SPARTAN™

Machine Training Manual

Table of Contents

Version 1.26.2026

Section 1

Measuring Tooling and Filling in the Tool Library

Overview

The user will need to gather information from the tool set after assembly. This can be done manually or by using an off-line tool measuring system (various tooling vendors offer this as a option to save time on personally measuring tooling). If a user has an offline measuring system, see "Tool Measure Settings" in the operation manual for instructions on importing tool data from an off-line tool measuring system.

If the tool(s) are being measured manually, the user should only measure tools using a digital caliper vs a tape measure, to ensure the highest level of accuracy. All tools will need to be 'fine-tuned' following the procedure "Tooling Setup" (section 3 of the Machine Training Manual).

Tool Data

Use the following information in the procedure below to enter tool data. The user can make copies of the worksheet near the end of this manual to record your tooling data.

- **Tool Description:** Unique name for each tool for organization. Example - a core tool with a 2" diameter- the user should name the tool "***CORE 2.0***". A shaped tool would be named with a tool position in the set, and what profile it is. Example: ***P1 T30 R10*** for a position #1 30mm T profile with a 10mm radius.
- **Mill or Drill:** This distinguishes the use of the tool in the machine and in a "Style"
- **Tool ID #:** This will be the digital location of the tool in the "Tool Library" list on the machine
- **Tool Pocket Number:** This will be the physical position in the tool rack (in a pocket clip)
- **Tool diameter:** The measurement of the diameter of the tool, for core, mill, and gauge tools this is the *active cutting diameter* of the tool. For shaped tools- this will be the diameter of the smallest radius.
- **Tool length:** The length measurement is dependent on what type of tool is getting measured. All measurements are made from the gauge line on the tool holder (near the holder's shoulder), and to the "used bottom" of the tool (see examples following for details)
  - End tools such as the core, gauging, and mill tools measure to the end of the tool
  - Shaped tools measure to the Z zero (Ride height) on the tool.

- Then **ADD** the gauge adder (the distance from the shoulder of the holder to the spindle face) to the measurement. This will approximately adjust to the gauge line on the cone and is different on certain manufacturers tool holders' cones but is often around .070-.090".
- **(XY) Stock REMOVAL*:** The amount of material the manufacturer recommends this tool to remove (given to the user in a manufacture's tool spec sheet along with RPM and feed rate settings)
- **(Z) Stock REMOVAL*:** The amount of material the manufacturer recommends this tool to remove in Z. Certain tool-sets do not use this value as it is built into the tool's shape.
- **(Z) Depth Offset:** Vertical dimension from the measured length to the Z=0 point, and varies by profile and tool type
- **Serial Number (OPTIONAL):** Some tooling has a serial number; this space is for the ability to record this for the tool.

Measuring diameter

Measuring length

Measuring Z zero

Measuring Utility Tools (Core, Milling, Zee, and Gauge/Stubbing)

1. **Tool diameter:** Measure the diameter of the tool in the area that the diamonds will be used for cutting.
2. **Tool length:** Measure from the top of the tool holder's shoulder to the end of the tool, then ADD the gauge line adder (often around .070-.090") to the measurement. This will approximately adjust to the gauge line on the cone. *Note: There is not a time when the user will want to include the length of the bolt if it goes through the tool.*

(Core Tool)

Tool Length

Tool Diameter

(Mill Tool)

Tool Length

Tool Diameter

(Gauge & Zee Tools)

Tool Length
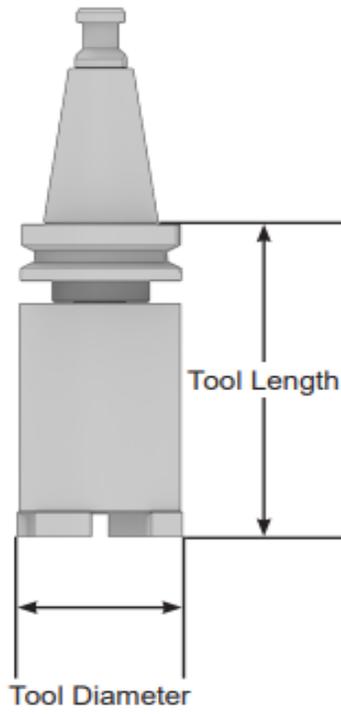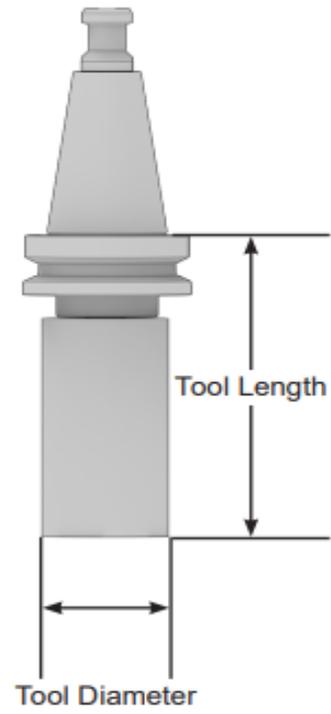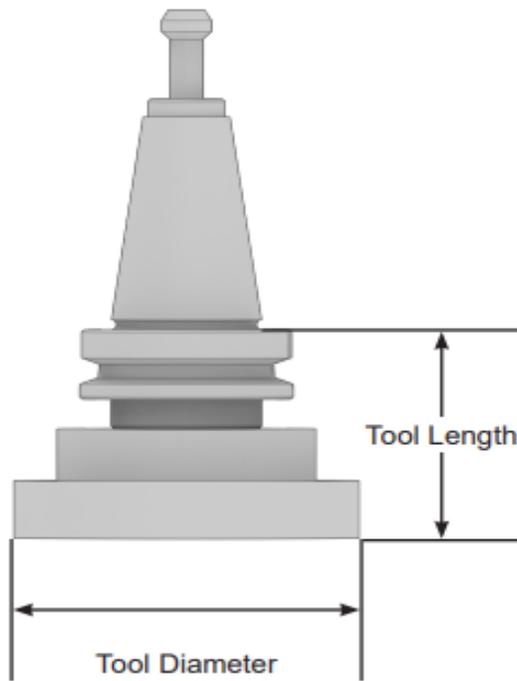
Tool Diameter

Measuring Shaped Tools

The following both explain and show examples of measuring various shape tools, measuring shape tools other than those shown will be similar.

1. **Tool diameter:** Always measure the SMALLEST diameter of the tool on a shaped tool.
2. **Tool length:** Also called the Ride Height, it is the point located on a shaped tool where the lower edge (polished face) of the material will contact the tool.
   a. Measure from the top of the tool holder's shoulder to the Z zero (ride height) of the tool
   b. then ADD the (gauge line adder (often around .070-.090") to the measurement. This will approximately adjust to the gauge line on the cone. *Note: There is not a time when the user will want to include the length of the bolt if it goes through the tool.*

Measuring Encapsulating Tools

Follow this example for measuring encapsulating tools.

(Encapsulated Tools)

Tool Length

(Material)

(Z zero or ride height)

Tool Diameter

encapsulating tools.

Measuring Non-Encapsulating Tools

Follow this example for measuring

(Non-Encapsulated Tools)

Tool Length

(Material)

(Z zero or ride height)

Tool Diameter

Spartan Tool Library Worksheet							
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Library Location Number	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	**Tool Pocket Number**	**Tool Diameter**	**Tool Length**	**(XY) Stock REMOVAL***	**(Z) Stock REMOVAL***	**(Z) Depth Offset**	**Serial Number**
This information is specific to the tool manufacturer.							

Entering Tool Data

The Spartan user will be using the ***Spartan Tool Library Worksheet*** to add tools into the **Tool Library**.

NOTE: The RPM and feed rates will be entered in the "Style" Library later.

Add or Edit a Tool

1. If the Spartan is not powered up, do so now.
2. Log in a user account with the correct permissions allowed to edit the tool library.
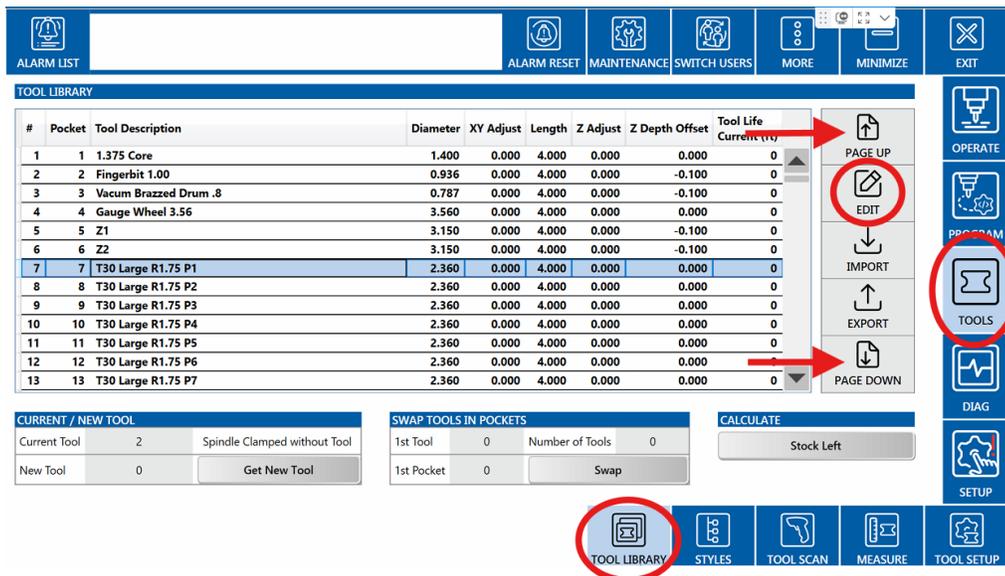
3. Select the **Setup** primary tab then selects the **Tool Library** secondary tab.
4. Use the **Page Up** or **Page Down** buttons until you see the Tool ID (library line number) you are looking for. Select the line number to edit, when selected, the tool number will be highlighted.
5. Select the Edit button
6. Enter tool data needed by the library
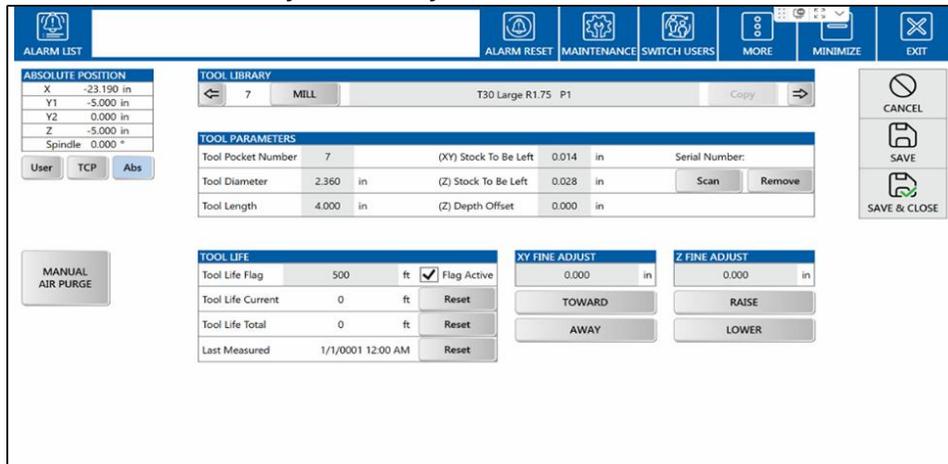
NOTE: You can copy tooling data from a previously entered tool by using the left or right arrows to the Tool ID # and press the Copy button. A popup will open allowing you to select what data you wish to copy and what location to copy to.
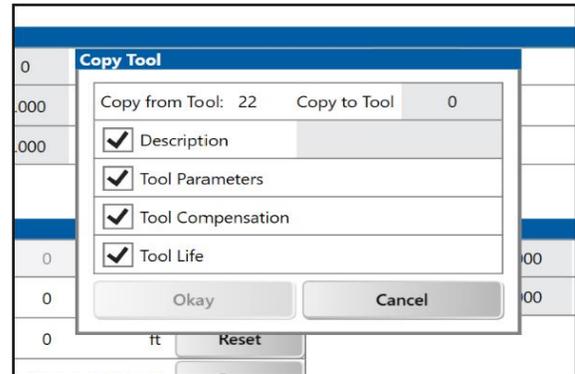
Extra Information in the library

- **Mill or Drill**
    - All tools in the library are classified as either "**MILL**" or "**DRILL**". Since most tools are milling tools- the screen opens defaulted to "MILL". Tapping the box toggles the display between "MILL" and "DRILL.
- **Tool Life**
    - If you want to use the OPTIONAL Tool Life indicator, select the check-box, if not make sure checkbox is not selected.
    - Press the Reset buttons to reset tooling life for new tools or after dressing a tool

Once all the tool data has now been entered; to finish:

- Select "Cancel" to exit the tool edit screens without saving your data, or
- Select "Save" to add additional tools. *NOTE: navigate between tools with the forward and back buttons.* **Save before switching tools**.
- Select "Save & Close" to save and exit the tool edit screen.

"Stock Left" Calculation Compensation Explained [(click to watch)](#)

A set of shape tools usually consist of seven tools: four metal shaping tools and three polishing tools. Each metal shaping tool removes a certain amount of material which is determined by the tool's manufacturer. "Stock Left" will determine the amount of material-to-leave for all tools that follow that tool. The objective is to be at the finished edge after the final metal shaped tool has been run. Thus a tool's material-to-leave value is determined by adding the removal amount of all tools that follow that tool.

The "Stock Left" calculator calculates the values for each tool and then saves the value into the tool's library information. Below is an example of how a tool's compensation value is computed. The goal, for the "metals" in the set, is to be at the finished size, on the edge being ran, after the final metal tool has shaped the material.
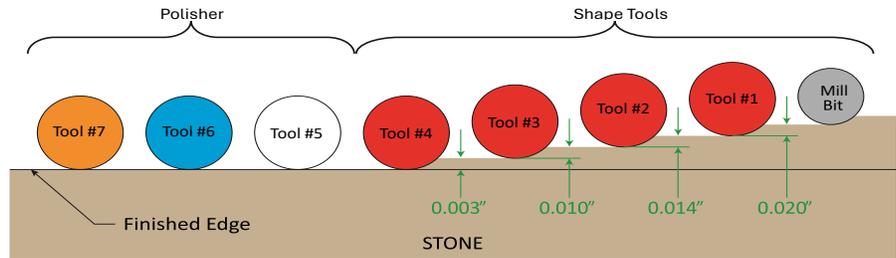
In this example, the tooling manufacturer has specified all removal rates for the metal tooling as listed below. *NOTE: removal rates and the number of tools and tool sets can vary. Always check the manufacturer's recommendations before making any calculations.*

Tool #1 removes 0.020

Tool #2 removes 0.014

Tool #3 removes 0.010

Tool #4 removes 0.003

Tool Number	Compensation Amount	NOTE
4	0.000	After tool #4, the material is reduced to the finished edge. The polishers do NOT remove a significant amount of material.
3	0.003	Material removed by tool #4
2	0.013	Sum of the material removed by tools #3 and #4 (0.010 + 0.003 = 0.013)
1	0.027	Sum of the material removed by tools #2, #3, and #4 (0.014 + 0.010 + 0.003 = 0.027)

NOTE: The mill bit's "stock left" is NOT normally included in tool selection when using the compensation function.

"Stock Left" for mill bit: Sum of the removal amounts of tool #1 - #4 (0.020 + 0.014 + 0.010 + 0.003 = 0.047). The value of .047" would typically reside in a "style" rather than in the library because if the user was going to run the same tool for a different purpose, such as cutting a seam with the same mill bit used in a sink set, the user may not want to have stock to be left or need a different value for less tools in the seam set.

Using the "Stock Left" Calculator (click to watch)

Tool sets should, typically, be in consecutive order in the tool library to be able to use the compensation calculator function.

1. From the Tool Library screen press the **Stock Left** button.
2. Select the "1st Tool" field and enter the Tool ID # of the first tool in the set.
3. Enter the number of tools in the set in the "Number of Tools" box (e.g., the first tool's # is 1 with 7 tools total).

4. Enter the Horizontal and Vertical Removal values for Tools that are *metals* in the set. Some tools do not require vertical removal by the tooling manufacturer, making this it optional- but optimal for tunning of tooling
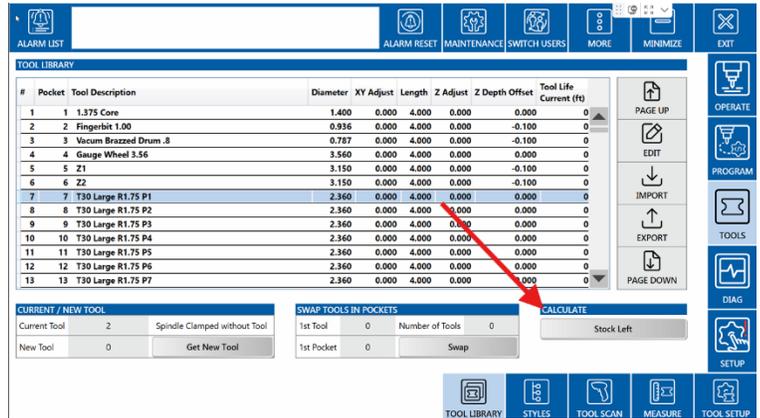
Number of Tools	7		1st Tool	7
Tool Number	XY Removed	XY Left	Z Removed	Z Left
7	0.020	0.027	0.010	0.014
8	0.014	0.013	0.007	0.007
9	0.010	0.003	0.005	0.002
10	0.003	0.000	0.002	0.000
11	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000
13	0.000	0.000	0.000	0.000

Verify the values are correct and then press the "Accept" button to save the values and return to the Tool Library screen or press the "Cancel" button to exit without saving.

Okay	Cancel

Section 2

Creating MACHINING STYLES

Overview

Machining "Styles" allows the user to program and control the Spartan. The "Styles" are a saved group of tools that can be ran as a set with running parameters (RPM, feed rate, stock removal rates, etc.) that are provided by the tooling's manufacturer. The user will need to use a defined "style" to run a "Tool Setup" procedure (section 3) and to apply to sinks or seams programmed at the machine (see Section 4 and section 5).

NOTICE
— ALWAYS USE THE MANUFACTURERS SETTINGS FOR YOUR TOOLS —
THE SETTINGS SHOWN IN THIS PROCEDURE AND ELSEWHERE IN THE MANUAL ARE USED AS EXAMPLES ONLY! DO NOT USE FOR YOUR TOOLING!

The following procedure explains the basic steps to define several different machining styles. The user will need to follow *ALL* manufacturers' recommended settings for the tools at any time on the Spartan.

Different Usage Means Different Styles

Styles can be made up of a single tool or a group of tools, but all tooling that is used to program on the Spartan HMI (control) needs to be entered as a style.

- A style example with a single tool may be a drilling style for faucet holes.
- An example style with multiple tools will be the most common for profiling and seams. However, in both profiling and seams, the user could have multiple similar styles for various uses.
    - Different materials may require different feed rates
    - Certain materials may require less tools (thin stone may not need a gauge wheel)
    - If the user has more than a .250" on an edge to remove multiple passes could be used with the Z wheels, or a finger-bit could cut off excess

Creation of a Style

After turning on the HMI's Park Industries user interface, the user will;

- First select the "Tools" main tab on the right-hand side of the screen
- Second select the "Styles" sub tab on the bottom of the screen
- Then choose the "ADD" option near the right to open a blank "style" window
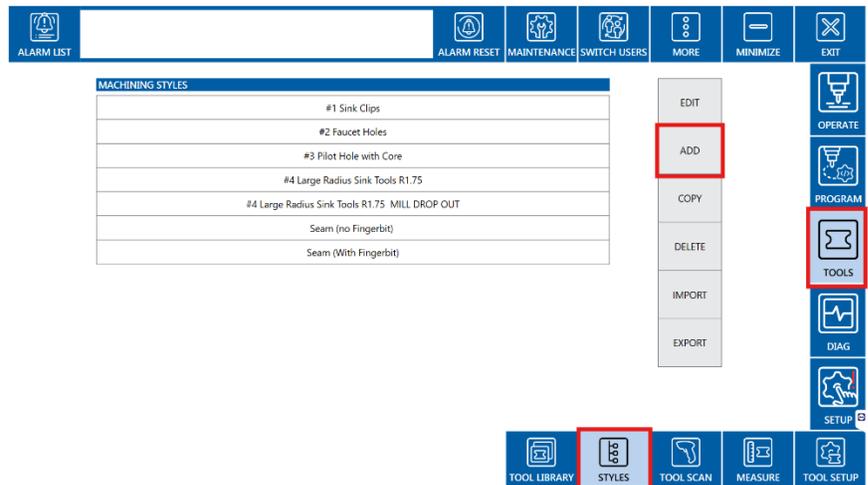
Note: The user would be able to edit or copy existing styles if needed. The user would also be able to save or restore styles from a USB or network by using "Export" or "Import "buttons.

The MACHINING STYLE Window

It would always be in the user's best interest to have an organizational structure in the Spartan's styles. That includes in the name and order of the tools

- **DESCRIPTION-** this is the name this style will use in the different areas of the HMI's programming screens. The style list organizes itself in an alphanumerical sorting that can be used to keep similar styles in groups.

- **EDIT-** allows the user to change running parameters of each tool in the style
- **ADD-** allows the user to increase or change out the tools in the style (once the user is done adding, the screen will immediately go into **EDIT**)
- **DELETE-** allows the user to delete a tool from the style
- **MOVE UP & MOVE DOWN-** allows the user to organize the desired running order of the tools in the style
- **Close (looks like a blue X)-** this acts as a close and a save feature

14

Filling in Tooling Manufacture's Running Parameters

The tool data entered will tell the Spartan how to run this specific tool when using this style, you can use the same tool for multiple styles with very different settings, for example you can have a style where a core bit will drill through the material or you can have a style where that same core bit will stop at a preset distance before drilling through.

The only item that can be defined for the style group as a whole is the Description.

      Description: this is the name this style will use in the different areas of the HMI's programming screens. The style list organizes itself in an alphanumerical sorting that can be used to keep similar styles in groups. Unique names for each Style is helpful for understanding the tools used and where the style should be used.

An example name for a style could be **"Small Radius T30R3 20mm R.625 MILL DROP OUT"**

This name would suggest:

- It is for sinks with tight/small corners that are only 5/8 inch or larger
- It is a T profile meant for 30mm stone (or 3cm)
- It would be likely to contain both a drill and finger-bit pass to remove a sink drop for a counter that would not contain a precut hole from a saw/ saw jet

Items defined for each individual tool in the style group are:

- **Safe Rapid**: The height that the tool will rapid at in the X and Y axis
- **Rapid Down To:** The Z height that the tool will rapid to in Z typically set to 4, unless the tool is a core, mill, or gauge wheel, then use 2.
- **Final Depth:** The Z height that the tool will cut at, usually zero unless coring, or gauging.

The following information is specific from the tool manufacturer.

- **RPM*:** The speed that the tool will spin at, use manufacturers values.
- **Feed rate*:** The speed that the tool will travel in the X and Y axis, use manufacturers values.
- **Stock Left Adder.*:** The amount of material the tool will leave behind. Typically zero unless running a pre-pass, mill bit, or Z wheel.
- **Cut At Tool Center**: When checked the tool path will use ONLY the diameter (at the time of program creation) set in the tool library and the "Stock Left Adder" in the Style. When NOT checked the tool path will use the diameter *PLUS* any values in the Stock Left *and* the Style. Stock Left values are usually calculated when using the Compensation Calculator. *Note: mostly, only a gauge/stubbing wheel uses tool center in sink styles*

The following items have separate entries for the lead in and lead out.

- **Lead In & Lead Out:** In this section you can enter the lead in & lead out data used when creating tool paths for this tool in this style.
- **Overlap:** Normally set to .5 inch.
- **Link Start and End Position:** When checked the starting and ending positions will be set to the same location.

  - **Type:** Both (Line and Arc) are typically used for most cuts
  - **Angle:** Approach and Retract angle is normally 90 on inside Geometry (sinks)
  - **Line Multiplier**:** 1.5 times the tools radius is the minimum value recommended.
  - **Arc Multiplier**:** 1.5 times the tools radius is the minimum value recommended.
  - **Copy Lead In / Out buttons:** makes the lead in and out use the same parameters

** Based on the tool radius of the tool being used

EXAMPLE drilling tool-

EDIT TOOLING PARAMETERS
⇐   1   DRILL   1.375 Core   ⇒

TOOL INFO

| Diameter | 1.400 in | Stock Left | 0.000 in |
| Length | 4.000 in | Z Depth Offset | 0.000 in |

- the TOOL INFO window should just have a *Diameter* and *Length* from the Tool Library. *Stock Left* would be left at 0.000 in since it is not getting used for a milling bit, and *Z Depth Offset* would typically be set to 0.000 in as well.

LEAD IN & LEAD OUT

| Overlap | 0.500 | in | ☑ Link Start and End Position |

LEAD IN			LEAD OUT	
Type	Line-Arc	>>	Type	Line-Arc
Angle	90		Angle	90
Line Multiplier*	1.500	<<	Line Multiplier*	1.500
Arc Multiplier*	1.500		Arc Multiplier*	1.500

*Based on tool radius   *Based on tool radius

TOOL STYLE

Safe Rapid	5.000	in
Rapid Down To	4.000	in
Final Depth	0.200	in
RPM	2200	RPM
Feed Rate	2.0	in/min
Stock Left Adder	0.000	in
☑ Cut At Tool Center		

- the TOOL STYLE window will contain the program's running heights and tooling manufacturer information – which may need to be converted into inches depending upon the brand of tool. *(NOTE: on a drill and a Gauge wheel, the final depth will typically not be at 0.000 inches. The user would want to not go all the way through for faucet holes, or go plenty far through for a sink pilot hole (for a finger-bit to follow))*

Working Parameters for Thin Wall Core Bits – Granite and Engineered Stone

Diameter	R.P.M.	Feed Rate MM/Minute
25mm	2,600	30–40mm
30mm	2,400	40–50mm
32mm	2,400	40–50mm
35mm	2,200	50mm
38mm	2,200	50mm

Example of Tool Specs from Tooling Manufacturer for a Core Drill

.EXAMPLE rougher tool-

EDIT TOOLING PARAMETERS
⇐   2   MILL   Fingerbit 1.00   ⇒

TOOL INFO

| Diameter | 0.936 in | Stock Left | 0.000 in |
| Length | 4.000 in | Z Depth Offset | -0.100 in |

- The TOOL INFO window would have a *Diameter* and *Length* from the Tool Library, but could also have a *Z Depth Offset* depending on how the length was measured. *Stock Left* would typically be left at 0.000 in the library. If it was for a cooktop it could stay without leaving stock in the style, but in a sink- a rougher would instead use *Stock Left Adder* (in the TOOL STYLE window) to leave material for a profiling set.

LEAD IN & LEAD OUT

| Overlap | 0.500 | in | ☑ Link Start and End Position |

LEAD IN			LEAD OUT	
Type	Line-Arc	>>	Type	Line-Arc
Angle	90		Angle	90
Line Multiplier*	1.500	<<	Line Multiplier*	1.500
Arc Multiplier*	1.500		Arc Multiplier*	1.500

*Based on tool radius   *Based on tool radius

TOOL STYLE

Safe Rapid	5.000	in
Rapid Down To	4.000	in
Final Depth	0.000	in
RPM	5500	RPM
Feed Rate	12.0	in/min
Stock Left Adder	0.060	in
☐ Cut At Tool Center		

- The TOOL STYLE window will contain the program's running heights and tooling manufacturer information – which may need to be converted into inches depending upon the brand of tool.
- The LEAD IN & LEAD OUT window is where the user can set how tools should enter a sink cutout.

Diameter	Diamond Height	Material Hardness	R.P.M.	Feed Rate/Minute
22mm	35mm	3-5	5,500-6,000	30–40cm

Example of Tool Specs from Tooling Manufacturer for a Finger-bit

EXAMPLE profiling (shaped) tool-

- The TOOL INFO window would have a *Diameter* and *Length* from the Tool Library, but could also have a both the *Stock Left* *Z Depth Offset* depending on how the length and width was measured. The *Stock Left* may not be used if the tool measurement comes from an offline measuring system that is altering the *Diameter* of the tool to accomplish the same removal amounts

EDIT TOOLING PARAMETERS
14   MILL   T30R3 sink 20mm R.625 P1

TOOL INFO

Diameter	0.790 in	Stock Left	0.014 in
Length	4.000 in	Z Depth Offset	0.000 in

LEAD IN & LEAD OUT

Overlap 0.500 in   ☑ Link Start and End Position

	LEAD IN			LEAD OUT	
Type	Line-Arc		Type	Line-Arc	
Angle	90		Angle	90	
Line Multiplier*	1.500		Line Multiplier*	1.500	
Arc Multiplier*	1.500		Arc Multiplier*	1.500	
*Based on tool radius			*Based on tool radius		

TOOL STYLE

Safe Rapid	5.000	in
Rapid Down To	4.000	in
Final Depth	0.000	in
RPM	6500	RPM
Feed Rate	39.0	in/min
Stock Left Adder	0.000	in
☐ Cut At Tool Center		

- The TOOL STYLE window will contain the program's running heights and tooling manufacturer information – which may need to be converted into inches depending upon the brand of tool. Be careful not to double up on the *Stock Left Adder* with the *Stock Left* - from the TOOL INFO window.
- The LEAD IN & LEAD OUT window is where the user can set how tools should enter a sink cutout.

Position	Slab Thickness	RPM	Stock removal	Feed Rate/Minute
Position 1 METAL	2cm–4cm	6,500–7,000	Shaping	31–39" (800–1,000mm)
Position 2 METAL	2cm–4cm	6,500–7,000	0.0078–0.0118' (0.2–0.3mm)	31–39" (800–1,000mm)
Position 3 METAL	2cm–4cm	6,500–7,000	0.0078–0.0118" (0.15–0.2mm)	31–39" (800–1,000mm)
Position 4 METAL	2cm–4cm	5,500–6,000	0.0019" (0.05mm)	39" (800mm)
Position 5 ERASER	2cm–4cm	4,200	0.001–0.002" (0.03–0.05mm) Pressure	31–39" (800–1,000mm)
Position 5 EXPRESS	2cm–4cm	3,300	0.001–0.002" (0.03–0.05mm) Pressure	47–59" (1,200–1,500mm)
Position 6 EXPRESS	2cm–4cm	3,300	0.001–0.002" (0.03–0.05mm) Pressure	47–59" (1,200–1,500mm)
Position 7 EXPRESS	2cm–4cm	3,300	0.001–0.002" (0.03–0.05mm) Pressure	47–59" (1,200–1,500mm)

Example of Tool Specs from Tooling Manufacturer for a Shaped Tool

Section 3

Tool Setup

Overview

The "Tool Setup" screen is where the user of the Spartan can create a program specifically for setting up tooling that will be used for edge details. Such tooling could be simplified as tools that use their diameter to cut the stone such as a Z wheel or a profiling tool, and even a gauge wheel – but not a core bit or drill bit or tools that are designed for cutting on the bottom of the tool. The goal in this screen, for the user, would be to run a test edge to verify that the tool's diameters, lengths, and the amount each tool needs to take off the stone (removal amount) is correct before running sinks or seams. *Note: the tool setup program on this screen is designed to run on a piece of stone that is 22-inches deep and is the correct thickness for the user's tooling.*
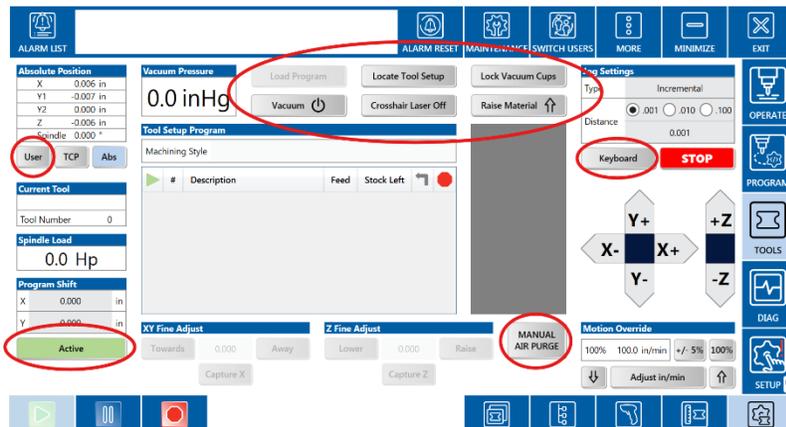
Tool Setup Program Window

1. **Machine Style**- select the style for testing (click in the grey field next to "Machining Style" to see available styles)
2. **"Play" (looks like a green play button)** - allows the user to select which tools in the style should run. *Note- drills or core bits cannot be selected.*
3. **# Column** - shows which tool Number from the library will be used

4. **Description** - shows which tool, associated with the ID, will be used
5. **Feed** - shows the feed rate set in the style
6. **Stock Left**- shows the amount the tool should be leaving from the finished edge once all tools are ran
7. **Leave Early (looks like a left turn arrow)** – can be checked to set staggered paths to allow the user to measure verify how much each tool is touching the stone
8. **Stop Before (looks like a stop sign)**- can be checked to have the Spartan machine stop the tool before running down the edge of the stone. This is used to visually verify the diameter and length of the tools before running to help prevent

19
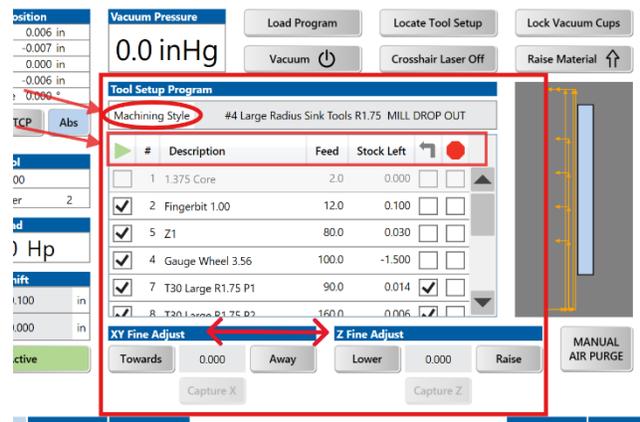
costly measurement mistakes. The "Capture X" and "Capture Z" are designed to work with the Stop tool as they will calculate the difference from the end point of the program (made from the Stop Before button) and an adjusted tool location to offer a different way of adjusting tooling than the "Fine Adjust" buttons.

9. **Fine Adjust Buttons**- once clicking on a tool in the style, these buttons allow a tool length (Z Fine Adjust) and diameter (XY Fine Adjust) to be corrected without the need to leave the "Tool Setup" screen.

Screen Buttons useful for tool setup WHILE RUNNING

- **Crosshair Laser On/Off**- turn the laser used for locating on or off
- **Locate Tool Setup**- moves the machine's laser to the location recommended to place your *MATERIAL* for tool setup. *Note: the tool setup program will run on the left side of the stone. This is not for the Pod Location!!!*
- **Raise Material/ Lower Material**- raises and lowers the pneumatic roller table
- **Lock Vacuum Cups**- activates the brakes on the movable sink vacuum cups
- **Vacuum**- activates the vacuum system
- **Activate for Program Shift**- can be used to run the program in a different location other than default (wanting to run further from pods, or need to bump tool program into stone for recutting)
- **User**- once activated, the machine will show the location of the spindle in reference to the point the "User" button was first selected
- **Keyboard**- allows the user to use the arrow keys on the keyboard and "page up" or "page down" keys to control the XYZ axis. The "+" and "-" on the NUM PAD control the speed/ movement amount . Use of this button with the door open will always change the Jog settings to "Incremental" (displayed picture)

- **MANUAL AIR PURGE**- causes the spindle to purge water from inside the spindle with air pressure. *Note: Do NOT have any person's face by spindle when performing!*

20
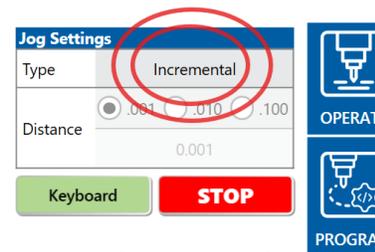
The Process of "Tool Setup"

SPARTAN allows the user to set staggers, stop tools in an area to visually verify diameters and lengths of tools, and adjust tooling to make for perfect running conditions.

The User will need to be signed in at least as a "Maintenance User" to complete tool setup.

Note: *This process will assume tools are mounted properly on 40 taper tool holders and that the following will be done before attempting to run a tool setup program.*

1. *The HMI's "TOOLS→TOOL LIBRARY" screen has been filled in for selected tools*
    a. *Diameters*
    b. *Lengths*
    c. *Stock To Be Left (both XY & Z)*
    d. *Offsets*
    e. *Pocket location (if being used)*
2. *The HMI's "TOOLS→STYLES" screen has been filled in for selected tools*
    a. *Named "Style"*
    b. *Tools added and ordered correctly*
    c. *"Tool Style" is filled in with correct cutting parameters*

First Step- Selecting your Machining Style [(click to watch)](#)

- Click "Machining Style" *(click in the grey field next to "Machining Style" to see available styles)*
- Select the Machining Style needing to be run in a tool setup program
- Click "Accept"
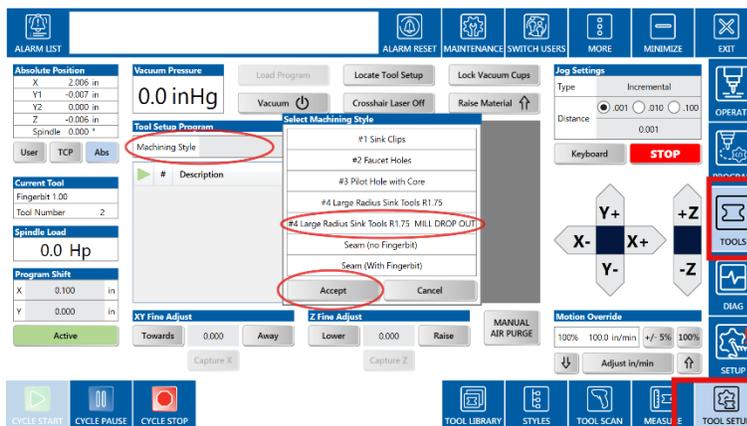
Second Step- Setting The Metals to Run First… and why [(click to watch)](#)

Some tool styles do not require a polisher (like a seaming tool set, or a cooktop cut out with a core bit and a finger bit). In those scenarios, the user can have all tooling selected to run in the "Tool Setup Program" window, other than a drill tool. If a set has polishing tools and the tools of the style have never been set up before- it would be wise for the user to ensure that the metals are properly set up first as metal tools are slightly more forgiving to improper tooling measurements. Metals also lay the foundation for proper polishing, which is why it would be typically recommended not include the polishers right away.

- **Selecting tools to Run**- keep a checkmark on the left-hand side, under the green play button in the "Tool Setup Program" window, for just your utility tools (Z Wheel rougher and Fingerbits etc.) as well as, if the user is doing a profile tool set, the first positions of the profile that are metals (typically the first 4-5 tools depending on what set was purchased from the tooling vendor).
- **Uncheck the polishing tools**- some tooling does not require a polisher (like a seaming tool set). This step would not be necessary for those styles.
- **Leave Early**- other than for the first tool and a gauge wheel, click the "Leave Early" button (looks like a left turn arrow) to set a staggering of the tools on the edge of the stone. This stagger will allow the user to be able to run the tool set up program without needing to manually stop each tool. This will also allow the user room to measure the appropriate stock removal rates that are designated by the tooling vendor. *Note: This stagger is designed to run on a piece of stone that is 22-inches deep and is the correct thickness for the user's tooling*.
- **Stop Before (optional but recommended)**- The user can use the "Stop Before" stop sign on one tool at a time. This function, if checked, will cause the tool that contains a stop sign to have the machine complete the lead-in portion of the toolpath -to be in line with the stone- but stop before actually touching the stone. This will cause the program to end and go no further allowing the user to open the door and visually verify
    - that the tool looks to be removing the correct amount of material
    - that the tool looks to be at the correct height in the stone

Note: Again, this is based off that a 22 Inch set up piece of stone of the correct thickness. For the program to recognize any changes selected- "LOAD PROGRAM" must be clicked to re-write the correct selections back into the NC of the HMI.

- **Load Program**- must be selected/ reselected anytime the user has made a change to the options in the "Tool Setup Program" window that need to be implemented before the user can choose "CYCLE START".

Third Step- Setting The Stone in the Correct Location [(click to watch)](#)

The size of stone preferred for setting up tooling on the SPARTAN is 21-22 inches deep by 36 inches (or longer) to allow the use of vacuum cups. Typically, all three movable cups would be pushed toward the center of the machine, with the actual tool setup program running 18.100" to the left of the center datum (this is a little over 1 inch to the left of the left vacuum cup if the cup was moved all the way in on its bearing rail towards the center of the machine).

1. Select "Crosshair Laser On" till the laser turns on
2. Click on the "Locate Tool Setup" button on the top middle of the screen with "Program Shift" activated if necessary
3. PULL OUT ANY ROLLER WHEELS THAT WILL INTERFERE WITH TOOL SET UP PROGRAM (in the default position, most tools will not need the rollers removed)
4. Push all vacuum cups toward the center of the machine
5. The user should remove all body appendages from the machine- then press the "Raise Material" button to have the roller table and sink trap door come to the raised position
6. Load a test piece of at least the minimum recommended size (21x36 Inches)
   a. Roll/ place the test piece against the roller fence towards the back of the machine and keep the left edge of the stone in line with the laser edge.
      i. This will give the user 1.000 inch of "cup-to-stone edge clearance" to set up tooling
      ii. If tooling that requires a larger "cup-to-stone edge clearance" is being ran for tool setup- the user may use "Program Shift" to adjust the location of the tool set up program (likely in the X Negative direction) and the "Locate Tool Setup" button will show the new location of where to place the left side of the stone
   b. Hold the piece against the rollers while pressing the "Lower Material" button *(Note: always be careful of any pinch points while handling the material in any machine to avoid personal injury!!!!!)*
   c. Double check that all vacuum pods are positioned far enough under the stone to not leak vacuum pressure and that the left side of the stone has enough room for tool clearance
      i. Select "Lock Vacuum Cups"
      ii. Select "Vacuum" (make sure vacuum's negative pressure comes into an acceptable running range and that the stone has been pulled down tight)

        iii. Double check that the rear rollers have resistance and that the user has pushed the piece against the rollers to keep it straight

Fourth Step- Run the Metals [(click to watch)](#)

1. Click "Load Program" with...
    a. tools selected to run that you wish to set up (do not include the polishers)
    b. 2nd tool through last metal tool leaving early to set stagger (except a gauge wheel)
    c. "Stop Before" turned on for the first tool that has never been set up
    d. (OPTIONAL) "Program Shift" activated using the location the user desires
2. Press "CYCLE START"
    a. User can hold down "CYCLE PAUSE" or
    b. "CYCLE STOP" at any time in preparation to stop machine movement. If one does not experience improper running results, the user can slide their cursor off the "CYCLE PAUSE" or the "CYCLE STOP" button to remain in the running program.
3. Allow tools that have been set up to run, waiting for the machine to stop on which ever tool had "Stop Before" selected. Once machine has stopped- the user may open the door to visually verify the position of the tool in relation to the stone. This position is based on the user entered length and diameters from in the library.
    a. The user could activate the "User" button in the top left of the screen and the "Keyboard" activation in the top right of the screen to actively check fitment of tooling to the stone's edge
    b. If the tool needs adjusting at this point the user could utilize the **Towards, Away, Lower** and **Raise** fine adjust button(s) to adjust accordingly. However, if the position of the tool is considerably far from where it was expected to be, a remeasurement of the library fields or new considerations of the style properties may be required.
4. Once the first tool has been visually verified, the next tool to be ran in the tool setup program can be selected to have the "Stop Before". This will allow the previous tool to be ran down the edge of the stone and change the program
    a. The user will need to select "LOAD PROGRAM" for the program to recognize any changes selected.
    b. The user will need to also re-activate the "Program Shift" if it was used
5. The user will run the next portion of the program- repeating steps 2-5 as many times as needed to complete the metals until they have all been tested, measured and adjusted according to tooling manufacturer specifications. ***Note: at any time while running a program, the user can hold the "CYCLE PAUSE" or the "CYCLE STOP"***

button incase the tool would perform in an unexpected way (too much horsepower, running at the wrong height, or hogging off too much stone). If one does not experience improper running results, the user can slide their cursor off the "CYCLE PAUSE" or the "CYCLE STOP" button to remain in the running program.

Absolute Position

X	2.006 in
Y1	-0.007 in
Y2	0.000 in
Z	-0.006 in
Spindle	0.000 °

User | TCP | Abs

Current Tool

Fingerbit 1.00

Tool Number 2

Spindle Load

0.0 Hp

Program Shift

X	0.100	in
Y	0.000	in

Inactive

Vacuum Pressure

0.0 inHg

Tool Setup Program

Machining Style	#4 Large Radius Sink Tools R1.75 MILL DROP OUT

▶	#	Description	Feed	Stock Left		●
☐	1	1.375 Core	2.0	0.00		
☑	2	Fingerbit 1.00	12.0	0.100		
☑	5	Z1	80.0	0.030		
☑	4	Gauge Wheel 3.56	100.0	-1.500		
☑	7	T30 Large R1.75 P1	90.0	0.014		
☑	8	T30 Large R1.75 P2	160.0	0.006		

XY Fine Adjust

Towards | 0.000 | Away

Capture X

Z Fine Adjust

Lower | 0.000 | Raise

Capture Z

Load Program | Locate Tool Setup | Lock Vacuum Cups

Vacuum ⏻ | Crosshair Laser Off | Raise Material ⇧

MANUAL AIR PURGE

Jog Settings

Type | Incremental

● .001 ◯ .010 ◯ .100

Distance | 0.001
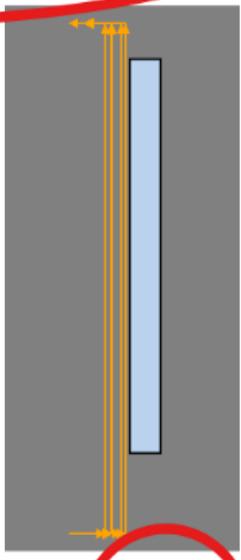
Keyboard | **STOP**

Y+ | +Z

X- | X+ | -Z

Y-

Motion Override

| 100% | 100.0 in/min | +/- 5% | 100% |

⇩ | Adjust in/min | ⇧

Top Navigation

ALARM LIST | ALARM RESET | MAINTENANCE | SWITCH USERS | MORE | MINIMIZE | EXIT

OPERATE

PROGRAM

TOOLS

DIAG

SETUP

Bottom Navigation

CYCLE START | CYCLE PAUSE | CYCLE STOP | TOOL LIBRARY | STYLES | TOOL SCAN | MEASURE | TOOL SETUP

Finish the Edge Styles and Polishers (if needed)

Remember, not all styles have polishers. For the styles that do however, the process will continue from the metal set up. The User should ensure that all the metals are properly adjusted and that they have been used to establish the left edge of the stone completely all the way down without a stager and while still vacuumed in place. This gives a fully prepped edge to set the polishers on.

1. Click "Load Program" with…
    a. tools selected to run that you wish to set up (do not include the metals)
    b. all polishers leaving early to set stagger to see proper polisher-to-stone contact
    c. "Stop Before" turned on for the first tool that has never been set up
    d. (OPTIONAL) "Program Shift" activated using the location the user desires
2. Press "CYCLE START"
    a. User can hold down "CYCLE PAUSE" or
    b. "CYCLE STOP" at any time in preparation to stop machine movement
3. Allow tools that have been set up to run, waiting for the machine to stop on whichever tool had "Stop Before" selected. Once machine has stopped- the user may open the door to visually verify the position of the tool in relation to the stone. This position is based on the user entered length and diameters from in the library.
    a. The user could activate the "User" button in the top left of the screen and the "Keyboard" activation in the top right of the screen to actively check fitment of tooling to the stone's edge
    b. If the tool needs adjusting at this point the user could utilize the **Towards**, **Away**, **Lower**, and **Raise** fine adjust button(s) to adjust accordingly. However, if the position of the tool is considerably far from where it was expected to be, a remeasurement of the library fields or new considerations of the style properties may be required.
4. Once the first tool has been visually verified, the next tool to be ran in the tool setup program can be selected to have the "Stop Before". This will allow the previous tool to be ran down the edge of the stone and change the program
    a. The user will need to reload "Load Program" for the program to recognize any changes selected.
    b. The user will need to also re-activate the "Program Shift" if it was used
5. Users will run the next portion of the program- repeating steps 2-5 as many times as needed to complete the polishers until they have all been tested for proper fitment and spindle loads according to tooling manufacturer specifications.

Setup of Drilling Tools [(click to watch)](#)

When the user of the Spartan is setting up a drilling tool, a style that uses only drill tools must be selected. The drills will run down to the "FINAL DEPTH" height designated in the style selected and can be verified by touching off the top of the vacuum cups or a gauged piece of stone with a known thickness. *Note: no stone needs to be loaded into the Spartan for drill setup.*

1. Click "Load Program" with...
    a. A style with only drill tools
    b. Tools selected to run that you wish to set up
    c. "Stop Before" turned on for the first tool that has never been set up
2. Press "CYCLE START"
    a. User can hold down "CYCLE PAUSE" or
    b. "CYCLE STOP" at any time in preparation to stop machine movement
3. Allow the tool to run, waiting for the machine to stop at the "FINAL DEPTH" set in the style (preferably a custom Z value of a gauged piece of stone with a known thickness OR at either Z 0.000 for vacuum cup height). Once machine has stopped- the user may open the door to visually verify the position of the tool in relation to the vacuum cups or a piece of stone of known thickness. This position is based on the user entered length and diameters from in the library.
    a. The user could activate the "User" button in the top left of the screen and the "Keyboard" activation in the top right of the screen to actively check the height of the drill tool
    b. If the tool needs adjusting at this point the user could utilize the Lower, and Raise fine adjust button(s) to adjust accordingly. However, if the position of the tool is considerably far from where it was expected to be, a remeasurement of the library fields or new considerations of the style properties may be required.
4. Once the first tool has been visually verified, and adjusted, the next tool to be ran in the tool setup program can be selected to have the "Stop Before"
    a. The user will need to reload "Load Program" for the program to recognize any changes selected.
    b. Users will run the next portion of the program- repeating steps 2-4 as many times as needed to complete the rest of the drilling tools until they have all been tested for proper running heights.

Section 4

## Creating a Parametric Sink on the HMI

Overview

Creating a sink in the machine's library will allow the user to create a program without the need for independent CAD/CAM software. It also will allow the user to set up each sink with specified holes for both faucets and sink anchor holes for fastening the sink. A machining style can also be selected for the required sink edge profile. *Note: the user must be signed into the HMI to create a sink (minimum level is "maintenance".)*

Creation of Sink in Library (click to watch)

After turning on the HMI's Park Industries user interface, the user will;

- First select the "Program" main tab on the right-hand side of the screen
- Second select the "Sink Library" sub tab on the bottom of the screen
- Then choose the "Create Sink" option near the right to open the "Create New Sink Profile" window, where you can select to start creating either an oval, rectangular or L-shaped sink
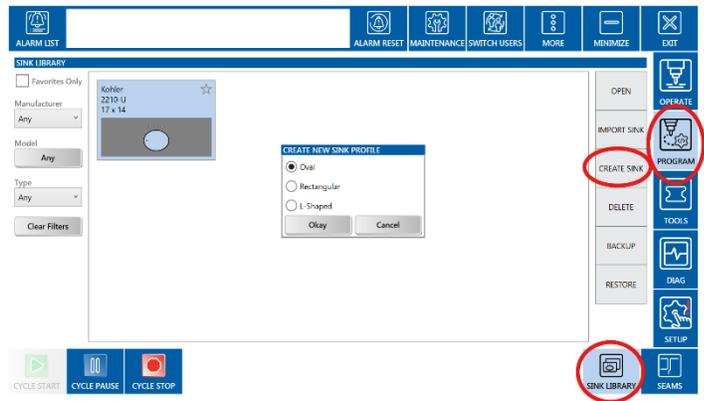
The "About" Screen (click to watch)

The user will be asked the "Type" of sink and to choose the "Manufacturer" of the sink they are creating as well as having the option to specify any model numbers (including from other manufacturers) with sinks the same shape and size.
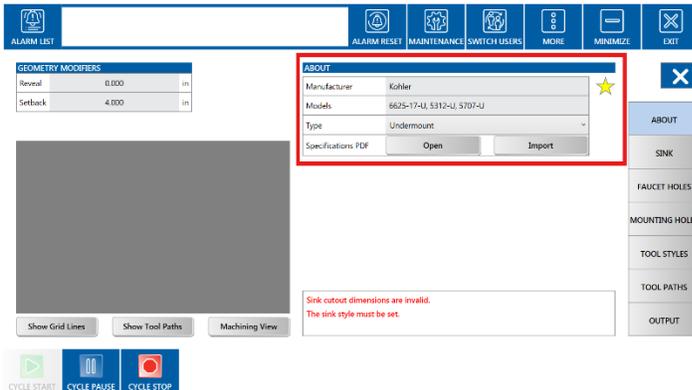
The *Specifications PDF* area can be used to "Import" a .pdf file from a website or one personally generated by a CAD software to make it easily accessible to "Open" later.

Note: The "Type" of sink can be set for further filtering of sinks in the future once the sink library is more populated. The user can also "Star," or favorite, the sink to make it easier to find for popular usage in jobs.

The "Sink" Screen [(click to watch)](#)

This is where the size and main dimensions of the sink will be stored as well as the setback (how far back into the counter) the user would like to place their sink hole.

- Type in the length(X) and width(Y) dimension(s) of your sink
- Each Radius can be controlled individually or as a group
- The Reveal field is where you can adjust the size of the sink opening if the original (flush mount) size needs to be altered for an overhang or reveal (positive and negative values can be used)
- The setback is the distance from the machine's origin fence (or datum) where the user is typically going to place the front edge of the sink

Note: Best practice could be considered that the "flush mount" size is what is placed in the sinks "SIZE" fields so that the "REVEAL" can be used to dimensionally alter for a positive or

negative overhang amount if desired. Be careful of large reveal (positive or negative amounts) that will alter the size of the sink's radiuses.
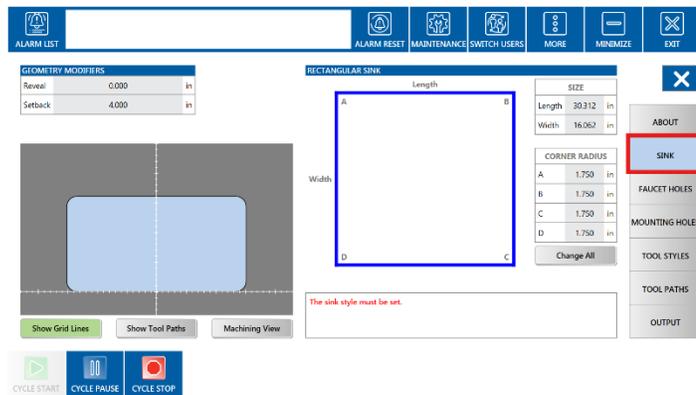
All editable field values are used to describe the sink's position with the polished/ finished face of the stone up. This allows the user to program and set the sink and see a mock rendering of the *__installed__* sink (polished face up).

In the bottom left corner of the screen, the user can click on "SHOW GRID LINES" to aid in visualization to see a representation of the setback chosen for the sink. Using the other buttons "SHOW TOOL PATHS" and "MACHINING VIEW" - are also available for viewing as the user continues through the **"Tool Styles"** and **"Tool Paths"** screen tabs to complete the sink.
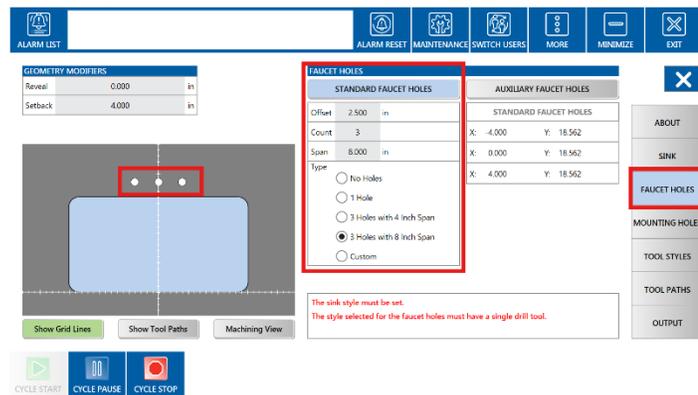
The "Faucet Holes" Screen (click to watch)

The user can choose the amount of the faucet holes and the locations using either a standard faucet hole pattern or customized spread of holes by changing X & Y locations that reference from the front edge of the ***Sink hole*** (setback), or even a combination of both.

Using Standard Holes option: (click to watch)

- **Offset**- the distance from the edge of the sink sizes in the "SINK" screen (this offset ignores the "reveal" value. Often- value will be 2.500 inches
- **Count**- how many holes the pattern will consist of
- **Span**- distance between the center of each **outer** hole in the pattern
- **Type**- there are three common patterns provided with the addition of customizing the pattern

  Note: the default X Y location will populate from the sink's width (from the SINK screen) plus the offset distance. If the Reveal value is changed to number other than 0.000 inches, then the location will be modified according to what was chosen in the "Offset" function

Using Auxiliary Faucet Hole option: (click to watch)
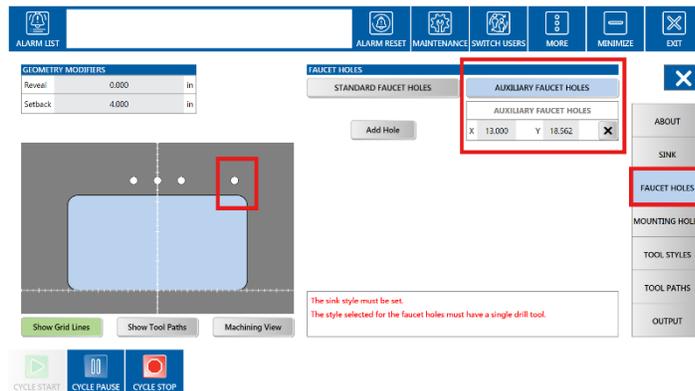
Works much like the "Standard Holes" option of 1 Hole except that the X & Y location can be changed per each additional hole location. This is commonly used for soap dispensers, RO systems, and other auxiliary sink add-ons.
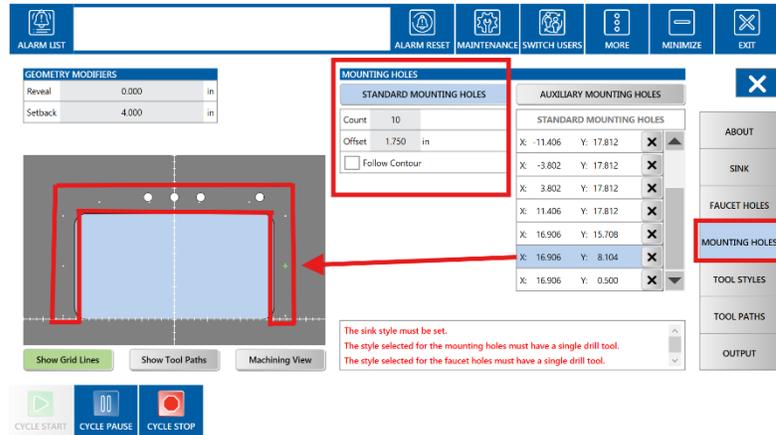
The "Mounting Holes" Screen (click to watch)

This screen allows the user to create either a specific location, or pattern, of holes that can be drilled with a small core bit for the purpose of mounting sinks with sink clips.

Using Standard Holes option:

(click to watch)

- **Count**- how many holes the pattern will consist of
- **Offset**- the distance from the edge of the physical (flush) sink to the center of the holes
- **Follow Contour**- if option is checked- the pattern of the mounting holes will follow the sink's rim in relation to the sink's shape. Holes will also populate near the front edge of the sink. Commonly used in oval sinks, and some L-shaped sinks, but not simple rectangular sinks.
  Note: Single or multiple holes can be deleted as necessary to prevent mounting hole locations that interfere with faucet holes or other features.

**Using Auxiliary Mounting Holes option: (click to watch)**
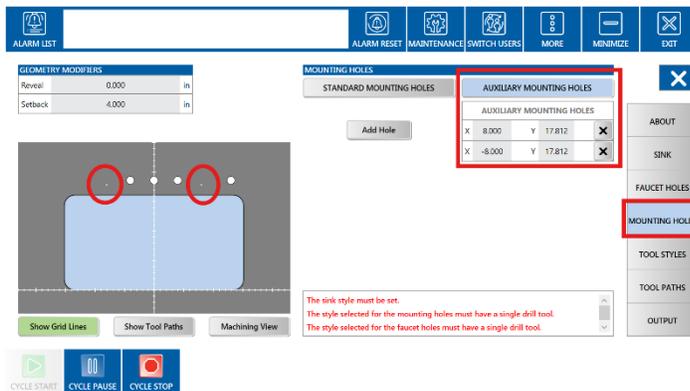
This feature utilizes the customizable X & Y location fields to properly place mounting holes in locations that would not interfere with faucet holes or sink rims.

The "Tool Styles" Screen [(click to watch)](#)

Once the sink, faucet and mounting holes are in their proper positions, the "Styles" (saved groups of tools) may be selected in this tab. There can be up to three different styles used in one sink drawing listed below


- **Sink Style**- style for milling and profiling out the sink
- **Faucet Hole Style**- style with a core bit for faucets
- **Mounting Hole Style**- style for sink clip mounting holes
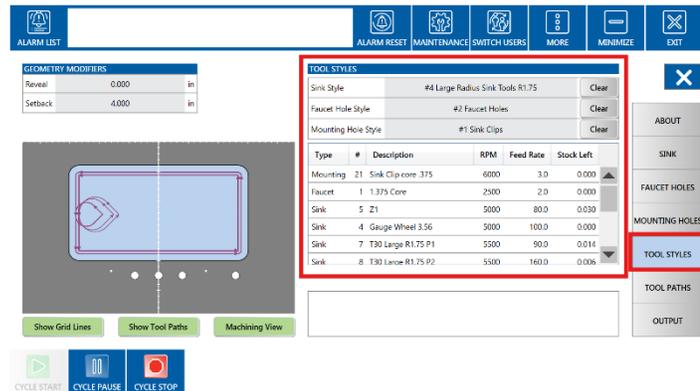
Note: For creation of "Styles" see section 2. The order that the tools are listed will be the running order of the program.

The "Tool Path" Screen [(click to watch)](#)

For the tools being applied, the user can choose an entry point for the tools in the "Sink Styles" such as which side (right or left) of the sink, or if the entry should be rotated around the sink hole for better clearance with tool lead ins/outs. It would also be common for the user to double check the sink program by toggling the "Machining View" button on and off, to have a clear understanding of the polished/ finished face up as well as its position at the machine.
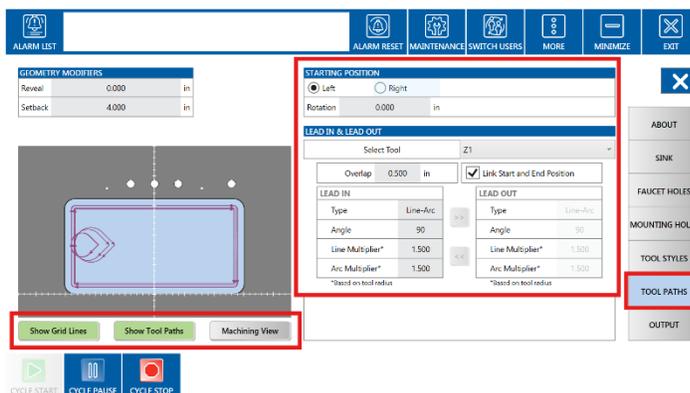
- **Starting Position**- user should typically try to choose the largest side of the sink for the safest entry of tools.
- **Rotation**- if the sink is an odd shape, rotation allows the user to position the entry point of the tools to a better suited location other than left or right side *(this amount is in inches)*

- **Lead In & Out**- while typically the styles used in a sink hole will have lead in and lead out saved in the "Styles" configurations, this option allows the user to modify
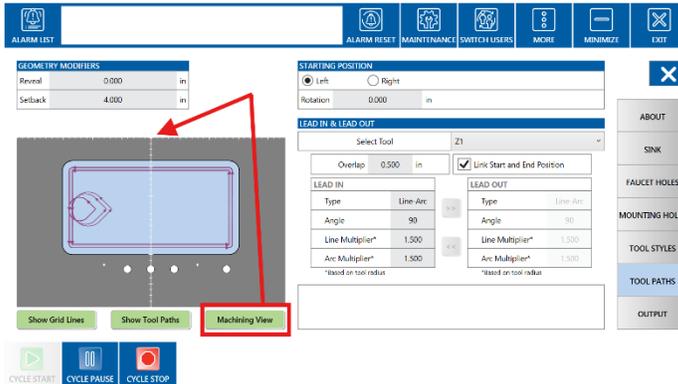
settings for the individual sinks to gain clearance in certain oddly shaped sinks. Most sink shapes, tools would be safer at an angle of 90 degrees or linked to a common enter and exit position

Note: As a reminder- the user can toggle back and forth with the different viewing settings buttons, under the sink rendering, for clarification of what will happen at the machine versus how the finished face would look afterwards. At any time, the sink can be saved with the progress that was done on the project in preparation for sending the sink to the Job Queue.

The "Output" Screen [(click to watch)](#) Currently-needing two videos

The Output Screen is typically the last step for a sink that has been properly programmed. This screen is where the user would send the program to the Queue or to a folder to be ran later. The Output Screen also contains a tool for checking against bad running practices where a tool set is programmed "too tight" into a corner where the size of the tool does not fit. The purpose is to try and correct programing errors that do not follow best practices with tools in corners. *Note: This is not a catch all scenarios!!! The user of the Spartan, that is programing, should understand the tool radius: geometry radius concepts as well as certain alterations, in a sink's shape, can inhibit the tool's usability. This section will be written to explain both "IMPORTED SINKS" (meaning from a CAD/ CAM software) and the "CREATE SINK" (parametric sink) button.*

- **Arc Radius Tool Size Check**- this function will allow the user to control and see what happens in sink hole's radii where tools cannot fit properly. If the tools being used in the style have a larger radius than the sink has drawn, this **"Arc Check"** will highlight effected sink Radii. This can help minimize programing errors if used correctly.
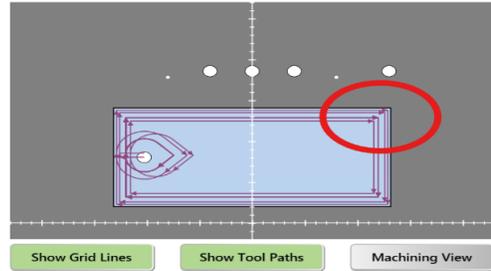

  - **REVEAL AMOUNT-** if the sink has been given a

34
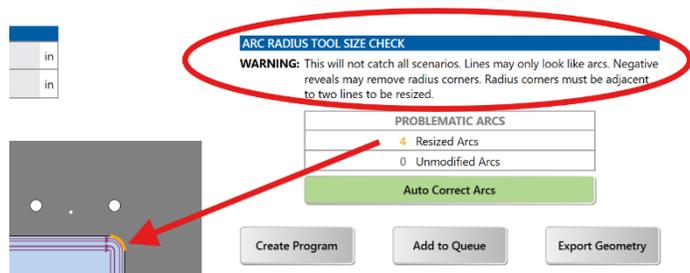
larger value (positive or negative) that changes the shape of the sink to no longer contain radii in the corners, the *"Arc Radius Tool Size Check"* will not be able to help the user alter corners for tooling.
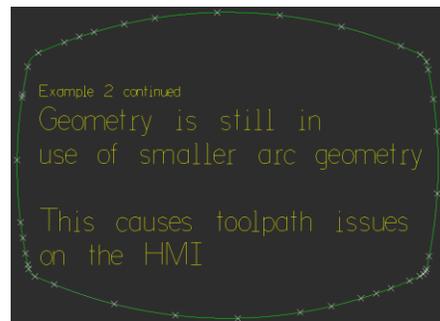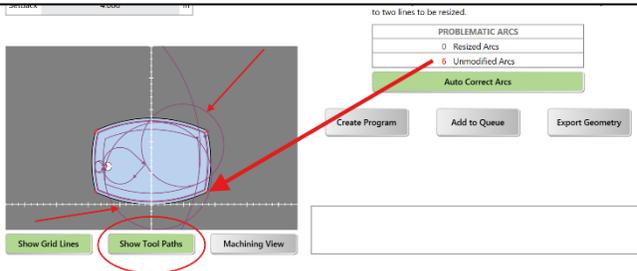
- o **TANGENCY**- if there is a scenario that does not have a tangent arc in the corner, by either a large (positive or negative) reveal or from importing a sink from a CAD software, the Arc Radius Tool *will not be able* to help the

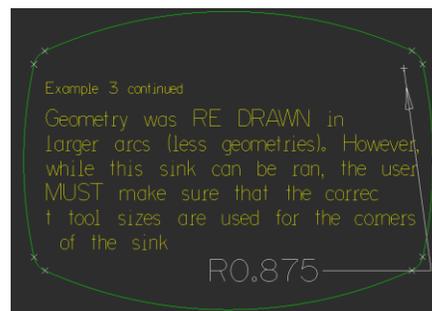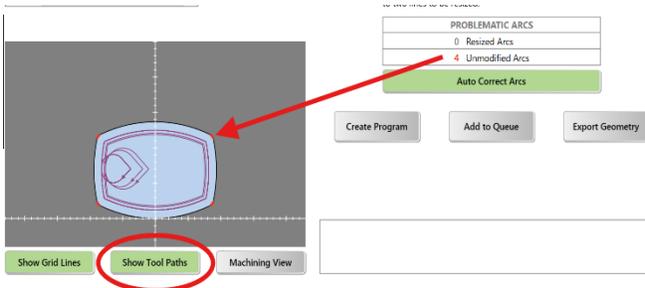| GEOMETRY MODIFIERS		
Reveal	-2.000	in
Setback	4.000	in

Example 1- *No radius drawn in import, or to large of a value in "Reveal"(in this case, the value is too large of a negative value).*

user add a radius *that should have* came from the CAD software! The arcs in the corners of a sink should also be ordered "Line, Arc, Line" if the Arc Radius Tool is expected to perform.

- o **TIPS**- Re draw the sink to have less arcs and use the "Reveal" function with caution. Always double check with   "Show Tool Paths"

Example 2- *Imported sink did not follow the order of "Line, Arc, Line" for tangency. It also could be drawn with too many splines. Notice the tool paths look quite odd.*

Example 2 continued
Geometry is still in use of smaller arc geometry

This causes toolpath issues on the HMI

Example 3 continued
Geometry was RE DRAWN in larger arcs (less geometries). However, while this sink can be ran, the user MUST make sure that the correct tool sizes are used for the corners of the sink
R0.875

- **Create Program**- makes both a program and a sticker that contains a QR code to load the program. The program that is made will be created using all of the sink's parameters active at the time of choosing "Create Program". The Parameters grabbed include; size, reveal, setback location, styles and tool parameters, faucets and mounting holes. The sticker with the QR code can be used many times on countertops that would use the same parameters and can be placed on the needed counters before being run on the Spartan for job planning.
    - **The sticker will contain**
        - Project name (user can change name to be more specific)
        - Manufacturer
        - Models of sink
        - Style names
        - Time stamp
- **Add to Queue**- places sink program in "OPERATE- QUEUE" for job processing
- **Export Geometry**- this button saves the geometry used for the sink and holes in addition to the "ABOUT" screen information into "*.PSINK*" file. This file is used to transfer sinks from one SPARTAN to another.
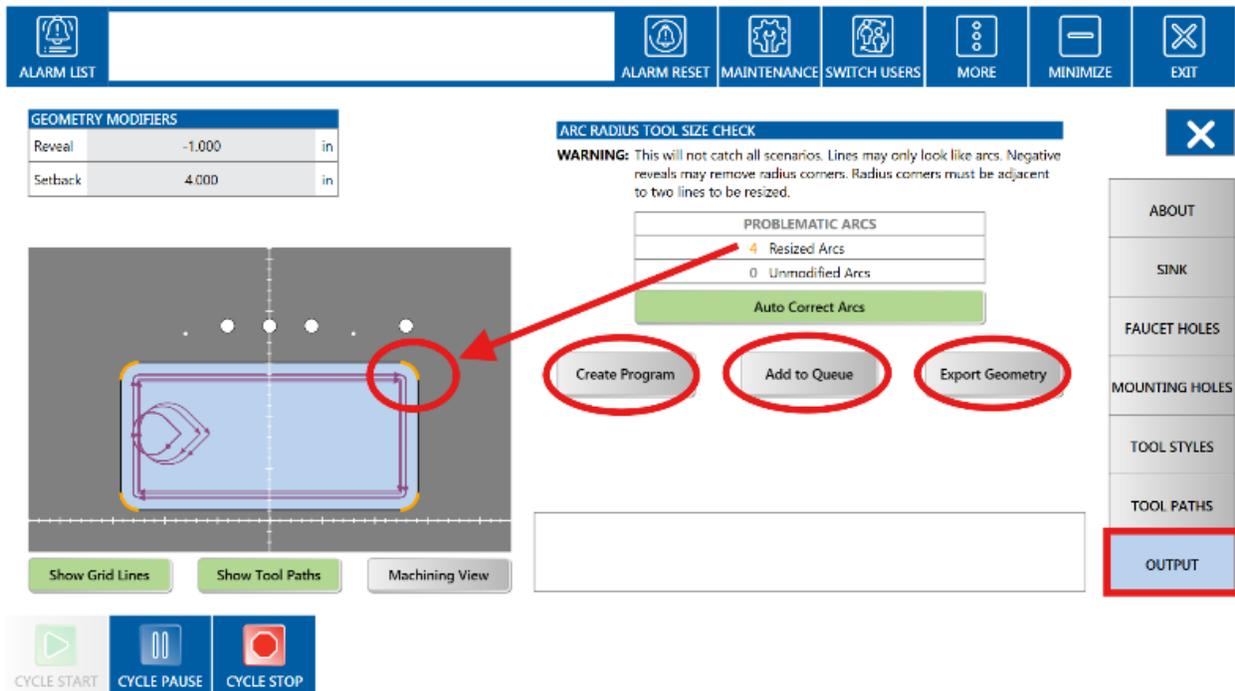
Section 5

Creating a Seams Program

Overview

The Spartan was created with being able to do seams in mind. The user can set up for either a standard straight (butt) seam or a euro seam and chosen parameters for proper placement. *Note: Running a seam will result in table rollers needing to be removed in some instances.*

Programming a Seam

After turning on the HMI's Park Industries interface, the user will;

- First select the "Program" main tab on the right-hand side of the screen
- Second select the "Seams" sub tab on the bottom of the screen
- Lastly choose the type of seam needed for the counter from the drop-down options near the left side of the screen (notice you will have to choose which side you are about to run as well- for every seam has two pieces)
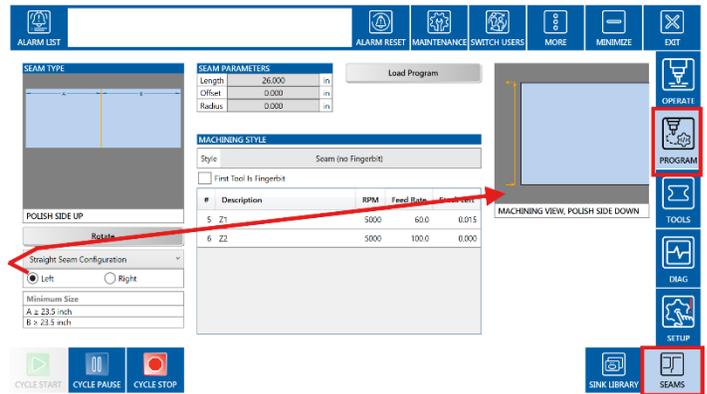
Straight Seam

The most common types of seams for most shops, a standard straight (butt) seam.
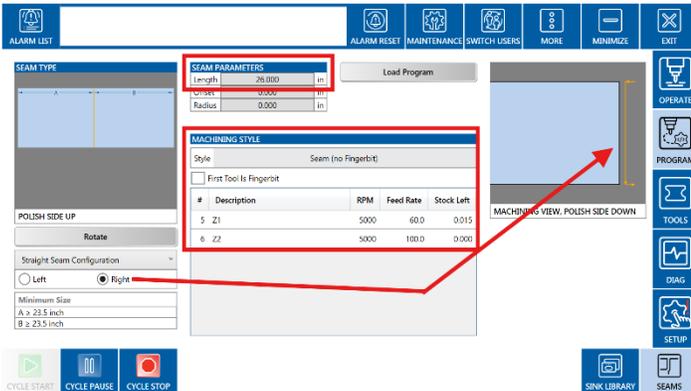
- The user should choose a "Style" of typically just Z wheels if there is less than .250 inches to remove from the edge. More than .250 inches would either require a Fingerbit to run first, or multiple passes with the rougher Z wheel.
- The user will also need to set the length of the cut needed in the "Seam Parameters" field. Max length is 26.00inches

Euro Seam

There are two configurations of Euro Seams to choose from. The viewing window on the left-hand side of the screen shows both with polished face up. Once choosing a side- the user must also choose "Male" or "Female" for each piece of the seam.

- **Seam Parameters**
    - **Length**- distance from end of profiled edge to wall
    - **Offset**- the distance, from the seam's corner to the long part of the euro seam. Max offset is 1.75inches
    - **Radius**- the radiused portion of the euro seam. User needs to apply a larger radius than the largest tool's diameter in the user's style *(Note: the combination of the offset and the radius must allow at least a .125" line segment to remain on the seams corner to act as a lead in for tools)*
- **Style**- if the counter piece *from the user's saw*, does not have the same shape of the euro seam already precut, a Fingerbit is recommended to cut the shape before the Z Wheels
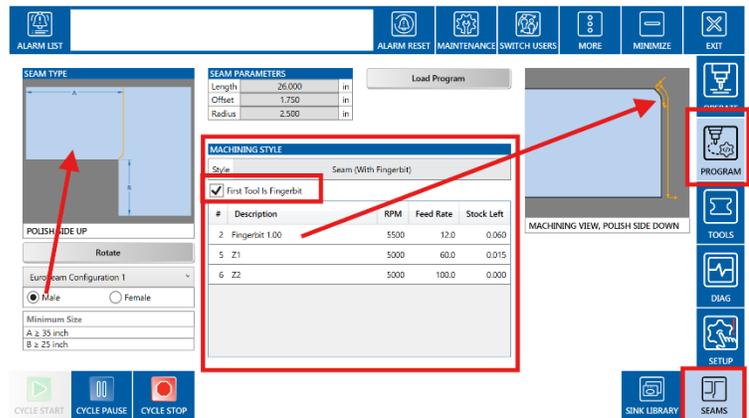
- **First Tool Is Fingerbit**- if checked, this option takes the fingerbit and adds a *mill relief*, using the fingerbit to first back-feed 6.xxx inches in from the radiused end, before running along the edge from the opposite side, to reduce chances of cracking the edge of the stone
- **Female vs Male**- the user must choose which shape they are wanting to cut from the two options.
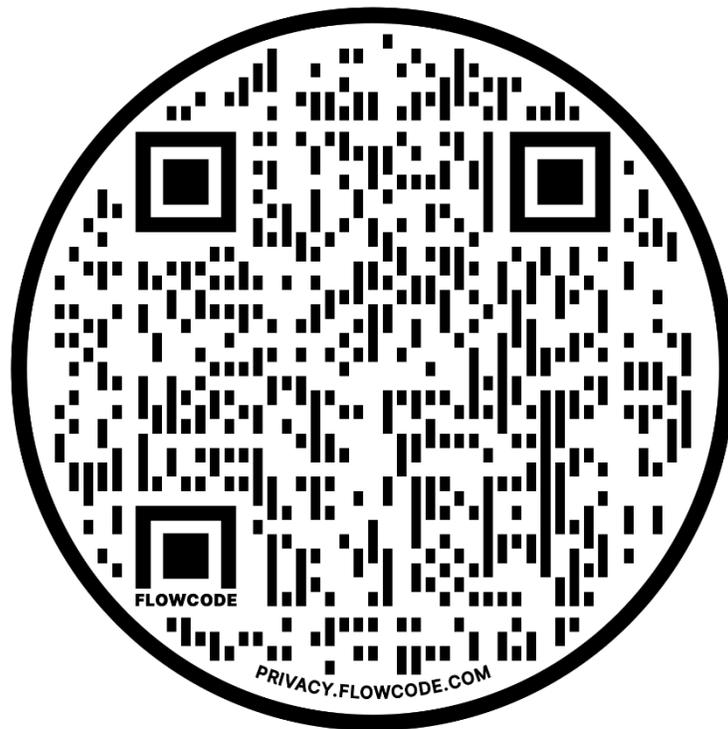- **Load Program**- Click to add immediately to the "OPERATE→ RUN" screen and skip the Queue.
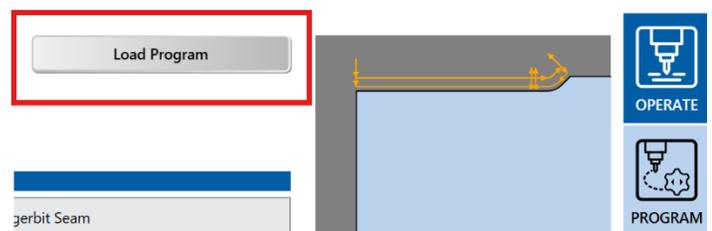
Section 6

Extras

Many videos exist on our website's service and training guides. Using the link below, or the QR Code- visit us online to learn more about what Park has to offer!

https://www.parkindustries.com/stone/service-support/guides/

FLOWCODE

PRIVACY.FLOWCODE.COM

SERVICE & TRAINING GUIDES

			Spartan Tool Library Worksheet				

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Library Location Number	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

Tool Description:						**MILL or DRILL** **(Circle one)**	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number

This information is specific to the tool manufacturer.

Spartan Tool Library Worksheet							
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Library Location Number	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
Tool Description:						MILL or DRILL (Circle one)	
Tool ID #	Tool Pocket Number	Tool Diameter	Tool Length	(XY) Stock REMOVAL*	(Z) Stock REMOVAL*	(Z) Depth Offset	Serial Number
*This information is specific to the tool manufacturer.							

Submit Your Sink DXF Files for the SPARTAN™

If you have a DXF file for a sink you cut, Park Industries® can help add it to your SPARTAN™. Submit your sink DXF file through our online form and our team will review it to ensure it's ready to use on your machine.

This submission process is designed specifically for SPARTAN users and supported by the Park Industries® service and training teams.

Note: Sink DXF clean-up applies only to the SPARTAN CNC Sink & Seam Router.

Scan the QR Code:

or visit: www.parkindustries.com/service-support/spartan-sink-dxf-submissions/

800.328.2309 // Proudly Made in Minnesota

Park Industries®
Precision. Performance. Peace of Mind.

Park Industries®