used (or CALLED) several times throughout the MAIN program, while the operator only has to program the SUB one time. In this way, time and effort are greatly reduced. Up to 70 subprograms are allowed with a single MAIN program.

6.10.2 CREATING and USING SUBPROGRAMS

For EXAMPLE: A workpiece must be center-drilled, drilled, then counterbored. As you know, each of the 3 tools must go to the same hole positions. If there are 10 hole positions, that would mean programming 30 hole locations (10 for each tool). By using the **SUB**, the 10 hole locations can be put into the SUB, and the SUB can be **CALLED** 3 separate times from the MAIN PROGRAM, once during each Tool. In this manner, the hole locations must be programmed ONLY ONE TIME. All Subs must be input AFTER the end of the MAIN program.

Another good use for SUB: If the outside contour of a workpiece must be roughed out with a roughing mill, then finished with a finishing mill, the contour should be put into a SUB. Then, in the MAIN program, the SUB can be called twice, once for each tool. The TOOL DIAMETER can be set to .5300" for the 1/2" roughing mill, and set to .5000" for the 1/2" finishing mill. Tool #1 will leave .0150" per side, for Tool #2 to clean-up to size.

STRUCTURE OF MAIN and SUB PROGRAMS:

Each MAIN program and SUB program must have a BEGINNING and an END:

```
MAIN PROGRAM:    1    Dim Abs
                 2    Rapid    X5.0000   Y-5.0000
                 3    Call     1
                 4    Rapid    X6.0000   Y-6.0000
                 5    Call     1
                 6    Rapid    X7.0000   Y-5.0000
                 7    Call     1
END MAIN PROG.   8    EndMain
SUB PROGRAM:     9    Sub      1
                10    Z -.0625
                11    Dim Incr
                12    Line     X.375     Feed 2.3
                13    Line     Y.375
                14    Line     X-.375
                15    Line     Y-.375
                16    Dim Abs
                17    Z .1000
END SUB PROG.   18    EndSub
```

Notice how the MAIN program begins at block 1, and runs through block 8. The SUB program begins at block 9, and runs through block 18.

When the MAIN program reaches block 3, the CNC will automatically "jump" to block 9, execute the SUB, and when it reaches block 18, "jump" back to the MAIN program to block 4.

The maximum number of subprograms in a program is 70.

When Soft-Key F8 (SUB) is pressed from the MAIN edit screen, the SUB soft-key strip appears:

F2: SUB: Used to BEGIN a SUB. Each sub must have a beginning. When pressed, the operator must give the desired subprogram number (from 1 to 9999). It is best to start with 1 and if other subs are necessary, use 2,3,4,5, etc..

F3: CALL: Used to CALL (execute) a SUB while in the MAIN program. One sub can be called from another sub (known as "nesting". Nesting is allowed 10 levels deep). When pressed, the operator must give the desired subprogram number (from 1 to 9999) to be called.

F4: END: Used to END a SUB. Each sub must have an end. Any MAIN program that uses subprograms must also have an END. When pressed, the operator may choose EndSub or EndMain (to END the SUB program, or to END the MAIN program). Use the ARROW up-down keys to move onto the desired entry, and press ENTER. The block will go into the program.

F5: LOOP: Used to LOOP (repeat) a subprogram. See section 6.11 below for LOOP function.

F6: RMS: Used to ROTATE, MIRROR, and/or SCALE a subprogram. See section 6.12 below for RMS functions.

F7: DWELL: Allows a timed dwell (or infinite dwell) to be input into the program. Time is in seconds. Infinite dwell is DWELL 0.0 - the program will stop until START key is pressed.

F8: M-CODE: Allows a miscellaneous machine function (such as spindle off, an indexer, work light, clamp, etc.) to be activated through the program. This must be pre-wired and set by the machine builder. Check your machine manual for available M-Codes and their functions. Typical M-CODES are: MCode 3 = Spindle On Fwd, MCode 4 = Spindle On Rev, MCode 5 = Spindle Off, MCode 8 = Coolant On, MCode 9 = Coolant Off.

F9: PREV: Used to change the CRT - screen to the PREVIOUS screen.

6.11 LOOPS (REPEAT FUNCTION)

SUB

F5 (Loop) while in the SUB Soft-Key strip allows the operator to LOOP (or repeat) a SUBPROGRAM. When pressed, the LOOP key will prompt the operator for the following information:

Only SUB programs may be looped.

Inputs:

Sub #: The number of the SUB program to be looped.

Loops: The number of TIMES to loop the sub.

X Incr: The incremental DISTANCE along the X axis to move, inbetween each LOOP of the sub.

Y Incr: The incremental DISTANCE along the Y axis to move, inbetween each LOOP of the sub.

Tool #: The tool number from the Tool Page.

See section 6.16.5 below for a program example of LOOP.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the SUB MENU returns to the Soft-Key strip. Soft-Key F9 (Prev) will bring the PREVIOUS Soft-Key strip to the screen, the MAIN edit MENU.

6.12 ROTATE, MIRROR, and SCALE

RMS

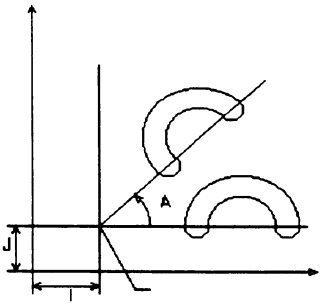
F6 (RMS) while in the SUB Soft-Key strip allows the operator to ROTATE, and/or MIRROR IMAGE, and/or SCALE a subprogram. All of these functions are grouped together on one input screen, for simplicity. Any ONE or ALL THREE of the functions may be used at the same time. When pressed, the RMS key will prompt the operator for the following information:

Only SUB programs may be rotated, mirrored and/or scaled.

Inputs:

Sub #: The number of the SUB program to be ROTATED, and/or MIRRORED, and/or SCALED.

Loops: The number of TIMES to loop the sub, used for ROTATE only. This works along with ANGLE (below) to set up an automatic loop that increments at a specified angle (instead of an increment of X-Y, as in a normal LOOP).



StartAngle: The FIRST angular position to execute the SUB at, such as 0 degrees, or 30 degrees, etc. This allows the subprogram to be initially programmed at zero degrees (3 o'clock position, usually the easiest location to program), but executed at another angular location. Used for ROTATE only.

Angle: The incremental ANGLE between loops. Used for ROTATE only.

X Center: The origin of the rotation along the X axis (used for ROTATE only).

Y Center: The origin of the rotation along the Y axis (used for ROTATE only).

Mirror X: Use the +/- key to toggle the entry to YES or NO, for MIRROR image on the X axis.

Mirror Y: Use the +/- key to toggle the entry to YES or NO, for MIRROR image on the Y axis.

X Scale: The scaling FACTOR for the X axis (i.e; 2 for twice, .5 for half).

Y Scale: The scaling FACTOR for the Y axis (i.e; 2.0 for twice, .5 for half).

Tool #: The tool number from the Tool Page.

See section 6.16.6 below for a program example of ROTATE.

After the data is keyed in, press **F10** to "save" the block. When F10 (save) is pressed, the block is input into the program, the cursor advances to the next block, and the SUB MENU returns to the Soft-Key strip. Soft-Key F9 (Prev) will bring the PREVIOUS Soft-Key strip to the screen, the MAIN edit MENU.

NOTES:

1. Rotate, Mirror Image, and/or Scaling will be automatically "turned off" when the subprogram is finished.

2. If ARCS are included in the SUB to be SCALED, the scaling FACTOR MUST BE EQUAL for the X and Y axes.

6.13 MISC - F9

MISC

F9 (MISC) while in the MAIN EDIT Soft-Key strip allows the operator to perform miscellaneous editing functions on the program he/she is working with. Included under MISC are; COMMENT, SEARCH, PAGE-UP, PAGE-DOWN, BEGIN, END, QUIT, and PREVIOUS.

When Soft-Key F9 (MISC) is pressed from the MAIN edit screen, the MISC soft-key strip appears:

F2: COMMENT: Used to place a "comment block" in the program. A comment block could consist of a message to the operator, such as "* .5000 END-MILL", or "* OPERATION 2 of 3". When pressed, the operator may use the CNC keyboard keys AND/OR the ASCII table (shown under F2-ASCII). The (*) character is automatically inserted at the beginning of the comment. This is necessary so the CNC will not try to execute something it does not understand. See section 5.3 above for use of the ASCII table.

F3: SEARCH: Used to SEARCH a block number (or certain characters, such as "SUB 1", or "TOOL# 5"). When pressed, the operator must give the desired block number (or use the ASCII table to search a word. See section 5.3 above for use of ASCII table). The cursor will move to the given block number (or word) when ENTER is pressed. If a NUMBER is entered, the CNC assumes you want to search a BLOCK NUMBER.

F4: PG-UP: Used move the program text UP, one page (9 blocks) at a time. This is a quick way to browse through the program NOTE: The ARROW up-down keys will also move the cursor, one block at a time.

F5: PG-DN: Used move the program text DOWN, one page (9 blocks) at a time. This is a quick way to browse through the program NOTE: The ARROW up-down keys will also move the cursor, one block at a time.

F6: BEGIN: Used to quickly move the cursor to the BEGINning of the program.

F7: END: Used to quickly move the cursor to the END of the

program.

F8: QUIT: Used to exit a program **WITHOUT SAVING** any of the changes/additions/deletions done in the current programming session. If changes were made, a prompt will ask the operator "Are you SURE you want to quit ?". Pressing F1 (Yes) or F2 (No) will end the function. In this way if a mistake is made, and the operator would like to start the edit session over again, QUIT allows he or she to do so.

F9: PREV: Used to change the CRT - screen to the PREVIOUS screen.

6.14 EXIT - F10

EXIT

F10 (EXIT) while in the MAIN EDIT Soft-Key strip allows the operator to EXIT the program, while the CNC automatically SAVES the program in memory. When pressed, the program will be saved into memory, and the CRT - screen will change back to the PROGRAM (directory) screen.

6.15 EDITING an EXISTING PROG.: RE-EDIT, INSERT, DELETE, SKIP

When a program needs to be changed (edited) one of four things may be done to change it:

1. Change an existing block(s) - (Re-Edit).
2. Add a block(s) - (Insert).
3. Delete a block(s) - (Delete).
4. Skip a block (insert a (*) character in front of it).

All of the above functions are done through the EDIT mode.

6.15.1 RE-EDIT (Changing an Existing Block)

To change an existing block, the CNC must be put into EDIT mode, with the correct program. The CURSOR may then be MOVED to the suspect BLOCK, then ENTER must be pressed.

When ENTER is pressed on the existing block, the data on that block is brought back up to the "entry" area of the CRT - screen (the upper-half of the screen), and the changes may then be made.

On blocks such as "Dim Abs" or "Unit Inch", pressing enter while on the block will toggle to the opposite (i.e; Dim Incr or Unit MM). This is because there are only two choices for that

particular block to be.

After making the change(s), press F10 (SAVE) to put the block back into the program. Then, when F10 (EXIT) is pressed, the program is saved, with the change(s) included.

6.15.2 INSERT (Adding a Block)

To INSERT (add) a block, the CNC must also be in EDIT mode, with the correct program. The CURSOR may then be MOVED to the position in the program where the new block is to be put.

Now, the operator simply chooses what type of block he/she wants to insert (Drill, Pocket, Mill) and programs it.

After programming the new block(s), press F10 (SAVE) to put the block into the program. The program blocks BELOW the newly added block will be automatically pushed down one block number, for each new block added. When F10 (EXIT) is pressed, the program is saved, with the new blocks(s) included.

6.15.3 DELETE (Removing a Block)

To DELETE (remove) a block, the CNC must be in EDIT mode, with the correct program. The CURSOR may then be MOVED to the block(s) in the program that is to be removed.

Now, the operator simply presses the CLEAR key to DELETE the block. (Remember, if a block is accidentally deleted the QUIT function may be used. See section 6.13 above for QUIT).

After deleting block(s), press F10 (EXIT), and the program is saved, with the changes included.

6.15.4 SKIP (Ignore or Skip a Block)

To SKIP (ignore) a block, the CNC must be put into EDIT mode, with the cursor on the block to be skipped. Press the **0** numeric key, and a * will appear at the beginning of the block. The * will cause the CNC to ignore that particular block. To remove the * press the **0** key again.
This function can be useful while proving programs, or to skip holes or pockets in similar workpieces.

6.16 SAMPLE PROGRAMS

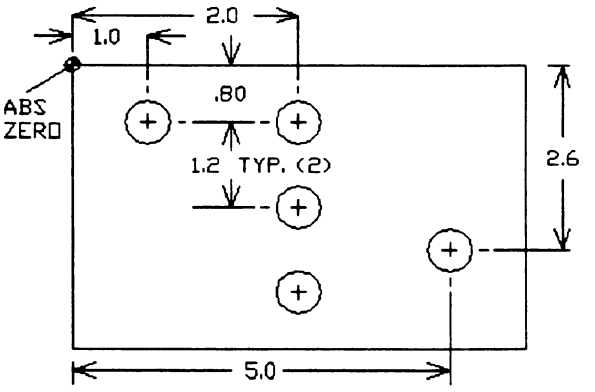
Included below are several "sample" programs, intended to help the operator learn the Series 1100 3-axis CNC's programming concepts. Please follow along, and use the examples as a guide now, and in the future. As you quickly get familiarized with the 1100

CNC, programming will become as easy as 1-2-3.

It is suggested that you actually key is each sample program, run in DRAW graphics, even LOAD it and run it in the AUTO mode (without a tool in the spindle). This way, you can practice programming and operating the Series 1100 control.

6.16.1 EXAMPLE #1: DRILL

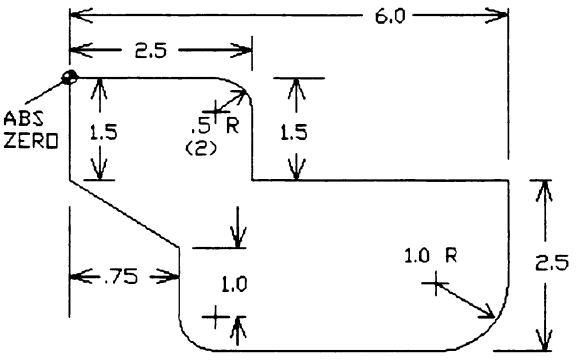
This DRILL example will use a 3/8" twist drill as Tool #1. A "Peck" drilling cycle will be used.




```
 1  DimAbs
 2  PeckDrill  Zdepth -.6500  StartHgt .1000  Peck .1500  Feed 12.8
          Tool# 1
 3  Rapid X1.0000 Y-.8000
 4  X2.0000
 5  Dim Incr
 6  Y-1.2000
 7  Y-1.2000
 8  Dim Abs
 9  X5.0000 Y-2.6000
10  DrillOff
11  Tool# 0
12  Rapid Z0.0000
13  X-3.0000 Y1.0000
14  <End Of Program>
```

6.16.2 EXAMPLE #2: MILL

This MILL example will use a 1/2" dia. end mill as Tool #1. Assume that the end-mill will climb-cut around the outside of the part. The 2 sharp inside corners will have the tool radius (not shown on drawing).

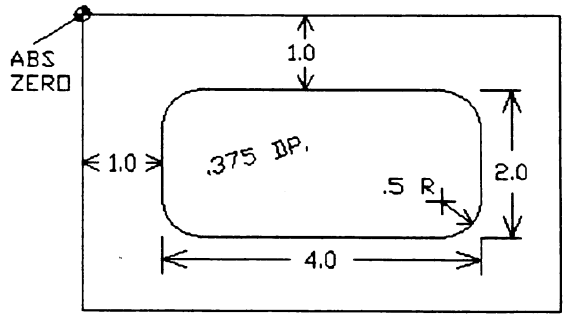



```
 1  DimAbs
 2  Rapid X-.3500 Y.0000  ToolComp Left  Tool# 1
 3  Z.1000
 4  Line Z-.2500  Feed 40.0
 5  Line X2.0000  Feed 15.0
 6  Arc Cw X2.5000 Y-.5000  Radius .5000
 7  Line Y-1.5000
 8  X6.0000
 9  Dim Incr
10  Line Y-2.5000  CornerRad 1.0000
11  Dim Abs
12  Line X.7500  CornerRad .5000
13  Y-2.5000
14  X.0000 Y-1.5000
15  Y.3500
16  Line X-.35  ToolComp Off
17  Rapid Z0.0000  Tool# 0
18  X-2.0000 X2.0000
19  <End Of Program>
```

NOTE: If a roughing mill and a finish mill were required, the contour (block 1 through block 18, excluding the TOOL on block 1) could be put into a SUB, then CALLED once for Tool #1, and once for Tool #2.

6.16.3 EXAMPLE #3: POCKET

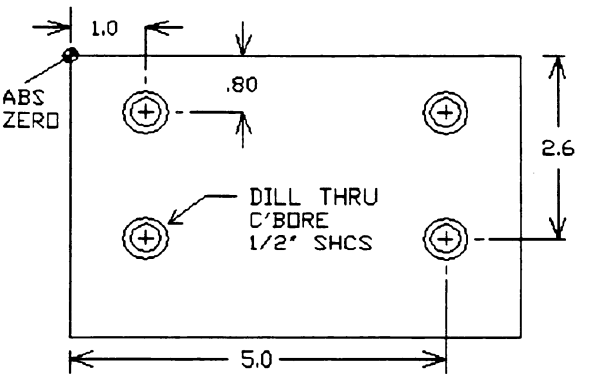
This POCKET example will use a 1/2" dia. end mill as Tool #1.




```
1  DimAbs
2  RectPock  XCenter 3.0000  YCenter -2.0000  StartHt .1000
     Length 4.0000  Width 2.0000  ZDepth -.3750  Ccw
     CornerRad .5000  Stepover .3000  DepthCut .1875
     FinStock .0150  RoughFeed 14.0  FinFeed 10.0  Tool# 1
3  Rapid  Z0.0000  Tool# 0
4  X-3.0000 Y3.0000
5  <End Of Program>
```

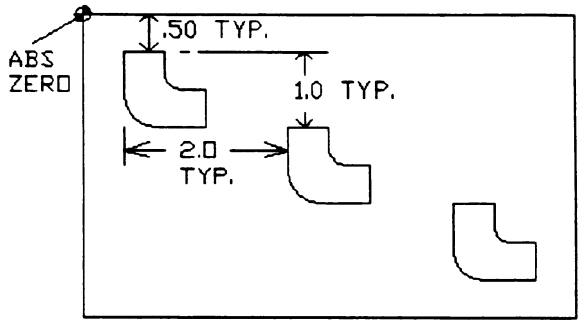
6.16.4 EXAMPLE #4: SUB using DRILL

This SUB program example will use a #4 centerdrill as Tool #1, a 17/32" drill as Tool #2, and a counterbore tool for a 1/2" S.H.C.S. (Socket Head Cap Screw).




```
 1  Tool# 1
 2  BasicDrill  Zdepth -.1800  StartHgt .1000  Feed 7.0
 3  Call 1
 4  Tool# 2
 5  PeckDrill  Zdepth -.6500  StartHgt .1000  Peck .125  Feed 11.5
 6  Call 1
 7  Tool# 3
 8  Boring  Zdepth -.2800  StartHgt .1000  Dwell 0.7  Feed 4.0
 9  Call 1
10  EndMain
11  Sub 1
12  DimAbs
13  Rapid  X1.0000 Y-.8000
14  X5.0000
15  Y-2.6000
16  X1.0000
17  DrillOff
18  Tool# 0
19  Z0.0000
20  X-1.0000 Y1.0000
21  EndSub
22  <End Of Program>
```

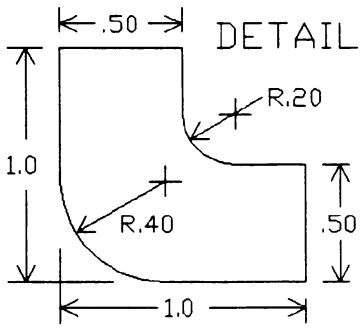
6.16.5 EXAMPLE #5: LOOP a SUB



This LOOP a SUB program example will use a 1/8" dia. end mill as Tool #1. Block #2 in the program will rapid the tool to a spot INSIDE the first window, then the LOOP/SUB will take effect.

The fact that the SUB is programmed in INCREMENTAL allows the shape to be cut anywhere on the workpiece.

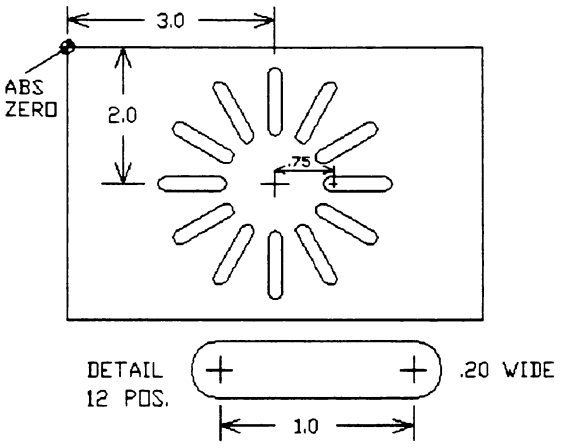
```
 1  DimAbs
 2  Rapid X.7500 Y-.7500  Tool#1
 3  Loop Sub# 1  #Loop 3  XIncr 2.0000  YIncr -1.0000
 4  Rapid Z0.0000  Tool# 0
 5  X-2.0000 Y2.0000
 6  EndMain
 7  Sub 1
 8  DimAbs
 9  Rapid Z.1000
10  Line Z-.0625  Feed 1.5
11  Dim Incr
12  Line Y.2500  ToolComp Left  Feed 2.1
13  X-.2500
14  Line Y-1.0000  CornerRad .4000
15  X1.0000
16  Y.5000
17  Line X-.5000  CornerRad .2000
18  Y.5000
19  X-.2500
20  Line Y-.2500 ToolComp Off
21  Dim Abs
22  Rapid Z.1000
23  EndSub
24  <End Of Program>
```



6.16.6 EXAMPLE #6: ROTATE a SUB

This ROTATE a SUB program example will use a 1/8" dia. end mill as Tool #1.

NOTE the Dim Abs (set ABSOLUTE dimensions) in Block #1. This is important, to set the ANGLE as absolute.




```
 1  Dim Abs
 2  RMS Sub# 1  #Loops 12 StartAngle 0.0  Angle 30.0
     XCenter 3.0000  YCenter -2.0000  Tool# 1
 3  Rapid Z0.0000  Tool# 0
 4  X-2.0000 Y2.0000
 5  EndMain
 6  Sub 1
 7  Dim Abs
 8  Rapid X3.7500 Y-2.0000
 9  Z.1000
10  Line Z-.1250  Feed 1.2
11  Dim Incr
12  Line Y.1000  ToolComp Left  Feed 2.2
13  Arc Ccw X.0000 Y-.2000 Radius .1000
14  Line X1.0000
15  Arc Ccw X.0000 Y.2000 Radius .1000
16  Line X-1.0000
17  Line Y-.1000  ToolComp Off
18  Dim Abs
19  Rapid Z.1000
20  EndSub
21  <End Of Program>
```

NOTE: The slot at the 3 o'clock position was programmed into the SUBPROGRAM. The RMS block then rotates the slot based on degrees between (30), and # of times (12), and center-point (X3 Y-2).

6.17 "HOT-KEYS" for QUICK PROGRAMMING

Shown below is a table of "Hot-Keys", that allow quicker access of some CNC functions, such as RAPID, LINE and ARC.

Normally, to access a LINE move, from the MAIN Edit menu, the operator would have to press F5 (Mill), then press F3 (Line).

A quicker way to access these functions, is through the NUMERIC keys on the CNC keyboard (keys 1, 2, 3, 4, 5, etc.). When pressed, each of them will directly access a menu, without having to navigate through the sub-menus:

FOR EXAMPLE:

The following table shows the numeric key's functions, while at the MAIN EDIT MENU:

Key	Function	Definition of Function
1	Rapid	Define a RAPID program block
2	Line	Define a LINE program block
3	Arc	Define an ARC program block
4	Feed	Set a feedrate
5	Tool	Activate a tool # (Tool 0 included)
6	MCode	Activate an M-code (ie; MCode 5, spdl off)
7	Unit	Set Inch or Millimeter programming
8	Dwell	Program a dwell
9	Plane	Select a Plane (XY, XZ, YZ).

This "Hot-Key" scheme allows the operator to quickly access functions with minimal menu navigation.

IMPORTANT NOTE:

The "Hot-Key" scheme may also be used while in Manual Mode. When used while in Manual Mode, the operator can access MDI moves directly, without entering into an MDI program. See section 4.7 above for more details.

7.0 CALCULATOR MODES

7.1 WHAT IS CALCULATOR MODE ?

CALC modes are 3 separate calculators that allow the operator to find answers to math and trig problems he or she faces while programming.

The Soft-Key strip of the RAPID, LINE, and ARC screens allow the operator to access the CALCULATOR modes, as well as F7 of the MAIN soft-key strip.

CALC

F7 (Calc) allows the operator to access any one of the three (3) CALC modes. When pressed, a window will appear on the screen with the prompt "Select Type of Calculator:"

1. Standard Calculator (+,-,*,/, and TRIG functions).
2. Right-Triangle Calculator (find side lengths or angles of a right triangle).
3. Geometry Calculator (build points, lines, circles).

The 3 calculators are shown by "icons" (figures) on the screen. The CALC on the LEFT side of the window is the STANDARD calculator. CENTER of the window is the RIGHT-TRIANGLE calculator. On the RIGHT side of the window is the GEOMETRY calculator.

The left-right ARROW keys may be used to move the cursor (dark background) to the desired CALCULATOR. When ENTER is pressed, the chosen calculator will become active.

7.2 STANDARD MATH CALCULATOR MODE

The STANDARD calculator allows addition (+), subtraction (-), multiplication (*) and division (/), as well as trigonometric functions such as Sine, Cosine and Tangent.

When selected, the STANDARD calculator will appear in the center of the CRT - screen, and the Soft-Keys will contain the necessary functions needed to run the calculator. This calculator uses the same principles used in standard hand-held calculators used in every-day life.

The Soft-Keys of the STANDARD calculator are explained below:

F1: ADD: Used to ADD values. EXAMPLE: PRESS 4.1875 + 3.8125 ENTER. The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store).

F2: SUBTRACT: Used to SUBTRACT values. EXAMPLE: PRESS F8 (CE

- Clear Entry) to clear the previous number, if any. PRESS 4.1875 - 3.8125 ENTER. The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store).

F3: MULTIPLY: Used to MULTIPLY values. EXAMPLE: PRESS F8 (CE - Clear Entry) to clear the previous number, if any. PRESS 4.1875 * 3 ENTER. The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store).

F4: DIVIDE: Used to DIVIDE values. EXAMPLE: PRESS F8 (CE - Clear Entry) to clear the previous number, if any. PRESS 4.1875 / 3 ENTER. The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store).

F5: OPEN (Left) PAREN: Used in an equation to open a localized calculation. EXAMPLE: PRESS F8 (CE - Clear Entry) to clear the previous number, if any. PRESS 3 * (6 + 2) ENTER. The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store).

F6: CLOSE (Right) PAREN: Used in an equation to close a localized calculation. See above EXAMPLE.

F7: FUNC: Used to do a TRIG or other function on a number keyed in, or on an answer that was calculated. EXAMPLE: PRESS F8 (CE - Clear Entry) to clear the previous number, if any. PRESS 30 FUNC (F7) select Cosine (by using ARROW down key and ENTER). The answer will appear. In order to "recall" the answer at a later time, the operator must STORE the answer by pressing F9 (Store). Other functions such as Square Root, Square, and more are included in a second pull-up window - MORE...

F8: CE - Clear Entry: Used CLEAR an answer or number form the calculator's active line.

F9: STORE: Used to save (store) an answer in a special memory, so it may be later "recalled" into a program.

F10: EXIT: Used to EXIT the calculator mode, to the previous screen (if STORE was pressed before EXIT, the last number stored will be "pasted" into the entry-space where the cursor was, before entering the calculator.

NOTE: Pressing the ARROW-UP key while in this calculator allows the operator to RECALL a value from any one of the calculators.

7.3 RIGHT-TRIANGLE (TRIG) CALCULATOR MODE

The RIGHT-TRIANGLE (TRIG) calculator allows the operator to find side lengths or angles of a right triangle.

When selected, the TRIG calculator will appear on the CRT - screen, with a sample triangle in the upper-left corner of the screen.

There are six (six) parts to any RIGHT-TRIANGLE, including 3 sides (or legs), and 3 angles - one of which is 90 degrees. The SIDES are labeled A,B,C and the ANGLES are labeled D,E, and F - 90 degrees. Angle "F" is set to 90 degrees, and cannot be changed. This calculator uses the same principles used in standard trigonometry.

The "prompt area" at the left of the screen is where the operator enters the KNOWN values. The operator must fill in exactly two (2) values, either sides and/or angles. Too many entries or not enough, will result in an error message.

Once the KNOWN values are keyed in, the operator muse press F7 (FIND), and the CNC will calculate and display the answers in the corresponding triangle "part" (A,B,C,D,E). Each value calculated by the CNC will have a (*) next to it so the operator can tell which values were calculated. A triangle corresponding to the calculated values will appear in the graphics area, with the ABCDEF labels shown also.

To STORE a value for a later RECALL, the operator must place the cursor on the value he/she wishes to store, and press F9 (STORE).

Previously stored values may be RECALLED into any of the triangle "parts" by placing the cursor on that part, and pressing F2 (RECALL). To clear ALL values (except F), press F8 (CLR-ALL). The CLEAR key on the CNC keyboard will only clear the value where the cursor rests. The Soft-Keys of the TRIG calculator are explained below:

F2: RECALL: When pressed, this key allows the operator to RECALL any value previously "stored", from any one of the calculators. The value will be placed into the position where the cursor rests.

F7: FIND: When pressed, the CNC will calculate and display the answers in the corresponding triangle "part" (A,B,C,D,E). Each value calculated by the CNC will have a (*) next to it so the operator can tell which values were calculated. A triangle corresponding to the calculated values will appear in the graphics area.

F8: CLR-ALL: Used to CLEAR ALL the values from the prompt area (except F). To clear a single value, place the cursor on that value, and press the CLEAR key on the CNC keyboard.

F9: STORE: Used to save (store) an answer in a special memory, so it may be later "recalled" into a program.

F10: EXIT: Used to EXIT the calculator mode, to the previous screen.

NOTES:
1. Two parts of a triangle (plus "F") must be known to solve any problem in the right-angle calculator mode.

7.4 GEOMETRY CALCULATOR MODE

7.4.1 WHAT is a GEOMETRY CALCULATOR ?

The GEOMETRY calculator allows the operator to build "geometry" (points, lines and circles) in order to find tough-to-find answers.

The operator can create the pieces of geometry that form a workpiece, using points. lines and circles. He/she simply selects an Icon (graphic image) with the cursor (using the ARROW keys), and presses ENTER. The display will then ask for a dimension such as length of line, radius of arc, XY coordinate of point, etc..

Ultimately, it is the **POINTS** that are necessary to program a workpiece. A point could be the "intersection of two lines", the intersect of a line and arc, etc..

While using the GEOMETRY calculator, the operator can see the points, lines and circles appear on the CRT - screen as they are programmed.

> When selected, the GEOMETRY calculator will appear in the center of the CRT - screen, and the Soft-Keys will contain the necessary functions needed to run the calculator. This calculator uses the same principles used in standard geometry.

The screen is similar to the DRAW graphics screen in layout, except the soft-key functions are related directly to CALCULATOR, and the "icons" are shown on the left.

As the cursor (arrows on CNC keyboard) is moved up, down, left and right, its location will be shown by a dark background.

"Geometry" can be placed onto the screen, to be used later with the "recall" function. Geometry may consist of the 3 basic elements; points, lines, and circles. When entered by the operator, these elements will appear on the screen in gray dotted points, lines, and circles.

Geometry can be used to find the intersection of lines and arcs, whose intersection is not given on the blueprint. Using Geometry can eliminate many trig functions. For example, the operator can place two circles on the screen, each with different centers and radii. He/she can then place a line tangent to both these circles, and then place points at the intersections of the lines and circles. These points may then be "recalled" into the part-program.

Geometry can be very useful when working with blueprints with limited information. Many combinations of geometry can be entered, to help find intersections.

The **Geometry Calculator** uses mostly **ABSOLUTE dimensions**. The Abs/Incr setting of the CNC has NO EFFECT on the Geometry Calculator itself. Some incremental definitions exist, such as a point defined from another existing point. Please TRY TO INPUT each one of the following points, lines, and circles, so that you may fully understand each one.

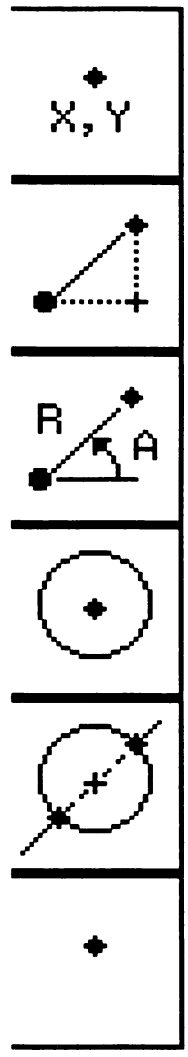
NOTE: Only POINTS may be RECALLED into a program. The point may be an intersection of 2 lines, the intersection of a line and arc, the center of a circle, etc...

7.4.2 POINTS

The LEFT-hand vertical column of six (6) icons are for creating POINTS. Each one has a specific way to describe a point. To select a point definition, simply move the cursor onto one of them, and press ENTER. A prompt will appear asking the operator to input values, and as he/she keys in the numbers, and presses ENTER, the point will appear on the screen. If the point does not appear, a "fit-zoom" may be done (using F5), to show all the points, lines, and circles that exist.

Shown below are **POINTS**. Lines and circles will be explained on the following pages.

Point type #1 defines a point by XY ABSOLUTE dimension. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for X value, then Y value. The operator should key in each dimension, pressing ENTER after each. The point will appear on the screen. This is to create a **NEW POINT**.



Point type #2 defines a point, also by XY dimension, but it allows the XY values to be from ABS zero, OR from another point. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for X value, Y value, then "from point:_" (origin point). To define from absolute zero, simply press ENTER without keying in a point number. To define from another existing point (incrementally), simple key in the NUMBER of that point. The operator should key in each dimension, pressing ENTER after each. The point will appear on the screen.

Point type #3 defines a point by radius and angle dimensions, and allows the R and A values to be from ABS zero, OR from another point. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for radius value, angle value, then "from point:_" (origin point). To define from absolute zero, simply press ENTER without keying in a point number. To define from another existing point (incrementally), simply key in the NUMBER of that point. The operator should key in each dimension, pressing ENTER after each. The point will appear on the screen.

Point type #4 defines a point as being the center of an existing circle. When the cursor is placed on this icon, and ENTER is pressed, the prompt will ask for "Number of circle:_". The operator must key in the number of the circle, then press ENTER. The point will appear on the screen, at the exact center of the circle.

Point type #5 defines a point at the intersection of two other elements (line-line, line-circle, circle-circle). When the cursor is placed on this icon, and ENTER is pressed, the prompt will ask for "Number of first element:_" then "Number of second element:_". The operator must key in the NUMBER of the 1st element, then press ENTER, NUMBER of the 2nd element, then ENTER. All intersect possibilities will appear on the screen, and each will be numbered. After choosing one (by its number), the point will appear on the screen, at the intersection of the 2 elements.

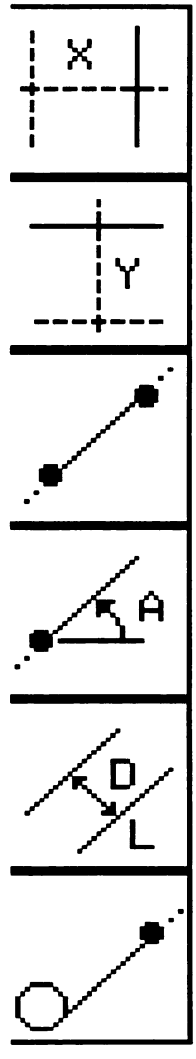
Point type #6 is ONLY used to pick an EXISTING point for the construction of other elements, such as lines and circles. For example, if a circle is to be input (see below for circles), one of

the prompts for the circle will be "Select center point definition:_", (to define the circle center). At this time, the points column will be active, and the operator must choose a point definition, to describe the center of the circle. This icon can also be used in other definitions, such as "line between two points". See next section(s) for further details.

7.4.3 LINES

The CENTER vertical column of six (6) icons are for creating LINES. Each one has a specific way to describe a line. To select a line definition, simply move the cursor onto one of them, and press ENTER. A prompt will appear asking the operator to input values, and as he/she keys in the numbers, and presses ENTER, the line will appear on the screen. If the line does not appear, a "fit-zoom" may be done, to show all the points, lines, and circles that exist.

Shown below are **LINES**. Circles will be explained on the following pages (see above for points).



Line type #1 defines a vertical line by X ABSOLUTE dimension. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for X value. The operator should key in each dimension, pressing ENTER after each. The line will appear on the screen.

Line type #2 defines a horizontal line by Y ABSOLUTE dimension. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for Y value. The operator should key in each dimension, pressing ENTER after each. The line will appear on the screen.

Line type #3 defines a line between 2 points. When the cursor is placed on this icon and ENTER pressed, the prompt will change to "Select end point definition:_", and the cursor will move to the POINTS group, described above. Any one of these point definitions can be chosen to describe which 2 points the LINE will go through. If point definition #1 is chosen, the operator may key in the X-Y value of the point(s) that the line goes through (point def. #6 allows to choose existing points to go through). The line will appear on the screen after all the data is entered. See section 7.4.2 above for Points.

Line type #4 defines a line through a point at an angle. When the cursor is placed on this icon and ENTER pressed, the prompt will ask for angle, then the prompt will change to "Select point definition:_", and the

cursor will move to the POINTS group. Any one of these point definitions can be chosen to describe which point the line will go through, at the given angle. Press ENTER after each prompt. The line will appear on the screen after all the data is entered. See section 7.4.2 above for Points.

Line type #5 defines a line at a parallel distance (D) from another existing line. When the cursor is placed on this icon and ENTER pressed, the prompt will ask for the number of the line to be offset from, then the offset distance (+ or -). Press ENTER after each prompt. The line will appear on the screen. See section 7.4.2 above for Points.

Line type #6 defines a line tangent to a circle, through a point. When the cursor is placed on this icon and ENTER pressed, the prompt will ask for "Number of circle:_", then the prompt will change to "Select point definition:", and the cursor will move to the POINTS group. Any one of these point definitions can be chosen to describe which point the line will go through, tangent to the circle. Press ENTER after each prompt. The line will appear on the screen after all the data is entered. See section 7.4.2 above for Points.

7.4.4 CIRCLES

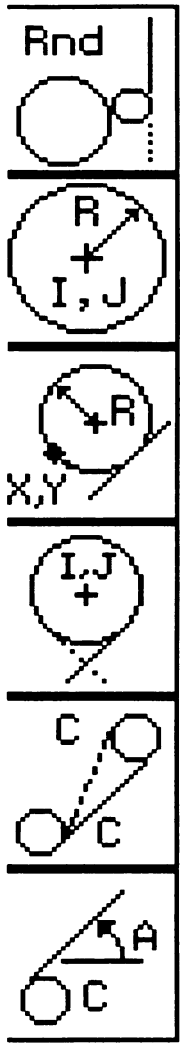
The RIGHT vertical column of six (6) icons are for creating CIRCLES (or line tangent to circle). Each one has a specific way to describe a circle (or tangent line). To select a circle definition, simply move the cursor onto one of them, and press ENTER. A prompt will appear asking the operator to input values, and as he/she keys in the numbers, and presses ENTER, the circle will appear on the screen. If the circle does not appear, a "fit-zoom" may be done, to show all the points, lines, and circles that exist.

Shown below are **CIRCLES**. Points and Lines are explained on the above pages.

Circle type #1 defines a circle by TANGENCY between two other lines and/or circles. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for R value, 1st element #, then 2nd element #. The operator must answer each prompt, pressing ENTER for each. All of the tangent possibilities will appear on the screen, and each will be numbered. After choosing the desired tangency # and pressing ENTER, the circle will appear on the screen.

Circle type #2 defines a circle by Radius and center point. When the cursor is placed on this icon, and ENTER pressed, the

prompt will ask for R value, then the prompt will change to "Select center point definition:_", and the cursor will move to the POINTS group. Any one of these point definitions can be chosen to describe the center point. The operator must answer each prompt, pressing ENTER. The circle will appear on the screen.



Circle type #3 defines a circle passing through a point, tangent to a line. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for R value (radius of the circle), the # of line of to be tangent to, then the prompt will change to "Select point definition:_", and the cursor will move to the POINTS group. Any one of these point definitions can be chosen to describe which point the circle will pass through. The operator must answer each prompt, pressing ENTER. All tangent possibilities will appear on the screen, and each will be numbered. After choosing the desired number, the circle will appear on the screen.

Circle type #4 defines a circle by XY center point, and tangent to a line. When the cursor is placed on this icon, and ENTER pressed, the prompt will ask for the number of line to be tangent to, then the prompt will change to "Select center point definition:_", and the cursor will move to the POINTS group. Any one of these point definitions can be chosen to describe the center point. The operator must answer each prompt, pressing ENTER. The circle will appear on the screen.

Circle Icon type #5 and #6 **<u>define lines</u>**. Type #5 defines a line tangent to 2 circles. When the cursor is placed on this icon and ENTER pressed, the prompt will ask for the NUMBER of a circle, then the NUMBER of the second circle to be tangent to. Press ENTER after each prompt. **<u>Note:</u>** 1/2 of the tangent possibilities will appear on the screen, and each will be numbered. After choosing the desired tangent line, the line will appear on the screen. **<u>Note:</u>** To obtain any other tangent possibilities, reverse the order of the element numbers. Then, the other 1/2 of the tangent possibilities will be shown.

Circle Icon type #6 also defines **<u>a line</u>**. #6 defines a line tangent to a circle at an angle. When the cursor is placed on this icon and ENTER pressed, the prompt will ask for angle, then the number of a circle to be tangent to. Press ENTER after each prompt. The line will appear on the screen.

The Soft-Keys of the GEOMETRY calculator are explained below:

F5: DISPLAY: Allows the operator to Zoom, Window-Zoom, etc.. See section 9 for details on the DISPLAY key.

F7: GEOMETRY: Allows the operator to VIEW a list of the geometry, delete one ITEM of geometry, or delete ALL the geometry. Use the ARROW keys to move to the desired selection, and press ENTER.

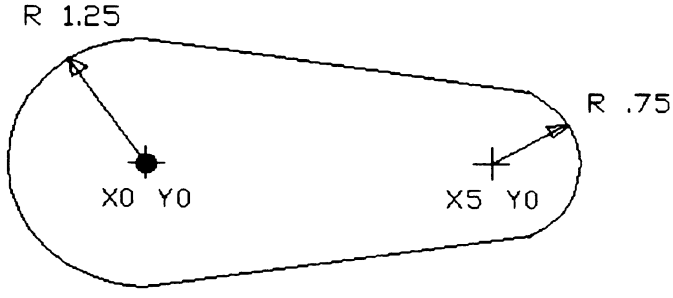
F10: EXIT: Used to EXIT the calculator mode, to the previous screen.

NOTE: Point values and circle centers are stored automatically, therefore, the store key does not appear.

EXAMPLE: The following example shows how to find the unknown points to program the shape shown in the figure below. The points needed are where the lines intersect the arcs.

This example could be a slot milled into a workpiece, or a male shape. In either case, the same points must be found. X0 Y0 is set at the center of the large radius. Watch the prompts after every ENTER, as you try this example.

Keystrokes:



1. F2 PROGRAM (if necessary)
2. F2 Create
3. key in 123 press ENTER
4. F4 EDIT
5. F7 CALC, select the Geometry Calc
6. Move the cursor to the second CIRCLE icon, press ENTER
7. The prompt asks for the radius value. Press 1.25 ENTER. The cursor is now active on the "points" icons, asking how to describe the center point of the circle. Since we know the center to be X0-Y0, select the top point icon by pressing ENTER. Now the prompts ask for X value and Y value, so press 0 ENTER 0 ENTER. The circle will appear on the screen. If it does not, press F5 (Display) and FIT (to do an AUTO-FIT zoom).
11. Press ENTER again, to select the same circle icon again.

12. Press .75 ENTER for the radius, Press ENTER to select the top points icon, press 5 ENTER 0 ENTER (for the center point location), then press F5 ENTER (to Fit on screen).
13. Cursor down to circle icon #5 (line tangent to 2 circles), press ENTER.
14. Press 1 ENTER 2 ENTER (for the NUMBERS of the circles to be tangent to), and 2 tangent possibilities will appear. Press 1 ENTER to select the correct tangent line.
15. Select the same icon (line tangent to 2 circles) again by pressing ENTER. Now, key in 2 ENTER 1 ENTER, and the other 2 tangent possibilities appear. Select 1 ENTER.
16. Cursor over to the 5th point icon (point at the intersection of 2 elements), and press ENTER to select.
17. Press 1 ENTER 3 ENTER, and the point (#5) will appear. This is one of the points needed to program this contour.
18. To find another necessary point, press ENTER (to select the intersect icon again).
19. Press 2 ENTER 3 ENTER, and the point (#6) will appear.
20. To find the third necessary point, press ENTER (to select the intersect icon again).
21. Press 2 ENTER 4 ENTER, and the point (#7) will appear.
22. To find the last necessary point, press ENTER (to select the intersect icon again).
23. Press 1 ENTER 4 ENTER, and the point (#8) will appear.

Now, the 4 necessary points are known. Pressing F7, the selecting GEOMETRY LIST will show the data on the elements just entered. Points #5,6,7 and #8 are what is needed.

These four points may be RECALLed into the program, one at a time, whenever needed. See section 7.5.2 below for recall function.

7.5 STORE and RECALL

STORE and RECALL allow answers found in any of the CALCULATOR modes to be memorized by the CNC and RECALLED into the program, into any place the operator needs it.

7.5.1 STORE

STORE

F9 (Store) allows answers to be saved (stored) in a special memory, so it may be "recalled" into a program by the RECALL key described below. Used in the STANDARD and RIGHT-TRIANGLE calculators. (Geometry calculator automatically stores all info).

7.5.2 RECALL

RECALL

F2 (Recall) allows values previously calculated in any CALC mode to be "recalled" into area(s) of the input screens for RAPID, LINE, ARC, Z-MOVE, HOME and SET-ZERO.

When F2 (Recall) is pressed, the screen will prompt the operator as to which CALCULATOR to recall the value(s) from. Three (3) choices are present, the standard CALC, the right-triangle CALC, and the geometry CALC. One of these calculators can be chosen by using the left-right ARROW keys. Place the cursor (dark background) on the desired CALC and press ENTER.

Any value that was "stored" may now be recalled (Geometry CALC does not require storing). The most recent "stored" value will be the first one, the next recent will be the second one, and so on.

The value will be "recalled" (or "placed") into the space on the input screen where the cursor was before the RECALL function was initiated. **NOTE:** RECALL will not function if the cursor is on an item which is cannot be numeric, such as CW/CCW, ToolComp, etc.. (this is most common in the ARC menu).

The examples below assume that a CALC mode was used to store information prior to using the RECALL function. See section 7 for CALC modes.

EXAMPLE #1:

While in the RAPID menu, the cursor is resting on the X value, which is now blank. F2 (Recall) is pressed. The "standard" or "right-triangle" calculator is chosen, and a value of .7071 is shown on top of the "answer window". When ENTER is pressed, that value (.7071) will be automatically input into the X space on the input screen. This saves the operator having to memorize or write down values that are calculated.

EXAMPLE #2:

While in the ARC menu, the cursor is resting on the X value, which is now blank. F2 (Recall) is pressed. The "geometry" calculator is chosen, and several points that were "created" in the geometry calculator are shown. The arrow up-down keys may be used to move the cursor onto the desired X-Y point (such as point #4 described in the example above). When ENTER is pressed, a second window will appear, asking the operator if he/she wants "Both X and Y" values, "X Only" value, or "Y Only" value. Whatever is chosen will be automatically input into the X and/or Y spaces on the input screen. This saves the operator having to memorize or write down values that are calculated.

NOTES:

1. The cursor must be placed on the "space" that the answer is to be placed.

2. Answers in the standard and right-triangle CALC modes must be "stored" using the STORE Soft-Key, before they may be RECALLED.

3. Values in the geometry calculator are stored automatically.

8.0 RUNNING PROGRAMS (CUTTING PARTS) AUTO - S.STEP

8.1 MESSAGE LINE

As shown in section 2.3 above, the CRT - Screen is divided into two main parts, TOP and BOTTOM. Near the very bottom, just above the Soft-Key labeled boxes, is the MESSAGE LINE.

The MESSAGE LINE is where messages to the operator will appear, such as "Tool Change", "Program Ended Normally", and other requests by the CNC to the operator. Error messages also appear on the Message Line.

For most messages, the operator must press the START key to continue the program. This allows as much time as necessary for the operator to change a tool, or any other manual operation required.

See section 2.3 above for the screen figure.

8.2 SINGLE-STEP MODE

Single-Step mode (referred to as S.STEP), is a mode of the CNC control that allows only one block (or 1 move) of data to be executed each time the START key is pressed. In this way, each block of the program can be inspected as it is executed, and XYZ end-points can be checked before the next block begins execution.

It is important to check which program is **LOADED** (from the LOAD key described in section 5.7 above), before going into S.Step mode. The LOADed program will be the one that is going to be used in S.Step OR Auto modes. Make sure the program you want to cut with is LOADed. Check the upper-right status area, the word **PROGRAM:**

S.Step mode is useful in "proving" the program. It is generally used after the program is checked in Draw mode, and before the piece-part run begins in Auto mode.

S.Step Mode may be aborted at any time by pressing F4 (MANUAL). Pressing F4 will abort the program AND the S.Step mode. The machine axes and functions will remain as they were when the S.Step mode was aborted. Exceptions to this rule are: ToolComp will be canceled, and "canned cycles" like pocketing or drilling will be aborted.

The START and HOLD keys on the CNC keyboard will begin and halt execution of a block when pressed. The HOLD key only halts the program, it does not permanently abort the program. To resume the program after pressing HOLD, simply press the START key.

8.2.1 Entering S.STEP Mode: F5

S.Step Mode can be entered into from the CNC's main soft-key

strip, from either Manual or Auto modes. Pressing F5 will set the CNC to S.Step mode. A slight delay may occur.

Soft-key **F7 MOTION** toggles on/off for S.Step by MOTION, or S.Step by BLOCK.

MOTION ON : One MOTION block will be executed for each press of the Start key. Non-motion blocks (blocks without axes moves or positions, like ZERO-SET, will be executed in sequence until a motion block is reached. A MOTION block has a command to MOVE the machine. Thus the term "Single-Step by MOTION".

MOTION OFF : One single block of data is executed for each press of the Start key, regardless of its contents. Thus the term "Single-Step by BLOCK".

The S.Step CRT - screen is the same as that of the Manual Mode (see section 2.3 above). It shows the machine status as well as some lines of the program being executed.

Below are the soft-key descriptions with details of their functions.

8.2.2 SEARCH: F3

While in S.STEP mode, before a program has started execution, F3 will allow the operator to SEARCH a "block" in the LOADed program. Search only works in the forward direction.

SEARCH allows the operator to start the program at a known block, and can be used for a "mid-program start". Care should be taken and the operator should ensure that the block searched for will start or continue the program from a safe position. The CNC will begin execution from the searched block when Start is pressed.

When F3 (Search) is pressed while idle in the S.Step mode, a prompt will appear on the "Message line": **Enter block to search: _**
The operator should then key in the block number, and then press ENTER, or use the ASCII table to key in a "word" to search. See section 5.3 above for use of ASCII table.

The Search key (F3) will not appear on the screen if the program is halted by the Hold key. The program must be idle, finished, or aborted to use F3.

EXAMPLES:

1) To search for block 125 : Press F3, press 125, press ENTER. The CNC will position the program at block 125. The Start key must be pressed to start the program from that block.

NOTE: The CNC arrow-up/down keys also provide a block-by-block

search of the program while the CNC is idle. Neither F3 or the arrow keys will work if the CNC is halted by the Hold key. The program must be idle, finished or aborted by F4 to use a search function.

8.2.4 MANUAL: F4

Pressing F4 (Manual) will ABORT the program AND the S.Step mode. The CNC will change to Manual Mode.

As explained above, S.Step Mode may be aborted at any time by pressing F4 (MANUAL). Pressing F4 will abort the program AND the S.Step mode. The machine axes and functions will remain as they were when the S.Step mode was aborted. Exceptions to this rule are: ToolComp will be canceled, and "canned cycles" will be aborted. See section 4 above for Manual Mode.

8.2.5 AUTO: F6

Pressing F6 (Auto) will change the mode from S.Step to AUTO. If F6 is pressed (while in S.STEP mode), the CNC will stop at the end of the current move, and by pressing START once more, the program will continue to run the blocks sequentially in the AUTO mode.

The operator may switch from S.Step to Auto, or from Auto to S.Step at any time during the execution of a program without aborting the program. In this way, a critical area of a program may be carefully inspected in S.Step before switching to the Auto mode. See section 8.3 below for Auto Mode.

8.2.6 MOTION: F7

Soft-key F7 (Motion) toggles on/off for S.Step by MOTION, or S.Step by BLOCK.

MOTION ON : One MOTION block will be executed for each press of the Start key. Non-motion blocks (blocks without axes moves or positions, like DWELL 2.5), will be executed in sequence until a motion block(s) is reached. Thus the term "Single-Step by motion".

MOTION OFF : One single block of data is executed for each press of the Start key, regardless of its contents. Thus the term "Single-Step by block".

The MOTION Soft-key will be highlighted when Motion is ON. When Motion is OFF (Step by block), the key will NOT be highlighted.

8.2.7 DRAW: F10

Pressing F10 (Draw) will set the CNC into Real-Time Graphics (Draw While Cutting) Mode. It is VERY important to understand that

MACHINE MOVEMENT WILL OCCUR if Draw is entered into from S.Step or Auto Modes (See section 9 below - Draw Mode Details).

Complete details of the Draw mode are explained below in section 9 of this manual.

NOTES ON S.STEP MODE:

1. S.Step mode is a useful tool in the "proving" out of a program before the piece-part run is made. It allows only ONE block of program to be executed each time the Start key is pressed.

2. The active program block will be HIGHLIGHTED when in S.Step mode.

3. To ABORT the S.Step mode and program, press F4 (Manual).

4. Draw (F10) mode from the S.Step mode WILL PRODUCE MACHINE MOVEMENT. This is knows as Real-Time Draw Graphics.

5. The START and HOLD keys begin and halt execution of a block while in S.STEP mode. HOLD only 'stops' the program, does not abort it.

8.3 AUTO MODE

Automatic mode (referred to as AUTO), is a mode of the CNC control that allows continuous production of workpieces. In this way, when the START key is pressed, the program will run continuously, until a "prompt" on the message line appears, or until the program is finished.

It is important to check which program is **LOADED** (from the LOAD key described in section 5.7 above), before going into Auto mode. The LOADed program will be the one that is going to be used in S.Step OR Auto modes. Make sure the program you want to cut with is LOADed. Check the upper-right status area, the word **PROGRAM:**

Auto Mode may be aborted at any time by pressing F4 (MANUAL). Pressing F4 will abort the program AND the Auto mode. The machine axes and functions will remain as they were when the Auto mode was aborted. Exceptions to this rule are: ToolComp will be canceled, and "canned cycles" like pocketing and drilling will be aborted.

The START and HOLD keys on the CNC keyboard will begin and halt execution of a block when pressed. The HOLD key only halts the program, it does not permanently abort the program. To resume the program after pressing HOLD, simply press START key.

8.3.1 Entering AUTO Mode: F6

Auto Mode can be entered into from the CNC's main soft-key strip, from either Manual or S.Step modes. Pressing F6 will set the CNC to Auto mode.

The Auto CRT - screen is the same as that of the Manual Mode (see section 2.3 above). It shows the machine status as well as some lines of the program being executed.

Below are the soft-key descriptions with details of their functions.

8.3.2 SEARCH: F3

While in AUTO mode, before a program has started execution, F3 will allow the operator to SEARCH a "block" in the LOADed program.
SEARCH allows the user to start the program at a known block, and can be used for a "mid-program start". Care should be taken and the operator should ensure that the block searched for will start or continue the program from a safe position. The CNC will begin execution from the searched block when Start is pressed.

When F3 (Search) is pressed while idle in the S.Step mode, a prompt will appear on the "Message line": **Enter block to search: _**
The operator should then key in the block number, and then press ENTER, or use the ASCII table to key in a "word" to search. See section 5.3 above for use of ASCII table.

The Search key (F3) will not appear on the screen if the program is halted by the Hold key. The program must be idle, finished, or aborted to use F3.

EXAMPLES:

1) To search for block 125 : Press F3, press 125, press ENTER. The CNC will position the program at block 125. The Start key must be pressed to start the program from that block.

NOTE: The CNC arrow-up/down keys also provide a block-by-block search of the program while the CNC is idle. Neither F3 or the arrow keys will work if the CNC is halted by the Hold key. The program must be idle, finished or aborted by F4 to use a search function.

8.3.4 MANUAL: F4

Pressing F4 (Manual) will ABORT the program AND the Auto mode. The CNC will change to Manual Mode.
As explained above, Auto Mode may be aborted at any time by pressing F4 (MANUAL). Pressing F4 will abort the program AND the S.Step mode. The machine axes and functions will remain as they were when the Auto mode was aborted. Exceptions to this rule are:

ToolComp will be canceled, and "canned cycles" will be aborted. See section 4 above for Manual Mode.

8.3.5 S.STEP: F5

Pressing F5 (S.Step) will change the mode from Auto to S.STEP. If F5 is pressed (while in AUTO mode), the CNC will stop at the end of the current move, and by pressing START once more, the program will continue to run the blocks one-at-a-time in the S.STEP mode.

The operator may switch from Auto to S.Step, or from S.Step to Auto at any time during the execution of a program without aborting the program. In this way, a critical area of a program may be carefully inspected in S.Step before switching to the Auto mode. See section 8.2 above for S.Step Mode.

8.3.7 DRAW: F10

Pressing F10 (Draw) will set the CNC into Real-Time Graphics (Draw While Cutting) Mode. It is VERY important to understand that **MACHINE MOVEMENT WILL OCCUR** if Draw is entered into from S.Step or Auto Modes (See section 9 below - Draw Mode Details).

Complete details of the Draw mode are explained below in section 9 of this manual.

NOTES ON AUTO MODE:

1. The active program block will be HIGHLIGHTED when in Auto mode.

2. To ABORT the Auto mode and program, press F4 (Manual).

3. Draw (F10) mode from the Auto mode WILL PRODUCE MACHINE MOVEMENT. This is knows as Real-Time Draw Graphics.

4. The START and HOLD keys begin and halt execution of a block while in AUTO mode. HOLD only 'stops' the program, does not abort it.

8.4 STARTING AND STOPPING the PROGRAM

The START key will start the program running, whether in S.Step or Auto modes. As stressed above, MAKE SURE the program loaded is the one you want to cut with. The active or "working" program name can be seen behind the word **PROGRAM:** on the status screen, in the upper-right area. See section 5.7 above for LOAD.

The HOLD key will temporarily halt the program. The program will continue to run when the START key is pressed again.

The hold key does NOT abort (terminate) the running program.

8.5 ABORTING the PROGRAM

To terminate, or ABORT a running program, the **MANUAL** (F4) key must be pressed. This will allow the operator to gain access to the other modes of the CNC, such as Edit, Program, simulation Draw, MDI, and more.

When pressed, the machine axes and functions will remain as they were when the Auto or S.Step mode was aborted. Exceptions to this rule are: ToolComp will be canceled, and "canned cycles" will be aborted. See section 4 above for Manual Mode.

8.6 REAL-TIME DRAW (DRAW WHILE CUTTING)

Real-Time Graphics (Draw While Cutting) allows the TOOL PATH to be viewed on the CRT - screen WHILE the moves are being executed. The standard Draw Mode (simulation draw, accessed from the Program page) should be run first to check for errors, and to size the part on the screen.

ALL graphic parameters must be set in the standard Draw Mode. The parameters cannot be set or changed in Real-Time Draw mode.

The Soft-key display and functions will be that of the Auto or Single Step mode. The CLEAR key can be used to clear the screen anytime.

See section 9 below for complete details on Draw Mode.

9.0 DRAW MODE DETAILS

9.1 WHAT is DRAW MODE ?

Draw mode (part graphics), is a method of proving a program before any material is cut. This helps eliminate scrap, and helps reduce the chances for a potentially serious mistake.

DRAW allows the operator to view the part edge and/or tool path from different views, zoom in on a particular area of the part, and generally inspect the moves the machine will make.

Draw graphics can be run while the machine is idle (Simulation Graphics), or while the machine is cutting (Real-Time graphics). Whether or not the machine will run depends on which mode DRAW was entered into from. It is VERY important to understand this difference. See below.

9.2 ENTERING DRAW MODE:

Draw graphics is a specific Mode of the CNC control, and can be entered into from the EDIT screen, or from the AUTO or S.STEP screens. There is a **DISTINCT DIFFERENCE** in how the graphics mode is entered:

If entered from **EDIT** screen: (F2) The CNC will draw the part **without** machine movement (Simulation Graphics).

If entered from **AUTO** or **S.STEP**: (F10) The CNC will draw the part **while it is actually machining** the part (Real-Time Graphics).

It is very important to understand this difference, as it will distinguish whether the machine will move or not while running DRAW graphics.

Draw is USUALLY used from the EDIT screen, so the program can be reviewed before any machining takes place (WITHOUT machining - "SIMULATION" graphics). If desired, the operator may set the CNC to "draw while cutting", by entering the Draw graphics mode DIRECTLY from the AUTO or S.STEP modes (WITH machining - "Real-Time" graphics). In the latter case, the program must be LOADed using F6 from the PROGRAM page. This section of the manual will cover SIMULATION draw mode only. See section 8.6 above for real-time DRAW mode.

9.3 DRAW MODE SCREEN:

After pressing the DRAW soft-key (F2) from the EDIT mode, the CNC will be in SIMULATION DRAW Graphics mode, and the Soft-Key

strip will change. Pressing F2 again (or F10) will DEACTIVATE the DRAW mode, and the Soft-Key strip will change back to the EDIT Soft-Keys, but the graphic box will remain.

The DRAW screen is the same as that of the Manual, Auto and S.Step modes, with the exception that the UPPER-RIGHT quarter of the CRT - screen will be used to show the moves in the program. Any messages will be shown near the center-line of the CRT screen, in the upper-half.

Inside the bottom half of the screen is where the blocks of the program being run are displayed. While graphics are running, the blocks will scroll up the screen. When F2 is OFF, the blocks may be edited. Also here is the Soft-key strip display.

The moves are distinguished by type, that is, RAPID or FEED moves:

Rapid: DOTTED line.

Feed: SOLID line or arc.

ToolComp moves: (solid or dotted, depends on feed or rapid). NOTE: Compensation TOGGLE must be set to USE or BOTH for compensated moves to be DRAWN. See section 9.7 below.

Drilled Holes: Shown as a cylinder, provided that a TOOL has been defined in the Tool Page, and is activated with a Tool# command (such as Tool# 1, Tool# 2, etc..).

SOFT-KEYS IN THE DRAW MODE:

Below are the soft-key descriptions with details of their functions.

Many of the settings of the DRAW mode interact with each other, and can be set using the Parms (F9 - Parameters) soft-key. As always, it is recommended that the entire chapter of DRAW Mode be reviewed before operation. This will help eliminate questions that may arise.

Any Soft-Key that is currently active will be highlighted in reverse video.

9.4 RUN: F3

Pressing F3 (Run) will START the selected program running, and the Graphics will begin to "DRAW" the selected program.

WHILE THE PROGRAM IS RUNNING, several selections can be toggled or set:

F1 (Auto) will set the graphics Run mode to "Auto" (it will execute blocks continuously, one after the other). In Auto, the blocks are executed sequentially, and will continue to be run until the ending block is reached (or if "Real-Time Draw", until a message (manual tool change, program stop, etc.., is reached).

F2 (Block) will set the graphics Run mode to "Single-Step by Block". In Block, only one block of program is executed for each press of the Start key, whether it is a movement block or not. In this way, the program blocks may be reviewed one at a time as they are executed (providing Text is toggled On), before allowing the program to continue.

F3 (Motion) will set the graphics Run mode to "Single-Step by Motion". In Motion, one MOTION will be executed for each press of the Start key. Blocks without an axis move (i.e; DWELL 2.2) will be executed, but the graphics will continue through those blocks until a motion commanded move is reached. In this way, motions can be reviewed one at a time, before allowing the program to continue.

F4 (Text) is a "toggle" type 'switch' to turn the program text on or off. With Text On (text Soft-Key highlighted), the blocks of program are displayed in the bottom half of the screen, as they are executed. With Text Off, program blocks will not appear, and the graphics will run faster.

F5 (Tool) is a "toggle" type 'switch' to turn the "dynamic" tool display on or off. With Tool On (highlighted), the tool (as defined in the Tool Page) will be displayed as it is moved through the part program. With Tool Off, the tool(s) will not appear, and the graphics will run faster.

F6 (Rapid) is a "toggle" type 'switch'. It allows the operator to "turn-off" the rapid moves. When ON, the rapid moves in the program will be shown. When OFF, no rapid moves are shown.

F7 (Start) is the key to re-start the graphics to Run, if it has been stopped by the Motion mode, Block mode, or the Hold key. The Start key on the CNC keyboard has the same effect.

F8 (Hold) key may be used to halt the graphics at any time it is running. In this way, the part can be "stopped" being drawn, for inspection of moves. Pressing F7 will re-start the graphics running. The Hold key on the CNC keyboard has the same effect.

F9 (Cancel) can be used to ABORT the running of the program. Pressing F9 will abort the program and return the screen to the main Draw mode soft-key strip.

NOTE: Any active soft-keys will be highlighted.

9.5 VIEW: F4

Pressing F4 (View) brings the first "pull-up window" in Draw mode to the screen, allowing the operator to choose which view the program will be drawn in. As with all pull-up windows, cursor up/down to the desired choice, then press ENTER.

XY will display the TOP VIEW when Drawing the program. This is considered to be the XY view.

XZ will display the FRONT VIEW when Drawing the program. This is considered the XZ view.

YZ will display the SIDE (END) VIEW when Drawing the program. This is considered the YZ view.

Iso will display an Isometric view of the program as it is run. The vantage point is from the lower-right quadrant of the Cartesian Coordinate System (quadrant IV).

9.6 DISPLAY: F5

Pressing F5 (Display) brings the display "window" to the screen, allowing the operator to set items related to the graphics screen, such as zoom size, erase screen, etc. The items under Display (F5) are detailed below:

Fit will cause an "auto-zoom" or auto-fit of the executed blocks on the screen. This will 'fit' all of the moves onto the graphics screen, allowing the operator to view all of the moves made in a program (providing the Start and End blocks are set accordingly). In this way, the entire part can be quickly shown without setting a zoom scale factor.

Window allows a section of the part to be easily "zoomed" in on. When window is pressed, a 'window' (of default size) will appear on the screen. The arrow keys will allow the window to be moved around the screen, and soft-keys F5, F6, F7, and F9 allow the window to be expanded, compressed, reset to original size, or for the process to be aborted. Once a desired window position and size is reached, pressing F10 or ENTER will cause the graphics to fit only the moves included in the "window" on the screen. In this way, a small area of moves on a part can be closely inspected.

Half acts as Fit (explained above), except it zooms the part graphics moves to half (1/2x) of the previously set size.

Double acts as Half (explained above), except it zooms the part graphics to twice (2x) the previously set size.

Scale allows the user to scale the graphic representation on the screen to a desired value. It has no effect on the actual part size. After pressing scale, the cursor will await entry of a scale factor, such as 1.5, .25, 5, etc. When ENTER is pressed, the graphics will begin to draw when F3 (Run) is pressed, or if the "auto-run" toggle is active. In this way, the operator may get an ideal size of the part to show on the screen.

Erase allows the graphics and the text (if any) to be completely erased from the graphics and text areas of the screen. Erase is a one-shot process, and has no other use.

9.7 PARMS (Graphics Parameters): F9

Pressing F9 (Parms) allows certain parameters of the graphics mode to be turned On or Off, such as tool display, grid, axes lines, ToolComp, text, etc. When F9 is pressed, a window will appear on the screen with the following settings:

Text toggles the program text on or off. With Text On, the blocks of program are displayed in the bottom half of the screen as they are executed. With Text Off, program blocks will not appear, and the graphics will run faster.

Tool toggles the static tool display on or off. With Tool On, the static tool (as defined in the Tool Page) will be displayed as it is moved through the part program. With Tool Off, the static tool(s) will not appear, and the graphics will run faster. If NO TOOL# is in the program, no tool will be shown, regardless of this setting.

ToolComp allows the operator to set whether the compensated moves (ToolComp Left or Right) in a program will be shown or not, or to show BOTH compensated and non-compensated moves. This allows the operator to compare the non-compensated (programmed) moves with the compensated (offset) moves. Often when a programming error is made, it is due to the path of the compensated part elements. The selections made here do not affect the part program. Selections for ToolComp are detailed below:

a) Ignore sets the ToolComp in the graphics to Off. In this case, the path drawn on the screen will not show the compensated moves (if any) in the program.

b) Use sets the ToolComp in the graphics to On. In this case, the path drawn on the screen will show the compensated moves. It will also show moves that are programmed (without ToolComp), but comparison is not possible until ToolComp is set to Both (see below).
c) Both sets the ToolComp in the graphics to Both. In this case, both the programmed AND the compensated moves are drawn on the screen. The program is actually DRAWN TWICE, first WITHOUT ToolComp, and second WITH ToolComp. In this way, the two paths may be compared to help eliminate programming errors or undercuts with cutter diameter compensation.

Rapid allows the rapid (dotted) lines to be toggled On/Off. This has no effect on the program, only the rapid lines shown in graphics. This can help eliminate screen clutter in long programs with many rapid moves.

Axes toggles the axes display (X,Y,Z) lines On or Off. This shows the axes of the Cartesian (XYZ) Coordinate System.

Grid toggles the 2-D Grid to be turned On or Off. The Grid distance can be set to the desired value by entering a LENGTH. See below.

Grid size allows the GRID size to be set. It is entered in units (inch or mm). When pressed, the Length setting will prompt for a value (1 ; .5 ; .25 ; etc.). When ENTER is pressed, the grid will appear. Note that the GRID cannot be shown in ISO view.

Mode Pressing Mode allows the operator to set the graphics mode to Auto, Block or Motion:
a) Auto will set the graphics mode to Auto. In Auto, the blocks are executed sequentially, and will continue to be executed until the ending block is reached (or if Draw was entered into from Auto or S.Step, until a message (manual tool change, program stop, etc.., is reached).
b) Block will set the graphics mode to Block. In Block, only one block of program is executed for each press of the Start key, whether it is a movement block or not. In this way, the program steps may be reviewed one at a time as they are executed before allowing the program to continue.
c) Motion will set the graphics mode to Motion. In Motion, one MOTION will be executed for each press of the Start key. Blocks without an axis move (i.e; DWELL 3.3) will be executed, but the graphics will continue past those blocks until a motion commanded move is encountered. In this way, motions can be reviewed one at a time, before allowing the program to continue.

Run toggles the "automatic run" function On or Off. With Run On, each time a soft-key from the MAIN soft-key display is

set, the program will automatically re-draw itself. This prevents having to press F3 (Run) on the main display each time a function is set.

Erase toggles the "automatic erase" function On or Off. With F7 On, each time the program is Run, it will first erase the graphics and text area of the Draw screen. If it is Off, the screen WILL NOT be erased when the program is run, but can be erased through the Display window on the main soft-key display.

Start/End N# allows the operator to choose which BLOCKS of the program he/she wishes to view. In this way, the whole program need not be drawn.
For this setting, blocks are referred to as N#'s (N1, N10, N25, etc..). Details on setting Blocks:
a) Start when pressed, allows the operator to key in the beginning N#. This is the block the graphics will START execution from. Keying in -1 (negative 1) will set the beginning block to START, which means the start of the program.
b) End when pressed, allows the operator to key in the ending N#. This is the block the graphics will STOP execution on. Keying in -1 (negative 1) will set the ending block to END, which means the end of the main program.

NOTE: When setting Start and End blocks, ALWAYS use blocks from the MAIN PROGRAM ONLY. The CNC will execute subprograms if they are called from the main program. The END block should never be beyond the end of the main program. The BLOCKS function is NOT possible while using REAL-TIME graphics.

9.8 EXIT: F10

Pressing F10 (Exit) once will turn off Draw mode, twice will exit the CNC from the graphics mode to the previous mode of operation.

Depending on which mode the DRAW mode was entered into from (the EDIT page, or AUTO/S.STEP), the CNC will return to that mode when F10 (EXIT or DRAW) is pressed.

If DRAW was entered into from AUTO or S.STEP, the CNC will continue to run the program (in that mode) after F10 is pressed.

9.9 EDITING WHILE IN DRAW

Pressing F2 (Draw) will DEACTIVATE the DRAW mode, and the

Soft-Key strip will return to the EDIT Soft-Keys.

EDITING IS ONLY POSSIBLE when F2 (DRAW) is "off" (not highlighted).

With F2 off, a program can quickly be edited and re-drawn to check the tool(s) path.

9.10 Notes on DRAW Mode:

1. DRAW mode is an effective method of proving a program before cutting material. It is recommended that all programs be run in Draw mode (SIMULATION: without cutting) before actually machining the part. This can help eliminate mistakes.

2. Active soft-keys will be highlighted.

3. Start and Hold keys on the CNC keyboard have the same effect as the Start and Hold Soft-Keys. Clear also applies.

4. Types of moves shown are as follows:

   Rapid: Dotted line
   Feed: Solid line or arc
   Compensated moves: (solid or dotted, depends on feed or rapid). NOTE: ToolComp TOGGLE must be set to USE or BOTH for compensated moves to be DRAWN. See section 9.7 above.

5. Notes on ENTERING the DRAW MODE:
   If entered from **EDIT** screen: The CNC will draw the part **without** machine movement.

   If entered from **AUTO** or **S.STEP**: The CNC will draw the part **while it is actually machining** the part.

   It is very important to understand this difference, as it will distinguish whether the machine will move or not while running graphics.

6. A program MUST BE LOADED to be run in Real-Time Draw mode.

7. At any time a "pull-up window" is present, if no choice is desired, pressing CLEAR on the CNC keyboard will exit the window.

8. For use of REAL-TIME (Draw while cutting) Draw mode, refer to section 8.6 above.

9. Drilled holes will be shown as a "cylinder". The cylinder will represent the programmed diameter of the tool(s), and the programmed depth of the hole(s).

10.0 USING the FLOPPY DRIVE and RS-232

10.1 USING the FLOPPY DISK DRIVE

The FLOPPY DISK drive is used to:

a) Store programs TO a floppy disk

b) Retrieve programs FROM a floppy disk

c) Update the System Software

The floppy disk drive is located on the computer cabinet, usually on the left side of the machine tool. A FLOPPY DISK must first be inserted into the FLOPPY DRIVE, before it can be used.

To put a disk IN, open the access door to the floppy drive, and GENTLY push the disk into the drive slot. It will pop in easily.

To get a disk OUT, push the small button located near the drive slot. The disk will pop partially out, and can be pulled out.

NOTE: Any floppy disk must first be FORMATTED before it can be used. Some disks are pre-formatted when bought. FORMATTING can be done through the "UTILITY" Soft-Key of the PROGRAM screen. See section 5.10.5 above for FORMAT function.

EXAMPLES: (Assumes the disk is formatted, and in the drive slot).

1. To STORE ONE program onto the floppy disk: While on the PROGRAM screen, place the cursor on the desired program, and press F9 (UTILITY), then select COPY. A window will appear asking which drive to copy to. Select drive **A:** . Drive A is the floppy drive, drive D is the CNC's, where your PART PROGRAMS are stored. The "other" choice may be used to change the program name, as well. DO NOT USE DRIVE C:
   The program will be copied onto the floppy disk. The floppy disk should be removed from the drive, and stored in a safe, dry place, away from anything magnetic.

2. To STORE SEVERAL programs onto the floppy disk: While on the PROGRAM screen, place the cursor on the desired programs, pressing ENTER while on each one. Pressing ENTER will "MARK" the programs. Then press F9 (UTILITY), then select COPY. A window will appear asking which drive to copy to. Select drive **A:** Drive A is the floppy drive, drive D is the CNC's computer. The "other" choice mat be used to change the program name, as well.
   The programs will be copied onto the floppy disk. The floppy disk should be removed from the drive, and stored in a

safe, dry place, away from anything magnetic.

3. To RETRIEVE ONE program FROM the floppy disk: While on the PROGRAM screen, press F7 (LOG), and select drive A: . All the part-programs on drive A: will be shown on the screen. Place the cursor on the desired program and press F9 (UTILITY), then select COPY. A window will appear asking which drive to copy to. Select drive **D:** . Drive A is the floppy drive, drive D is the CNC's computer. The "other" choice may be used to change the program name, as well.

   The program will be copied onto the CNC'c memory. Press F7 (LOG) again, and select drive D: . The program will now appear in the CNC's directory, on the PROGRAM screen. The floppy disk should be removed from the drive, and stored in a safe, dry place, away from anything magnetic.

4. To RETRIEVE SEVERAL programs FROM the floppy disk: While on the PROGRAM screen, press F7 (LOG), and select drive A: . All the part-programs on drive A: will be shown on the screen. Place the cursor on the desired programs pressing ENTER while on each one. Pressing ENTER will "MARK" the programs. Then, press F9 (UTILITY), then select COPY. A window will appear asking which drive to copy to. Select drive **D:** . Drive A is the floppy drive, drive D is the CNC's computer. The "other" choice may be used to change the program name, as well.

   The programs will be copied onto the CNC'c memory. Press F7 (LOG) again, and select drive D: . The programs will now appear in the CNC's directory, on the PROGRAM screen. The floppy disk should be removed from the drive, and stored in a safe, dry place, away from anything magnetic.

10.2 COMMUNICATIONS - RS-232

COMMUNICATIONS, is **RS-232**, and when selected (through the F9-Utility window), the screen will change to the Communications page. Communications is used to transfer programs into or out of the CNC to an offline system, in **ASCII** format. The offline system must be plugged into the RS-232-C port of the computer.

COMMUNICATIONS is accessed through the UTILITY window of the PROGRAM screen. From the Manual mode, press F2, F9, then select Communications.

Before Communication can take place, the parameters of the CNC shown at the top of the Comm screen must match that of the offline system. These Comm parameters can be set by pressing **F5** (Param) of the Comm page soft-keys. The following parameters can be set, and their selections are listed:

Port : Set by builder. Should not be changed. See service manual.

Baudrate: 110, 150, 300, 600, 1200, 2400, 4800, 9600, 19,200.

Parity: None, Odd, Even.

Data Bits: 5, 6, 7, 8.

Stop Bits: 0, 1, 2.

Software Handshake (XON,XOFF): On, Off.

Data Type: ASCII or Binary.

The above parameters can be set by pressing F5 (Param) of the Comm soft-key strip. Only one value for each category can be set. Once the parameters are set, press F10 (Prev), then **F1** (Receive) or **F3** (Send) can be executed:

Receive (F1) will prompt the operator for a program name. The incoming data will be stored in the given program. If the program name given already exists in the PROGRAM screen's directory, a message will appear "Program Already Exists", and will allow the operator to choose a NEW program name (Rename), or OVERWRITE the existing program. If OVERWRITE is chosen, the existing program is erased from memory, and the incoming program will have its program name. The ASCII table may be used to give a program name. See section 5.3 above for use of the ASCII table.

Once the program name is keyed in and ENTER is pressed, the CNC will display "READY TO RECEIVE", and transmission from the offline system can begin.

Send (F3) will prompt the operator with whichever existing program in the CNC memory that the CURSOR IS ON, and ask if this is the program he/she wishes to send. If it is indeed the program to be sent, pressing F1 (Select) will begin the transmission to the offline system. F3 (Change) will allow the operator to select a different program from CNC memory to be sent (ASCII table may be used to give name. See section 5.3 above for use of ASCII table).

The offline system should be ready and waiting before F1 (SELECT) is pressed, as transmission out of the CNC will begin immediately.

DNC (F4) is **D**irect **N**umerical **C**control. See section 14 below.

NOTES:

1. See your service manual for port designations and other technical information regarding RS-232.
2. During Send or Receive, soft-key F4 (Text) appears. This allows the operator to view the data as it is sent/received. F1 (Hold) and F2 (Resume) will halt and resume transmission when pressed.
3. F10 (Prev) should be pressed when transmission is complete.

Also, press F10 (Exit) from the Comm screen to return to the PROGRAM page.

4. **TestLnk (F7)** allows testing of the RS-232 link.

10.3 G-CODE CONVERTER

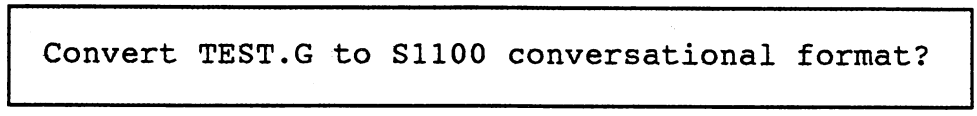
The G-CODE converter is accessed through the UTILITY (F9) window of the PROGRAM directory screen, item labeled **"Gcode To S1100"**. It can be used to "translate" a G-coded program into the Series 1100's machinist's language. It is designed specifically to translate the Anilam Series 1400's code, but will also translate G-code from any CNC control that uses industry-standard code. Since "canned cycles" differ from one control manufacturer to another, only Anilam's Series 1400 CNC's canned cycles are supported.

10.3.1 USING the CONVERTER

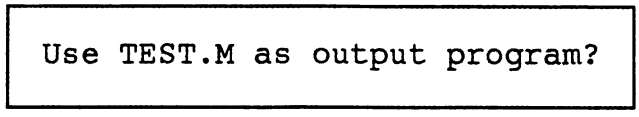
To convert a G-code program, the program must be in the 1100's USER directory. This can be done by copying the G-code program from a floppy disk, or downloading via RS-232. In either case, the G-code program must NOT have an extension of .M. The file extension .M is reserved just for S1100 "machinist's language" format programs. The G-code program SHOULD have a file extension of .G.

You will need to change the program display mode to see programs that do not have a .M extension. This is done by pressing the **Display** softkey (F8) in the **Program** page. Once you see the program you want to convert (the .G program), move the cursor onto that program.

The conversion process is accessed through the **Utility** softkey in the **Program** page. Press **Utility** (F9) and select the entry **"G-code To S1100"**. Assuming you have selected a program named TEST.G for conversion, the following screen will display:



The **Yes, No** and **Cancel** softkey set is displayed. Selecting **No** or **Cancel** stops the conversion. Selecting Yes will display the following screen:



For the new converted program, the system will create a new program with the same name, but with a .M extension. Thus, TEST.G will get converted into TEST.M. Selecting **Cancel** stops the conversion. Selecting **No** allows the user to enter another program name. Selecting **Yes** continues the conversion with the current program name. If the output program already exist the screen will show the following:

```
Output program TEST.M already exists.

Overwrite program?
```

Selecting **No** or **Cancel** stops the conversion. Selecting **Yes** will continue the conversion and overwrite the existing program.

While the G-code program is being converted a status screen shows the progress of the conversion:

```
S1100 PROGRAM CONVERSION UTILITY

    Input program: TEST.G
   Output program: TEST.M

  Percent complete: 55.2%
 Input block count: 221
Output block count: 235

      Error count: 8

-------------------------------

      Press F9 To Cancel
```

The status screen shows:

```
     Input program: program being converted
    Output program: program being created

  Percent complete: how much of the input has been
       processed
 Input block count: how many lines of G-code have been
      processed
Output block count: how many lines of S1100 code have been
      created

        Error count: how many blocks could not be
      converted
```

At any time during the conversion **Cancel** can be pressed to stop the conversion. If **Cancel** is pressed the output program will exist up to the point when **Cancel** was pressed.

The size of the program that can be converted is only limited by the amount of program (memory) space available. If the error **Disk Full!** appears, the operator can try running the converter from floppy and saving the program on the USER drive (or run from the USER drive and save the program on a floppy).

To create more space you can move programs to a floppy, and/or delete *.S.

10.3.2 WHAT GETS CONVERTED?

The following table shows what G-code addresses get converted:

G CODE	S1100 Conversational
G90/1	Dim Abs/Incr
G17/8/9	Plane XY/XZ/YZ
G75	FramePock (Series 1400)
G76	HolePock (S1400)
G77	CircPock (S1400)
G78	RectPock (S1400)
G79	DrillBHole (S1400)
G80	DrillOff

G CODE	S1100 Conversational
G81	BasicDrill (S1400)
G82	*Not Supported*
G83	PeckDrill (S1400)
G84	*Not Supported*
G85	BoreDrill (S1400)
G86	*Not Supported*
G87	ChipBreak (S1400)
G88	*Not Supported*
G89	*Not Supported*
G179	DrillPatt (S1400)
Tt	Tool# t
G4 Tt	Dwell t
Oo	Sub o
M99	EndSub
M98 Pp	Call p
M2	EndMain
Mm	Mcode m
Ff	Feed f
G0/1	Line with XY/XR/YR/RC format
G2/3	Arc with XYIJ/XYR/IJC format
G5	Ellipse with XYIJABL format (S1400)
G6	Spiral with XYIJL format (S1400)
G40/1/2	ToolComp Off/Left/Right (part of next Rapid, or Line, block)

Any other address not mentioned above will generate an error.

NOTE: Something to be careful with are circle centers. Traditionally, G-code systems support either incremental [S1400 or Fanuc (tm)] or absolute circle centers, not "modal" circle-centers. If the operator converts a program that has incremental circle centers but his S1100 is set to absolute, his program will not run correctly. The operator must set his circle center parameter to match those of the program (or, change the off-line post-processor

to output modal circle centers). Note that using the XYR format will not cause any problems (ie; no circle center). For most offline CAD/CAM systems, the post-processor can be set to generate arcs with XYR format or centers that match the S1100 setting.

10.3.3 WHEN ERRORS are GENERATED

When an error is detected the converter will put the guilty block as a "comment" in the S1100 program. The comment will have the format:

* * Can't convert "xxxxxxxx"

where "xxxxxxx" would be the actual text it cannot convert.

When this happens, **it is important that the operator fix** the error by looking at the suspect block, and converting it 'manually' by deciphering the meaning of the G-code block, and adding a new block with the machinist's language equivalent.

11.0 USING the "OFFLINE" KEYBOARD

11.1 WHAT IS an "OFFLINE" KEYBOARD ?

An "offline" keyboard is simply an additional keyboard that may be used **IN PLACE OF THE CNC KEYBOARD**. It is commonly known as a QWERTY keyboard, because the first six (6) letters are the letters Q W E R T Y.

In some cases of personal preference, the operator may choose to use either keyboard. It is possible with the Series 1100 CNC control.

All functions of the CNC keyboard are accessible with the offline keyboard with the exception using the HandWheel, and the E-Stop. See section 11.3 below for usage.

11.2 INSTALLATION

The "offline" keyboard is installed by placing its plug into the socket located in the rear of the CNC CONSOLE. The E-Stop should be pushed in before plugging in the offline keyboard.

The TOGGLE switch located near the socket must be switched into the opposite position, to activate the offline keyboard. Wait 5 seconds before pressing any keys, then press F10. Once activated, the CNC keyboard will no longer be active.

To de-install the offline keyboard, push the E-Stop in, unplug the keyboard, then switch the TOGGLE switch to the opposite position. Wait 5 seconds before pressing F10.

11.3 USAGE

Please read section 11.2 above before proceeding.

The chart below shows the CNC keyboard equivalents on the "offline" keyboard:

1100 CNC keyboard	Offline keyboard	Description
CLEAR	Alt-C	Clear message/MDI
START	Alt-S	Start program/MDI
HOLD	Alt-H	Hold program/MDI
ABS/INCR	Alt-E	Toggle ABS/INCR
+/-	-	Tollge Arc CW/CCW and ToolComp L/R/Off
SERVO RESET	Alt-R	Reset servo motors
SPINDLE OFF	Alt-O	Stop spindle
SPINDLE ON FWD	Alt-F	Start spindle FWD
SPINDLE ON REV	Alt-G	Start spindle REV

FEEDRATE KNOB	Alt-0 thru 9, and A,B,D	Set feedrate override knob.
JOG SELECT:		
RAPID	Alt-F1	Set Rapid
FEED	Alt-F2	Set Feed
JOG 100	Alt-F3	Set Jog 100
JOG 10	Alt-F4	Set Jog 10
JOG 1	Alt-F5	Set Jog 1

Moving the axes:

X+	Alt-X	Jog X positive
X-	Alt-K	Jog X negative
Y+	Alt-Y	Jog Y positive
Y-	Alt-L	Jog Y negative
Z+	Alt-Z	Jog Z positive
Z-	Alt-N	Jog Z negative
stop X,Y,Z movement	Alt-T	stop any movement

NOTES:

1. When using the offline keyboard, program names may include letters and numerals, and also the characters !@#$%^&(){}
2. While in the PROGRAM directory, if the first letter or number of a program is pressed, the cursor will automatically go to the first program beginning with that character.
3. The use of drives and paths while copying is possible with the offline keyboard.

When **SHIFT** key is pressed (only with "offline keyboard") while in the TOOL Page, secondary soft-key functions are available:

SHIFT F3 : Will CLEAR the entire TOOL TABLE, after confirmation.

SHIFT F10 : Will allow exit of the Tool Page WITHOUT SAVING any changes that have been made since entering the Tool Page.

When **SHIFT** key is pressed (only with "offline keyboard") while in the PROGRAM screen, secondary soft-key functions are available:

SHIFT F2 : Will allow creation of a subdirectory.

SHIFT F3 : Will allow deletion of any file(s) in the memory. A path or wildcard should be given (i.e; *.S, or d:\user*.s

12.0 TEACH MODE

12.1 WHAT is "TEACH" MODE ?

TEACH mode is a special mode that allows the operator to put the ACTUAL MACHINE POSITION into the program as a block. Whatever is in the X-Y-Z display can be "digitized" into the part program.

It may be useful if an unknown bore location is indicated, and the operator wishes to make its position part of the program.

or

A part without a blueprint must be reproduced, the operator can manually move the machine axes around the part, taking points as he/she deems necessary.

12.2 ENTERING into TEACH MODE

TEACH mode accessed through the EDIT mode. A program must be created (unless using an existing program), and the EDIT mode must be accessed.

After pressing EDIT, see Soft-Key F1 is called TEACH. When Pressed, the CNC will be in TEACH MODE, and the TEACH soft-key will be highlighted.

The CRT-SCREEN will be similar to EDIT mode, except the AXIS POSITIONS and JOG STATUS will be displayed in the top-half of the screen. The Soft-Keys of TEACH are described below in section 12.3

12.2.1 EXITING TEACH MODE

Pressing F1 (TEACH) will exit the teach mode, and return the CNC to the EDIT mode. Pressing F10 (EXIT) will return the CRT-screen to the PROGRAM directory screen.

12.3 TAKING IN DATA USING TEACH MODE

To enter the axes positions into the program using TEACH mode, put the CNC in TEACH mode as described above. Once in TEACH mode, the Soft-Keys will change as follows:

F1 **TEACH** remains highlighted, telling the operator he/she is in the TEACH mode.

F2 **Rapid**, when pressed, will input a Rapid block into the program, with the current X-Y-Z position. The cursor advances, waiting for the next position to be input using F2, F3 or F4.

F3 **Line**, when pressed, will input a Line block into the program, with the current X-Y-Z position. The cursor advances, waiting for the next position to be input using F2, F3 or F4.

F5 **Modal**, when pressed, will input a block into the program, with the current X-Y-Z position, but without the Rapid or Line definition. This is known as a "MODAL" move. When the program is later run in AUTO or S.STEP, the CNC's move to this position will be in whatever mode (rapid or line-feed) that was previously set. The cursor advances, waiting for the next position to be input using F2, F3 or F4.

NOTE: Soft-key F8 (HandWhl) may appear if installed. The HANDWHEEL mode may be used while in TEACH Mode. HandWheel Mode is described in section 4.11 above.

13.0 GLOSSARY / INDEX

13.1 GENERAL

This chapter describes the terminology used within this programming/operations manual and the Series 1100 CNC control. The descriptions are general, and only meant to refer the operator to the appropriate section(s) of the manual, so a more in-depth understanding of the term can be reached.

The definitions given are only as used in this manual and the Series 1100 CNC system. They are listed in alphabetical order and include the section(s) of the manual where the term may be found.

EXAMPLE: 6.7.3 = Chapter 6, section 7.3

13.2 GLOSSARY OF TERMS

A

A: The drive designation for floppy disks. 5.8

Absolute: A dimension type given from a fixed reference point. Opposite of incremental. 3.3

Angle: Expressed in degrees, usually with radius definition. Used in the Polar coordinate system. 3.4

Arc: A circular path of motion. 6.7

Arrow keys: Keys which move the screen cursor. Located on the CNC keyboard. 2.1

ASCII: A table used to access all the letters of the alphabet, used for creating, renaming programs. 5.2

Auto: CNC mode in which piece parts are produced. Control executes blocks sequentially and continually. 8.3

B

Baud rate: Speed at which characters are sent/received through the RS-232 port. 10.2

Begin: An editing function to move to the beginning of a program. 6.13

Bits: A computer term that describes the length of a character. A number of bits make up a byte, usually 7 or 8. Needed for B.P.C. in RS-232 settings. 10.2

Block: A single line of a program. It contains instructions to the CNC. A basic element of the program. 6.0

Bolt-Hole: A drilling function that allows a bolt-hole pattern to be easily defined. 6.5

C

C: The computer's disk drive designation. Not for Use by the operator. 5.8

Calculator: A mode of the CNC where answers may be found to math and trig problems. 7.0

Call: To execute a subprogram. 6.10

Cal-Z: Used to automatically input a Tool Length Offset into the Tool Page. 3.5

Cancel: Used to abort a programming process. 6.0

Canned cycle: A permanent program macro in the CNC memory accessed by the programmer to machine common shapes/moves with minimal input. 6.5, 6.6

CheckDisk: Used to view information about the computer's storage area. 5.10

Circular: Motion of an arc. 6.7

Comm: Short for Communications or RS-232. A method of sending programs in/out of the CNC's computer. 10.2

Comment: A "note" to the operator that may be placed in a program. 6.15

Compensation: Used for tool diameter and length. When active, CNC uses diameter and length offsets applied to programmed moves. 3.5

Coordinate: Describes a point in space. In Cartesian coordinate system, described by X, Y and Z. In Polar coordinate system, described by radius and angle. 3.2, 3.4

Corner rounding: A function that will automatically round programmed move intersections. 3.6

Create: Used to create a new part program. 5.3

CRT: Cathode Ray Tube. The CNC's display screen.

Cursor: A pointer that shows the location of input characters or selections to be made.

D

Delete: To remove blocks or programs from CNC memory. 5.4, 6.15

Default: The initial condition of a parameter or variable when the CNC is first turned on. III 5

Diagnostics: A series of routines to help troubleshoot problems related to software/hardware. See OEM manual.

Directory: An area of memory on a hard or floppy disk. Helps organize data for easier access.

Display: Used to switch the program display to 4 different types of views. 5.9

Double: A function of Draw graphics that doubles the size of the drawn moves on the screen. 9.6

Draw: Graphics mode. Used to prove-out part programs. Shows tool paths. Can also be used while cutting. 9.0

Drill: A window where drilling functions are selected. 6.5

Dwell: A programmed temporary stop. Defined in seconds. Also used in some drilling/boring cycles. 6.7

E

Edit: A mode of the CNC where programs are entered into the memory. Also means to change an existing program. 5.5, 6.15

Ellipse: A type of movement. 6.7

End: An editing function to move to the end of a program. 6.13

Enter: A key on the CNC keyboard used to input a block into memory. When pressed, the cursor advances. 6.0

Erase: A function of Draw graphics used to clear the screen of

text and drawn moves. 9.6

E-Stop: Emergency stop. Button (usually large red) that disables the servo motors and all M-functions. 2.1

Exit: To leave a mode or screen of the CNC. It will bring the screen to the previous screen. 5.11, 6.14, 9.8

Extension: A suffix used for file organization. Each "file" has a Name and an Extension (FILENAME.M). The extension can be up to 3 characters long. 5.9

F

Feed: Also called feedrate, it is the velocity of axis travel while in linear or circular interpolation (not rapid). 6.7

File: A program in the memory of the computer is a file. Each file has a name. User files have an extension of .M

Find: A function of the Tool Page which will search a particular Tool #. 3.5

Finish Height: A Z-reference dimension used in canned cycles. 6.5, 6.6

Fit: A function of DRAW and CALC modes used to auto-fit the moves onto the screen. 9.6, 7.4

Floppy: Refers to a medium of data storage. Floppy disk. Floppy drive is where the disk is inserted into. 5.8, 5.10

Format: Refers to "formatting" a disk. 5.10

Function key: Also referred to as soft-key or Fn key, these keys provide various functions. Found directly under the CRT. 2.2

G

G-code: Another type of programming language. 5.10

Graphics: A visual representation of programmed moves. Also referred to as Draw. 9.0

Grid: A display of dots or lines used in Draw graphics to help the user get a feel for the actual size of a drawn part. 9.0

H

Half: A function of Draw graphics which halves the size of the drawn piece-part. 9.0

Halt: Halted; To temporarily stop execution of a movement or program by the Hold key. 8.4

Handshake: A parameter used during RS-232 transmission. 10.2

Hold: A key used to temporarily halt a program. 2.1

Home: A function that sends the machine axes to a fixed position. The position is set by parameter, and is referred to as Machine Home. 6.7

Hot-Key: Used to quicken the programming process. 6.17

I

Icon: A graphical representation of a type of Geometry input in the Geometry Calculator. 7.4

Incremental: Distance from point to point. Incremental mode uses position in reference to previous position. Opposite of Absolute. 3.3

In-Position: A check the CNC makes to determine if it has reached target. See OEM manual.

Input: Data that is entered into the CNC memory. 6.0

Insert: To add a block to a program. 6.15

Interpolation: Movement of axes using a pre-determined rate. Used for linear feed and circular feed.

IPM, MMPM: Abbreviations for various types of feedrates: IPM = Inch per minute ; MMPM = Millimeter per minute.

J

Jog: To move the machine axis via the Manual panel. Done in rapid/feed continuous, and increments. 4.3

K

Keyboard: The alpha-numeric key section on the CNC used for data entry. Also auxiliary keyboard. 2.1, 11.0

L

Line: A type of movement done in a straight line or angle. 6.7

List: To show the contents of a program. 5.6

Load: To move a program into the working memory of the CNC. 5.7

Loop: To instruct the CNC to repeat a move or series of moves. 6.11

M

M-code: A coded instruction to the control or machine to perform a specific function. 6.10

Manual: A mode of CNC used to manually command the machine. 4.0

Mark: To highlight programs for copying, deleting, etc. 5.0

MDI: A special mode of the CNC. 4.6, 4.7

Message: An instruction, or error, alerting the operator to perform a task or notify of an error. 4.8

Misc: A soft-key used for miscellaneous editing functions. 6.13

Mold rotation: A cycle in the CNC which rotates subroutines around an axis to produce a mold core or cavity. 3 axis only. 6.6

More: To select MORE milling functions, or MORE RAPID, LINE or ARC formats. 6.7

Motion: A form of Single-Step mode. 8.2, 9.7

N

Nesting: When a subprogram is called from another subprogram. The second is "nested" to a level of two deep. 6.10

O

Offline keyboard: A secondary keyboard, usually of the typewriter (Qwerty) type which can be plugged into the CNC and used instead of 11.0

Override: To adjust feed or rapid rate(s) by use of a rotary switch. 4.9

OEM: Original Equipment Manufacturer. The builder/fitter of the machine tool and/or CNC control. See OEM manual

P

Parameter: A setting in the CNC control. Usually refers to a permanent setting changed only by the OEM. Also term for a single word (item) in a macro. VI 4.7

Parity: A setting used in RS-232 communications. Usually Even. Odd, or None. 10.2

Paste: What the RECALL function of the calculators do. 7.5

Pattern: A drilling function that allows a pattern of holes to be easily defined. 6.5

Plane: One of three flat imaginary surfaces consisting of two of the 3 basic axes. 3 axis only. 6.7

Pocket: A term used to describe a cavital area milled into a piece-part. Usually circular or rectangular. Canned cycles are provided for ease of programming. 6.6

Port: An I/O device within the CNC computer. See OEM manual.

Previous: A soft-key function (usually F9) which brings the CRT screen back one screen, to the previous screen

Print: To download a file(s) to a printer. 5.10

Program: A file in the CNC memory. 5.0

Q

Quit: Used to exit a program without saving the changes made to it in the current editing session. 6.13

R

Radius: A parameter of a circle or arc, radius determines the incremental distance from center to circumference. 6.7

Rapid: A commanded movement done at the machine's fastest rate. Used for positioning only. 4.3, 6.7

Receive: To cause a program to be received from an offline device through the RS-232 port on the computer. 10.2

Rename: To change the name of a file (program). 5.10

Rotate: To index a plane of the Cartesian (XYZ) coordinate system angularly around a point. 6.12

RS-232: A method of sending/receiving programs out/in to the CNC control. Accessed by Comm soft-key. 10.2

S

Save: To input a block of into the program. 6.0

Scale: Scaling is used to enlarge or shrink dimensions in a program. 6.12

Scroll: To move the cursor up or down on the screen.

Search: To find a block number while in the Auto/S.Step modes. 8.0

Send: To cause a program to be sent to an offline device through the RS-232 port on the computer. 10.2

Servo: A motor or motor system used to drive the machine axes. See OEM manual.

Skip: Allows selected blocks of the program to be skipped. 6.15

Servo Reset: A key used to turn ON the servo system. 2.1

Set-up Utility: A part of the software that allows configuration of certain aspects of the CNC control. See OEM manual.

Set Zero: Used to shift the absolute zero-point (part zero) from one point on the axes grid to another. 6.7

Shift: A key on the offline keyboard that allows secondary functions of the soft-keys to be performed.

Single-step: (S.Step) CNC mode in which program steps are executed one at a time, for each press of the Start key. 8.2

Soft-key: Function key. A horizontal strip of keys directly below the CRT that provide functions. 2.2

Software: The internal code in the computer.

Spiral: A type of movement. 6.7

Start: A key used to begin or continue the execution of a program. 2.1

Start Height: A Z-reference dimension used in canned cycles. 6.5, 6.6

Status: The condition of the CNC. 2.3

Stepover: Refers to the width of cut on pocketing canned cycles. 6.6

Subdirectory: A directory within another directory. To create one, press SHIFT-F2 (using offline keyboard) from the PROGRAM screen.

Subprogram: A separate (secondary) program included into a main program file. 6.10

System Info: Used to view info on the CNC system. 5.10

T

Teach: To input axes positions and various data from the Edit mode into a program file. 12.0

Text: A toggle to turn ON or OFF the program text while in DRAW mode. 9.7

Tool Offset: Describes the diameter (or radius) and the length offset of a tool. 3.5

Tool Length Offset: Usually describes the distance and direction from the bottom of the tool to the top of the workpiece. 3.5

U

Unmark: To de-highlight a program or section of program. 6.0

Undelete: Used to retrieve a program that was accidentally deleted. 5.10

Unit: Used to switch to INCH or MM programming. 6.7

Utility: Util; A part of the File page function, allows file-handling functions to be performed. 5.10

V

View: A planar (XY-XZ-YZ) view used in Draw graphics. 9.5

W

Window: A function of DRAW and CALC modes, used to zoom-up on an area. 9.6, 7.4

14.0 DNC FUNCTION

14.1 GENERAL

This chapter describes the DNC function. DNC is an acronym for Direct Numerical Control (otherwise known as Continuous Downloading). This function is only used when a program that is larger than the CNC's available memory, is necessary to be run.

It is possible to run a program that is larger than the CNC's available memory using DNC.

14.2 Before Using DNC

Before using the DNC function, all set-up procedures must be completed, such as setting Absolute pert-zero and Tool Page entries. Once the DNC function is started, it may be aborted by pressing F4 (DNC), then F10 (Exit) twice. This will revert the CNC back into Manual Mode.

14.3 Accessing and Running DNC

DNC is accessed through the UTILITY (F9) soft-key while in the PROGRAM screen (program directory). After pressing UTILITY, select COMMUNICATIONS. This is the normal RS-232 Communications mode detailed in section 10.2 above.

Soft-key F4 is labeled DNC. To begin the DNC process, press F4, then press F1 receive. The CNC is now ready to begin receiving the program from an offline (host) computer. **NOTE**: Be sure the transmission parameters are set correctly (such as baud rate, parity, etc.). These parameters are detailed in section 10.2 above.

Once the transmission is started from offline, the CNC will act as usual, with Tool Changes, Single-Step Mode, and Real-Time Draw operations as described in previous sections.

To abort the DNC process at any time, press F4 (DNC), then F10 (Exit) twice. This will leave the CNC in MANUAL mode.

14.4 DNC Special Parameters

Special parameters exist for the DNC mode, in addition to the normal transmission parameters described in section 10.2 above. These are set from the factory, and should not need to be altered. If necessary, see the Series 1100 interface manual for details.

15.0 OFFLINE SOFTWARE OPTION

15.1 What is the Offline Software Option ?

The "Offline Software Option" is an option the customer may purchase, that allows the Series 1100 CNC control's software to be run on a desk-top personal computer (PC). A special software security device is included to make this possible.

This can be a handy item for creating and proving programs in a more comfortable environment than the shop floor. The Offline Software has all the functions of the CNC's software except motion control. It simulates machine motion. It may also be used to familiarize new operators or employees with the Series 1100 system.

15.2 Computer Requirements

Before loading the Offline Software option onto a desk-top PC, be certain the following PC requirements have been met:

- Minimum of 386-SX motherboard
- Math co-processor (except DX models)
- Minimum 4 Mb RAM
- Minimum VGA graphics
- 3-1/2" 1.44 Mb floppy disk drive
- At least 2 Mb hard disk drive space

With the above requirements met, the computer will run the Series 1100 Offline Software.

15.3 Loading the Offline Software

To load the "Offline Software" option onto a desk-top (or lap-top, notebook) PC, take the following steps:

1. Get to the DOS prompt **C:**
2. Place the Offline disk in the 3-1/2" drive
3. Type **A:INSTALL** then press **ENTER** (or B:INSTALL, depending on the floppy disk drive designation).
4. The installation utility will prompt the operator with certain questions, that are self explanatory.
5. Then, to start up the Series 1100 Offline software, be sure the computer is at the DOS prompt **C:\S1100** then type **PS11** then press **ENTER**. The S1100 software will begin. Follow the programming - operation manual for all details.
6. To EXIT the software, from the MANUAL Mode, press F10 (Exit), press ESCAPE, key in password (X2) then press ENTER.

15.4 Keystrikes for Offline Software

Since a CNC keyboard cannot be present on a desk-top PC, the CNC keys are simulated through a special series of keystrikes. These functions such as START, HOLD, CLEAR and others are executed through the "ALT" key with another character on the PC's keyboard.

Hold the ALT key and press S is equal to pressing START. ALT-C is CLEAR, ALT-H is HOLD. See section 11.3 above for full description of Offline Keyboard equivalents.

ADDENDUM TO SERIES 1100 PROGRAMMING MANUAL

P/N 70000300
November 30, 1993

CONTENTS:

1. "Hot-Key" function and Use:

1.1 What are "Hot-Keys" ?

The term "Hot-Key" is used in cases where a software function can be accessed by use of a "short-cut" while programming. These short cuts can save time and effort while keying in a part program.

Normally, in the Series 1100 Control, moves such as RAPID, LINE, ARC, and other functions such as FEED (feedrate) and TOOL are programmed by selecting a function key (F1 through F10) while in the EDIT mode.

By using a short cut ("Hot-Key"), these functions can be accessed WITHOUT ever pressing a Function Key (F1 through F10). "Hot-Keys" may be used in either EDIT or MANUAL modes. See below.

The keys used for "Hot-Key" functions are the NUMBER KEYS, number 1 through number 9 on the Series 1100 keyboard.

1.2 Functions possible through "Hot-Keys".

The following functions can be accessed by using a NUMBER KEY on the Series 1100 keyboard (number 1 through number 9):

Number Key:	Function:
1	RAPID move
2	LINE move
3	ARC move
4	FEED rate programming
5	TOOL activate / deactivate
6	M-CODE programming
7	UNIT select (inch or mm)
8	DWELL (timed dwell)
9	PLANE select (3-axis only)

Later versions of the Series 1100 keyboard show these functions in small print on each of the number keys.

1.3 Use in EDIT Mode.

While in the program EDIT mode (while keying in a part program), the "Hot-Keys" can be used freely. The functions are also accessible through the normal Function-Keys located directly below the CRT-Screen.

Any time while in EDIT mode, the above functions can be programmed by simply pressing the corresponding NUMBER key on the Series 1100 keyboard. Once a function is selected, it is programmed and "Saved" as usual.

The "Hot-Keys" may also be used in MDI mode.

1.4 Use in MANUAL Mode.

The "Hot-Key" functions may also be used while in MANUAL Mode. This can be useful in programming AND executing single-moves while in the manual mode, such as LINE moves, ARC moves, and FEED rate programming.

While in MANUAL Mode, select a function through use of a "Hot-Key", fill in the blanks, and SAVE. The Series 1100 will show a message and ask for the START key to be pressed. When pressed, the START key will cause the machine to execute the move programmed such as RAPID, LINE or ARC, or the function programmed such as FEED, TOOL, or M-CODE.

These moves and functions can be useful if squaring up a part or making a single cut while in Manual Mode, without making a program to do it. They are also useful during the set-up of the workpiece.

NOTE: When programming RAPID, LINE or ARC through a "Hot-Key" in MANUAL Mode, CornerRad and ToolComp are not allowed. These functions are available through the "Hot-Keys" in the EDIT and MDI modes.

Reminder...the Series 1100's MDI mode is also useful for setting up workpieces and / or making single cuts. See programming manual for full details on MDI mode.

2. DNC (Direct Numerical Control):

2.1 Overview: What is DNC ?

A typical DNC feature is designed to allow program execution without the program residing in the control's memory. This allows programs to be executed which are larger than the CNC's memory capacity. In such a case the program must be sent via RS-232 (Communications) from an offline system, which could be a computer, another CNC or any other device capable of RS-232

communications. DNC is also used in custom applications where a special device is attached to the CNC and the device needs to do something after the CNC executes each block. DNC mode in the Series 1100 provides DNC capability in such a way that both needs described above can be achieved.

2.2 Set-Up of the DNC Parameters.

Certain parameters MUST be set before the DNC mode can be used. Please check the parameters before using DNC mode. The following setup screen is available from the Series 1100's "Set-Up Parameters". The option of Set-Up occurs each time the CNC is turned on, before entering into the CNC's software. Within "Set-Up", the selection "Machine / Installation" will show a sub-selection of "DNC Set-Up". When selected, DNC Set-Up will show the following screen:

DNC Setup	
1. Execution Mode	Buffered/Drip Feed
2. Buffer size (in buffered mode)...	16K/32K/64K/128K/Max
3. Use DNC macro before block	Yes/No
4. DNC macro number	Onnnn [100-32000]
5. Ignore G41/2, G59 and Blueprint .	Yes/No
6. Wait for start	No/First/Every

2.2.1 Execution Mode

This parameter tells the CNC in what mode to execute the data being received via DNC. **Buffered** means that the buffer size specified in item 2 must be filled before execution of the first buffer of data begins. In this case the CNC will wait an amount of time proportional to the size of the buffer selected in item 2. When the initial buffer is full it will be transferred to the background for execution (execution buffer). While the execution buffer is executing, a second buffer, the swap buffer, is also being filled up. When all the data in the execution buffer is processed, the contents of the swap buffer is copied into the execution buffer. This should provide smooth transitions from one buffer to the next, causing no machine hesitations.

Drip feed means that each block will be executed as it is received (ie: no buffering). There will be no initial delay. However, there will probably be a slight hesitation from block to block (depending on the block just executed). The default setting for execution mode is **Buffered.**

2.2.2 Buffer size (in buffered mode)

This parameter will allow the user to specify the amount of memory that should be reserved for DNC in **Buffered** mode. The choices available will be:

16K
32K
64K
128K
Max

where **Max** means that the control will intelligently estimate what is the maximum amount of memory that can be allocated. Depending on the size of program being DNC'ed and the amount of RAM (not CMOS memory) available on the CNC, choosing **Max** could allow the entire program to be sent over before execution begins. The default setting is **32K.**

*** * NOTE * ***

For systems with a hard drive it will be more efficient if the program is sent over via RS232 in its entirety and then executed in normal Auto/S.Step mode.

2.2.3 Use DNC macro before block

This will allow the user to specify if he wants the macro given in item 4 to be executed *before* every block. The default setting is **No.**

2.2.4 DNC macro number

This will allow the user to specify a macro number to be ran at the end of each block. The macro should be stored in the **User macro file** specified in the **Control Software Setup.** Remember that the user macro file can be located in the system or user directory. The macro number must be between 100 and 32000. This capability would be very useful in custom applications where the user wants to do something after every block. Notice that to have the macro execute item 3 must be set to **Yes.** The default macro number is **100.**

2.2.5 Ignore G41/2, G59 and Blueprint

This will allow the user to enable/disable processing for "ToolComp" (tool radius compensation) and "CornerRad" (corner rounding). This entry should be set to **Yes** unless it is absolutely required. If it is set to **No**, even if the program does not contain either of these functions, the CNC must waste background cycles looking for them, resulting in slower machining / processing. Most offline Cad/Cam systems will output compensated points, so the Series 1100's ToolComp and CornerRad functions are not necessary in this case. The default setting is **Yes.**

2.2.6 Wait for start

This allows the user to specify if he wants the control to wait for a START key before execution or not. Selecting **No** will allow the DNC data to execute as soon as it is available. Selecting **First** will require the START key to be pressed before execution of the first block begins. Selecting **Every** will require the START key to be pressed before execution of every block in Drip Feed mode. In buffered mode the user can switch to Single Step to achieve the same effect. The Default setting is **First.**

NOTE: The Series 1100 control will stop on Tool Change blocks, program stop blocks, etcetera, as normally. The parameter described above only applies to the beginning of the program (No, First settings).

2.3 DNC Usage:

DNC mode is accessed from the user communication page. This allows the user to verify his communication parameters before starting the actual DNC. While in the Communications Screen, soft-key **F4** is labeled **DNC** (F4). This soft-key is a toggle that affects how the **Receive** function works. The servos must be on when the **DNC** softkey is pressed, otherwise, an error message will be given and the control will not go into DNC mode.

When the **DNC** softkey is OFF (not highlighted), pressing the **Receive** softkey will access the normal Recieve mode. When the **DNC** softkey is ON (highlighted) pressing the **Receive** softkey will engage DNC mode. At this point the user will be prompted to start sending data, as in:

"Start sending data from offline system."

The rest depends on how DNC was configured:

If **Execution mode** is set to **Buffered** the CNC will wait until the specified buffer is full. Once the buffer is full data will start executing or the CNC will wait for a START key depending on how the **Wait for START** option was configured. Once this initial buffer is executing the CNC will start filling a secondary (swap) buffer in background mode. Depending on the program, if the execution buffer is emptied much faster than the swap buffer can be filled up, eventually there will be hesitations. This situation can be alleviated by using a bigger buffer or adding more memory.

If **Execution mode** is set to **Drip Feed** the CNC will prepare a block for execution as soon as it is received. Once the block is ready it will start executing or the CNC will wait for a START key depending on how the **Wait for START** option was configured.

If **Use DNC macro at end of block** was set to **Yes** then the corresponding macro will be executed before execution of each block.

Anytime there is data executing in DNC mode, softkey set similar to the one in Auto/S.Step will be displayed. This will allow the user to switch between Auto or S.Step as necessary. To indicate

that the CNC is running in DNC mode the **DNC** softkey will be highlighted. To cancel DNC press the DNC softkey again to turn it off. This will restore the CNC back into the communications screen.

The data being received MUST be in the native format of the control (S1100 conversational format).

Any errors encountered during the download will cause DNC to stop. At this stage the user must exit DNC and correct the error before attempting DNC again.

2.4 Display

The incoming text will not be shown, so a ticker like the one displayed during **Receive** mode will be shown. The area just above the softkeys will display Total blocks received, % of Receive buffer full, and % of Execution buffer done. The program text and other data will show just like in Auto/S.Step.

.....END.....

Summary:

This document outlines changes to the 1100 Series 2 & 3 Axis Control Software made since the distribution of current programming manual. This addendum supersedes all earlier addendums to these manuals.

Contents:

NOTE: Holders of Addendum Version 3.10A are advised there are significant changes made to the Face Pocket cycle.

Edit Screen (Added Tool Hot Key)

Added use of keypad hot key to insert a tool number block into a program. With program editor active, pressing the #5 keypad key allows operator to insert a tool number block into the program. Block activates corresponding tool diameter and length offsets.

NOTE: This feature is also available when editing in the MDI mode.

Tool Page (Softkey Label Change)

Find softkey was (F7), moved to (F4).

Calib Z softkey was (F9), moved to (F8).

Draw Screen (Softkey Label Change)

Draw screen (F2) softkey label was changed from **Block** to **S.Step**, operation is unchanged, (advances the program one block at a time).

Draw Screen (Improved Popup Menus)

Inserted additional **Draw** screen, **Parm**, (F9) popup menus as shown, see *Figure 1, Parm, (F9) Popup Menu Additions, Circled*. Operators no longer need to key in "-1" to designate starting or ending blocks. **Start Of Program** and **End Of Program** menu selections automatically select starting or ending block. Choosing the **Other Block** selection prompts operator for a block number between 1 and 99999999.

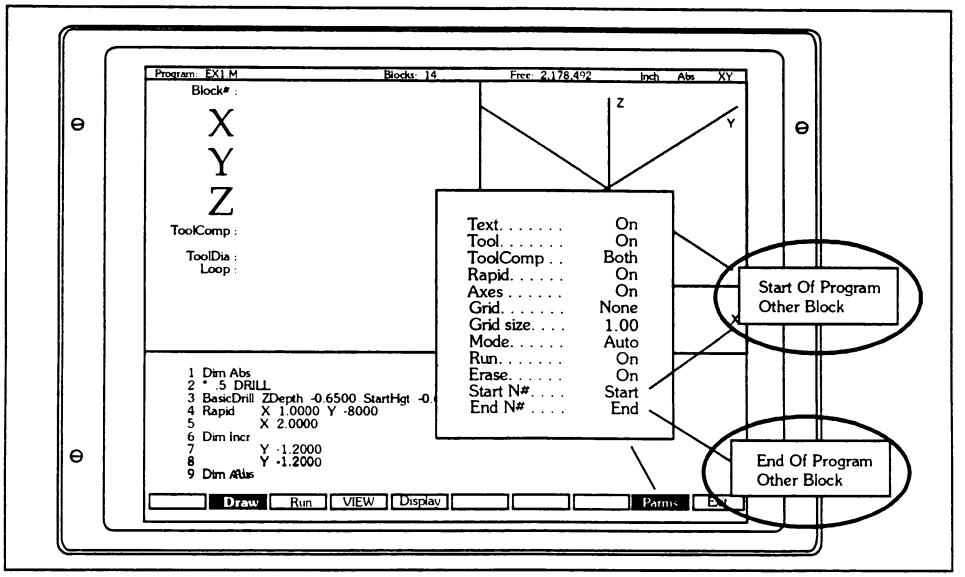


Figure 1, Parm, (F9) Popup Menu Additions, Circled

Hole - Mill Pocket (Added DepthCut Feature)

Programming A Hole - Mill Pocket Cycle

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START), .

Hole - Mill cycles simplify programming required to cut through holes, or clean up and counter bore existing holes.

When used, **StartHgt** and **ZDepth** must be used together. When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**.

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and executes an additional finish pass to the specified **Diameter** and **ZDepth**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

Program a **Hole** cycle as follows:

1. With the CNC in the **Edit** mode, press **Pocket,** (F4), (pop up menu displays).

2. Move highlight to mark **Hole** and press (ENTER), ENTER, (display prompts for labeled values as follows, see *Figure 2, Hole - Mill Pocket Screen Diagram*).

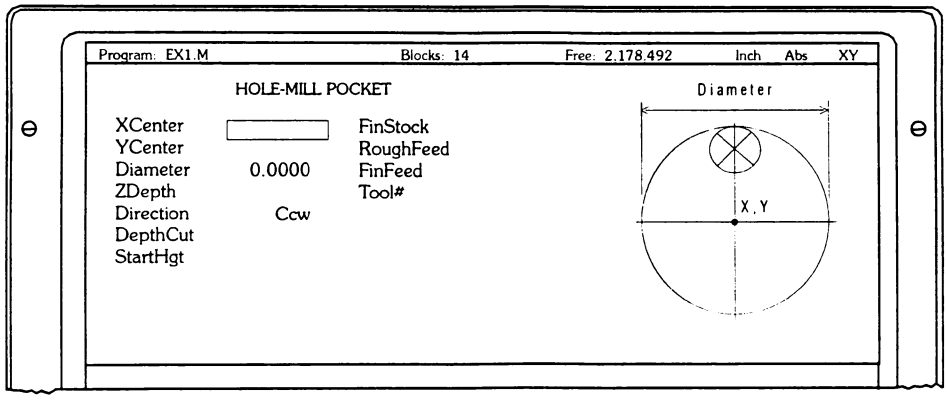


Figure 2, Hole - Mill Pocket Screen Diagram

XCenter X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.

YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.

NOTE: Keying in absolute center coordinates whenever possible is recommended.

Diameter	Diameter of the pocket
ZDepth	The depth of the finished pocket.
Direction	This setting allows the operator to choose between a counter clockwise (Ccw), or clockwise (Cw) direction. The setting is toggled by using the (TOGGLE), [+/-], key.
DepthCut	Z depth per pass, defaults to a single cut minus the finish stock if no value is entered.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Rectangular Profile (Added Canned Cycle)

Programming A Rectangular Profile Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START),  .

The rectangular profile cycle is provided for cleaning up the inside or outside profile of a rectangle. **Rect.Profile** cycles execute as follows:

1. CNC rapids to the ramp starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.
2. Machine feeds into the profile along Ramp #1, cuts profile to **Length**, and **Width** specified, then ramps away from the work along Ramp #2, see *Figure 3, Rectangular Profile Screen Diagram*.

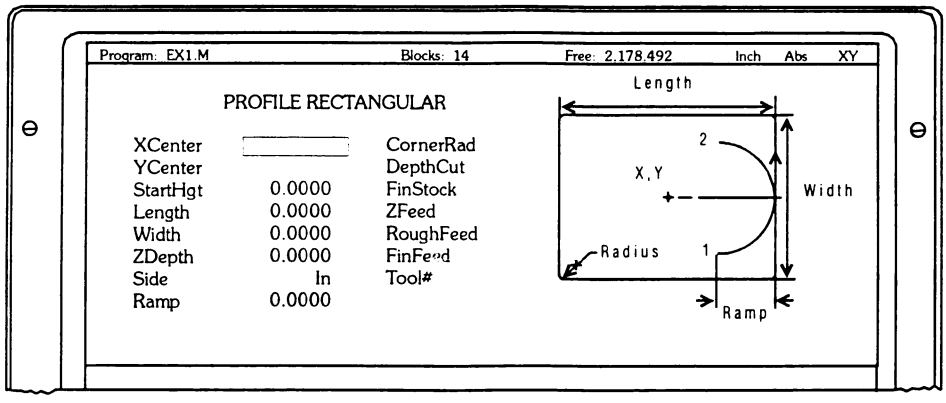


Figure 3, Rectangular Profile Screen Diagram

When cutting an inside profile, the tool ramps into the work as shown on the screen. When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2 as shown, see *Figure 4, Ramp Position For Outside Profile*. It is important to ensure clearance exists for outside ramps.

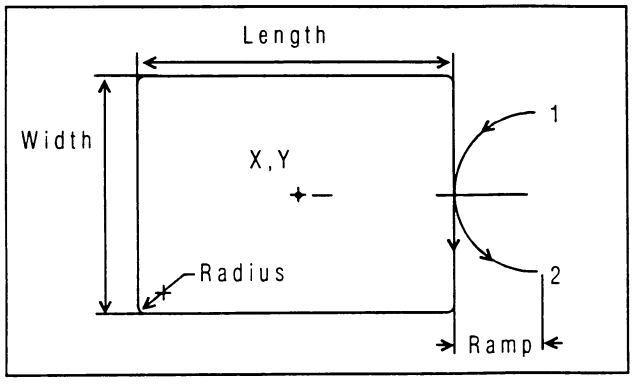


Figure 4, Ramp Position For Outside Profile

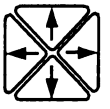
When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and at the end of the cycle, executes a finish pass to **ZDepth**, **Length**, and **Width**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected. **ZFeed** also reverts to the current feedrate if not programmed.

Program a rectangular profile as follows:

1. With the CNC in the **Edit** mode, press **Pocket**, (F4) (**Pocket** pop-up menu appears).
2. Using the (ARROW),  keys, position the highlight to mark **Rect.Profile** and press (ENTER), 

(**PROFILE RECTANGULAR** screen diagram displays, see *Figure 3, Rectangular Profile Screen Diagram*).
3. Key in **Rect.Profile** information labeled as follows:

XCenter	X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
Length	Finished length of rectangle.
Width	Finished width of rectangle.
ZDepth	Finished Z position of milled area.
Side	Setting for cutting on the inside of the profile, (In) or the outside (Out). This setting is toggled between selections using the (TOGGLE),  key.
Ramp	Radius of the ramp into and away from the cut, may be set to zero.
CornerRad	Corner radius setting. If a negative value is used, both the direction of cut, and the starting and end points are reversed.

DepthCut	Z move increment used for each pass.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.
ZFeed	Z axis feedrate (three axis only).
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active Tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Circular Profile (Added Canned Cycle)

Programming A Circular Profile Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START),  .

The circular profile cycle is provided for cleaning up the inside or outside profile of an existing circle. **Circ.Profile** cycles execute as follows:

1. CNC rapids to the ramp starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.
2. Machine feeds into the profile along Ramp #1, cuts profile to the **Diameter** specified, then ramps away from the work along Ramp #2, see *Figure 5, Circular Profile Screen Diagram*.

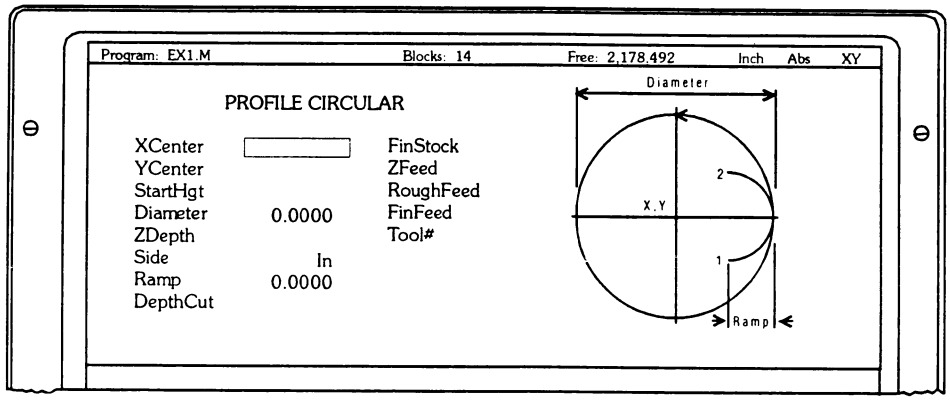


Figure 5, Circular Profile Screen Diagram

When cutting an inside profile, the tool ramps into the work as shown on the screen diagram. When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2 as shown, see *Figure 4, Ramp Position For Outside Profile*. It is important to ensure clearance exists for outside ramps.

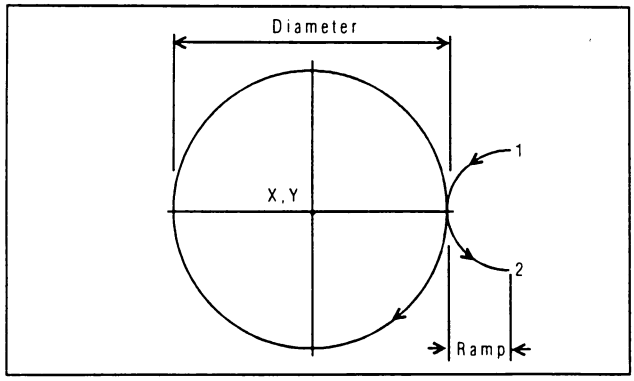


Figure 6, Ramp Position For Outside Profile

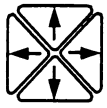
When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and at the end of the cycle, executes a finish pass to **ZDepth** and **Diameter**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected. **ZFeed** also reverts to the current feedrate if not programmed.

Program a rectangular profile as follows:

1. With the CNC in the Edit mode, press **Pocket**, (F4) (Pocket pop-up menu appears).
2. Using the (ARROW),  keys, position the highlight to mark **Circ.Profile** and press (ENTER),  (PROFILE CIRCULAR screen diagram displays, see *Figure 5, Circular Profile Screen Diagram*).
3. Key in Circ.Profile information labeled as follows:

XCenter	X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
Diameter	Finished diameter of circle. If a negative value is used, both the direction of cut, and the starting and end points are reversed.
ZDepth	Finished Z position of milled area.
Side	Setting for cutting on the inside of the profile, (In) or the outside (Out). This setting is toggled between selections using the (TOGGLE),  key.
Ramp	Radius of the ramp into and away from the cut, may be set to zero.
DepthCut	Z increment used for each pass.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.

ZFeed	Z axis feedrate (three axis only).
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active Tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Face Cycle (Added Canned Cycle)

Programming Facing Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START), [I].

Face pocket cycles simplify the programming of the repetitive moves required to face the surface of a part. **Face** cycles execute as follows:

The tool begins its cut one tool radius from the start point, moving in a direction appropriate for the selected stepover. See *Figure 7, Face Cycle Tool Approach*. The selected stepover determines the axes of the approach. **Face** cycles can start in any corner of the surface, and cut in any direction, depending on the sign (+/-) of the **Length** and **Width** values. To ensure complete facing of the surface, programming in a slightly larger **Length** and **Width** is recommended.

At the end of the cycle the tool rapids to **StartHgt**, then rapids back to XY start position.

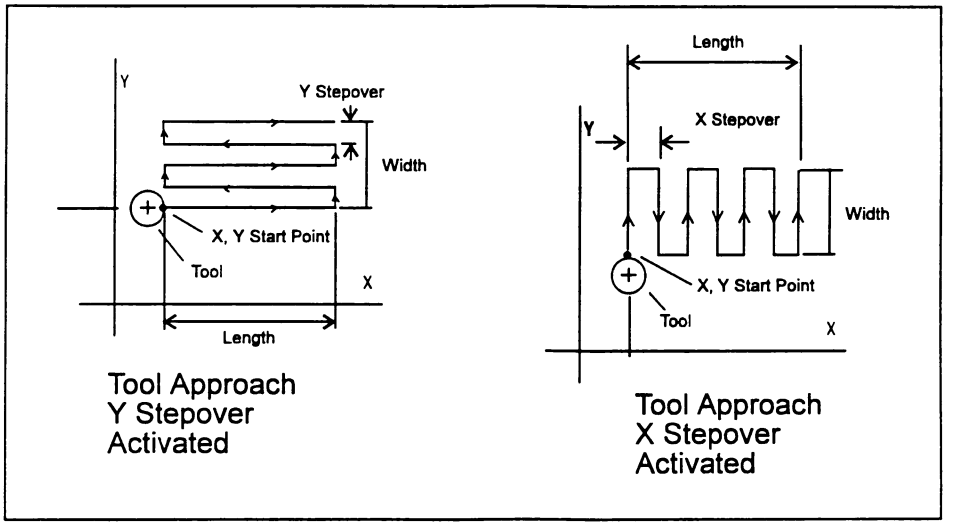


Figure 7, Face Cycle Tool Approach

1. With the CNC in the **Edit** mode, press **Pocket**, (F4), (pop up menu displays).
2. Move the highlight to mark **Face** and press (ENTER), [ENTER], (display prompts for labeled values, see *Figure 8, Face Pocket Screen Diagram*).
3. Key in **Face** cycle information labeled as follows:

StartHgt — Z starting position, position CNC rapids to before feeding into work.

ZDepth — The Z position of the faced surface.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

XStart X coordinate of facing cycle starting point, present position if no value entered.

YStart Y coordinate of the facing cycle starting point, present position if no value entered.

NOTE: Keying in absolute starting point coordinates whenever possible is recommended.

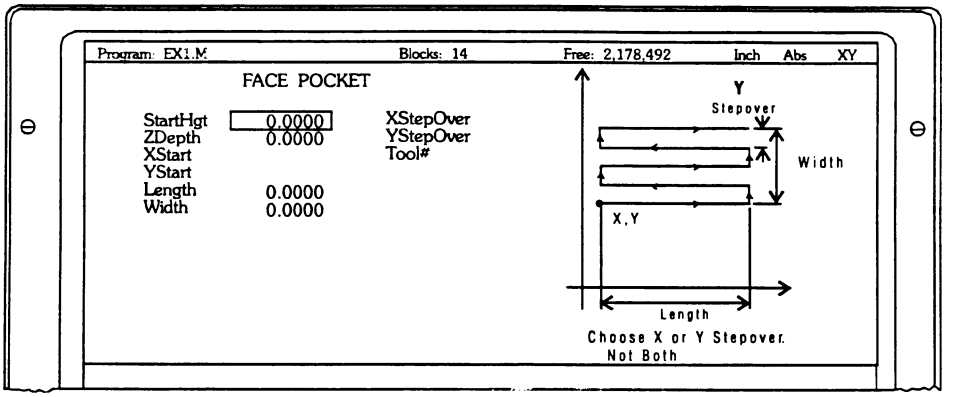


Figure 8, Face Pocket Screen Diagram

Length The X length of the faced surface.

Width The Y length of the faced surface.

XStepover X axis stepover is the size of cut along the X axis, value entered can be no greater than 70% of the active tool radius.

YStepover Y axis stepover is the size of cut along the Y axis, value entered can be no greater than 70% of the active tool radius.

NOTE: Enter either X or Y stepover only do not enter both.

Tool# Active tool.

NOTE: The CNC will allow the operator to key in incorrect stepover values, but the program may not run. Programs will stop and generate an error message if the stepover values are too large or if more than one stepover is entered.

Tapping Cycle (Added Canned Cycle)

Programming Tapping Cycles

The tapping cycle will only be available on 3 axis controls that have the spindle options, (M3, M4, & M5) installed. A **Spindle RPM** value, for the tool, must be entered on the tool page, for the cycle to operate. During execution, the CNC uses the **Spindle RPM** value from the tool page, and the threads per inch (or pitch) value from the cycle block to calculate the proper feedrate for tapping.

When the cycle runs, the CNC rapids to the **StartHgt**, and with spindle turning in the given direction, feeds to the **ZDepth**. The spindle then stops, and reverses out to the **ReturnHgt**. At **ReturnHgt** the spindle stops and changes back to the original direction. See *Figure 9, Tapping Cycle Screen Diagram*.

The **Tapping Cycle** can be used with all available patterns. A **Drilling Off** must be programmed to turn off the **Tapping Cycle** when it is no longer needed.

NOTE: StartHgt is 0.100 in.(2 mm) above the surface to be drilled.

1. With the CNC in the **Edit** mode, press **Drill**, (F3), (drill cycle pop-up menu appears).
2. Highlight **Tapping**, and press (ENTER), [ENTER], (CNC displays **TAPPING CYCLE** diagram).
3. Key in labeled values and press (ENTER), [ENTER], (display clears, new tapping cycle block is added).

NOTE: Once the operator defines a **TAPPING CYCLE** cycle, he must then program moves to tell it where to tap. The CNC will perform the cycle at the end point of all subsequent blocks until it sees a Drilling Off block.

4. Program subsequent moves to position the work to where the tapping is to occur, the CNC will tap a hole at the end point of every move.
5. After programming the last tapping move, press **Drill,** (F3), (drill cycle pop-up appears).
6. Highlight **Drilling Off**, and press (ENTER), [ENTER], (**Drill Off** block is included in program to take CNC out of the tapping mode).

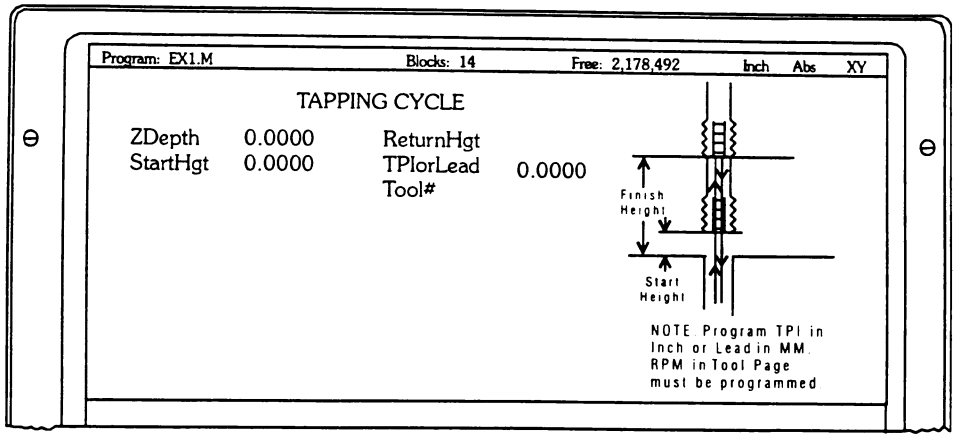


Figure 9, Tapping Cycle Screen Diagram

Labels for the **TAPPING CYCLE** screen are as follows:

ZDepth	Depth of tap.
StartHgt	Starting position.
ReturnHgt	Return height.
TPIorLead	TPI in **Inch** or Lead in **MM**.
Tool#	Active tool number.

Summary:

This document outlines changes to the 1100 Series 2 & 3 Axis Control Software made since the distribution of current programming manual. This addendum supersedes all earlier addendums to these manuals.

Contents:

NOTE: Holders of Addendum Version 3.10A are advised there are significant changes made to the Face Pocket cycle.

Edit Screen (Added Tool Hot Key)

Added use of keypad hot key to insert a tool number block into a program. With program editor active, pressing the #5 keypad key allows operator to insert a tool number block into the program. Block activates corresponding tool diameter and length offsets.

NOTE: This feature is also available when editing in the MDI mode.

Tool Page (Softkey Label Change)

Find softkey was (F7), moved to (F4).

Calib Z softkey was (F9), moved to (F8).

Draw Screen (Softkey Label Change)

Draw screen (F2) softkey label was changed from **Block** to **S.Step**, operation is unchanged, (advances the program one block at a time).

Draw Screen (Improved Popup Menus)

Inserted additional **Draw** screen, **Parm**, (F9) popup menus as shown, see *Figure 1, Parm, (F9) Popup Menu Additions, Circled*. Operators no longer need to key in "-1" to designate starting or ending blocks. **Start Of Program** and **End Of Program** menu selections automatically select starting or ending block. Choosing the **Other Block** selection prompts operator for a block number between 1 and 99999999.

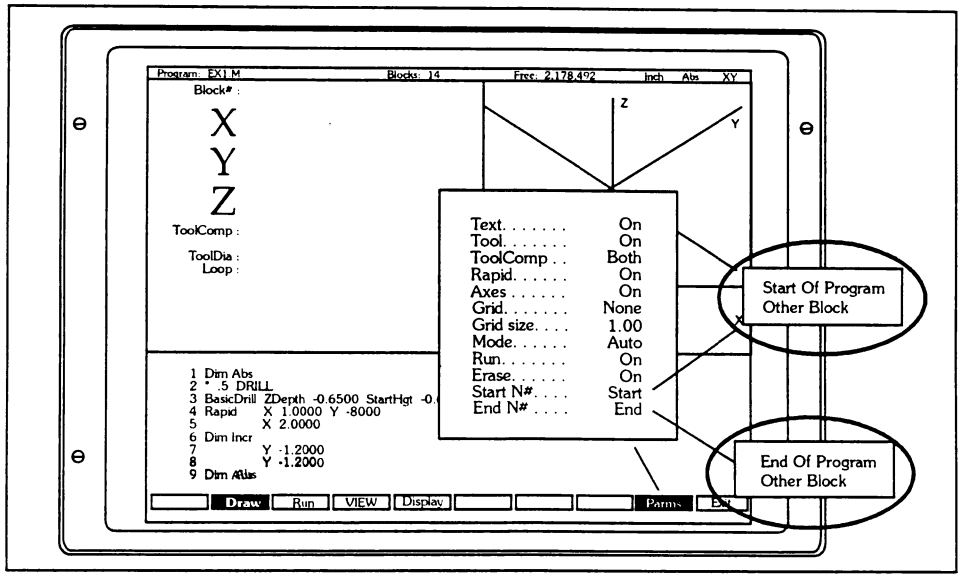


Figure 1, Parm, (F9) Popup Menu Additions, Circled

Hole - Mill Pocket (Added DepthCut Feature)

Programming A Hole - Mill Pocket Cycle

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START), .

Hole - Mill cycles simplify programming required to cut through holes, or clean up and counter bore existing holes.

When used, **StartHgt** and **ZDepth** must be used together. When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**.

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and executes an additional finish pass to the specified **Diameter** and **ZDepth**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

Program a **Hole** cycle as follows:

1. With the CNC in the **Edit** mode, press **Pocket,** (F4), (pop up menu displays).
2. Move highlight to mark **Hole** and press (ENTER), ENTER, (display prompts for labeled values as follows, see *Figure 2, Hole - Mill Pocket Screen Diagram*).

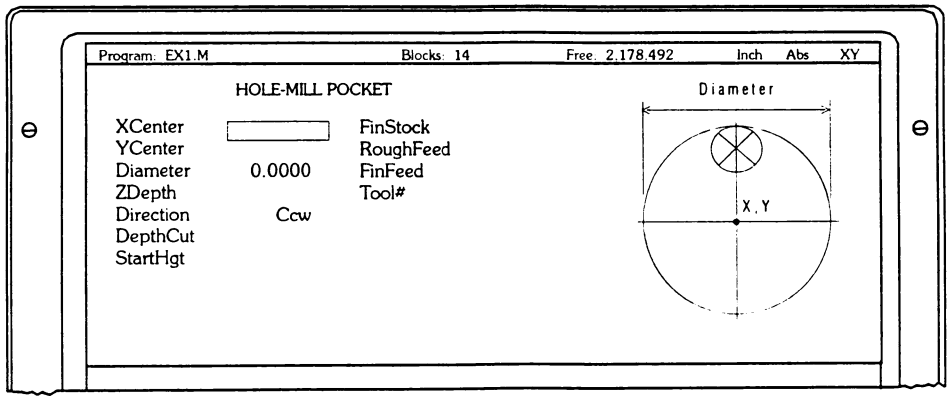


Figure 2, Hole - Mill Pocket Screen Diagram

XCenter X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.

YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.

NOTE: Keying in absolute center coordinates whenever possible is recommended.

Diameter	Diameter of the pocket
ZDepth	The depth of the finished pocket.
Direction	This setting allows the operator to choose between a counter clockwise (Ccw), or clockwise (Cw) direction. The setting is toggled by using the (TOGGLE), [+/-], key.
DepthCut	Z depth per pass, defaults to a single cut minus the finish stock if no value is entered.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Rectangular Profile (Added Canned Cycle)

Programming A Rectangular Profile Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START),  .

The rectangular profile cycle is provided for cleaning up the inside or outside profile of a rectangle. **Rect.Profile** cycles execute as follows:

1. CNC rapids to the ramp starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.
2. Machine feeds into the profile along Ramp #1, cuts profile to **Length**, and **Width** specified, then ramps away from the work along Ramp #2, see *Figure 3, Rectangular Profile Screen Diagram*.

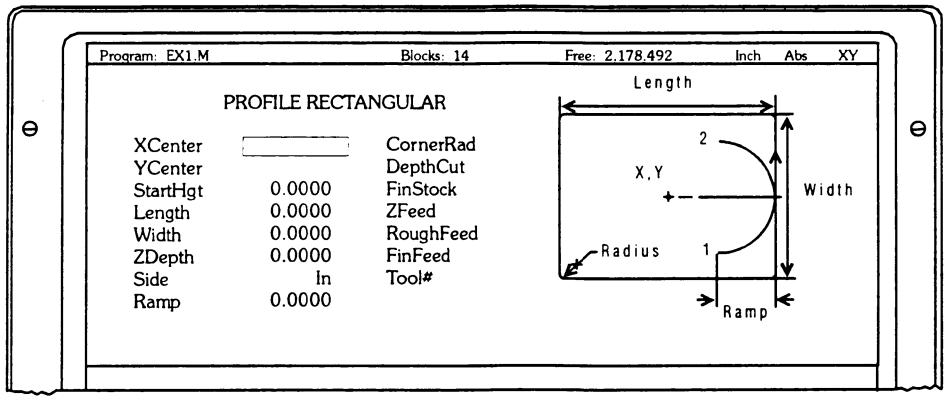


Figure 3, Rectangular Profile Screen Diagram

When cutting an inside profile, the tool ramps into the work as shown on the screen. When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2 as shown, see *Figure 4, Ramp Position For Outside Profile*. It is important to ensure clearance exists for outside ramps.

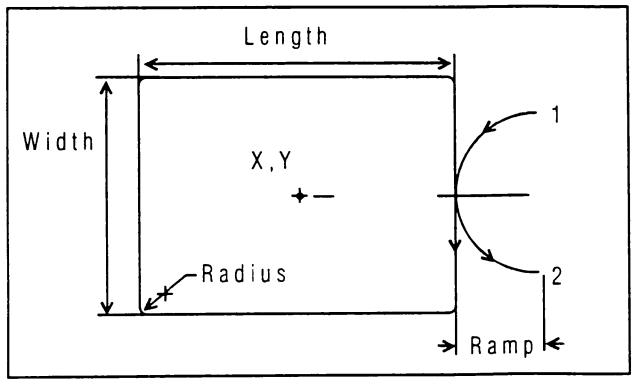


Figure 4, Ramp Position For Outside Profile

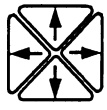
When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and at the end of the cycle, executes a finish pass to **ZDepth**, **Length**, and **Width**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected. **ZFeed** also reverts to the current feedrate if not programmed.

Program a rectangular profile as follows:

1. With the CNC in the **Edit** mode, press **Pocket**, (F4) (**Pocket** pop-up menu appears).
2. Using the (ARROW),  keys, position the highlight to mark **Rect.Profile** and press (ENTER),  (**PROFILE RECTANGULAR** screen diagram displays, see *Figure 3, Rectangular Profile Screen Diagram*).
3. Key in **Rect.Profile** information labeled as follows:

XCenter	X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
Length	Finished length of rectangle.
Width	Finished width of rectangle.
ZDepth	Finished Z position of milled area.
Side	Setting for cutting on the inside of the profile, (In) or the outside (Out). This setting is toggled between selections using the (TOGGLE),  key.
Ramp	Radius of the ramp into and away from the cut, may be set to zero.
CornerRad	Corner radius setting. If a negative value is used, both the direction of cut, and the starting and end points are reversed.

DepthCut	Z move increment used for each pass.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.
ZFeed	Z axis feedrate (three axis only).
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active Tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Circular Profile (Added Canned Cycle)

Programming A Circular Profile Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START),  .

The circular profile cycle is provided for cleaning up the inside or outside profile of an existing circle. **Circ.Profile** cycles execute as follows:

1. CNC rapids to the ramp starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.
2. Machine feeds into the profile along Ramp #1, cuts profile to the **Diameter** specified, then ramps away from the work along Ramp #2, see *Figure 5, Circular Profile Screen Diagram*.

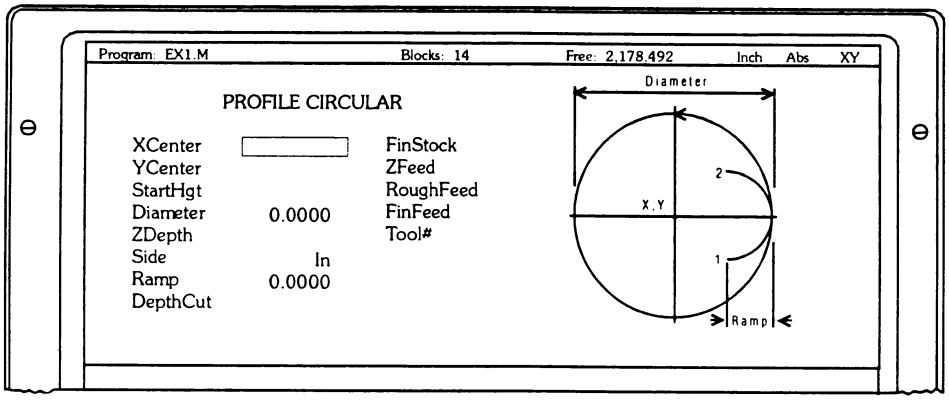


Figure 5, Circular Profile Screen Diagram

When cutting an inside profile, the tool ramps into the work as shown on the screen diagram. When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2 as shown, see *Figure 4, Ramp Position For Outside Profile*. It is important to ensure clearance exists for outside ramps.

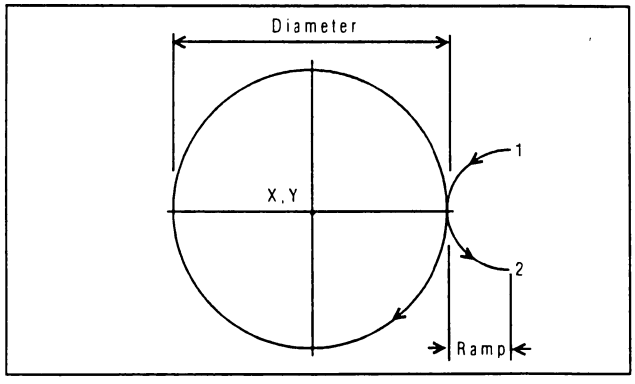


Figure 6, Ramp Position For Outside Profile

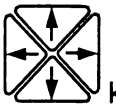
When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

NOTE: Required StartHgt is 0.100 in., (2 mm) above the surface being cut.

When **FinStock** is used, the CNC leaves the specified amount of material on the profile and depth, and at the end of the cycle, executes a finish pass to **ZDepth** and **Diameter**. If a negative **FinStock** value is keyed in, CNC will leave the specified amount but will skip the finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes the cycle's feed moves at the current feedrate. If a **RoughFeed** value is keyed in, only the **RoughFeed** rate is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected. **ZFeed** also reverts to the current feedrate if not programmed.

Program a rectangular profile as follows:

1. With the CNC in the Edit mode, press **Pocket**, (F4) (Pocket pop-up menu appears).
2. Using the (ARROW),  keys, position the highlight to mark **Circ.Profile** and press (ENTER),  (PROFILE CIRCULAR screen diagram displays, see *Figure 5, Circular Profile Screen Diagram*).
3. Key in Circ.Profile information labeled as follows:

XCenter	X coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
YCenter	Y coordinate of pocket center, if no coordinate is entered, pocket is centered at present position.
StartHgt	Z starting position, position CNC rapids to before feeding into work.
Diameter	Finished diameter of circle. If a negative value is used, both the direction of cut, and the starting and end points are reversed.
ZDepth	Finished Z position of milled area.
Side	Setting for cutting on the inside of the profile, (In) or the outside (Out). This setting is toggled between selections using the (TOGGLE),  key.
Ramp	Radius of the ramp into and away from the cut, may be set to zero.
DepthCut	Z increment used for each pass.
FinStock	Amount of stock left by the machine before the finish pass, zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass.

ZFeed	Z axis feedrate (three axis only).
RoughFeed	Rough pass feedrate.
FinFeed	Finish pass feedrate.
Tool#	Active Tool.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

Face Cycle (Added Canned Cycle)

Programming Facing Cycles

NOTE: On a 2 axis machine, whenever a Z move is required, the CNC will hold the program and prompt the operator to position the Z axis and press (START), .

Face pocket cycles simplify the programming of the repetitive moves required to face the surface of a part. **Face** cycles execute as follows:

The tool begins its cut one tool radius from the start point, moving in a direction appropriate for the selected stepover. See *Figure 7, Face Cycle Tool Approach*. The selected stepover determines the axes of the approach. **Face** cycles can start in any corner of the surface, and cut in any direction, depending on the sign (+/-) of the **Length** and **Width** values. To ensure complete facing of the surface, programming in a slightly larger **Length** and **Width** is recommended.

At the end of the cycle the tool rapids to **StartHgt**, then rapids back to XY start position.

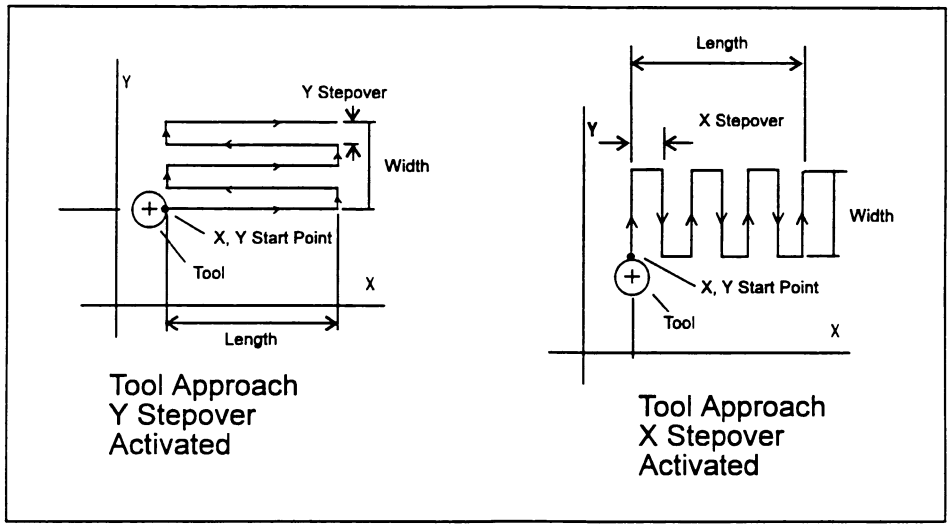


Figure 7, Face Cycle Tool Approach

1. With the CNC in the **Edit** mode, press **Pocket**, (F4), (pop up menu displays).
2. Move the highlight to mark **Face** and press (ENTER), , (display prompts for labeled values, see *Figure 8, Face Pocket Screen Diagram*).
3. Key in **Face** cycle information labeled as follows:

StartHgt — Z starting position, position CNC rapids to before feeding into work.

ZDepth — The Z position of the faced surface.

NOTE: ZDepth must be lower than StartHgt. The CNC will allow the operator to key in impossible positions, but the program will not run, it will halt and produce an error message.

XStart X coordinate of facing cycle starting point, present position if no value entered.

YStart Y coordinate of the facing cycle starting point, present position if no value entered.

NOTE: Keying in absolute starting point coordinates whenever possible is recommended.

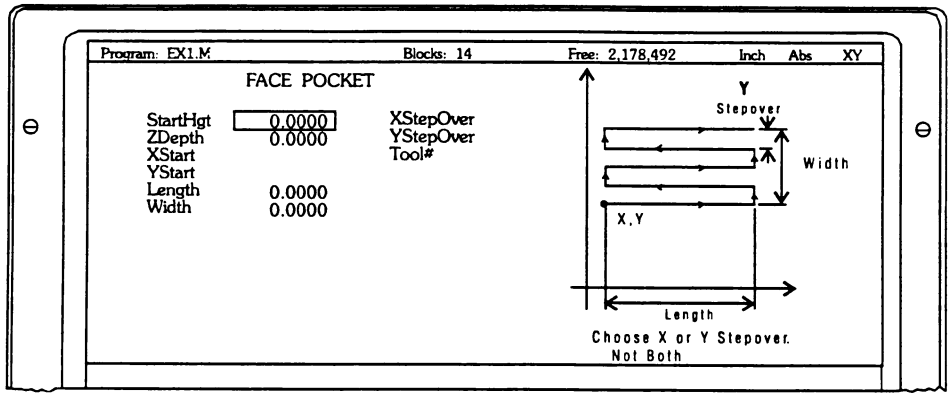


Figure 8, Face Pocket Screen Diagram

Length The X length of the faced surface.

Width The Y length of the faced surface.

XStepover X axis stepover is the size of cut along the X axis, value entered can be no greater than 70% of the active tool radius.

YStepover Y axis stepover is the size of cut along the Y axis, value entered can be no greater than 70% of the active tool radius.

NOTE: Enter either X or Y stepover only do not enter both.

Tool# Active tool.

NOTE: The CNC will allow the operator to key in incorrect stepover values, but the program may not run. Programs will stop and generate an error message if the stepover values are too large or if more than one stepover is entered.

Tapping Cycle (Added Canned Cycle)

Programming Tapping Cycles

The tapping cycle will only be available on 3 axis controls that have the spindle options, (M3, M4, & M5) installed. A **Spindle RPM** value, for the tool, must be entered on the tool page, for the cycle to operate. During execution, the CNC uses the **Spindle RPM** value from the tool page, and the threads per inch (or pitch) value from the cycle block to calculate the proper feedrate for tapping.

When the cycle runs, the CNC rapids to the **StartHgt**, and with spindle turning in the given direction, feeds to the **ZDepth**. The spindle then stops, and reverses out to the **ReturnHgt**. At **ReturnHgt** the spindle stops and changes back to the original direction. See *Figure 9, Tapping Cycle Screen Diagram*.

The **Tapping Cycle** can be used with all available patterns. A **Drilling Off** must be programmed to turn off the **Tapping Cycle** when it is no longer needed.

NOTE: StartHgt is 0.100 in.(2 mm) above the surface to be drilled.

1. With the CNC in the **Edit** mode, press **Drill**, (F3), (drill cycle pop-up menu appears).
2. Highlight **Tapping**, and press (ENTER), [ENTER], (CNC displays **TAPPING CYCLE** diagram).
3. Key in labeled values and press (ENTER), [ENTER], (display clears, new tapping cycle block is added).

NOTE: Once the operator defines a **TAPPING CYCLE** cycle, he must then program moves to tell it where to tap. The CNC will perform the cycle at the end point of all subsequent blocks until it sees a Drilling Off block.

4. Program subsequent moves to position the work to where the tapping is to occur, the CNC will tap a hole at the end point of every move.
5. After programming the last tapping move, press **Drill,** (F3), (drill cycle pop-up appears).
6. Highlight **Drilling Off**, and press (ENTER), [ENTER], (**Drill Off** block is included in program to take CNC out of the tapping mode).

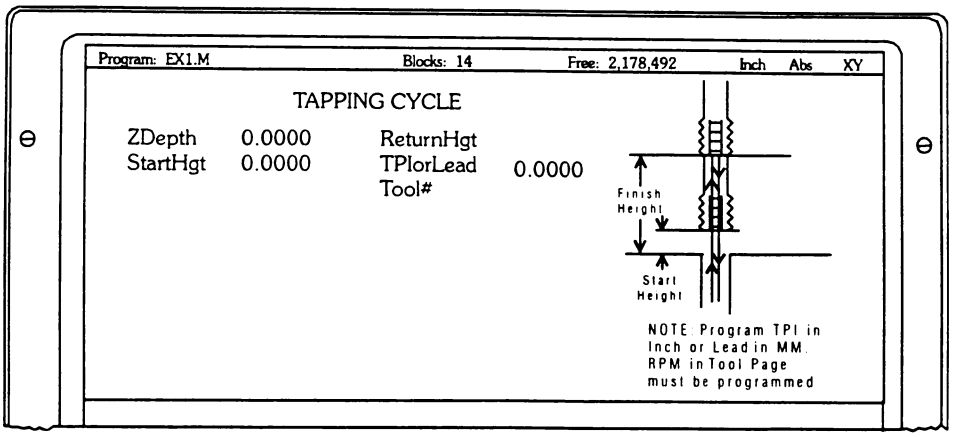


Figure 9, Tapping Cycle Screen Diagram

Labels for the **TAPPING CYCLE** screen are as follows:

ZDepth	Depth of tap.
StartHgt	Starting position.
ReturnHgt	Return height.
TPIorLead	TPI in **Inch** or Lead in **MM**.
Tool#	Active tool number.

Software Options Installation Procedure

P/N 70000317
SFTWR OPT INST PROC

Summary:

This document details the installation procedure of the Anilam Electronics Software Options Kit.

Kit Contents:

Item Name:	Part Number	Quantity
Installation Procedure	70000317	1 ea.
Installation Diskette	Serialized	1 ea.
Hardware Key	Serialized	1 ea.

NOTE: The software package on the installation diskette, and the hardware key are matched serially. The software will only run on a control if the key with the same serial number is installed.

CAUTION: With serialized installation disks, it is extremely important the disk be stored in a safe place. The disk alone cannot be replaced, in the event a disk is lost, the entire kit must be replaced.

Procedure:

CAUTION: During the installation procedure, the control will prompt to overwrite the existing configuration file. Under no circumstances should an equipment operator overwrite the configuration file. Overwriting the configuration file may disable the machine until it can be restored by qualified personnel.

1. Verify kit contents, and confirm disk serial number matches the serial number of the hardware key. Verify the ordered options from the shipping document.
2. Gain access to the 25 pin female, DB -25 printer port connector, inside the cabinet, on the computer chassis, and install the hardware key.

   NOTE: If an Anilam electronics hardware key is already installed remove it and replace with the key serialized for the new software package.
3. Gain access to the controls floppy disk drive and insert the installation diskette.
4. At the control console run the **Setup Utility.**
5. From the **Setup Options** menu, activate the **Machine/Installation** selection.
6. From the **Machine/Installation**, menu activate the **Software Upgrade** selection, (installation sequence begins, follow screen prompt instructions).

   NOTE: After installation of software option package, additional setup changes may be required depending on the options purchased.

   NOTE: With the installation of this kit, the original program disk that came packaged with the program manual is no longer of any value and should be discarded.