

PypeServer for Vernon Machines

This document covers topics specific to Vernon machine working well with PypeServer. Topics include:

- General network layout for using PypeServer with the machine
- Vernon MPM settings and PypeServer settings for running NC Code from PypeServer
- NC file monitoring for PypeServer part cut status
- Various methods for nesting in the pipe dead-zone

Table of Contents

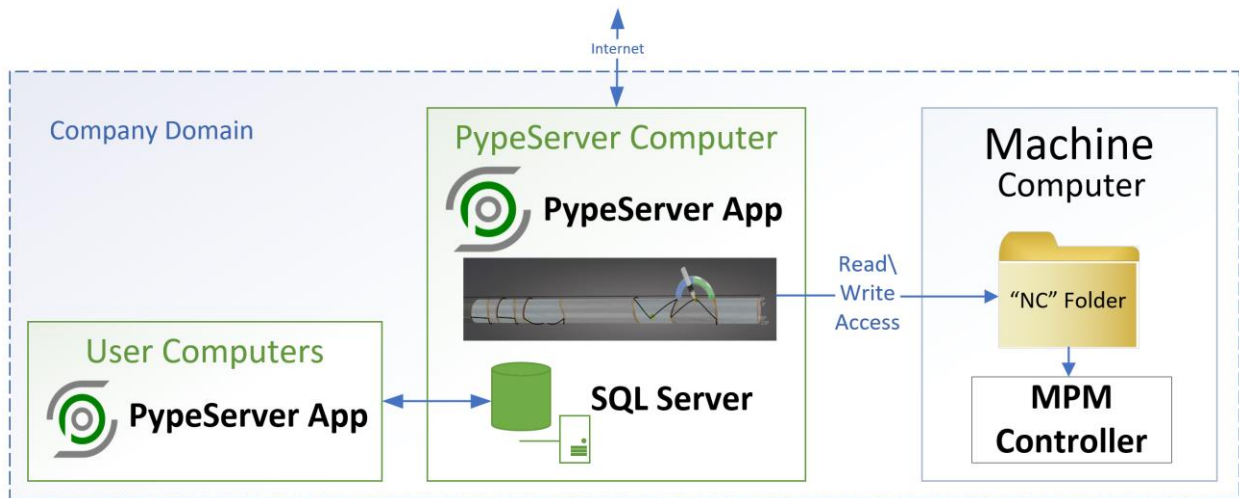
1	General overview of network layout.....	3
2	NC File use	4
2.1	Vernon Machine settings for NC: Enable the M0 (Machine Stop) command	4
2.2	Loading and Cutting a Program	5
2.3	PypeServer Machine Settings for Creating Vernon NC Files	7
3	Last Cut on End of Pipe “Use Far Pipe End”	10
3.1	Machine Setting: Cutting Dead Zone.....	11
4	Shift Pipe Forward after Each Part	12
4.1	Machine Setting.....	12
4.2	Per Pipe Setting – Required.....	12
4.3	Machine/User sequence after each part.....	13
5	Other Machine Settings.....	14
5.1	LeadIn/Out Settings.....	14
5.2	Is Metric = <checkbox>	14
5.3	Feedrate control adjustments	15
5.4	A and C axis feedrate control adjustments	15
5.5	Torch slowdown on tilt adjustment	15
5.6	Default Smoothing Passes and Default Smoothing Distance Machine Settings	16
5.7	Add Date Time To NC Files	16
5.8	Stagger Straight Cut Start Rotation Dist = <distance>.....	16
6	Torch Settings.....	17
6.1	General Tab	17
6.2	Tip Geometry Tab	18

7	Pause Machine for Torch Change for Fineline torches	19
7.1	Configure Torch Settings	19
7.2	NC Code (what you will see).....	20
7.3	Use	21
8	Swap Ground and Touch Sensor for Final Cuts	22
8.1	Initial Settings	22
8.2	Part Requirements.....	22
8.3	Sample GCode	24
8.4	Setting the sensor.....	25

1 General overview of network layout

PypeServer can be run from any Windows computer on a network. Users from different disciplines use PypeServer for different parts of the fabrication workflow. To allow these users access to PypeServer, the PypeServer SQL server needs to be on the company domain so that CAD programmers, Detailers, Shop Foremen and others can run the PypeServer application. PypeServer also needs to connect to the machine's NC folder with read/write access, so that PypeServer users can automatically place NC files (cut programs) into that folder.

The best configuration looks like this:



Note: Internet connectivity allows PypeServer support to assist in initial system configuration and ongoing support. PypeServer can work with your IT department to get PypeServer joined to the domain. For more IT-level information, see the PypeServer training document "Getting Started with PypeServer", section 2.

2 NC File use

For Vernon Machines running WinMPM, PypeServer outputs to an NC format. This format allows PypeServer to fully control machine motion and thereby enables PypeServer features such as nesting, torch beam divergence (angle), using the pipe far end as a final end-cut, shifting the pipe forward after each part is cut, and other features. (These features are discussed further in this document.)

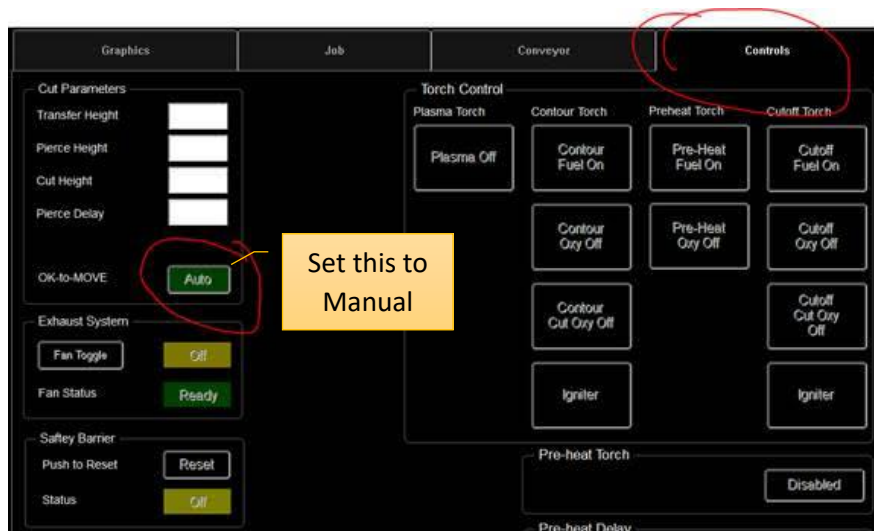
The WinMPM user's manual provides a section on NC programming for your better understanding of the actual NC codes in the PypeServer output files.

2.1 Vernon Machine settings for NC: Enable the M0 (Machine Stop) command

In some cases, PypeServer NC programs instruct the machine to stop while running the NC program so that the user can position the machine as instructed.

2.1.1 Vernon Newer VMD software:

Vernon's MPM software typically comes configured with an override set to ignore the machine stop M0 command. You'll need to disable this override to enable the use of M0 for PypeServer. **If you do not disable this override, the M0 commands in the PypeServer program will be ignored and the machine will not stop as instructed.**



To enable M0, change the "OK-to-MOVE" variable within the VMD main screen under the controls tab. When the system is in "AUTO", M00 is ignored. In "MANUAL", M00 will be enabled. This variable must be set prior to loading the NC file.

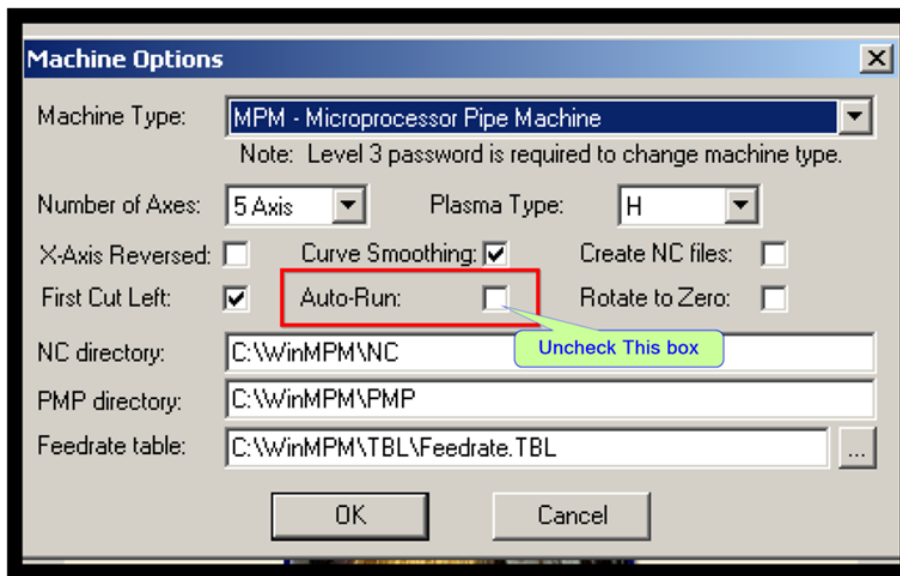
2.1.2 Enabling M0 in Older WinMPM Software

Text in *italics* in this section are from the WinMPM manual.

You need to uncheck the “Auto-Run” checkbox in Machine Options. Go to the Machine Options Page “Tools→Machine Options”, or by pressing “F6” and then “F2”

Machine options are machine particular and will override any machine options contained in .pmp program file. A Level 1 password is required to access the Machine Options windows. Some machine options require a password higher than Level 1 to change.

The Password by default is “aaa”, though it might be different for your machine.



6.2.1.8 “Auto Run”

If the “Auto Run” option is checked it will allow an entire pipe machine program (.pmp) to be run with “Cycle Start” switch only being pushed one time at the beginning of the program. ...

Location in the WinMPM Manual

This information is located in the WinMPM manual. It will be in different locations depending on which version you have.

2.2 Loading and Cutting a Program

2.2.1 Loading a program

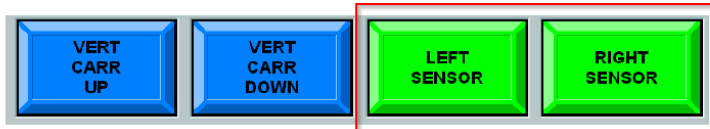
In the first screen that pops up on the MPM program, you’ll find a “Load NC File” button.

- Select the “Load NC” button
- Navigate to where the NC file is located and open the file
- Read the instructions in the NC file. It will tell you where to start the torch. In normal operations this start is just in from the edge of the pipe end where cutting is starting.
- Then Select the “Control Panel” button.

Note: In some older systems you cannot continue to see the NC file. Please see section [Machine Can View NC File When Running](#).

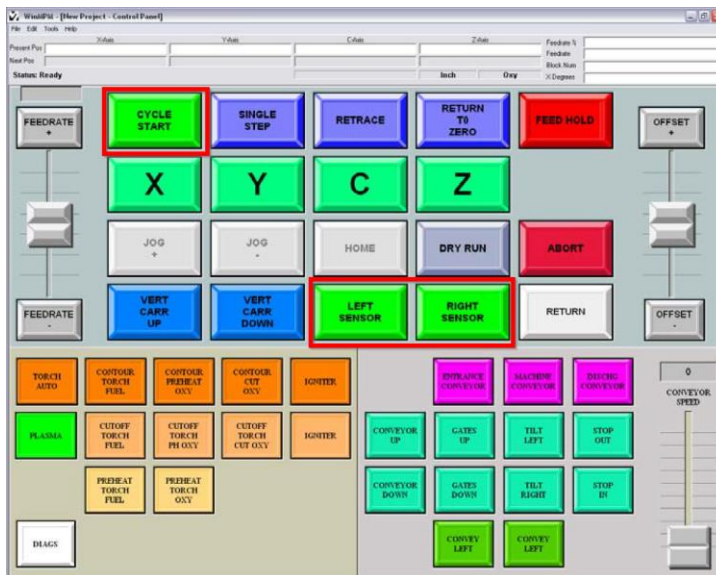
2.2.2 Select a Sensor

Before starting the program, you must select the **leading** sensor (the sensor that will touch down on the pipe at the start of the first cut). This cannot be done programmatically so it must be done in advance of running the NC program.



2.2.3 Hit start

This may require hitting start a few times.



2.3 PypeServer Machine Settings for Creating Vernon NC Files

This section covers the machine parameters highlighted below. The Parameters NOT highlighted are either set by PypeServer during setup or as needed per customer needs, and in some cases set automatically set by PypeServer for the Vernon machine. These parameters are seen by opening the settings → Machine Settings tab and (if you have multiple machines) selecting the correct machine. If you are at the machine kiosk, you will have rights to modify these settings. If you are at another computer (“remote”) you will need to enter the Admin password, which by default is “Admin”. This password can be changed per request.

The screenshot shows the 'System Settings' application with the 'Machine Settings' tab selected. The 'Machine' dropdown is set to 'Machine One'. The 'Cutting Dead Zone' is 20.000. The 'Folder or IP Address' is '\\VernonMachine\NC'. The 'Data Transfer Mode' is 'DumpToFile'. The 'Data Reporting Mode' is 'SystemWatchesFiles'. The 'Local Connection Required' checkbox is unchecked. The 'Min Cut Steps' is 2, 'Max Cut Steps' is 3000, 'Leadin Min Distance' is 0.2, 'Lead In-Out Arc Radius' is 0.2, and 'Leadin Steps' is 8. The 'Torch Head Type' is 111, 'Machine Positioning' is 1111111111000000, 'Machine Capabilities Filter' is 1111111111000000, 'Excluded Capabilities Filter' is 0000000000000000, and 'Is Metric' checkbox is unchecked. The 'System Properties' tab is also visible, showing various parameters like 'Name', 'A_MaxRotation', 'A_MinRotation', 'A_TorchZeroAngle', 'AdjustSpeedByStepCount' (highlighted in yellow), 'C_BlendThroughZero', 'C_MaxRotation', 'C_MinRotation', 'C_TorchZeroAngle', 'CarriageUpTimeSeconds', 'doACRotationFixesInPath', 'FileSyncTimeIntervalMinutes', 'FriendlyNameForMachineAppTab', 'LocateFarPipeEndAtLastPart' (highlighted in yellow), 'MaxA_AngChangePerSecond', 'MaxC_AngChangePerSecond', 'MaxNCArchiveFilesPerMachine', 'NCFileTag', 'ProcessRunningMachine', 'Requires_M0_OnFirstSensorsOn' (highlighted in yellow), 'ShowMachineAppInTab', 'StaggerStaightCutStartRotationDist', 'SwapGroundAndSensorAtEnd' (highlighted in yellow), 'TorchHeadType', and 'X_PipeRotationDir'.

This field has been moved to the System Properties tab

2.3.1 Adjust Speed By Step Count == True if running WinMPM, False if running VMD

Older Vernon machines run WinMPM (not the newer VMD). These older machines change feedrate based on the number of steps in a cut. So if you are running WinMPM, then set this to True so that your machine will respond properly to feedrate changes.

2.3.2 Requires_M0_OnFirstSensorsOn

Older Vernon machines require that, after any machine stop, that another stop (M0) is inserted after the torch-down seeking sensors are turned on in the GCode (with M12). If after you hit start on the first cut, your torch turns on without first seeking the pipe using the sensors, then set this to true. Note that users must hit the start button twice on starting the program.

2.3.3 Folder or IP Address = <Domain file share> (where the NC files are sent)

This is a common read/write file location shared between PypeServer and the Vernon MPM program. PypeServer will write NC files to this folder, and the machine operator will use MPM to open and load these PypeServer NC files. Ideally this share should map to MPM’s default NC folder. Depending on network and machine/MPM configurations, this may not be possible. If not possible, feel free to contact PypeServer and Lincoln to discuss.

To set up the share on your Vernon machine computer, go to the NC folder location and share it such that PypeServer can see that folder through a mapping, such as \\VernonMachine\NC

To access this share from PypeServer, enter the mapping to the machine as shown. Warning, it is not recommended to use network drive mappings (such as z:\), because if this changes, then the synchronization system will not work. If you do use network drive mappings. Do not change the drive letter.

2.3.4 NC File and PypeServer Part Status Synchronization

This section discusses how PypeServer can monitor files sent out to the machine to maintain the status of scheduled parts. Scheduled Part status can be:

- Not Nested
- Nested
- On Machine
- Cut
- Scrapped

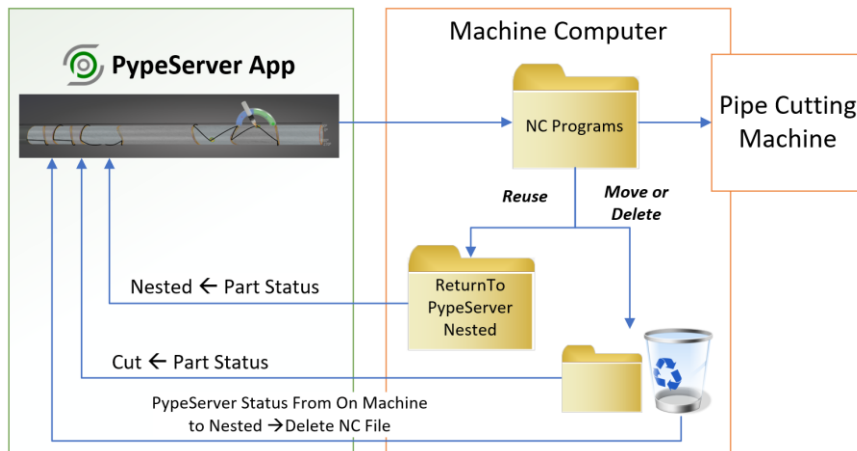
For more information, please see the training video “NC File Management and Sync”.

2.3.4.1 Data Reporting Mode = <selection>

Data Reporting Mode = SystemWatchesFiles : With this setting, PypeServer will monitor the “[Folder or IP Address](#)” folder, such that

- When files are deleted, the Scheduled Parts’ status are changed from “On Machine” to “Cut”.
- When files are move to the subfolder: “Return To PypeServer Nested”, the Scheduled Parts’ status are changed from “On Machine” to “Nested”
- When a user in PypeServer changes a pipe nesting that is currently “On Machine” back to nested, the NC file in the NC Programs folder (typically out at the machine) will be deleted.

This synchronization behavior is summarized in the following diagram:

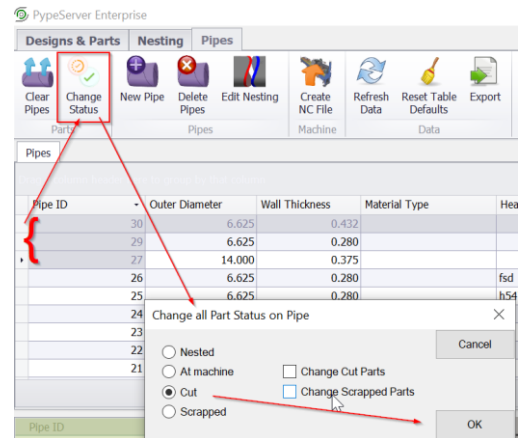


Data Reporting Mode = None

With no data reporting (file tracking), users will need to manage the status of pipes after they are cut.

If this mode is selected, then the user can (should) maintain Scheduled Part cut status in PypeServer by selecting one or more pipes and selecting the Change Status button as shown here:

Note that this is extra work that is performed automatically when Data Reporting Mode = SystemWatchesFiles.

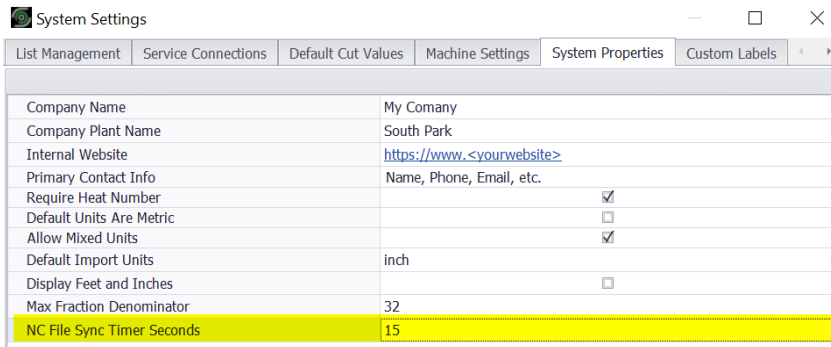


Other settings are not applicable:

- The **Data Reporting Mode = Automatic** is for machines directly connected to PypeServer, where there is no file transfer occurring. This is not applicable to the Vernon machine.
- The **Data Reporting Mode = DialogProcess** selection is for manual saws.

NC File Timer Interval Seconds = <minutes>

When you use this synchronization feature, you can set how often the system checks the NC file folder for synchronization. This setting is in the System Properties Tab and is a global setting for all machines, shown here:



For more information, please see the training video “NC File Management and Sync”.

2.3.5 Machine Can View NC File When Running = <True/False>

As of this writing, this feature has not been implemented. Please contact PypeServer for availability.

In some older WinMPM systems you cannot continue to see the NC file when running the program. In this case, PypeServer will create a second file that will provide a list of M0 commands in the program, in order. The user, when running the program, can view this list (in notepad or similar app), or print the list, and check off each M0 stop as they are reached.

Set this to False if you cannot see the NC file when running.

Machine Can View NC File When Running = False

Note that in most cases the reason for the machine pause is intuitive. One important pause is when using the feature to swap the ground and sensor to nest through the dead zone. In that case the machine must be aware of when to swap the sensor, or upon the next cut, will likely crash the torch head.

3 Last Cut on End of Pipe “Use Far Pipe End”

PypeServer can nest parts so as to use the far end of the pipe as the final (straight) cut of the last part. This allows PypeServer to nest “through” the “Dead-Zone”—where the ground is located. To use this feature:

- One-Time Setup:
 - You must specify the Dead-Zone in machine settings
 - You must select the way that the machine knows where the far end of the pipe is located.
- Each Pipe:
 - You must make sure no cuts go into the Dead-Zone
 - You must check the “Use Pipe Far End” on the Pipe. Checking this will change the Dead-Zone indicator to green and PypeServer will nest parts into that zone.

One configuration of nesting with the Far Pipe End looks like this:



Note: You can also check the Use Far Pipe End checkbox in the Nesting Settings dialog so that all new Pipes will have this checked.

Settings discussed in this section are highlighted here in Machine Settings. To see them you may need to enter the Machine Admin password.

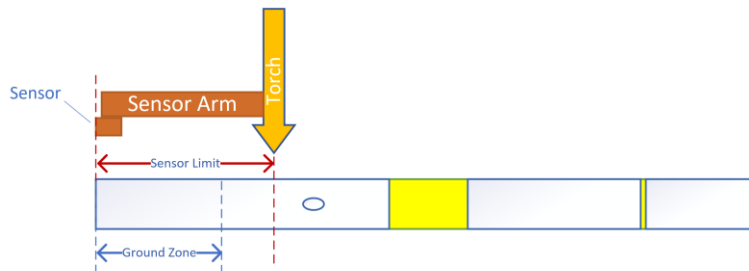
System Settings					
List Management	Service Connections	Default Cut Values	Machine Settings	System Properties	Custom Labels
Machine		Firmware Revision		1.02	
Machine One		Machine Name		Machine One	
Is Metric					
Min Pipe OD		3.500		A_MaxRotation	
Max Pipe OD		37.000		A_MinRotation	
Max Pipe Length		300.000		A_TorchZeroAngle	
Cutting Dead Zone		12.000		AdjustSpeedByStepCount	
Machine Zero to End of Pipe		0		C_BlendThroughZero	
Max Feed Rate		100.000		C_MaxRotation	
COM Port		COM6		C_MinRotation	
Folder or IP Address		\\Machine\WC		C_TorchZeroAngle	
Data Transfer Mode		DumpToFile		CarriageUpTimeSeconds	
Data Reporting Mode		SystemWatchesFiles		doACRotationFixesInPath	
Local Connection Required				FarPipeEndLocatingMethod	
Min Cut Steps		2		FileSyncTimerIntervalMinutes	
Max Cut Steps		3000		FriendlyNameForMachineAppTab	
				LocateFarPipeEndAtLastPart	
				True	

3.1 Machine Setting: Cutting Dead Zone

When you nest with “Use Far Pipe End” checked, the dead zone will become green to indicate that you can nest in it. However, PypeServer still needs to know the Cutting Dead Zone. The Cutting Dead Zone defines the area of pipe that the torch should not cut because it will hit the ground, and should not try to touch off its height because the active sensor will be off the end of the pipe.

Cutting Dead Zone is set this to the greater value of

- The distance that your ground goes into the pipe, plus enough to keep metal splash off your ground, or
- The distance from the torch to the end of the pipe, when the sensor is as far to the end of the pipe as you can go. It's important to measure this sensor distance so that PypeServer does not create cuts where the sensor arm will be off the pipe.



When not using the far pipe end (“Use Far Pipe End” unchecked), then this dead-zone will show up in PypeServer as a yellow area on the end of the pipe and PypeServer will not nest into it. It will look like this:



4 Shift Pipe Forward after Each Part

This feature allows the machine operator to shift the pipe forward after each part is cut. This is useful in various cases, including the cutting of large, heavy parts that need to be unloaded from the end of the machine, or cutting small parts where each part can be easily dropped off the end onto a cart.

4.1 Machine Setting

To enable this feature, you must, in Machine Settings, set the “Supports Shift Pipe Forward” parameter to true.

Supports Shift Pipe Forward For Each Part	True
---	------

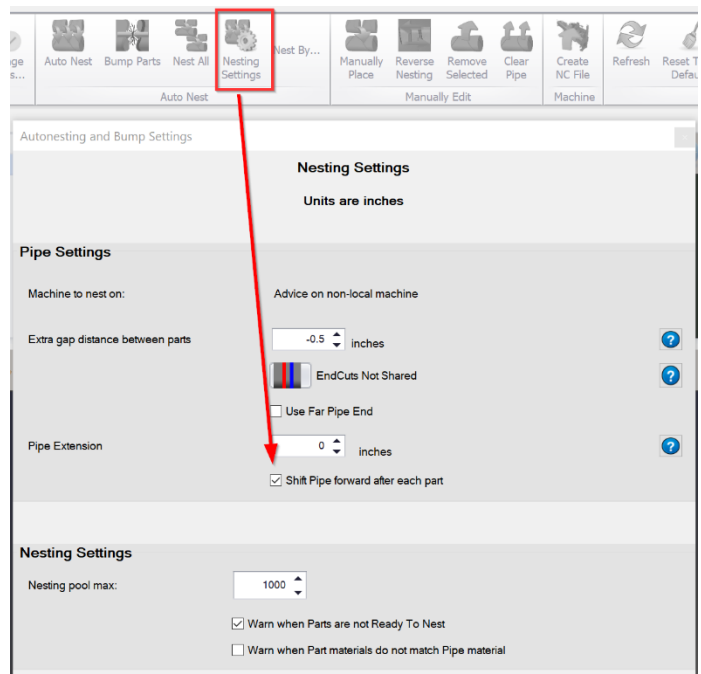
This will cause this feature to show up as the Pipe parameter.

4.2 Per Pipe Setting – Required

Once the Machine setting is set to true, you still need to check the Pipe’s “Shift Pipe Forward” checkbox, shown here:

Pipe ID	5
Customer	
Job	
Design Outer Diameter	8.000
Design Wall Thickness	0.125
Is Metric	<input type="checkbox"/>
Pipe Length	111.000
True Diameter	8.000
True Thickness	0.125
Remaining Length	103.773
Material Type	
Heat Number	c
Total Cut Time	1:12 min
Notes	
Machine Name	Machine One
Seam Angle	0.00
Nesting Settings	
End Cut Sharing	<input type="checkbox"/>
Extra Nesting Gap	0.000
Last cut is end of pipe	<input type="checkbox"/>
Dead Zone Extension	0.000
Shift Pipe Forward	<input checked="" type="checkbox"/>

Note: If you are shifting the pipe forward on every pipe, you can check the “Shift Pipe Forward after each part” checkbox in the Nesting Settings dialog to cause each new pipe to have the Shift Pipe Forward option automatically checked.



4.3 Machine/User sequence after each part

Here is the sequence of machine and user steps that occurs after a part is cut to resuming cutting the next part:

This sequence requires the user to hit cancel at the M0 twice, once to move the torch out of the way to remove the cut part (only if needed), and once to shift the pipe forward so the last pierce hole is under the torch.

1. **Program** finishes cutting a part.
2. **Program** moves torch Y position back to the start of last part, and moves the pipe X rotation to the pierce-hole of the last cut on last part
3. **Machine** stops with an M0
4. **Step four is used only if the user needs to jog the torch out of the way to clear the part. If you don't need to clear the part, just press OK.**
 - **User:** selects Cancel (The program is still running, but the user can jog the torch and move the pipe.)
 - **User:** jogs the torch down the carriage to get the torch out of the way
 - **User:** removes cut part
 - **User:** selects Run Program again.
 - **Machine** will return to the position before the jog in step 5.

Steps 5 and 6 are where you position the torch back over the pierce hole of the last cut. (Or if you are sharing endcuts, you position the torch on the edge of the last cut as if you were cutting that edge.)
5. **User:** selects Cancel (a second time)
6. **User:** raises the pipe and rolls it forward so that the torch is right over the pierce-hole of the last cut. Because it's hard to lower the pipe to match the exact torch y-position, you can move the gantry after setting the pipe back down on the cutting rollers.
 - **NOTES:**
 - Move the torch to position such that the torch stays in that position when restarting. On some machine this motion is performed using the Offset feature. When you hit restart, the machine should not move back to the previous (short) distance you moved/offset from. It is recommended that you test this feature on your specific machine.
 - Do not jog the torch because this is the starting position you are aligning the pipe to. If you jog the torch, then resuming the program will cause the torch to return to the pre-jogging position.
 - They pipe's X rotation should already be correct
 - If you have a little extra pipe on the tail end of the job, then it's OK to position the torch a little down the pipe from the last pierce hole, but not OK to position it farther toward the cut end.
7. **User (optional):** goes back to the NC window so they can see the GCode (a convenience)
8. **User:** hits Run Program again,
9. **User:** selects OK to the M0 stop to continue the program

5 Other Machine Settings

5.1 LeadIn/Out Settings

These settings are seldom changed. These are only limits. The actual leadin settings are made in the Torch settings, but they must fall within these limits.

For more information, please see the document “Torch End of Cut Leadout and Tuning” in the PypeServer training system.

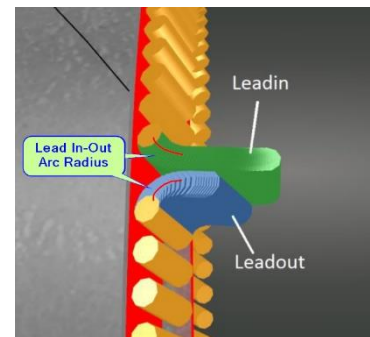
Leadin Min Distance	0.25
Lead In-Out Arc Radius	0.15
Leadin Steps	8

5.1.1 Leadin Min Distance = <distance>

This is the minimum distance allowed for a leadin in PypeServer. A good performing machine can have a short leadin.

5.1.2 Lead In-Out Arc Radius = <distance>

This is the radius of the turn made by the leadin. Note that this is not the overall length of the leadin. Having this as a small radius decreases the range of gouging, but the tradeoff is that if it's too tight, the machine may fault or have difficulty making turn smoothly.



5.1.3 Leadin Steps = <integer number>

This is the number of steps PypeServer creates for the leadin. This adjustment is not typically made by the user.

5.2 Is Metric = <checkbox>

If you set this to Metric, all your machine and torch settings will be interpreted as metric. You can still have standard (Imperial) measurements parts. Note that you do not need to set this to Metric just because your machine runs in metric. This is only a convenience to allow users to work in familiar units.

5.3 Feedrate control adjustments

There are three settings to control feedrate, shown here:

System Settings			
List Management	Service Connections	Default Cut Values	Machine Settings
Machine	Firmware Revision	1.02	Name
Vernon Machine	Machine Name	Vernon Machine	A Max Rotation
	Cutting Dead Zone	24.000	A Min Rotation
Admin logout...	Machine Zero to End of Pipe	0	Add Date Time To NC File Names
Validate Settings	Folder or IP Address	C:\MachineOutput\Vernon	Adjust Speed By Step Count
	Data Transfer Mode	DumpToFile	A Rotate Feed Rate Increase
	Data Reporting Mode	SystemWatchesFiles	Bed Wheel Axle Distance Apart
	Local Connection Required	<input type="checkbox"/>	Bed Wheel Diameter
	Min Cut Steps	2	Carriage Up Time Seconds
	Max Cut Steps	700	C Tilt Feed Rate Increase
	Leadin Min Distance	0.2	Default Smoothing Distance
	Lead In-Out Arc Radius	0.2	Default Smoothing Passes
	Leadin Steps	8	Friendly Name For Machine App Tab
	Machine Positioning	111	Locate Far Pipe End At Last Part
	Machine Capabilities Filter	11111111111000000	NC File Sync On
	Is Metric	<input type="checkbox"/>	NC File Tag
	Default Cutter Type	Plasma	Negate A Axis Rotation
			Pause After Pieces Shorter Than
			Pause Before Returning To Start
			Process Running Machine
Torch	Torch Name	Plasma on V1	Name
Plasma on V1	Cutter Beam Divergence Ang	0	C Bevel Feed Rate Decrease
	Kerf	0.100	Dwell Secs After Torch On
	Pierce Angle	10.00	Torch Tip Base Diameter

These feedrate adjustments are combined to calculate a net feedrate for any given cutting step.

5.4 A and C axis feedrate control adjustments

The “A Rotate Feed Rate Increase” and “C Tilt Feed Rate Increase” settings allow you (or PypeServer support) to increase the feedrate of the machine in proportion to the speed of rotation of these axes. These settings are provided because some Vernon machines slow down more than the specified feedrate when the A and C axes are moving quickly. These fields can have two types of data:

1. A number, which works as a multiplier based on the rotation speed. For many Vernon 7000-series (machines with serial numbers in the 7000 range), an appropriate A Rotate Feed Rate Increase is 3, and an appropriate C Tilt Feed Rate Increase is 1.5.
2. These settings can also define a non-linear spline by points, where X is the rate of degrees per second, and Y is the desired feedrate multiplier at that rate. For example the A Rotate Feed Rate Increase may be set to:

“0,1,10, 1.5,40,3”. This would provide a feedrate multiplication of one when the A-head is not rotating, and increase to 1.5x when the A-head is rotating at 10 degrees per second, and then ramping up to 3 when the A-head is rotating at 40 degrees per second.

5.5 Torch slowdown on tilt adjustment

The “C Bevel Feed Rate Decrease” field slows the head down as the torch leans over in beveling so as to compensate for cutting thicker material. A single number in this setting will defines a linear slowdown from no slowdown at zero degrees to the specified multiplier (slowdown) at 45 degrees.

Similar to the AC head feedrate multipliers, you can also enter a set of points to define a non-linear slow down ramp. For example: “0,1,15,0.9,30,0.7,45,0.6” would effect a non-linear progression through these feedrates:

0 tilt = 0 slowdown, 15 degrees = 0.9 of feedrate, 30 degrees = 0.7 of feedrate, 45 degrees = 0.6 of feedrate,

5.6 Default Smoothing Passes and Default Smoothing Distance Machine Settings

For best cutting on Vernon machines, the A-Head needs to rotate smoothly. Any rapid changes will result in the machine slowing down.

Also, for plasma torches, it is best to not rotate the torch through tight corners such that the torch stays in one place through the tight corner. This can occur in cuts such as an offset or tilted hole, a rectangle, or a y-trunk end-cut. To avoid this, the A-head rotation can be smoothed to effectively smooth transitions through areas where a true perpendicular torch cut would be less than optimal and typically over-burn.

The “Default Smoothing Distance” machine setting specifies the distance away from a given step that will be included in the smoothing algorithm. The higher the distance, the more each step is influenced by steps farther away. In some cuts, such as holes and rectangles, this setting can be overridden at the cut level by setting the Smoothing Distance for the cut.

The “Default Smoothing Passes” machine setting specifies how many passes the smoothing algorithm will run. The more smoothing passes, the smoother the A-head will rotate. However, in cases where directions change quickly, this will also increase the C-tilt from perpendicular across the pipe. For most cuts, 2 smoothing passes is sufficient. In some cuts, such as holes and rectangles, this setting can be overridden at the cut level by setting the Smoothing Passes for the cut.

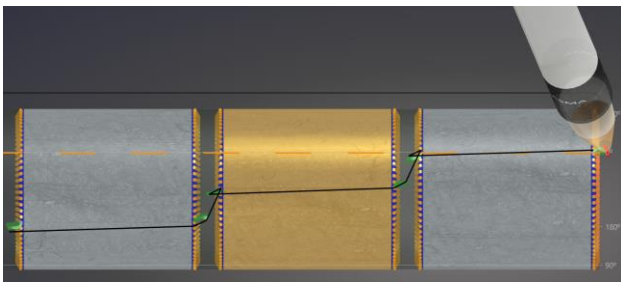
5.7 Add Date Time To NC Files

When true, nc files will look like this: Part 96 11_30_21 17_50.nc

When false, nc files will look like this: Part 96.nc

5.8 Stagger Straight Cut Start Rotation Dist = <distance>

When straight cuts are nested together without any rotation between starts and finish, the leadins and leadouts can overlap and cause the torch to burn incorrectly or fault as the leadin torch start hits the leadin and leadout gaps of the previous cut. This instructs PypeServer nesting to rotate the next part by a distance (around the pipe) so as to avoid this intersection. Shown below is a 4.5” pipe with StaggerStraightCutStartRotationDist = 1 inch.



6 Torch Settings

6.1 General Tab

General

Tip Geometry

Feedrate Decrease when Beveling

?

Dwell Seconds after Torch On.

?

Use Fineline Torch

☒

FineLine Torch Settings

Add FineLine Materials To Materials List

The **Feedrate Decrease when Beveling** field is described by the help button icon as follows:

C Bevel Feed Rate Decrease

This setting slows the feedrate as the bevel increases. If this setting is a number, then it defines a linear slowdown from no slowdown at zero degrees bevel, to this setting multiplied by feedrate 45 degrees. For example, a setting of 0.5 would slow a feedrate down to 75 at 22.5 degrees bevel, and 50 at 45 degrees bevel.

You can also enter a set of points to define a non-linear slow down ramp, where X = bevel and Y = feedrate multiplier. For example, for a Feedrate of 100, the settings "0,1, 15,0.9, 30,0.7, 45,0.6" defines a curved feedrate changes as follows:

Bevel	Feedrate multiplier	Feedrate
0	1	100
15	0.9	90
30	0.7	70
45	0.6	60

The **Dwell Seconds after Torch On** field is described by the help button as follows:

Dwell Secs After Torch On

If not zero, this will cause a machine pause (dwell) G-Code to be inserted after the plasma torch-on G-Code. Use this only if the machine begins moving without waiting for the torch to ignite.

Do not set this if the machine is automatically waiting because the added dwell will cause overburn and may also cause the torch to fault after it burns through the pierce. Most machines do not require a dwell after torch-on.

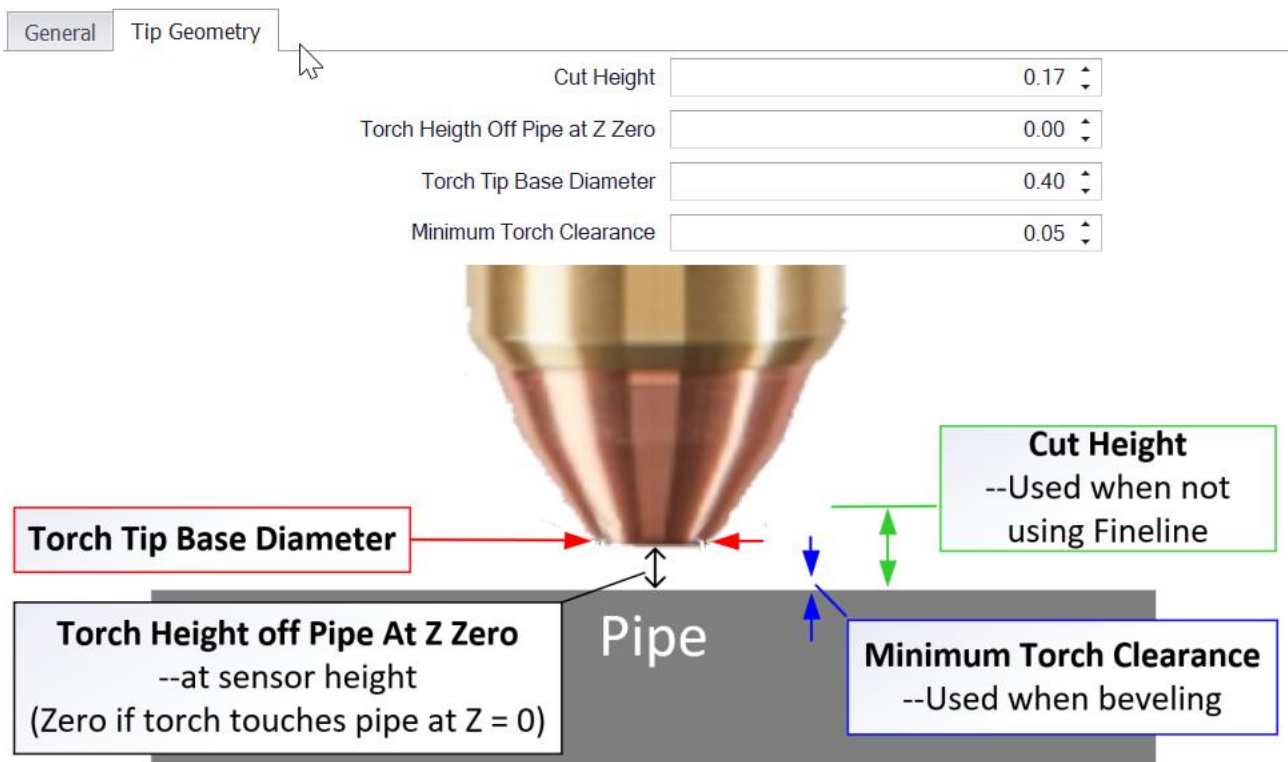
The machine instructions can use Fineline torch cut tables by checking the Use Fineline Torch checkbox.

6.2 Tip Geometry Tab

The image below is the “Tip Geometry” tab of the Torch Settings

NOTE: By default, Vernon 5 Axis machines only use the Z axis to move to the Zero position that is programmed by Vernon customer support. The torch needs to be a give height off the pipe with the torch at its correct pivot height. This height has been seen to be 3/16” of an inch, though Vernon support can provide the required height. If your torch height is off after setting your pivot height, then you can either loosen the torch in the collet and raise it to the height specified by Lincoln, or contact Lincoln support, who can adjust the Z-zero height in machine settings.

If your machine cannot adjust Z on the fly (again, the default for Vernon machines) then this section does not apply.

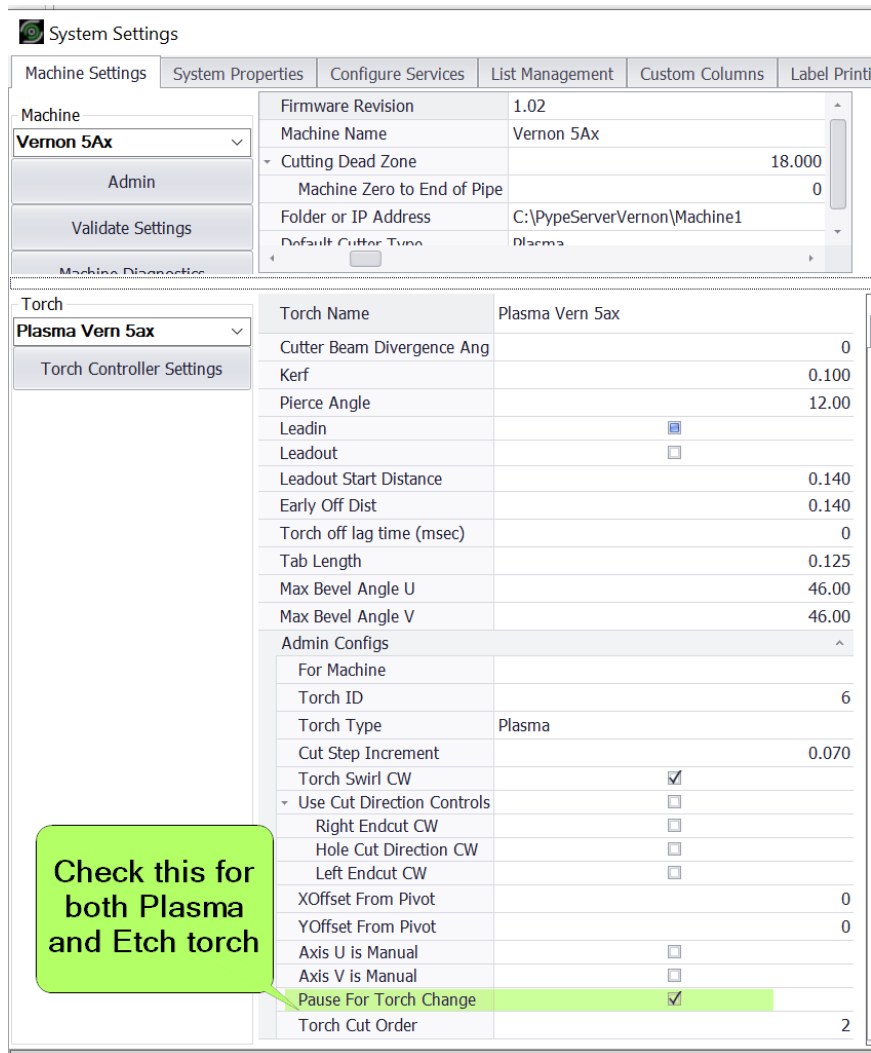


7 Pause Machine for Torch Change for Fineline torches

As of this writing, Fineline torches cannot automatically change from Cut to Mark (Etch). Users can still create parts with both Plasma cuts and Etch marks.

7.1 Configure Torch Settings

Both Plasma and Etch torches must be configured to **Pause for Torch Change**.



The screenshot shows the 'System Settings' application with the 'Machine Settings' tab selected. The 'Machine' dropdown is set to 'Vernon 5Ax'. The 'Torch' dropdown is set to 'Plasma Vern 5ax'. The 'Torch Controller Settings' section is expanded, showing various parameters. A green callout box with the text 'Check this for both Plasma and Etch torch' points to the 'Pause For Torch Change' checkbox, which is checked.

Machine	Firmware Revision	Machine Name	Cutting Dead Zone	Machine Zero to End of Pipe	Folder or IP Address	Default Cutter Type
Vernon 5Ax	1.02	Vernon 5Ax	18.000	0	C:\PypeServer\Vernon\Machine1	Plasma

Torch	Torch Name	Cutter Beam Divergence Ang	Kerf	Pierce Angle	Leadin	Leadout	Leadout Start Distance	Early Off Dist	Torch off lag time (msec)	Tab Length	Max Bevel Angle U	Max Bevel Angle V	Admin Configs
Plasma Vern 5ax	Plasma Vern 5ax	0	0.100	12.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0.140	0.140	0	0.125	46.00	46.00	For Machine
													Torch ID
													Torch Type
													Cut Step Increment
													Torch Swirl CW
													Use Cut Direction Controls
													Right Endcut CW
													Hole Cut Direction CW
													Left Endcut CW
													XOffset From Pivot
													YOffset From Pivot
													Axis U is Manual
													Axis V is Manual
													Pause For Torch Change
													Torch Cut Order

Note: The machine will not pause for torch change on the first cut.

7.2 NC Code (what you will see)

```
; ***** Cut 5640 *****
; Design 596, Name: With Etch
; Slit Cut 5640
; Torch Kerf 0.020
; Torch Beam Divergence 0
; Torch Pierce Angle 0
;
; Go to the start of the cut
N21000 G00 X-20.813 Y0.500
; To Restart the program at this cut, position the torch at the start
; of the cutpath and restart at this next G92 command
N21005 G92 X-20.813 Y0.500 ; Start prog here to restart at this cut
N21010 G00 C0.000 A0.000
N21015 F80.000
N21020 M12 ; Sensors On
N21025 G01 Z0.100 ; Z Down with sensors on
; G61 per Machine Property: "Exact Stop G61 Supported"
N21030 G61 ; Exact stop
; Pause per Machine Property: "Dwell Secs For Torch To Reach Sensor Height"
N21035 G04P4.000
; (Setting FineLine 170 to PLASMA using process 1020)
N21040 $usv(36)=0 ; (Material)
N21045 $usv(22)=25 ; (Plasma PSI)
N21050 $usv(70)=25 ; (Shield PSI)
N21055 $usv(21)=12 ; (Amps)
N21060 $usv(14)=0.1 ; (Cut or Mark Height)
N21065 $usv(72)=0.1 ; (Ignition Height)
N21070 $usv(15)=0.1 ; (Retract Distance)
N21075 $usv(151)=2 ; (Process Type)
N21080 $usv(154)=250 ; (Travel Speed)
N21085 $usv(153)=7 ; (Gas Combo)
;Fineline Kerf: 0
; Change torch to Etch
N21090 M22 ; Cycle Start Light On
N21095 M00 ; Program Stop
N21100 M23 ; Cycle Start Light Off
N21105 M64 ; Plasma On
; Dwell for pierce. Typically used when the pierce sensor is not working.
; Dwell per Torch Property: "Dwell Secs After Torch On"
N21110 G04P1.000
; Start of section: cut
N23001 G01 X-20.874 Y0.500 C0.000 Z0.100 A0.000 F250.000
N23002 G01 X-20.936 Y0.500 C0.000 Z0.100 A0.000
N23003 G01 X-20.997 Y0.500 C0.000 Z0.100 A0.000
```

Note: The machine will not pause for torch change on the first cut.

7.3 Use

When the machine pauses for a torch change, here (at the time of this writing) is where the change is made in the controller:



8 Swap Ground and Touch Sensor for Final Cuts

USER NOTE. This feature is still in development.

8.1 Initial Settings

8.1.1 SwapGroundAndSensorAtEnd = <True/False>

SwapGroundAndSensorAtEnd = True.

With the Vernon machine you can nest all the way to the far end of the pipe—through the “dead zone” where the ground is located and where the leading sensor needs something to touch—by

1. Pausing the machine,
2. Moving the ground to the trailing end of the final part,
3. Changing the torch height touch sensor from the leading to the trailing sensor.

PypeServer supports this in GCode by pausing the machine at the appropriate cut (step 1 above) and instructing the operator to swap the ground and torch height touch sensor. To enable this feature in PypeServer set the machine parameter **SwapGroundAndSensorAtEnd = True**. Seen in Machine Settings:

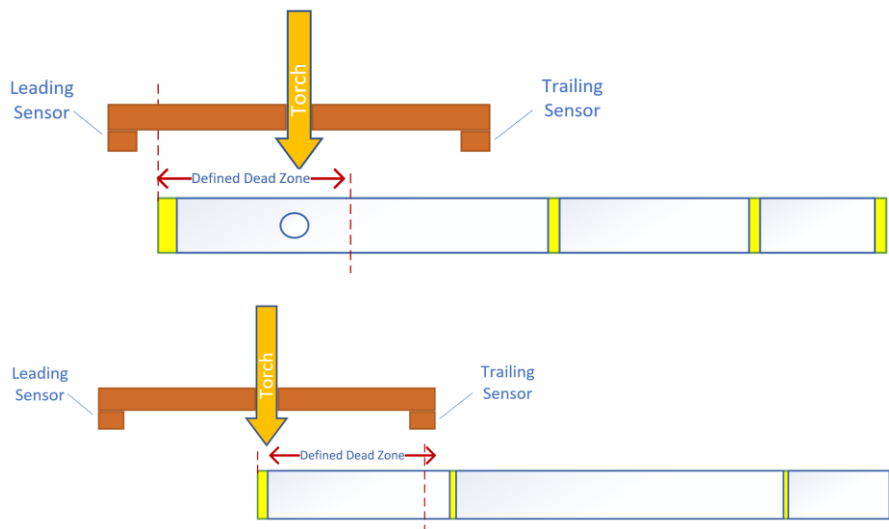
SwapGroundAndSensorAtEnd	True
--------------------------	------

8.2 Part Requirements

This section describes requirements for the final part. Not all parts will work for this feature.

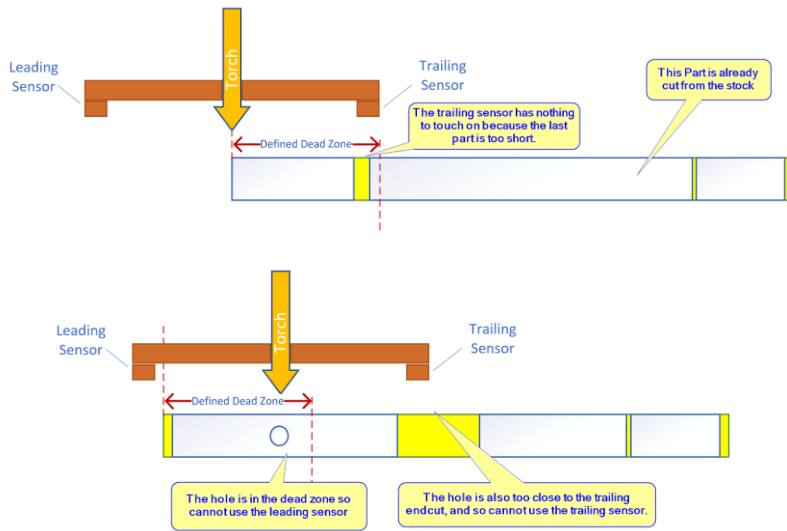
Your final part's first cut requiring this swap should be at least the distance from the torch tip to the trailing sensor, or there better be a piece of pipe for the machine to touch down on. Examples:

These will work. In both cases the trailing sensor can touch the trailing end of the final part.

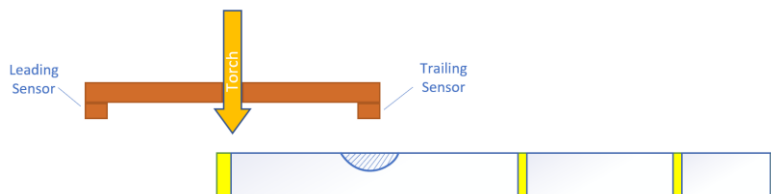


These will not work. One solution for these scenarios is to use a longer part with enough space from the first cut in the dead-zone to the trailing end of the part. Another would be to leave the previous cut part in place

such that the torch sensor can find that part, but you must make sure the sensor is not going to miss the pipe in a gap.



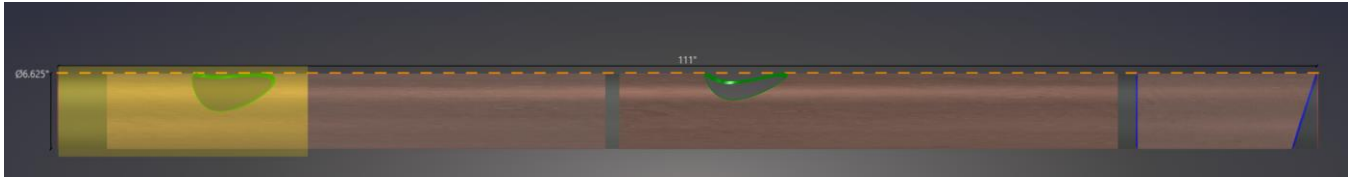
This may not always work. The operator must consider this scenario and check for this condition before starting any of the final cuts:



In this scenario, and the final cut is straight, user may consider rotating the pipe to get a sensor touch point.

8.3 Sample GCode

The following GCode for the following nesting:



Here's what you get before the start of the hole. The machine pauses and instructs you to swap the ground and sensor:

```
N63122 G01 X18.703 Y48.310 C-2.000 Z0.007 A0.007 F49.550
N64000 M16 ; Plasma Off
N64005 M13 ; Sensors Off
N64010 M33 ; Carriage Up
N64015 G04P 2.000 ; Dwell (Time Delay)
N64020 M34 ; Stop Carriage Up
N64025 G00 C0.000 A0.000
N71000 ; Part: ID:=62 Scheduled Part ID:: 143
N71005 ; Cut: Type=Hole, ID:=284
N71010 G70 ; Units are Inches
N71015 ;==== Swap the sensor and ground to the trailing end of the pipe before continuing.
N71020 ;==== Visually confirm that the trailing sensor will contact the pipe.
N71025 M22 ; Cycle Start Light On
N71030 M00 ; Program Stop
N71035 M23 ; Cycle Start Light Off
N71040 G00 X18.920 Y18.644 C-22.374 Z0.082 A-1.151 ; Go to the start of the cut
N71045 M12 ; Sensors On
N71050 G90 ; Absolute Positioning Mode
N71055 M15 ; Plasma On
N71060 ; Start of section: leadin
N73001 G01 X18.913 Y18.675 C-25.515 Z0.095 A-1.151 F30.940
N73002 G01 X18.905 Y18.706 C-27.753 Z0.105 A-1.151 F30.580
```

End of previous cut

Swap Instructions

Start of hole

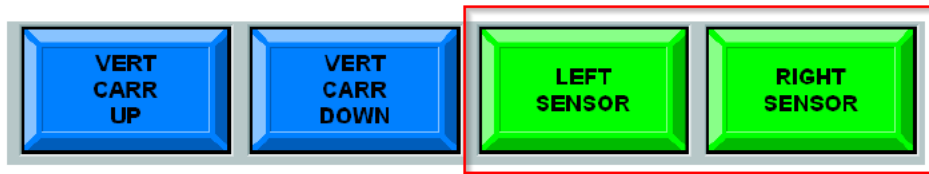
Here is what you get before the start of the final end-cut:

```
N73140 G01 X18.585 Y18.778 C-41.099 Z0.174 A-0.277 F28.460
N74000 M16 ; Plasma Off
N74005 M13 ; Sensors Off
N74010 M33 ; Carriage Up
N74015 G04P 2.000 ; Dwell (Time Delay)
N74020 M34 ; Stop Carriage Up
N74025 G00 C0.000 A0.000
N81000 ; Part: ID:=62 Scheduled Part ID:: 143
N81005 ; Cut: Type=Straight, ID:=283
N81010 G70 ; Units are Inches
N81015 ;==== Visually confirm that the trailing sensor will contact the pipe.
N81020 M22 ; Cycle Start Light On
N81025 M00 ; Program Stop
N81030 M23 ; Cycle Start Light Off
N81035 G00 X18.813 Y4.210 C0.000 Z0.000 A0.000 ; Go to the start of the cut
N81040 M12 ; Sensors On
N81045 G90 ; Absolute Positioning Mode
N81050 M15 ; Plasma On
N81055 ; Start of section: cut
N83001 G01 X18.642 Y4.210 C0.000 Z0.000 A0.000 F50.000
```


8.4 Setting the sensor

When instructed to set the sensor to the trailing sensor, you must set that manually in the Vernon machine software. Which end you set will depend on whether your machine is configured right or left-handed.

In WinMPM this will typically look like this:



In newer (VMD) versions the UI may look like this:

