

## Program Parameters

Program parameters are displayed on tabs for General 1, General 2, Milling 1, Milling 2, Holes, Probing, and Performance. The Performance tab in WinMax is active when the SelectSurface Finish Quality option is enabled. The programmer has the option to make changes to any or all of the program parameters and save them as user defaults. The user defaults and original WinMax defaults can be restored by using the appropriate softkey. Parameters can be altered with the Change Parameters data block during program execution. Refer to Conversational Part Programming for details.

Refer to NC Part Programming for information about NC Parameters.

Softkeys on the Program Parameters screen are:

- **SAVE AS USER DEFAULTS F4**—saves the selected field's value as the user-defined default value
- **RESTORE USER DEFAULTS F5**—restores the user-defined values to a field that has been populated with other values
- **RESTORE WINMAX DEFAULTS F6**—restores the WinMax-defined values to a field that has been populated with other values.



Change Parameters program blocks load the user-defined parameters set in Program Parameters.

- **NC Parameters F7**—accesses NC Configuration Parameters. These parameters are available only with NC Part Programming.

## General Parameters 1

These are the fields on the General 1 tab:

- **Retract Clearance**—Determines the Z coordinate to which the Z axis positions before rapid table positioning. This includes a tool moving from one drilled hole to another, or from one milling operation to another (programmed in separate data blocks or generated as a patterns operation).
  - The default is the maximum programmable Z travel. This is the difference between the Z-Axis MAX Travel and the Z-Axis MIN Travel as indicated on the Machine Specifications screen.
  - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

If the next operation has a different Z Start value, the CNC always retracts to the highest dimension. When a Position block is programmed, the tool always retracts to The safety plane programmed as Z Top of the Safety Work Region.

- **Rapid Traverse**—Determines the feedrate that the table (X and Y axes) moves between one point in the part program to the next point in the program (rapid table positioning).
  - The default is 400 ipm (10160.0 mm/min).
  - The Range MAX value is user-defined in the Maximum Rapid Traverse Rate field on the Machine Specifications screen. The Range MIN value is 0.1 ipm (2.54 mm/min).
- **Peck Clearance Plane**—Determines the relative distance to the previous peck level. In conversational programming, the tool retracts to Z Start after each peck. The tool then rapids down to a position which is the Peck Clearance distance above the previous peck level before plunging to the next peck level at plunge feedrate.

Peck Clearance Plane only applies to conversational programs.

- The default is 0.05 inches (1.27 mm).
- The range is 0 through 99.9999 inches (0 through 2514.6 mm).
- **Chord Error**—Determines the maximum distance the cutter deviates from the true arc path.
- **Override Lockout**—Disables the Axis Feed dial on the jog unit of Hurco controls when set to On. The default is Off.

## General Parameters 2

These are the fields on the General 2 tab:

- **Depletion Retract**—Specifies the dimension above the part surface to which the Z axis retracts. The Z axis retracts while waiting for additional data to be transmitted into the current program during execution of an NC part program.
  - The default is 0.005 inches (.127 mm).
  - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

Depletion Retract only applies to NC programs.

- **Interrupt Cycle Z Retract**—Retracts the Z axis to Retract Clearance when you press the Interrupt Cycle console button on a Hurco control.
  - Select No to keep the spindle in its current position when the button is pressed.
  - The default is Yes.
- **First Peck Offset**—Permits modifying the depth of the first peck in milling and hole operations. Use this feature whenever a first peck needs to be deeper or shallower than subsequent pecks.

The permitted range is -10.0000 to + 10.0000 inches or -254.00 to +254.000 millimeters.

The First Peck Offset value is added to the operation's peck depth in calculating the first peck only. Use a positive First Peck Offset value for deeper peck and negative value for shallower peck. A First Peck Offset of 0.000 will run the pecks normally, without any First Peck Offset.

For example, if the peck depth in a drill operation is set to 0.2000 inches and First Peck Offset is set to +0.0500 inches, then the first peck will be 0.2500 inches down from Z start plane and all subsequent pecks will be 0.2000 inches deep. If the First Peck Offset is set to -0.0500 inches, the result is a first peck only 0.1500 inches down from Z start plane and every subsequent peck will be 0.2000 inches deep.

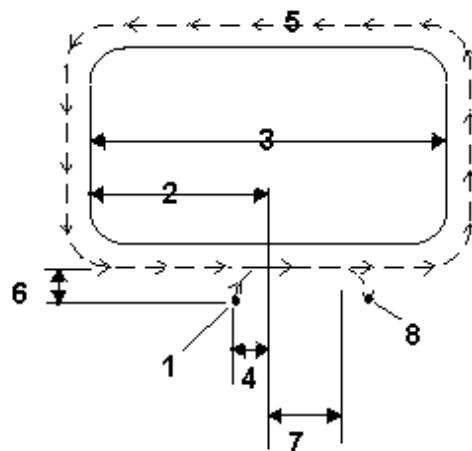
- **Move to Safe Pos During TC**—Indicates whether or not the table will move to the right/front of the machine when the operator is changing a tool. If this field is set to Yes when a part program block calls for a tool change, the table will move out of the way.
- **Include Offset Z in Tool Zero Cal**—Indicates whether or not the Offset Z value in Part Setup is added to the zero calibration value when tool lengths are adjusted. Default is Yes.

## Milling Parameters 1

Milling parameters apply to cutter motions during conversational milling operations only.

Milling 1 Parameters fields are defined as follows:

- Blend Offset**—Determines the distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This field is used in milling circles, frames, and ellipses.
  - The default is 0.1250 inches (3.175 mm).
  - The range is 0 through 1.0 inch (0 through 25.4 mm).
- Blend Overlap**—Determines the distance the tool travels past the entry point before it is withdrawn from the part. This field is used in milling circles, frames, and ellipses.
  - The default is 0.1250 inches (3.175 mm).
  - The range is 0 through 1.0 inch (0 through 25.4 mm).



1	Start Point
2	1/2 X Length
3	X Length
4	Blend Offset
5	Tool Path
6	Blend Offset
7	Blend Overlap
8	End Point

Figure 11. Blend Offset and Blend Overlap

- Finish Feed (%)**—Allows you to specify a different feed for finishing operations without changing the tool. The specified percentage is a multiplier of the feed entered in Tool Setup. This multiplier is applied whenever the tool is entered for the finishing operation of a milling block. (See example below.)



If this parameter is changed in either Program Parameters or with a Change Parameters block, all existing data blocks that are affected by the multiplier will be updated.

- **Finish Speed (%)**—Allows you to specify a different speed for finishing operations without changing the tool. The specified percentage is a multiplier of the speed entered in Tool Setup. This multiplier is applied whenever the tool is entered for the finishing operation of a milling block. (See example below.)

⇒ If this parameter is changed in either Program Parameters or with a Change Parameters block, all existing data blocks that are affected by the multiplier will be updated.

#### Finish Feed / Finish Speed example:

Program Parameters: Finish Speed %=120, Finish Feed %=80

Tool Setup: Speed=5000, Feed=100

When a program block is created, the speed is automatically set to 5000 and the feed is set to 100. When the tool is entered into the finishing operation, the multipliers are applied, and finish speed is set to 6000 (5000 x 120%), finish feed is set to 80 (100 x 80%).

⇒ The Finish Feed and Finish Speed parameters are not applied if the roughing feed or speed in a data block is changed.

Values entered manually into the Finish Feed or Finish Speed fields in the data block take precedence over these parameters.

If the Tool & Material Library option is enabled, separate roughing and finishing defaults can be set for each tool. If finishing defaults are defined for a tool, those values take precedence over the Finish Feed and Finish Speed multiplier parameters.

- **Finish XY**—Determines the amount of material in the X-Y axis direction to be removed by the finish pass.
  - The default is 0.2 mm (0.007874 inches).
  - The range is 0 through 25.4 mm (0 through 1.0 inches).

⇒ Stock is removed up to a **maximum** of 90% of tool diameter.

For example, if 25 mm is entered in the Finish XY field, and the tool has a diameter of 10 mm, the XY stock removed on the finish pass will be 9 mm (90% of the 10 mm tool diameter), despite the number entered in the Finish XY field.

- **Finish Z**—Determines the amount of material in the Z axis dimension to be removed by the finish pass.
  - The default is 0.1 mm (0.003937 inches).
  - The range is 0 through 1.0 inch (0 through 25.4 mm).

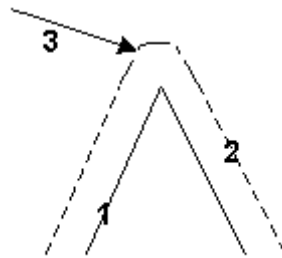
- **Milling Direction**—Determines the milling type. Select Conventional or Climb milling for canned milling cycles (e.g., frame, circle, and ellipse) and for contours (e.g., line, arc).
  - The default is Climb.
  - The choices are Conventional or Climb.
- **Default Pocket Overlap**—Determines the cutter step-over movement in a pocket milling operation. After the first pass, the tool follows a path produced by offsetting the boundary by the tool radius, plus the pocket overlap for each pass while avoiding islands inside the boundary.
  - The default is 50%.
  - The range is 0 through 99%.
- **Cutter Comp Param**—Determines the programmed tool automatically follows the finished contour of the part with cutter compensation. Without cutter compensation, the center line of the programmed tool follows the print line.
  - **Insert Arc**—Inserts a tangent arc to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect).



<b>1</b>	Programmed Tool Path
<b>2</b>	Cutter Compensated Path
<b>3</b>	Using the Insert Arc Parameter

**Figure 12. Cutter Compensation using the Insert Arc parameter**

- **Insert Line**—Joins the cutter compensated lines and arcs as described below:
  - Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.
  - Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.
  - Two arc segments have tangent lines (to the arcs) inserted and extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.



1	Programmed Tool Path
2	Cutter Compensated Path
3	Completed Path Using the Insert Line Parameter

**Figure 13. Cutter Compensation using the Insert Line parameter**

## Milling Parameters 2

The Milling 2 Parameters fields are related to the Helical Plunge Option:

- Mill Plunge Type
- Mill Plunge Ramp Slope
- Mill Plunge Helix Radius
- Finish Plunge Type
- Finish Plunge Ramp Slope
- Finish Plunge Helix Radius
- Operator Specify Pocket Start
- Inward Pocket Plunge Near Center

Refer to Helical Plunge in *WinMax Mill Options* for details about these parameters.

## Holes Parameters

The Holes Parameters fields are defined below:

- **Bore Orient Retract**—Determines the distance the boring tool moves away from the part surface at the end of the boring cycle. Used only when a Bore Orient data block is included in the part program.
  - The default is 0.05 mm (0.019685 inches).
  - The range is 0 through 99.9999 inches (0 through 2514.6 mm).
- **Drill Dwell**—Determines the pause (dwell) in seconds before the tool retracts at the bottom of a drill operation. The most often changed Holes Parameter is Drill Dwell. This parameter controls the length of time the drill stays at the bottom of a hole after it has drilled the hole. This parameter is not used for NC programs.
  - The default is 0.5 seconds.

- The range is 0 through 20 seconds. (Set this to 0.0 seconds, and the drill immediately pulls out of the hole after it is drilled.)
- **Bore Dwell**—Determines the pause in seconds before the tool retracts at the bottom of a Bore operation. This parameter is not used for NC programs.
  - The default is 1.0 seconds.
  - The range is 0 through 20 seconds.

## Probing Parameters

The Probing Parameters are:

- **Automatic Tool Monitoring**—indicate if tools that were calibrated with the probe should be automatically checked with each tool change.
- **Zero Cal (Length) Tolerance**—indicate the zero calibration (tool length) used when checking for a defective tool.
- **Diameter Tolerance**—indicate the diameter tolerance used when checking for a defective tool.
- **Retain Probed Part Setup**—allows the probed part setup and/or tool calibrations to be retained for new program runs. Choices are:
  - **Do Not Retain**—no updates will be made to part zero or tool lengths.
  - **Retain All**—retains part setup and tool calibrations.

⇒ Previous versions of WinMax may have included additional selections which have subsequently been removed; older programs that contained a selection other than **Do Not Retain** or **Retain All** will be converted to Retain All upon opening, and a message will appear informing the user.

For example, the following table shows Part Setup values for a sample program before and after a Probed Part Setup is executed:

Original Part Setup values	Probed Part Setup values
Part Zero X = 20.0000	Part Zero X = 19.1234
Part Zero Y = 10.0000	Part Zero Y = 11.1111
Probe Z = 15.0000	Probe Z = 16.5555
Skew Angle = 0.0000	Skew Angle = 2.0045
Zero Cal = 18.0000	Zero Cal = 17.4444
(Tool Setup)	(Tool Setup)

If Retain Probed Part Setup is set to **Do Not Retain**, the original values for Part Setup and Zero Cal will be restored. Part Setup and Tool Setup will contain the original data (20, 10, 15, 0, and 18).

If Retain Probed Part Setup is set to **Retain All**, Part Setup and Tool Setup will contain the probed values (19.1234, 11.1111, 16.5555, 2.0045, and 17.4444).





OFFSET Z is not affected in any way by the probe block or the parameter setting.

## Performance Parameters

Surface Finish Quality (SFQ) is enabled with the SelectSurface Finish Quality option. SFQ parameters can be modified in either Conversational or NC programming. The default SFQ for roughing is 80 and finishing is 20. Recommended values are:

SFQ	Desired Result
1-20	High precision parts /finishing
21-79	Good surface quality / finishing, semi-finishing
80-100	High throughput / roughing

**Table 4. Recommended SFQ values**



If SelectSurface Finish Quality is not enabled, conversational roughing tools use SFQ of 80 and conversational finishing tools use SFQ of 20; NC default is 50.

Smoothing Tolerance specifies the maximum allowable deviation from the tool path. The range is 0.0000 to 0.0500 inches (0.0000 to 1.2700 mm); default is 0.0005 inches (0.01270 mm). This corresponds to NC code G05.2.