MILLPWR G2
PROGRAM TRAINING

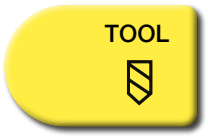
HEIDENHAIN CORPORATION
333 East State Parkway, Schaumburg, IL 60173
1274 Hunt Road, Ashville, NY 14710

Notes

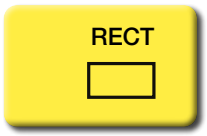
MILLPWR^{G2}

Easy-to-Master Programming

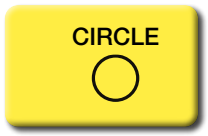
Standardized menus make it easy to program common features like lines, arcs, custom pockets, hole patterns, and islands. To add a feature to your program, simply press a function key and then follow the prompts. MILL**PWR^{G2}** also includes an “engrave” option for engraving alphanumeric characters vertically, horizontally, diagonally or along an arc.



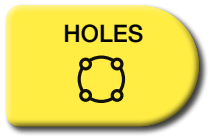
Program tools by diameter, length, type, direction, speed



Pocket, frame, face, slot

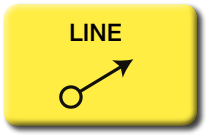


Pocket, frame, ring, helix

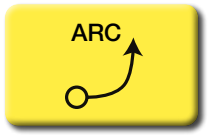


Full/partial circles, linear row/ column, rectangular frame and array

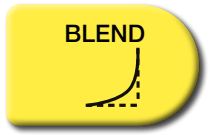
Drill, bore, position



Defined by “from” and “to” points or by angles



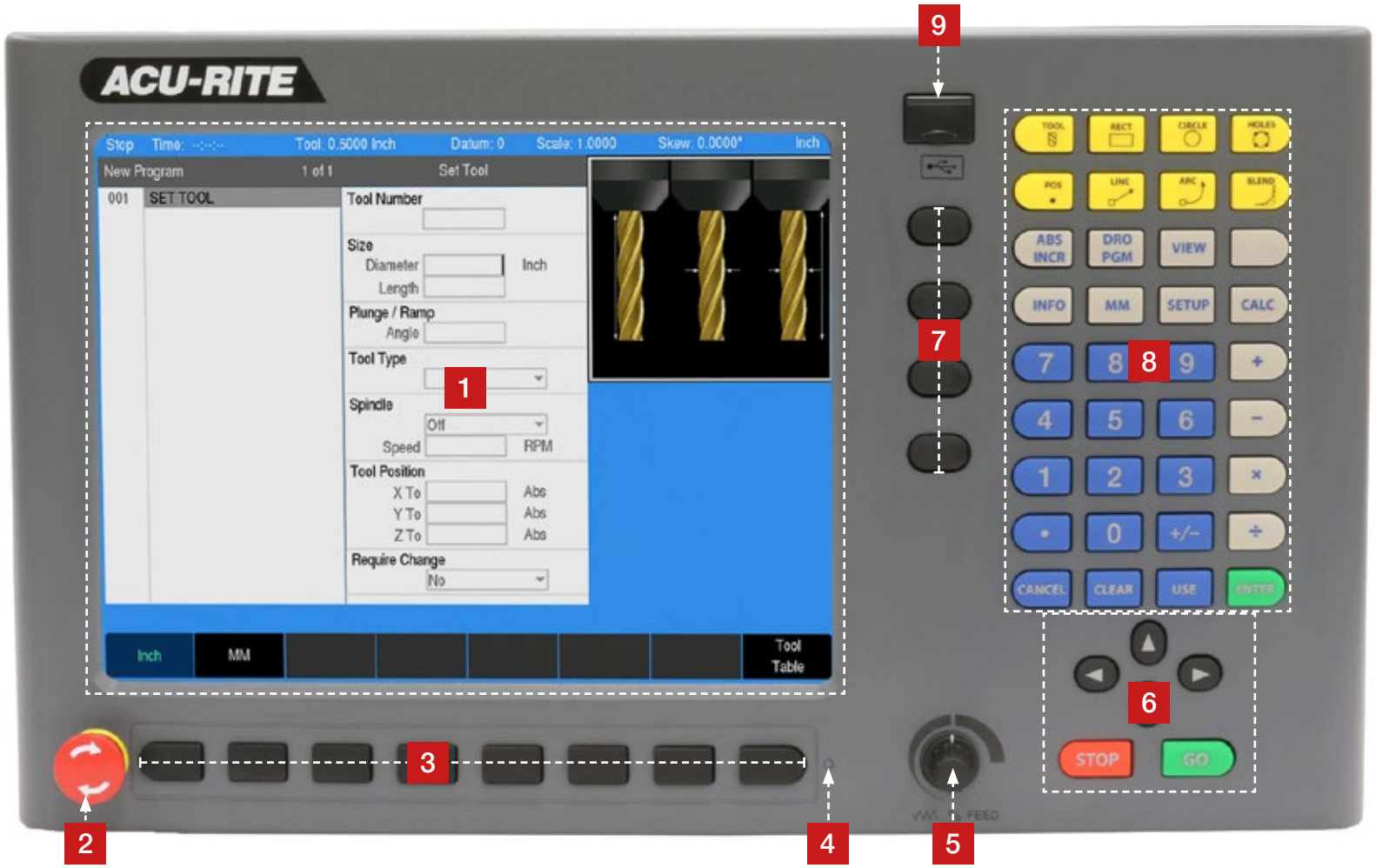
Defined by “from” and “to” points or by sweep, center or 3-points



Insert a corner radius between two lines, two arcs or lines and arcs

MILLPWR G2

The Operating Console



1 12.1" Flat-Panel Color Display

2 Emergency “E” Stop

3 Soft Keys

4 Power On / Error Indicator Light

5 Potentiometer for Feed Rate Override

6 Go, Pause/Stop and Navigation/Move Table Arrow Keys

7 Axis Keys

8 Console Keypad

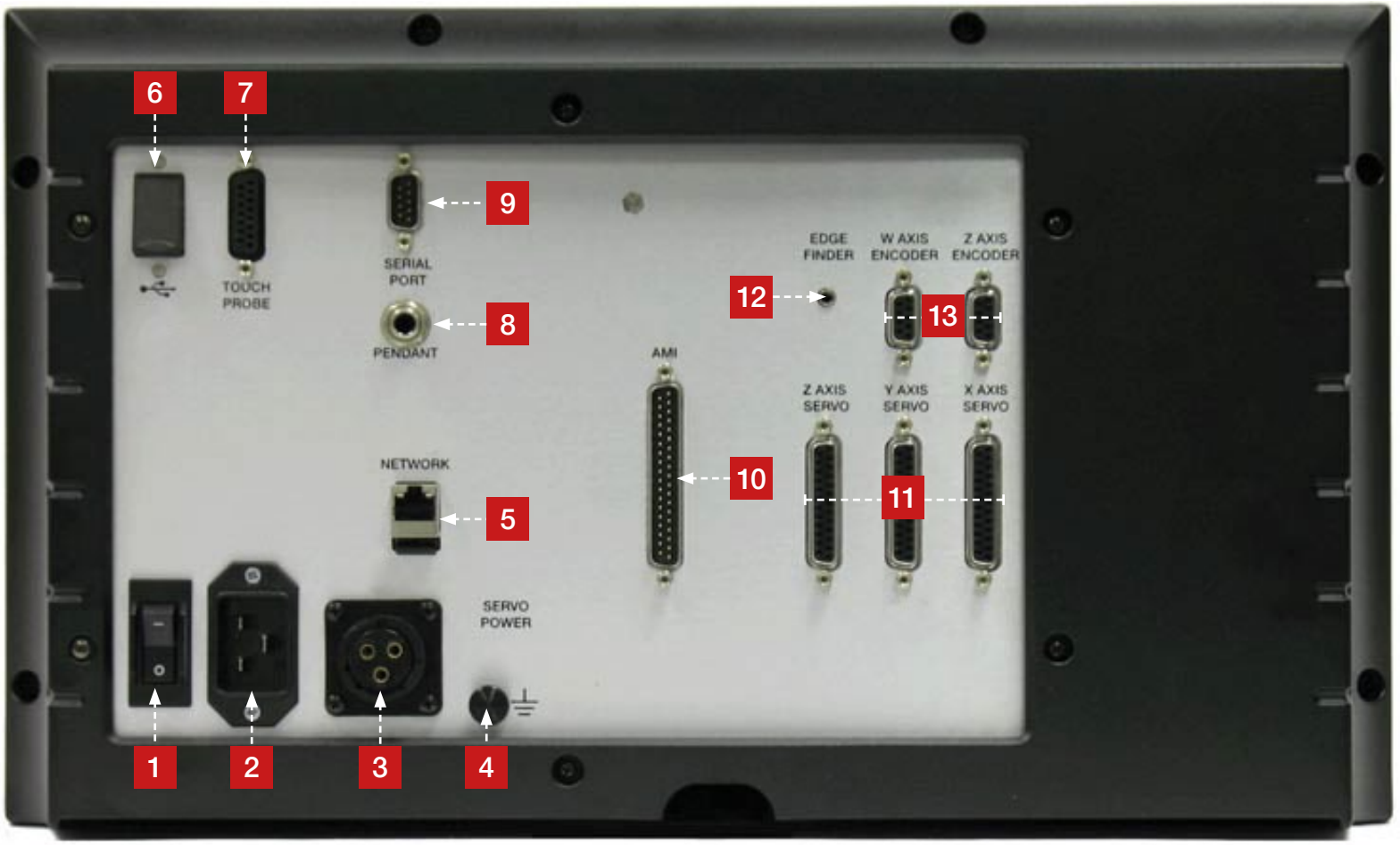
9 USB Port

PPT SLIDES 3-11

Notes

MILLPWR G2

The Operating Console



1. Power Switch
2. Power Connector
3. Servo Power Output
4. Earth Ground Terminal
5. Ethernet Port
6. USB Port
7. KT 130 Edge Finder
8. Remote Stop/Go Switch
9. RS-232-C Connector
10. Auxiliary Machine Interface (AMI)
11. Servo connector (X, Y and Z)
12. KT 3D Grounding Edger Finder
13. Encoder Inputs (W and Z axis)

PPT SLIDES 12-24

Notes

Disengage Z Axis Feature

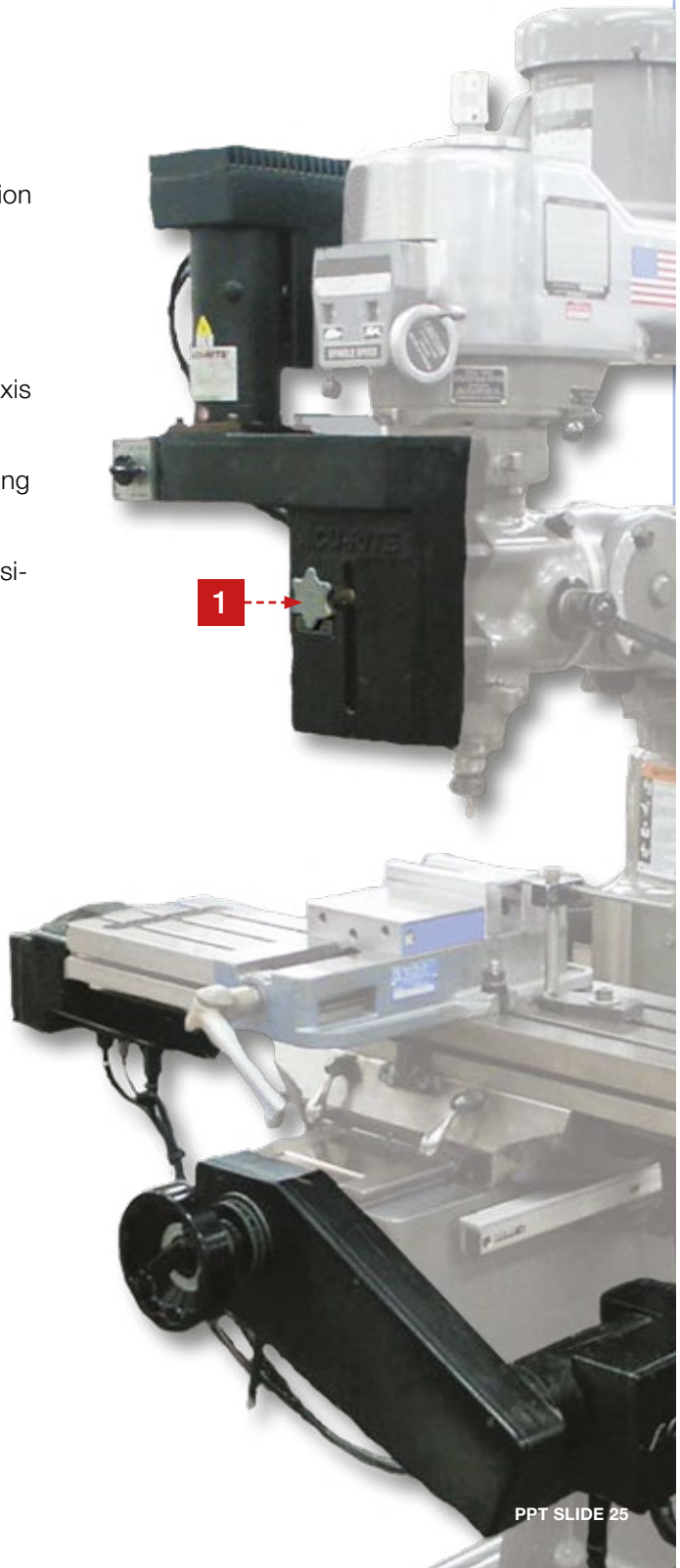
MILL**PWR**^{G2} provides the flexibility to switch between 2 axes and 3 axes operation within a single program and without changing setup parameters.

Disengaging the Z axis drive

1. Raise the quill, then loosen the quick release knob [1] on the front of the Z axis drive system.
2. Leave the Z BEGIN field blank when programming a step, or a one time milling operation.
3. When the operation is then performed, you will be prompted to manually position the quill when necessary.

Re-engaging the Z axis drive

1. Raise the quill handle to seat the ball nut into the nut block.
2. Tighten the quick release knob.
3. Lower the quill below the upper limit switch.



PPT SLIDE 25

Notes

Find Home

Unlike many controls, MILL**PWR**^{G2} does not require a move to a fixed machine home position. It instead establishes home using a distance encrypted reference pattern. This pattern allows the control to reestablish work piece zero, also called datum, from any location. During the homing process each axis will move about 1 to 2 inches. This is enough to learn the machine's current position.

- During start up, the Find Home soft key is provided.
- If the find home step is not performed at initial start up, it can be initiated at any time during operation. Press the Datum soft key, then press the Home soft key, and then press the Find Home soft key.
- A 3-axis system will move the table and quill. They will automatically move a few inches along the Z, Y, and then X to find home.
- A 2-axis system will move the table. The table will automatically move a few inches along the Y, and then X to find home. Then the control will prompt you to move the Z quill manually to home it.
- If a W-Axis scale is coupled with the Z, the control will then prompt you to move the W manually to find home.

Position-Trac™ will accurately re-establish work piece zero after a power loss, or shut down. When home has been found, the tool's position (relative to the most recent datum) will be displayed and soft limits reestablished.

PPT SLIDES 26-27

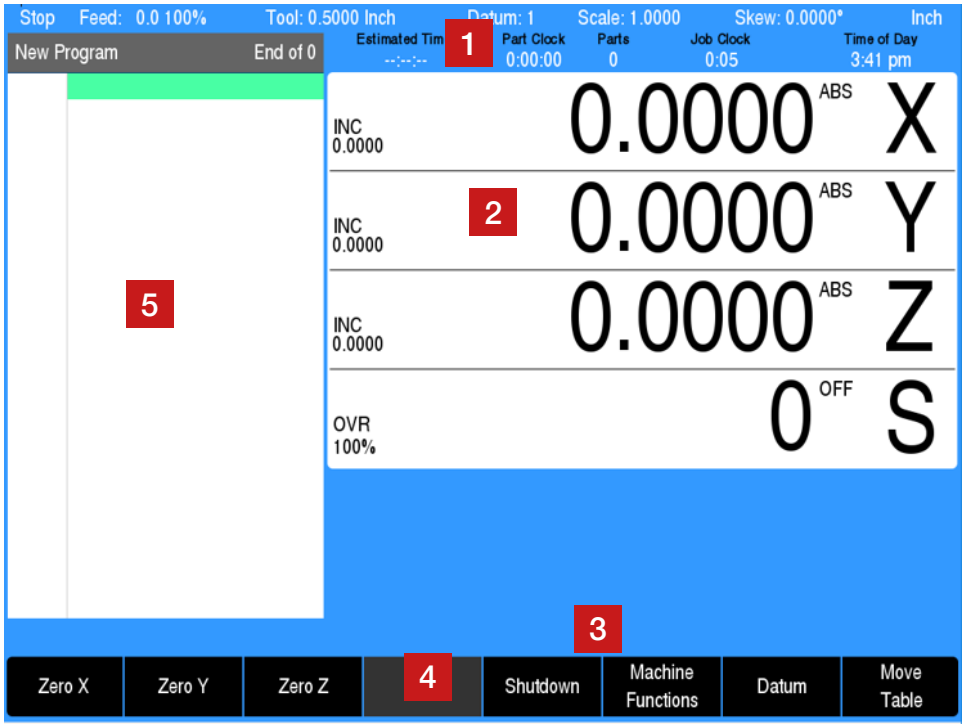
Notes

The Digital Readout (DRO)

Soft key functions are shown in the lower display area (4) and change as needed as you use the control.

Pressing the key directly below the Soft keys label activates the described function.

The DRO mode allows the use of manual machining operations along with the milling function hard keys. The Move Table Feature allows you to move the table and quill at rapid or at a set feed rate to power feed



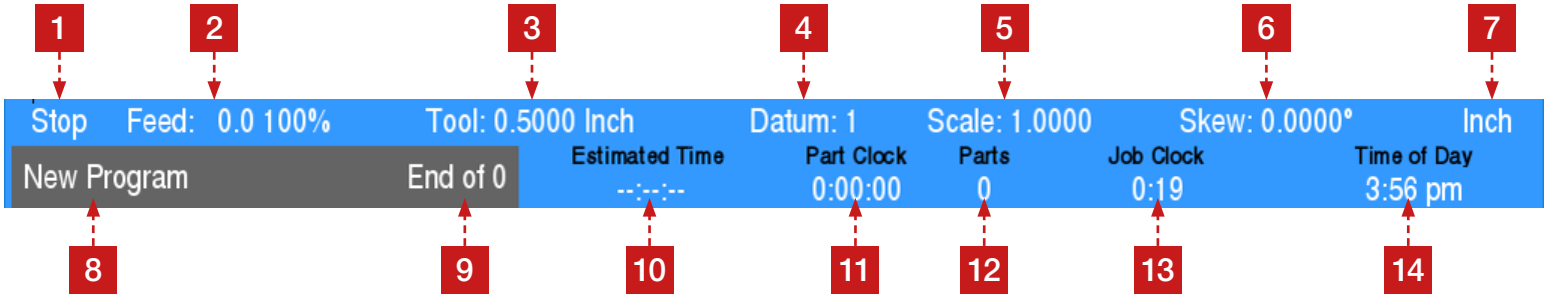
1. **Status Bar Display** for Servo Motor Status, Feed Rate, Tool, Datum, Scale, Skew, (Inch/MM), Estimated Time, Part Clock, Parts (run), Job Clock, and Time of Day
2. **Axes Display** (current position)
3. **Operator Intervention Message** line (OIM)
4. **Soft keys display area**
5. **Dialogue box display area** for milling functions

Available inactive soft key: black key with white text

Unavailable soft key: gray key with grey text

Active soft key: blue key and green text

Notes



Status Bar Display

1. **Servo Motor Status**
   On, Stop, Run, and Pause, indicates the servo motor status.

2. **Feed**
   Rate indicates actual feed rate. Percentage indicated the percent of feed rate override adjustment from the feed rate override knob.

3. **Tool**
   Indicates the current tool diameter and units. In PGM mode, indicates the programmed tool to be used for the highlighted step.

4. **Datum**
   Indicates the current datum in use.

5. **Scale**
   Indicates the amount of scaling factor being applied. 1.0000 equals no scaling.

6. **Skew**
   Indicates the amount of skew angle being applied to the X, Y plane.

7. **Inch/MM**
   Indicates current units in use. The current feed rate is displayed in these units per minute.

8. **Title of program**

9. **Line position / total lines**

10. **Estimated Time**
    Indicates approximate run time for the current program at 100% feed rate override.

11. **Part Clock**
    Times the cumulative running time of the current program. The timer is reset when a program is loaded.

12. **Parts**
    (Run) shows the number of times the current program was run to create a part.

13. **Job Clock**
    Shows total elapsed time for the current job (started and stopped by the operator).

14. **Time of Day**
    Shows the current time.

Notes

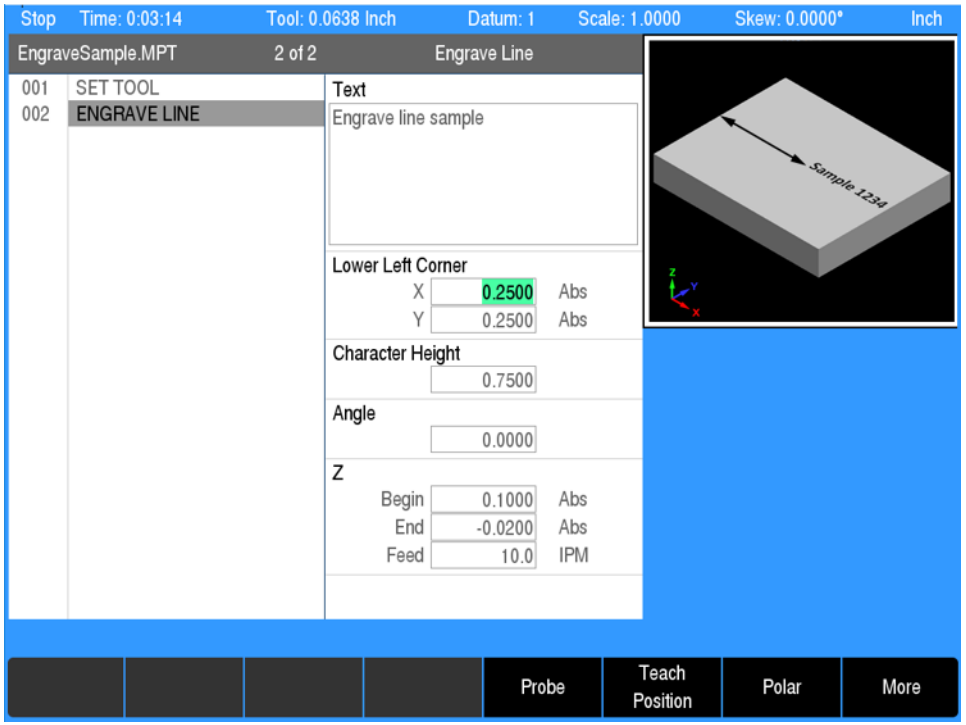
Context Sensitive Help

When assistance is needed, the User Manual can be displayed. To use this help, in this example, the console is in PGM mode, and a Linear Engraving cycle is being programmed.

INFO

Using Context Sensitive Help

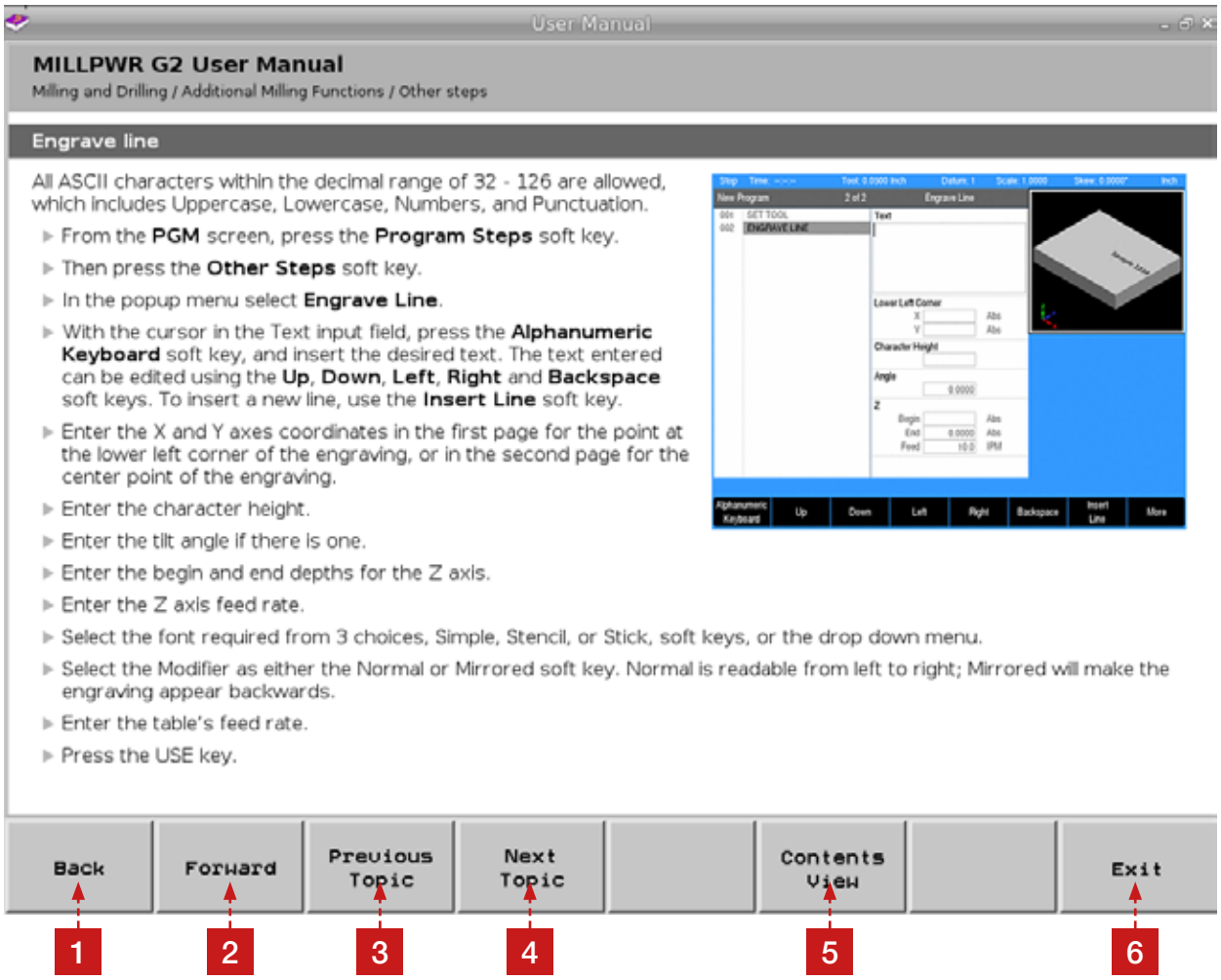
- Press the INFO key to open the on screen User Manual.
- A window will appear open to the section in the User Manual pertaining to the activity being performed.



Notes

Context Sensitive Help: Use of the soft keys

1. The **Back** soft key navigates one page back in history per key press. History is not cleared when the Help Screen is exited.
2. The **Forward** soft key navigates one page forward in history per key press.
3. The **Previous Topic** soft key navigates one Topic up in the contents window per key press.
4. The **Next Topic** soft key navigates one Topic Down in the contents window per key press.
5. The **Contents View** soft key is a toggle key to show/ hide the contents view on the left hand side of the display.
6. To close, press the **Exit** soft key.



NOTE: The **Previous Topic** and **Next Topic** soft keys will select the previous and next topic in the manual contents even if the **Contents View** is hidden.

Notes

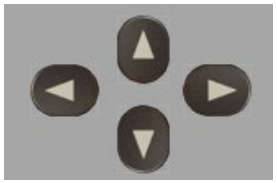
Jog Modes

Pressing the MOVE TABLE soft key activates the jog mode. Jogs are performed at the currently active feed rate. Any time Jog mode is active, pressing the CLEAR key opens a window that allows you to set the feed rate. All axes can be moved simultaneously.

Continuous Moves

To Jog the machine using continuous movement, press the Move Table soft key. Make sure none of the incremental Jog keys are selected.

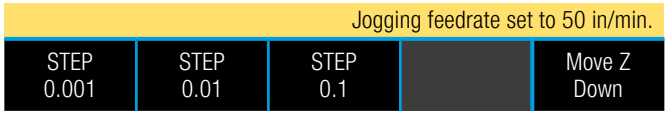
Press the appropriate ARROW key or a Move Z Soft Key to move in your desired direction. The machine will move at the current feed rate until the arrow key is released. The feed rate override knob can be used to control the feed rate.



Step Moves

Jog at one of the three predetermined step increments by pressing the appropriate soft key, then pressing an ARROW key or Move Z Soft Key to move in that direction. Each press of a direction key will initiate a move at the chosen increment. Press another step key to change the step distance or press the active step key to resume continuous jogging.

While the table is in motion, pressing the GO key will lock the feed and direction. This allow you hands free operation as the control feeds. Press the EXIT soft key to stop the control feed.

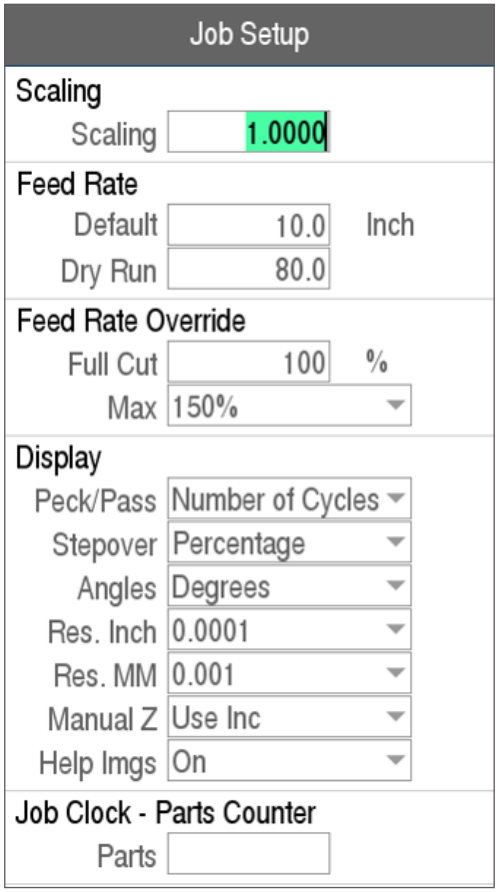


Notes

Job Setup

Parameters related to the job at hand are located in Job setup. These parameters are accessed from the DRO screen by pressing the SET-UP key.

SETUP



Scaling

This parameter allows you to adjust the finish size of a program by a scaling factor. The default is 1:1 or no scaling. A factor larger than one will grow the part size while a number less than one will shrink the part size. This is useful when creating a mold cavity for a material that shrinks to size after injection.

Feed Rate

- **Default** setting is the feed rate that will appear in a new program step. Choose a rate that you will use most of the time.
- **Dry Run** is the feed rate at which the machine will run when Dry Run is selected within Run Options.

Feed Rate Override

- **Max** is the maximum amount of override allowed when adjusted by the override knob.
- **Full Cut** percentage reduces the feed rate by this percentage when the tool is not able to stepover by the amount specified in a canned cycle.

Display

- **Peck:** This parameter changes the information requested for a peck between Number of cycles or Distance per peck.
- **Stepover:** Choose to display Percent of tool diameter, or distance per step.
- **Angles:** Choose Degrees Decimal or Degrees Minutes Seconds DMS. Use the decimal key to separate the degrees, minutes, and seconds. i.e. 47.22.12.
- **Res. Inch:** Choose the DRO Display resolution, 0.0001, 0.0005, or 0.0010.
- **Res MM:** Select Display resolution, 0.001, 0.005, or 0.010.
- **Manual Z:** Select Switch to Incremental, or Do Nothing. Incremental switches the DRO to show the incremental distance to the end of each move. Do nothing keeps the display at the current selected value of either Abs or INCR.
- **Help Images:** Automatic or On changes the graphic help screen display when programming.

PPT SLIDES 41-45

Notes

Job Setup

Parameters related to the job at hand are located in Job setup. These parameters are accessed from the DRO screen by pressing the SETUP key.

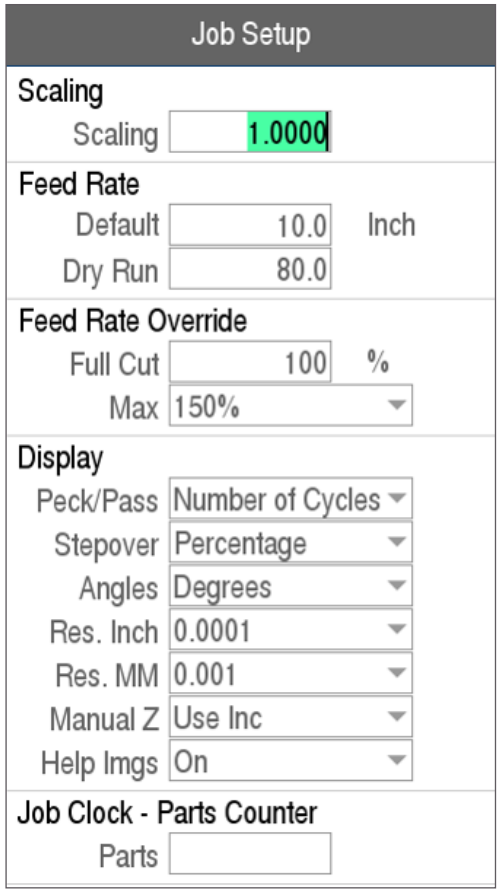
SETUP

Job Clock – Parts Counter

The job clock controls are accessed when this field is highlighted. Enter the starting number for the parts to be counted and adjust the number as needed during a run.

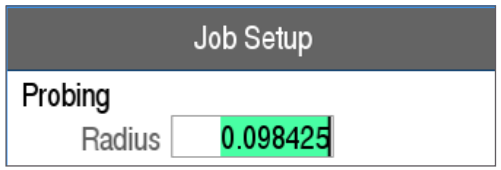
Pause Clock	Resume Clock	Reset Clock	More		Tool Table	Message Log	Install Setup

Job Clock and Parts Counter Keys



Probing Radius

Enter the radius of your electronic touch probe here. The radius will be adjusted for when probing cycles are used.



Notes

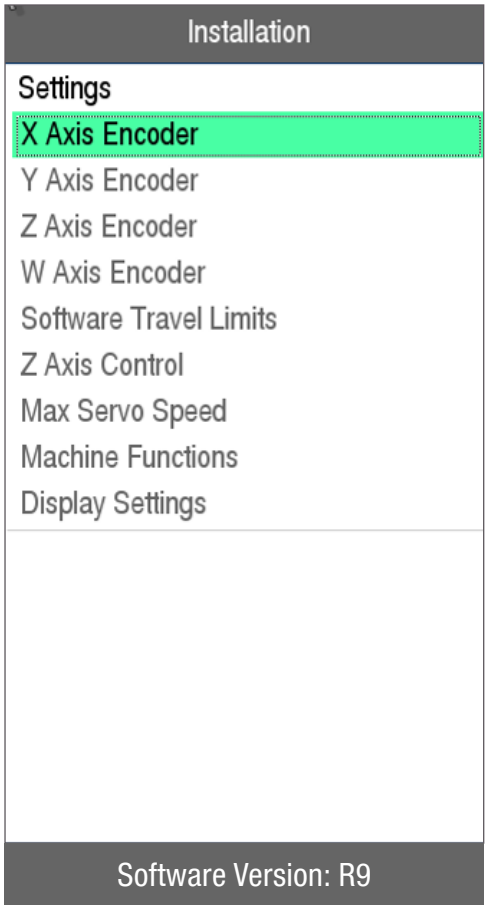
Install Setup

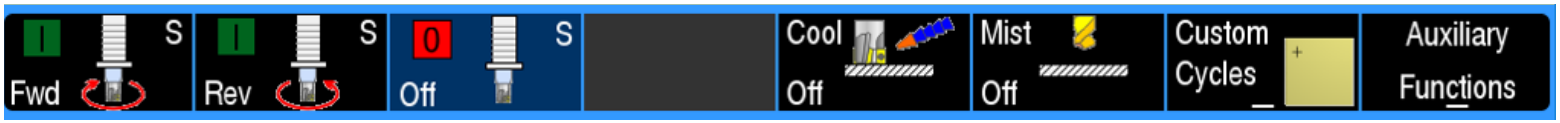
Parameters that effect machine performance are listed under Install Setup. These parameters are password protected to prevent unintentional modification. They are accessed from the DRO screen by pressing the SETUP key then the Install Setup soft key.

Operator password: 8891

Settings

- **X Y Z W Encoder:** These parameters are used for establishing the encoder resolution and count direction. If changed, the control can become unpredictable.
- **Table limits:** These parameters establish machine travel limits once home is established.
- **Z Axis Control:** Use this to disable and enable the Z-Axis. When disabled, the control is restricted to two axis operation.
- **Max Servo Speed:** Set the maximum servo speed of the control with this parameter. Tuning is effected with by this parameter and resetting to 100 ipm before tuning is recommended if changed.
- **Machine Functions:** Provides access to coolant and spindle function keys when the optional Auxiliary Machine Interface (AMI) is installed. Some spindle functionality requires an electronic variable speed spindle. Four additional outputs are available as well. Your current software version can be found here.



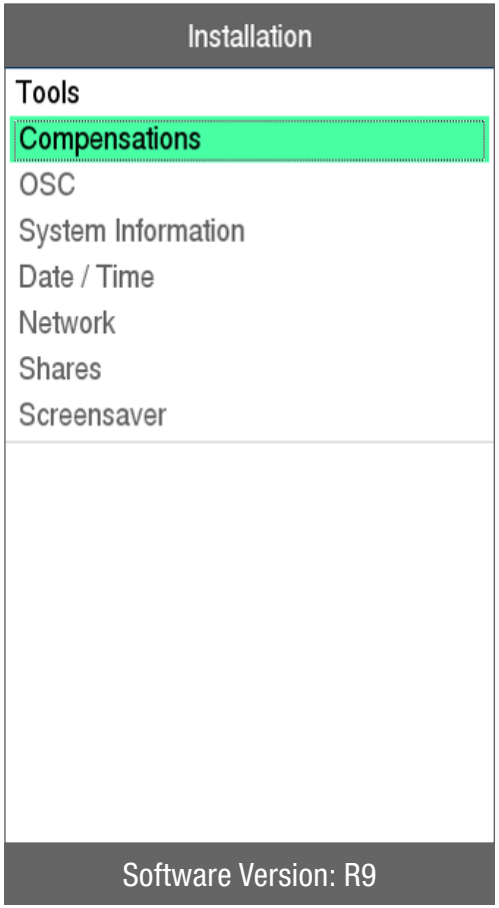


Notes

Install Setup

Tools

- **Compensation:** Adjust for minor linear error discrepancies between machine and control or enter a non-linear error compensation table determined by laser calibration.
- **OSC:** This is the oscilloscope where servo tuning via the analog wizard and machine accuracy can be measured.
- **System Information:** Lists hardware, features, and your serial number.
- **Date / Time:** Sets the local time and date.
- **Network:** Provides settings necessary to join to a data network.
- **Shares:** Sets up sharing within the network.
- **Screensaver:** Access to the system screen saver features.

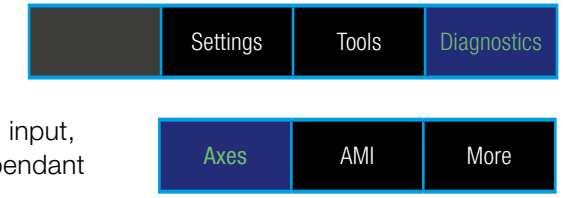


Notes

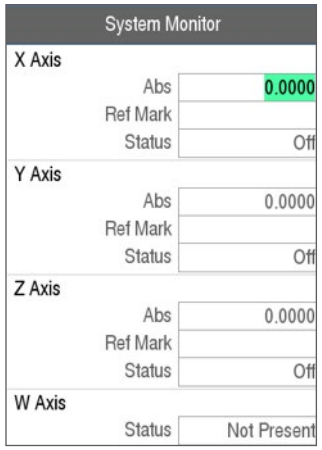
Install Setup

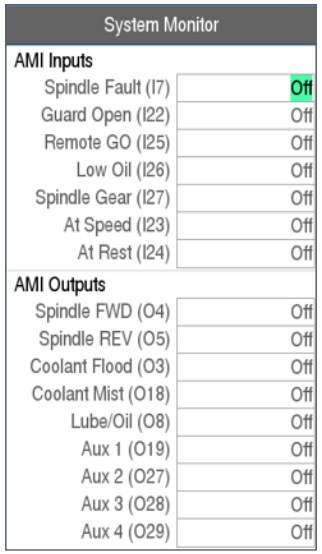
Diagnostics

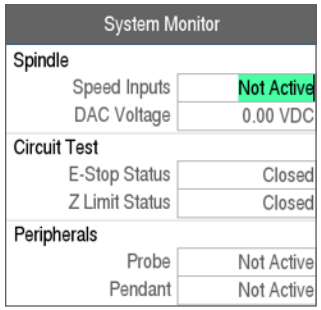
- **System Monitor:** Machine position and motor status and reference mark verification. AMI inputs and outputs, spindle speed input, e-stop status, probe status, and pendant status.

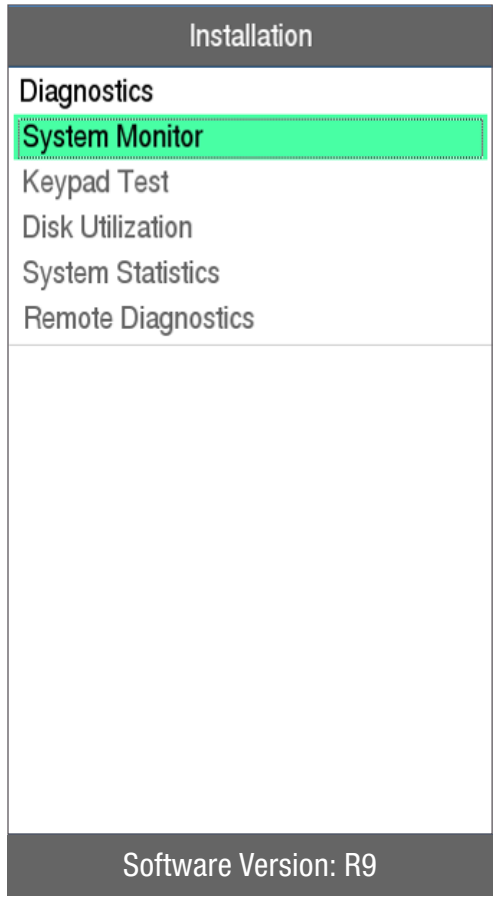


System monitor features









- **Keypad Test:** Test key functionality, press, release, and function.
- **Disk Utilization:** Used vs Free space, number of folders and files.
- **System Statistics:** Time on, Time servos active, Distances traveled.
- **Remote Diagnostics:** Allows a field service technician remote access to a control.

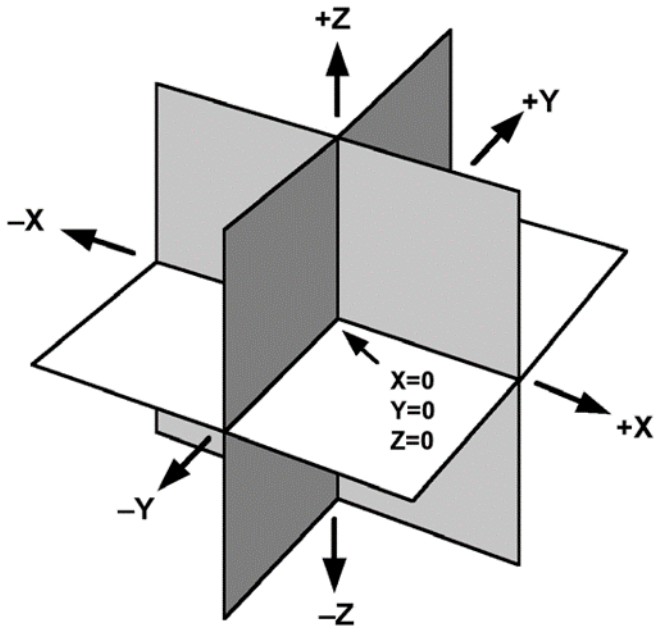
Notes

Axis Conventions

Count Direction

Machine movement is described with positive and negative numbers. Standard milling conventions use positive and negative count directions for the X, Y and Z-axis described by the following:

- **X axis:** The table moves to the left (tool moves to the right) for a positive count direction.
- **Y axis:** The table moves toward you (tool moves away from you) for a positive count direction.
- **Z axis:** The quill moves up from the table surface for a positive count direction.



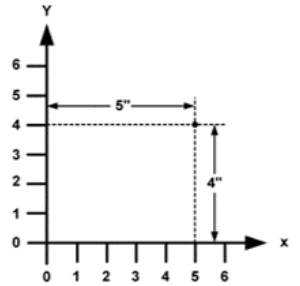
Cartesian Coordinates

A cartesian coordinate is a position described with X by Y values.

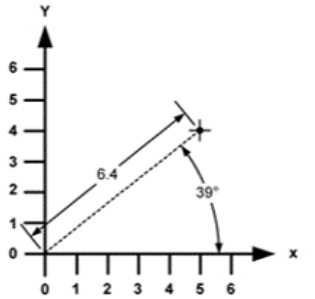
Polar Coordinates

A polar coordinate is a position described with an angle by radius. When you are dealing with angles, polar input may simplify programming.

Reference Point			
Point			
	X	5.0000	Abs
	Y	4.0000	Abs
Z			
	End	0.0000	Abs



Reference Point			
Point			
	R	6.4031	Abs
	A	38.6598	Abs
Z			
	End	0.0000	Abs



Notes

Absolute & Incremental Work Piece Positioning

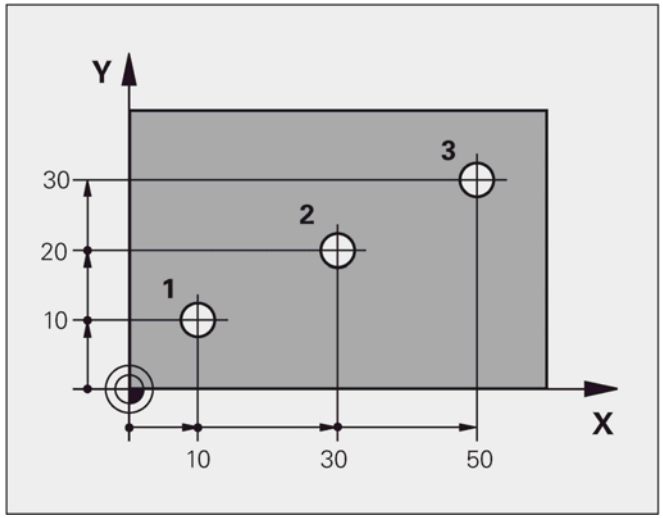
Absolute Work Piece Positions

Absolute coordinates are position coordinates that are referenced to a datum also known as work piece zero.

Absolute positions always refer to datum.

Example 1: Holes dimensioned in absolute coordinates

Hole 1	**Hole 2**	**Hole 3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm



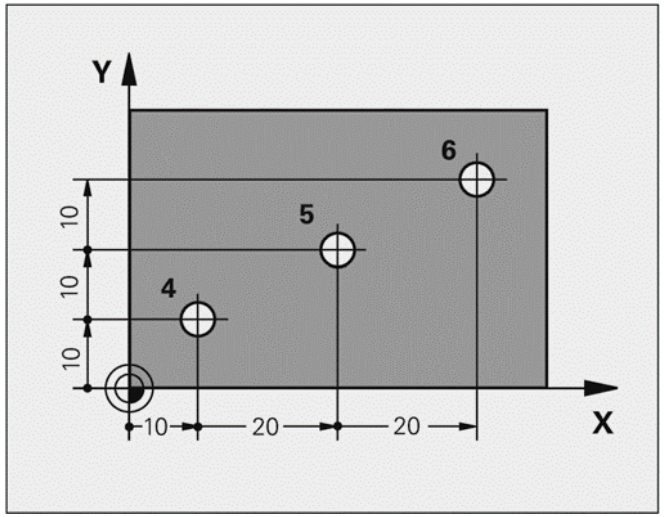
Example 1

Incremental Work Piece Positions

Incremental coordinates typically reference current location but can also reference a previous step within the program.

Example 2: Holes dimensioned in incremental coordinates

Hole 4 (Absolute)	**Hole 5 Inc from 4**	**Hole 6 Inc from 5**
X = 10 mm	X = 20 mm	X = 20 mm
Y = 10 mm	Y = 10 mm	Y = 10 mm



Example 2

Teach Positions

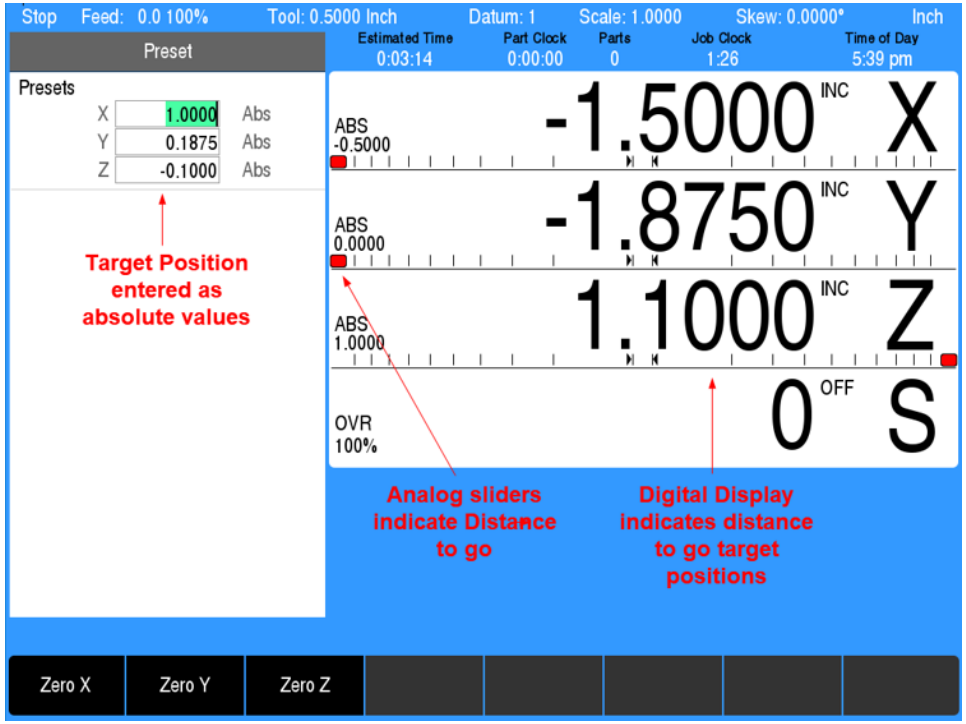
With the exception of Datum, whenever an X, Y, or Z coordinate is being entered, the Teach Position soft key will appear. When pressed, the current absolute position of the spindle is inserted in the active axis field. This allows you to transfer existing feature locations into a program step. Use the key-pad to adjust the taught position in order to compensate for the tool tip use to locate the feature and to adjust as otherwise required.

Notes

DRO Position Preset

To access the PRESET function, when in the DRO:

- Press the ABS/INCR key on the front panel to switch to the incremental mode.
- Press the Preset soft key or one of the axis keys to open the PRESET dialogue.
- Enter an X, Y, and or Z absolute value. The distance to this location will be preset in the incremental display. Manually move the machine to incremental zero to achieve this position.
- Press the CANCEL key at any time to cancel and exit the PRESET dialogue.



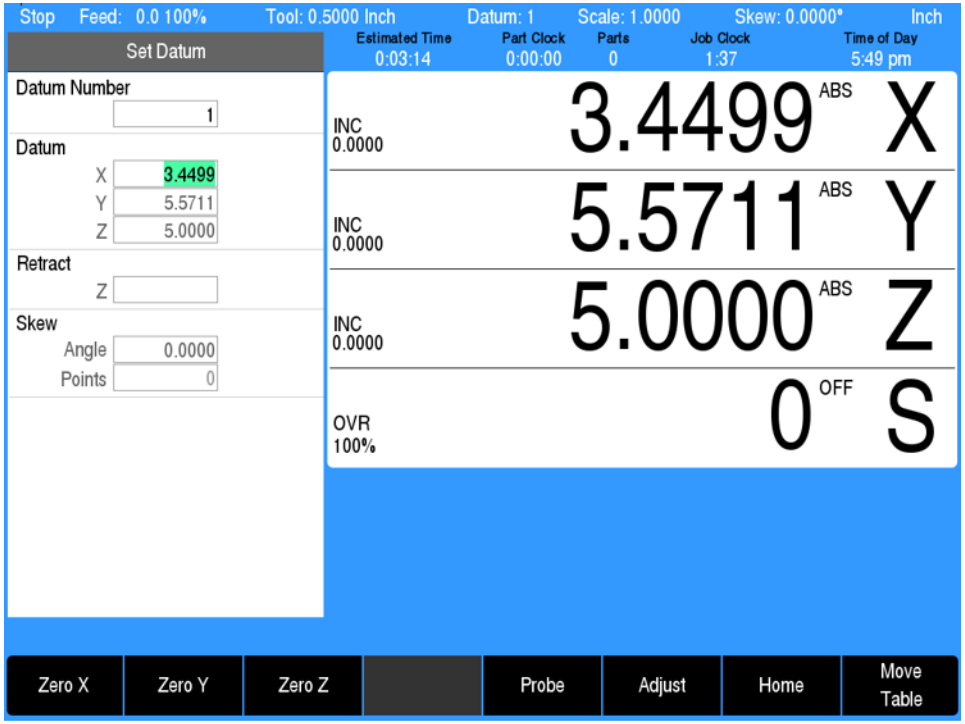
Soft keys are available to zero the incremental display for quick "zero and go" operations.

PPT SLIDE 57

Notes

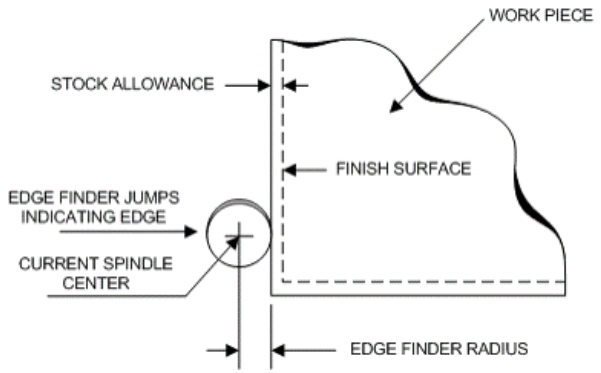
Establishing a Datum

Where you locate your datum will vary from job to job. Choose a location that best suits the job at hand. Placing it where the majority of your dimensions refer, will speed up programming and reduce errors. There may be a specified datum or multiple datum on a print. MILLPWRG2 allows you to set up to 99 datum locations. Insert an edge finder, indicator, or tool, into the spindle. Choose the device that best suits your needs. i.e. an indicator is best for locating a center point.



To establish Datum at a corner of your work-piece:

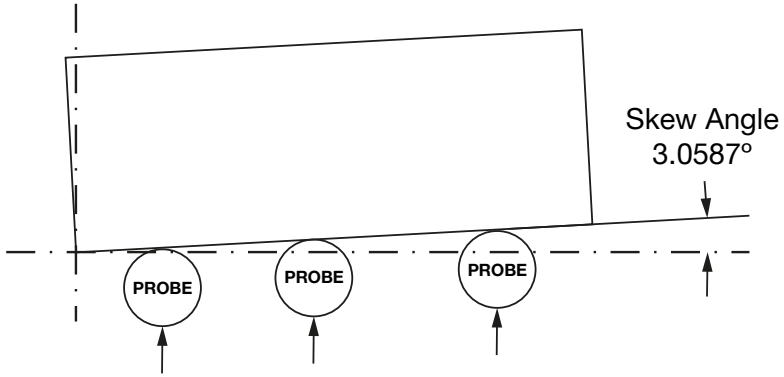
- From the DRO mode insure you are in absolute and press the Datum soft key.
- Enter a datum number from 1 to 99 as the datum to be defined.
- To locate an edge as illustrated to the right, move your tool or edge finder until it touches the work piece.
- Press the appropriate soft key to zero the axis you are setting, then raise the quill and move to center the tool on the edge or instead add or subtract the radius of the tool being used plus any stock allowance. If the datum is several inches further in the part you can enter this distance as well. Once entered, this location is set and you can maneuver to the other axis and repeat this process.
- When both the X and Y are located, place the tool you wish to use in the spindle and touch the top of the work piece.
- Press Zero Z to set the datum at the top of your work piece or enter the distance to the desired datum in the Z field.



Notes

Skew

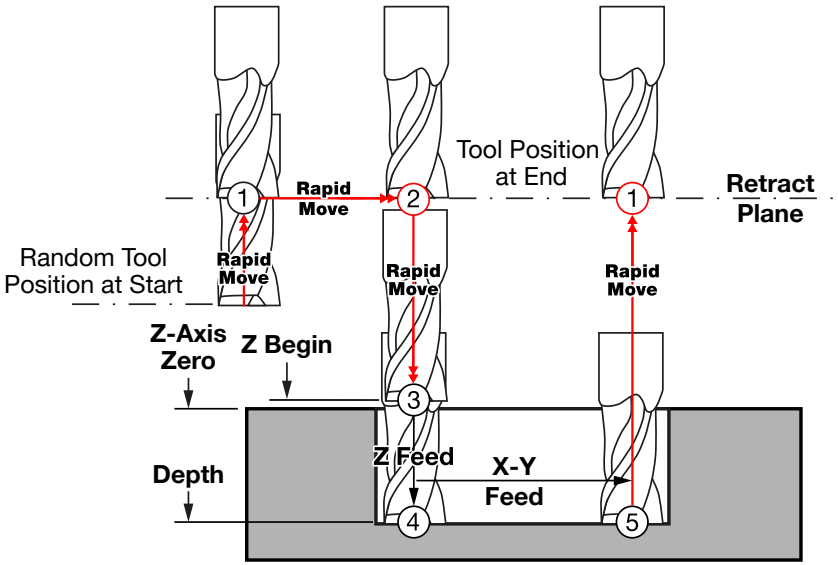
- The skew function compensates for the angle of your work piece when it is not parallel with either the X or Y axis.
- To compensate for the skewed part, touch off on two or more points along one axis, either X or Y.
- Use an electronic edge finder to enter points with each touch. With a mechanical edge finder or indicator, locate the edge and press the Teach Position soft key to enter points.



- Skewing adjusts the edge you choose making it parallel with the nearest axis of travel. Do not use points along a curve, along two different lines, or on a 45 degree angle.
- With rough surface, enter multiple points to more accurately calculate the skew angle. Locating points that are farther apart is also more accurate than points that are near each other.
- The skew feature does not work with G-code programs. Remove any skew angle prior to running a G-code program.

Retract

The retract is a plane above datum zero from where the tool begins a cycle and then returns to upon completion. All rapid movement between cycles occurs at this plane. By setting a retract position, you can ensure that the tool is clear when moving from one position to the next. Enter a Z axis retract position sufficient to clear the work piece and fixturing or press the CLEAR key to clear any value. With no value, the quill will revert to the upper travel limit between steps.



PPT SLIDES 61-62

Notes

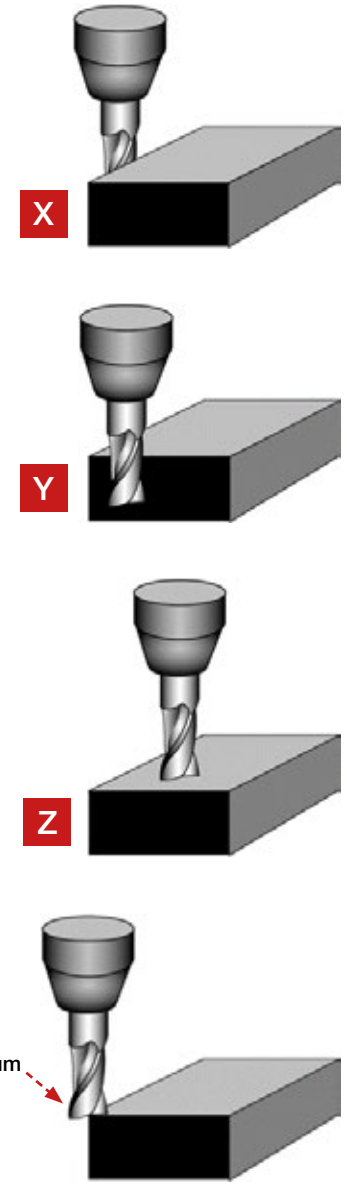
Example: Setting Datum at the Top Front Left Corner

Datum

- Lower the tool then hand crank the table along the X axis, slowly spinning the tool by hand as you go (Place the spindle in neutral for this). Stop when the tool contacts the part.
- Using the keypad, enter into the X field the location of the spindle center from the edge (tool radius). In the illustration to the right, the value would be negative because the tool's center is on the negative side of the edge.
- Press ENTER or the down arrow to move to the next field. Position the tool in front and move the Y axis, slowly spinning the tool by hand as you go until the tool contacts the part.
- Using the keypad, enter into the Y field the location of the spindle center from the edge (tool radius). In the illustration to the right, the value would be negative because the tool's center is on the negative side of the edge.
- Press the ENTER or the down arrow to move to the next field. Position the tool so that its tip touches the top surface of the part.
- Press the Z = 0 soft key or enter a zero in the Z axis field.
- Press ENTER. With the data entered, the cursor will highlight the next field.

Retract

The retract is a plane above datum zero from where the tool begins a cycle and then returns to upon completion. All rapid movement between cycles occurs at this plane. By setting a retract position, you can ensure that the tool is clear when moving from one position to the next. Enter a Z axis retract position sufficient to clear the work piece and fixturing or press the CLEAR key to clear any value. With no value, the quill will revert to the upper travel limit between steps.



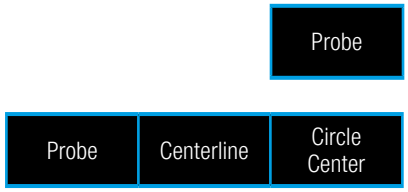
Notes

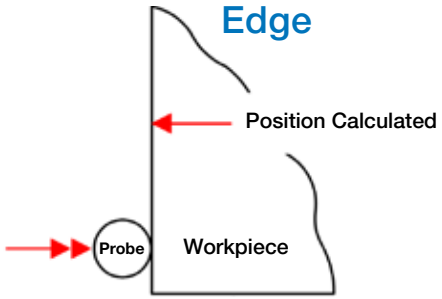
Using the Probing Cycles

When in the Datum field, three probing cycles are available to assist you.

- Press the Probe soft key to access three probing cycles: Edge, Centerline and Circle Center.
- Press the Use Tool soft key if you are not using a KT130 or KT3D touch probe.
- Press the Edge, Centerline, or Circle Center soft key to select the operation you wish to use.
- Move to the edge of your work piece. If using a touch probe, contact with the part will trigger the entry or if using a mechanical edge finder press the Teach

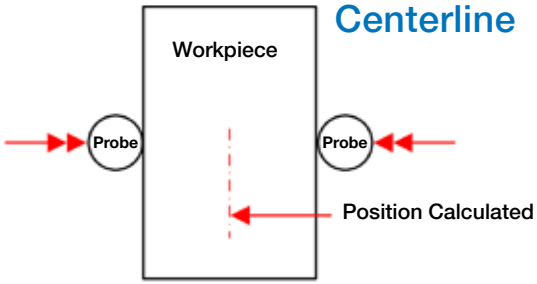
  soft key when the indicator triggers.





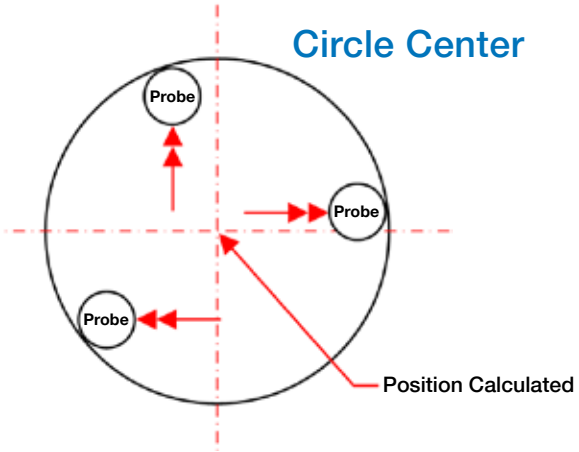
Edge

Sets the datum with a single trigger of the edge finder. The system will prompt to “Move to edge.” Start movement until the edge finder is triggered. Once triggered, the system will exit the probing mode and set datum for the moving axis. The probe tip compensation is automatically adjusted by the value entered in the job setup parameter. This also works when using a probe while in the regular datum entry field.



Centerline

Sets the datum with 2 triggers of the edge finder, bisecting the distance between the two points. The axis that is used for the first trigger must also be used for the second. Centerline works for only one axis of travel at a time.



Circle Center

Sets the datum using 3 points of a circle. This operation functions in the X - Y axis by touching three points along a circle or radius. Both X and Y will be established with this operation. The farther apart your three points are, the more accurate the center point will be.

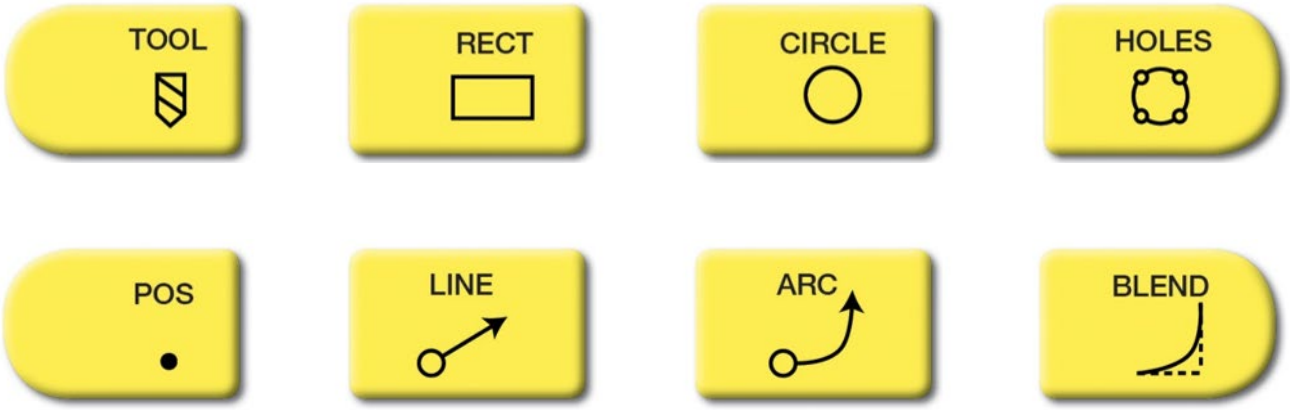
PPT SLIDES 65-68

Notes

MILLPWR G2

Single-Step Milling Functions

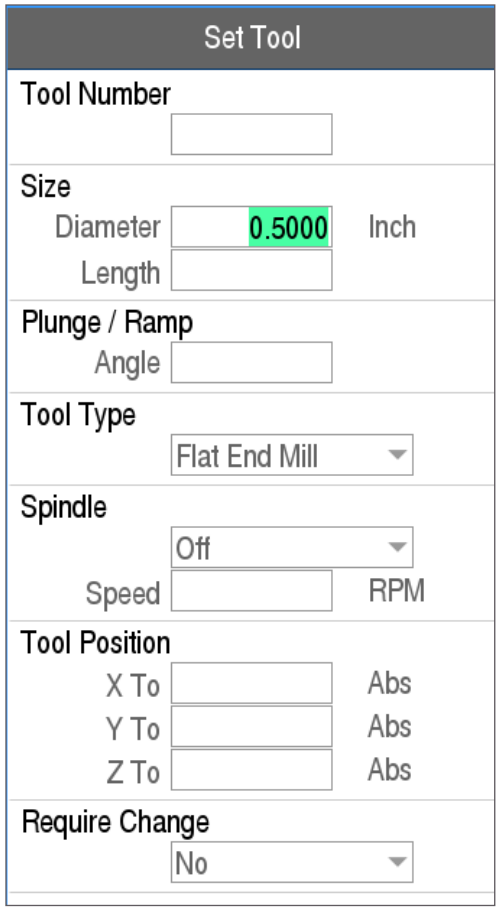
Single-Step Milling Functions can be run as stand alone operations. With the exception of the blend key, each of these keys offer at least one single-step cycle.



Notes

Set Tool

From the DRO, press the tool key to open the tool dialog window. Enter the tool information then press Use or GO to initiate the tool change. When completed the current tool data will update to indicate the new tool being used. In a program press Use to save the tool step in a program.



Tool Number

Enter a number to select a pre-established tool from the tool library. Do not use a number if defining the tool within the step.

Diameter

Used when left or right offset is used. Diameter is the only mandatory information required.

Length

An offset used to offset the tool relative to Z datum.

Plunge / Ramp Angle

Sets the angle the tool enters the cut.

Tool Type

Helps you differentiate tools with similar diameters.

Spindle

Direction and speed can be entered and you will be prompted to set these at run time or G2 will be activate the spindle if equipped to do so.

Tool Position

A location the machine will move to prior to pausing for the change. Enter one or more axis value as needed.

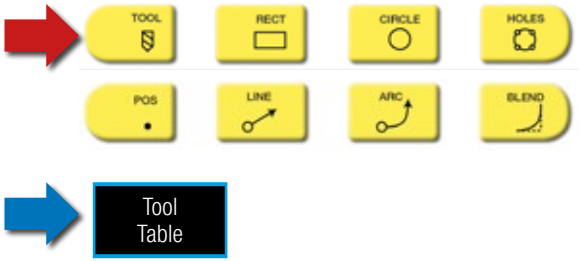
Require Change

- When set to “yes” will force the tool change even when the current tool matches the new tool.
- When set to “no,” the tool change is skipped when the tool matches.

Notes

Tool Library

From the DRO, press the tool key to open the tool dialog window then press the Tool Table soft key to enter the tool library. Arrow down to the tool number you wish to edit and press ENTER.



Tool Number

Use the arrow keys to highlight the tool number you wish to add or edit.

Diameter

Used when left or right offset is used. This is the only mandatory information required.

Length

An incremental adjustment used to offset the tool relative to Z datum. After insuring no offset is active, touch a tool to datum zero and press the Teach soft key to calibrate.

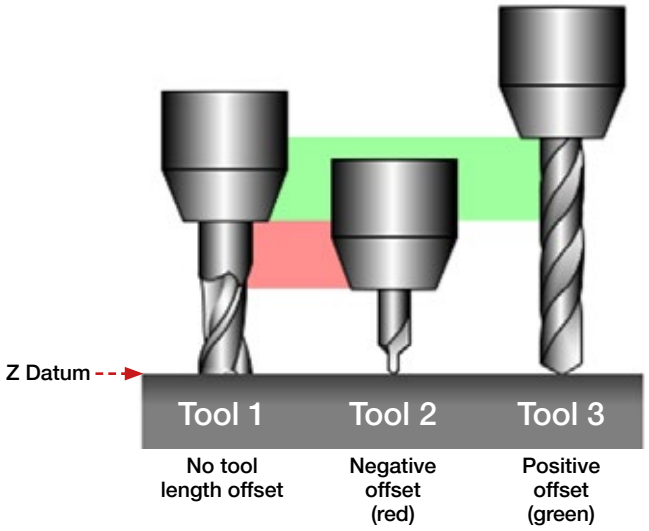
Type

Helps you differentiate tools with similar diameters. Press enter and use the arrow keys to select.

Tool	Diameter	Length	Type	Plunge Angle
1	0.25	0	Flat End Mill	5
2	0.3126	0	Flat End Mill	15
3	0.5	0	Flat End Mill	90
4	0.75	0	Roughing Mill	25
5	0.125	0	Pilot Drill	0
6	0.25	0	Pilot Drill	0
7	0.25	0	Drill	0
8	0.4126	0	Drill	0
9	6	0	Fly Cutter	0
10	0.5	0	Special Mill	30
11	0	0		30
12	0	0		30

Plunge Angle

The angle of entry a tool will use when feeding to programmed depth. 0° = 90°.

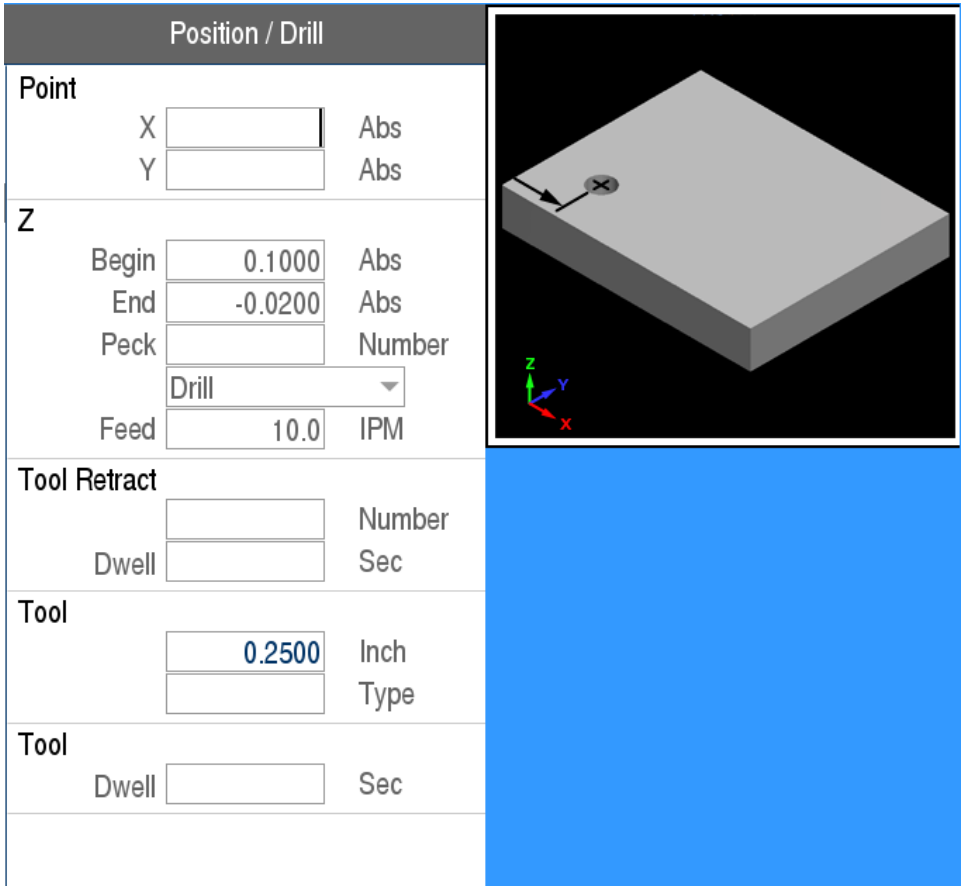


PPT SLIDES 72-73

Notes

Position / Drill

The POS key opens the POSITION / DRILL dialogue. The Position / Drill function will move the table to specific position and execute a drilling, boring, or positing (2-Axis drilling) operation.



Point

Enter the X and Y position to locate the single drill operation.

Z

- **Begin** is where the tool will rapid to. From there it will **Feed** at the given rate to the End value. Feed The given rate to the
- **Peck** will lift the tool by 0.01” at every peck you define by number or distance. With the Drill option selected, the tool will feed down and rapid up. With bore selected, the tool will feed in both directions. With position selected the operation reverts to a two axis operation pausing at each location until GO is pressed, useful for manually tapping.

Tool Retract

Causes the tool to lift to the Z begin. When retracted, the tool will dwell for the number of seconds entered to allow the tool and work piece to cool down if needed.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.

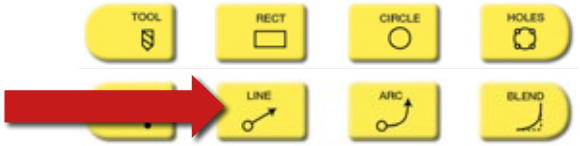
Dwell

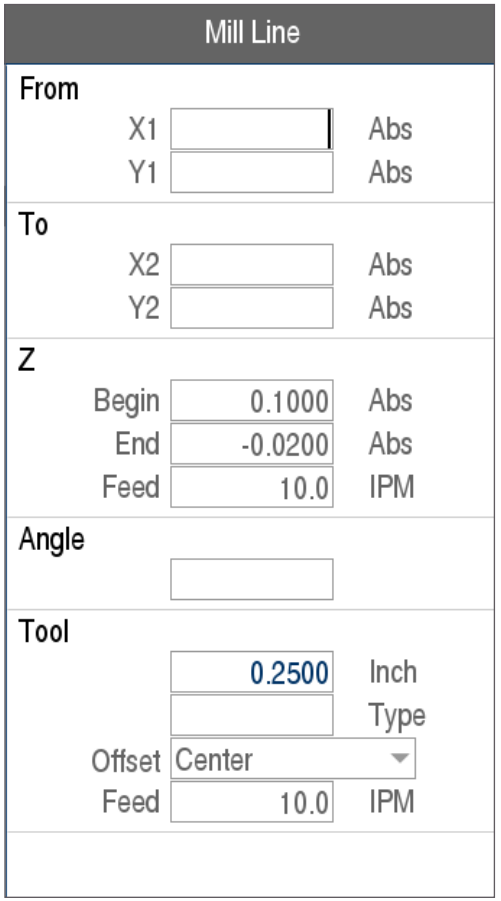
The number of seconds the tool will dwell at the bottom of the cycle and is useful to insure the bottom is finished and smooth.

Notes

Mill Line

The LINE key opens the line dialogue box allowing you to define a line from beginning to end. This step is capable of running as a stand alone operation in both DRO and PGM mode and can also be also be part of a continuous path in PGM mode when programming a continuous path. The begin point, Z information, offset and feeds are carried forward in PGM mode so you do not need to enter redundant information. These can be changed if needed.





From

X1 Y1 Define the start point of the line. These can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

To

X2 Y2 The end of the line and can be cartesian or polar, absolute (ABS) or incremental (INC) from the begin point or an external reference point.

Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the End value at the **Feed** provided.
- This feed is for the Z axis only. The feed below is for the X, Y plane.
- The tool will then proceed to the “To” point. It then rapids up to the retract plane set in datum.
- If the line is part of a continuous tool path the tool will not lift at the end but continue along the established path.
- Changing the “End” will break up a continuous path.

Angle

Allows you to create a line by programming to an X or a Y point at this given angle. i.e. from X0, Y0, to X2 at 20 degrees.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to remind you of which will be used.

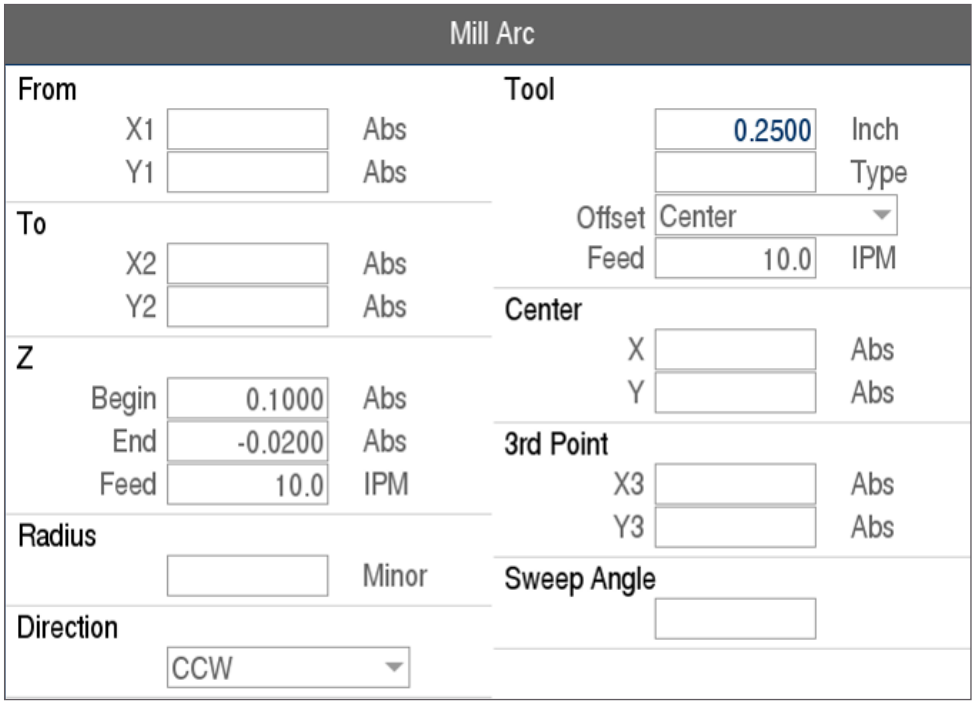
- **Offset:** Allows you to specify left, right, or center. Changing the offset will breakup a continuous path.
- **Feed:** Establishes the rate at which the tool will feed along the X, Y plane. This can be changed without breaking a continuous path.

PPT SLIDES 76-78

Notes

Mill Arc

The ARC key opens the arc dialogue allowing you to define an arc with both beginning and end. This step is capable of running as a stand alone operation in both DRO and PGM mode and can also be part of a continuous path in a program. When used in a continuous path, the begin point, Z Information, offset and feeds are carried forward so you do not need to enter redundant information. These can be changed if needed.



From

X1 Y1 define the start point of the arc. These can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

To

X2 Y2 is the end of the arc and can be cartesian or polar, absolute (ABS) or incremental (INC) from the begin point or an external reference point.

Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value at the Feed provided.
- This feed is for the Z axis only. The feed below is for the X, Y plane.
- The tool will then proceed to the “To” point. It then rapids up to the retract plane set in datum.
- If the line is part of a continuous tool path the tool will not lift at the end but continue along the established path.
- Changing the “End” will break up a continuous path.

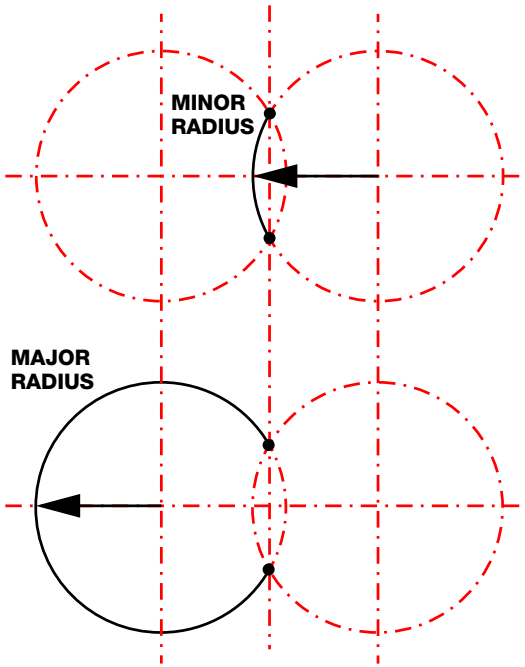
Notes

PPT SLIDES 79-80

Mill Arc

Radius

Establishes the size of the arc. Minor or Major can be established. Minor arcs are less than 180 degrees while Major arcs are greater than 180 degrees.



Mill Arc

From			**Tool**		
	X1	Abs		0.2500	Inch
	Y1	Abs			Type
To				Offset: Center	
	X2	Abs		Feed: 10.0	IPM
	Y2	Abs	**Center**		
Z				X	Abs
	Begin: 0.1000	Abs		Y	Abs
	End: -0.0200	Abs	**3rd Point**		
	Feed: 10.0	IPM		X3	Abs
Radius				Y3	Abs
		Minor	**Sweep Angle**		
Direction					
	CCW				

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction.

Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to remind you of which will be used.
- **Offset:** Allows you to specify left, right, or center. Changing the offset will breakup a continuous path.
- **Feed:** Establishes the rate at which the tool will feed along the X, Y plane. This can be changed without breaking a continuous path.

Notes

Mill Arc

Center

Allows you to define the center point of an arc. This is useful if a radius is not defined on your print. Do not enter a radius prior to entering the center point.

3rd Point

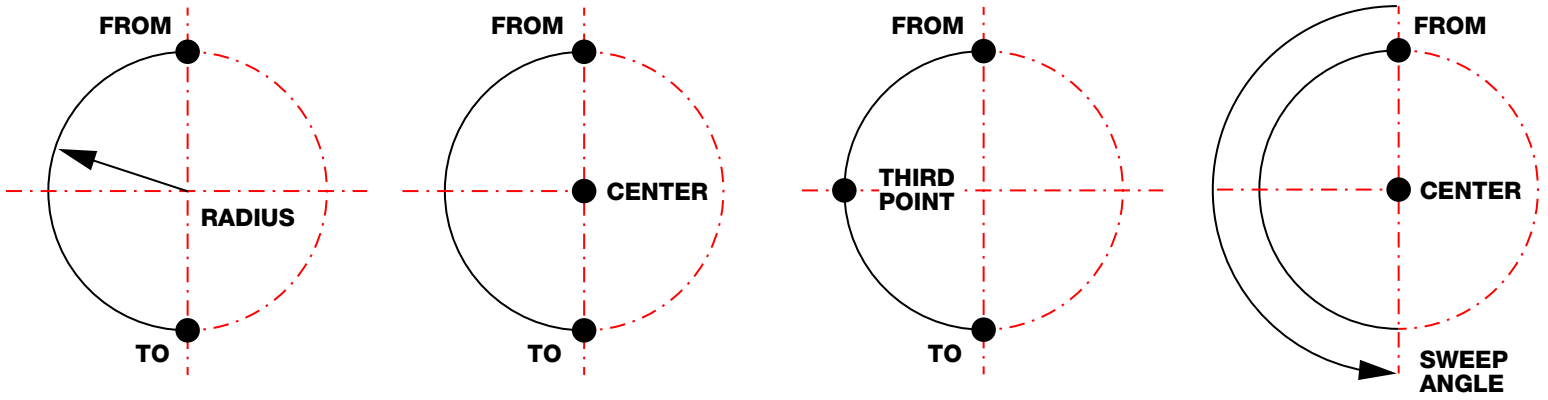
Provides you with another method of defining an arc. By programming a “From,” “To” and “3rd Point,” an arc can be determined. The “3rd Point” does not have to be within the “From” and “To” points.

Sweep Angle

An alternative entry for when the “From” or “To” point is unknown. If a “From” point is entered with a radius and a sweep angle, the “To” point can be calculated.

Mill Arc

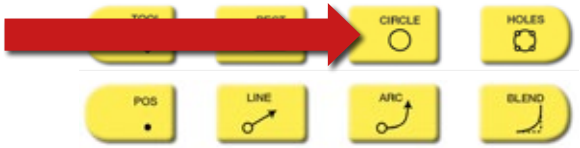
From			**Tool**		
	X1	Abs		0.2500	Inch
	Y1	Abs			Type
To				Offset: Center	
	X2	Abs		Feed: 10.0	IPM
	Y2	Abs	**Center**		
Z				X	Abs
	Begin: 0.1000	Abs		Y	Abs
	End: -0.0200	Abs	**3rd Point**		
	Feed: 10.0	IPM		X3	Abs
Radius				Y3	Abs
		Minor	**Sweep Angle**		
Direction					
	CCW				



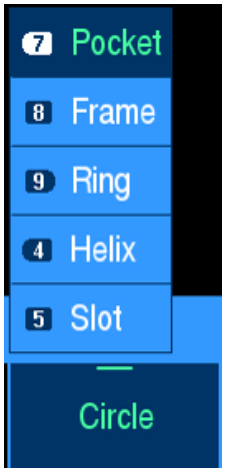
PPT SLIDE 83

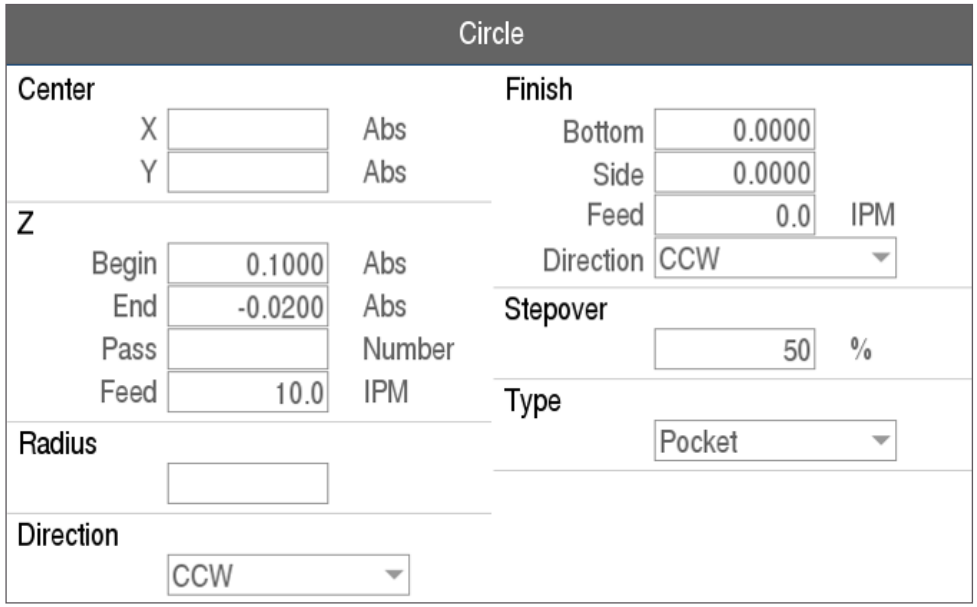
Notes

Circle Pocket



The CIRCLE key activates a pop up list of circle features. Highlight the Pocket option and press ENTER or press 7 on the keypad to select Pocket. Circle pocket allows you to define a complete stand alone circular pocket. This step is capable of running in both DRO and PGM mode.





Center

X Y define the center point of the circle. This can be cartesian or polar absolute (ABS) values relative to Datum Zero.

Z

- Begin is where the tool will rapid to. From there it will feed at the given rate to the End value at the Feed provided.
- This feed is for the Z axis only.
- Pass will divide the operation by the number of times or by the distance you program providing multiple passes to final depth.

Radius

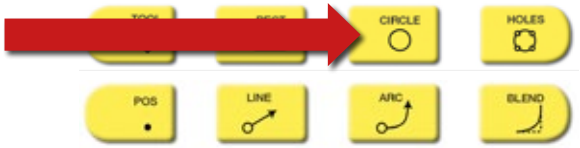
Establishes the size of the pocket.

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction.

Notes

Circle Pocket



Tool

Cannot be changed here but shows the tool information of the last tool step programmed to remind you of the tool that will be used.

Feed

Establishes the rate at which the tool will feed along the X, Y plane.

Finish

- **Bottom** amount of stock to leave for the final finish pass
- **Side** amount of stock to leave for the final finish pass
- **Feed** rate and **Direction** to use

Stepover

Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

Type

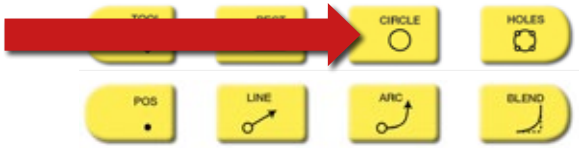
Allows you to choose contour if you wish to use the shape to define an Island within a custom pocket.

Circle

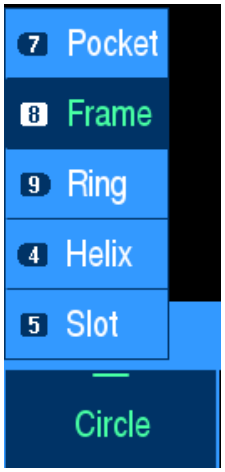
Center			**Finish**		
X		Abs	Bottom	0.0000	
Y		Abs	Side	0.0000	
Z			Feed	0.0	IPM
Begin	0.1000	Abs	Direction	CCW	
End	-0.0200	Abs	**Stepover**		
Pass		Number		50	%
Feed	10.0	IPM	**Type**		
Radius				Pocket	
Direction					
	CCW				
Tool					
	0.2500	Inch			
		Type			
Feed	10.0	IPM			

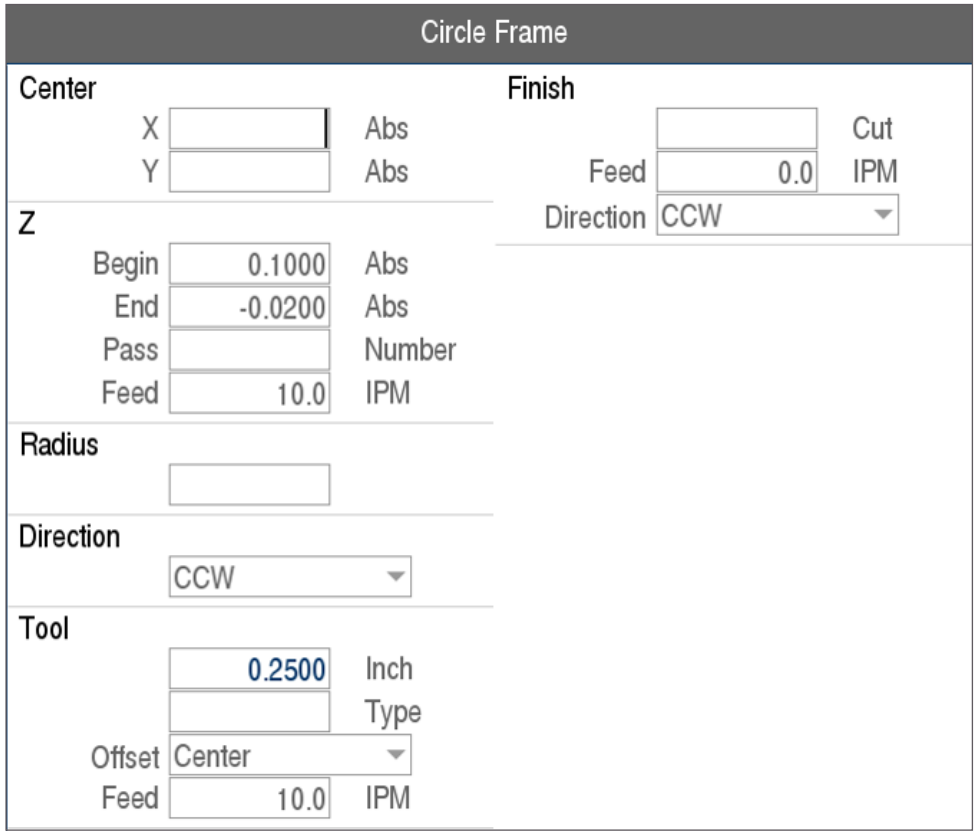
Notes

Circle Frame



The CIRCLE key activates a pop up list of circle features. Highlight the Frame option and press ENTER or press 8 on the keypad to select Frame. Circle frame allows you to define a complete stand alone circular frame. This step is capable of running in both DRO and PGM mode.





Center

X Y define the center point of the circle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value at the **Feed** provided.
- This feed is for the Z axis only.
- **Pass** will divide the operation by the number of times or by the distance you program providing multiple passes to final depth.

Radius

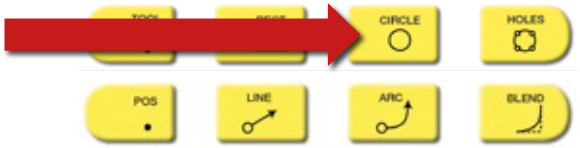
Establishes the size of the arc. Minor or major can be established.

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction.

Notes

Circle Frame

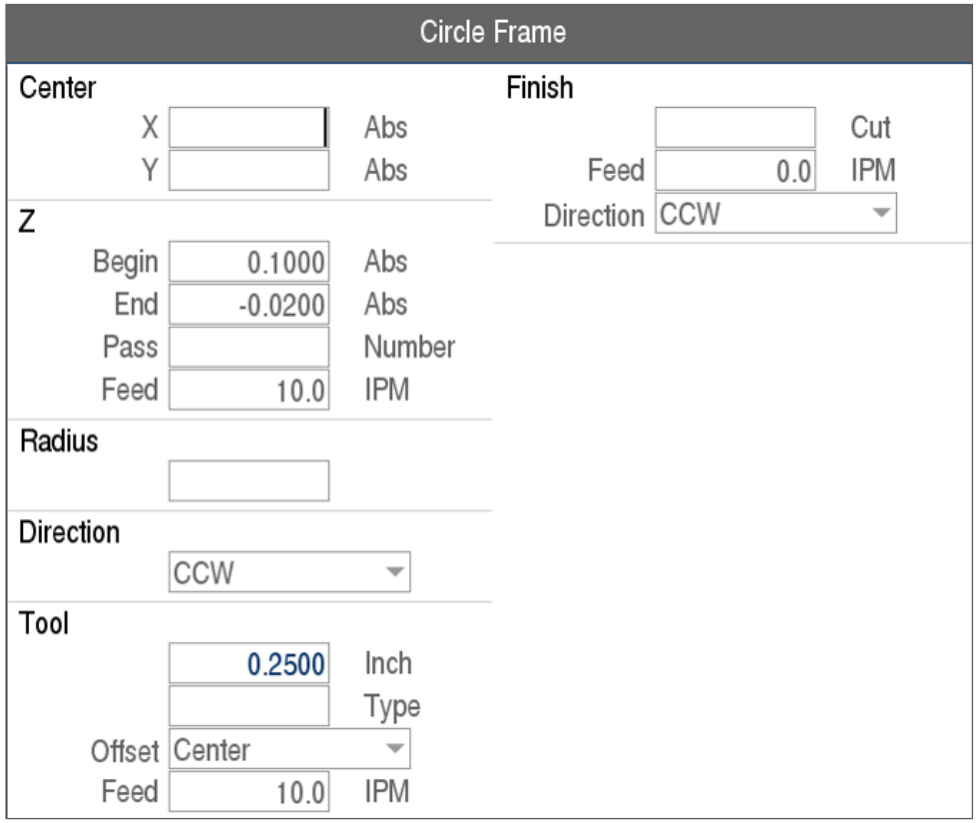


Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.
- **Offset:** Allows you to specify left, right, center, inside, or outside.
- **Feed:** Establishes the rate at which the tool will feed along the X, Y plane.

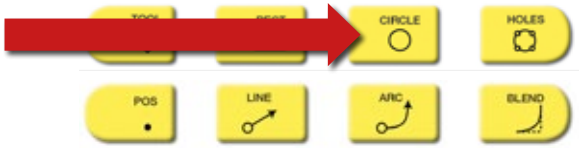
Finish

- **Cut:** Establishes the amount of stock to leave for the final finish pass
- **Feed** rate and **Direction** to use

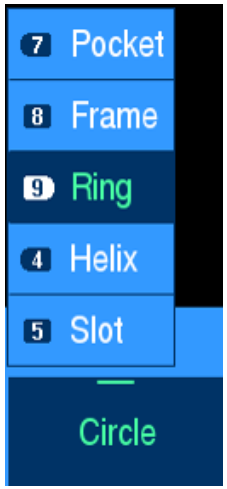


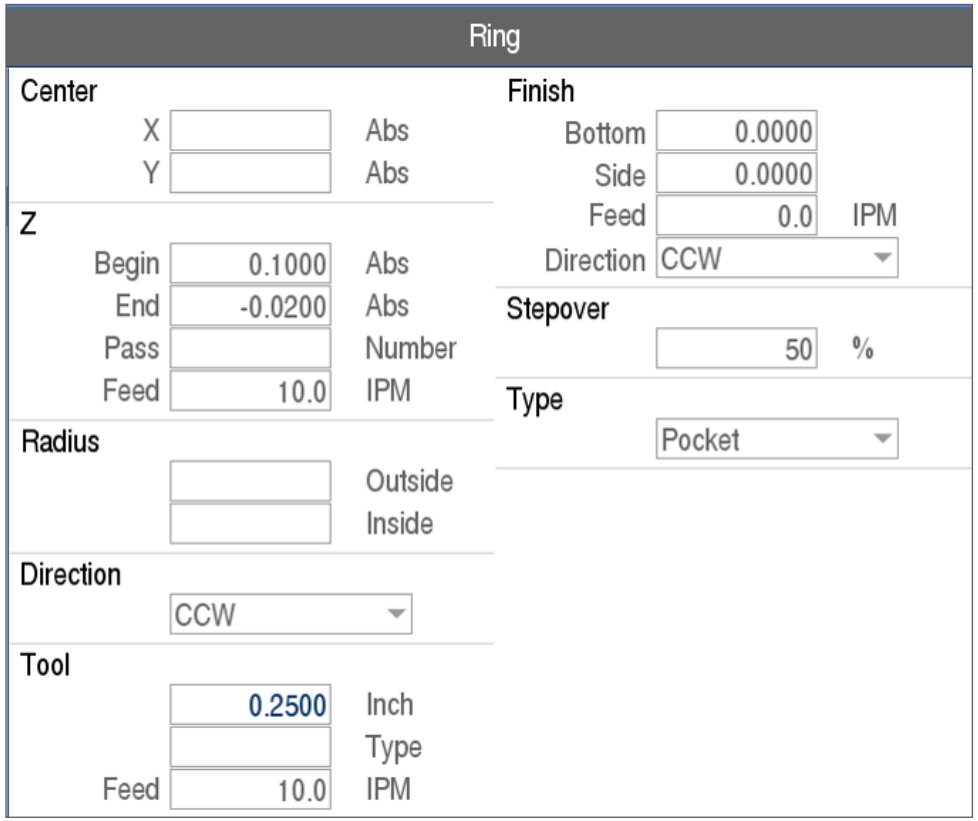
Notes

Circle Ring



The CIRCLE key activates a pop up list of circle features. Highlight the Ring option and press ENTER or press 9 on the keypad to select ring. Circle ring allows you to define a complete stand alone circular ring. This step is capable of running in both DRO and PGM mode.





Center

X Y define the center point of the circle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Z

- **Begin** is where the tool will rapid to. From there it will feed at the given **Feed** rate to the **End** value.
- This feed is for the Z axis only.
- **Pass** will divide the operation by the number of times or by the distance you program providing multiple passes to final depth.

Radius

- Establishes the size of the ring.
- **Outside** establishes the outer radius.
- **Inside** establishes the inner radius.

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction.

Notes

Circle Ring

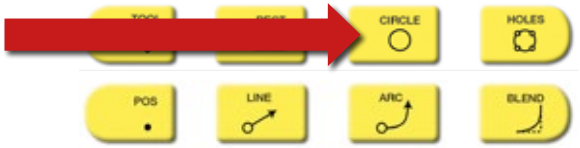
Tool
Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.

Feed
Establishes the rate at which the tool will feed along the X, Y plane.

Finish
Establishes the amount of stock (Cut) to leave for the final finish pass along with the Feed rate and Direction to use.

Stepover
Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

Type
Allows you to choose contour if you wish to use the shape to define an Island within a custom pocket.

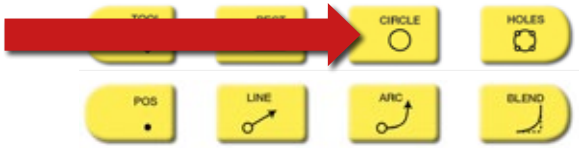


Ring

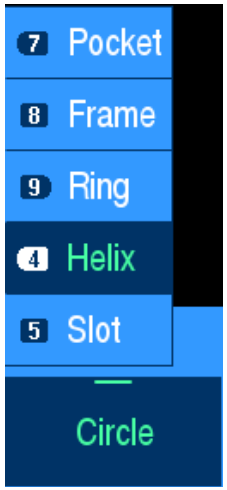
Center			**Finish**		
X		Abs	Bottom	0.0000	
Y		Abs	Side	0.0000	
Z			Feed	0.0	IPM
Begin	0.1000	Abs	Direction	CCW	
End	-0.0200	Abs	**Stepover**		
Pass		Number		50	%
Feed	10.0	IPM	**Type**		
Radius				Pocket	
		Outside			
		Inside			
Direction					
	CCW				
Tool					
	0.2500	Inch			
		Type			
Feed	10.0	IPM			

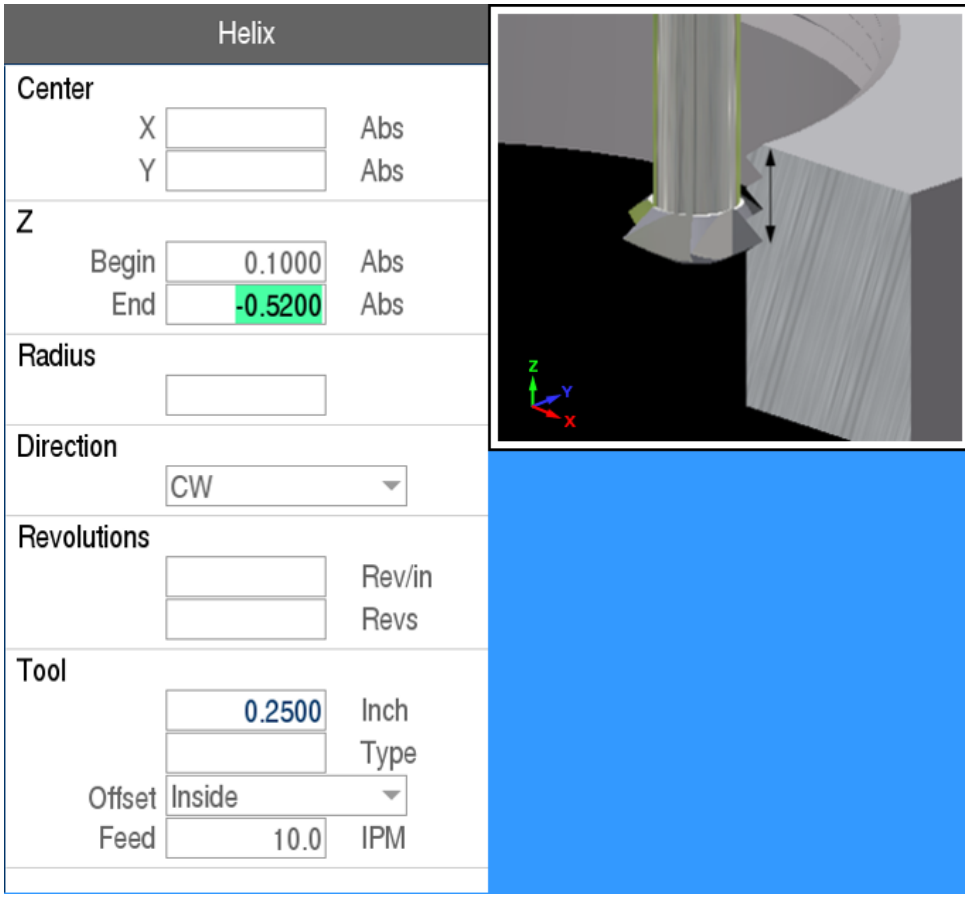
Notes

Helix



The CIRCLE key activates a pop up list of circle features. Highlight the Helix option and press ENTER or press 4 on the keypad to select Helix. Helix allows you to define a complete stand alone Helix. This step is capable of running in both DRO and PGM mode. Helix allows you to mill threads both inside and outside, left and right handed.





Center

X Y define the center point of the circle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Z

- **Begin:** Where the tool will rapid to.
- **End:** Feeds along the given pitch to the end value provided then rapids up to the retract plane as set in datum.

Radius

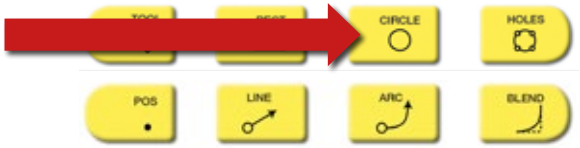
Establishes the size of the Helix.

Direction

Establishes clockwise (CW) for right handed or counter clockwise (CCW) for left handed direction.

Notes

Helix



Revolutions

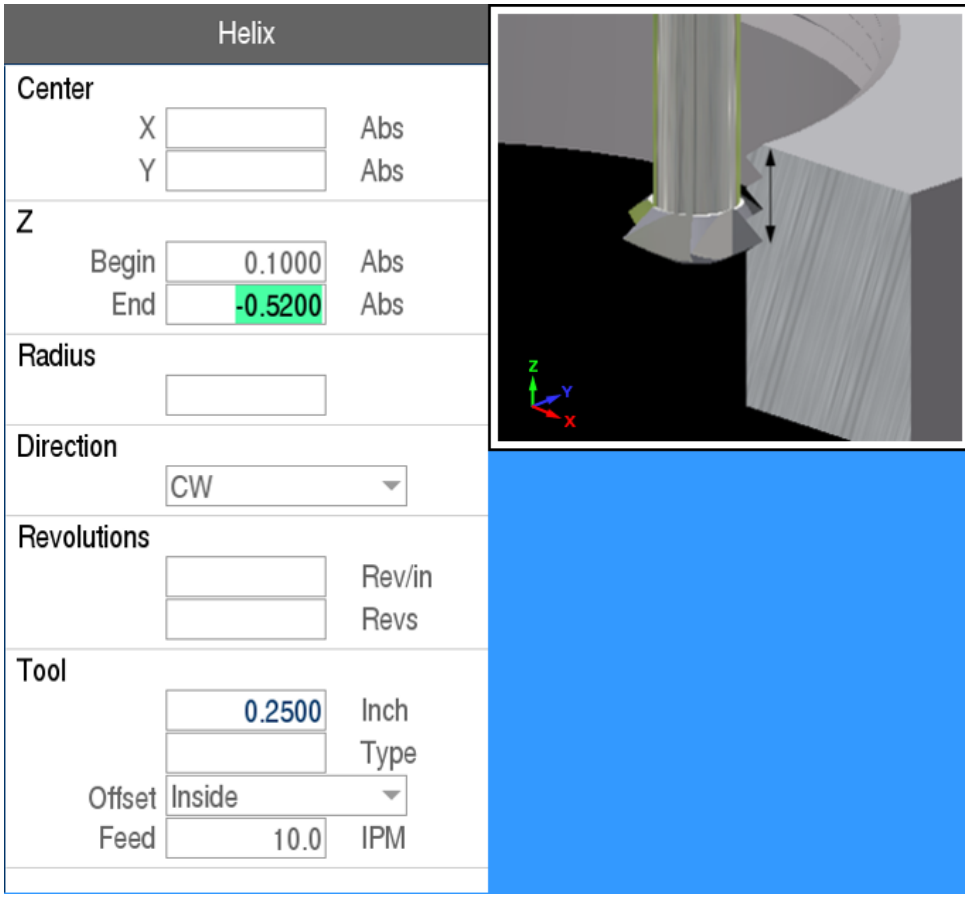
- **Rev / in** (revolution per inch) establishes a thread pitch
- **Revs** (total number of revolutions) if you are simply clearing a hole.

Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.
- **Offset:** Allows you to specify inside or outside.
- **Feed:** Establishes the rate at which the tool will feed along the X, Y plane.

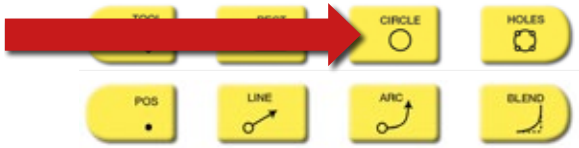
Start Angle

Establishes the starting angle along the circumference of the Helix. Zero degrees is at 3 o-clock. No angle defaults to zero degrees.



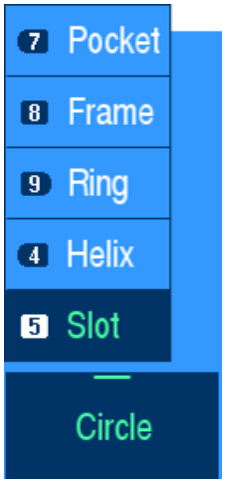
Notes

Radial Slot



The CIRCLE key activates a pop up list of circle features. Highlight the Radial Slot option and press ENTER or press 5 on the keypad to select Radial Slot.

Radial Slot allows you to define a complete stand alone Radial Slot. This step is capable of running in both DRO and PGM mode.



Center

X1 Y1 define the center point of the Radial Slot. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Z

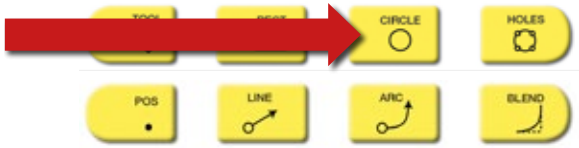
- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the End value at the **Feed** provided.
- This feed is for the Z axis only.
- **Pass** will divide the operation by the number of times or by the distance you program providing multiple passes to final depth.

Radial Slot

1st Arc Center			**Type**		
X1		Abs		Pocket	
Y1		Abs	**Sweep Angle**		
2nd Arc Center					
X2		Abs	**Tool**		
Y2		Abs		0.2500	Inch
Z					Type
Begin	0.1000	Abs	Feed	10.0	IPM
End	-0.0200	Abs	**Center**		
Pass		Number	X		Abs
Feed	10.0	IPM	Y		Abs
Radius			**Finish**		
			Bottom	0.0000	
Direction			Side	0.0000	
	CCW		Feed	0.0	IPM
Slot Width			Direction	CCW	
			Stepover		
				50	%

Notes

Radial Slot



Radius
Establishes the arc radius of the slot.

Direction
Establishes the sweep direction of the radial slot.

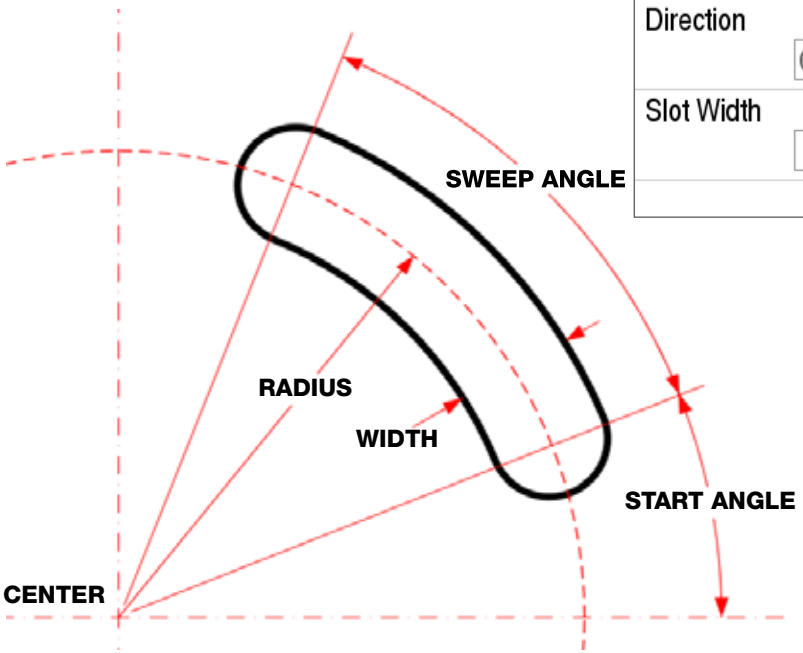
Slot Width
Establishes the width of the radial slot. It must be wider that the tool indicated.

Start Angle
Establishes the start of the radial slot.

Sweep Angle
Establishes the angular length relative to the start angle.

Radial Slot

1st Arc Center			**Type**	Pocket	
X1		Abs	**Sweep Angle**		
Y1		Abs			
2nd Arc Center			**Tool**		
X2		Abs		0.2500	Inch
Y2		Abs			Type
Z			Feed	10.0	IPM
Begin	0.1000	Abs	**Center**		
End	-0.0200	Abs	X		Abs
Pass		Number	Y		Abs
Feed	10.0	IPM			
Radius			**Finish**		
			Bottom	0.0000	
Direction			Side	0.0000	
	CCW		Feed	0.0	IPM
Slot Width			Direction	CCW	
			Stepover		
				50	%



PPT SLIDES 98-99

Notes

Radial Slot

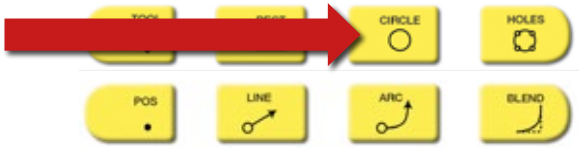
Tool
Shows the tool information of the last tool programmed to insure you the correct tool will be used.

Feed
Establishes the X, Y feed rate.

Finish
Establishes the amount of stock on the Bottom and Side to leave for the final finish pass along with the Feed rate and Direction to use.

Stepover
Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

Type
Allows you to choose contour if you wish to use the shape to define an Island within a custom pocket.



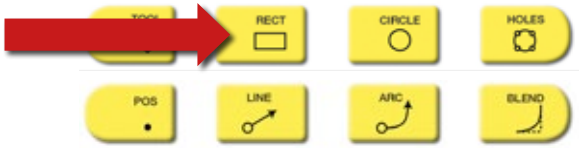
Radial Slot

1st Arc Center			**Type**		
X1		Abs		Pocket	
Y1		Abs	**Sweep Angle**		
2nd Arc Center					
X2		Abs	**Tool**		
Y2		Abs		0.2500	Inch
Z					Type
Begin	0.1000	Abs	Feed	10.0	IPM
End	-0.0200	Abs	**Center**		
Pass		Number	X		Abs
Feed	10.0	IPM	Y		Abs
Radius			**Finish**		
			Bottom	0.0000	
Direction			Side	0.0000	
	CCW		Feed	0.0	IPM
Slot Width			Direction	CCW	
			Stepover		
				50	%

PPT SLIDES 99-101

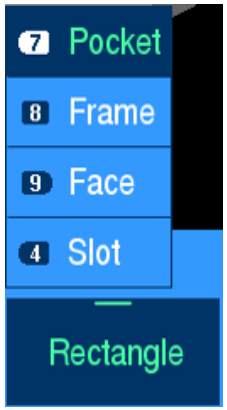
Notes

Rectangle Pocket



The Rectangle key activates a pop up list of rectangular features. Highlight the Pocket option and press ENTER or press 7 on the keypad to select Pocket.

Rectangle pocket allows you to define a complete stand alone rectangular pocket. This step is capable of running in both DRO and PGM mode.



1st Corner

X1 Y1 defines the anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not enter if a center point is to be entered.

Size

X Y defines the length and width of the rectangle. Values can be positive and negative. Do not enter if using a 2nd Corner.

2nd Corner

X2 Y2 defines a second anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or an incremental (INC) move from an external reference point. With a given angle the corners remain fixed.

Rectangle

1st Corner			**Tool**		
X1		Abs		0.2500	Inch
Y1		Abs			Type
Size			Feed	10.0	IPM
X			**Center**		
Y			X		Abs
2nd Corner			Y		Abs
X2		Abs	**Angle**		
Y2		Abs		0.0000	
Z			**Finish**		
Begin	0.1000	Abs	Bottom	0.0000	
End	-0.0200	Abs	Side	0.0000	
Pass	2	Number	Feed	0.0	IPM
Feed	10.0	IPM	Direction	CCW	
Corner			**Stepover**		
Blend		Radius		50	%
Direction			**Type**		
	CCW			Pocket	

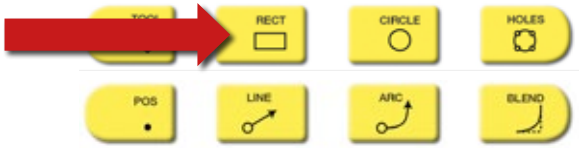
Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value stopping for each pass.
- This feed is for the Z axis only.
- **Pass** establishes the number of cuts between the Z Begin and End.
- **Feed** establishes the Z rate of motion for the Z axis only. The feed below is for the X, Y plane.

PPT SLIDES 102-104

Notes

Rectangle Pocket



Corner Blend Radius

Establishes the radius of each corner. Left blank and the tools radius will be left at each corner.

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction for the rough cut.

Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.
- **Feed** establishes the rate at which the tool will feed along the X, Y plane.

Center

X Y define the center point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner if you plan to use a center value.

Rectangle

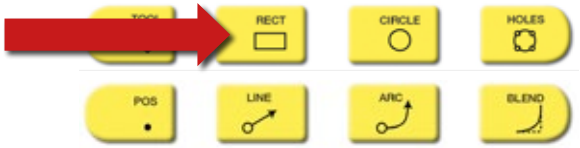
1st Corner			**Tool**		
X1		Abs		0.2500	Inch
Y1		Abs			Type
Size			Feed	10.0	IPM
X			**Center**		
Y			X		Abs
2nd Corner			Y		Abs
X2		Abs	**Angle**		
Y2		Abs		0.0000	
Z			**Finish**		
Begin	0.1000	Abs	Bottom	0.0000	
End	-0.0200	Abs	Side	0.0000	
Pass	2	Number	Feed	0.0	IPM
Feed	10.0	IPM	Direction	CCW	
Corner			**Stepover**		
Blend		Radius		50	%
Direction			**Type**		
	CCW			Pocket	

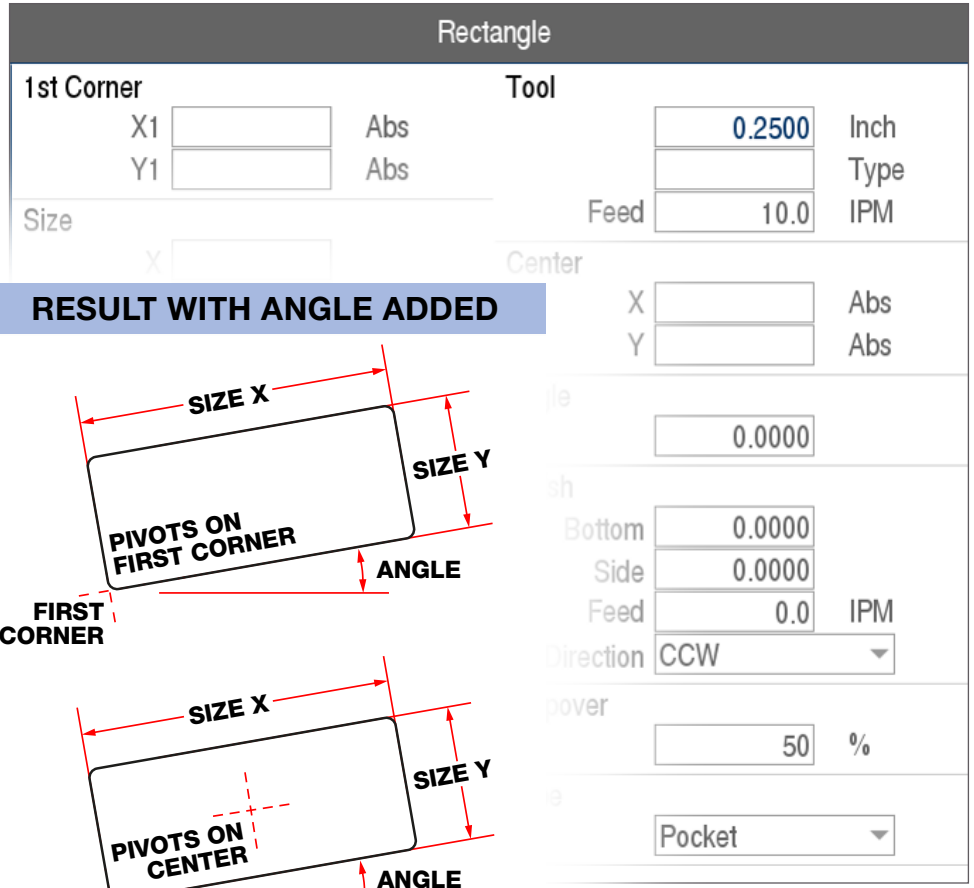
Notes

Rectangle Pocket

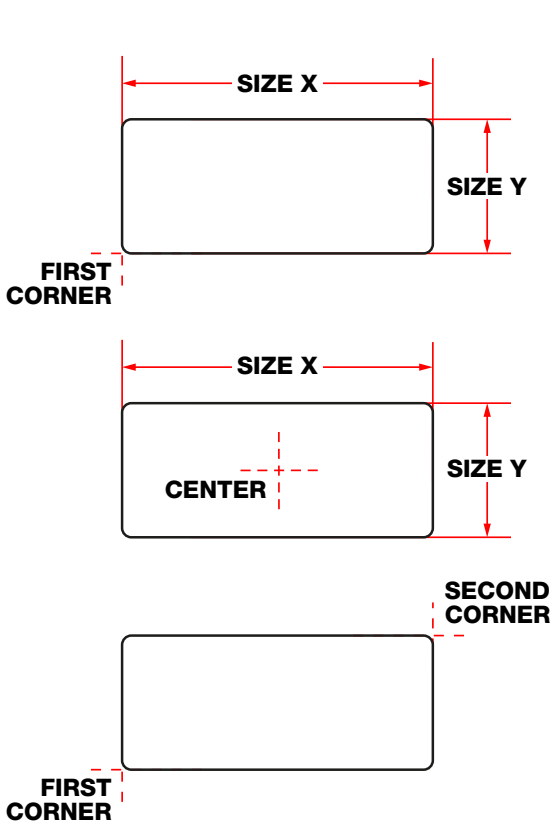
Angle

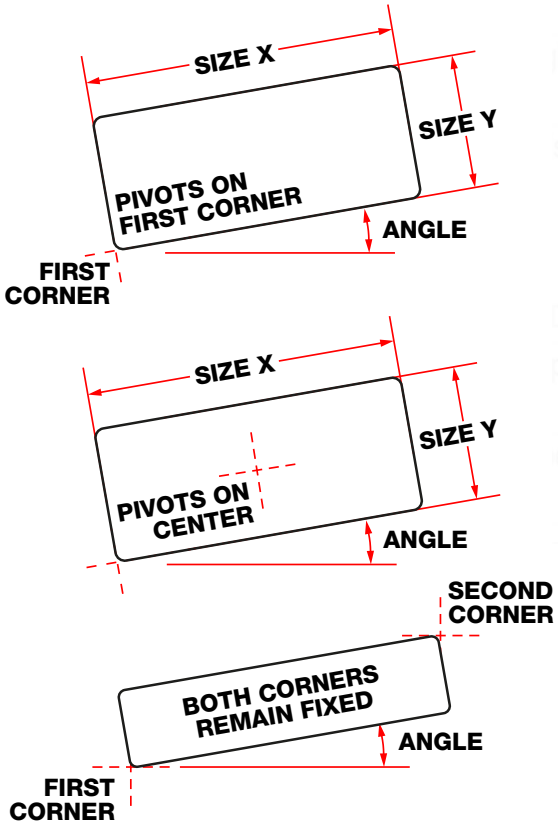
Pivots the rectangle as shown.





PROGRAMMED PARAMETERS	RESULT WITH ANGLE ADDED





PPT SLIDE 106

Notes

Rectangle Pocket

Finish

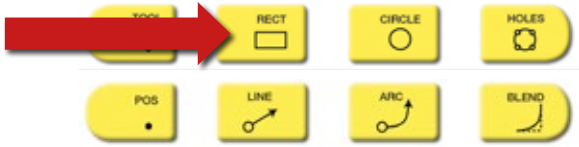
- Establishes the amount of stock (cut) to leave for the final finish pass
- **Feed** rate and Direction to use

Stepover

Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

Type

Allows you to choose contour if you wish to use the shape to define an Island within a custom pocket.



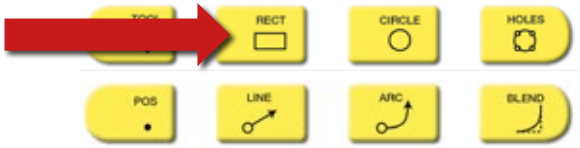
Rectangle

1st Corner			**Tool**		
X1		Abs		0.2500	Inch
Y1		Abs			Type
Size			Feed	10.0	IPM
X			**Center**		
Y			X		Abs
2nd Corner			Y		Abs
X2		Abs	**Angle**		
Y2		Abs		0.0000	
Z			**Finish**		
Begin	0.1000	Abs	Bottom	0.0000	
End	-0.0200	Abs	Side	0.0000	
Pass	2	Number	Feed	0.0	IPM
Feed	10.0	IPM	Direction	CCW	
Corner			**Stepover**		
Blend		Radius		50	%
Direction			**Type**		
	CCW			Pocket	

PPT SLIDE 107

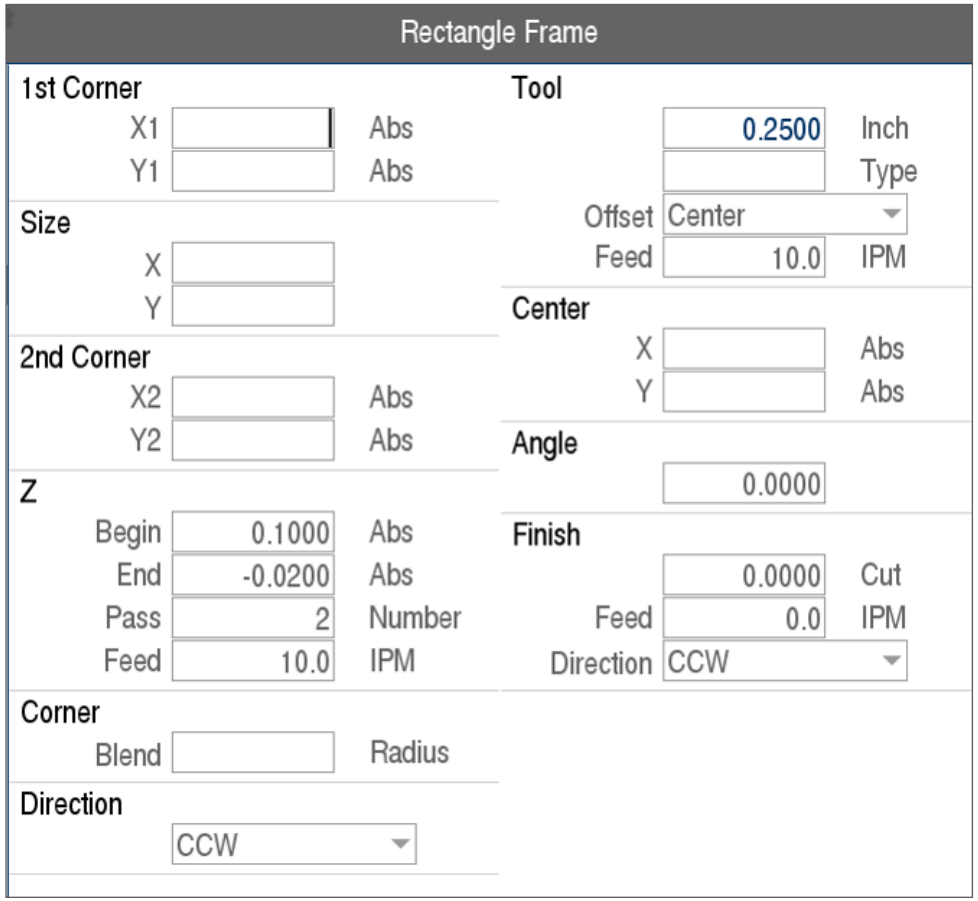
Notes

Rectangle Frame



The Rectangle key activates a pop up list of rectangular features. Highlight the Frame option and press ENTER or press 8 on the keypad to select Frame.

Rectangle Frame allows you to define a complete stand alone rectangular frame. This step is capable of running in both DRO and PGM mode.



1st Corner

X1 Y1 defines the anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not fill in if a center point is to be entered.

Size

X Y defines the length and width of the rectangle. Values can be positive and negative as needed. Do not enter values if using a second corner.

2nd Corner

X2 Y2 can be used to define a second anchor point for the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. With a given angle the corners remain fixed.

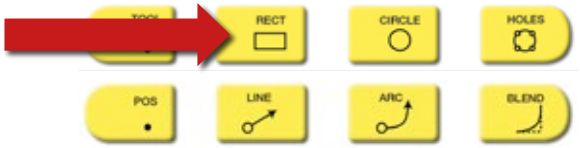
Z

- **Begin** is where the tool will rapid to. From there it will **Feed** at the given rate to the End value stopping for each pass.
- **Pass** establishes the number of cuts between the Z Begin and End.
- **Feed** establishes the Z rate of motion for the Z axis only. The feed below is for the X, Y plane.

PPT SLIDES 108-110

Notes

Rectangle Frame



Corner Blend Radius

Establishes the radius of each corner. Left blank and the tools radius will be left at each corner. Chamfer can also be selected here.

Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction for the rough cut.

Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.
- **Feed** establishes the rate at which the tool will feed along the X, Y plane.

Center

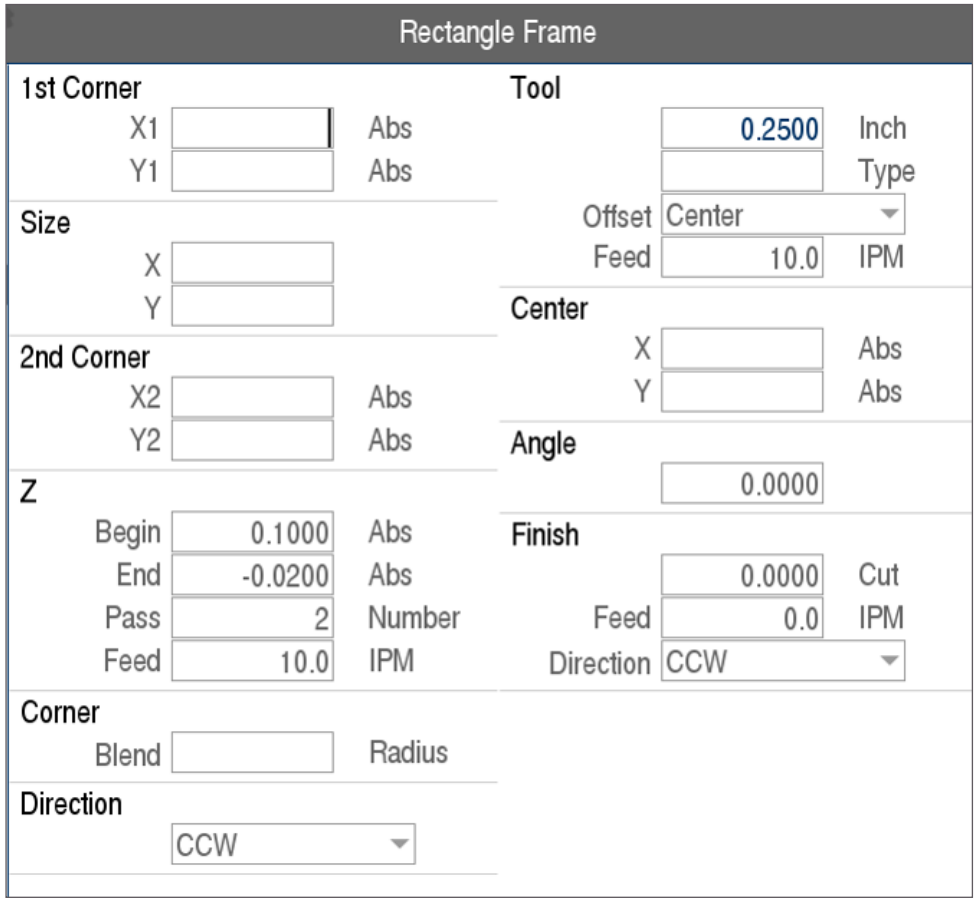
X Y define the center point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner if you plan to use a center value.

Angle

Pivots the rectangle as shown earlier.

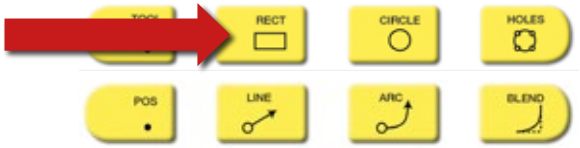
Finish

- Establishes the amount of stock (cut) to leave for the final finish pass
- **Feed** rate and **Direction** to use

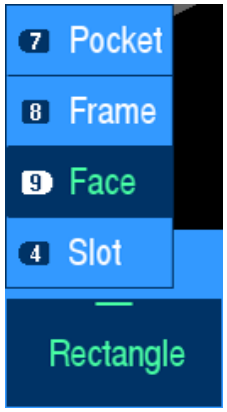


Notes

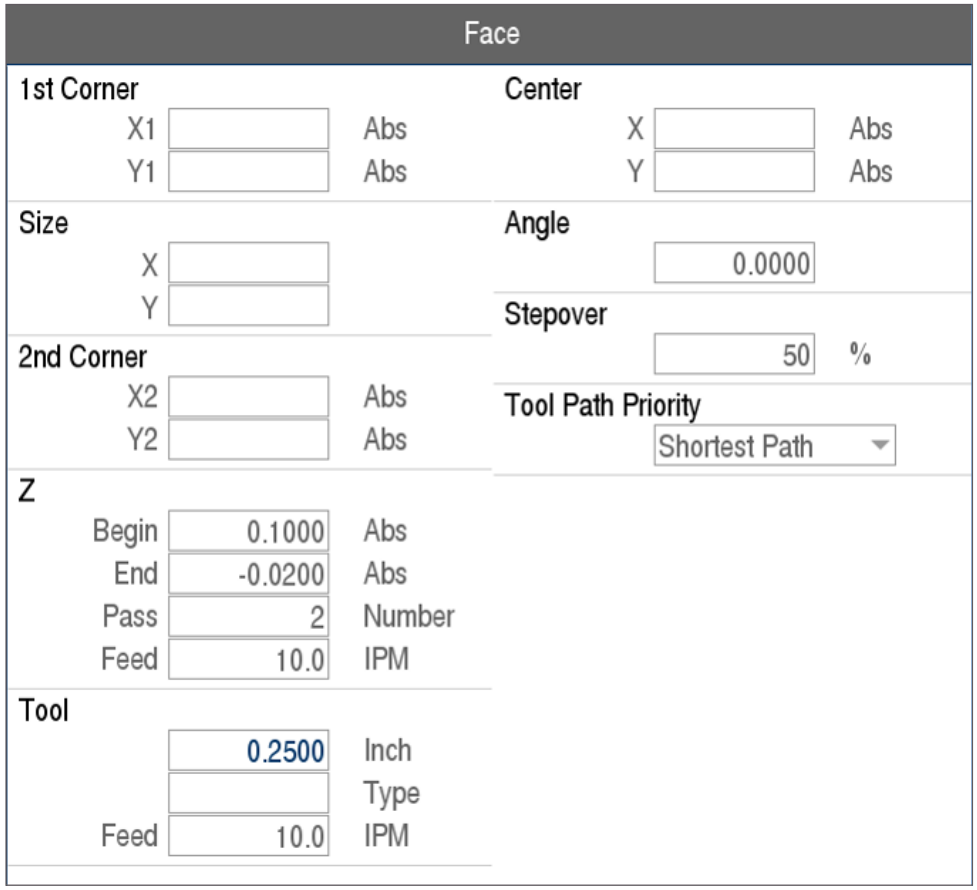
Rectangle Face



The Rectangle key activates a pop up list of rectangular features. Highlight the Face option and press ENTER or press 9 on the keypad to select Face.



Rectangle Face allows you to define a complete stand alone rectangular facing operation. This step is capable of running in both DRO and PGM mode..



1st Corner

X1 Y1 defines the start point of the facing operation. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Size

X Y defines the length and width of the rectangle. Values can be positive and negative as needed. Do not use if using a second corner.

2nd Corner

X2 Y2 can be used to define a second anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. With a given angle the corners remain fixed.

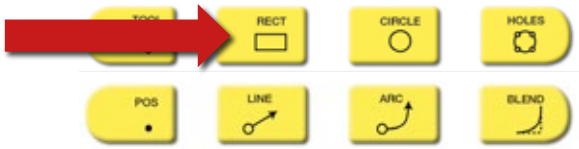
Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value stopping for each pass.
- **Pass** establishes the number of cuts between the Z Begin and End.
- **Feed** establishes the Z rate of motion for the Z axis only. The feed below is for the X, Y plane.

PPT SLIDES 112-114

Notes

Rectangle Face



Tool

- Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.
- **Feed** establishes the rate at which the tool will feed along the X, Y plane.

Center

X Y define the center point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner if you plan to use a center value.

Angle

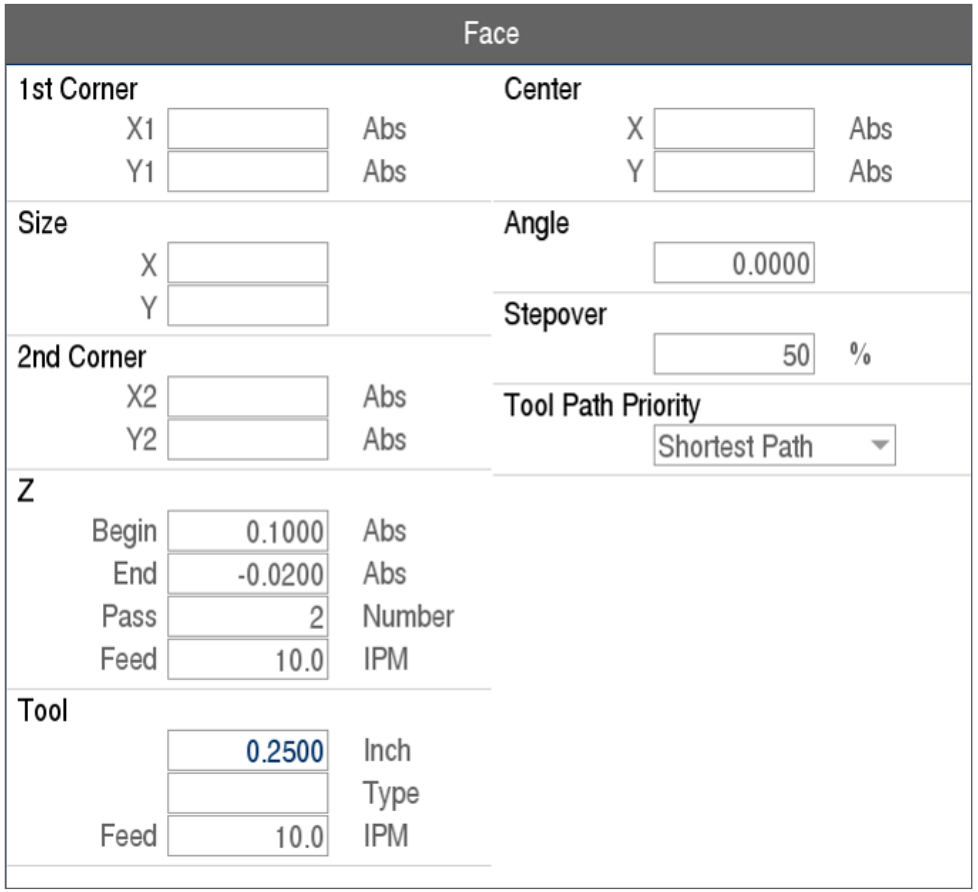
Pivots the rectangle as described earlier.

Stepover

Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

Tool Path Priority

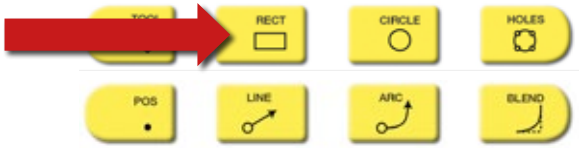
Allows you select either the shortest path or select even spacing over across the face.



PPT SLIDES 114-115

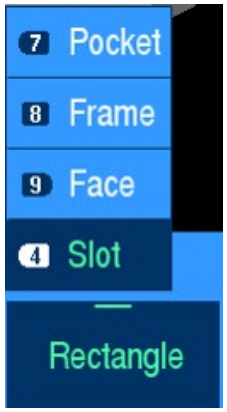
Notes

Rectangle Slot



The Rectangle key activates a pop up list of rectangular features. Highlight the Slot option and press ENTER or press 4 on the keypad to select Slot.

Rectangle Slot allows you to define a complete stand alone rectangular slot. This step is capable of running in both DRO and PGM mode.



Slot					
1st Arc Center			**Tool**		
X1		Abs		0.2500	Inch
Y1		Abs			Type
2nd Arc Center			Feed	10.0	IPM
X2		Abs	**Center**		
Y2		Abs	X		Abs
Z			Y		Abs
Begin	0.1000	Abs	**Angle**		
End	-0.0200	Abs		0.0000	
Pass		Number	**Finish**		
Feed	10.0	IPM	Bottom	0.0000	
Direction			Side	0.0000	
	CCW		Feed	0.0	IPM
Slot Width			Direction	CCW	
			Stepover		
Slot Length				50	%
			Type		
				Pocket	

1st Arc Center

X1 Y1 defines the anchor point of the slot. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not fill in if a center point is to be entered.

2nd Arc Center

X2 Y2 can be used to define a second anchor point for the slot. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. When both centers are entered the angle is calculated.

Z

- **Begin** is where the tool will rapid to before feeding at the given rate to the **End** value stopping for each pass.
- **Pass** establishes the number of cuts between the Z Begin and End.
- **Feed** establishes the Z rate of motion for the Z axis only. The feed below is for the X, Y plane.

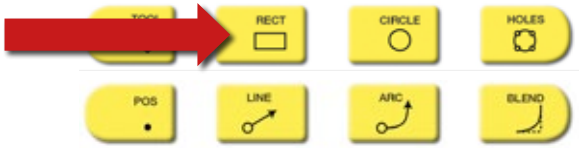
Direction

Establishes clockwise (CW) or counter clockwise (CCW) direction for the rough cut.

PPT SLIDES 116-118

Notes

Rectangle Slot



Slot Width
Establishes how wide the slot will be.

Slot Length
Establishes how long the slot will be.

Tool
- Displays the tool information of the last tool step to insure you the correct tool will be used.
- **Feed:** Establishes the feed rate along the X, Y plane.

Center
X Y define the center point of the slot. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner if you plan to use a center value.

Angle
Pivots the slot as shown earlier.

Finish
Establishes the amount of stock (cut) to leave for the final finish pass along with the feed rate and direction to use.

Stepover
Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

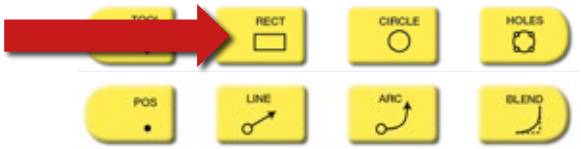
Type
Allows you to choose contour if you wish to use the shape to define an Island within a custom pocket.

Slot

1st Arc Center			**Tool**		
X1		Abs		0.2500	Inch
Y1		Abs			Type
2nd Arc Center			Feed	10.0	IPM
X2		Abs	**Center**		
Y2		Abs	X		Abs
Z			Y		Abs
Begin	0.1000	Abs	**Angle**		
End	-0.0200	Abs		0.0000	
Pass		Number	**Finish**		
Feed	10.0	IPM	Bottom	0.0000	
Direction			Side	0.0000	
	CCW		Feed	0.0	IPM
Slot Width			Direction	CCW	
			Stepover		
Slot Length				50	%
			Type		
				Pocket	

Notes

Rectangle Slot



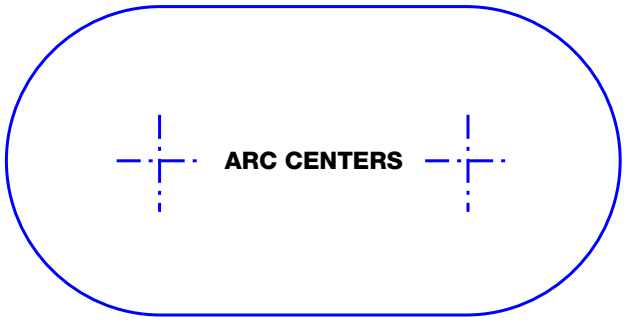
Slot Width

- Center, width, Length, Angle.
- 1st Arc Center, 2nd Arc Center, Width.
- 1st Arc Center, Width, Length, Angle.

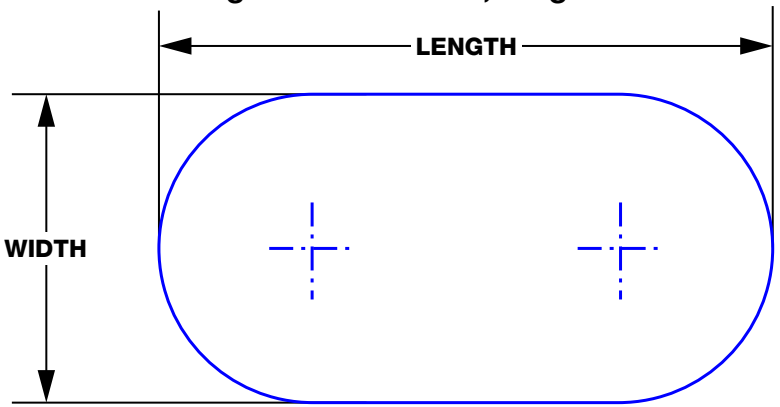
Requirements

1. Length and width must be positive values.
2. Width must be less than length.

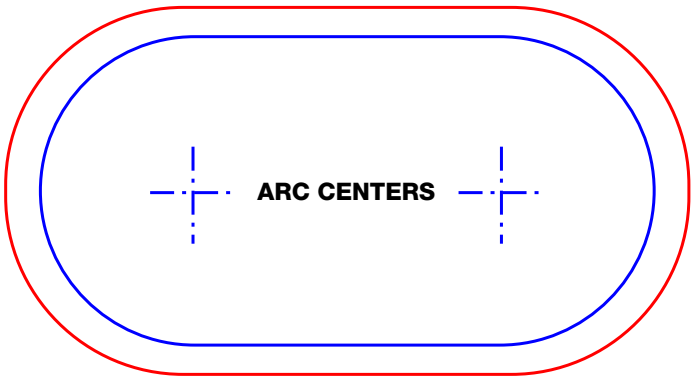
Programmed from arc centers



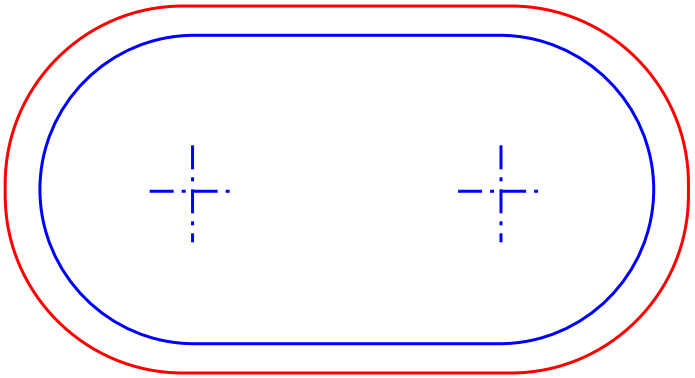
Programmed center, length and width



Adjustment to size does not change the arc center locations



Adjustment to size does not change the arc center locations

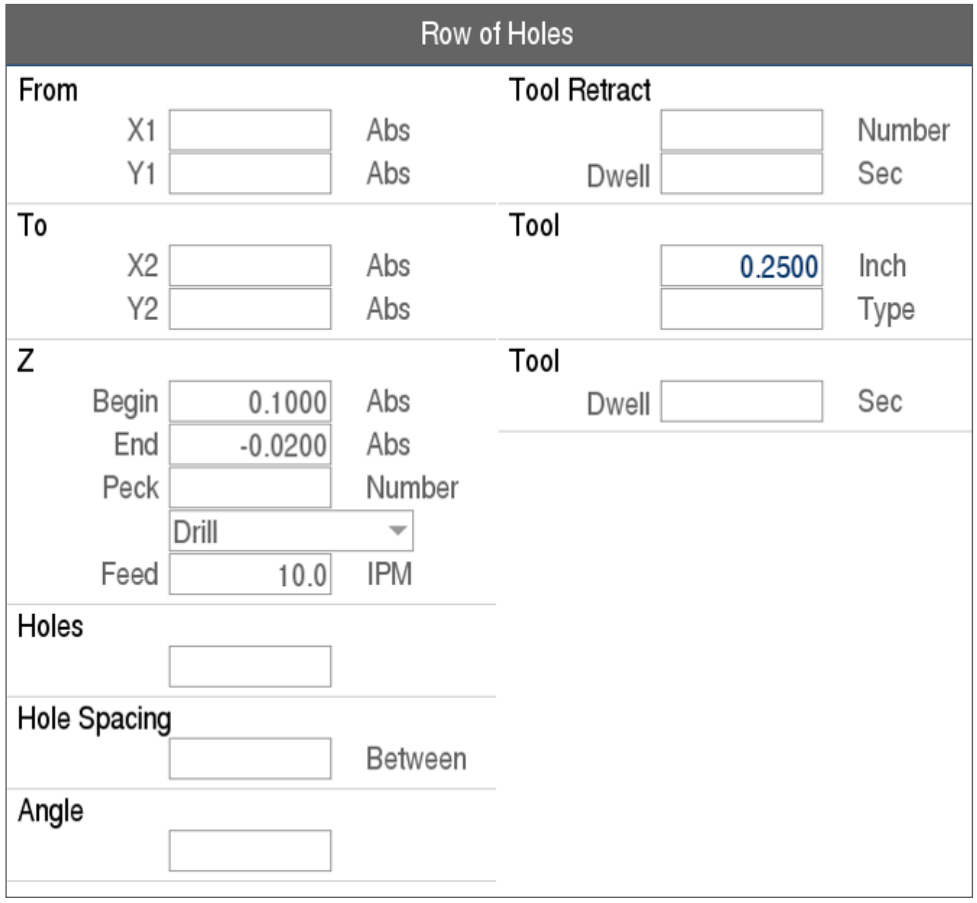


PPT SLIDE 120

Notes

Row of Holes

The HOLES key activates a pop up list of Hole features. Highlight the Row of Holes option and press ENTER or press 7 on the keypad to select Row of Holes. Row of Holes allows you to define a complete stand alone Row of Holes. This step is capable of running in both DRO and PGM mode.



From

X1 Y1 define the start point of the row. These can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

To

X2 Y2 is the end of the row and can be cartesian or polar, absolute (ABS) or incremental (INC) from the begin point or an external reference point. Do not enter if using Hole Spacing.

Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value at the Feed provided.
- This feed is for the Z axis only.
- When finished, the tool then rapids up to the retract plane set in datum.
- **Peck** lifts the tool slightly breaking the chip then resuming.
- Choosing **Drill** feeds down and rapids up, Bore feeds in both directions, and **Position** causes the routine to work as a two axis operation pausing at each location until GO is pressed.

Holes

Establishes the number of holes in the row.

PPT SLIDES 121-123

Notes

Row of Holes

Hole Spacing

Can be used alternatively to the **To** point. When used the To is calculated and is best used along with angle.

Angle

Allows you to create a row by programming along an angle using a From point, number of holes, spacing, and angle.

Tool Retract

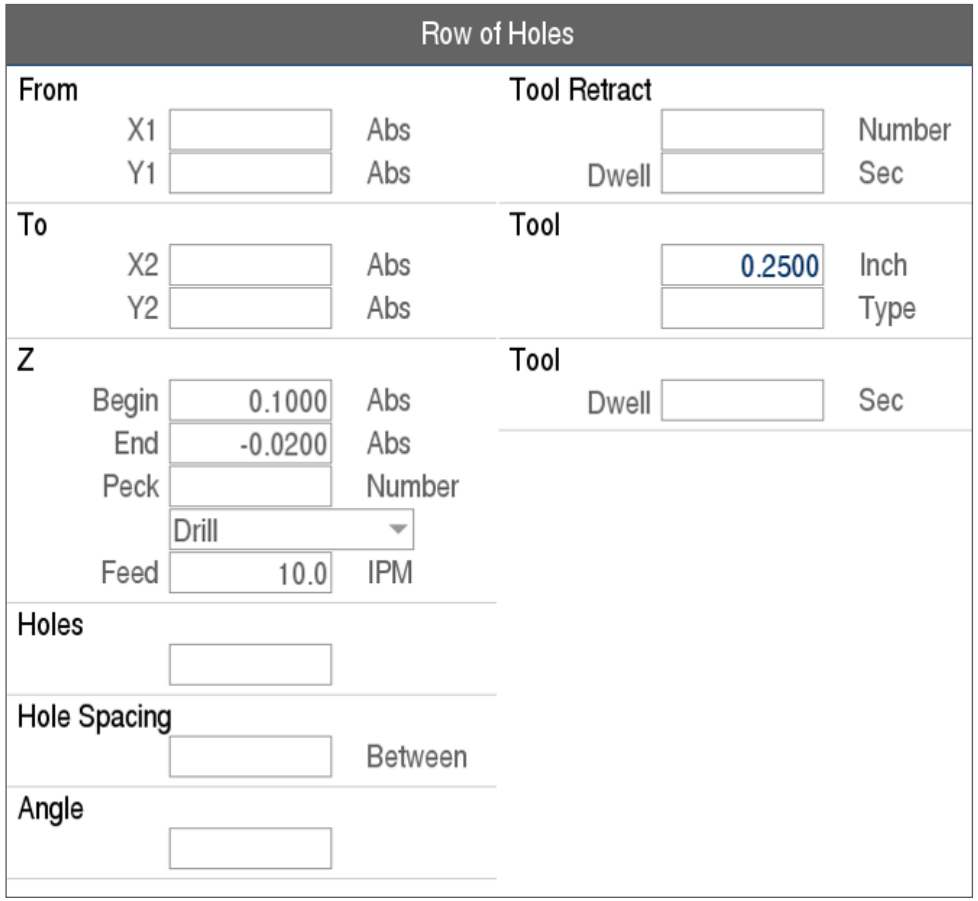
Establishes the number of times the tool will lift to the Z begin. When here it will dwell for the time entered in seconds before resuming allowing the tool and work to cool. The retract can be programmed along with peck to create the optimum drilling cycle.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used. Displays information of the last tool step programmed to insure you the correct tool will be used.

Dwell

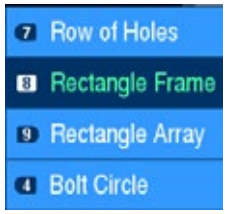
Holds the tool at the Z End for the number of seconds (Sec) provided to finish the bottom of the holes.

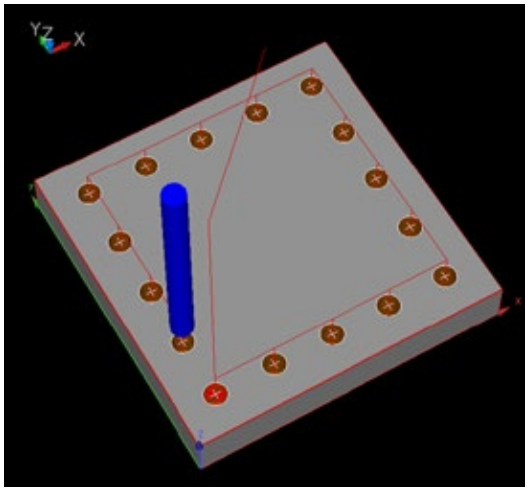


Notes

Rectangle Frame of Holes

The HOLES key activates a pop up list of Hole features. Highlight the Rectangle Frame option and press ENTER or press 8 on the keypad to select Rectangle Frame. Rectangle Frame allows you to define a complete stand alone Rectangle Frame of holes. This step is capable of running in both DRO and PGM mode.





Hole Frame					
1st Corner			Hole Spacing		
	X1	Abs		X	Between
	Y1	Abs		Y	Between
Size			Center		
	X			X	Abs
	Y			Y	Abs
2nd Corner			Angle		
	X2	Abs		0.0000	
	Y2	Abs	Tool Retract		
Z					Number
	Begin	Abs		Dwell	Sec
	End 0.0000	Abs	Tool		
	Peck	Number		0.2500	Inch
	Drill				Type
	Feed 10.0	IPM	Tool		
Holes				Dwell	Sec
	X				
	Y				

1st Corner

X1 Y1 defines the anchor point of the rectangular hole pattern. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not fill in if a center point is to be entered.

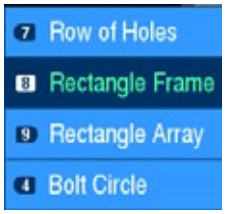
Size

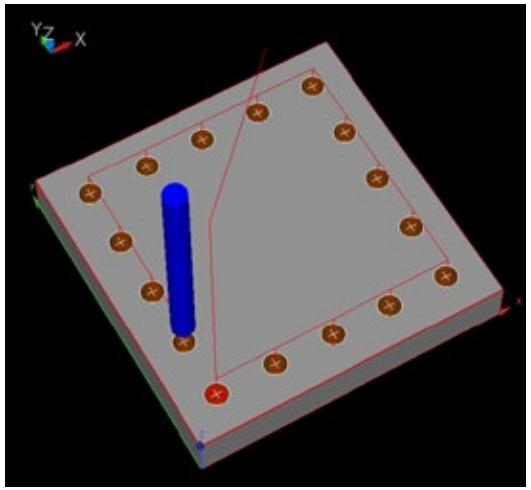
X Y defines the length and width of the rectangle and can be positive and negative as needed. Do not use if using a second corner or Hole Spacing is to be entered.

Notes

Rectangle Frame of Holes

The HOLES key activates a pop up list of Hole features. Highlight the Rectangle Frame option and press ENTER or press 8 on the keypad to select Rectangle Frame. Rectangle Frame allows you to define a complete stand alone Rectangle Frame of holes. This step is capable of running in both DRO and PGM mode.





Hole Frame					
1st Corner			**Hole Spacing**		
X1		Abs	X		Between
Y1		Abs	Y		Between
Size			**Center**		
X			X		Abs
Y			Y		Abs
2nd Corner			**Angle**		
X2		Abs		0.0000	
Y2		Abs	**Tool Retract**		
Z					Number
Begin		Abs	Dwell		Sec
End	0.0000	Abs	**Tool**		
Peck		Number		0.2500	Inch
	Drill				Type
Feed	10.0	IPM	**Tool**		
Holes			Dwell		Sec
X					
Y					

1st Corner

X1 Y1 defines the anchor point of the rectangular hole pattern. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not fill in if a center point is to be entered.

Size

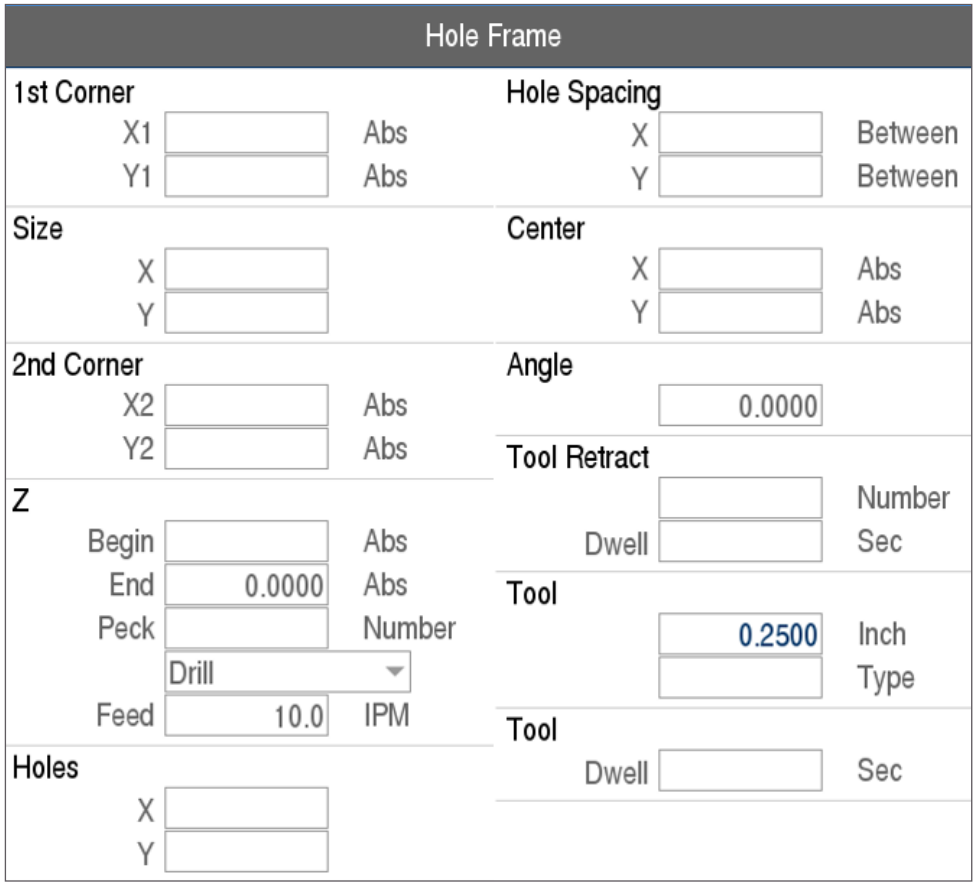
X Y defines the length and width of the rectangle and can be positive and negative as needed. Do not use if using a second corner or Hole Spacing is to be entered.

Notes

Rectangle Frame of Holes

2nd Corner

X2 Y2 can be used to define a second anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. When used with first corner, hole spacing will be calculated. With a given angle the corners remain fixed.



Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value at the Feed provided.
- This feed is for the Z axis only.
- When finished, the tool then rapids up to the retract plane set in datum.
- **Peck:** Lifts the tool slightly breaking the chip then resuming.
- There are three drilling options. Choosing **Drill** feeds down and rapids up, **Bore** feeds in both directions, and **Position** causes the routine to work as a two axis operation pausing at each location until GO is pressed.

Hole Spacing

Can be used alternatively to the To point. When entered, the To point is calculated. This option makes setting a frame on an angle simple.

Center

X Y define the center point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner when using a center location.

Notes

Rectangle Frame of Holes

Angle

Allows you to define the frame by programming along an angle using a From Point or Center, Number of Holes, Hole Spacing, and Angle.

Tool Retract

Establishes the number of times the tool will lift to the Z begin. When there it can dwell for the time entered in seconds (Sec) before re-suming, allowing the tool and work to cool. The retract can be programmed along with peck to create the optimum drilling cycle.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.

Dwell

Holds the tool at the Z End for the number of seconds (Sec) provided to finish the bot-tom of each hole.

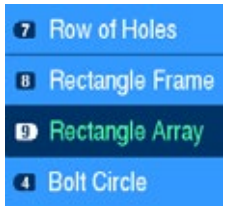
Hole Frame

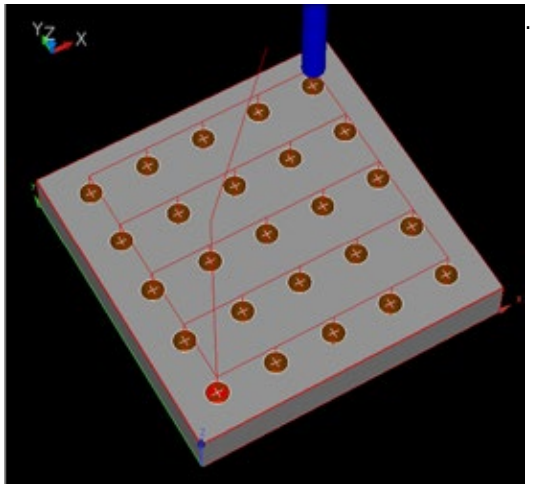
1st Corner			**Hole Spacing**		
X1		Abs	X		Between
Y1		Abs	Y		Between
Size			**Center**		
X			X		Abs
Y			Y		Abs
2nd Corner			**Angle**		
X2		Abs		0.0000	
Y2		Abs	**Tool Retract**		
Z					Number
Begin		Abs	Dwell		Sec
End	0.0000	Abs	**Tool**		
Peck		Number		0.2500	Inch
	Drill				Type
Feed	10.0	IPM	**Tool**		
Holes			Dwell		Sec
X					
Y					

Notes

Rectangle Array of Holes

The HOLES key activates a pop up list of Hole features. Highlight the Rectangle Array option and press ENTER or press 9 on the keypad to select Rectangle Array. Rectangle Array allows you to define a complete stand alone Rectangle Array of holes. This step is capable of running in both DRO and PGM mode.





Hole Array					
1st Corner			**Hole Spacing**		
	X1	Abs		X	Between
	Y1	Abs		Y	Between
Size			**Center**		
	X			X	Abs
	Y			Y	Abs
2nd Corner			**Angle**		
	X2	Abs		0.0000	
	Y2	Abs	**Tool Retract**		
Z					Number
	Begin	Abs		Dwell	Sec
	End 0.0000	Abs	**Tool**		
	Peck	Number		0.2500	Inch
	Drill				Type
	Feed 10.0	IPM	**Tool**		
Holes				Dwell	Sec
	X				
	Y				

1st Corner

X1 Y1 defines the anchor point of the rectangular hole Array. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. An angle entered under More below will pivot upon this corner. Do not fill in if a center point is to be entered.

Size

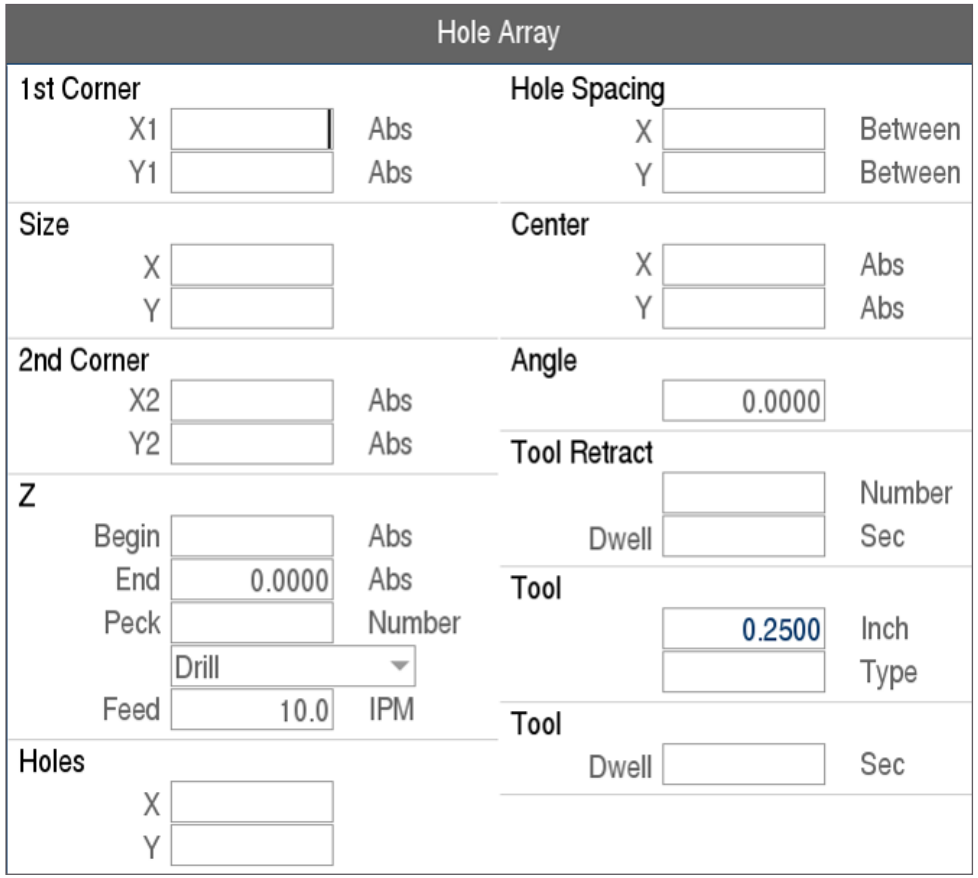
X Y defines the length and width of the rectangle and can be positive and negative as needed. Do not use if using a second corner or Hole Spacing is to be entered.

Notes

Rectangle Array of Holes

2nd Corner

X2 Y2 can be used to define a second anchor point of the rectangle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. When used with first corner, hole spacing will be calculated. With a given angle the corners remain fixed.



Z

- **Begin** is where the tool will rapid to. From there it will feed at the given rate to the **End** value at the Feed provided.
- This feed is for the Z axis only. When finished, the tool then rapids up to the retract plane set in datum.
- **Peck:** Lifts the tool slightly breaking the chip then resuming.
- There are three drilling options. Choosing **Drill** feeds down and rapids up, **Bore** feeds in both directions, and **Position** causes the routine to work as a two axis operation pausing at each location until GO is pressed.

Holes

X Y establishes the number of holes in the X and Y Array.

Hole Spacing

Can be used alternatively to the To point. When entered, the To point is calculated. This option makes setting an Array on an angle simple.

Center

X Y define the center point of the array. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point. Do not program a first corner or second corner if you plan to use a center value.

Notes

Rectangle Array of Holes

Angle

Allows you to define the array by programming along an angle using a From Point or Center, Number of Holes, Hole Spacing, and Angle.

Tool Retract

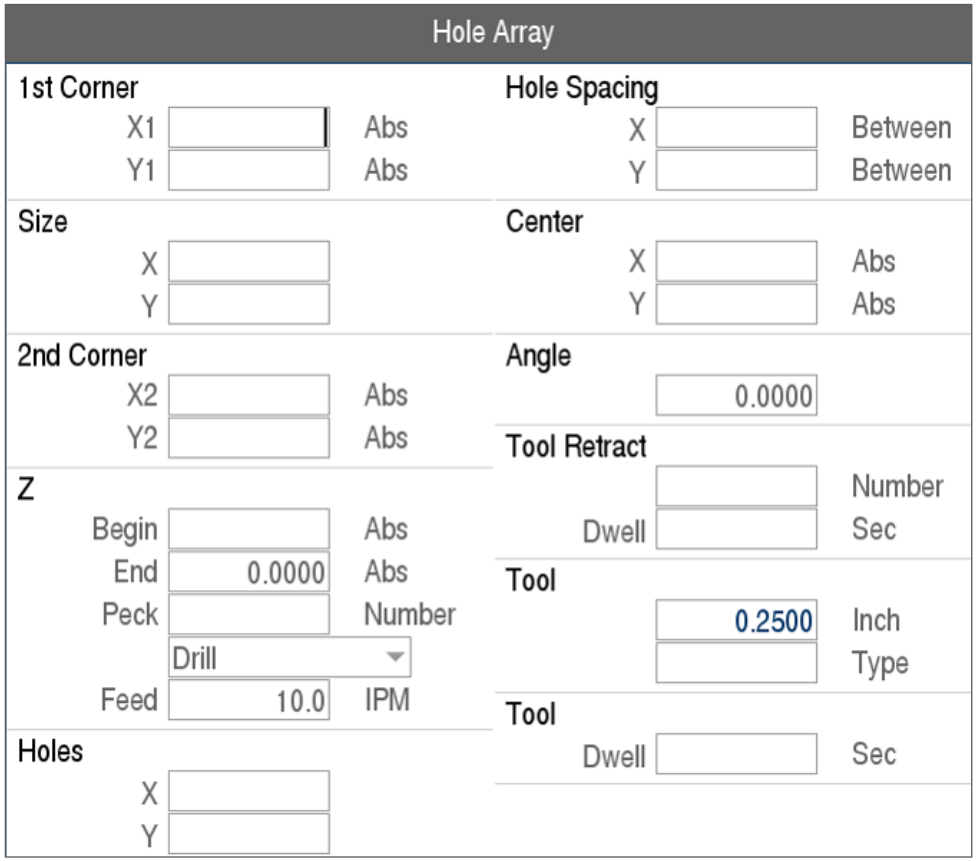
Establishes the number of times the tool will lift to the Z begin. When there it can dwell for the time entered in seconds (Sec) before re-suming, allowing the tool and work to cool. The retract can be programmed along with peck to create the optimum drilling cycle.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.

Dwell

Holds the tool at the Z End for the number of seconds (Sec) provided to finish the bot-tom of each hole.

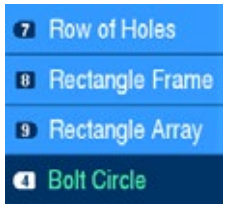


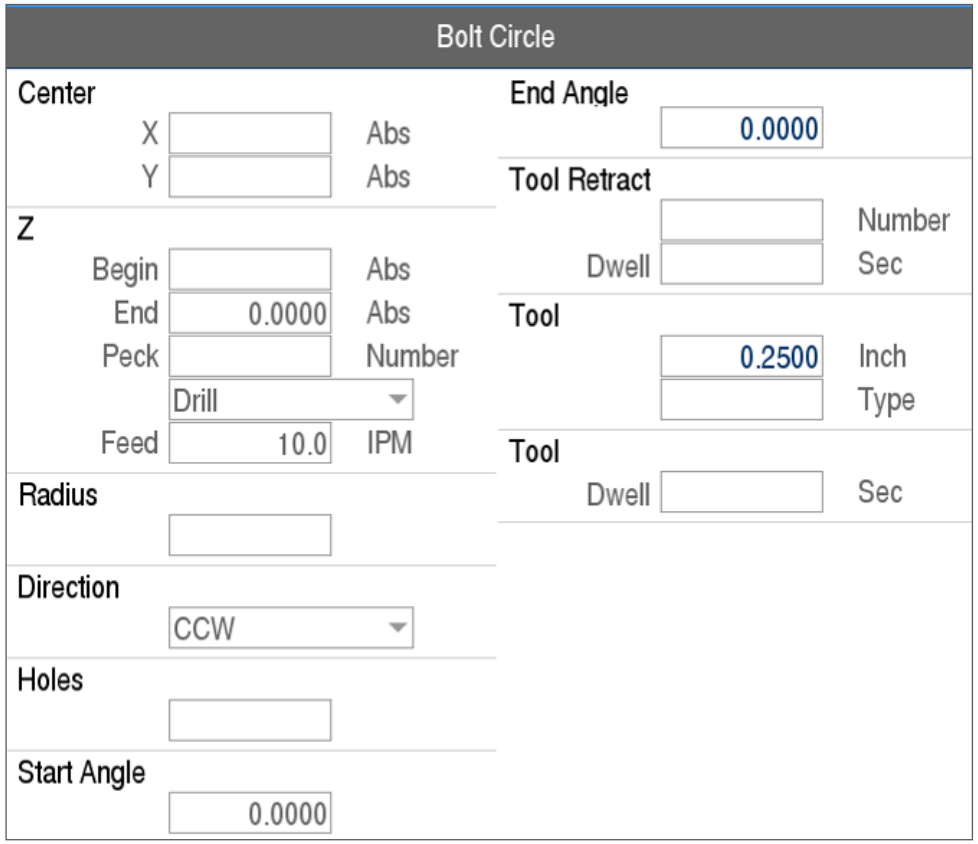
Notes

Bolt Circle

The HOLES key activates a pop up list of Hole features. Highlight the Bolt Circle option and press ENTER or press 4 on the keypad to select Bolt Circle.

Bolt Circle allows you to define a complete stand alone Bolt Circle. This step is capable of running in both DRO and PGM mode.





Center

X Y define the center point of the Bolt Circle. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Z

- **Begin** is where the tool will rapid to before it slows to the Z **feed** provided. When finished, the tool then rapids up to the retract plane set in datum.
- **Peck:** Lifts the tool slightly breaking the chip then resuming.
- There are three drilling options. Choosing **Drill** feeds down and rapids up, **Bore** feeds in both directions, and **Position** creates a two axis operation pausing at each location until GO is pressed.

Radius

Establishes the size of the Bolt Hole Pattern.

Holes

Establishes the number of holes in the pattern.

Start Angle

Used to place the first hole along the radius.

Notes

Bolt Circle

End Angle

Places the last hole along the radius. Leave blank for a full circle pattern.

Tool Retract

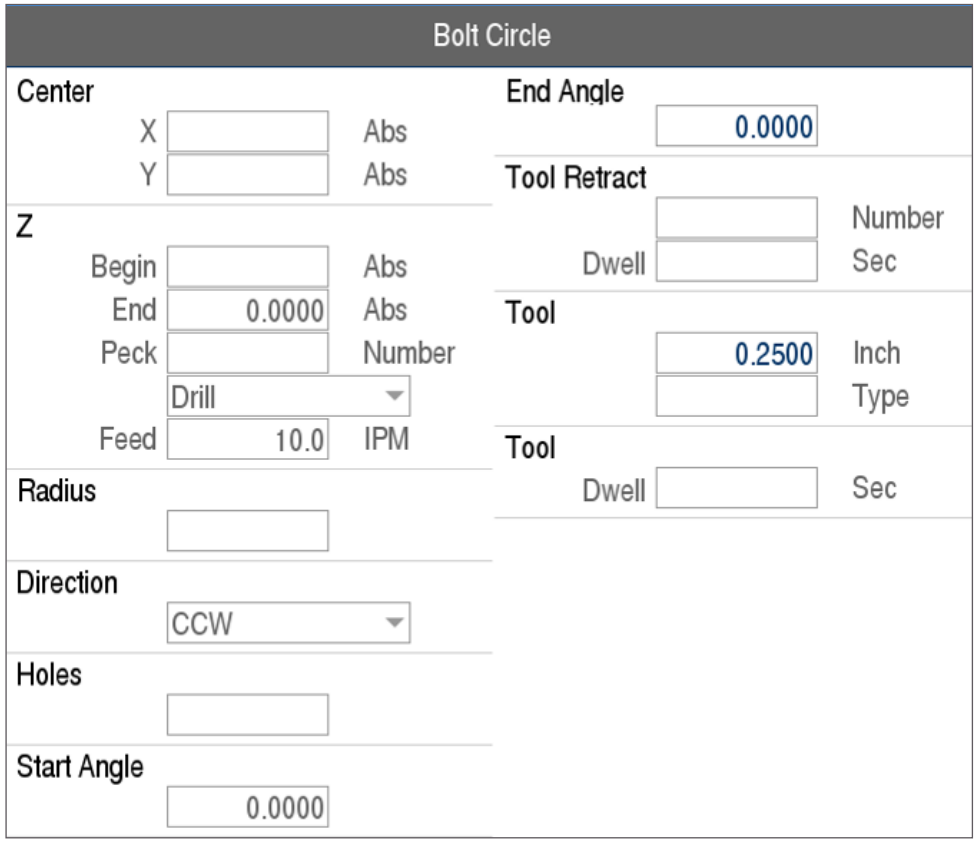
Establishes the number of times the tool will lift to the Z begin. When there it can dwell for the time entered in seconds (Sec) before re-suming, allowing the tool and work to cool. The retract can be programmed along with peck to create the optimum drilling cycle.

Tool

Cannot be changed here but shows the tool information of the last tool step programmed to insure you the correct tool will be used.

Dwell

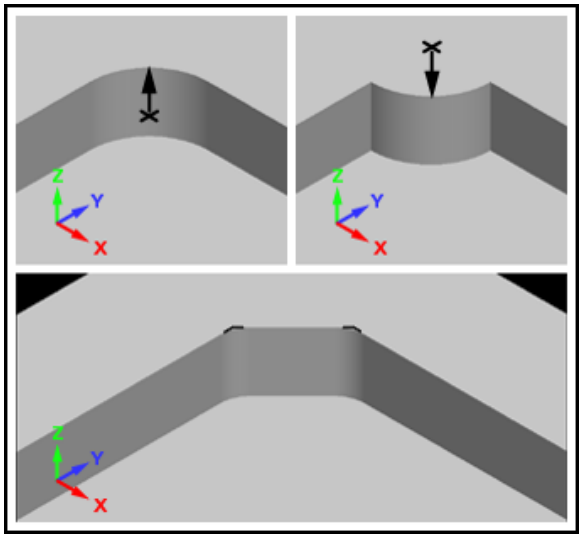
Holds the tool at the Z End for the number of seconds (Sec) provided to finish the bot-tom of each hole.



Notes

Blend, Inverted Blend & Chamfer

The BLEND key opens the Blend/Chamfer window allowing you to add a blend radius, inverted radius, or chamfer between two consecutive program steps in a continuous tool path. Chamfer to easily add a radius tangent to two lines and or arcs. The inverted chamfer is useful for leaving stock for a corner hole or to add interest to a profile. Chamfer creates a straight corner transition defined by length and width.



Type

Allows you to select Blend, Blend (Inv), or Chamfer

Steps

From and To indicate the steps to be blended. When inserting a blend between two existing steps, highlight the second of the two program steps. The blend will be inserted between the two. The steps must be continuous.

Radius

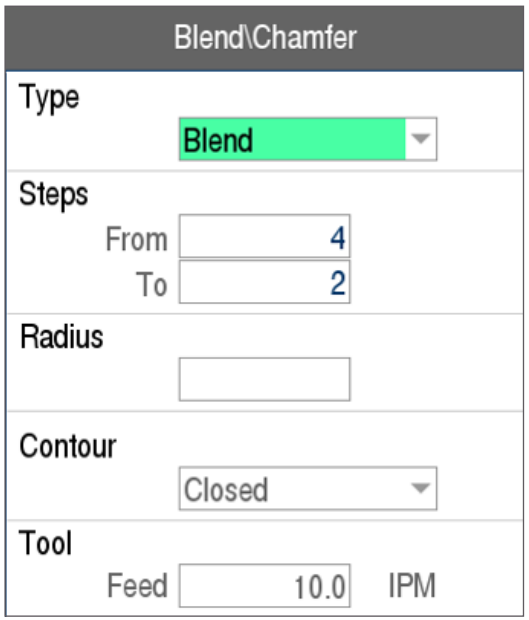
Defines the size of the blend. It must be large enough to fit between the features if a gap exists between the two. A blend must be less than 180 degrees.

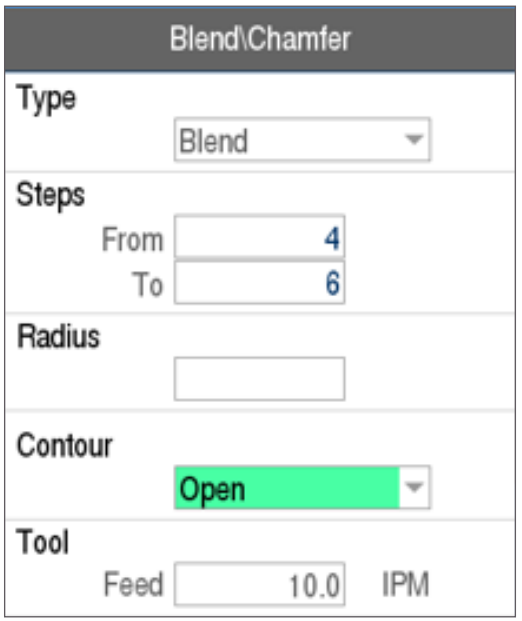
Contour

Use Open if blending with the next step in the path or choose Closed to blend with the first step in continuous path.

Size

Length 1 and Length 2 define a chamfer. Length 1 is the distance back along the from step starting from the intersecting point of the from and to step and Length 2 is the distance forward along the to step from the intersection.





PPT SLIDES 138-140

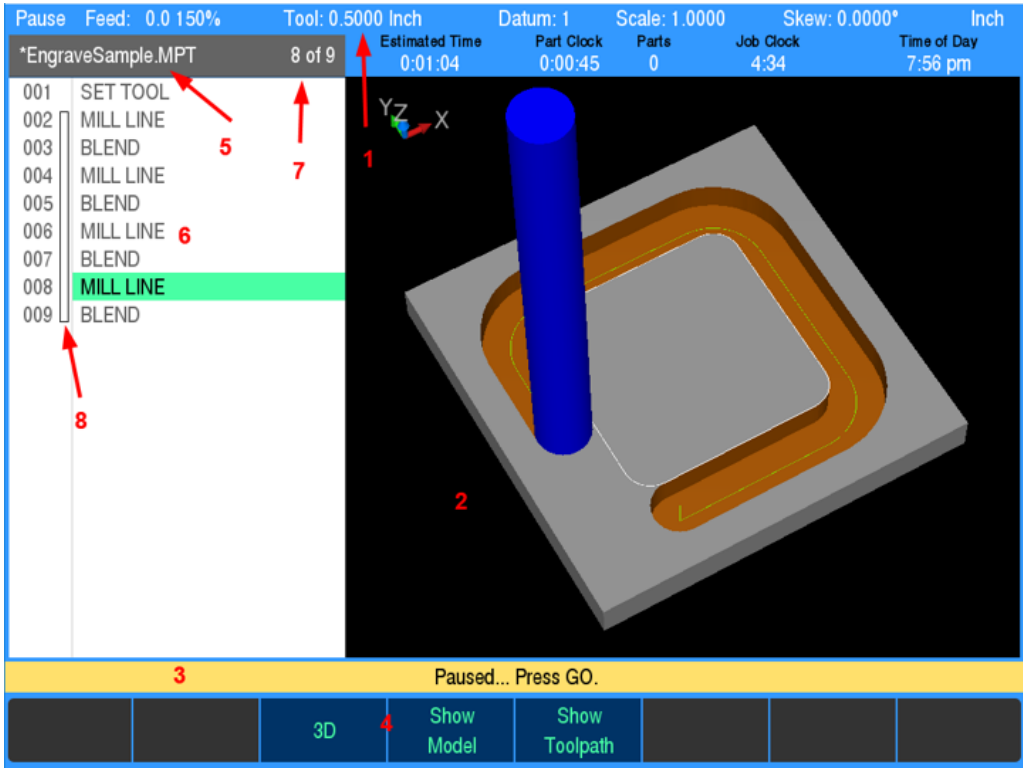
Notes

Program (PGM) Mode

When the DRO / PGM button is pressed the control switches between the digital readout (DRO) and program (PGM) mode. The program mode allows you to assemble multiple steps into a sequential set of operations.

DRO
PGM

1. **Status Bar:** Servo Motor Status, Estimated Time, Tool, Datum, Scale, Skew, (Inch/MM).
2. **Part Graphics:** Display 2D Line, 3D Line or 3D Solid (shown).
3. **Operator Intervention Message:** Prompts for an action required and displays errors.
4. **Soft Keys Display Area:** Indicates the function of the key below.
5. **Program Name**
6. **Program Steps**
7. **Step Range**
8. **Continuous Path Indicator:** Indicates when a tool path is continuous and is open ended or closed loop (shown).



The part graphics can be changed by pressing the view key and using the soft key functions. MILLPWRG2 uses a highlighted cursor to mark a field for selection or editing. The cursor will also change from a highlighted bar to a text cursor when data is entered. Use the ARROWS keys to move the cursor. The UP, DOWN arrows move the cursor through the fields. The RIGHT arrow will open a field that contains more choices, or subfolders in the folder tree. The LEFT arrow will close the menu, or subfolders.

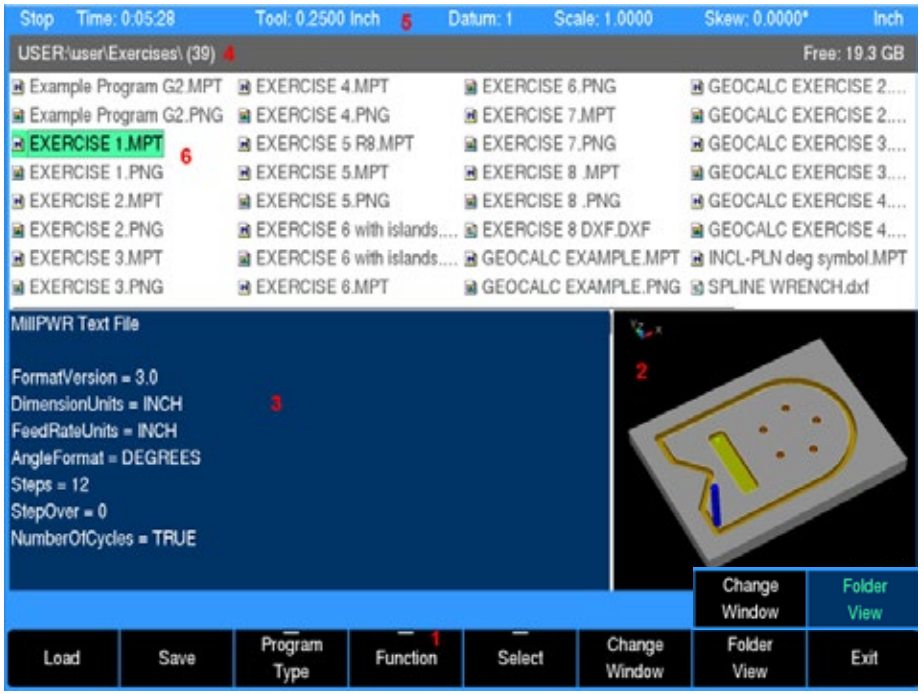
Notes

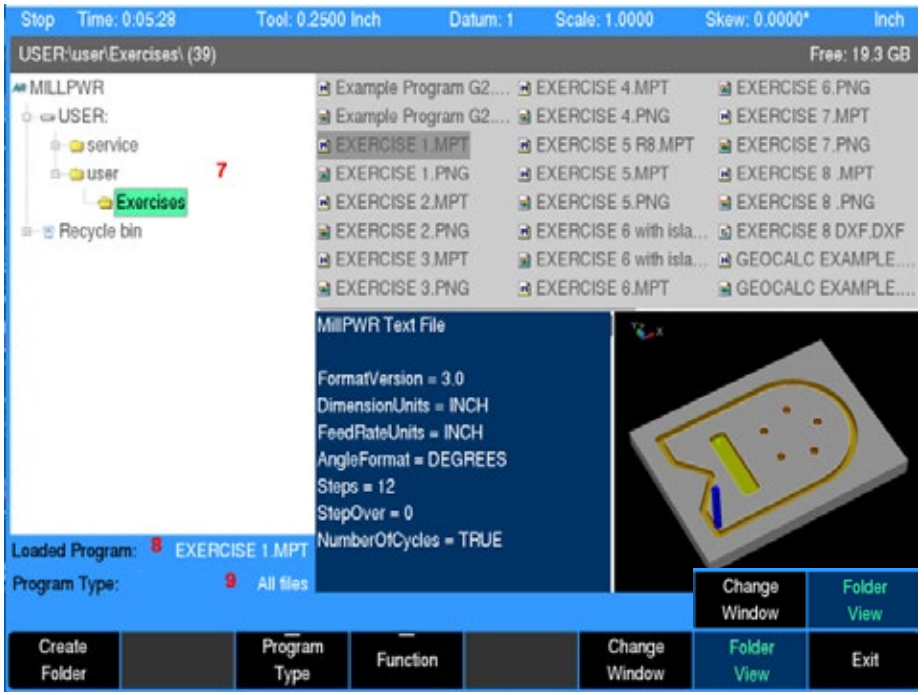
Program (PGM) Mode

Program Functions

When the Program Functions soft key is pressed the view switches to the program management screen. The program management area allows you to load, save, rename, delete and relocate programs. Access programs stored in the console, on a Network, and on USB memory devices. The default storage folder for the console is v:\user\.

1. **Soft Keys** change as required for the action you perform.
2. **Program Graphic Preview** available after a program has been drawn.
3. **Program Steps Preview** helps identify the program you seek.
4. **Current Folder** selected indicates which folder you are viewing.
5. **Status Bar** shows machine status.
6. **Program List** displays the list of all the programs stored in the current location (as filtered by type).
7. **Folder View:** Pressing this soft key opens the folder tree window and pressing the Change Window soft key will change the active window switching between Program List, Program Steps Preview, and Folder View. Use Change Window to select which window you would like to navigate.
8. **Loaded Program:** Shows the name of the program currently loaded.
9. **Program Type:** Shows the file type filter selected for the folder list.



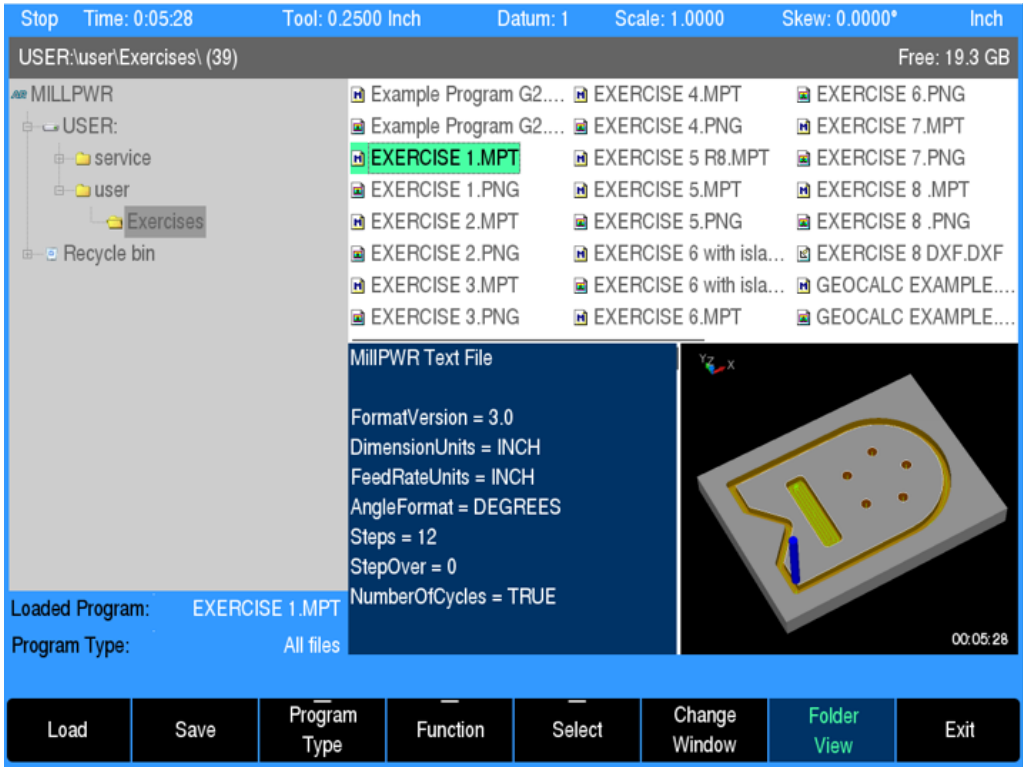


PPT SLIDES 143-144

Notes

Program List

Press the Change Window soft key to select the Program List window. When active the background will turn white. With the program list active the following soft keys are available.



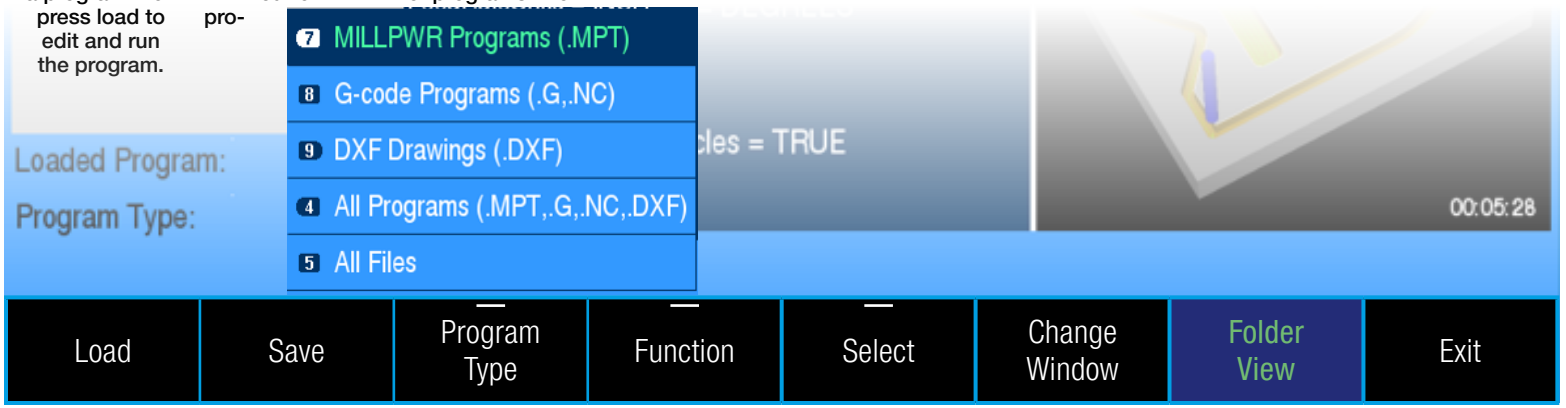
Notes

Program List

Use the arrow keys to highlight a program then press load to edit and run the program.

Press save to save the current pro-

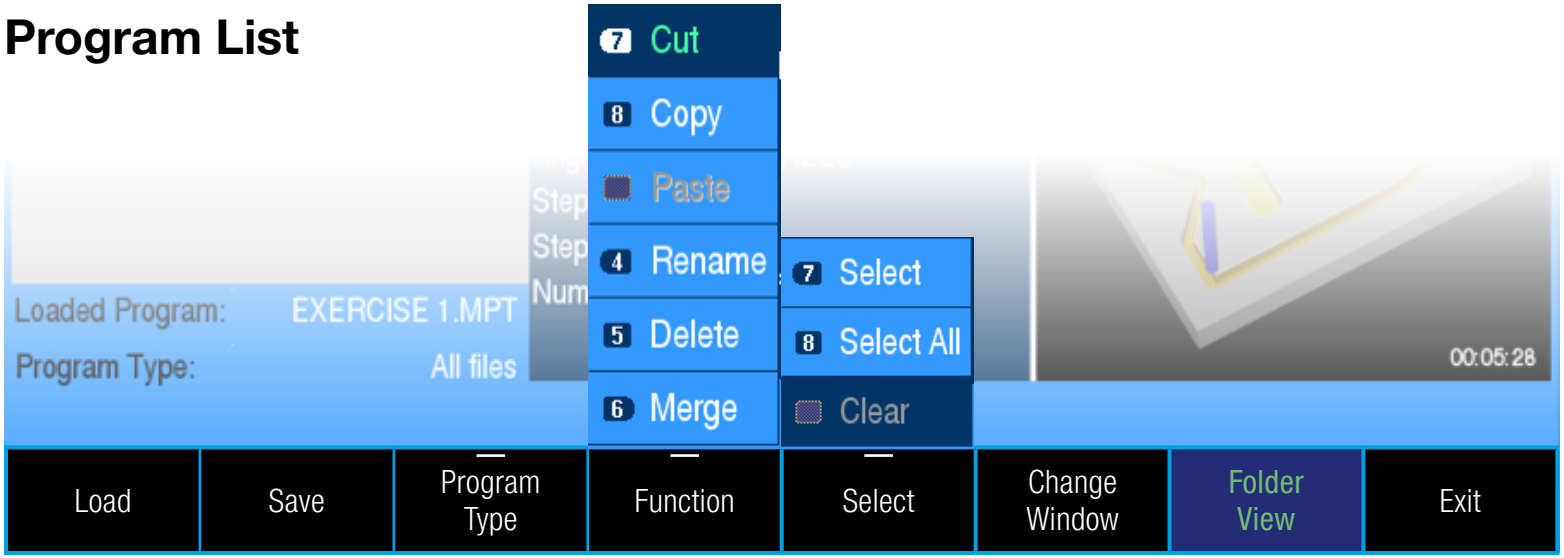
Allows you to filter a large list of programs into



- An arrow on a soft key indicates a list of options.
- Pressing the keypad number shown next to the option will activate that option.
- An inactive option is shown faded.
- An active soft key is shown depressed and with yellow text

Notes

Program List



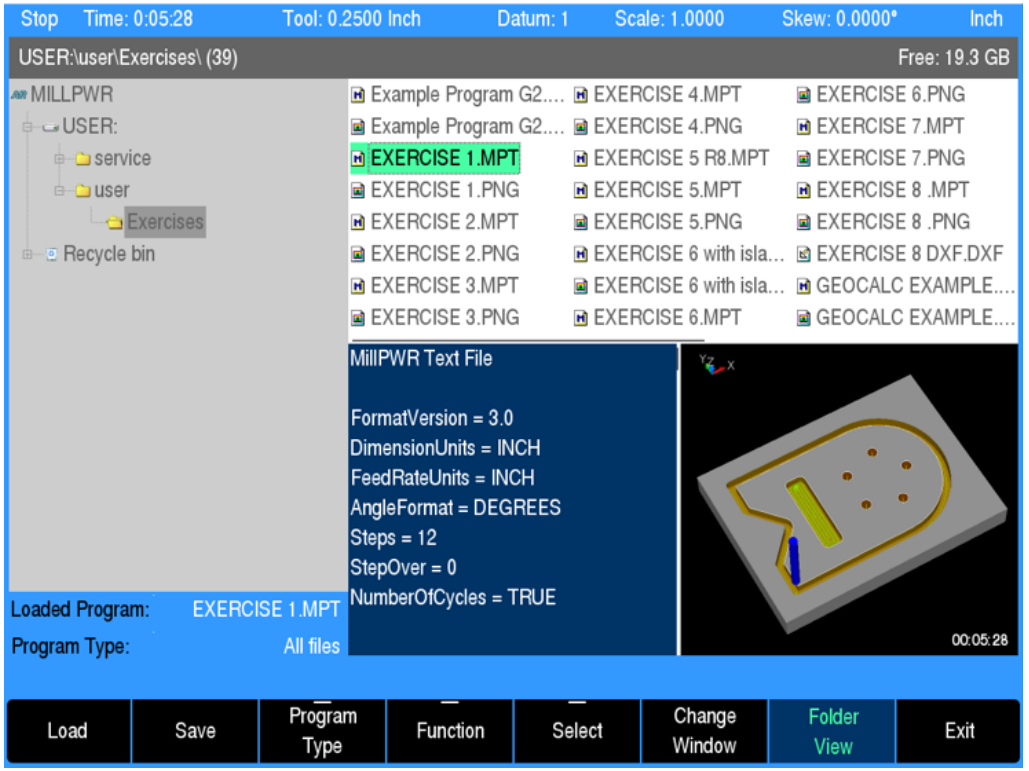
Function	Select	Change Window	Folder View	Exit
Allows you to cut, copy, paste, rename, delete, and merge programs.	Allows you to select more than one program to copy and paste elsewhere. Clear will unselect all selections.	Will change the active window switching between Program List, Program Steps Preview and Folder View.	Shows and hides the folder view (showing but with program list active in example).	Returns you to the program edit mode.

- An arrow on a soft key indicates a list of options.
- Pressing the keypad number shown next to the option will activate that option.
- An inactive option is shown faded.
- An active soft key is shown depressed and with yellow text

Notes

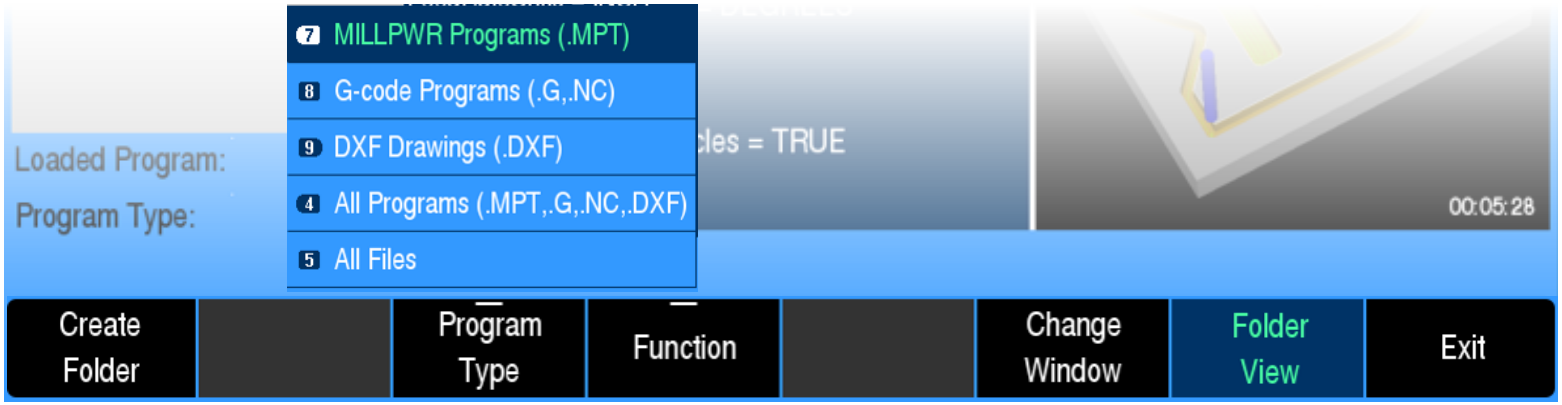
Folder List

Press the Change Window soft key to select the folder tree window. When active, the background will turn white. The folder tree can be used to access a connection to a network, USB memory devices, or folders on your MILL**PWR**^{G2}. Navigate through the folders using the up down arrow keys. Expand a folder with the right arrow key, collapse a folder with the left arrow key. Folders can be created on V:\User which is the MILL**PWR**^{G2} console default folder, on a USB device, on a network, or within sub-folders that have already been created. With the Folder tree window active the following soft keys are available.



Notes

Folder List



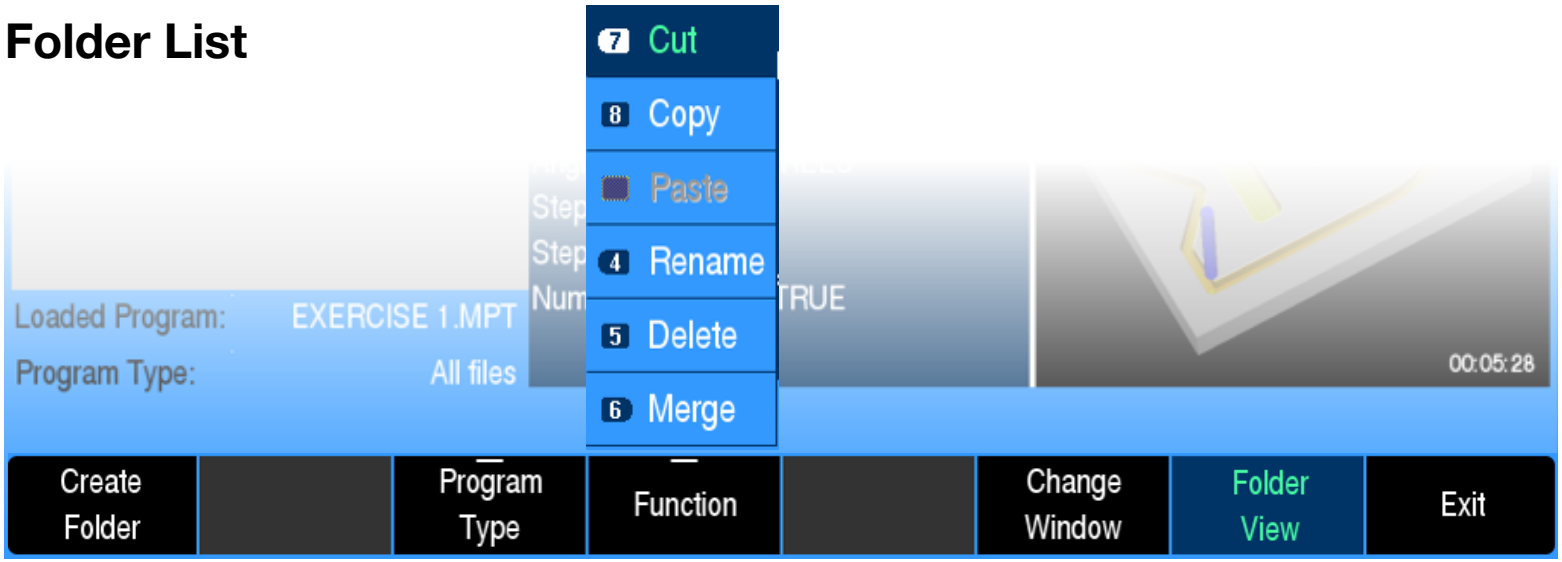
After navigating to where you wish to create a new folder, press Create Folder to create a new folder.

Allows you to filter a large list of programs into easy to navigate views segregated by file type.

- An arrow on a soft key indicates a list of options.
- Pressing the keypad number shown next to the option will activate that option.
- An inactive option is shown faded.
- An active soft key is shown depressed and with yellow text

Notes

Folder List



Allows you to cut, copy, paste, rename, delete, and merge programs.

Will change the active window switching between Program List, Program Steps Preview, and Folder View.

Shows and hides the folder view (the figure to the right shows an active folder view window).

Returns you to the program edit mode.

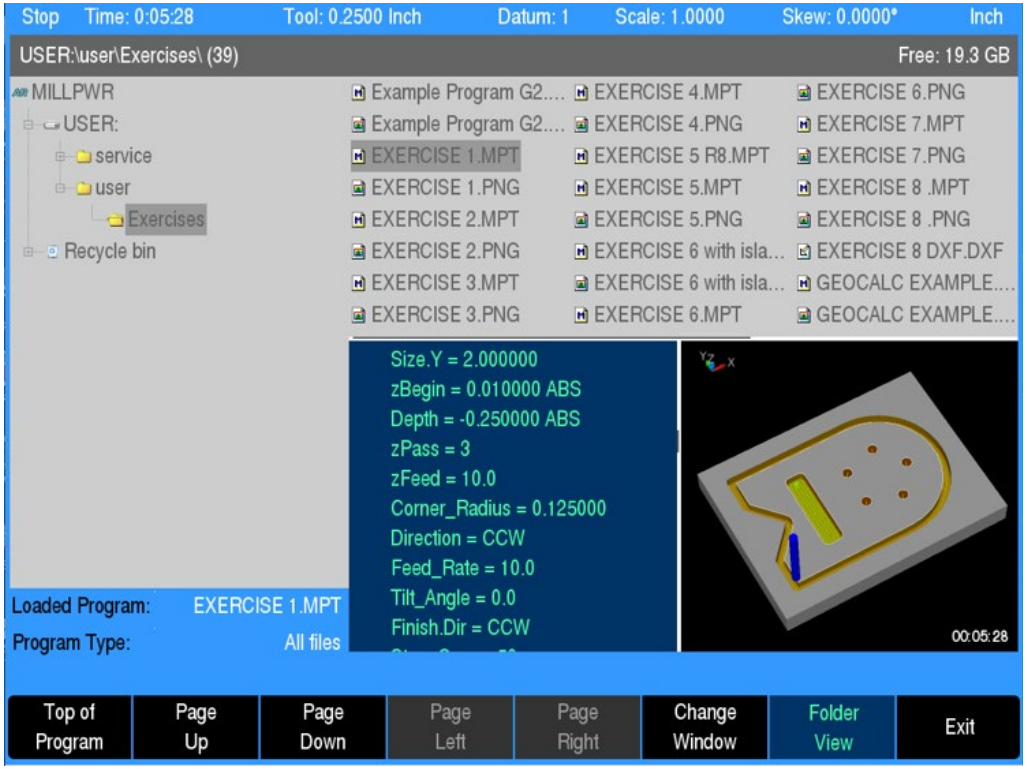
- An arrow on a soft key indicates a list of options.
- Pressing the keypad number shown next to the option will activate that option.
- An inactive option is shown faded.
- An active soft key is shown depressed and with yellow text

Notes

Program Preview

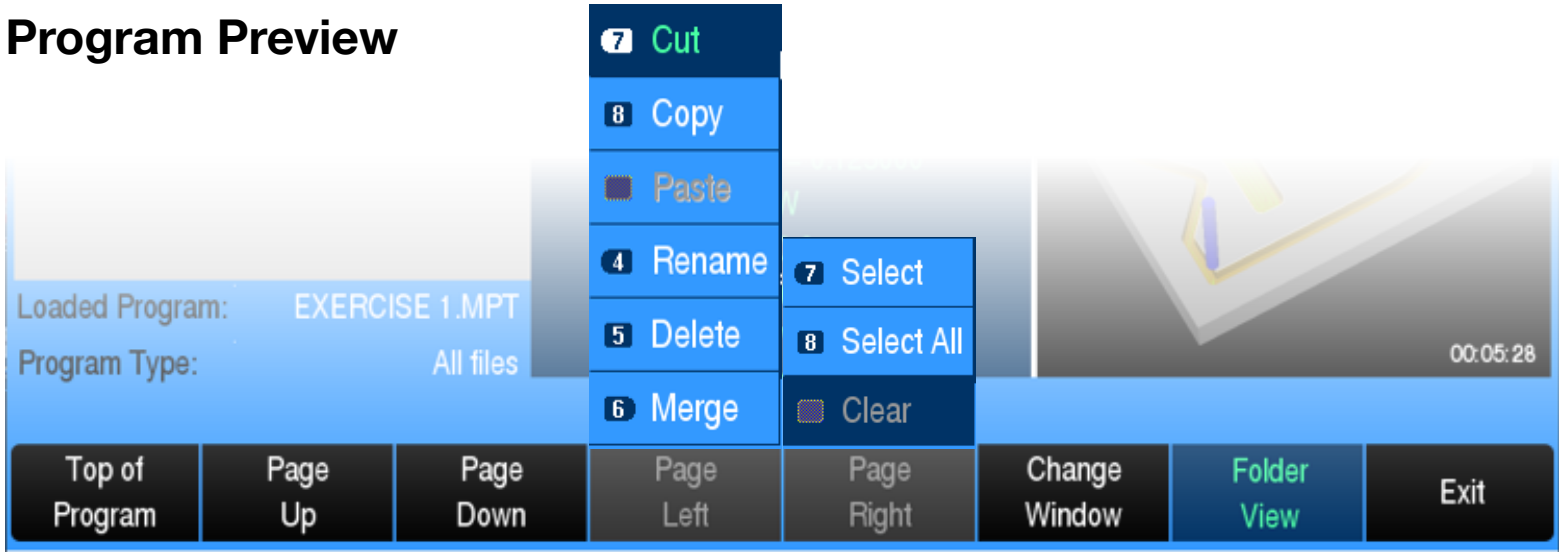
Press the Change Window soft key to select the program preview window. When active, the background will turn white. The program preview allows you to preview a program without opening it for editing. With the program preview window active the following soft keys are available.

Change Window



Notes

Program Preview



Top of Program	Page Up	Page Down	Page Left	Page Right	Change Window	Folder View	Exit
Returns you to the top of the program.	Navigates up one page with each press.	Navigates down one page with each press.	Navigates left one page with each press.	Navigates right one page with each press.		Shows and hides the folder view (showing with program preview active in example).	Returns you to the program edit mode.

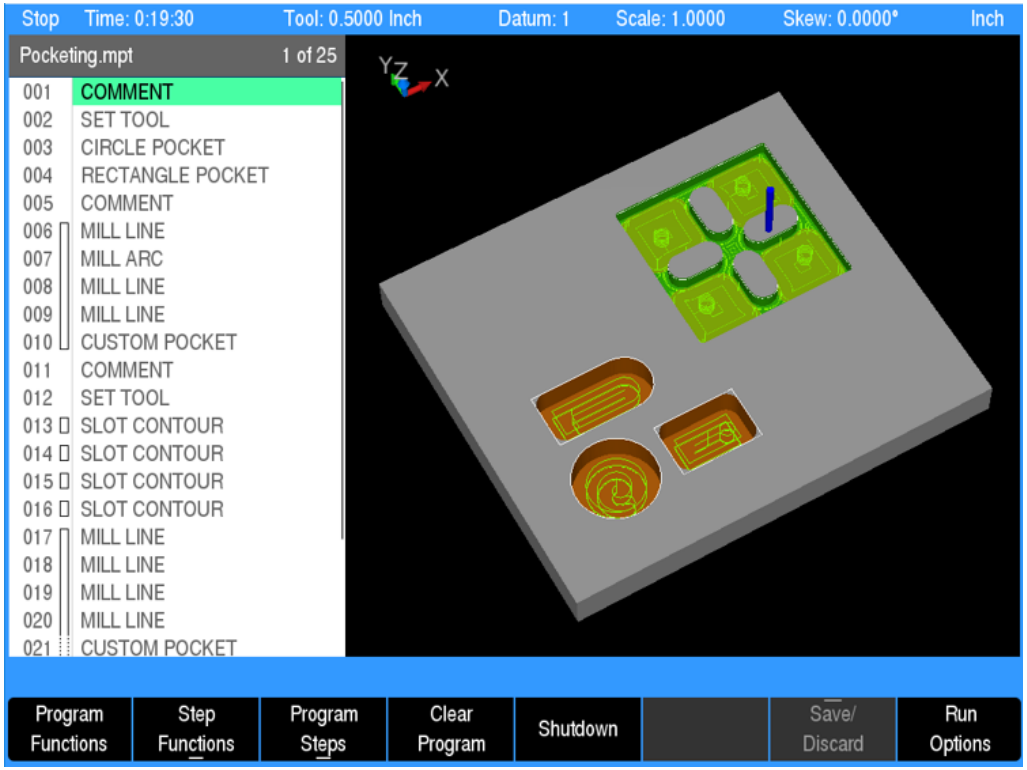
- An unavailable soft key is shown with dull yellow text.
- An available inactive soft key is shown with black text.
- An active soft key is shown depressed and with yellow text

Notes

Creating & Editing a Program

In programming mode you can assemble a sequence of events to machine a complex part. In addition to all the canned cycles previously mentioned, custom contours and pockets, engraving, comments can be entered for display at run time, and auxiliary functions can be triggered, and multiple tool changes programmed. All this is available at the control and easily programmed using information directly off of the print in hand.

The control also allows simple and advanced math calculation directly in any entry field using the keypad and the CALC key to access advanced calculator functions. The graphics provide you with a choice of 2D, 3D, and #d solid views and can be customized to suit your needs. The part graphics updates with every programmed step you create and also can be updated by pressing the decimal key.



For proofing your program Run Options allow you to run your program in graphics only, single step, and in dry run modes. You can skip a step by highlighting it and pressing the +/- key. Conversely, you can highlight any skipped step and press the +/- key to remove the skip designation. The feed rate override will also provide you with additional control throughout the process of confirming your program will run as expected.

Notes

Creating & Editing a Program

Step Functions

The triangle in the lower right corner of a soft key indicates that pressing it will open another layer of soft keys. The Step Functions soft key accesses the soft keys listed below.

Explode		Reverse Step	Reverse Path	Shift/Rotate Steps	Change Steps	Delete Steps	Copy/Move Steps
Pressing the Explode soft key will explode a highlighted program step into individual steps allowing you to modify them individually and also begin or resume machining from a specific step. You can explode the following functions: Hole Patterns, Repeat, Mirror, and Rotate Steps.		Switches the FROM and TO points and TOOL OFFSET of a highlighted step. Place the highlight curser on any step and press the softkey to reverse a step. The offset is reversed to keep the cutter compensation on the same side of the path when reversing direction.	Reverses any continuous tool path in the same manner as reverse step. Place the highlight curser within any path and press the softkey to reverse a step.				

- An unavailable soft key is shown with dull yellow text.
- An available inactive soft key is shown with black text.
- An active soft key is shown depressed and with yellow text.

Notes

Creating & Editing a Program

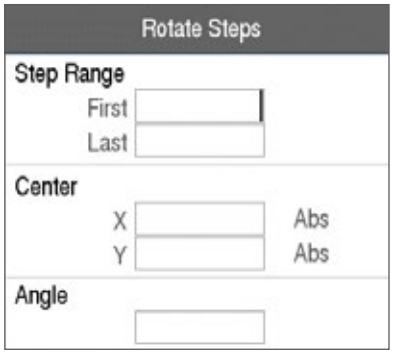
Step Functions

The triangle in the lower right corner of a soft key indicates that pressing it will open another layer of soft keys. The Step Functions soft key accesses the soft keys listed below.

Explode		Reverse Step	Reverse Path	Shift/Rotate Steps	Change Steps	Delete Steps	Copy/Move Steps

- An unavailable soft key is shown with dull yellow text.
- An available inactive soft key is shown with black text.
- An active soft key is shown depressed and with yellow text.

Pressing the Explode soft key will explode a highlighted program step into individual steps allowing you to modify them individually and also begin or resume machining from a specific step. You can explode the following functions: Hole Patterns, Repeat, Mirror, and Rotate Steps.



Notes

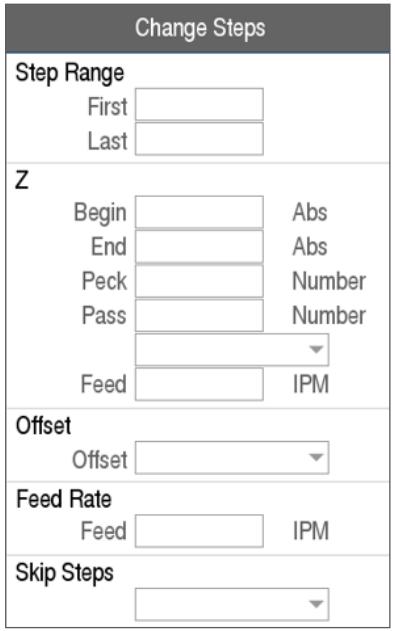
Creating & Editing a Program

Step Functions

The triangle in the lower right corner of a soft key indicates that pressing it will open another layer of soft keys. The Step Functions soft key accesses the soft keys listed below.

Explode		Reverse Step	Reverse Path	Shift/Rotate Steps	Change Steps	Delete Steps	Copy/Move Steps

- An unavailable soft key is shown with dull yellow text.
- An available inactive soft key is shown with black text.
- An active soft key is shown depressed and with yellow text.



Gives you the ability to change several steps simultaneously or mark them to be skipped. Enter the first and last step of the range you wish to change and define the changes to be made in the fields provided. You can change one or more of the variables using the fields provided. Steps that are set to be skipped will be shown greyed out with an asterisk. A skipped step cannot be edited. Once back to programming mode, highlighting a skipped step and pressing the +/- key will remove the skip tag. Conversely, you can highlight any step and press the +/- key to mark it to be skipped individually. .

Notes

Creating & Editing a Program

Step Functions

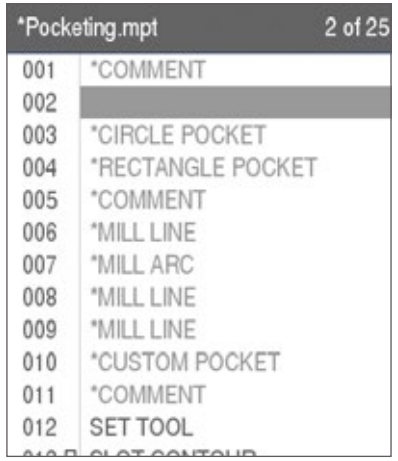
The triangle in the lower right corner of a soft key indicates that pressing it will open another layer of soft keys. The Step Functions soft key accesses the soft keys listed below.

Explode		Reverse Step	Reverse Path	Shift/Rotate Steps	Change Steps	Delete Steps	Copy/Move Steps

- An unavailable soft key is shown with dull yellow text.
- An available inactive soft key is shown with black text.
- An active soft key is shown depressed and with yellow text.

Delete Steps — Allows you to delete a range of steps you designate at the same time. The Clear key will delete a single highlighted step.

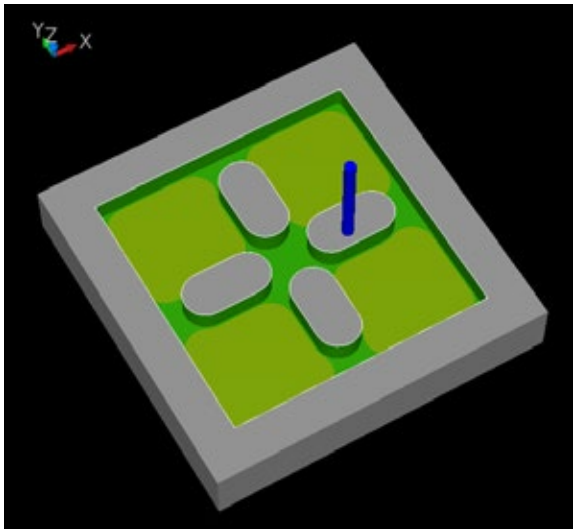
Copy/Move Steps — Makes it easy for you to duplicate or rearrange steps within your program. Place the cursor in the location you wish to copy or move to.



Notes

Program Steps

Pressing Program Steps opens soft key features that include alternative access to Tool, Position / Milling, Hole Pattern, Rectangle, and Circle and are to the same hard key functions listed earlier. In addition the following program steps are available.



Notes

Program Steps

Custom Pocket

Place the custom pocket step at the end of a continuous tool path.

Step Range

Will indicate the range of steps detected.

Staging Point (Optional)

Where the tool will rapid to above the pocket.

Z

- **Pass:** Establishes the number of cuts between the Z Begin and End.
- **Feed:** Establishes the Z rate of motion for the Z axis only.

Tool

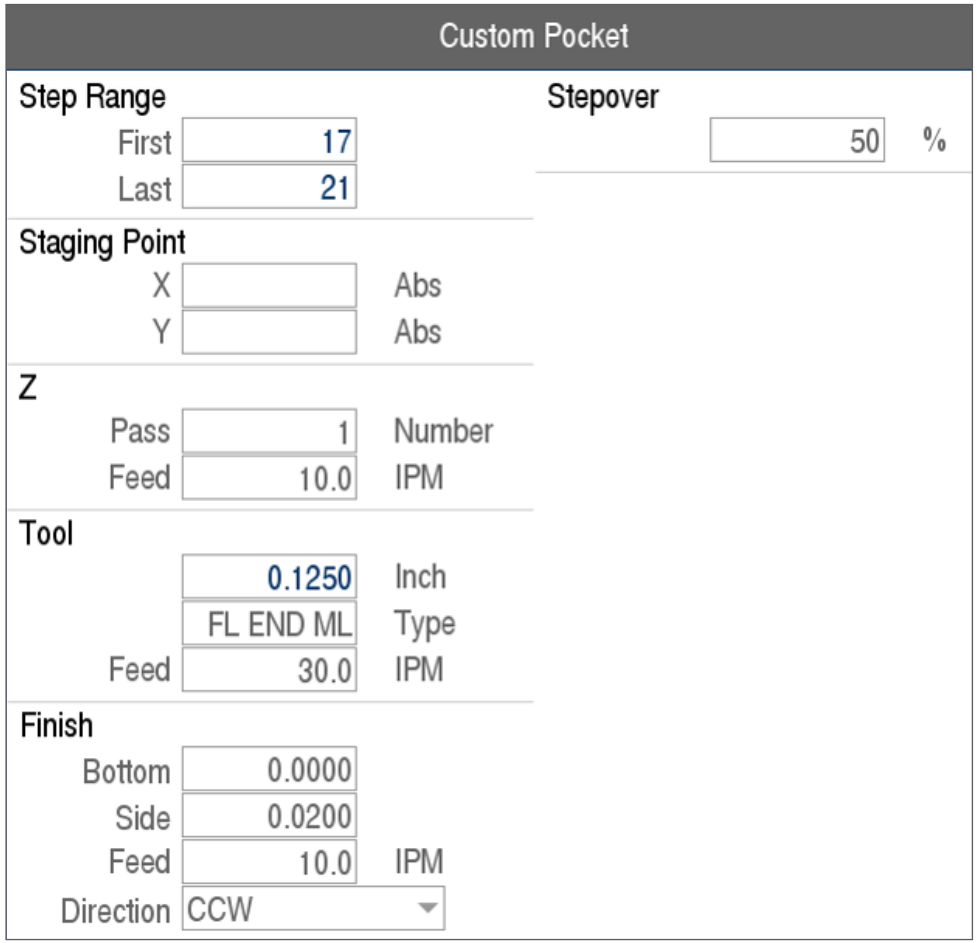
Feed in the Tool field establishes the rate at which the tool will feed along the X, Y plane.

Finish

Establishes the amount of stock (Cut) to leave for the final finish pass along with the Feed rate and Direction to use.

Stepover

Allows you to define the percentage of the tool to use per pass for the roughing cut. The default is 50%.

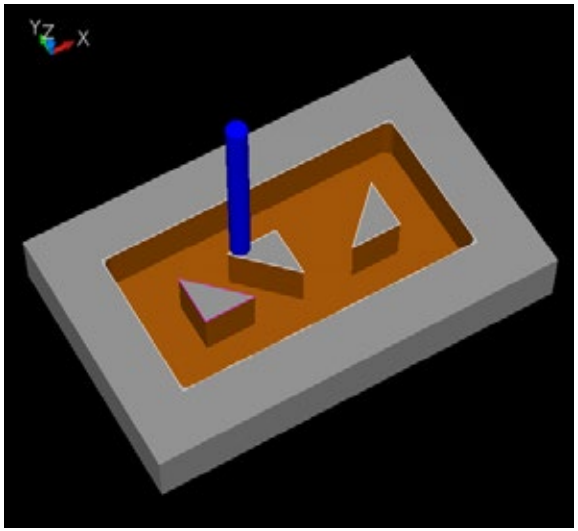


Notes

Program Steps

Island

Added to a closed tool path, Island defines an area to remain uncut within a custom pocket. Lines and arcs used to describe an island must immediately precede the island step and the Island step must immediately precede the steps for the custom pocket contour followed by the Custom Pocket Step. Anything placed in-between will interrupt the cycle. Place the cursor beneath your irregular tool path, press the Program Steps soft key then the Custom Pocket soft key. Highlight Island and press enter or press the 8 key to add a contour step to an irregular tool path.

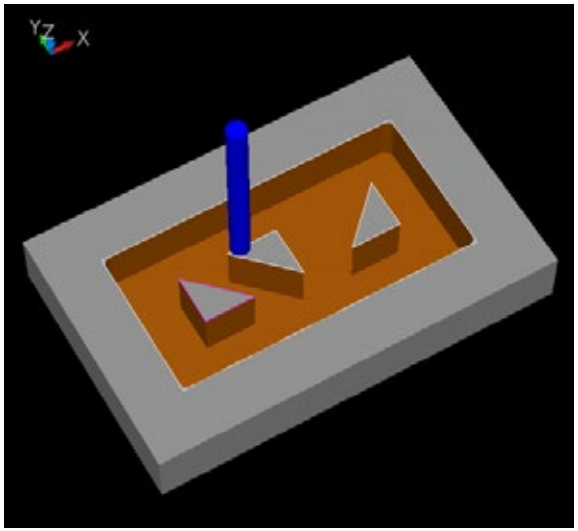


Notes

Program Steps

Island

Circle and rectangle pocket cycles and slot cycles can be used as islands when set as contour within the step.



Using lines and arcs to create Islands.

Island	
Step Range	
First	2
Last	4

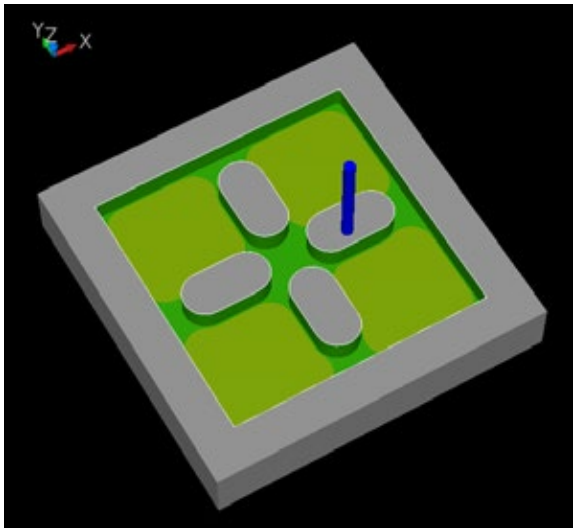
CustomPocket.mpt	5 of 19
001	SET TOOL
002	MILL LINE
003	MILL LINE
004	MILL LINE
005	ISLAND
006	MILL LINE
007	MILL LINE
008	MILL LINE
009	ISLAND
010	MILL LINE
011	MILL LINE
012	MILL LINE
013	ISLAND
014	MILL LINE
015	MILL LINE
016	MILL LINE
017	MILL LINE
018	MILL LINE
019	CUSTOM POCKET

Notes

Program Steps

Island

Circle and rectangle pocket cycles and slot cycles can be used as islands when set as contour within the step.



Setting a canned cycle to Contour for use as an Island.

012	SET TOOL
013	SLOT CONTOUR
014	SLOT CONTOUR
015	SLOT CONTOUR
016	SLOT CONTOUR
017	MILL LINE
018	MILL LINE
019	MILL LINE
020	MILL LINE
021	CUSTOM POCKET
022	SET TOOL
023	ROUGH MILL
024	SET TOOL
025	ROUGH MILL

Type

Contour

Notes

Advanced Custom Pocket Machining Features

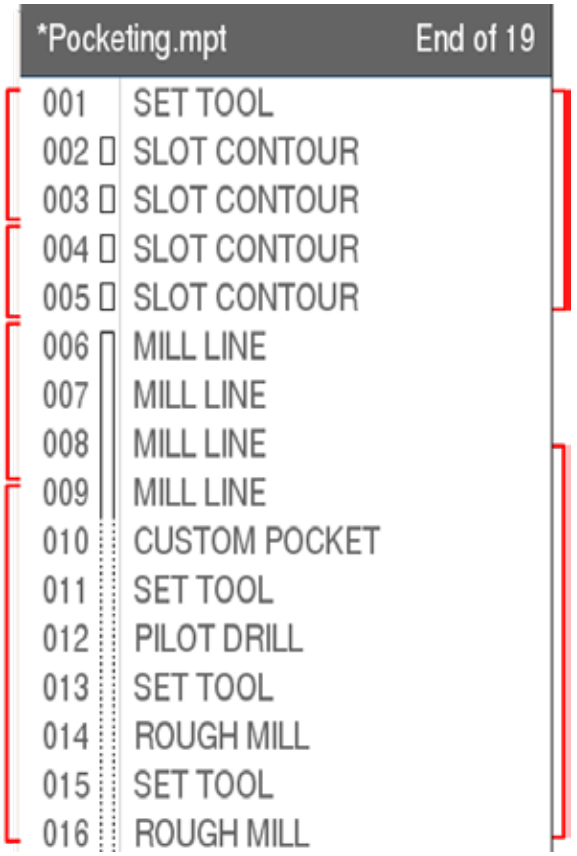
Contours are describes using Lines, Arcs, Blends, and Chamfers.

Canned cycles can be designated as contours and now used as Islands.

Contours can define Islands when followed by the Island step.

Contours define custom pockets when followed by the custom pocket step.

Advanced pocketing features can be added to a custom pocket. And are indicated by the dashed contour line extension.



All Islands must precede the contour of the custom pocket. Islands are allowed to overlap each other and overlap the custom pocket.

Advanced features must follow the custom pocket step.

If a tool change is needed the tool step must precede the advanced feature. If no tool step is added, the previous tool will be used.

Rough milling must precede finish milling.

Bottom finish must precede side finish.

Notes

Custom Pocket

Pilot Drill

(use if there is not room for a ramp in move)

The Pilot Drill option (Step 12) when used with a tool step (Step 11) will drill a pilot hole at every location where a tool needs to plunge. This allows you to use a tool that cannot center cut. The step will show you two tool fields. The first field is the preceding tool step and the second indicates the default tool (the tool that precedes this routine). During operation a prompt is displayed to use the pilot tool. Holes are then drilled at each calculated entry location. These locations are calculated with each run so any program change is automatically accounted for. The tool retracts to the active datum's retract position between holes.

Peck

The distance between each chip break move.

Feed

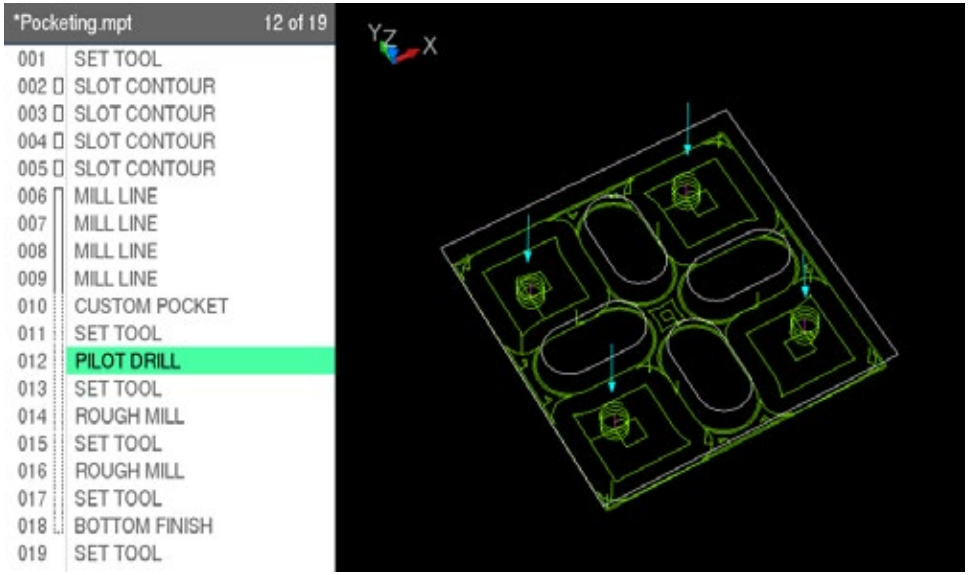
The rate of downward Z-Axis feed.

Tool

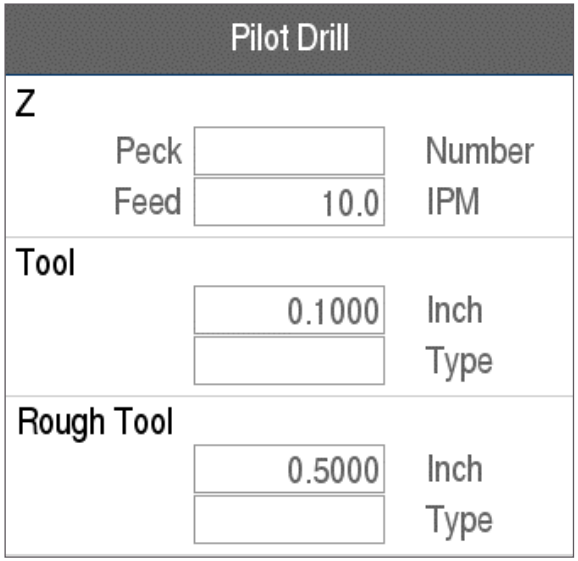
Indicates the tool to be used when pilot drilling

Rough Tool

Indicates the tool that will be used for roughing.



Line 12 - Pilot Drill Step will predrill at the locations where a tool will plunge. It updates these locations each time the program is edited.



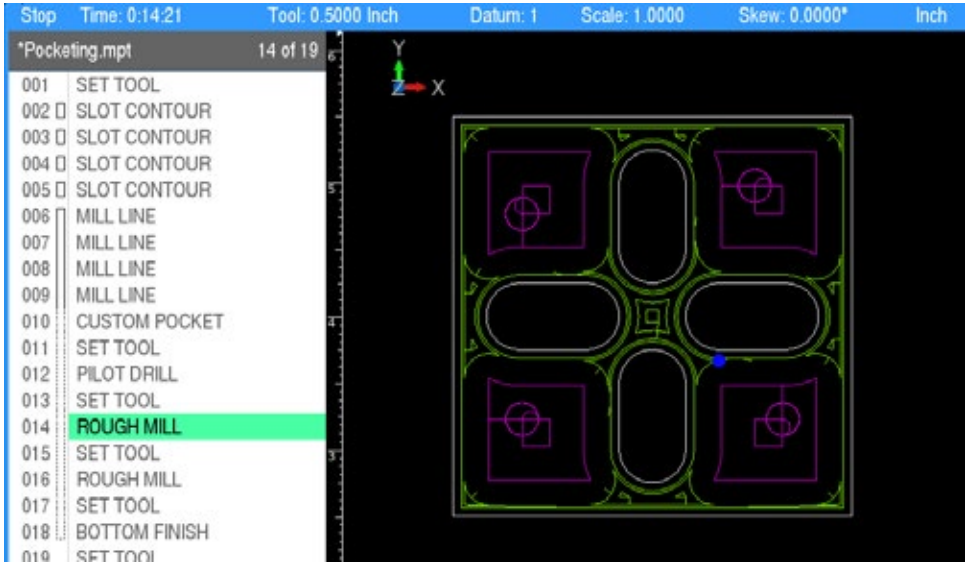
Notes

Custom Pocket

Rough Mill

(Clear a pocket with a large diameter roughing tool)

Used with a tool step (Step 13), Rough Mill (Step 14) allows you to clear a custom pocket using a large diameter tool and finish with the default tool or a finish tool specified later. You can have multiple rough tools programmed. Each tool should be smaller than the previous. G2 will use each subsequent roughing pass only within the areas left by the previous roughing cycle. The first rough cycle is indicated with the blue arrows.



Line 14 – 1st Rough Mill Step

Pass

The number of passes or the distance per pass until final depth is reached.

Feed

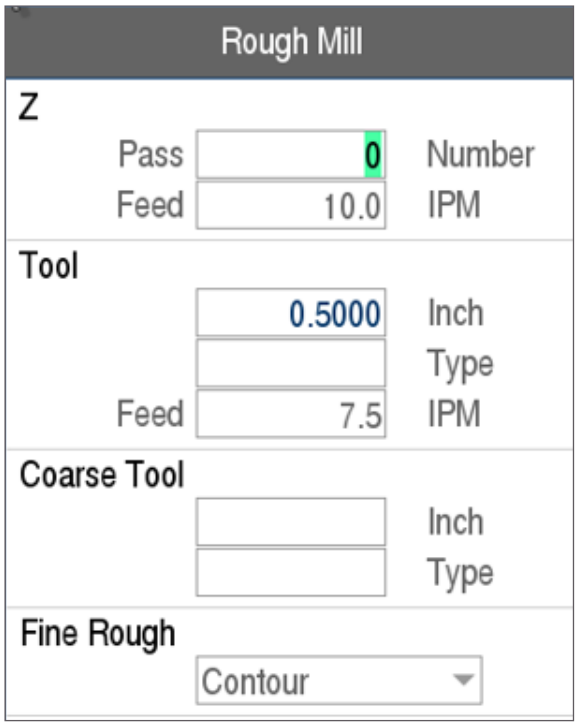
The rate of interpolation rate in the XY plane.

Tool

Indicates the tool to be used for this Rough Mill step.

Coarse Tool

Indicates the tool used in a prior Rough Mill step. Since this is the first rough mill step, no prior rough tool is listed.



PPT SLIDES 187-188

Notes

Custom Pocket

2nd Rough Mill

(Clear areas the larger rough cycle cannot reach (Optional))

Used with a tool step (Step 15), Rough Mill (Step 16) continues to rough clear the custom pocket in areas the first cycle could not reach. Each roughing tool should be smaller than the previous. The same tool used for the last roughing cycle will be used for finish cycles if no additional tool change step is programmed.

Pass

The number of passes or the distance per pass until final depth is reached.

Feed

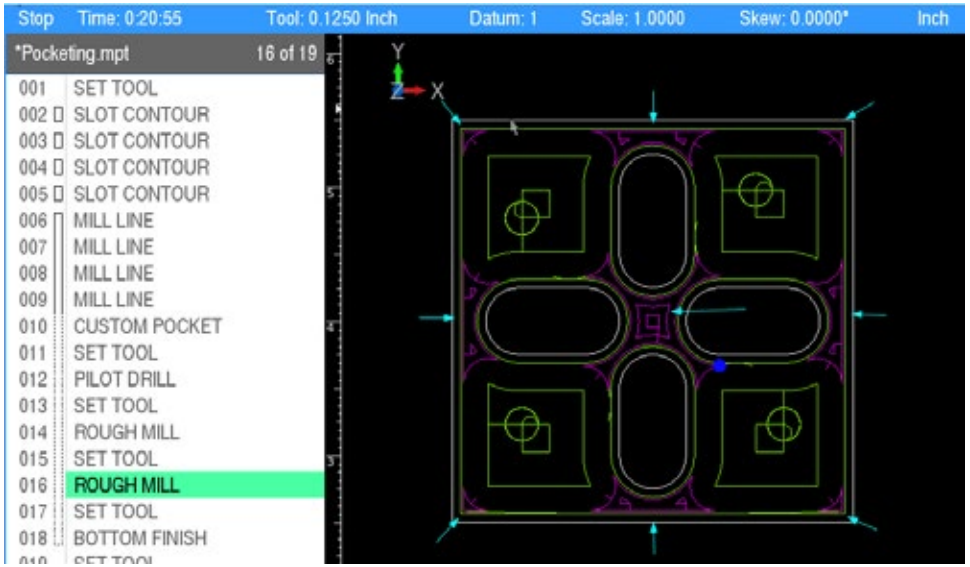
The rate of interpolation rate in the XY plane.

Tool

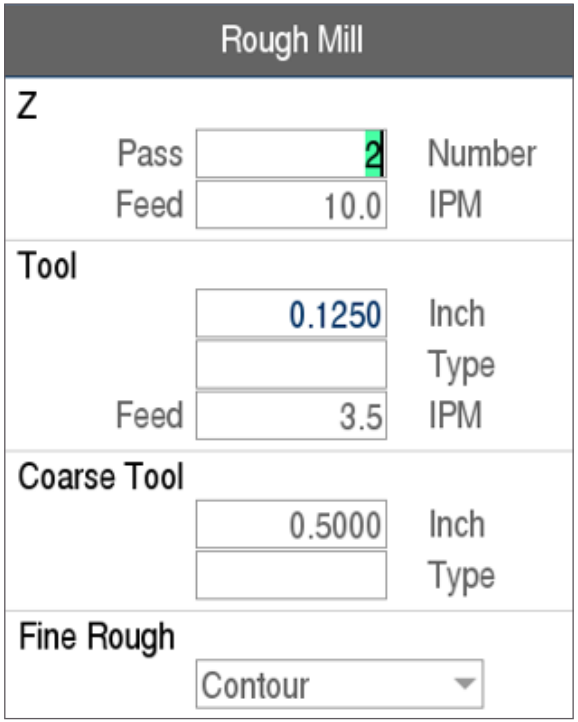
Indicates the tool to be used for this Rough Mill step.

Coarse Tool

Indicates the tool used in a prior Rough Mill step.



Line 16 – 2nd Rough Mill Step



PPT SLIDES 189-190

Notes

Custom Pocket

Bottom Finish & Side Finish

Bottom Finish (Line 17) and Side Finish (Line 18) allow you to complete a custom pocket routine. You can program more than one side finish. Specify the amount to remain for the second side finish in the Side field. If you do not specify a tool along before the finish steps the operation will proceed with the last current tool. You can specify a tool for the bottom finish and one for each side finish step if you wish. In this example the same tool will be used for the final rough pass and both finish operations.

Z

Feed: The rate the tool will feed between the Z Begin and End established in the contour steps that define the custom pocket

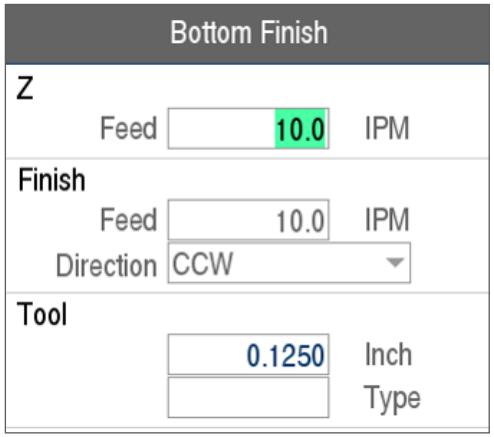
Finish

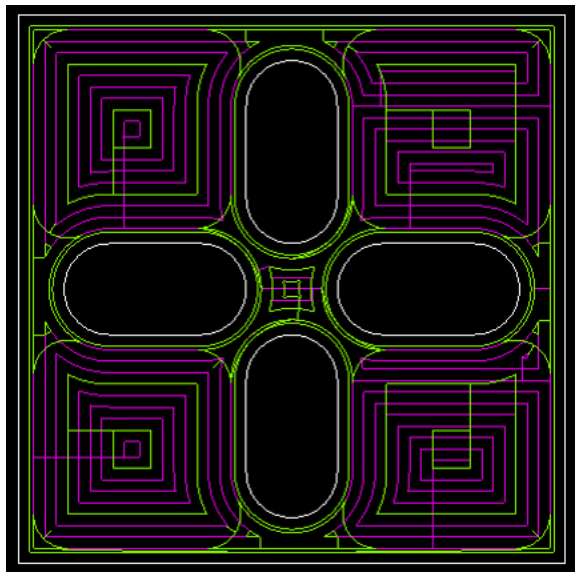
- **Cut:** Allows you to leave stock for a second side finish.
- **Feed:** The rate of interpolation rate in the XY plane.
- **Direction:** Establishes a climb cut or conventional cut.

Tool

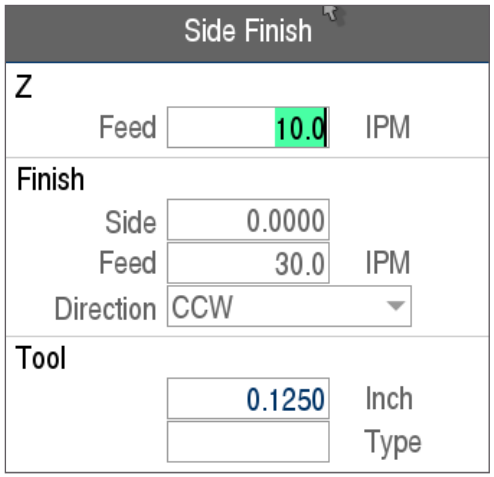
Indicates the tool to be used for the finish step.

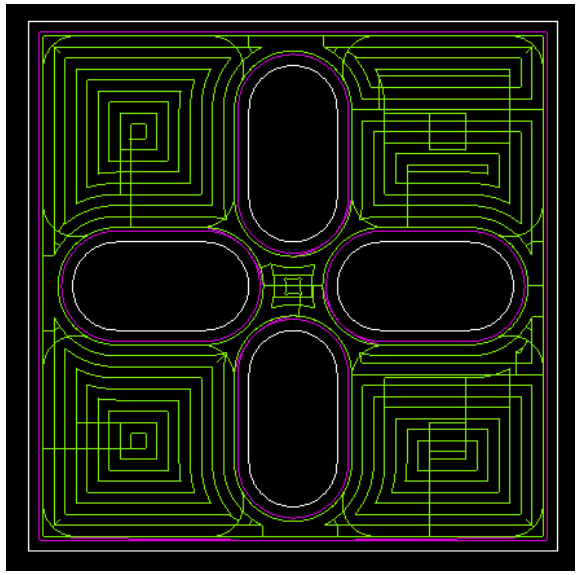
Line 17 – Bottom Finish





Line 18 – Side Finish





PPT SLIDES 191-192

Notes

Custom Pocket Review

A Custom Pocket can be very versatile ranging from a simply clearing a contour to a multi event process involving specialty tooling such as high helix finishing tools, tapered end mills, etc. The minimum requirement for a custom pocket is a closed path contour. Adding one or more islands allows you to clear an area around features.

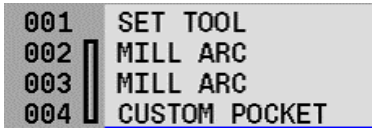
The **Pilot Drill** option allows you to use tooling not designed to plunge cut when room to ramp in is too limited. This options automatically determines the proper hole locations required every time the program is run adjusting when tool diameters are changed and the programmed contour is revised.

Rough Mill allows the use of custom tools to quickly and efficiently rough out an area reducing cycle time and extending tool life. It eliminates the need to program separate cycles one for roughing and another for finishing, reducing the amount of program need and editing later when part revisions are required. More than one roughing cycle can be added to a custom pocket allowing even greater flexibility.

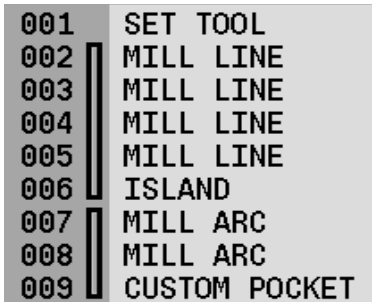
Bottom and Side Finish cycles allow for enhances finishing these cycles allow you to designate separate tools for bottom and side finishing. When not added to a custom pocket cycle, the default tool will perform both bottom and side finish cycles. Below are examples of custom pockets from the simplest to the complex.

Notes

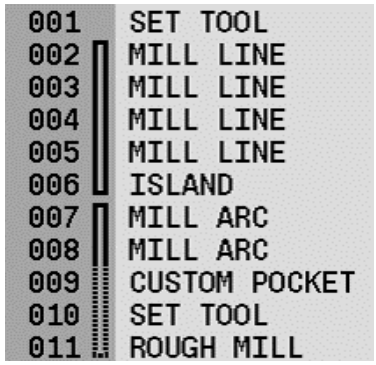
Custom Pocket Review



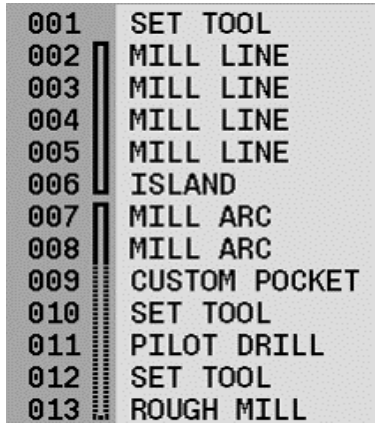
Tool (Step 1) will rough and finish the pocket as programmed.



Tool (Step 1) will rough and finish the pocket as programmed. Leaving the Island.



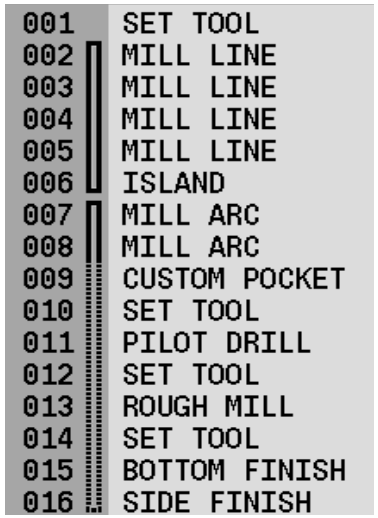
Tool (Step 10) will rough and tool (Step 1) will finish the pocket as programmed. Leaving the Island.



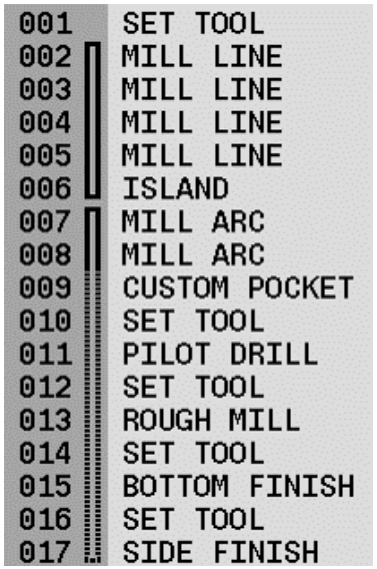
Tool (Step 10) will pilot drill then tool (Step 12) will rough then tool (Step 1) will finish.

Notes

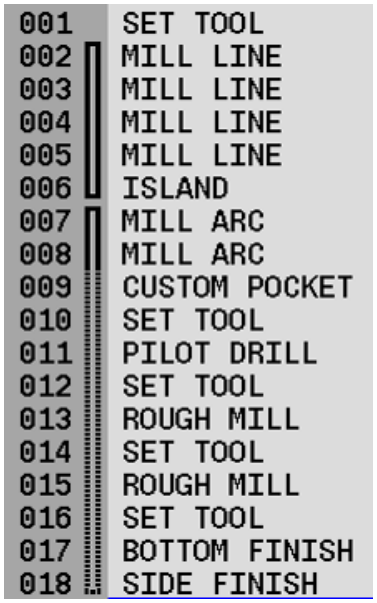
Custom Pocket Review



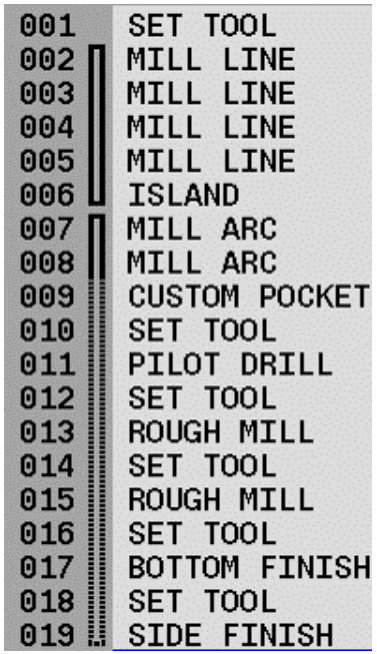
Tool (Step 10) will pilot drill then tool (Step 12) will rough then tool (Step 14) will bot-tom and side finish.



Tool (Step 10) will pilot drill then (Step 12) will rough then (Step 14) will bottom finish and (Step 16) will side finish.



Tool (Step 10) will pilot drill, (Step 12) will rough first, (Step 14) will rough second, (Step 16) will bottom finish and side finish.



Tool (Step 10) will pilot drill, (Step 12) will rough first, (Step 14) will rough second, (Step 16) will bottom finish and (Step 18) will side finish.

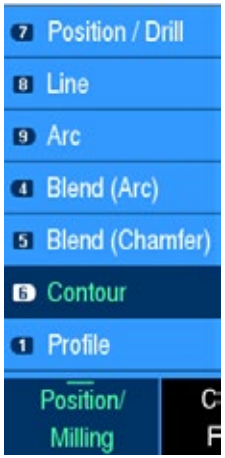
Notes

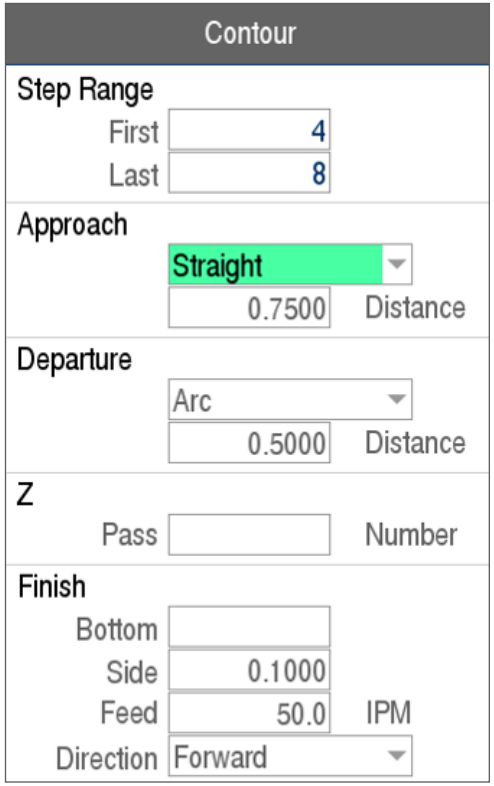
Program Steps

Contour

(Now located under Position/Milling)

When added to the end of a continuous tool path Contour provides you with the ability to add a lead on (approach), lead off (Departure) to the tool path. In addition, it also provides the ability to cut the contour in progressive passes and apply a finish pass like those found in canned cycles. Place the cursor beneath your tool path, press the Program Steps soft key then the Position/Milling soft key. Highlight Contour or press the 6 key to add a contour step to a custom tool path consisting of lines, arcs, blends, and bevels.





Step Range

Will indicate the range of steps detected.

Approach

Choose None, Straight, or Arc for the approach type and provide a distance off of the contour to use.

Departure

Choose None, Straight, or Arc for the departure type and provide a distance off of the contour to use. Choose none if you plan to finish in a reverse direction.

Pass

Establishes the number of cuts between the Z Begin and End.

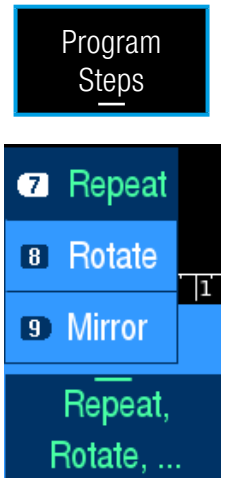
Finish

Establishes the amount of stock (**Cut**) to leave for the final finish pass for **Bottom** and **Side** finishing along with the **Feed** rate and **Direction** to use.

Notes

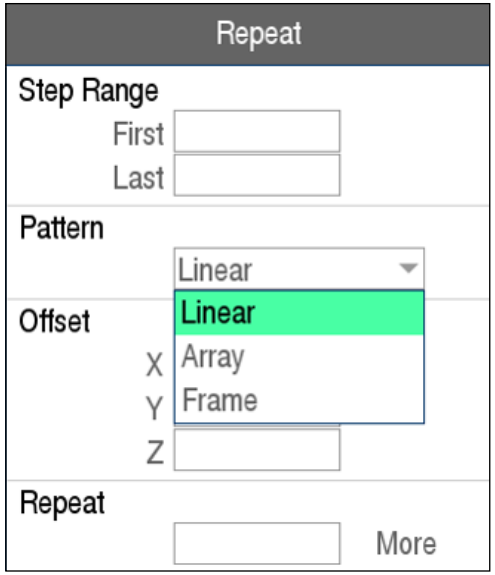
Program Steps

Pressing Program Steps opens soft key features that include alternative access to Tool, Position / Milling, Hole Pattern, Rectangle, and Circle and are to the same hard key functions listed earlier. In addition Repeat, Rotate, and Mirror are available.



Repeat

Allows you to run a set of program steps in multiple locations and/or in increasing depths. A repeat can be placed anywhere in the program after the original steps being repeated. Designate the step range, the X, Y, and or Z offset, any pattern option if desired, and the number of repeats.

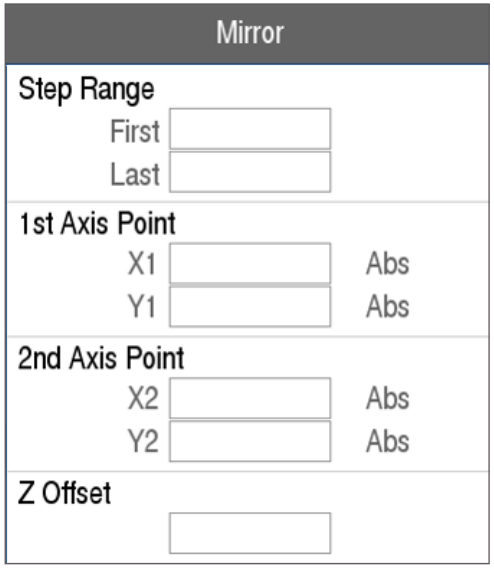


Rotate

Like repeat, rotate allows you to run a set of program steps in multiple locations around a common point and if required in increasing depths. A rotate must be at some point after the original steps. Designate the step range, X, Y center and any Z offset. Angle establishes the amount of rotation between the number of repeats.

Mirror

Allows you to flip a step range on an axis. A mirror step must be at some point after the original steps. Designate the step range. The 1st Axis Point is the begin point and the 2nd Axis Point is the end point that defines the reflection line that like the spine of a book anchors the mirror reflection. A change in Y defines a vertical line creating a left or right hand mirror. A change in X defines a horizontal line mirroring above or below. A change in both axes defines a diagonal line allowing a mirrored image on a bias.



PPT SLIDE 204-207

Notes

Program Steps

Engrave Line

Text
Enter the text to be engraved. Letters numbers and symbols are allowed. The Alphanumeric Keyboard Soft key activates a keyboard to enter the text.

Lower Left Corner
Locates the text on your work piece. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Character Height
Establishes the size of the text.

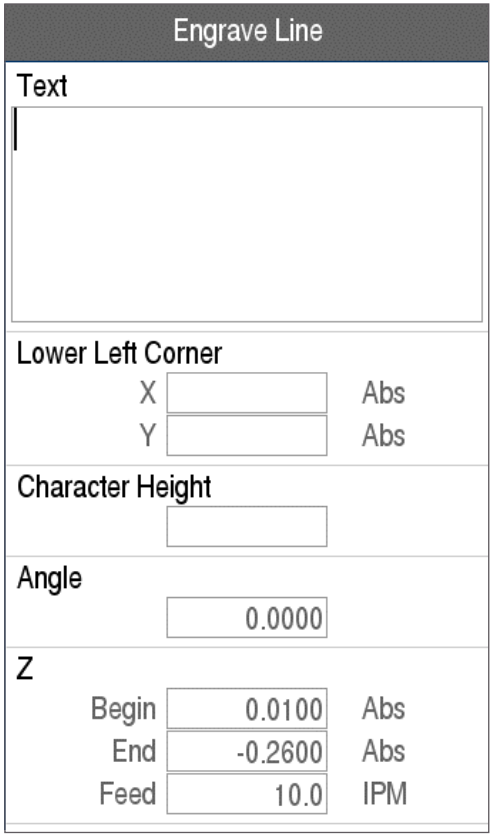
Z
- **Begin** is where the tool will rapid to.
- From there it will feed at the given rate to the **End** value at the **Feed** provided.
- This feed is for the Z axis only.
- When finished, the tool then rapids up to the retract plane set in datum.

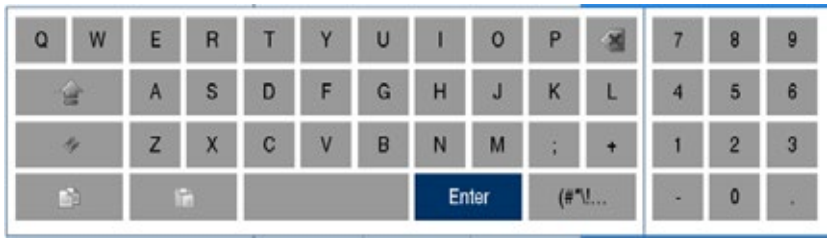
Style
Select font style Simple, Stick, or Stencil.

Modifier
Select Normal or Mirrored.

Tool
Feed is the rate of feed in the X, Y plane.

Font Options





A USB mouse and keyboard can be used for easy programming.

Notes

Program Steps

Engrave Arc

Text
Enter the text to be engraved. Letters numbers and symbols are allowed. The Alphanumeric Keyboard Soft key activates a keyboard to enter the text.

Center
Locates the text on your work piece. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.

Character Height
Establishes the size of the text.

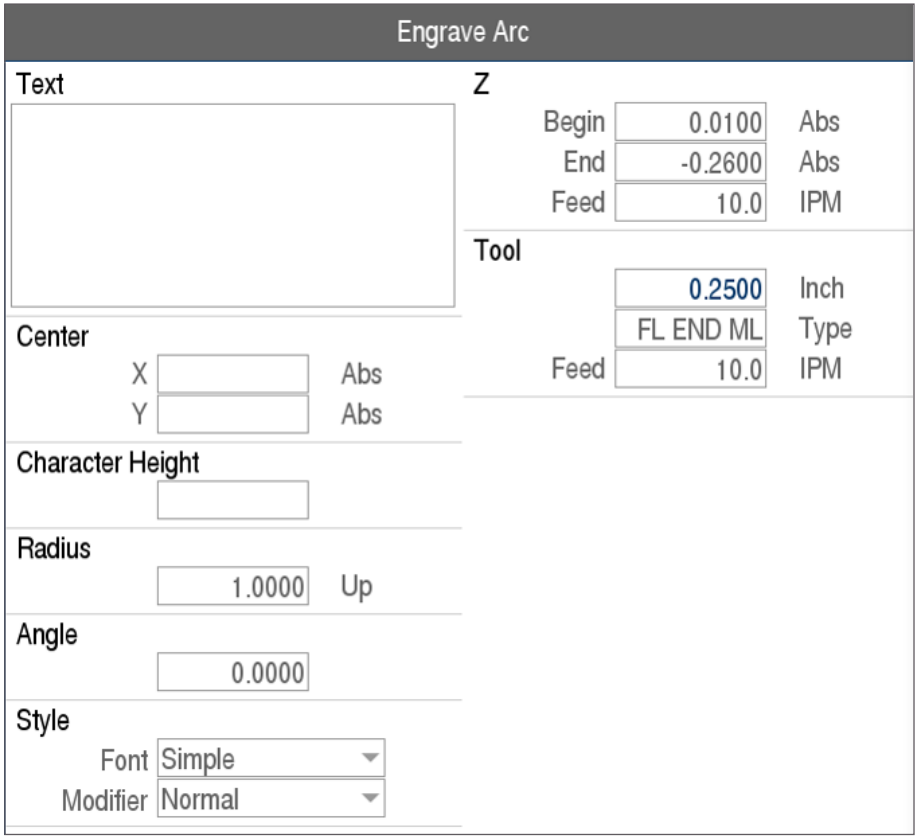
Radius
Up or Down establisher the arc that the text follows.

Z
- **Begin** is where the tool will rapid to.
- From there it will feed at the given rate to the **End** value at the **Feed** provided.
- This feed is for the Z axis only.
- When finished, the tool then rapids up to the retract plane set in datum.

Style
Select Simple Stick or Stencil.

Modifier
Select Normal or Mirrored.

Tool
Feed is the rate of feed in the X, Y plane.



Font Options

Notes

Program Steps

Comment

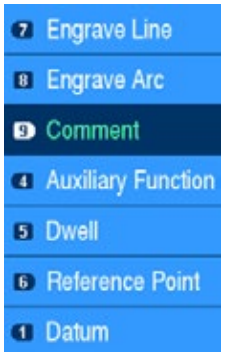
Allows you to keep setup notes, and prompt operators with specific instructions.

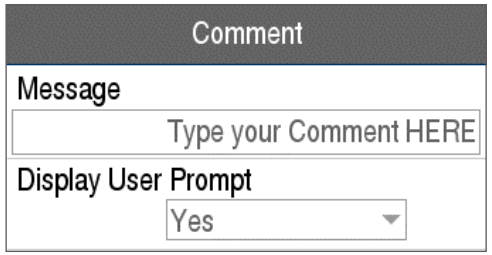
Message

The text you wish to keep as a reminder or prompt during program execution.

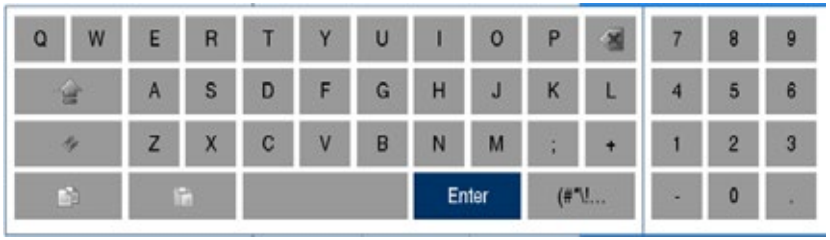
Display User Prompt

- When set to NO is skipped over during program execution.
- When set to YES, the message appears during program execution when the comment step is reached. The control then pauses until GO is pressed to resume program execution.





Alphanumeric Keyboard

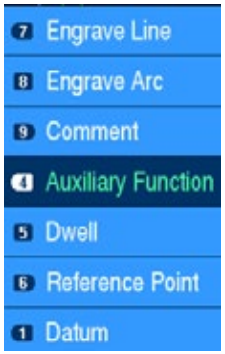


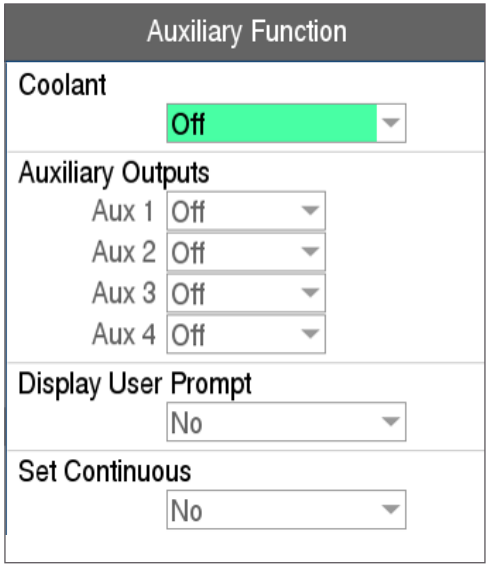
Notes

Program Steps

Auxiliary Function

When the optional Auxiliary Machine Interface (AMI) is connected to your MILLPWRG2, there are two coolant outputs and four auxiliary outputs available.





Coolant

Allows you to choose Off, Flood, or Mist. Coolant selection does not pause the program. Coolant is shut off upon the end of a program automatically.

Auxiliary Outputs

Allows you to signal one or more auxiliary devices such as an indexing table or other equipment. The output can be set to off, on, and pulsed for a specified time.

Display User Prompt

- Provides additional control by establishing timing sequences for the output activation.
- **No:** User prompt allows the output to turn on, off, or pulse without a pause or additional input. It is best when only coolant is being activated.
- **Before Step:** Causes the control to pause and prompt the operator to press GO. When GO is pressed, the functions execute as programmed.
- **After Step:** Will first execute the commanded functions then pause for a GO signal. The GO button, the remote stop go button, or a signal on the remote GO input of the AMI will tell G2 to resume. A programmable indexer connected to the remote go input can signal G2 to resume after it performs its operation.

Set Continuous

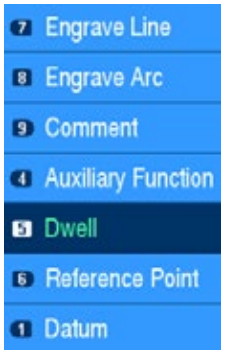
Tells G2 to perform this auxiliary function command without retracting or pausing the program, allowing you to turn on and off outputs 1 through 4 while machining within a contour.

Notes

Program Steps

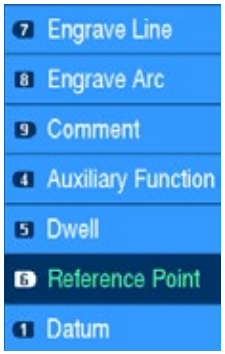
Dwell

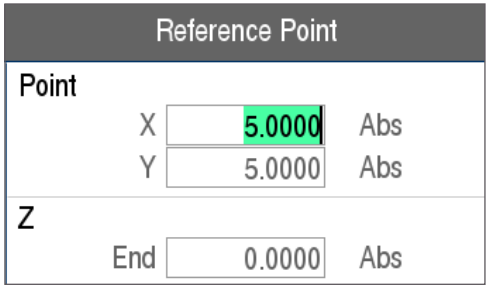
Allows you to program a pause in the program for as little as 0.1 seconds. A Dwell of 0 will cause the program to Pause until the operator presses GO. This dwell being independent of other cycles will occur at the programmed retract plane.



Reference Point

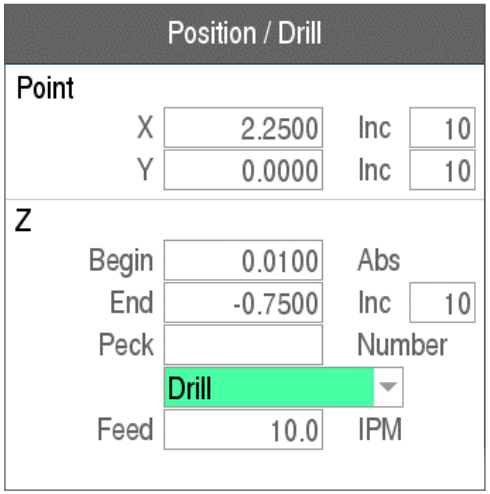
Can be used to graphically indicate center points, tangent points and other features. They can also be used as an external reference when programming incrementally. By programming several items incrementally from a reference point, the features can be relocated later as needed by simply changing the reference point position. A reference point in a continuous tool path will break the path, otherwise, reference points do not cause a pause in your program.





- **X Y:** Define the location of the reference point. This can be cartesian or polar, absolute (ABS) or incremental (INC) from an external reference point.
- **Z:** End is an offset from Z datum from which you can increment Z depth using incremental with an external reference.

The Position Drill example to the right shows X incrementing 2.2500” from step 12 and 0.0000” incrementally in Y, while Z is programmed to increment -0.7500” inches from the Z end of step 12. Any step containing X, Y, and Z values including a reference point can be used as an external reference when using incremental programming. Reference points can work like an alternate datum.



PPT SLIDE 215-216

Notes

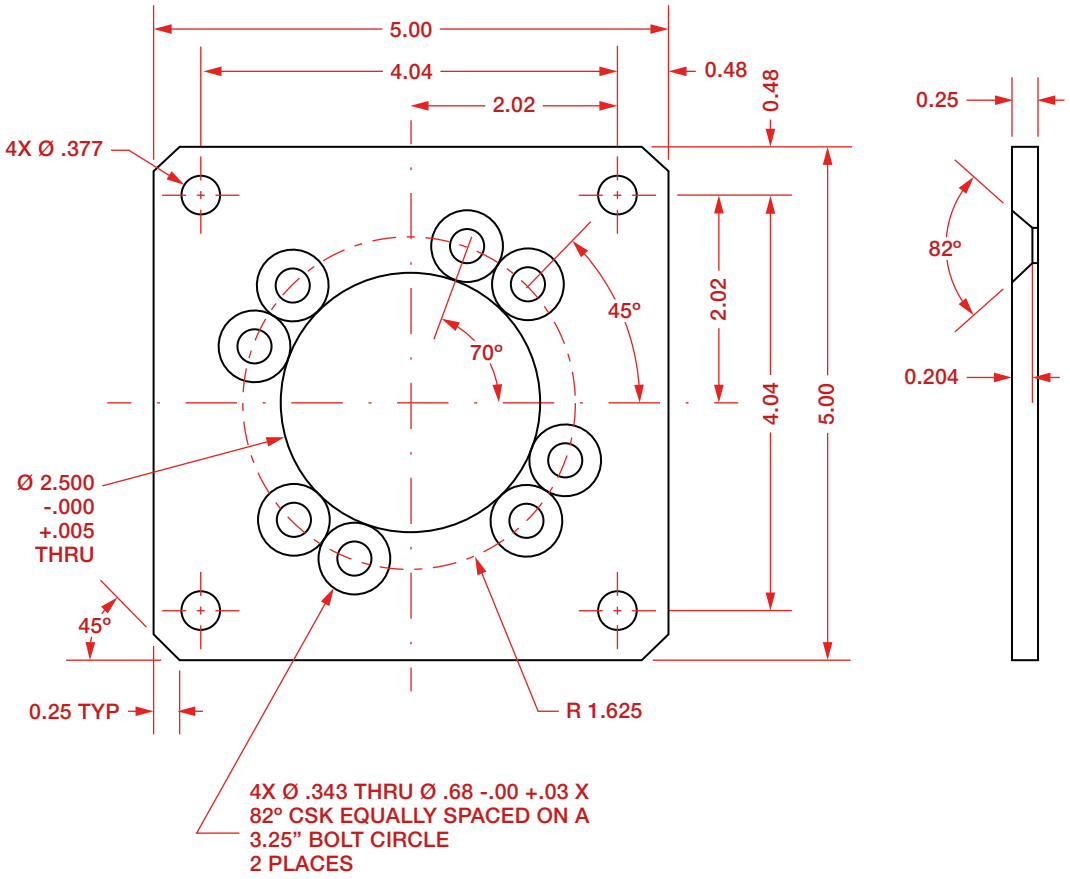
Sample Program

The following exercise will familiarize you with creating a program. A step by step explanation will guide you through programming the tools and the drilling and milling functions. The basic principals described here apply to all of the program functions in MILL**PWR**^{G2}.

Locating a Datum

The datum you choose may be a location indicated on the print or a location you choose as a matter of convenience. Choose a location where the majority of the dimensions refer. This will streamline data entry and reduce the amount of calculations needed. Although there is no datum identified on this print, you can see that the center of the part will simplify programming for the majority of the steps. Therefore we will use the center of the part as datum. This program could also be programmed from datum such as the upper left corner, the choice is yours to consider. Another consideration with every program is the order of operation. In this example we will pilot drill all holes, drill the through holes, then counter bore.

With the proper fixture plate (right), we can then place screws in two or three holes to hold down the part allowing the outer profile and inner bore to be machined without having to move clamps.



PPT SLIDE 217-219

Notes

Sample Program

DRO PGM

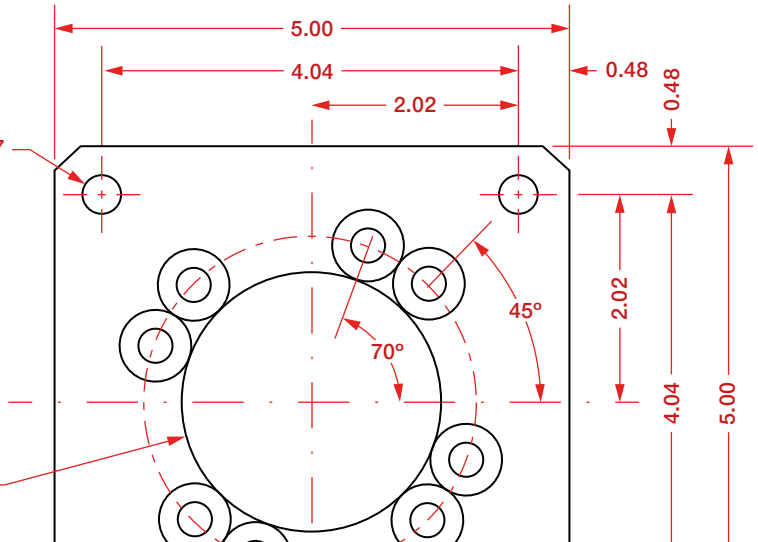
From the DRO press the DRO/PGM key to open the programming screen. Be sure INCH is displayed at the top of the screen before you begin programming.

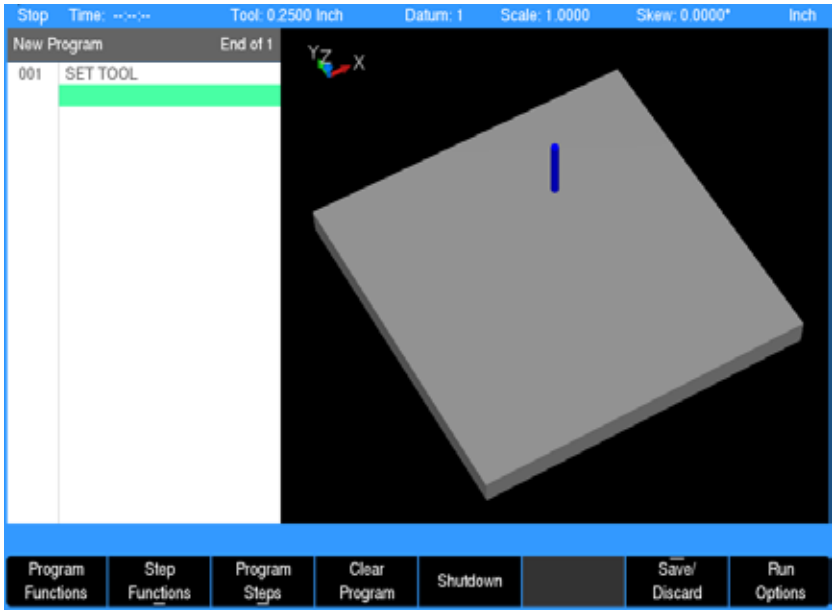
The first step for this program is to establish the tool. We will use a #3 pilot drill.

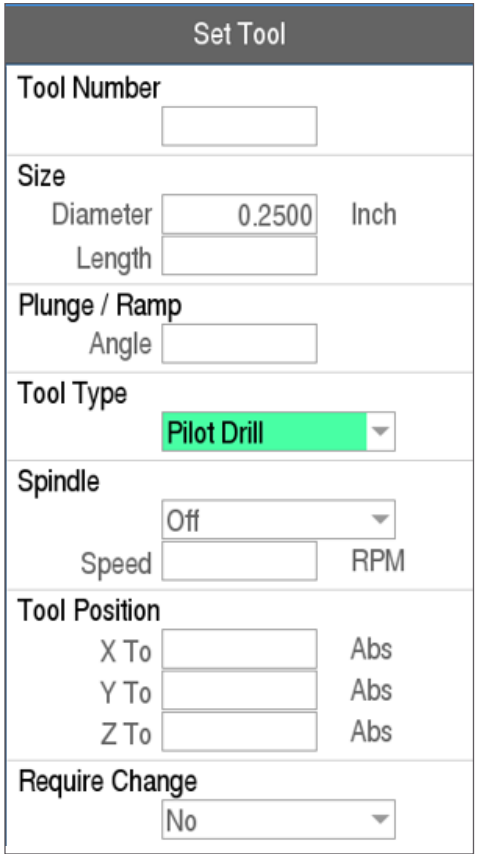
To choose a tool already entered in your tool library, simply enter the number in the Tool Number field and press USE. For this tool we will enter the diameter and type only. The length in unknown at this time and not necessary. We will establish Z-Zero when the tool is mounted and ready to drill.

TOOL

- Press the TOOL key.
- Enter the diameter of the drill 0.250".
- Arrow down to the tool type and press enter.
- Select Pilot Drill and press USE.



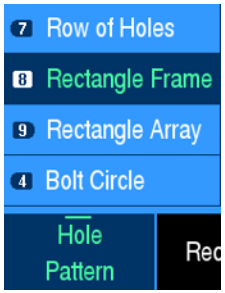


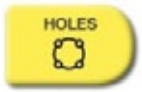


PPT SLIDE 220-221

Notes

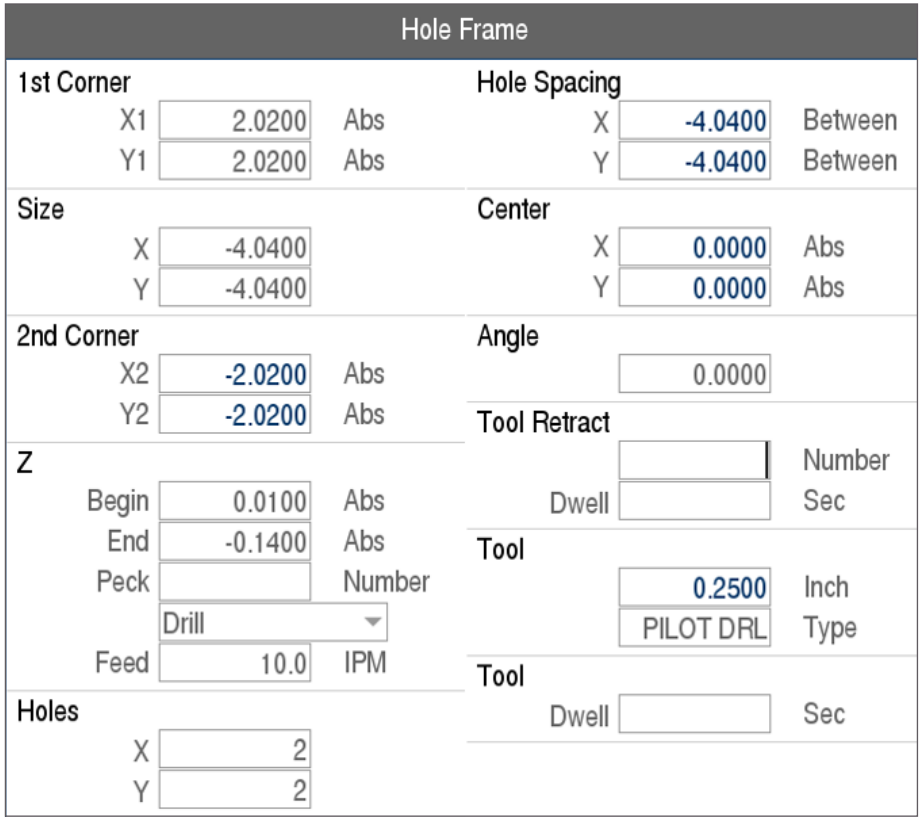
The four outer holes can be described as a rectangle frame pattern.

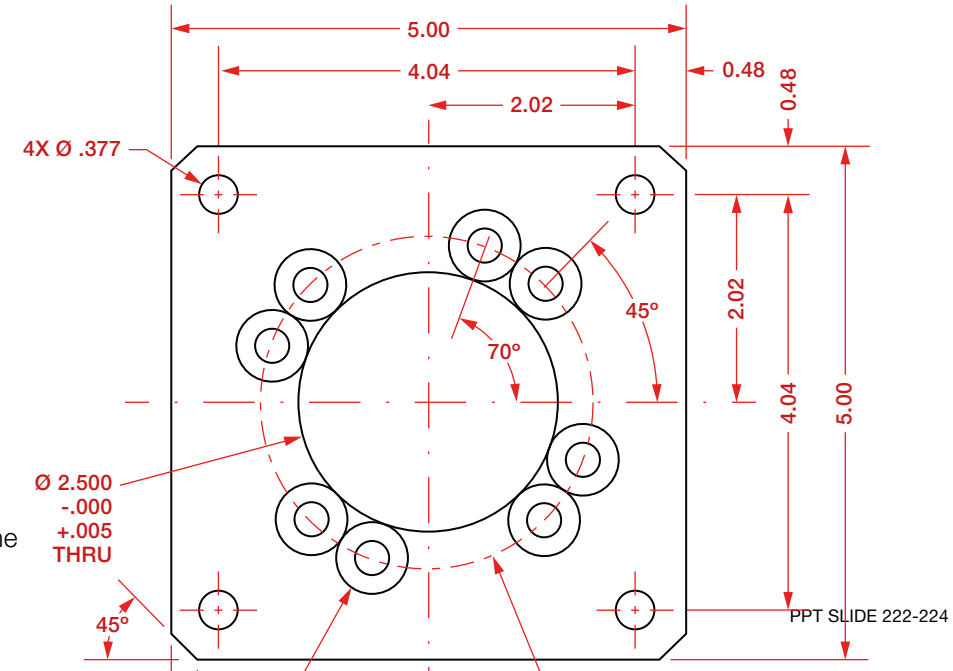




- Press HOLES then arrow down to Rectangle Frame or press 8.
- Enter 2.02 for the 1st Corner X1 location then arrow down or press ENTER and enter 2.02 for the Y1 location.
- Arrow down to Size and enter -4.04 for X and Y. Use negative values because the pattern falls to the left and below the first corner.
- We will establish the top of the part as Z zero so arrow down to Z Begin and enter .01.
- Arrow down to Z End and enter -.14. We will use the feed rate of 10.
- No pecking is required so leave this blank.
- Arrow down to Holes and enter 2 for both X and Y. You will notice the 2nd corner, Hole Spacing, and Center have been calculated and shown in blue.

The four outer holes will appear in the graphics and the block form will adjust to fit.



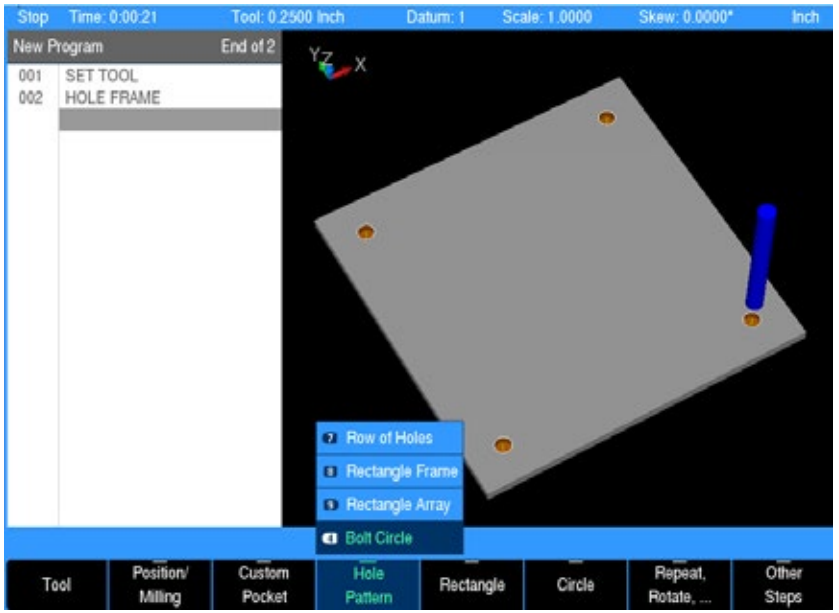


PPT SLIDE 222-224

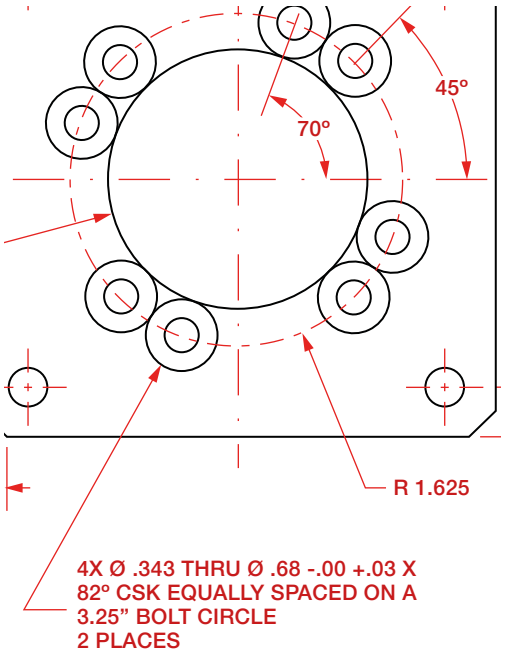
Notes

The next step is to program the first of the two hole patterns that can be described as a bolt hole circle.

- Press HOLES then arrow down to Bolt Circle or press 4.
- The center location is at X and Y Zero and the Z begin and End have carried over from the previous step.
- Arrow down to the radius.
- Enter 1.625 for the radius.
- Since this is a full circle the direction is not important.
- Arrow down to Holes and enter 4.
- Arrow down to the Start angle and enter 45. The end angle is automatically calculated.
- Press USE.



The first set of four holes will appear in the graphics.



Bolt Circle

Center			**End Angle**		
X	0.0000	Abs		315.0000	
Y	0.0000	Abs	**Tool Retract**		
Z					Number
Begin	0.0100	Abs	Dwell		Sec
End	-0.1400	Abs	**Tool**		
Peck		Number		0.2500	Inch
	Drill			PILOT DRL	Type
Feed	10.0	IPM	**Tool**		
Radius			Dwell		Sec
	1.6250				
Direction					
	CCW				
Holes					
	4				
Start Angle					
	45.0000				

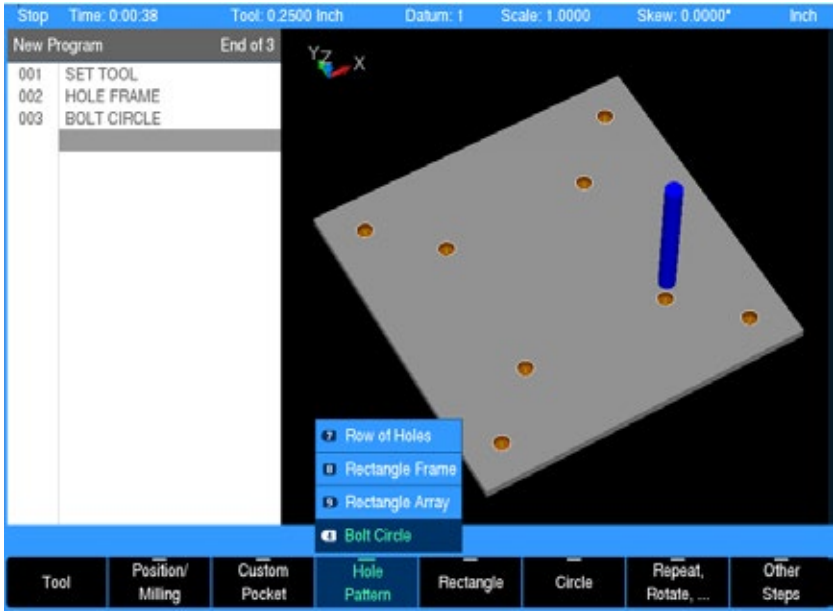
PPT SLIDE 225-227

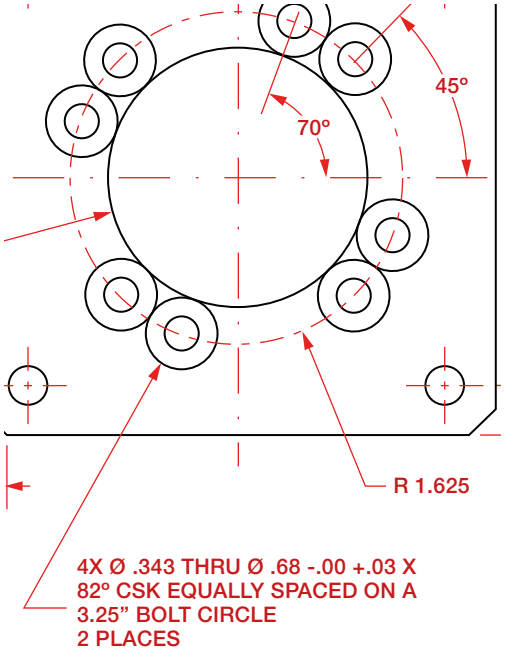
Notes

The next step is to program the second set of the two hole patterns that can be described as a bolt hole circle.

- Press HOLES then arrow down to Bolt Circle or press 4.
- All of the previous entries are carried forward.
- Arrow down to the Start angle and enter 70. The end angle is automatically calculated.
- Press USE.

The second bolt hole pattern will appear in the graphics.





Bolt Circle

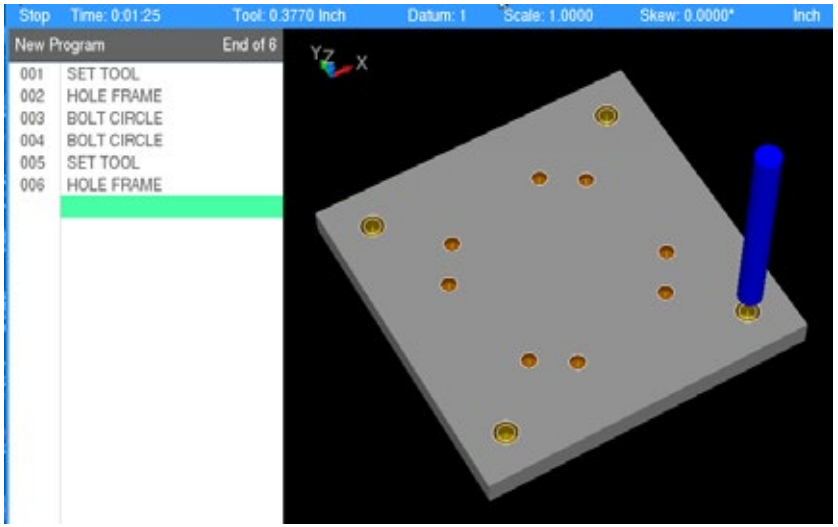
Center			**End Angle**	340.0000	
X	0.0000	Abs	**Tool Retract**		
Y	0.0000	Abs			Number
Z			Dwell		Sec
Begin	0.0100	Abs	**Tool**		
End	-0.1400	Abs		0.2500	Inch
Peck		Number		PILOT DRL	Type
	Drill		**Tool**		
Feed	10.0	IPM	Dwell		Sec
Radius					
	1.6250				
Direction					
	CCW				
Holes					
	4				
Start Angle					
	70.0000				

PPT SLIDE 228-230

Notes

The next step is to change the tool to a letter V drill (0.377") and program the frame set of holes.

- Press the TOOL key.
- Enter the diameter of the drill 0.377".
- Arrow down to the tool type and press enter.
- Select Drill and press USE.
- Press HOLES then arrow down to Rectangle Frame or press 8.
- All of the previous entries from the first frame of holes are carried forward.
- Arrow down to the Z End and change it to -0.400" to insure you drill through.
- Press USE.



Set Tool

Tool Number		
Size		
Diameter	0.3770	Inch
Length		
Plunge / Ramp		
Angle		
Tool Type		
	Drill	

Hole Frame

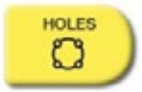
1st Corner			**Hole Spacing**		
X1	2.0200	Abs	X	-4.0400	Between
Y1	2.0200	Abs	Y	-4.0400	Between
Size			**Center**		
X	-4.0400		X	0.0000	Abs
Y	-4.0400		Y	0.0000	Abs
2nd Corner			**Angle**		
X2	-2.0200	Abs		0.0000	
Y2	-2.0200	Abs	**Tool Retract**		
Z					Number
Begin	0.0100	Abs	Dwell		Sec
End	-0.4000	Abs	**Tool**		
Peck		Number		0.3770	Inch
	Drill			DRILL	Type
Feed	10.0	IPM	**Tool**		
Holes			Dwell		Sec
X	2				

The second frame of holes will appear in the graphics.

Notes

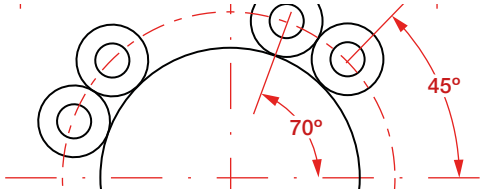
The next step is to change the tool to an 11/32” drill and program the two bolt hole patterns.

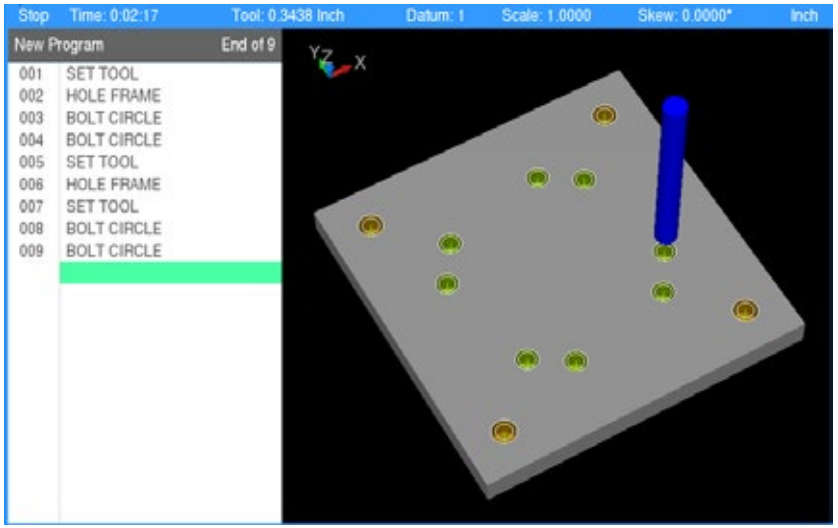
- Press the TOOL key.
- Enter the diameter of the drill 0.343”.
- Arrow down to the tool type and press enter. Select Drill and press USE.



- Press HOLES then arrow down to Bolt Circle or press 4.
- The previous entries from the Bolt Circle step are carried forward along with the last Z depth used.
- Arrow down to the Start Angle and enter 45. The end angle is automatically calculated.
- Press USE.
- Create another hole pattern with a 70 degree start angle.

The two bolt hole patterns will appear in the graphics.





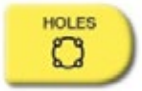
Center		
X	0.0000	Abs
Y	0.0000	Abs
Z		
Begin	0.0100	Abs
End	-0.4000	Abs
Peck		Number
	Drill	
Feed	10.0	IPM
Radius	1.6250	
Direction	CCW	
Holes	4	
Start Angle	45.0000	

Center		
X	0.0000	Abs
Y	0.0000	Abs
Z		
Begin	0.0100	Abs
End	-0.4000	Abs
Peck		Number
	Drill	
Feed	10.0	IPM
Radius	1.6250	
Direction	CCW	
Holes	4	
Start Angle	70.0000	

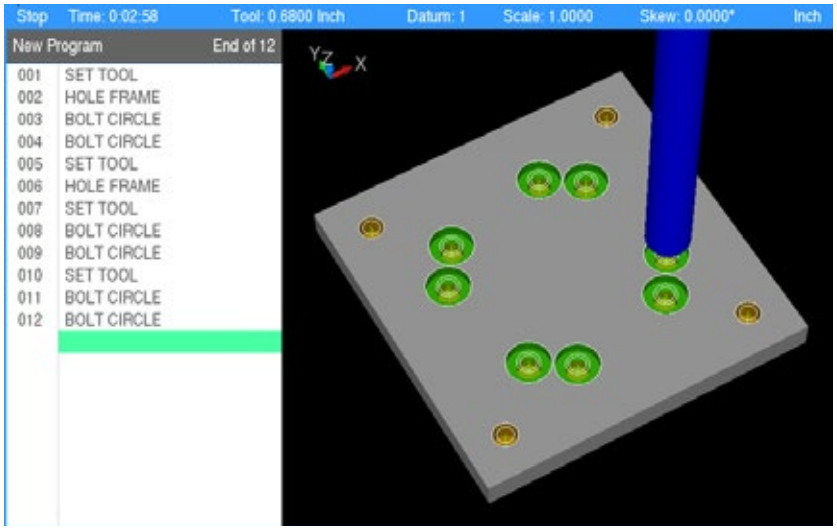
Notes

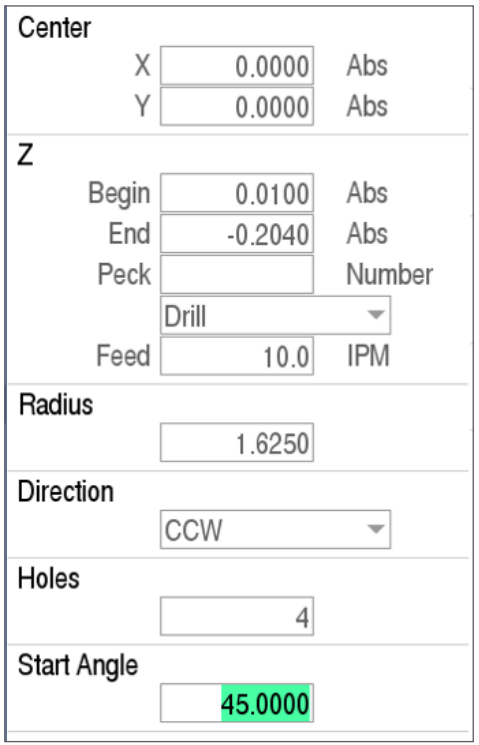
The next step is to change the tool to a counter sink drill and program the two bolt hole patterns again.

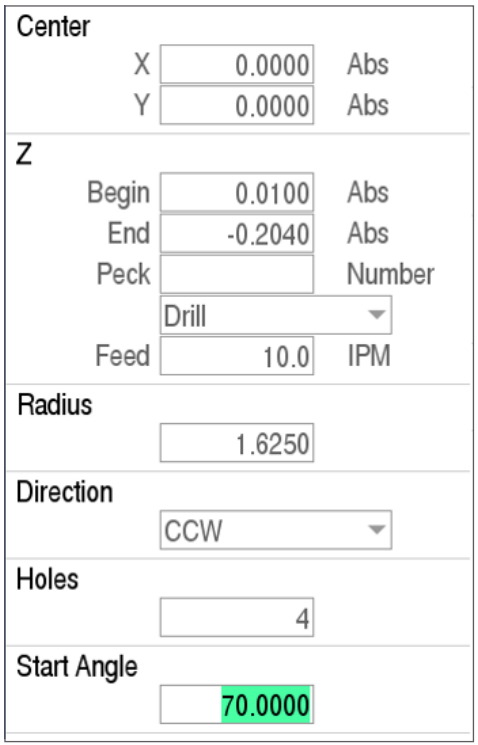
- Press the TOOL key.
- Enter a diameter of 0.680”.
- Arrow down to the tool type and press enter. Select Counter-Sink and press USE.



- Press HOLES then arrow down to Bolt Circle or press 4.
- The previous entries from the bolt Circle step are carried forward along with the last Z depth used.
- Arrow down to the Z Depth and change it to -0.204”.
- Arrow down to the Start Angle and enter 45.
- Press USE.
- Create another hole pattern with a 70 degree start angle.



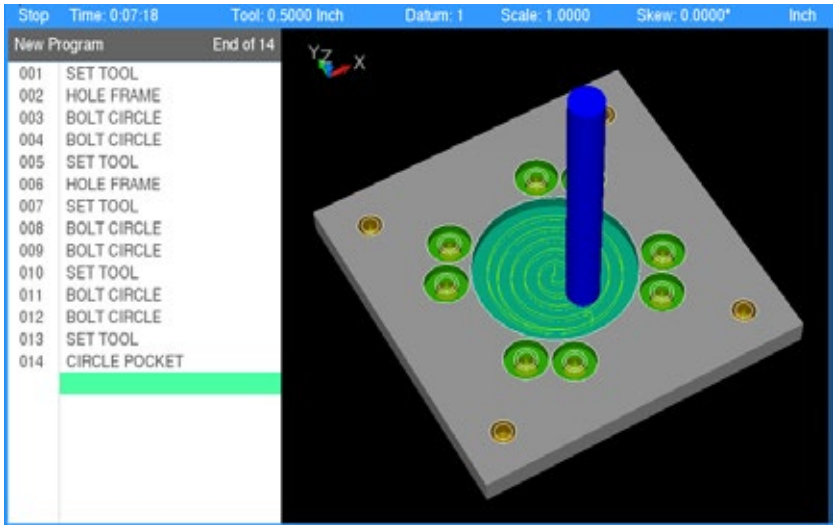


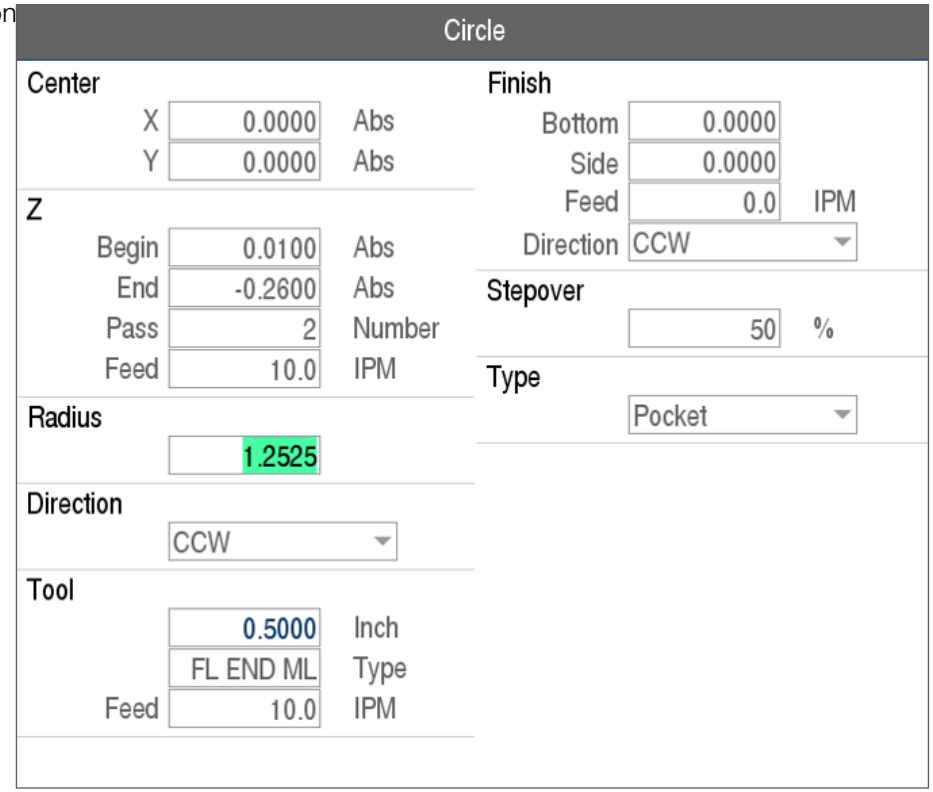


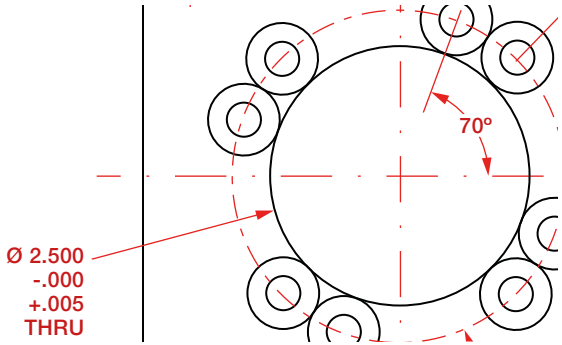
Notes

With the holes completed two or more bolts can be used to hold the work piece by screwing them down in tapped holes strategically placed in the fixture below. Now the center hole and outer profile can be machined without interference from clamps.

- Press the TOOL key.
- Enter a diameter of 0.500”.
- Arrow down to the tool type and press enter.
- Select Flat Endmill and press USE.
- Press CIRCLE then press enter for pocket.
- The previous entries from the Bolt Circle step are carried forward alon with the last Z depth used.
- Arrow down to the Z Depth and change it to -0.260”.
- Arrow down to Pass and enter 2.
- Arrow down to radius and enter 1.2525
- Press USE.







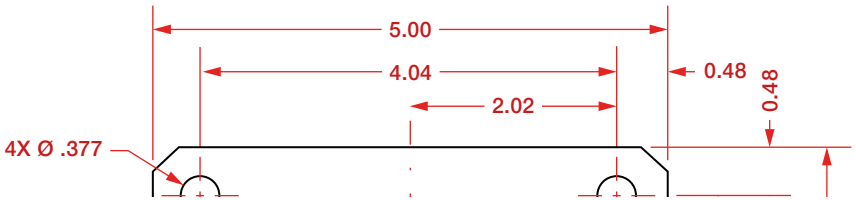
Notes

Rectangle Frame is the best option for the outer profile. Because the print clearly indicates the size of the rectangle and because we established Datum as the center we will use this information to program this frame. This outer profile could also be described using lines to describe the shape. If you are, for example, converting a DXF file then this would be the best solution.

- Press the Rectangle key.
- Choose Frame from the list by highlighting Frame and pressing Enter or by pressing the 8 key.
- Arrow down to the size and enter 5 for both X and Y.
- Review the Z information and insure it is the same as the previous circle pocket.
- Arrow down to Corner and select Chamfer and enter 0.25.
- Arrow down to Offset and select Outside.
- Arrow down to Center and enter zero for both X and Y.
- Press USE.

Rectangle Frame

1st Corner			**Tool**		
X1	-2.5000	Abs		0.5000	Inch
Y1	-2.5000	Abs		FL END ML	Type
Size			Offset	Outside	
X	5.0000		Feed	10.0	IPM
Y	5.0000		**Center**		
2nd Corner			X	0.0000	Abs
X2	2.5000	Abs	Y	0.0000	Abs
Y2	2.5000	Abs	**Angle**		
Z				0.0000	
Begin	0.0100	Abs	**Finish**		
End	-0.2600	Abs		0.0000	Cut
Pass	2	Number	Feed	0.0	IPM
Feed	10.0	IPM	Direction	CCW	
Corner					
Chamfer	0.2500	Length			
Direction					
	CCW				



Notes

The graphics will show a complete part and the block form for the graphics automatically sizes to include maximum depth, width, and length of the program. You can adjust the block form if you wish to better represent the stock size of your work piece.

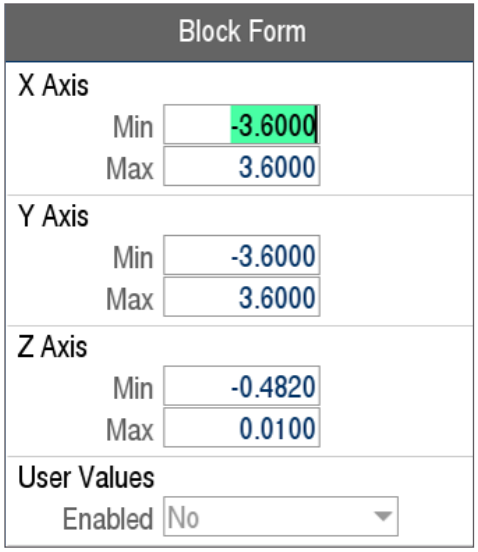
VIEW

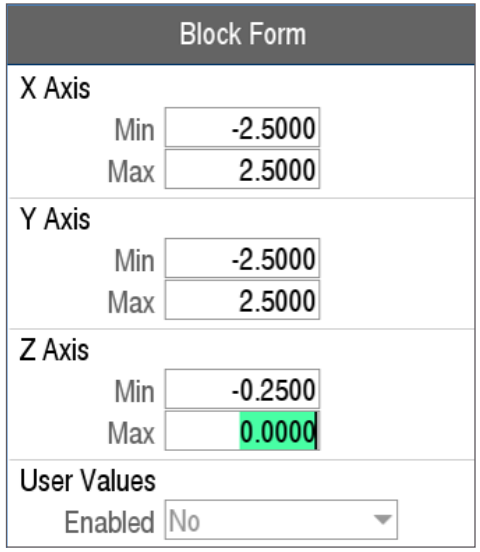
Block Form

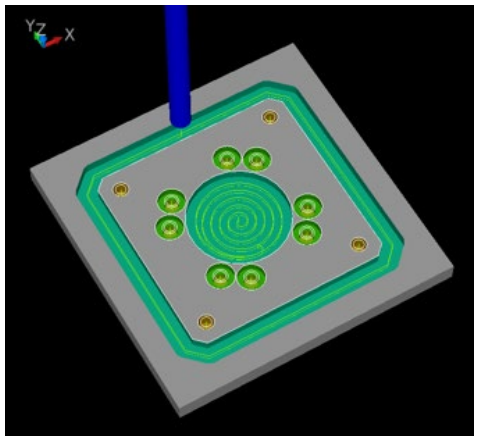
- Press the view Key then press the Block Form Soft Key. The form opens and you will find all the values are blue indicating calculated values.
- Enter the values that best describe the stock you will be using.
- Press USE.
- Press Cancel to exit view mode.

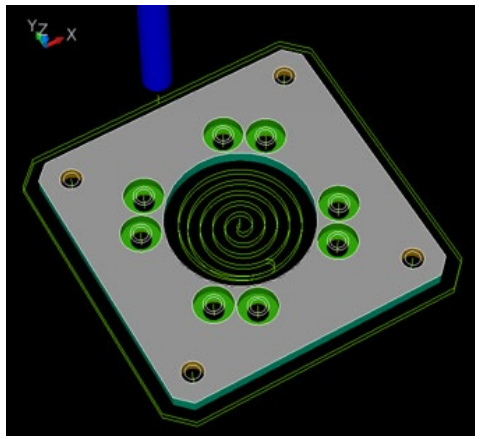
Calculated values are blue while entered values are black. Calculated values are best while you build your program because the view will automatically scale down as you add steps. You can have a mix of calculated and entered values.

Connecting a mouse to one of the USB ports allows you to left click and drag in the graphics to rotate the part and right click and drag to pan.



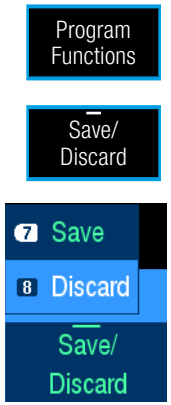






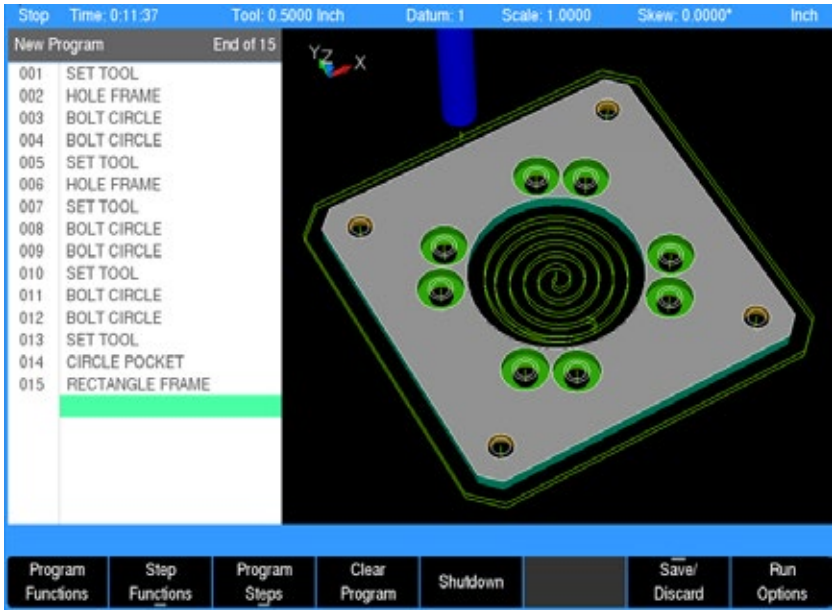
Notes

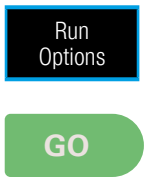
The program is now complete. At this point it is a good idea to save the program. An estimated run time is calculated in the top status bar. This does not include time for tool changes. It is highly recommended that you verify the program prior to cutting. You can preview the cutting operation graphically by selecting Graphics Only found under the Run Options soft key.



To save your work

- Press the Program Functions soft key.
- Press the Save / Discard soft key.
- Press Enter to save the program.
- Name the program “Example Program G2” and press USE.





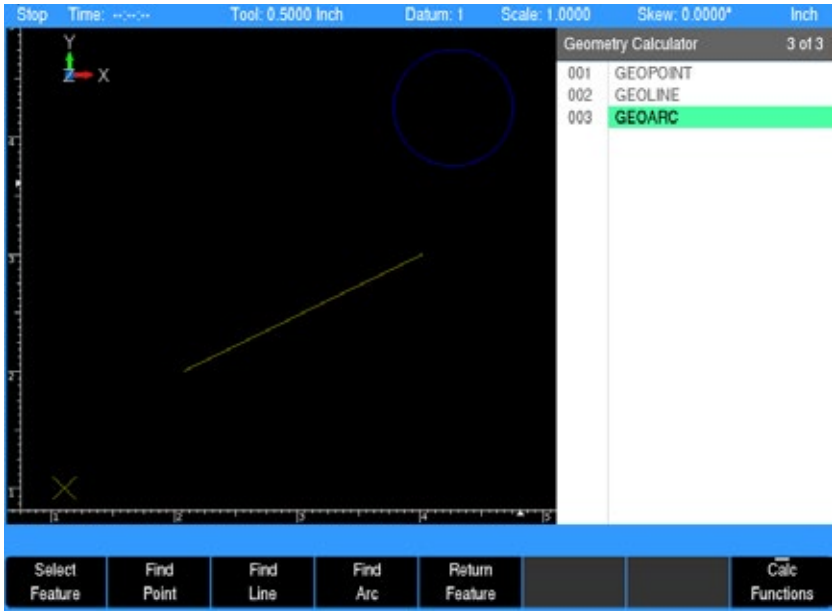
To proof the program

- Press the Run Options soft key and select Dry Run and Graphics Only.
- Insure you are on step 1 and press GO. When you are satisfied with your program insure Dry Run and Graphics Only are off and begin first piece machining.

Notes

Geometry Calculator

The geometry calculator's layout is similar to the program mode except the step list is on the right and the graphic window is on the left. This calculator is capable of creating points, lines, and arcs, relative to features you select. The capabilities provided here can allow you to program features when very little dimensional data is known. The sample program we will use is challenging to the most seasoned programmer. You will see that with the Geometry Calculator a challenge such as this is easily overcome.



Access the Geometry Calculator by pressing the CALC key, then the Geometry soft key when in the program edit screen. It is not accessible from the DRO.

Once in the geometry calculator you can press the Calc Functions key or simply begin creating geometry. When you press the Calc Functions soft key you can use the following functions:

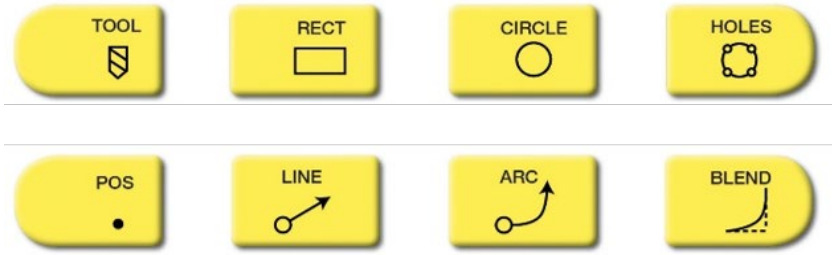
- **Clear Calculator:** Clears any geometry currently entered and displayed.
- **Save Calculator:** Saves your work allowing you to reload it if cleared.
- **Load Calculator:** Loads any saved items from above.
- **Load Program:** Loads your current program converting it to geometric steps. This saves time since you do not have to reenter work you already accomplished.

You can select these options using the arrow keys and enter key or press the corresponding number on the keypad that is displayed to the right of each option.

Notes

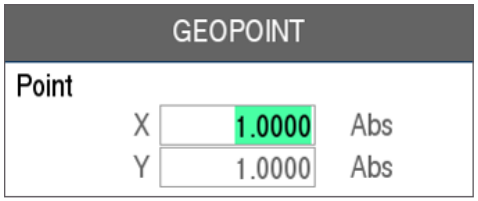
PPT SLIDE 247-249

The geometry calculator uses positions lines and arcs as building blocks. With one or two of these items selected you can then find additional positions, lines, and arcs relative the those selected. Once found they can be returned to your program. If you entered the geometry calculator while in a from, to, center, or third point in program step you can return a position directly into this step. If you enter the geometry calculator without being in an open program step, you can return a position as a drill position or return an entire line or arc to be used as part of a tool path.

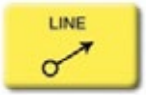


To create a position called a GeoPoint:

- Press the yellow POS key.
- A window will open asking for the X and Y coordinates. Z Axis data is not taken into consideration in the calculator. The Z Axis information, offsets, and feeds will be carried forward from the previous step as you return information to your program.
- Teach position allows you to transfer the current machine position to the GeoPoint and Polar allows you to define the position using a radius and angle from Datum zero.
- Enter the X and Y coordinates and press USE to complete.

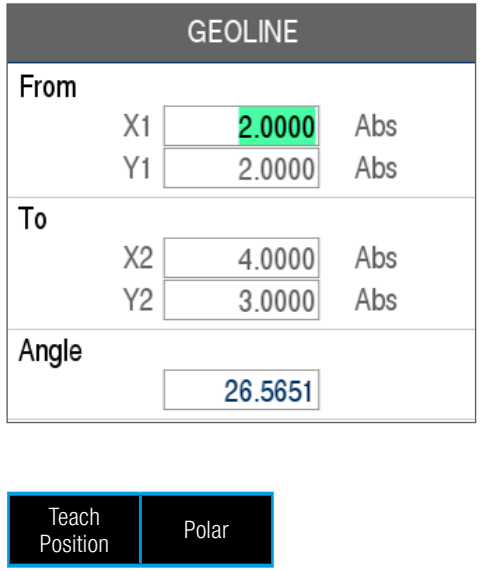


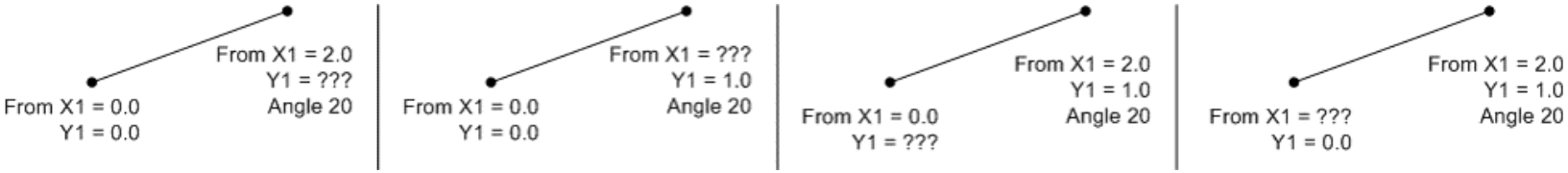
Notes



To create a GeoLine:

- Press the yellow Line key. Enter the information you have to define the line. It can be defined the same as a line when programming.
- **From:** X1 Y1 define the start point of the line. These can be cartesian or polar, absolute only.
- **To:** X2 Y2 is the end of the line and can be cartesian or polar, absolute only.
- **Angle:** Allows you to create a line by programming to an X or a Y value at this given angle. As you can see to the right, the X2 value has been calculated and appears in blue. By striking a line from X 2.0, Y 2.0 at 20° until Y 3.0 is reached we calculate the X2 value to be 3.7321.

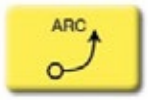




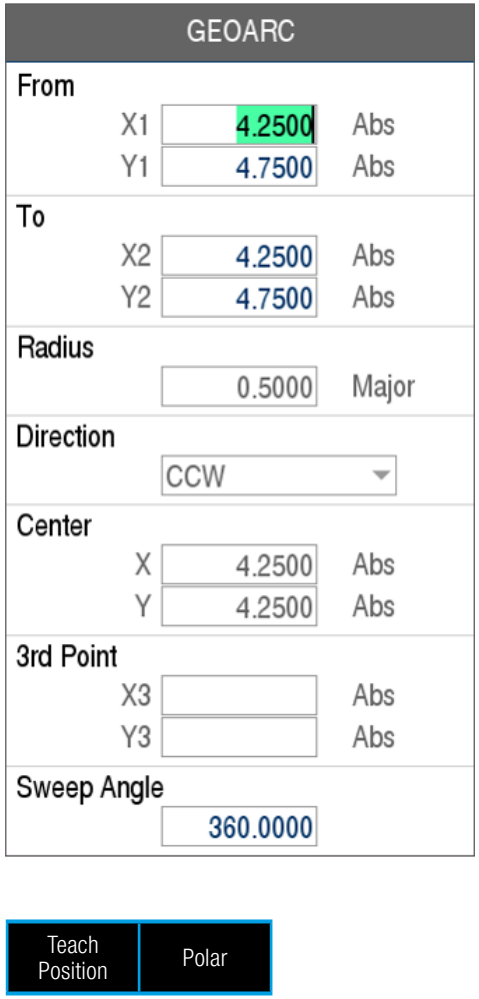
In each of these four examples, the unknown coordinate "???" when left blank will be calculated based on the angle provided.

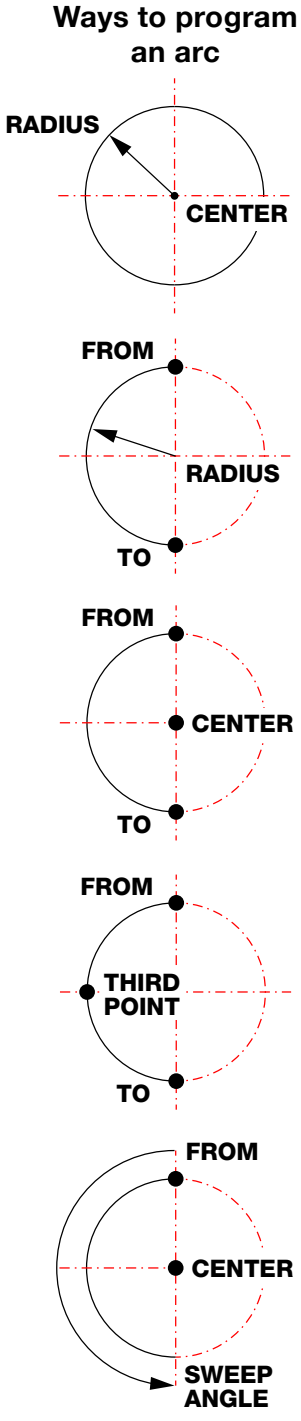
Notes

The ARC key opens the arc dialogue allowing you to define an arc. In the Geometry Calculator, an arc can be programmed using only the center, radius, and direction. This is very useful when your intent is to determine tangent points.



- **From:** X1 Y1 define the start point of the arc. These can be cartesian or polar, absolute only.
- **To:** X2 Y2 is the end of the arc and can be cartesian or polar, absolute only.
- **Radius:** Establishes the size of the arc. Minor or Major can be established. Minor arcs are less than 180 degrees while Major arcs are greater than 180 degrees.
- **Direction:** Establishes clockwise (CW) or counter clockwise (CCW) direction.
- **Center:** Allows you to define the center point of an arc. This is useful if a radius is not defined on your print. Do not enter a radius prior to entering the center point.
- **3rd Point:** Provides you with yes another method of defining an arc. By programming a from, to, and third point, an arc can be determined. The third point does not have to be within the From and To points.
- **Sweep Angle:** An alternative entry for when the from or to point is unknown. If a from point is entered with a radius and a sweep angle, the To point can be calculated.





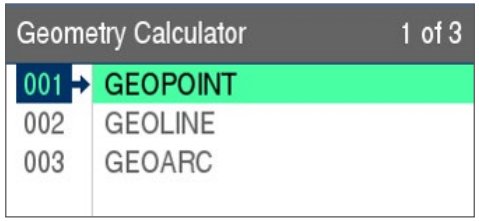
PPT SLIDE 252-255

Notes

After you create geometry or load your program into the calculator you can easily find relative features by selecting one or two items, then pressing the Find Point, Find Line, or Find Arc soft keys.

Select Feature	Find Point	Find Line	Find Arc	Return Feature

The chart to the right illustrates the possibilities depending on the combination of items selected. (A larger version appears on the next page.)



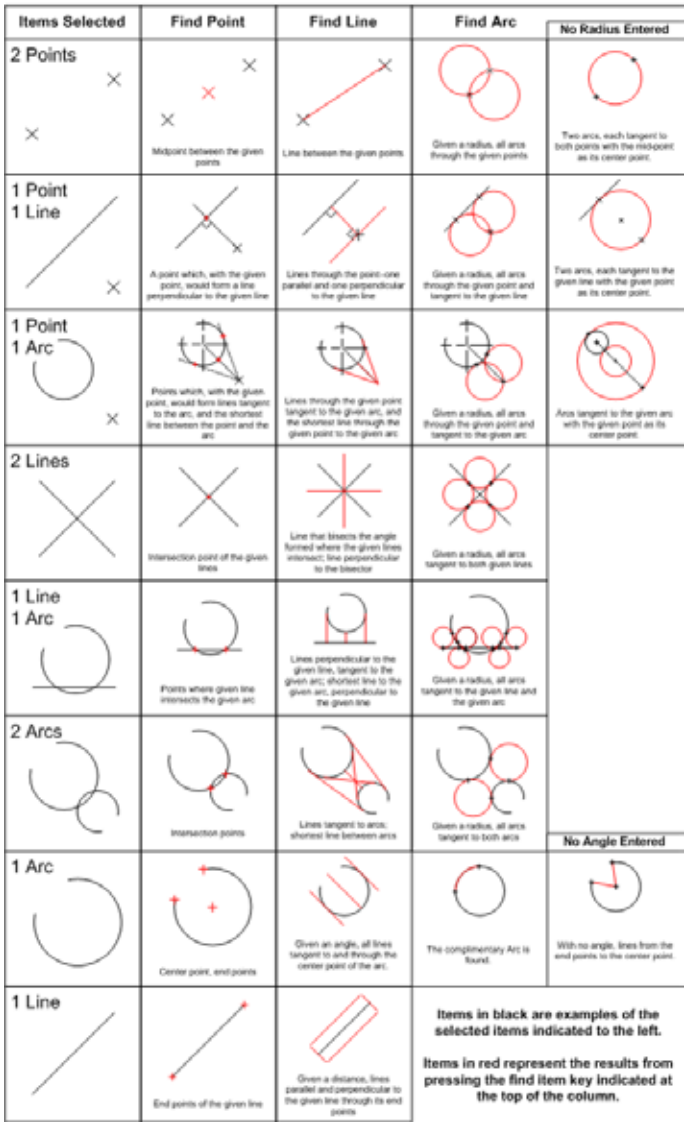
Select Feature

To select an item, highlight the item and press the Select Feature soft key. When selected, an arrow will appear beside the item as shown below. To deselect an item, highlight it again and press the Select Feature soft key. The arrow will disappear indicating the item is no longer selected. Because only one or two items can be selected at a time, when you select a third item, the first item will automatically become deselected. As you become familiar with this calculator this feature will become a great time saver.

Return Feature

The Return Feature soft key will return a highlighted feature to your program. If you entered the calculator while within a program step, the coordinate pertinent to the step is transferred. If you entered the calculator between two program steps or at the end of your program, the entire item will transfer as a separate program step.

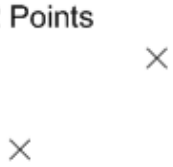
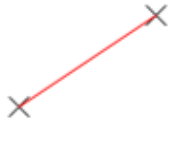
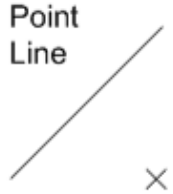
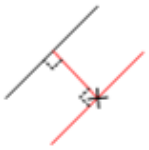
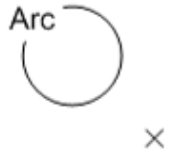
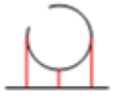
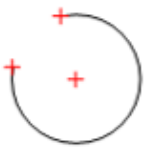
MILLPWR^{G2} Geometry Calculator Find Item Chart



PPT SLIDE 255-257

Notes

MILL**PWR**^{G2} Geometry Calculator Find Item Chart

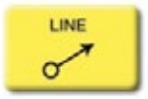
Items Selected	Find Point	Find Line	Find Arc	No Radius Entered
2 Points 	 Midpoint between the given points	 Line between the given points	 Given a radius, all arcs through the given points	 Two arcs, each tangent to both points with the mid-point as its center point.
1 Point 1 Line 	 A point which, with the given point, would form a line perpendicular to the given line	 Lines through the point--one parallel and one perpendicular to the given line	 Given a radius, all arcs through the given point and tangent to the given line	 Two arcs, each tangent to the given line with the given point as its center point.
1 Point 1 Arc 	 Points which, with the given point, would form lines tangent to the arc, and the shortest line between the point and the arc	 Lines through the given point tangent to the given arc, and the shortest line through the given point to the given arc	 Given a radius, all arcs through the given point and tangent to the given arc	 Arcs tangent to the given arc with the given point as its center point.
2 Lines 	 Intersection point of the given lines	 Line that bisects the angle formed where the given lines intersect; line perpendicular to the bisector	 Given a radius, all arcs tangent to both given lines	
1 Line 1 Arc 	 Points where given line intersects the given arc	 Lines perpendicular to the given line, tangent to the given arc; shortest line to the given arc, perpendicular to the given line	 Given a radius, all arcs tangent to the given line and the given arc	
2 Arcs 	 Intersection points	 Lines tangent to arcs; shortest line between arcs	 Given a radius, all arcs tangent to both arcs	
				No Angle Entered
1 Arc 	 Center point, end points	 Given an angle, all lines tangent to and through the center point of the arc.	 The complimentary Arc is found.	 With no angle, lines from the end points to the center point.
1 Line 	 End points of the given line	 Given a distance, lines parallel and perpendicular to the given line through its end points	**Items in black are examples of the selected items indicated to the left.** **Items in red represent the results from pressing the find item key indicated at the top of the column.**	

Geometry Calculator Example Program

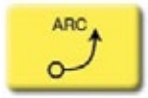
This programming example (below right) would be a challenge on any CNC platform. We will use the Geometry Calculator like you would use a compass and square to easily tackle this challenge. Begin this exercise from the program editing mode.

Begin at the left end of the bottom line and proceed around the part in a counterclockwise direction.

- Define a tool by pressing the TOOL key and program a half inch diameter.
- Press USE.

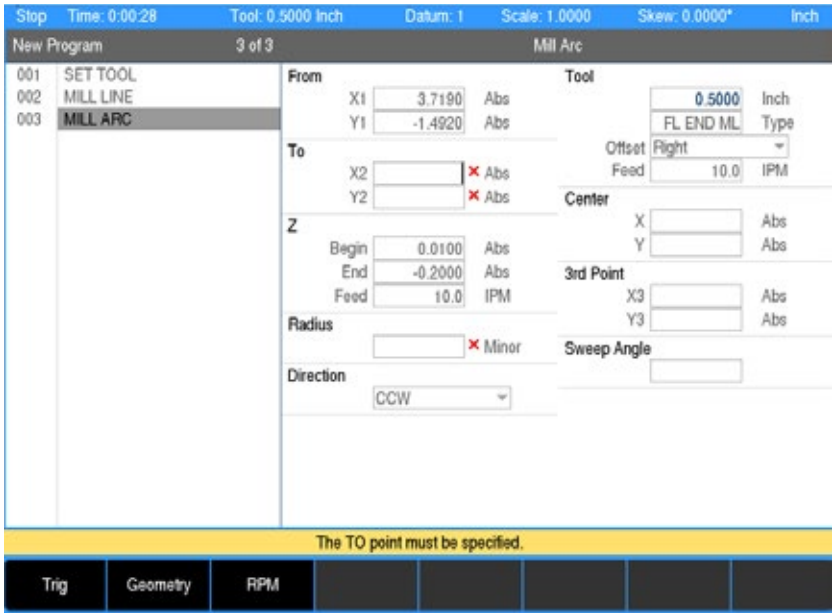


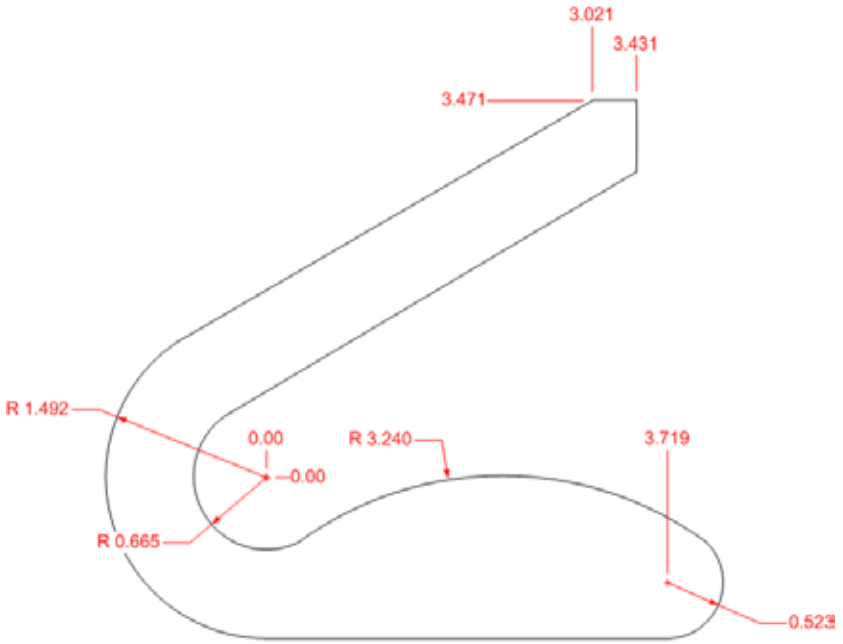
- Next press the LINE button. Program a line from the tangent point of the 1.492 radius to the tangent point of the 0.523 radius. Program the Z Begin at 0.01” and the Z End at -0.2” and a right offset.



- Begin an arc. The From point is carried forward from the line. The to point is not known. With the highlight in the To point, press the Calculator Key then the Geometry soft key.

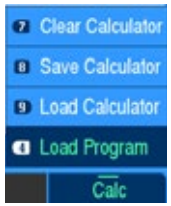
From	To
X = 0.0000	X = 3.7190
Y = -1.4920	Y = -1.4920

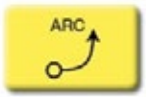


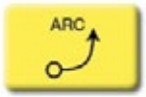


PPT SLIDE 258-259

Notes





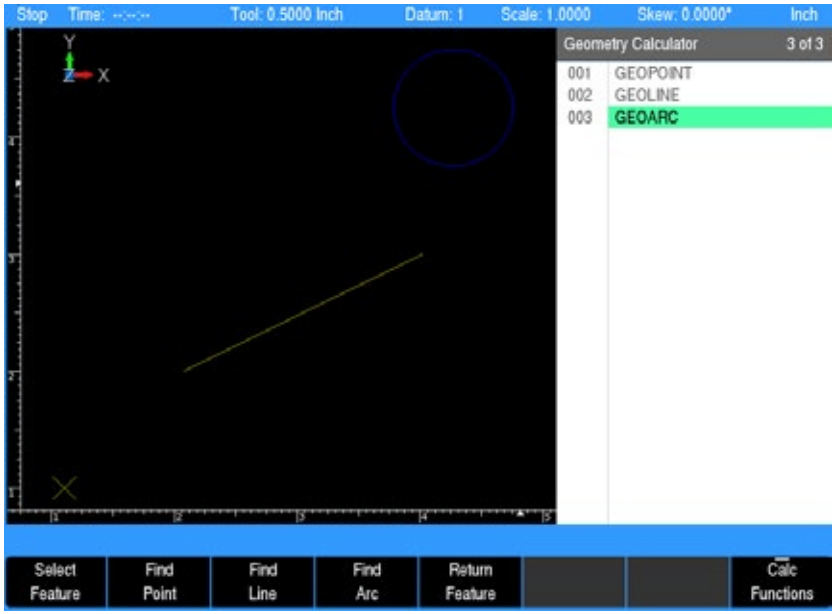


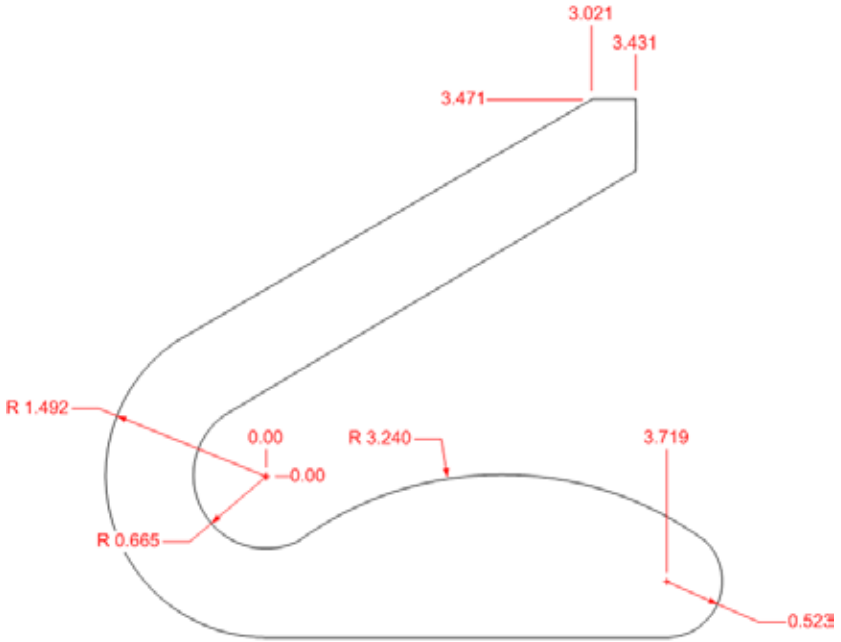
- Once in the Geometry calculator press the Calc Functions soft key and arrow down to Load Program and press Enter or press the 4 Key to bring the Line created in our program into the calculator.
- In the geometry calculator, the order of the geometry in the list is not important. We will begin constructing the geometry we can define.
- Press the Arc key and create an arc describing the inside bend centered on X,Y zero. Press USE.
- Create a second arc describing the right end. You can see that the Y center from zero is 0.523 inches above the -1.490 inch tangent point.

  0.523 - 1.490 = -0.969

- The radius is 0.523. Select CCW and press USE.

Center	Radius
X = 3.7190	R = 0.5230
Y = -0.9690	Dir = CCW



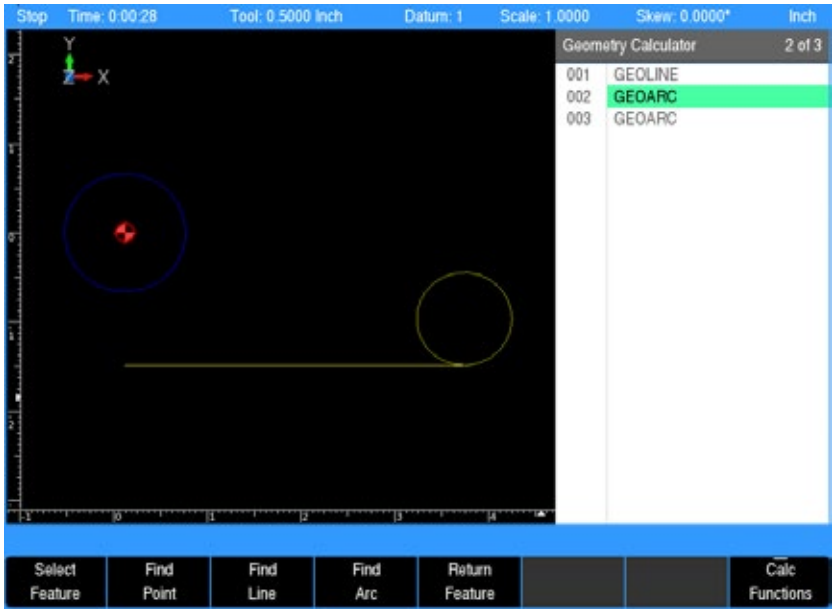


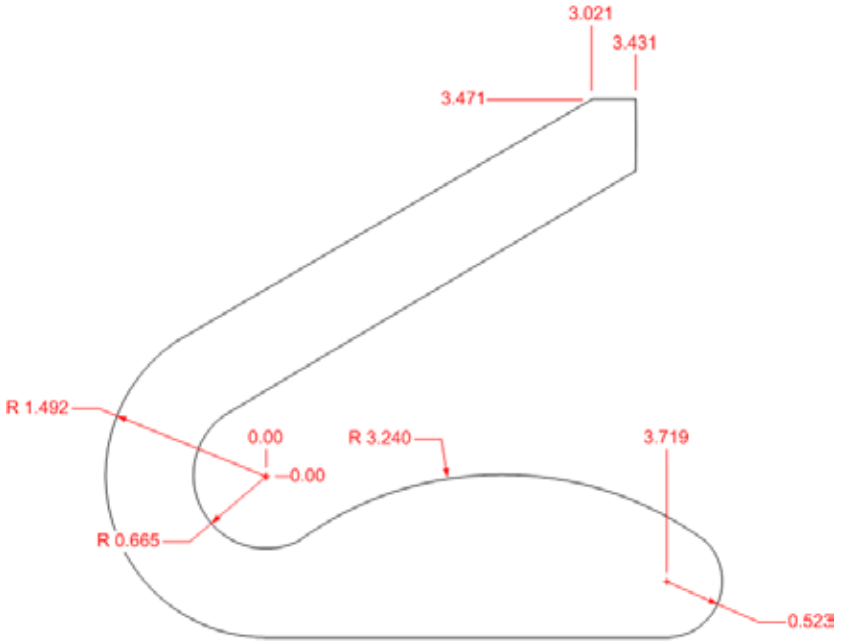
PPT SLIDE 260

Notes

Select Feature

- Select each of the two arcs you created by highlighting them and pressing the Select Feature soft key.



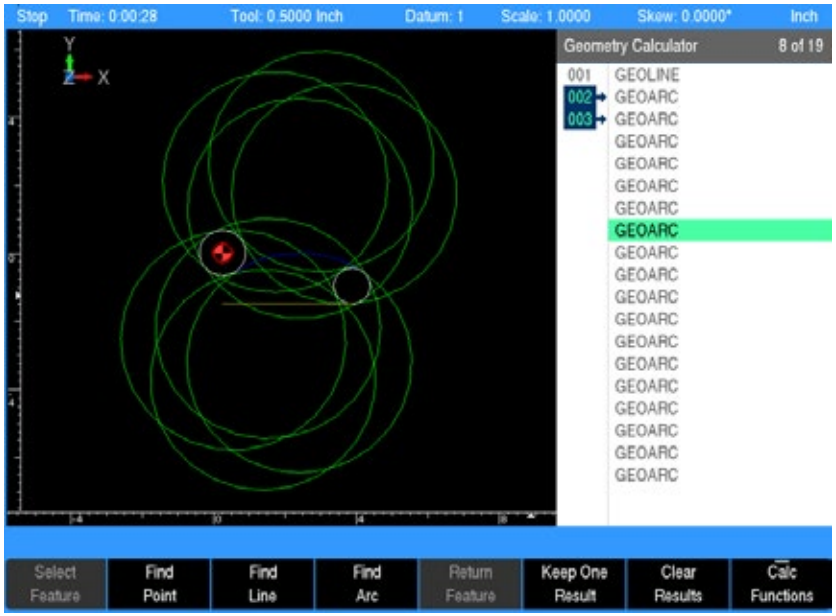


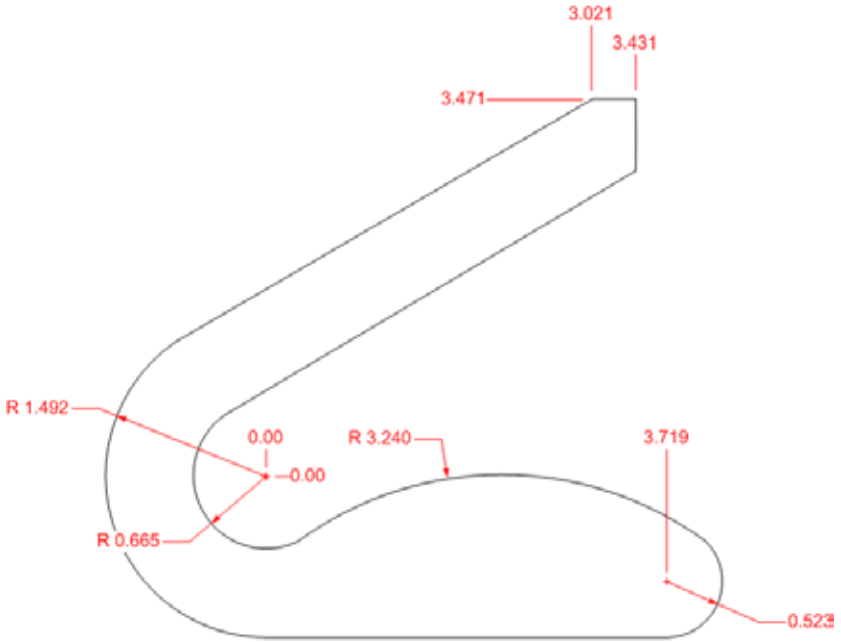
PPT SLIDE 261

Notes

Find Arc

- Press the Find Arc soft key and enter a radius of 3.240. All possible variations of this arc will appear.





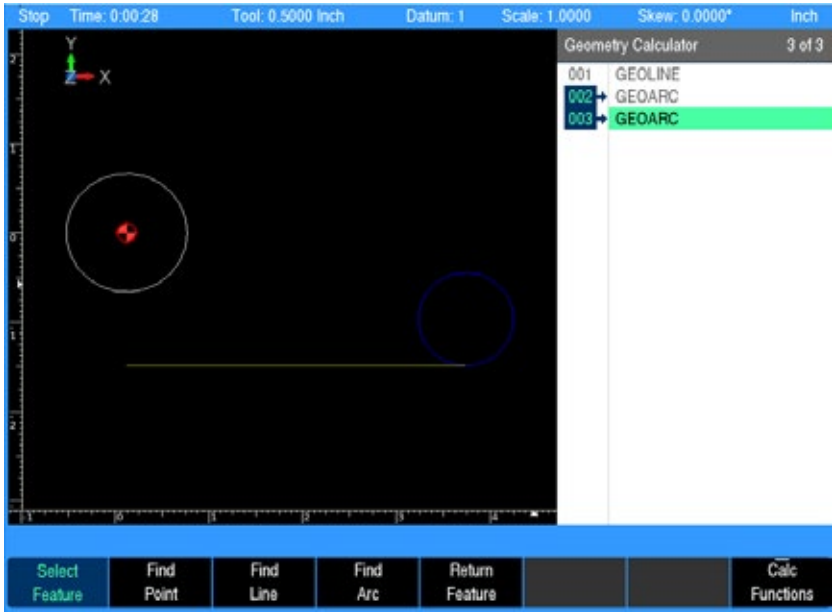
PPT SLIDE 262

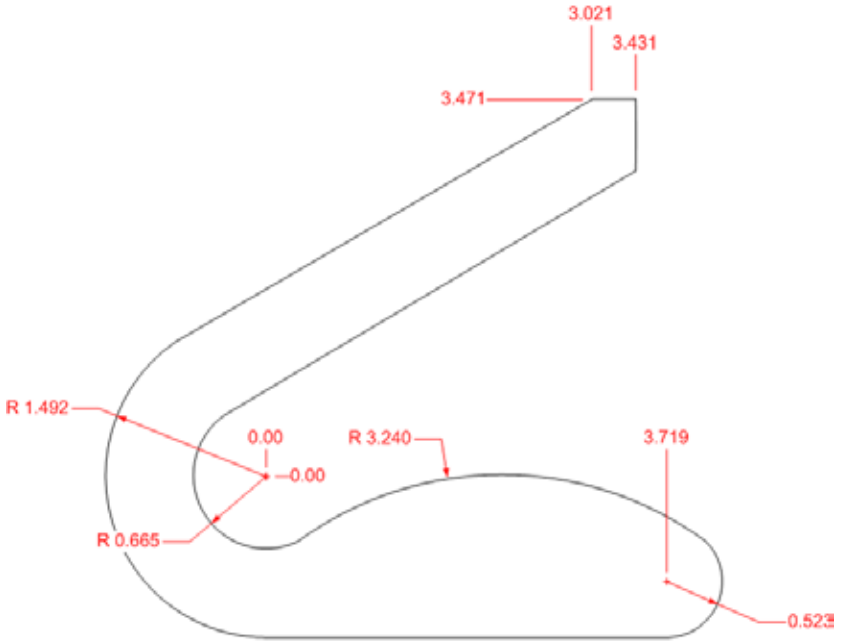
Notes

Keep One Result

Clear Results

- Highlight the desired arc and press the Keep One Result soft key, followed by the Clear Results soft key. The kept arc is now part of your geometry list and the remaining found arcs are discarded.

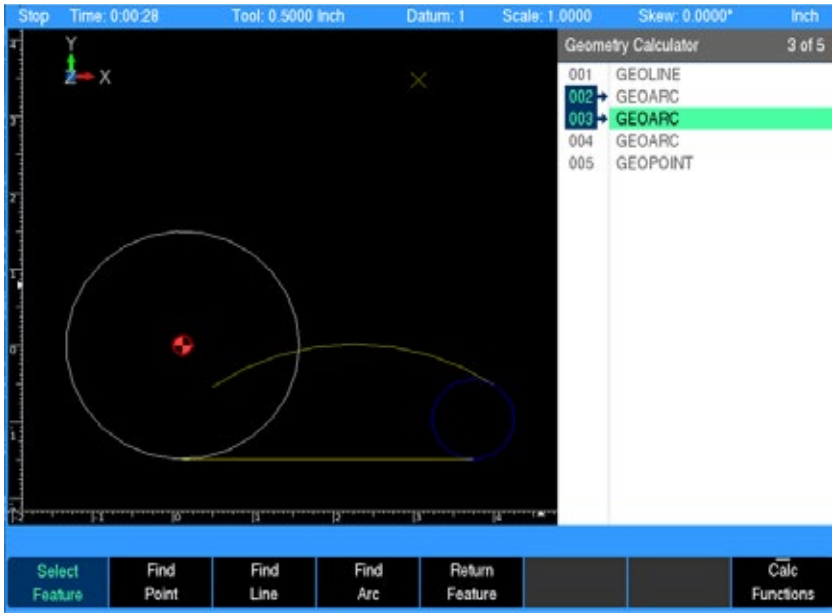


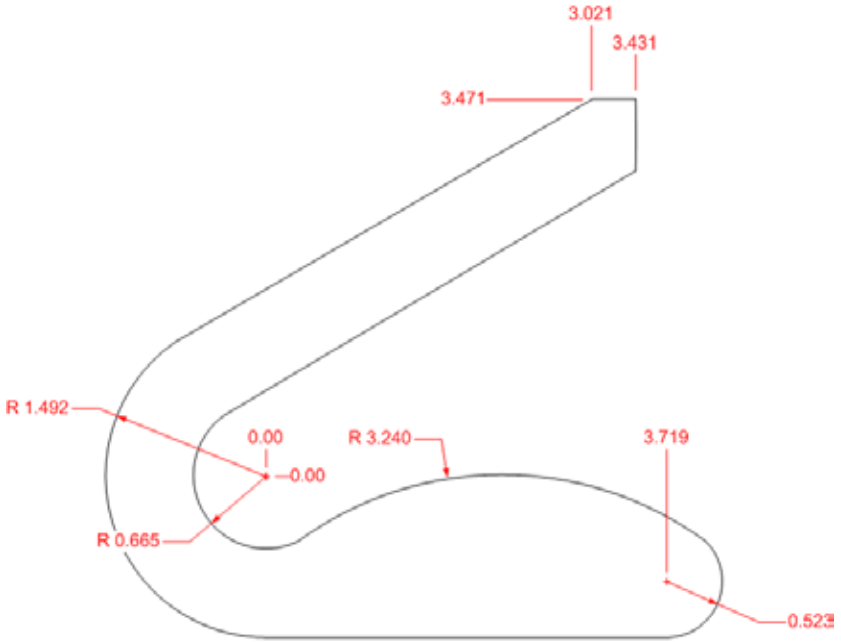


PPT SLIDE 263

Notes

- Create a Point describing the left end of the top line as shown on the print.
- Press the POS key and enter Position X = 3.0210, Y = 3.4710.
- Press USE.
- Change the left arc to a radius of 1.492.
- Highlight the arc (step 2) and press ENTER.
- Change the radius to 1.492 and press USE.

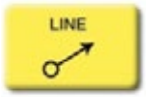




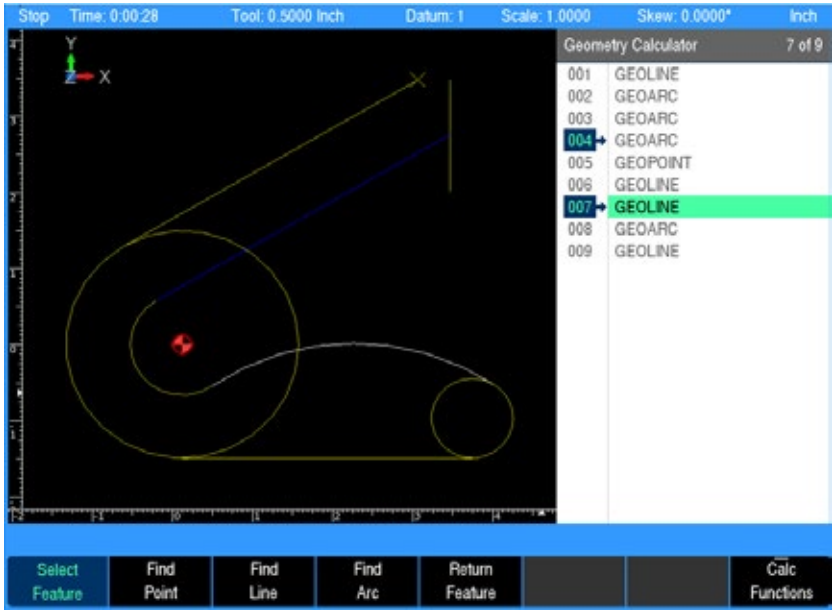
PPT SLIDE 264

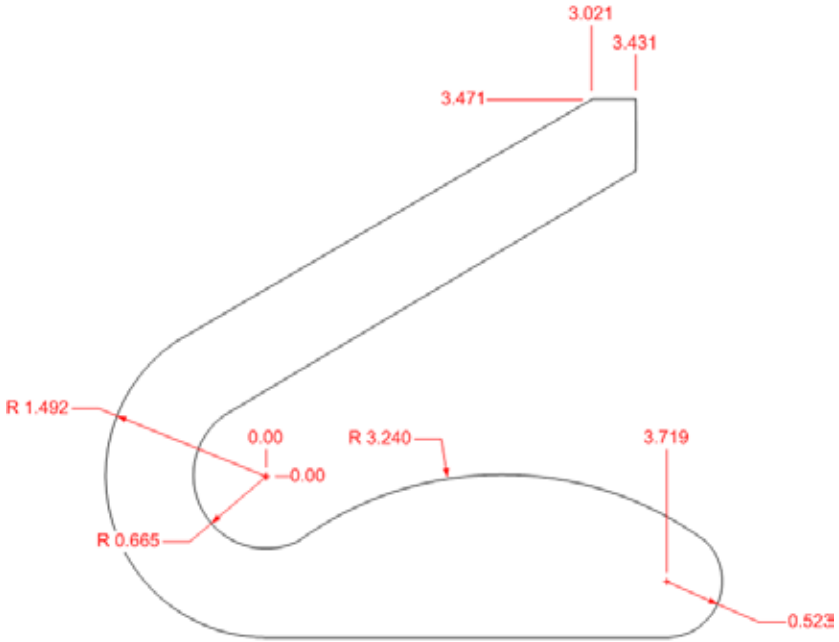
Notes

- Select the point (step 5) and the 1.492 arc (step 2) then FIND LINE. Keep the desired line.
- Select only the newly found line then FIND LINE. Enter a distance of 0.827. (1.4920r - 0.6650r = 0.8270) Keep the desired line.
- Select the new found line and the 3.240 radius (step 4) and FIND ARC with a radius of .665 Keep the desired arc. Clear the remaining results.



- Create a vertical line from the top right of the part. From X1 = 3.431, Y1 = 3.471, To X2 = 3.431 Y = 2.000. The length is not critical. You only need to insure it is well placed in X.





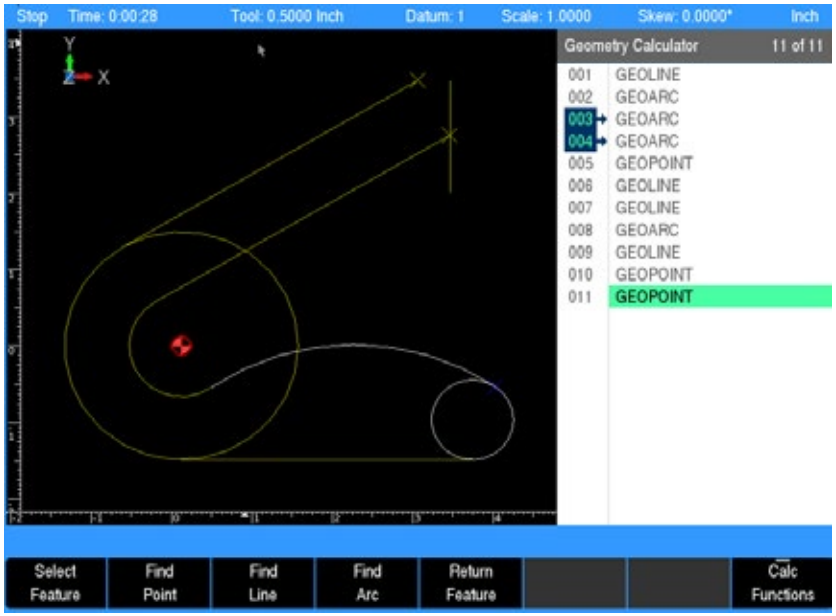
PPT SLIDE 265

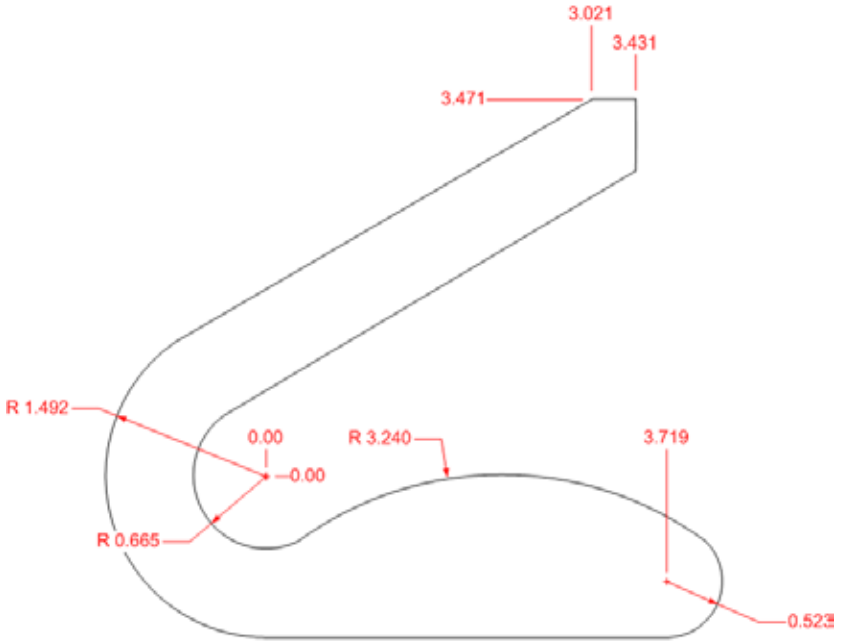
Notes

Keep One Result

Return Feature

- Select this line and the line tangent to the 0.665 radius (step 7) then press Find Point. Highlight and keep the found point.
- Select the 0.523 arc (step 3) and the 3.240 arc (step 4) and press find point. Keep the found point. Remember, this is where we left off in our program when we entered the geometry calculator.
- Highlight this point (step 11) and press Keep One Result, then press Return Feature.





PPT SLIDE 266

Notes

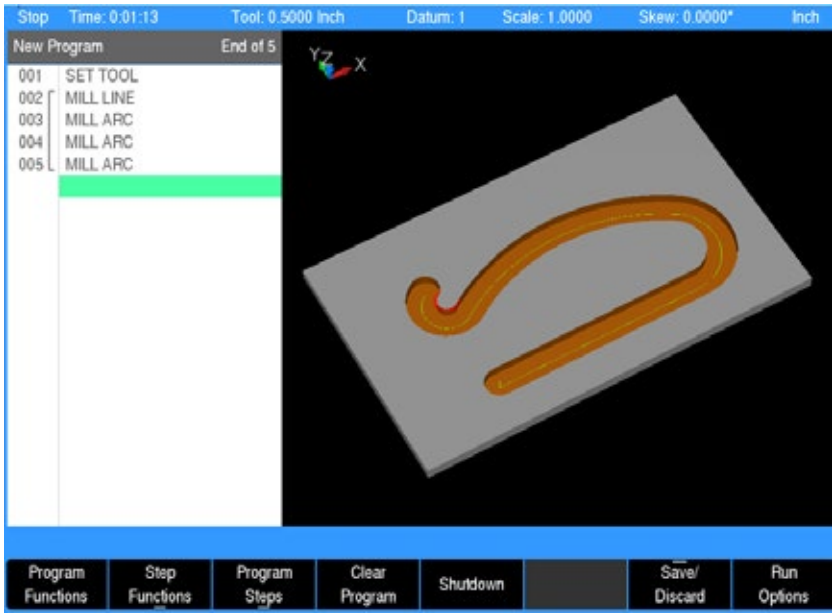
CALC

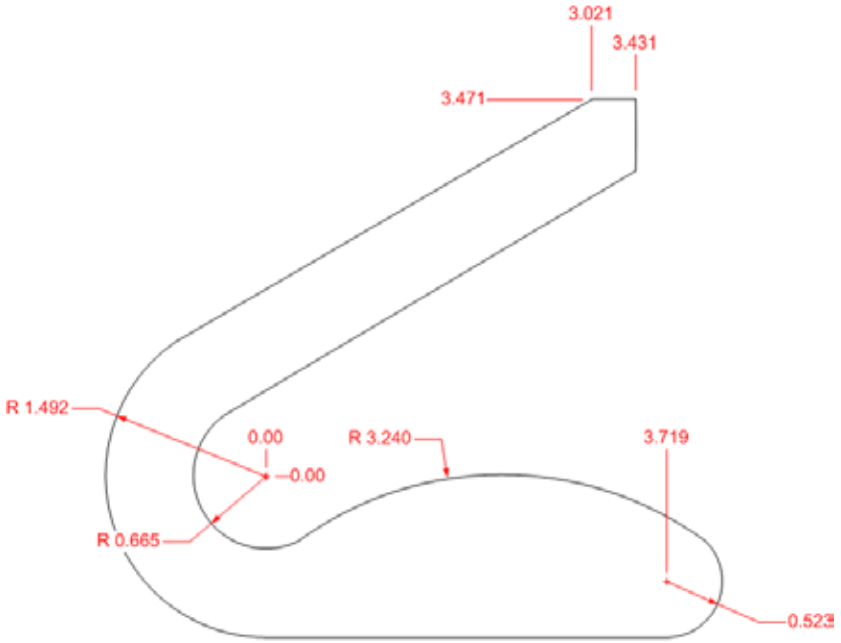
Geometry

Return Feature

- The coordinates of the found point are returned to the waiting Mill Arc step.
- Enter the 0.523 radius.
- Insure the direction is CCW.
- Press USE.
- Return to the Geometry Calculator by pressing the CALC button then the Geometry soft key.
- Highlight the 3.240” radius Geoarc (step 4) and press the Return Feature soft key.
- Press USE.
- Highlight the 0.665” radius Geoarc (step 8) and press the Return Feature soft key.
- This arc has been returned with a CCW direction and the end points reversed. We can easily fix this.

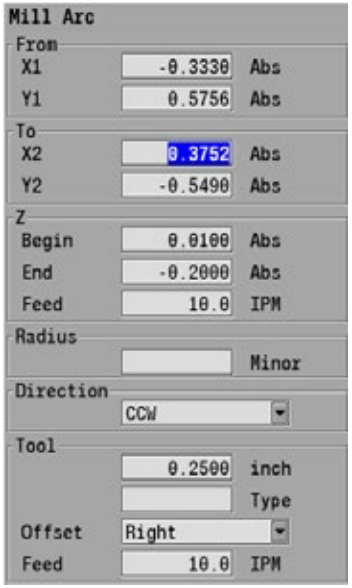
From			
	X1	3.7190	Abs
	Y1	-1.4920	Abs
To			
	X2	4.0107	Abs
	Y2	-0.5349	Abs
Z			
	Begin	0.0100	Abs
	End	-0.2000	Abs
	Feed	10.0	IPM
Radius			
		0.5230	Minor
Direction			
		CCW	



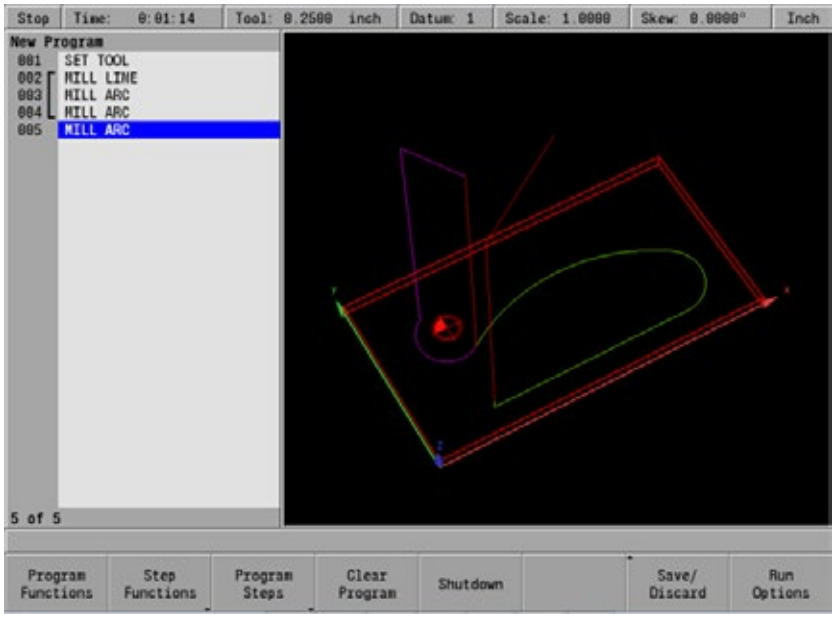


Notes

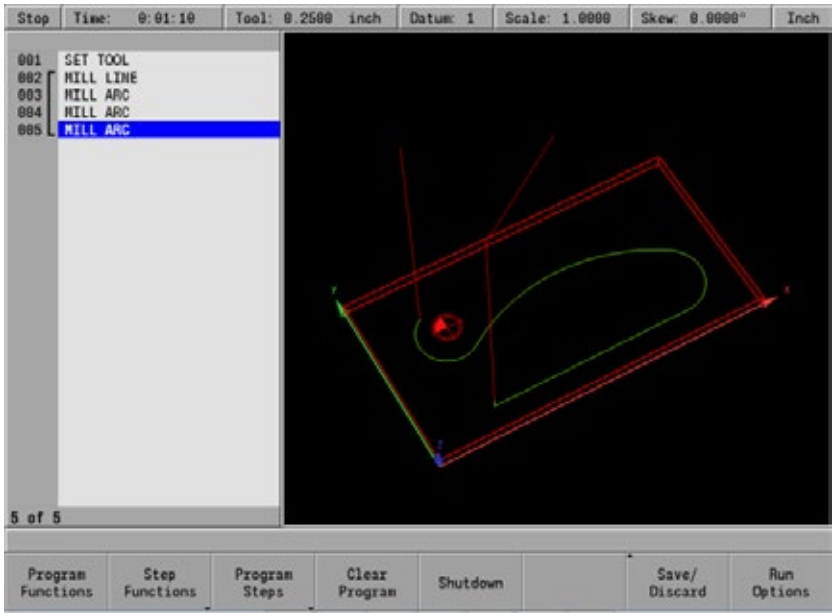
- Highlight the Offset field and choose Inside.
- Press USE.
- Now Highlight the MILL ARC, step (Step 5), and press the Step Functions soft key then press the Reverse Step soft key. The Reverse Step functions flips the step without the need for extensive editing.



Before Reverse Step



After Reverse Step



PPT SLIDE 268

Notes

With the continuous tool path maintained by reversing the arc in program step 5 we can continue building our path.

CALC

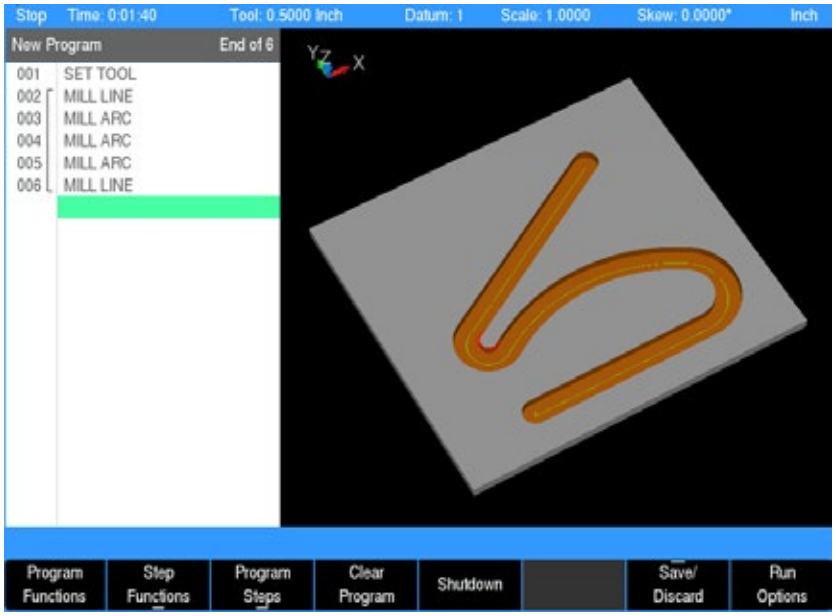
Geometry

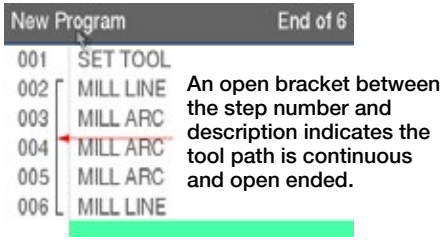
Return Feature

- Insure your curser is under the last step of the program and press the CALC key then the Geometry soft key to return to the geometry calculator.
- Highlight the GEOLINE (Step 7) and press the Return Feature soft key.
- This step also needs to be reversed. Before doing so highlight the offset and switch it to Left. This is needed because the reverse step function automatically reverses the offset to maintain a continuous tool path. When reversed, the offset will become Right once again.

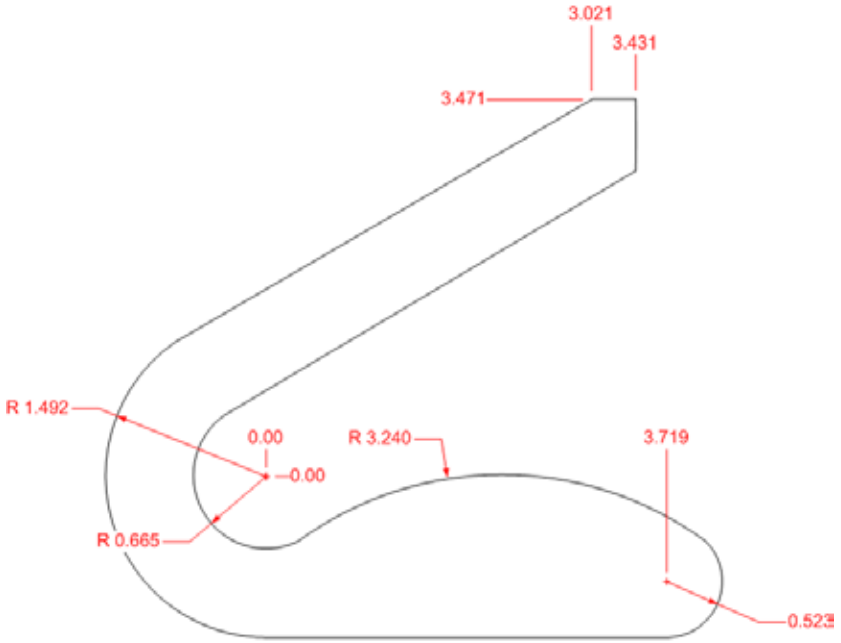
Reverse Step

- Press USE then highlight the MILL LINE step and press the Step Functions soft key then press the Reverse Step soft key.





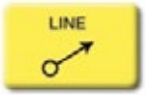
An open bracket between the step number and description indicates the tool path is continuous and open ended.



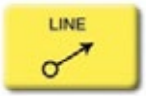
PPT SLIDE 269

Notes

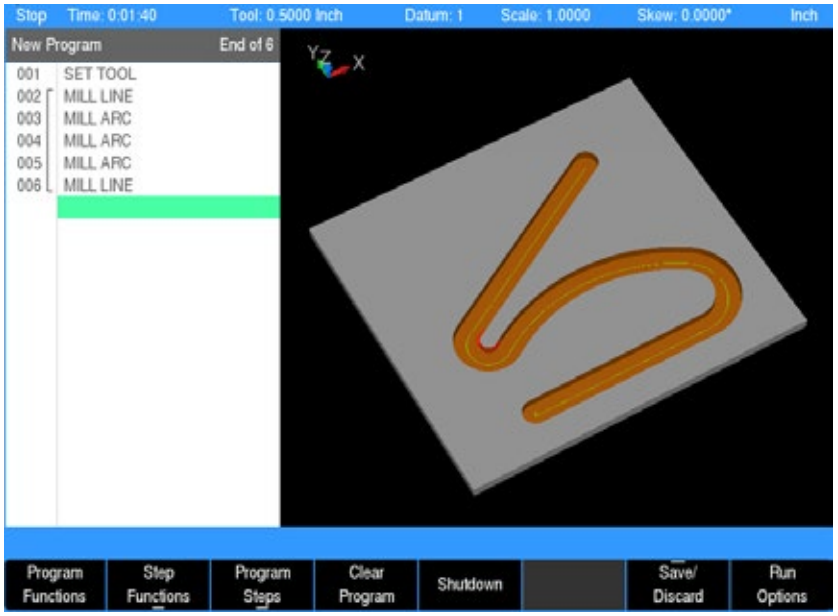
We can continue building our path. Because we know the end point for the next two step we will program the lines without returning to the calculator.

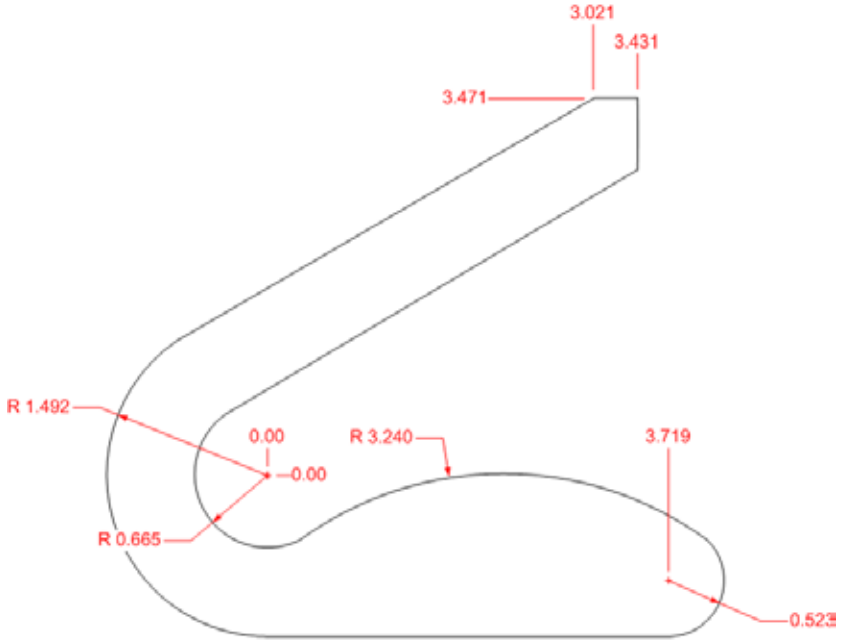


- Press the LINE key.
- The from point, offset, Z information and speeds are carried forward.
- Enter the end point of X2 = 3.431” , Y2 = 3.471”.



- Press USE.
- Press the LINE key.
- The from point, offset, Z information and speeds are carried forward.
- Enter the end point of X2 = 3.021” , Y2 = 3.471”.
- Press USE.





PPT SLIDE 270

Notes

For the next step, return to the geometry calculator and return the line, step 6.

CALC

Geometry

Return Feature

ARC

CALC

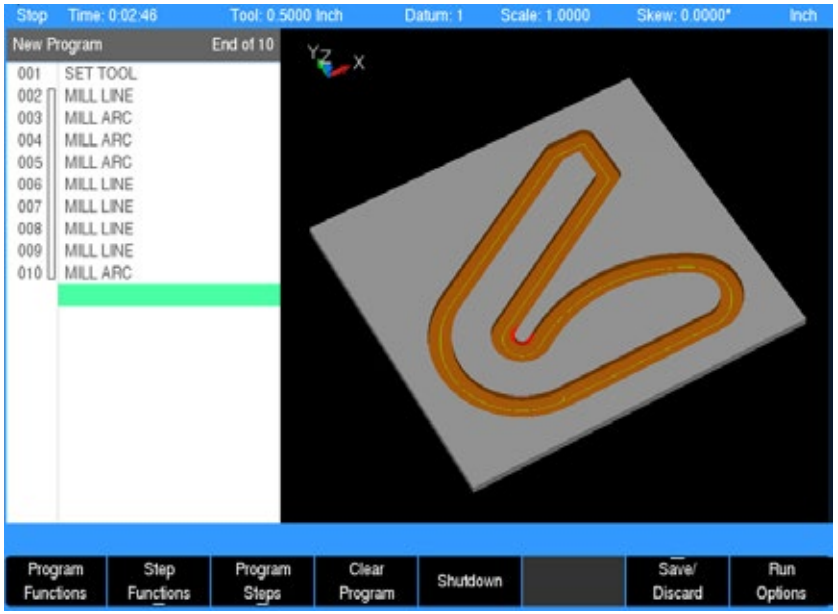
Geometry

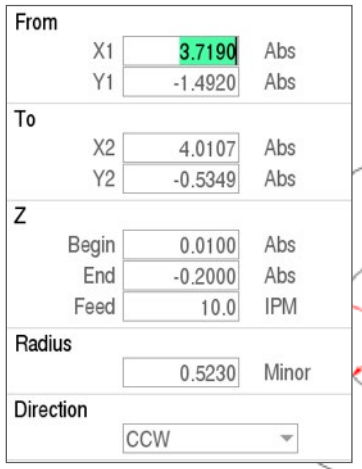
Select Feature

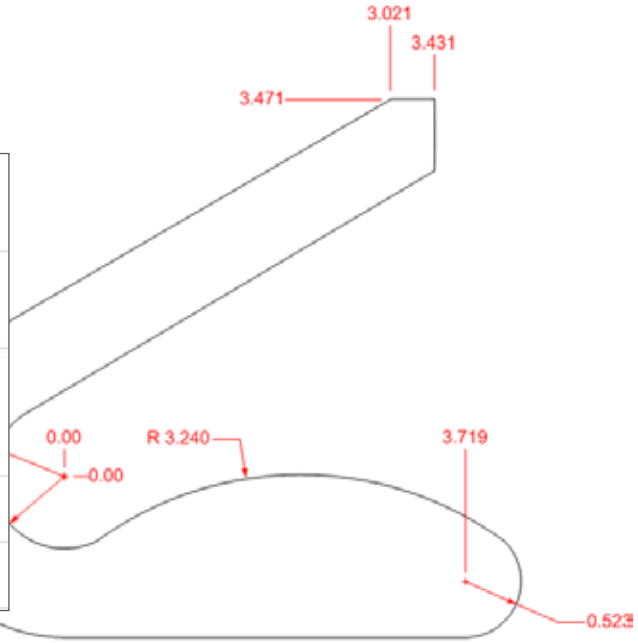
Find Point

Keep One Feature

- Insure your curser is under the last step of the program and press the CALC key then the Geometry soft key to return to the geometry calculator.
- Highlight the GEOLINE (Step 6) and press the Return Feature soft key.
- Press USE.
- Finally while still in the program editor, press the ARC key to create an arc. The end point is what we require.
- Press the CALC key then the Geometry soft key to return to the geometry calculator.
- Highlight steps 1 and 2 and press the Select Feature soft key to select both.
- Press the Find Point soft key.
- Highlight the found point and press the Keep One Feature soft key
- Press the return Feature soft key.
- Arrow down to the radius and enter 1.492”
- Arrow to the Direction and choose CCW
- Press USE.







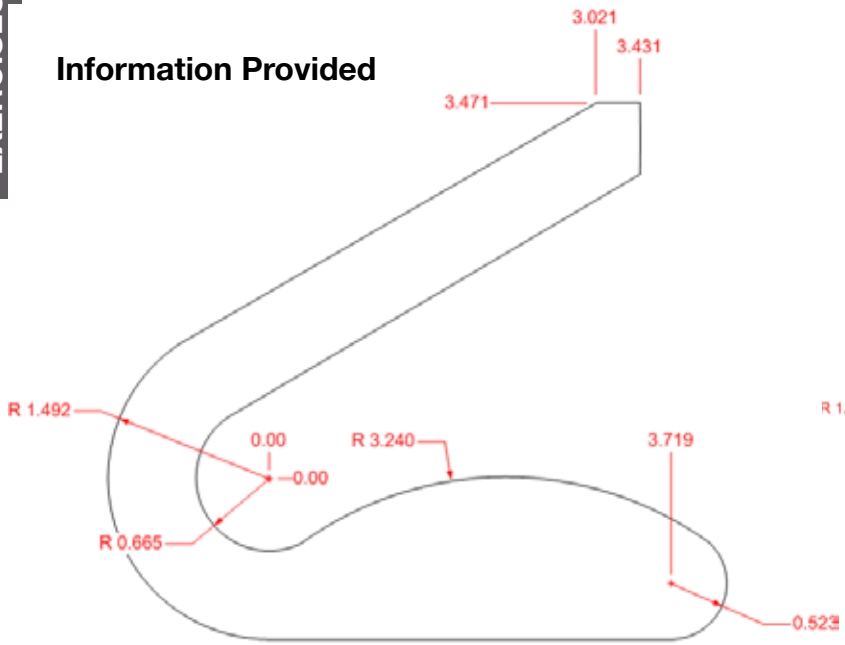
PPT SLIDE 271-272

Notes

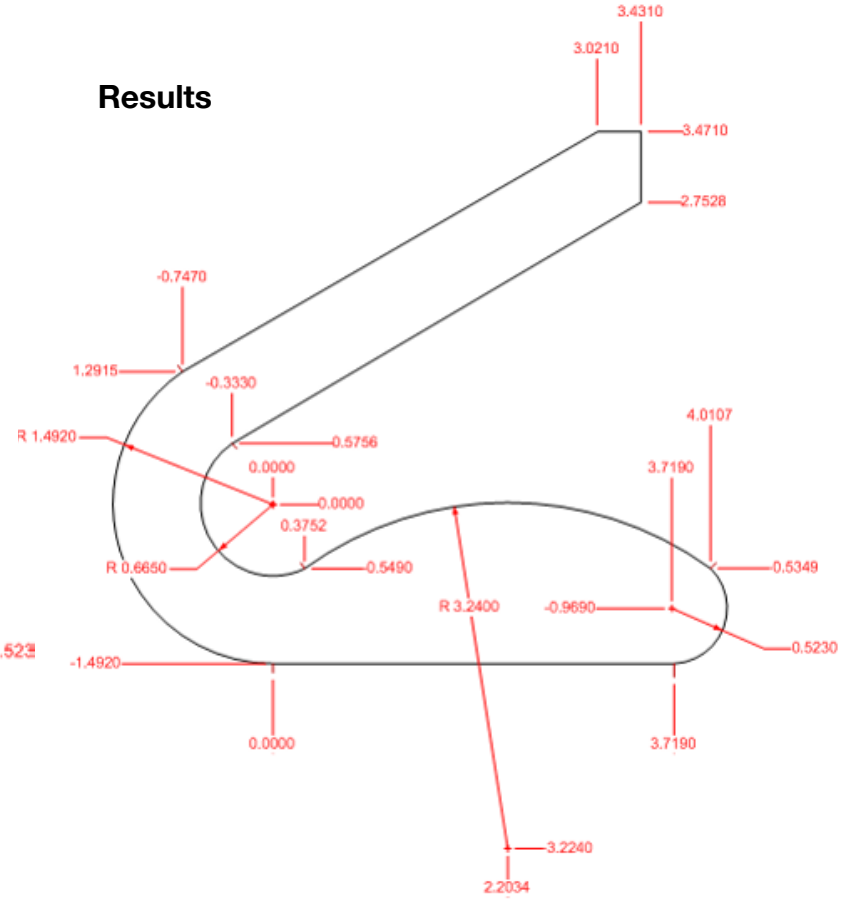
Geometry Calculator Example Program

With the use of the geometry calculator we were able to use the information provided on the left and determine the rest of the dimensions as shown on the right.

Information Provided



Results



Notes

Geo Exercise 1

Finding unknown tangents

CALC

Geometry

- Program the appropriate tool for the job.
- Program a line from zero to 2 inches.
- Begin another line. With the curser in the TO field, press the CALC key then the Geometry soft key.
- In the Geometry calculator, program a line from X, Y zero to Y 2 at 70°.
- Program a second line from X 2, Y 0 zero to Y 2 at 110°.

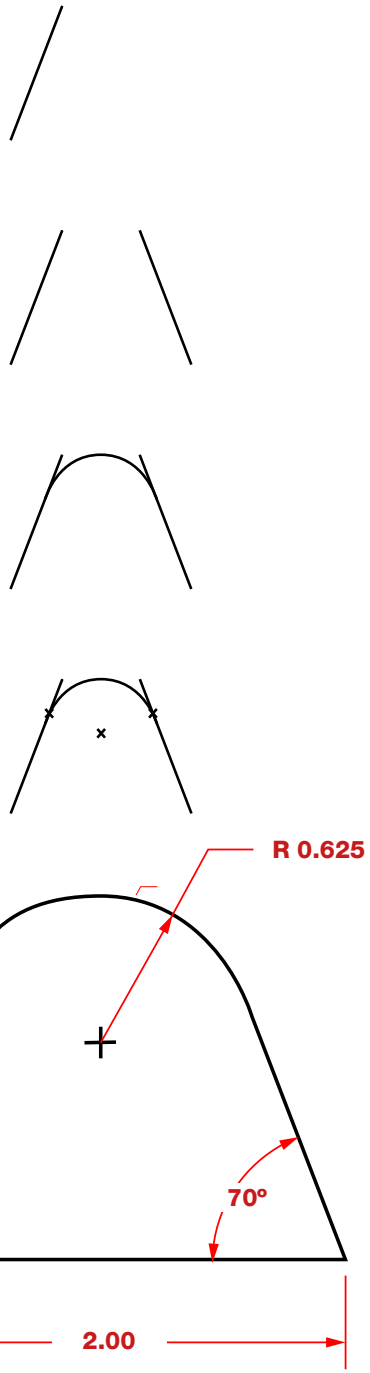
Select Feature

Find Arc

- Using the arrow keys and the Select Feature soft key, highlight and select both lines then press the Find Arc soft key. Enter a radius of .625".
- Choose the desired arc from the list of found arcs and press the Keep One soft key.

Select Feature

- Using the arrow keys and the Select Feature soft key, highlight and select the new arc only. Press the Find Point soft key. Press the Keep All Results soft key.
- Highlight the point representing the right tangent point of the arc and press the Return Feature soft key. The point is now transferred into your program step. Press USE.
- Program an arc. With the curser in the TO field, press the CALC key then the Geometry soft key.
- Highlight the point representing the left tangent point of the arc and press the Return Feature soft key. The point is now transferred into your program step. Enter a radius of .625 and insure the direction is CCW. Press USE.
- Program a line back to X, Y zero.



Notes

Geo Exercise 2

Creating a 1” Hexagon

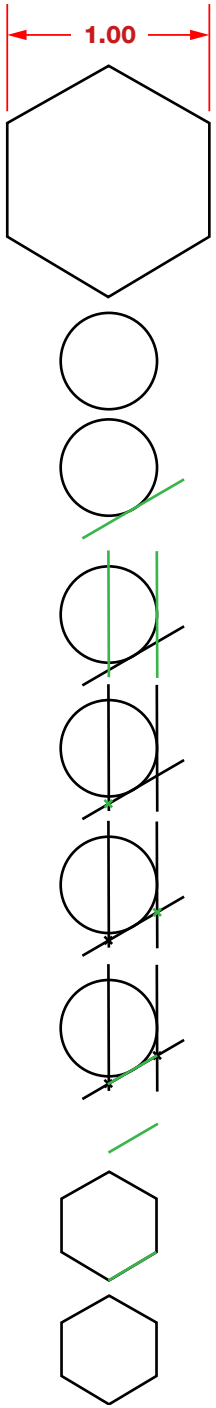
CALC

Geometry

- Program the appropriate tool for the job.
- Press CALC then GEOMETRY to enter the geometry calculator.
- Create a 1” diameter ARC in the calculator.
- Select the arc, find line at 30°, and keep the bottom line.
- Find line at 90° and keep the right and center line.
- Select the 30° and the center 90° lines, find point, and keep it.
- Select the left edge 90° and the 30° lines, find point, and keep it.
- Select the two points, find line, and keep it.

Return Feature

- Highlight the found line, and press RETURN FEATURE.
- Rotate the line around zero at 60° repeating five times.
- Explode the rotate.

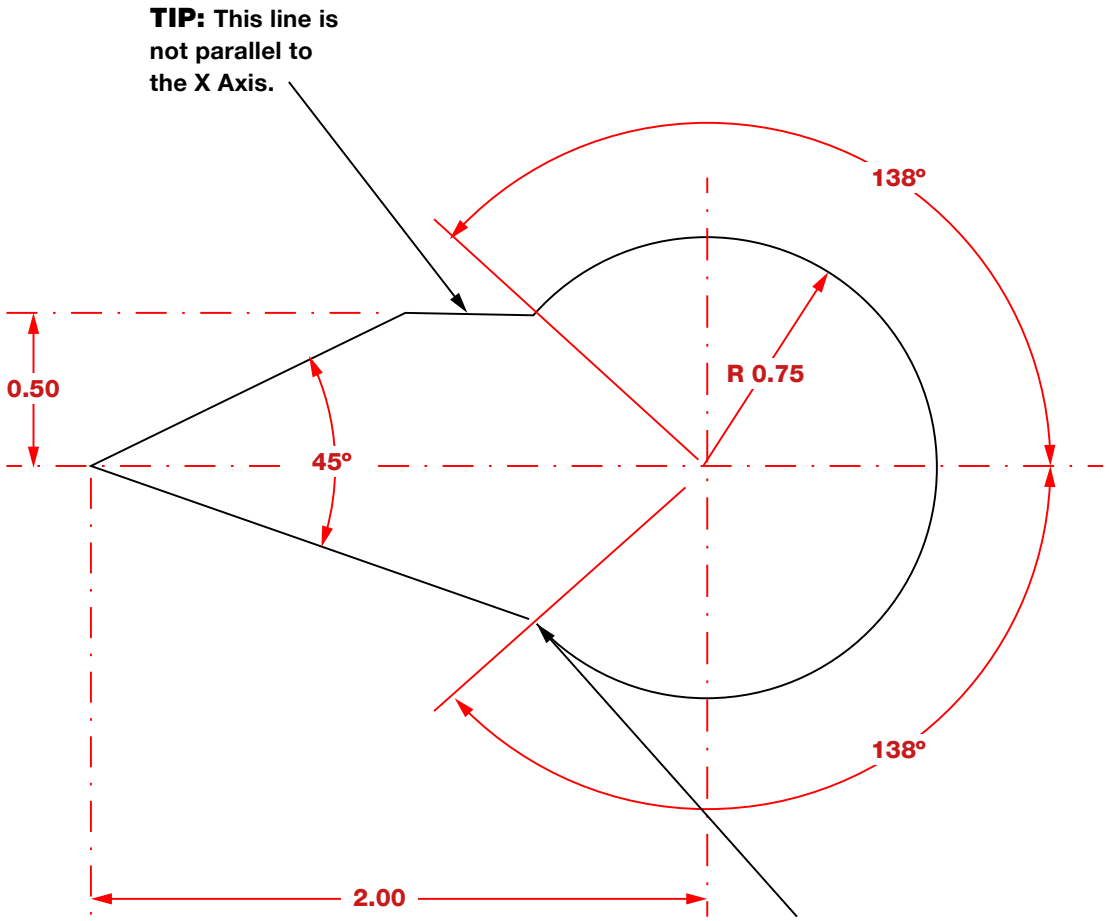


Notes

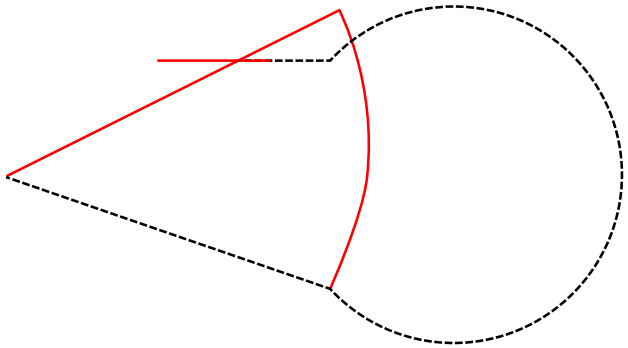
Geo Exercise 3

In the Calculator...

- Draw an arc with a 45° sweep angle
- Find the endpoints and center point of the arc.
- Find the line between center point and end point.
- Draw a line .5” above zero.
- Find intersection of the two lines.



TIP: Use polar coordinate
R = .75 A = -138°
to program this point.



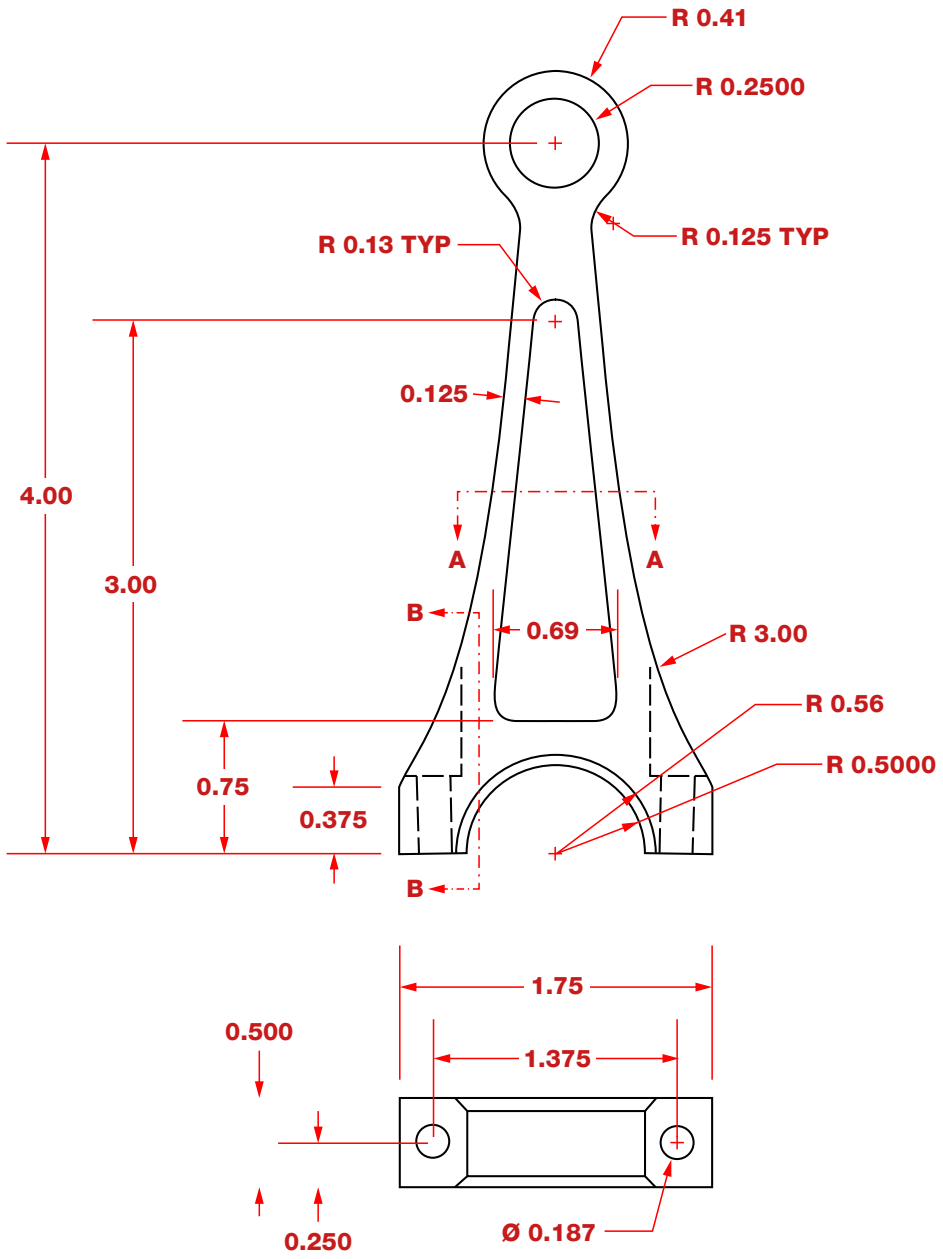
In the Calculator

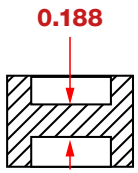
1. Draw an arc with a 45° sweep angle
2. Find the endpoints and center point of the arc.
3. Find the line between center point and end point.
4. Draw a line .5" above zero.
5. Find intersection of the two lines.

Notes

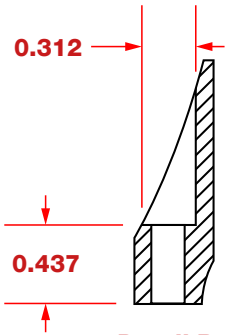
Geo Exercise 4

Create the outer profile and the center relief cut





Detail A

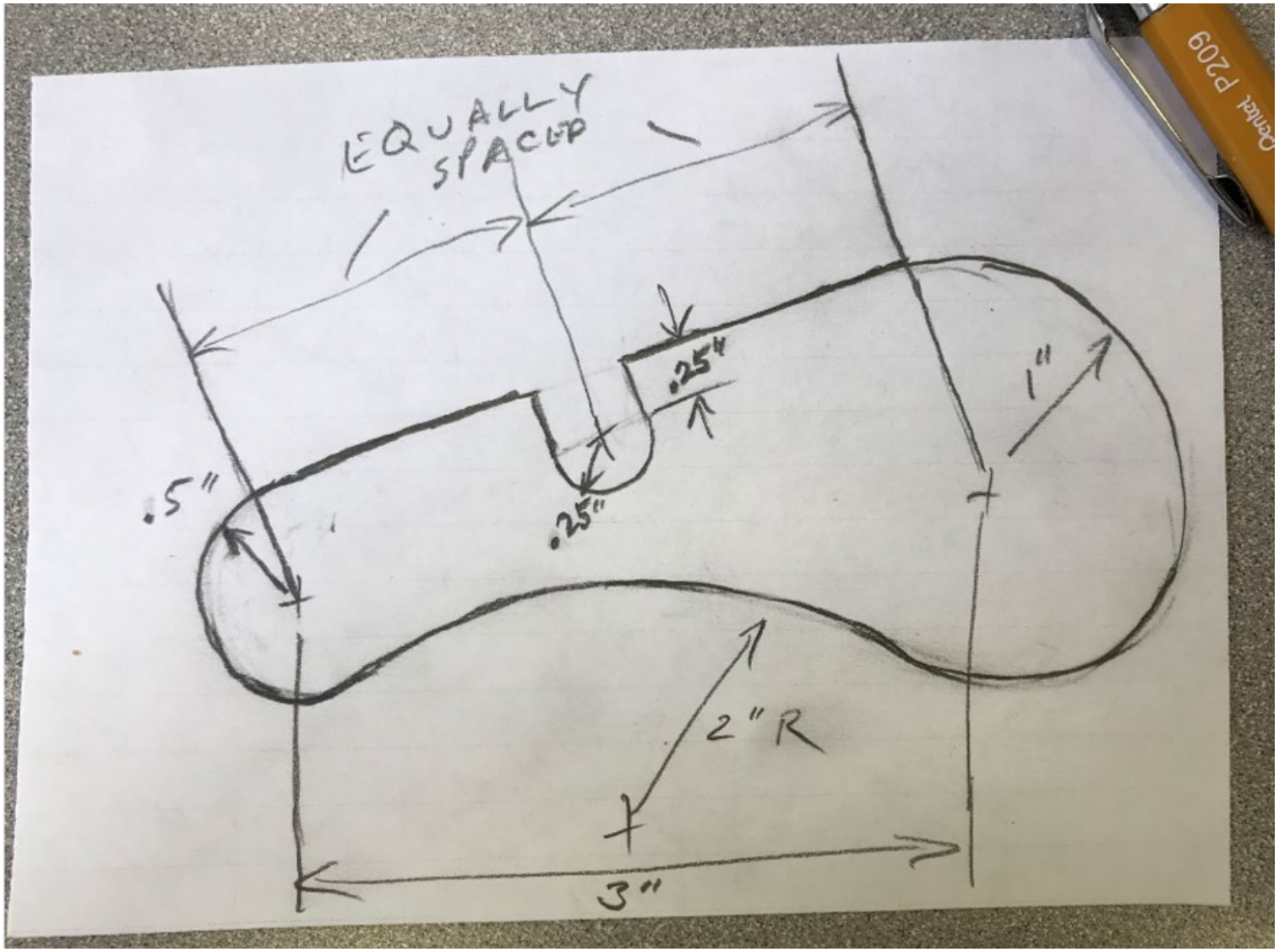


Detail B

Notes

Geo Exercise 5

You're handed a sketch and asked to cut this out of ½" aluminum

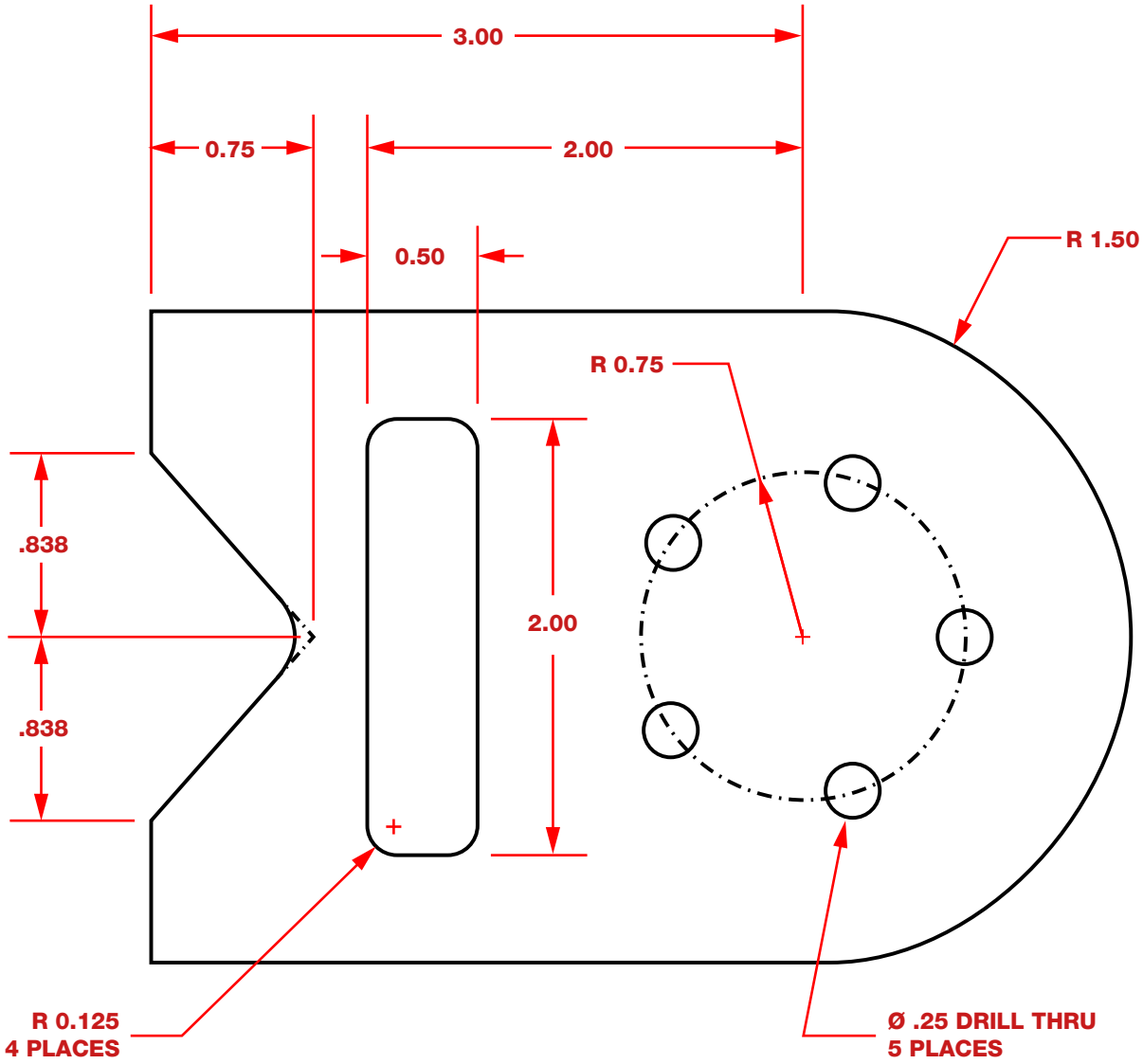


Notes

Exercise 1

Notes (unless otherwise specified)

- All dimensions are in inches.
- All radii are 0.25.

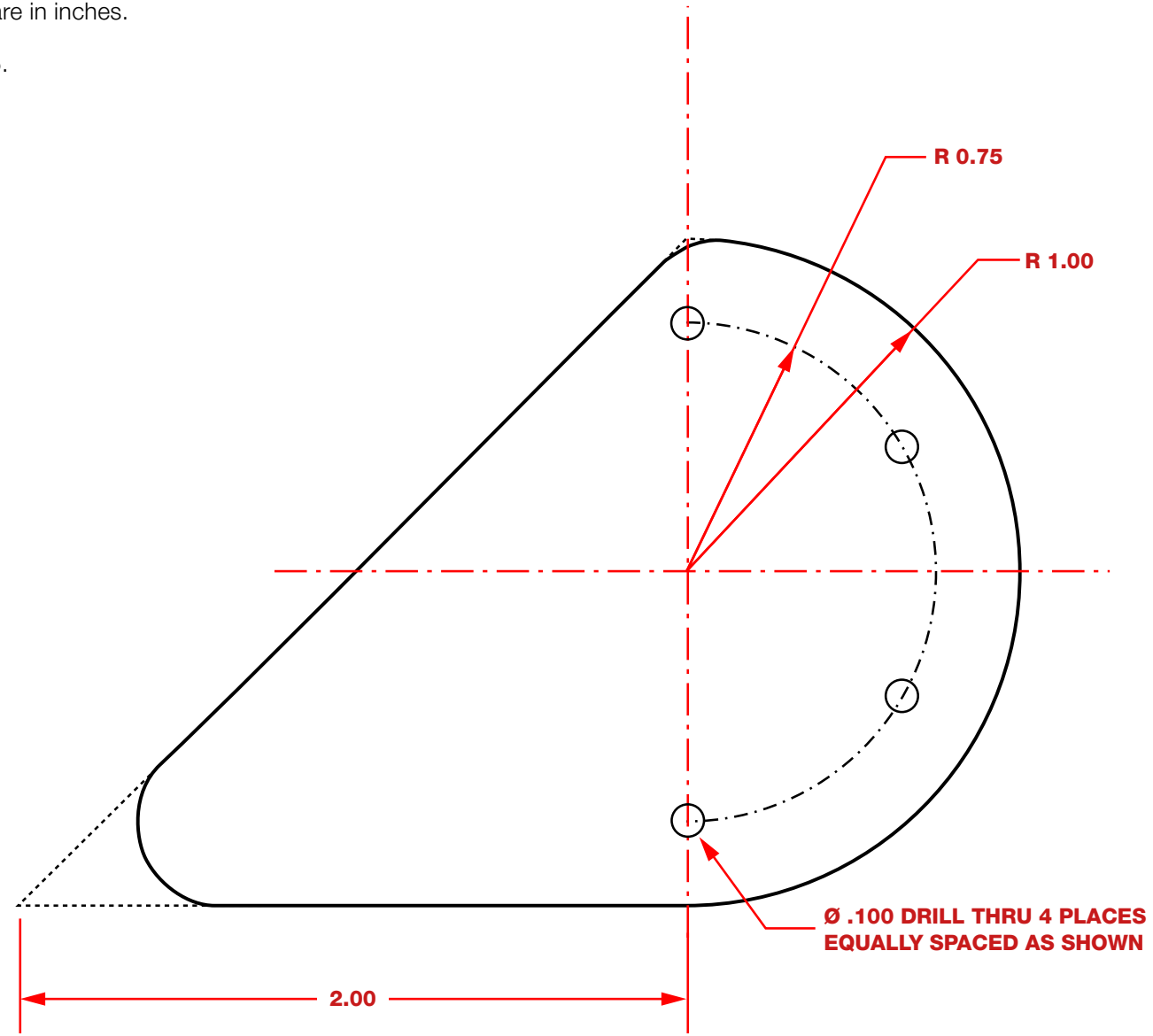


Notes

Exercise 2

Notes (unless otherwise specified)

- All dimensions are in inches.
- All radii are 0.25.

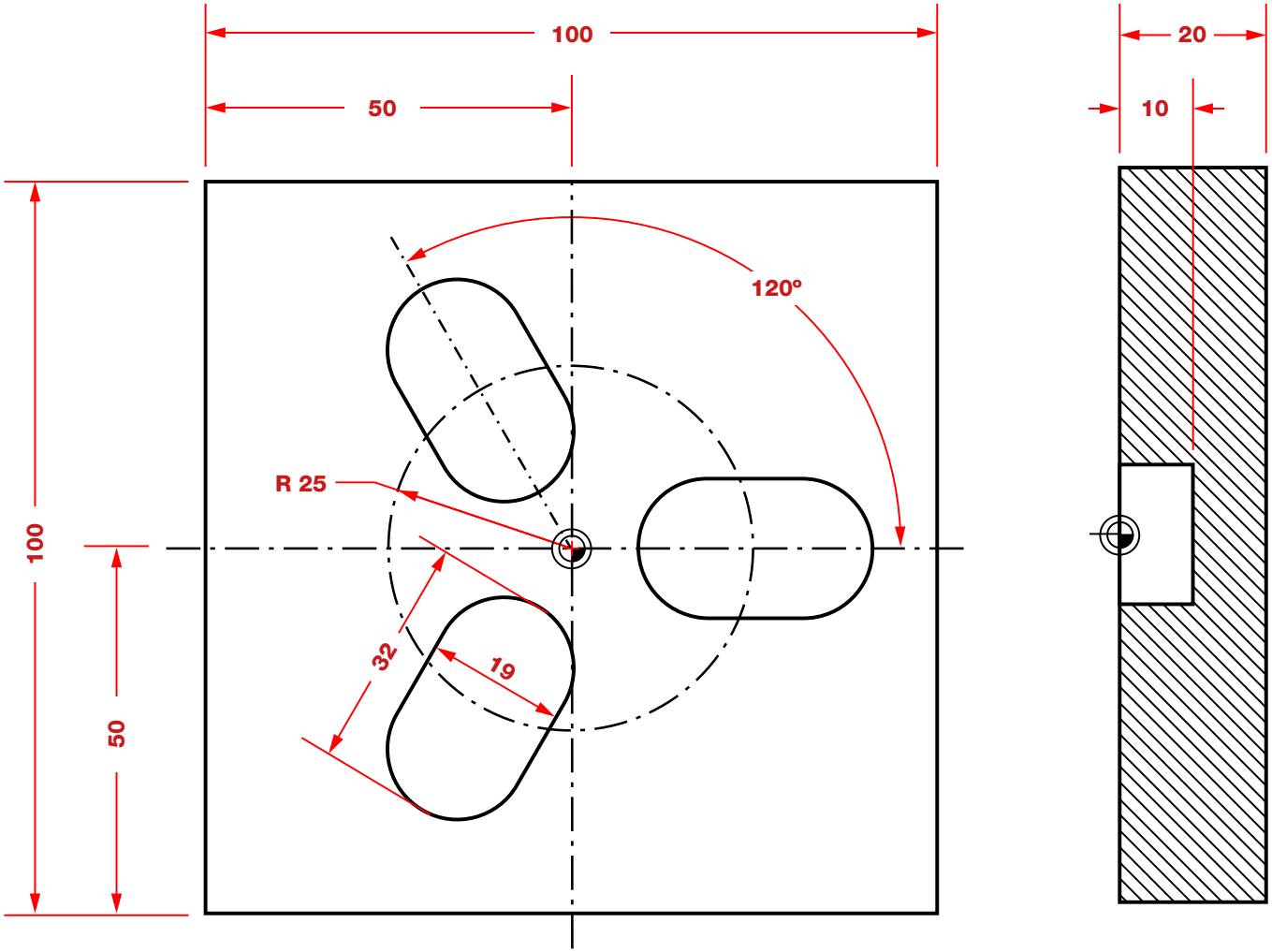


Notes

Exercise 3

Notes (unless otherwise specified)

- All dimensions are in millimeters.

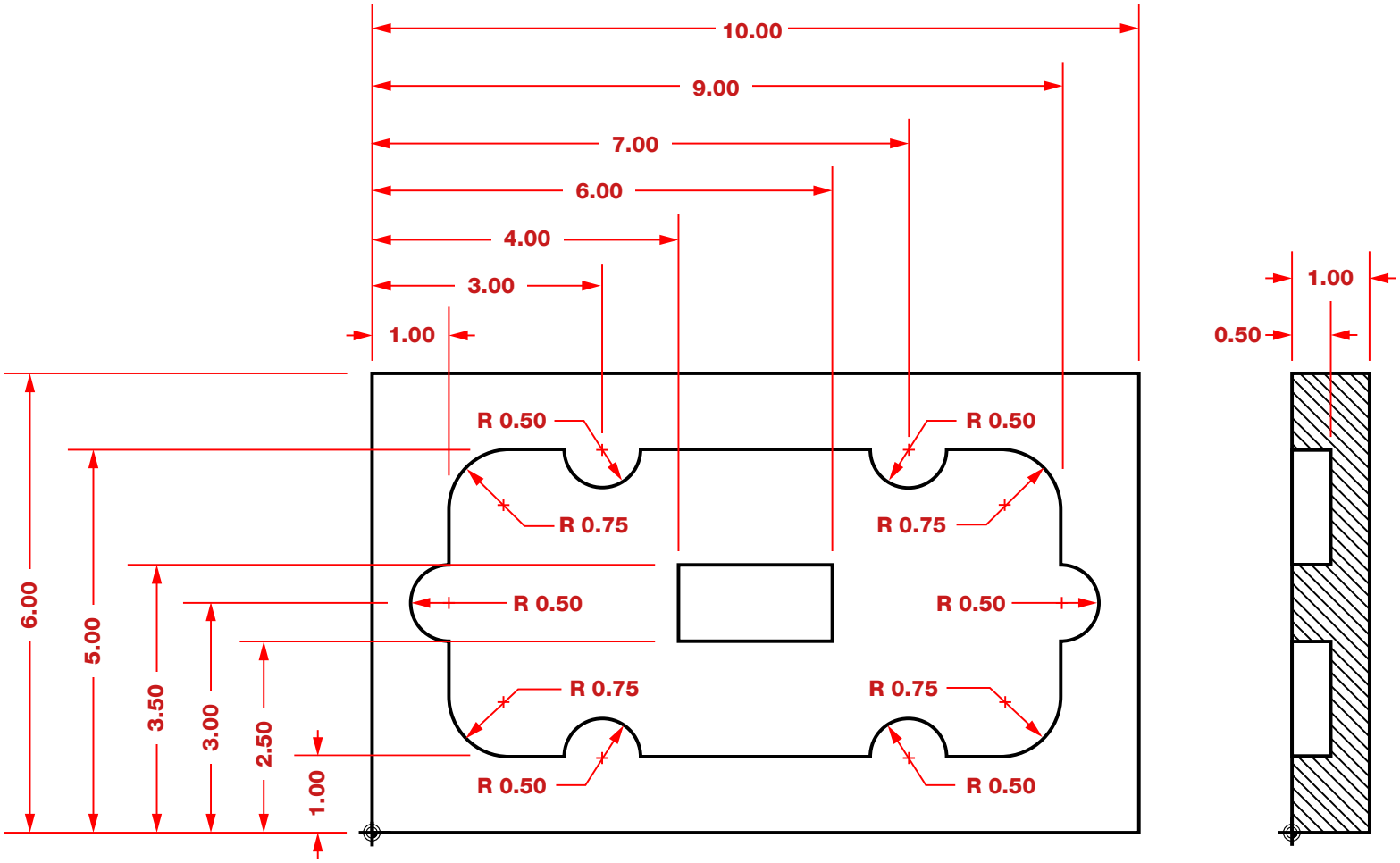


Notes

Exercise 4

Notes (unless otherwise specified)

- All dimensions are in inches.
- Tool radius allowed on inside corners.

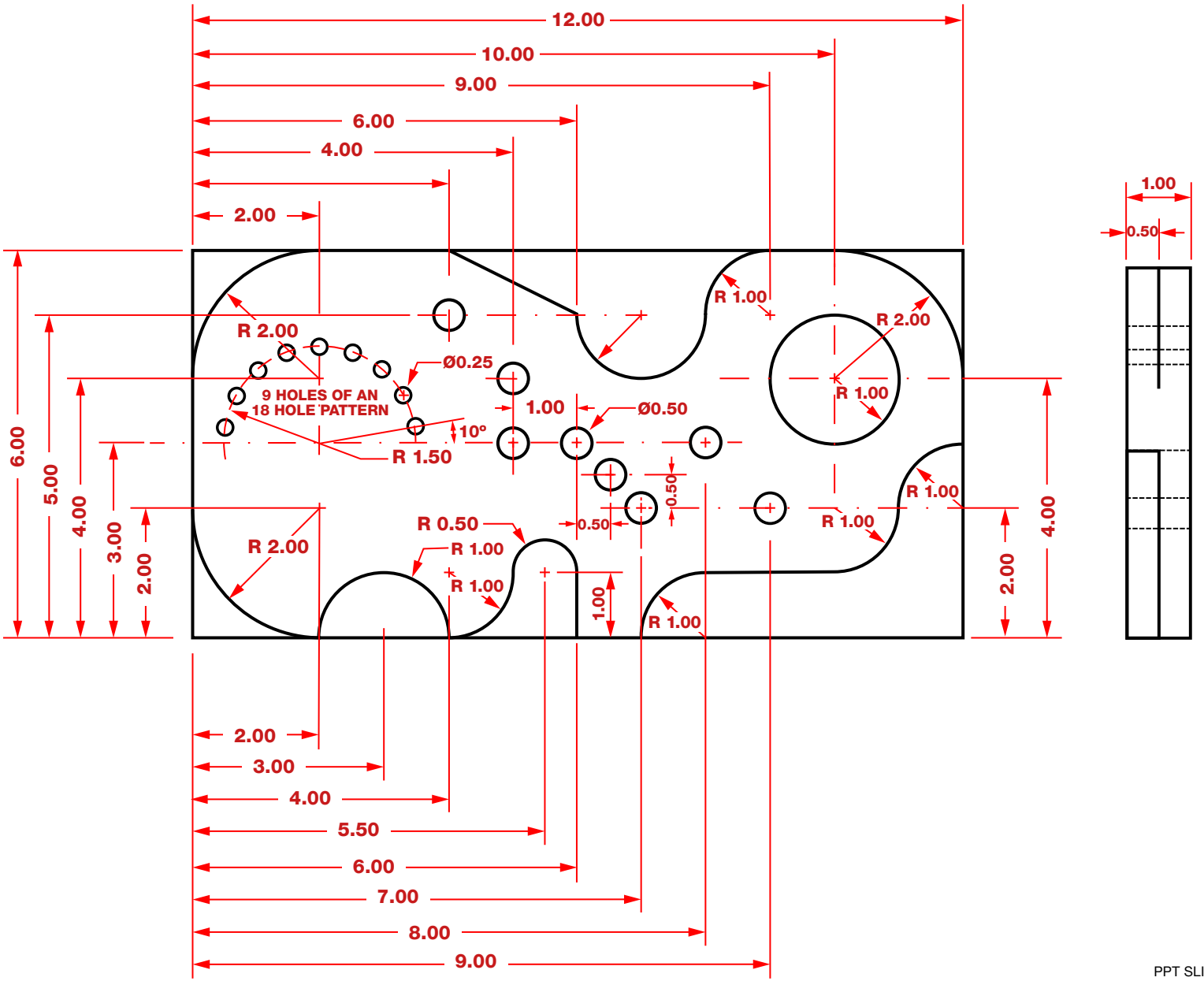


Notes

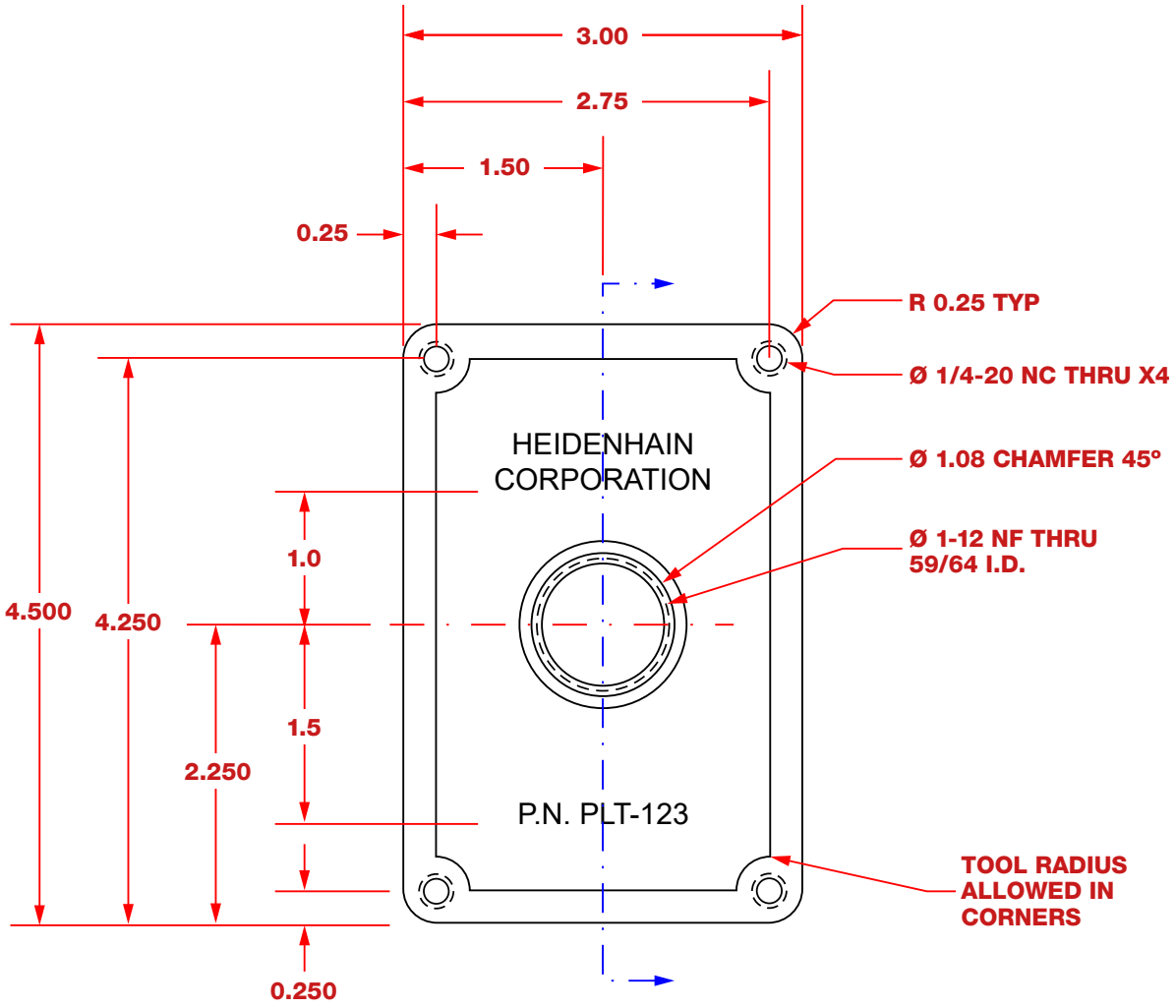
Exercise 5

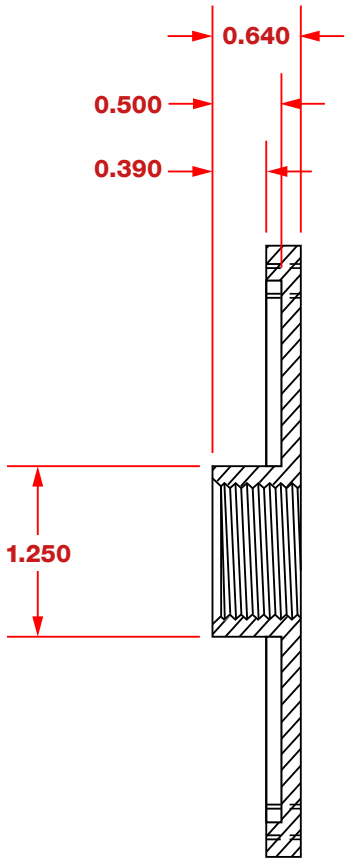
Notes (unless otherwise specified)

- All dimensions are in inches.



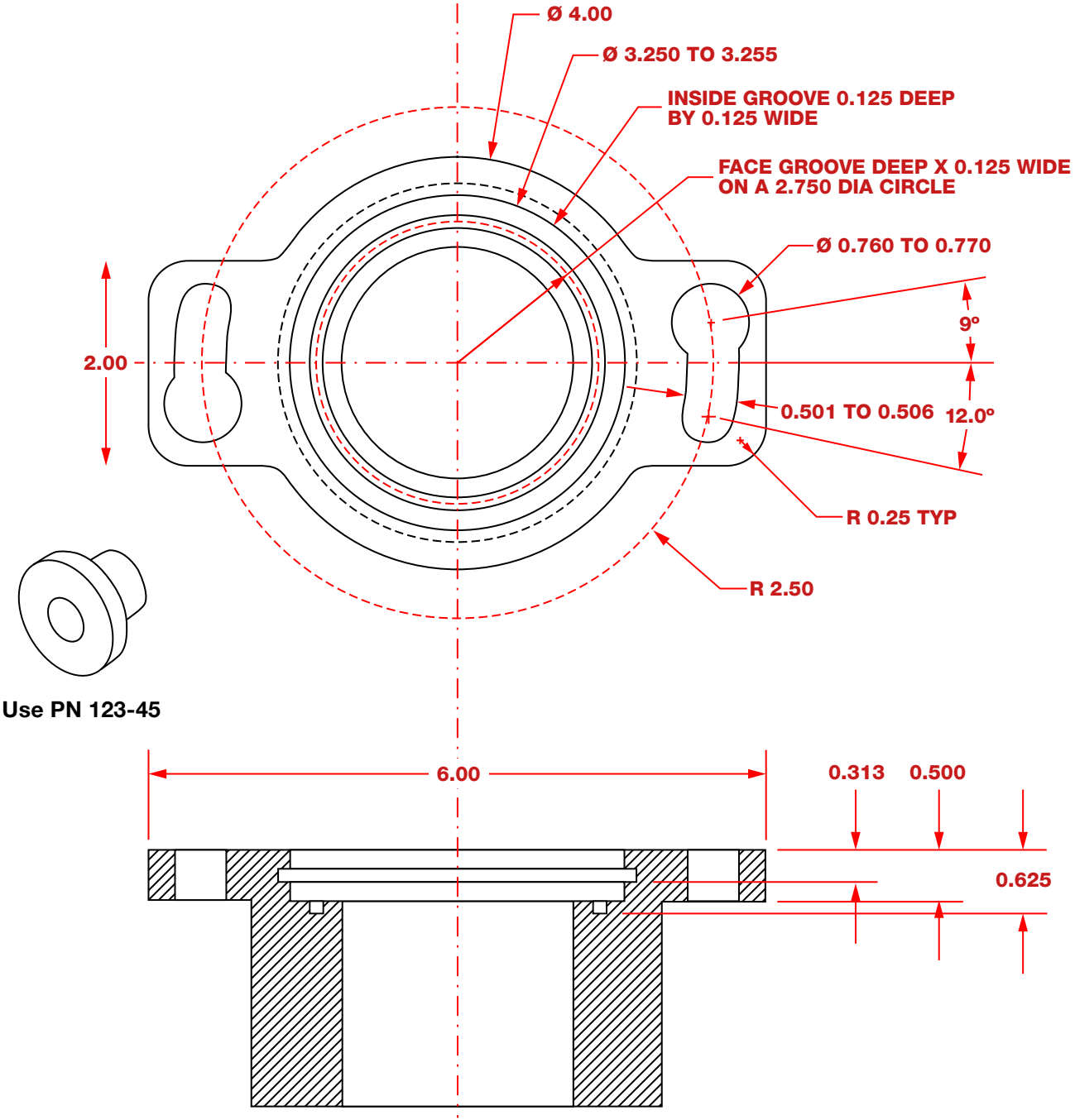
Exercise 6





Notes

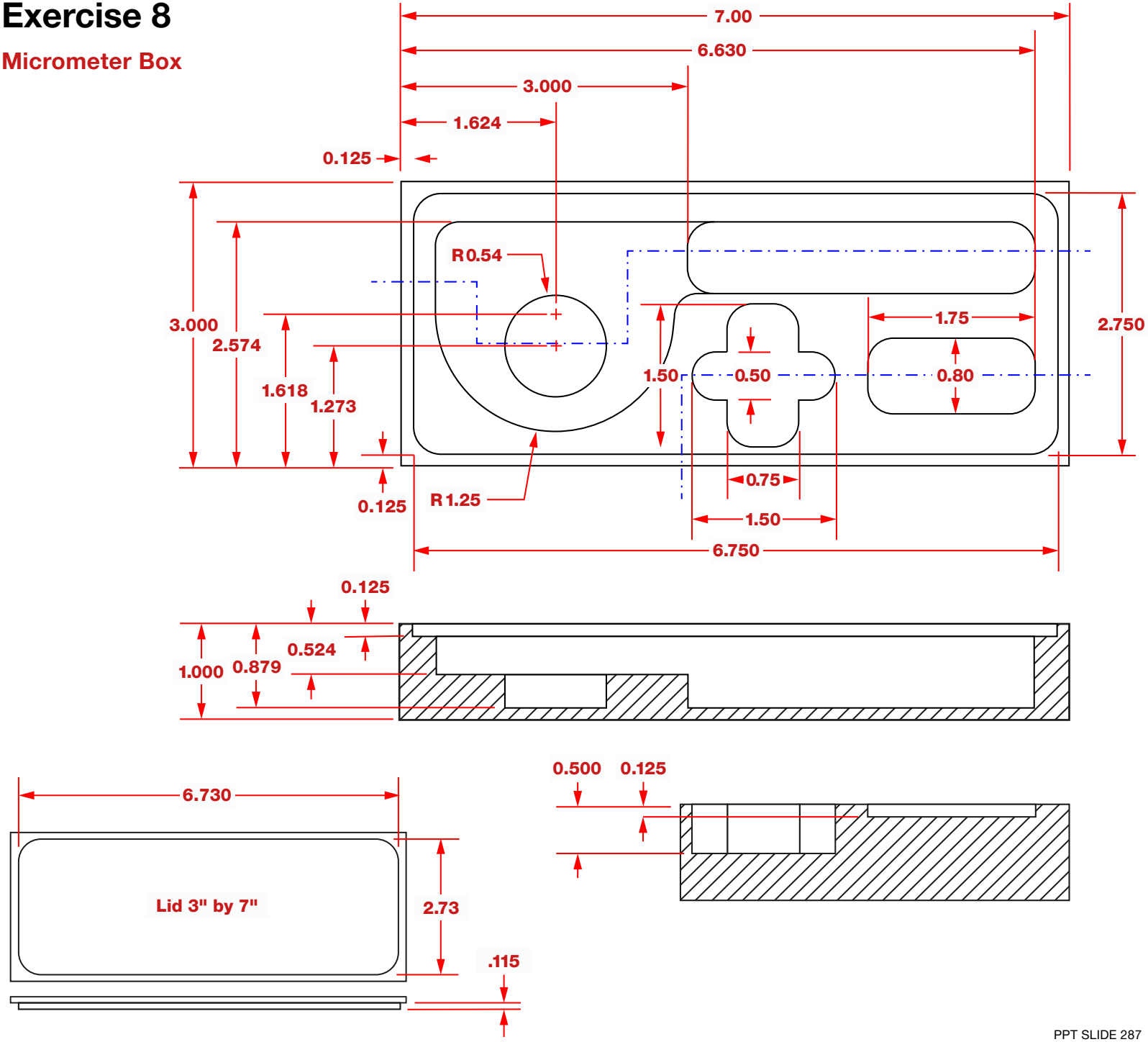
Exercise 7



Notes

Exercise 8

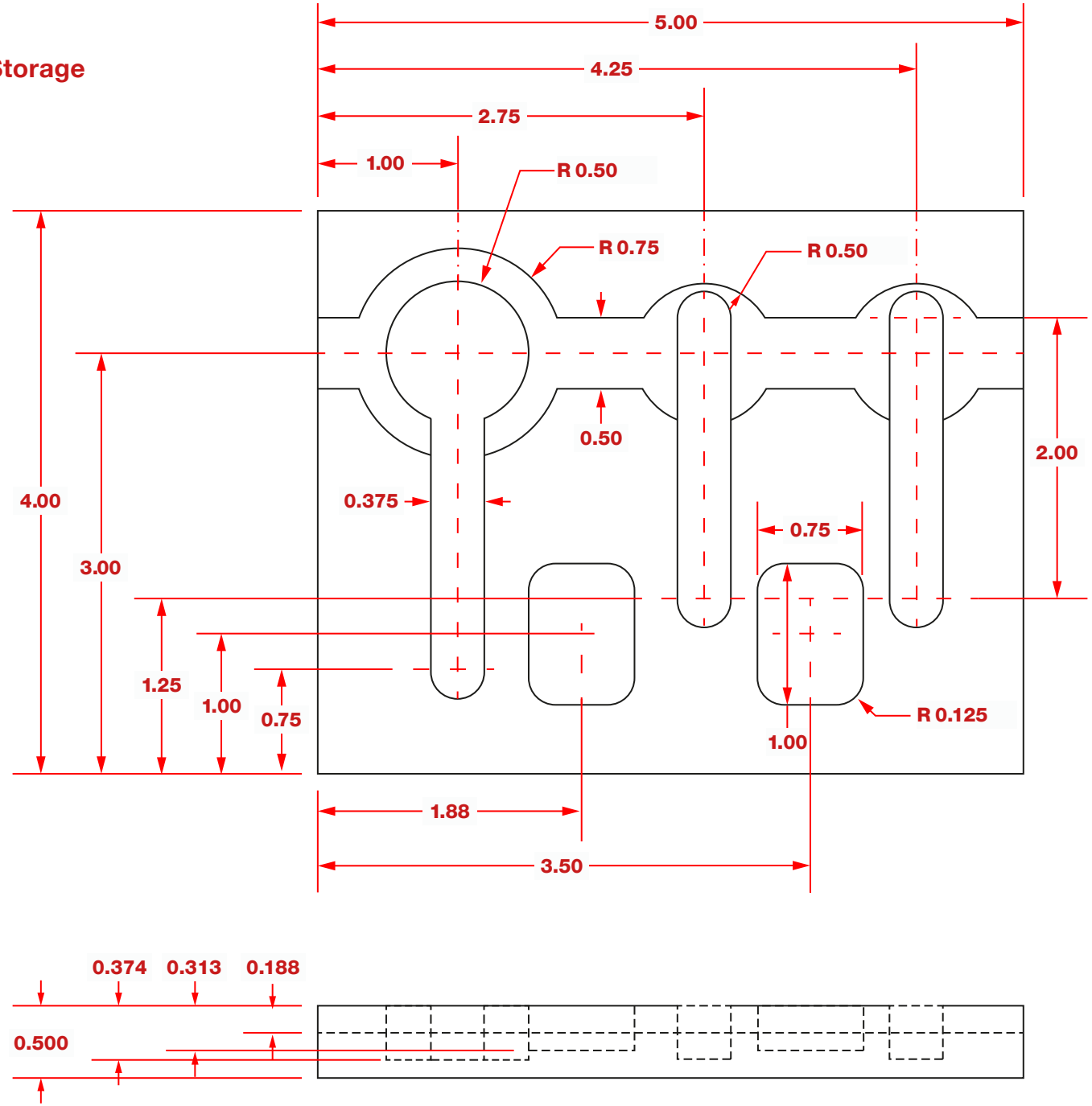
Micrometer Box



Notes

Exercise 9

Dial Indicator Storage

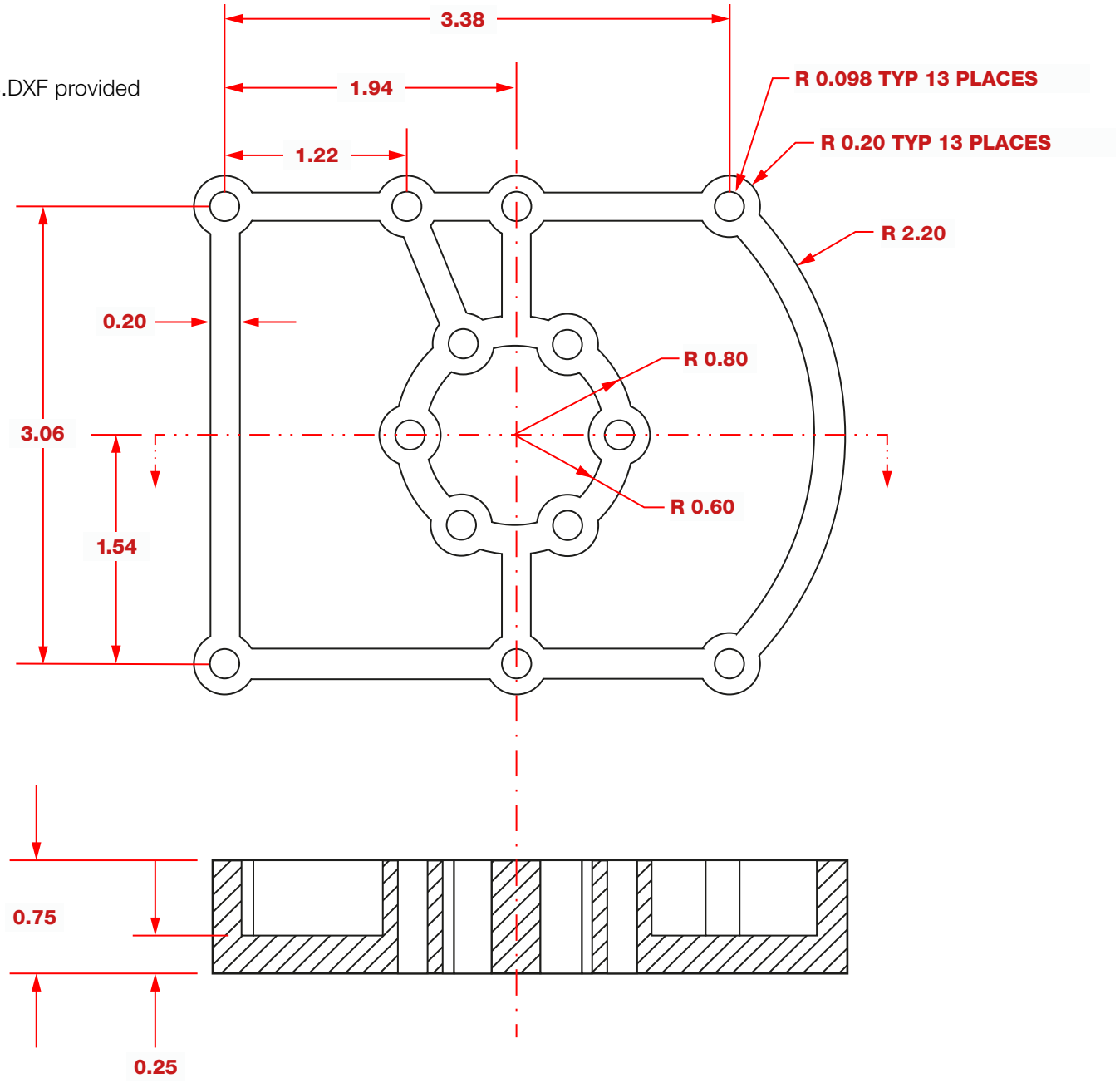


Notes

Exercise 10

DXF Import

- File EXERCISE 8.DXF provided

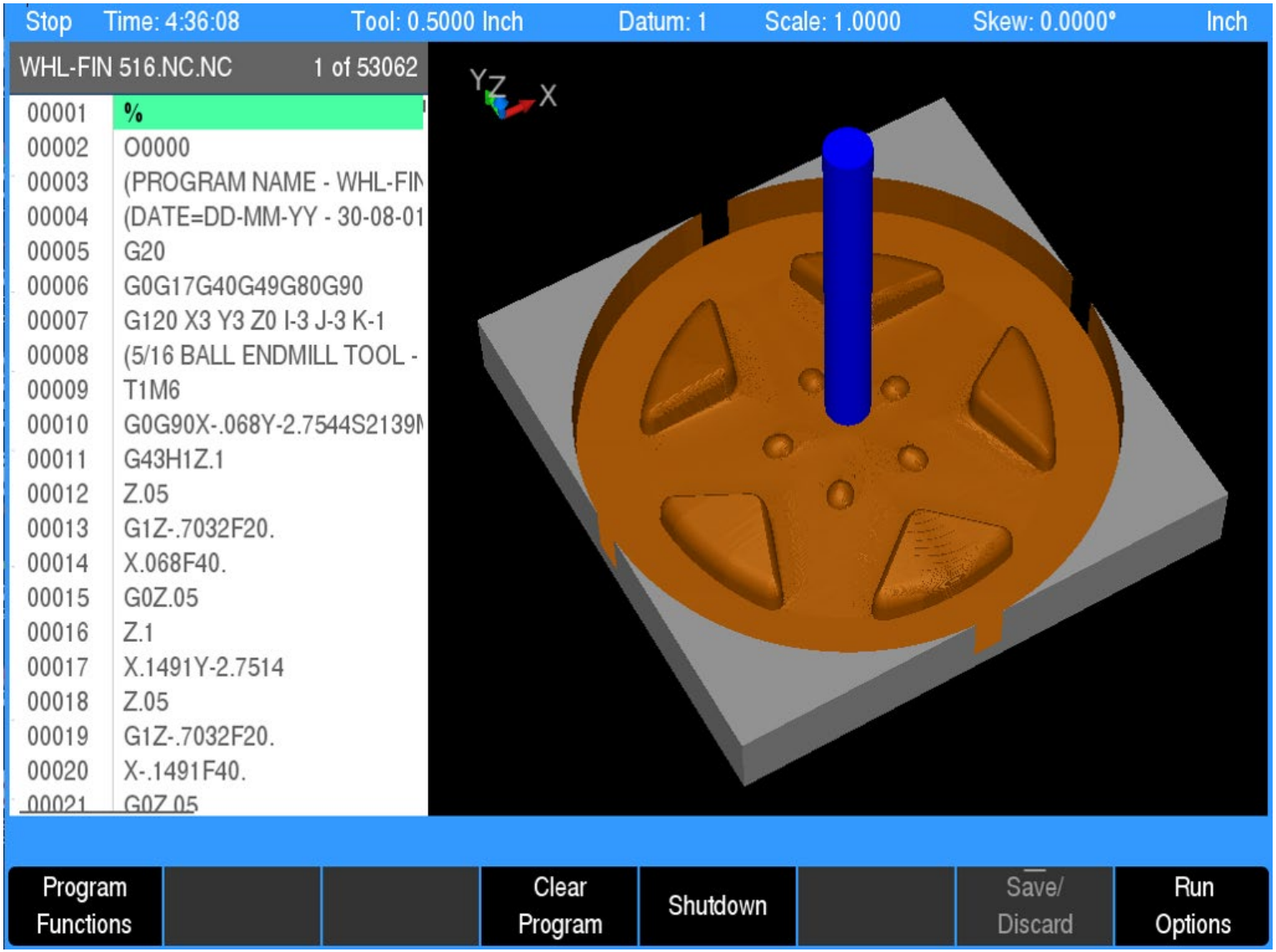


Notes

Exercise 11

G-Code Editing

- NC file provided



Notes

Review

- Startup
- Digital Readout Usage
- Jogging and Positioning
- Tool Library
- Canned Cycle Operations
- Programming
- Geometry Calculator

PPT SLIDE 291

Notes

ACU-RITE®

Made in the USA

www.acu-rite.com

A member of the HEIDENHAIN family of brands