

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:

	<u>Page</u>
Edit Screen (Added Tool Hot Key)	2
Tool Page (Softkey Label Change)	2
Draw Screen (Softkey Label Change)	2
(Improved Popup Menus)	2
Hole - Mill Pocket (Added DepthCut Feature)	3
Rectangular Profile (Added Canned Cycle)	5
Circular Profile (Added Canned Cycle)	8
Face Cycle (Added Canned Cycle)	11
Tapping Cycle (Added Canned Cycle)	13

<p>NOTE: Holders of Addendum Version 3.10A are advised there are significant changes made to the Face Pocket cycle.</p>
--

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.

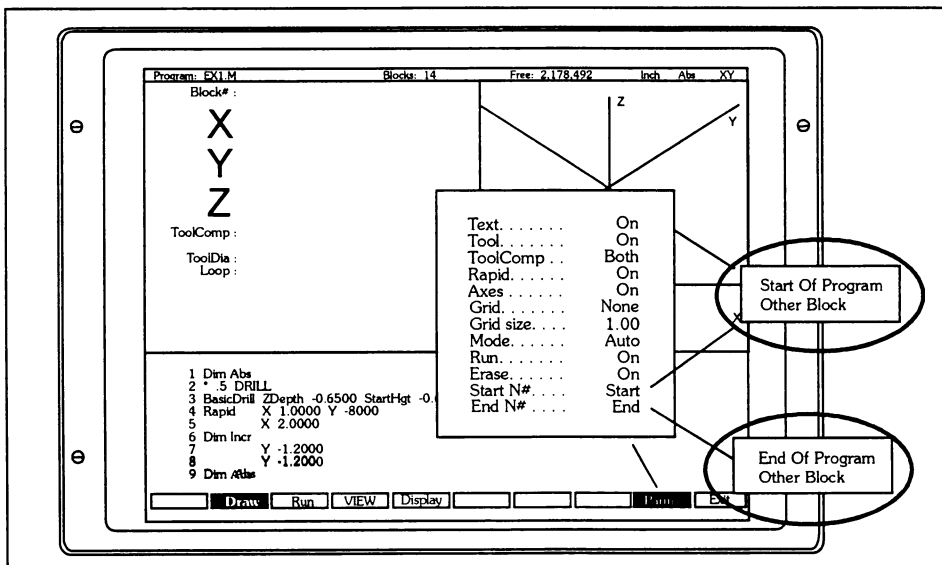



Figure 1, Parm, (F9) Popup Menu Additions, Circled

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), , (display prompts for labeled values as follows, see *Figure 2, Hole - Mill Pocket Screen Diagram*).

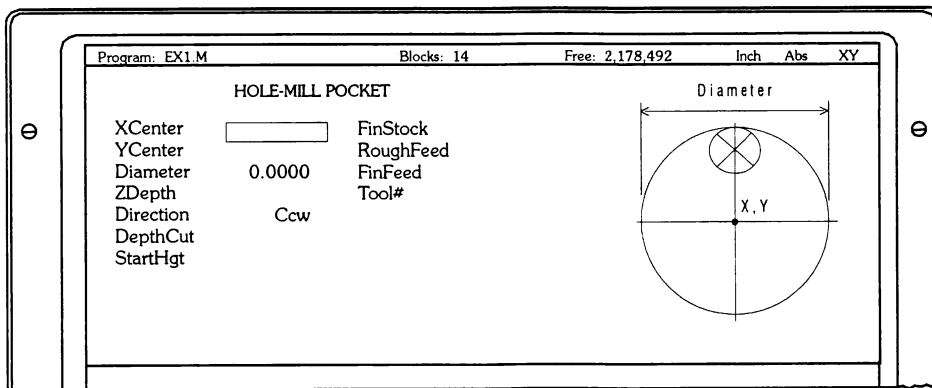


Figure 2, Hole - Mill Pocket Screen Diagram


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](#).

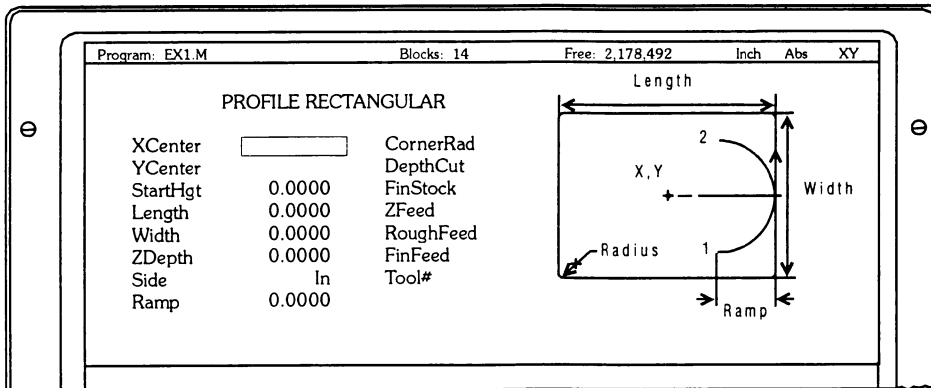


Figure 3, Rectangular Profile Screen Diagram

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.

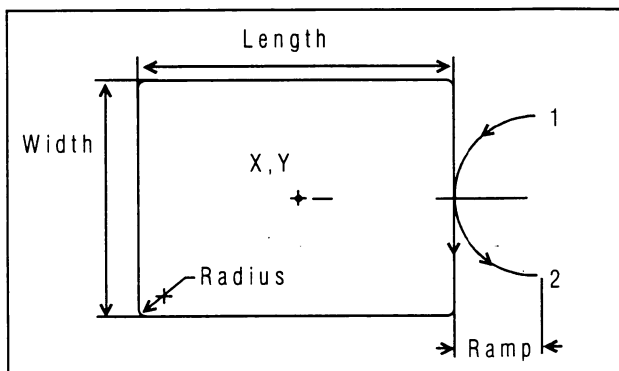


Figure 4, Ramp Position For Outside Profile

When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

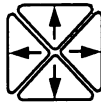
NOTE: Required **StartHgt** is 0.100 in., (2 mm) above the surface being cut.

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*.

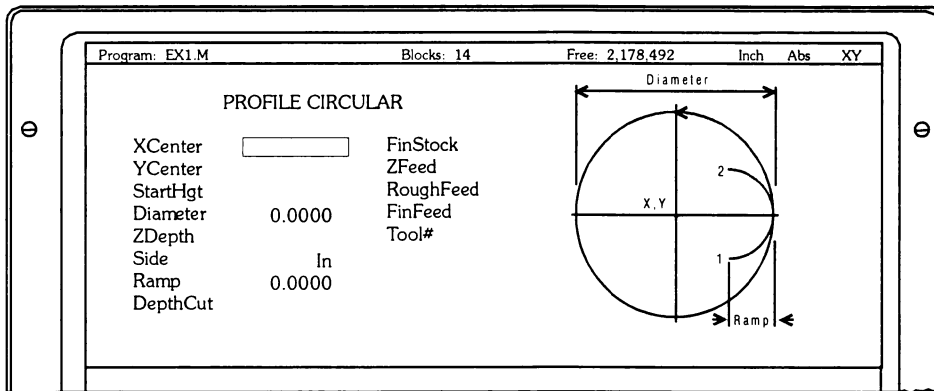


Figure 5, Circular Profile Screen Diagram

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.

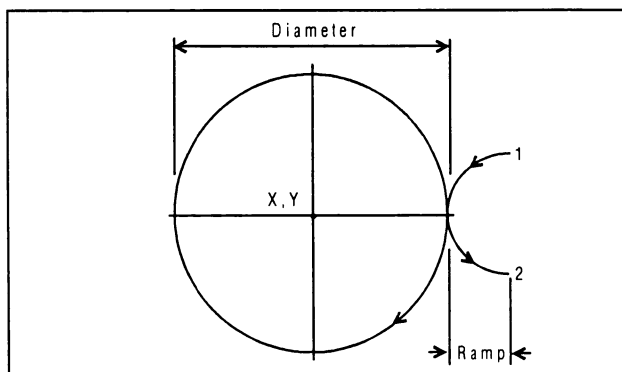


Figure 6, Ramp Position For Outside Profile

When **DepthCut** is keyed in, the CNC executes the number of passes needed to get from the **StartHgt** to the **ZDepth**,

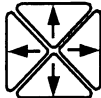
NOTE: Required **StartHgt** is 0.100 in., (2 mm) above the surface being cut.

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.

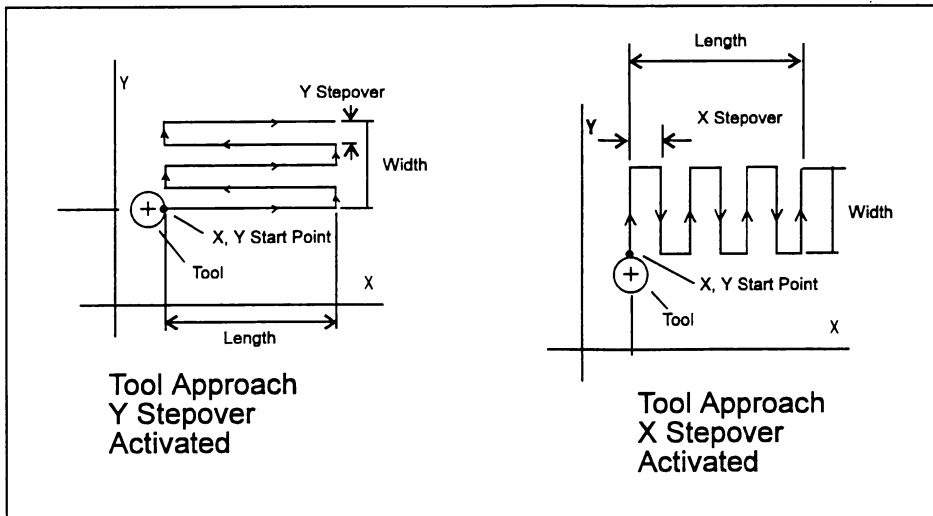



Figure 7, Face Cycle Tool Approach

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.

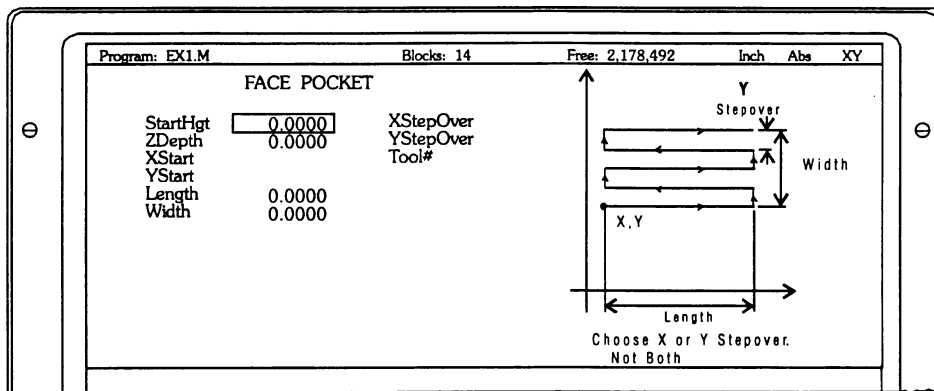


Figure 8, Face Pocket Screen Diagram

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), , (CNC displays **TAPPING CYCLE** diagram).

3. Key in labeled values and press (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), , (**Drill Off** block is included in program to take CNC out of the tapping mode).

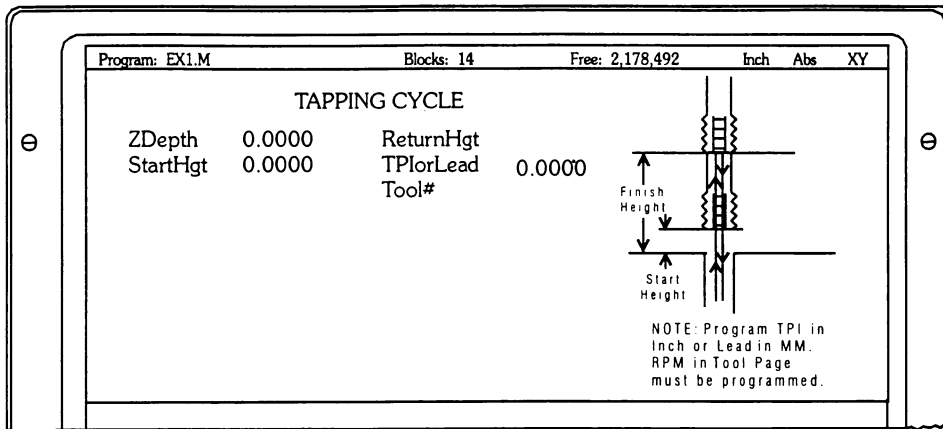


Figure 9, Tapping Cycle Screen Diagram

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.