

Setup and Use of Gang Tooling on Sherline Lathes and Chucker Lathes with Acorn Control

Setting up Acorn Control for Gang Tooling with Sherline Lathes
These instructions are a modification of the original Centroid instructions. They have been changed to better suit Sherline users.

Whether you are using one of our lathes or our chucker lathes, you will most likely be using tools that are mounted on both the front side (operator side) and the back side of the part. Therefore, we suggest that you follow these instructions and set your machine up for “Gang Tooling.”

Setup in Wizard or with Parameters

Select Gang and Orientation in the Wizard, or in the parameters.

- To get to the Wizard. Click on Utility F7, Acorn Wizard F10. Wait for the Wizard to come up. Click on Spindle #1 as shown below (see Figure 1) and change Lathe Orientation to “Gang” and Tool Orientation to “Front.”

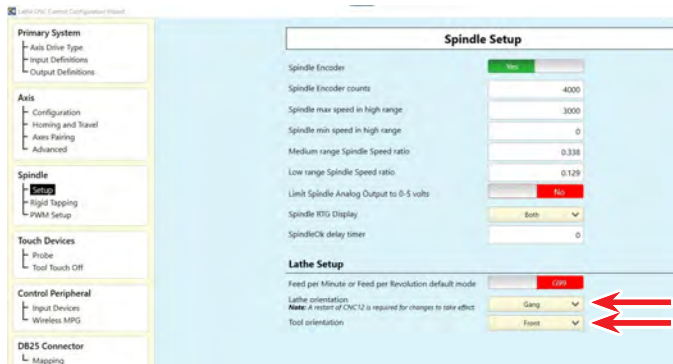


FIGURE 1

- To get to the parameters. Click on Setup F1, Config F3, enter password “137,” Enter, Parms F3. Now cursor to the desired parameter and enter the value.

The parameter for gang tooling is P163=1. This parameter allows you to choose the “Approach” side for your tool as either “Front” or “Rear” (see Figure 2).

Tool Orient	Tool Type	Approach	X Offset
OD	Turn	Front	0.0000
OD	Turn	Rear	0.0000
OD	Cutoff	Front	-3.0897
FFace	Groove	Front	0.0000

FIGURE 2

For Front Orientation P1=1. On our lathe and chucker lathe, the direction of travel may need to be reversed. In this case add (2) as stated on the parameter page and change this parameter to 3.0000 (see Figure 3).

000	-8.0000	020
001	→ 3.0000	021
002	0.0000	022

FIGURE 3

For Rear Orientation P1=0. In this case, we selected Front, because the main tools are all front-side tools. This also keeps the axis directions in the “Front orientation” as shown below.

If you don’t want to be asked for tool change set P6=1 for ATC (see Figure 4).

005	0.0000	025
006	→ 1.0000	026
007	0.0000	027

FIGURE 4

In addition to changing P6 to 1.000, you will also need to remove any “M06” commands from

your program wherever you have a tool change. **Examples:** Instead of “T0505 M06” change it to “T0505.” Otherwise the control will still stop and wait for you to click on “Cycle Start” before each tool will proceed.

With front orientation, the axes will have these directions (see Figure 5):



FIGURE 5

NOTE: It is critical that you understand the X-axis directions and how they apply to your front-side and rear-side tools.

- A. **With front-side tools:** To offset your tool using “Tool Wear,” the offset amount will be Negative to shift the tool toward the center of the part, and Positive to shift the tool away from the center of the part.
- B. **With rear-side tools:** The offset directions will be reversed. The offset amount will be Positive to shift the tool toward the center of the part, and Negative to shift the tool away from the center of the part.
- C. In addition to the X-axis direction, your program will have X+ dimensions for the front-side tools and X- dimensions for the rear-side tools.
- D. **For boring bars:** If you will be boring your part on the positive side of the part centerline (X+ side), you will describe your boring tool as a front-side tool. If your boring tool is aligned to bore on the rear side of the part centerline (X- side), Then, you will describe your boring tool as a rear-side tool.

Setup Tool Change (G28)

We suggest setting up your tool change position at this time so you can use the “Tool Check” as a quick and easy way to move all of your tools to a clearance position.

1. To get to the WCS Table page to set your G28 position, click on (Setup F1, Part F1, WCS Table F9, and Origin F2).
2. Jog your crossslide to a clearance position in the Z-axis first. This position should be decided using your longest tool. Move your longest tool to a position where there is adequate clearance between this tool and the front of your part.
3. Press Alt D to see the Machine Current Position in the DRO.

4. Courser down to the Z-axis under “Return #1 (G28).” Then, click on “Teach F2” to input the machine position for the Z-axis (see Figure 6)



FIGURE 6

5. Now, move the X-axis to a clearance position that will work for all of your tools (this is generally going to be in the middle of all of your tools).
6. Now, courser down to the X-axis on (G28) and highlight it. Then, click on “Teach F2” again. These steps will input the machine position for the X-axis.
7. An alternative to the “Teach” button would be to double-click on the desired axis on (G28) and input the axis value from the Machine DRO screen (see Figure7).

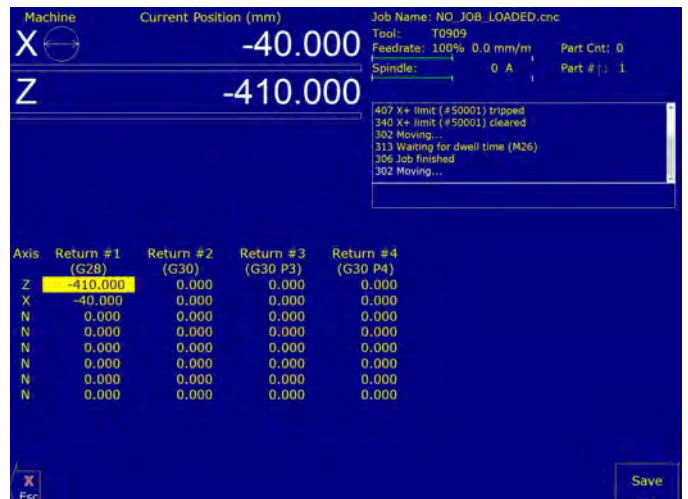


FIGURE 7—Return #1 (G28) table.

8. Now you are set. Any time that you are touching off a tool or moving your tools in Jog or MDI mode, you can click on the “Tool Check” icon and your machine will move to this G28 Return position. Just make sure that

your tools are all at a clearance position in the Z-axis before you click on the Tool Check icon (see Figure 8).

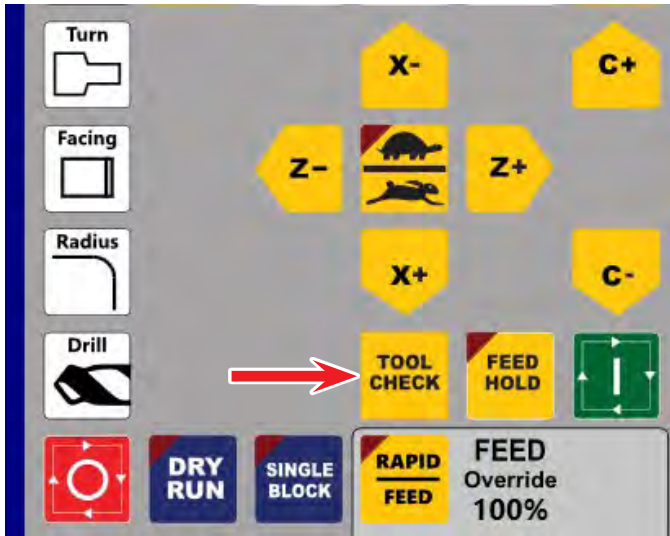


FIGURE 8

NOTE: Another reason to use G28 is that you will see the rapid moves in the Graph while making a program starting from G28 position. Then, you can be sure to have clearance for the tool. If one of the red rapid lines goes across the part, you will have a crash. Better to change the program then before hitting cycle start.

Now you have to decide if you want to use G28 for tool change or not.

By default, Intercon Setup is setup >Suppress G28 for tool change = No<.

That means if you change to another tool you have a simultaneous X Z rapid move to the G28 position. You can suppress this with >Suppress G28 for tool change = Yes<, or set P114=1.

But in both cases, you have to make a clearance move in Z before changing to the next tool to prevent the tool hitting your work piece and make the first move in X with the new tool to a safe Z position.

You can also set to stop the spindle during a tool change: Scroll down to find it on the Intercon Setup (Suppress G28 for tool change :No, see Figure 9).

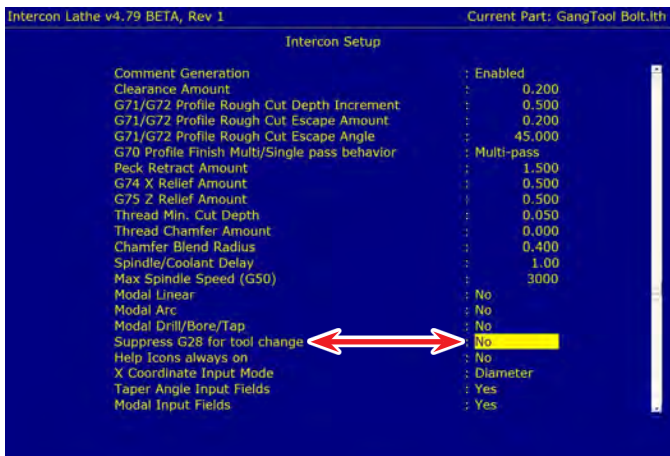


FIGURE 9

Tool Setup

We will be using tool #2 as our Reference tool to set our “part zero” in the X and Z. Tool #2 is our facing and turning tool. We chose this tool to set the X and Z “part zero” because this is the tool that will be machining the face and the OD of the part. All other tool operations will be done in relation to these surfaces and this tool.

Setting X-Ref First

1. First go to the Tool Wear and set all of the X and Z tool wear values to (0.0000). To do this click on Setup F1, Tool Offset F2, Tool Wear F9. Double Click on any X or Z value that is not (0.0000) and change it to (0.0000). Click on Save F10 (see Figure 10).



FIGURE 10

2. Use Jog to move your tool into position. In this case we are using tool #2 which is a Rear-side tool. Turn the spindle on. Use “INCR” to jog your X-axis in until you hear/see it cut the part. Use the Jog in “CONT” mode in the Z to take a skim cut on the OD at the X touch off position. Jog away from the front of the part (in Z-axis only). Leave your X-axis at the cut position.
3. Measure the cut diameter with micrometers. We measured (.5000”).
4. Click on Setup F1, Part F1, and then Set X F8.
5. Now this is important. The actual diameter is measuring .5000. However, tool #2 is a Rear tool. Therefore, the value that we enter for the X Diameter will need to be a Negative value to define which side of the part the tool is on. Click on the Part Position and enter (-.5000) (see Figure 11).

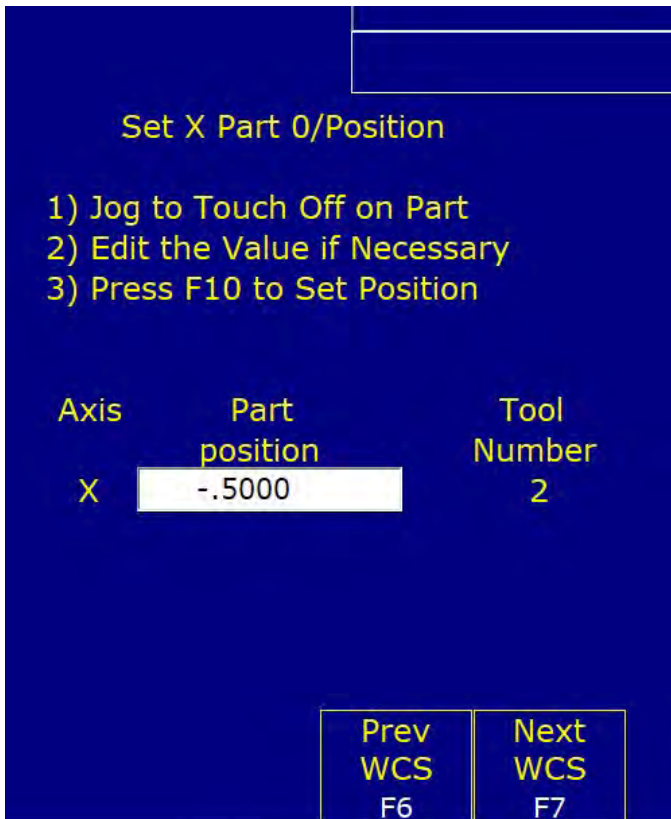


FIGURE 11

6. Change the “Tool Number” to the reference tool number (in our case Tool #2)
7. We set the “Set All WCS” to “Yes.” In order to change from No to Yes, press the space bar (see Figure 12). Then, click on Set F10.

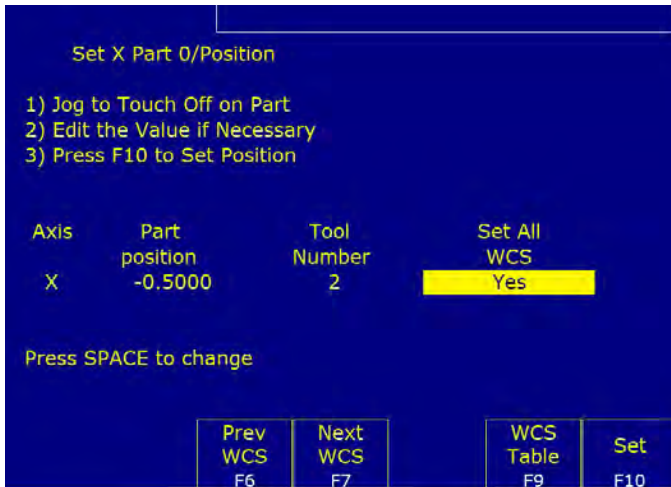


FIGURE 12

8. Hit ESC or click on “X Esc” to change to the Z-axis.
9. We did a face cut on the front of the part to set the Z Part Zero position. With the Ref Tool (#2) still at the cut position, Highlight the Part Position. This will default to (0.0000) and be in Yellow. Make sure the “Tool Number” is correct. Then, click on Set F10 (see Figure 13). This will set the current cut point to Part Z Zero.

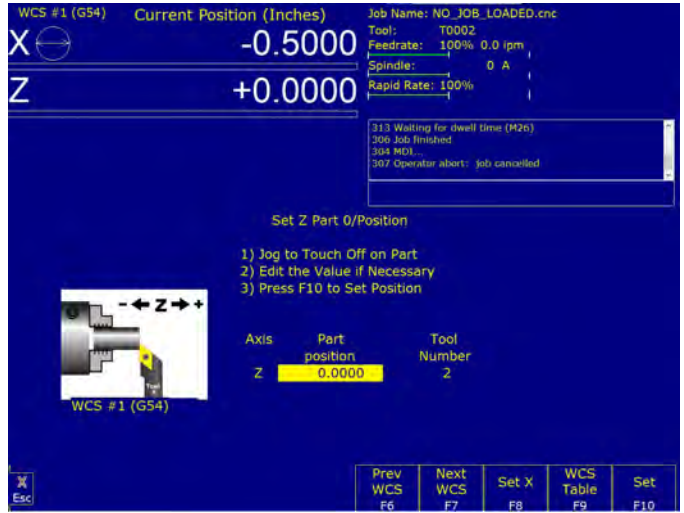


FIGURE 13

Setting up Tool Lib (Approach and Nose Vector)

1. Define the tools you are using. It is very important to have the approach and nose vector correct.
2. If you highlight a turning tool that has the Approach set to Front, you will see the Nose Vectors (3, 8, and 4) as shown below. Choose the vector number that represents the orientation of your tool to the part and enter that number for the Nose vector (in this example, “3”), (see Figure 14).

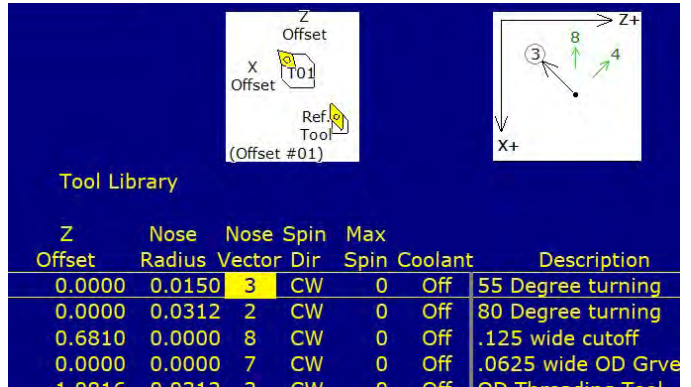


FIGURE 14

3. If you highlight a turning tool with a “Rear Approach,” the nose vectors (2, 6, & 1) will be shown for your choices (see Figure 15).

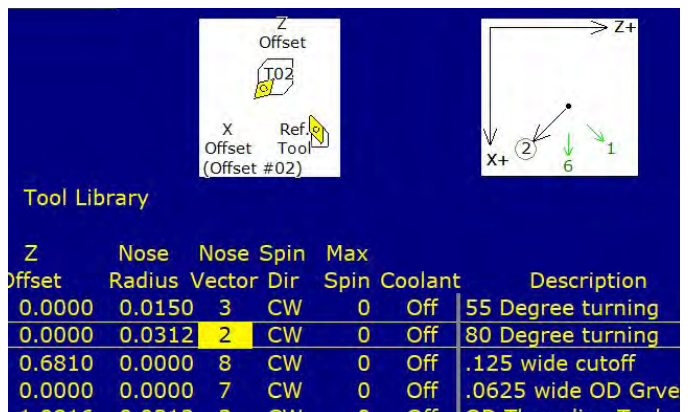


FIGURE 15

Measure Tools with Skim Cut

We will be setting the X Diam and the Z Ref with T0202 (our reference tool).

1. First, I like to call up each tool that I am setting so the control knows which tool I am using and the DRO position screen will show the DRO position for that tool. To do this, click on MDI F3. Then, type in the tool that you are using (in this example, "T0202"), (see Figure 16).

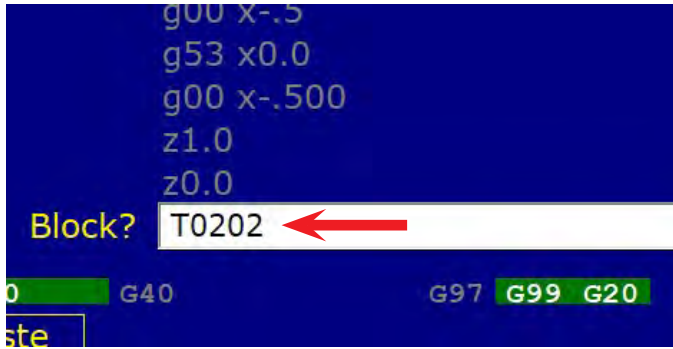


FIGURE 16

2. We just finished setting the "Part Zero" for the Z-axis with our reference tool #2.

Tool #2 should be at the part Z zero position, so we will start by measuring tool #2 in the Z-axis.

3. To get to the Tool Library (Setup F1, Tool Offset F2, Measure Tool F2). Now highlight the tool and the axis that you are going to measure (in this example we are highlighting tool #2, Z Offset). As you can see below the DRO is at Z0.0000. Now click on Measure Offset Z F6 (see Figure 17).

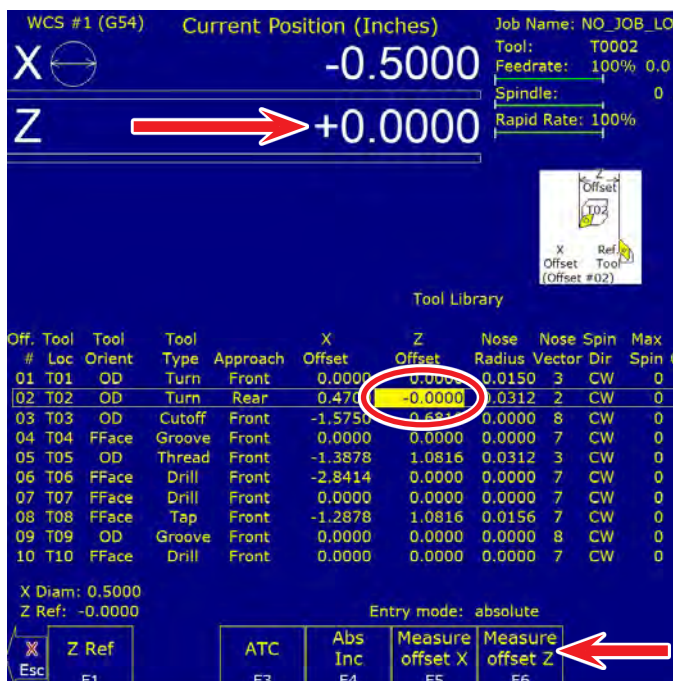


FIGURE 17

4. Then, click on Measure Here F10 (see Figure 18).

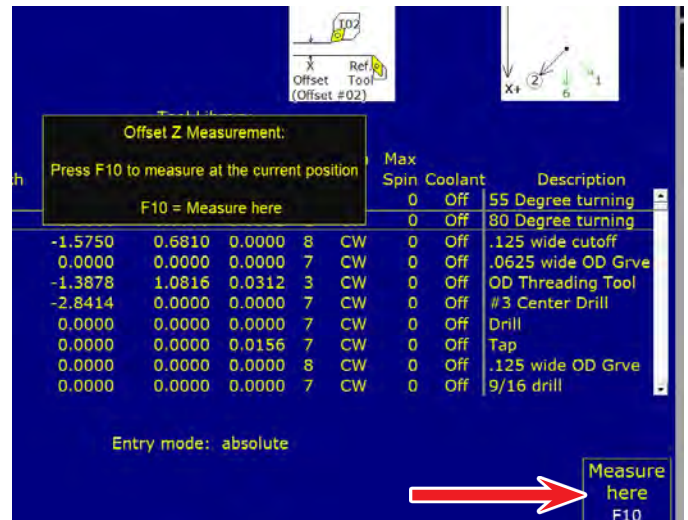


FIGURE 18

5. After you click on Measure Here the screen will go back to the main choices. At this time, we want to set the Z Reference with our Reference Tool (Tool #2). The cursor will still be on tool #2, Z Offset. Click on Z Ref F1 (see Figure 19).

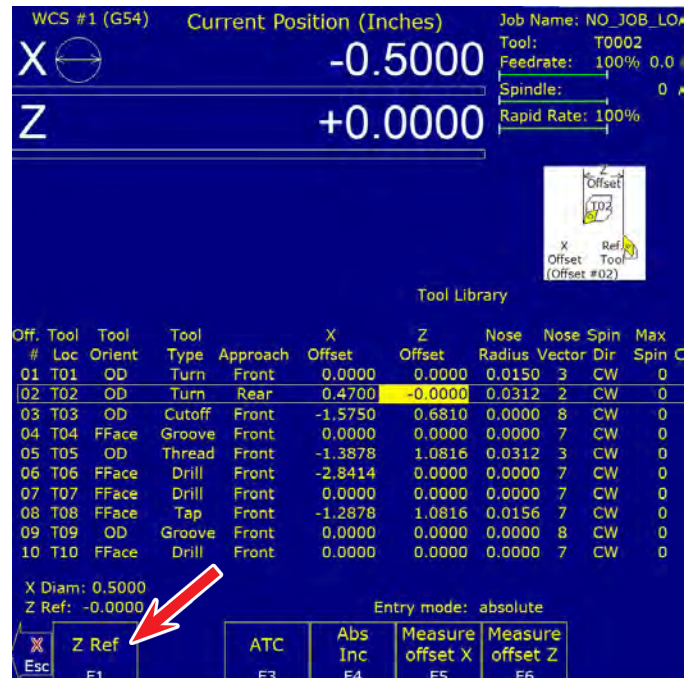


FIGURE 19

6. If your tool is at Z Zero (which ours is), click on Save F10. If you have moved your tool away from the face by .010" (as an example), Then, enter .010 into the white box and Then, click on Save F10 (see Figure 20).

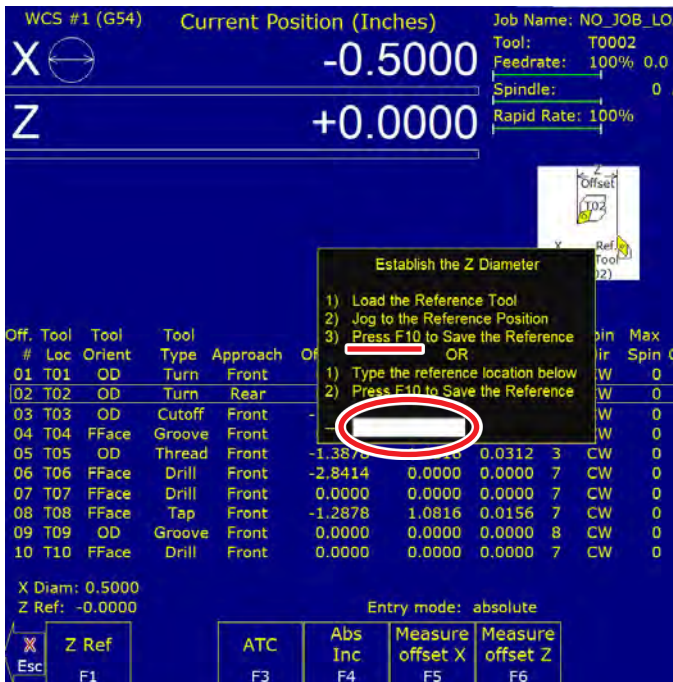


FIGURE 20

7. Now we have set our reference tool #2 at Z0.000, at the Part Z Zero position. All other tools will now be set in reference to this tool #2 position. The offset values for each tool will be the Distance and Direction Difference from each tool, to tool #2.
8. Now we are going to set the X diameter with our reference tool #2.
 - A. Either make another turning cut on the OD of the part and measure it as we did to achieve the Part zero in the X-axis earlier, or move your tool in the X until it touches the OD of the part that was turned to .5000. We are going to move the tool until it touches the .5000 diameter.
 - B. With the tool in the touch position we are going to establish the X Diameter. Cursor to the X Offset for the tool that you are setting (tool #). Then, click on X Diam F1.

NOTE: This Next Set Is Very Important. When the “Establish the X Diameter” window comes up you will enter the part diameter where the tool is touching into the white box. Because tool #2 is a Rear-side tool, you will enter a negative value to define to the control which side of the part this measurement is taken from. Enter (-.5000). Then, click on Save F10 (see Figure 21).

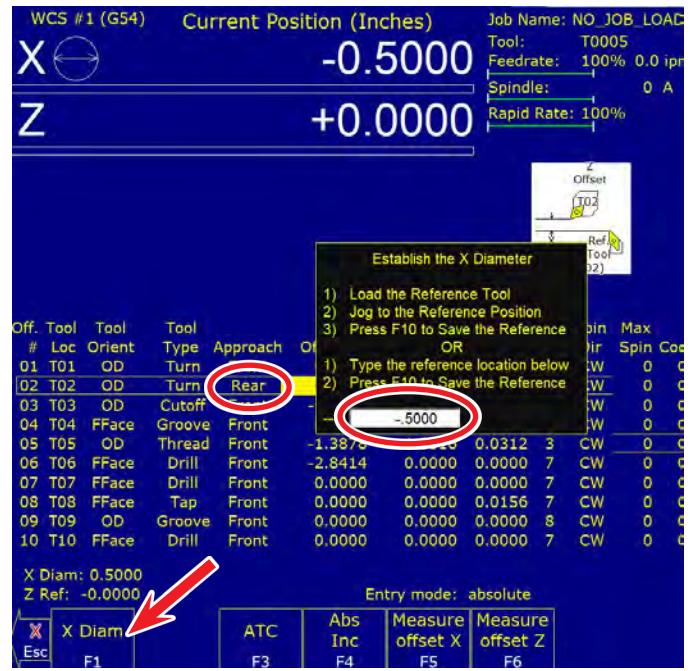


FIGURE 21

C. After you click on Save F10, the screen will show “X Diam: -0.5000” (see Figure 22).

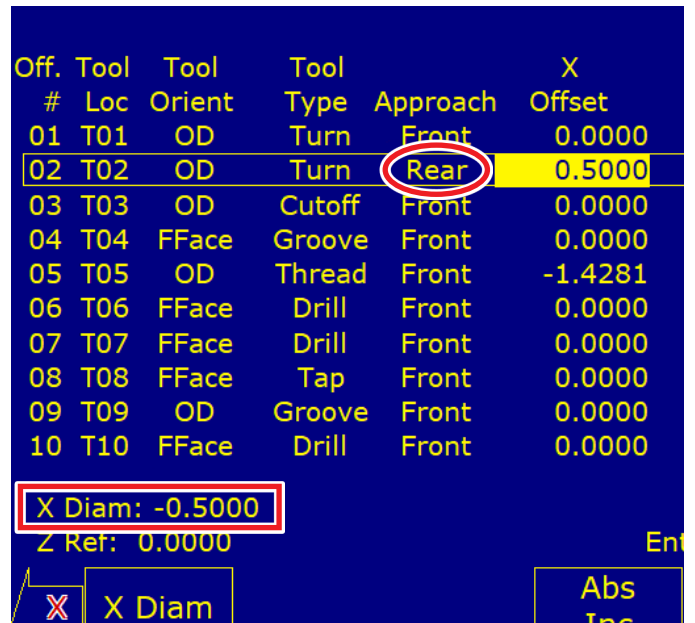


FIGURE 22

D. Click on “Measure Offset X F5”, the “Measure Here F10” (see Figure 23).

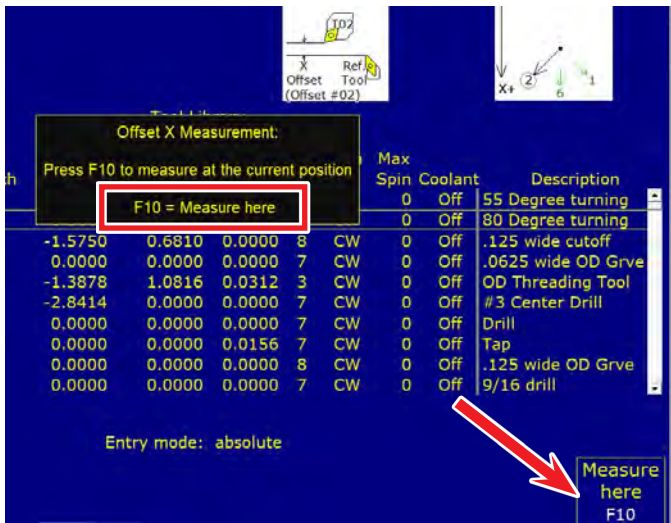


FIGURE 23

9. Now, we are going to set the rest of the tools. The rest of the tools are front-side tools. We are going to set tool #5, which is a threading tool.

A. Click on MDI F3. Then, type in the tool that you are using (in this example “T0505”). This will set the control and position to tool #5

B. Now move tool #5 to the side of the part using Jog Mode. Bring it in until it touches the side of the part that you turned to .5000.

NOTE: Very important! All of the rest of the tools are front-side tools. Therefore, you will need to set the X Diameter using a Positive Value!

C. With the tool in the touch position we are going to establish a New X Diameter. Cursor to the X Offset for the tool that you are setting (tool #). Then, click on X Diam F1.

When the dialogue box opens, enter (.5000), (see Figure 24).

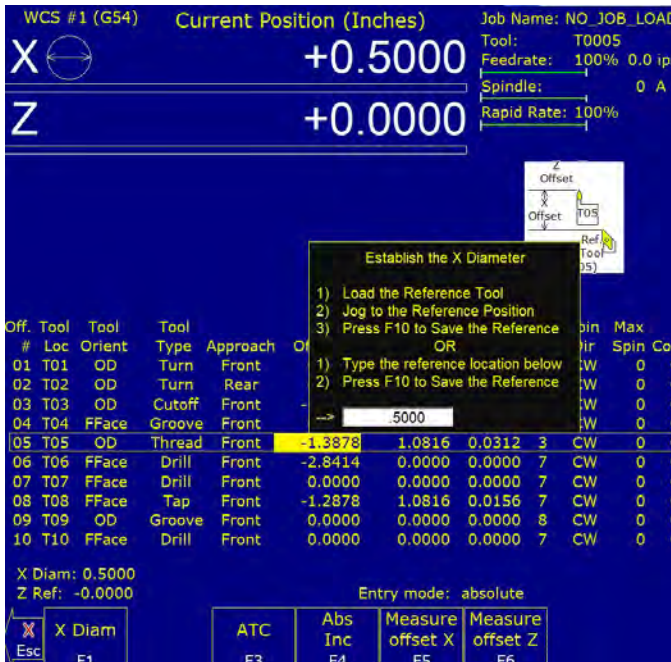


FIGURE 24

D. After you click on Save F10, the screen will show “X Diam: 0.5000” (see Figure 25).

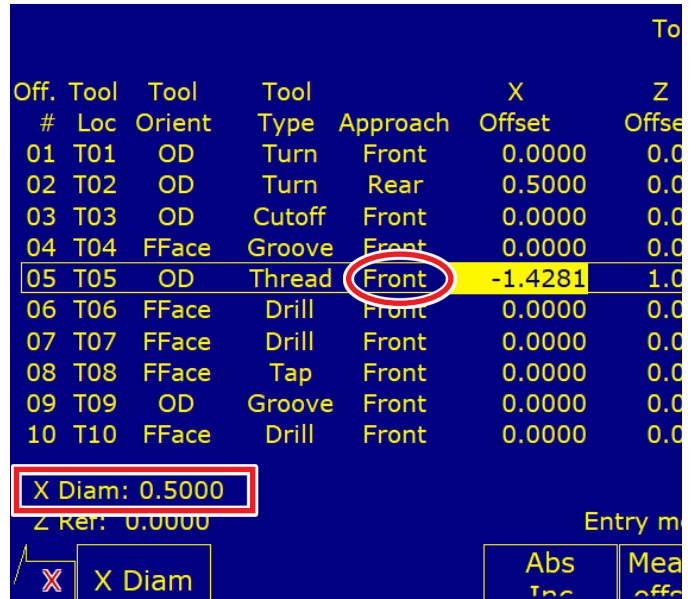


FIGURE 25

E. Now we are going to set the tool offset for tool #5 in the X-axis. With the tool in the touch position click on “Measure Offset X F5”, the “Measure Here F10” (see Figure 26).

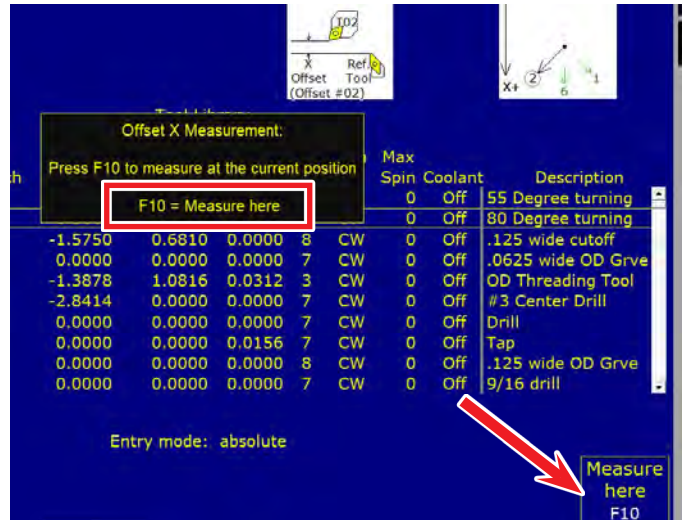


FIGURE 26

F. Now Jog the tool in the Z-axis until it clears the part. Then, move it in on the X-axis. Then, jog it in INCR until the tool touches the front of the part.

G. Cursor over to highlight Offset Z for tool #5. Now click on Measure Offset Z F6.

Then, click on Measure Here F10 (see Figure 27).

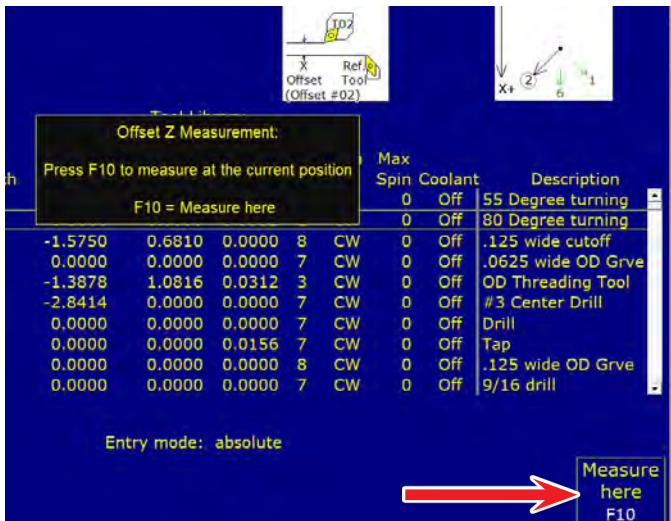


FIGURE 27

- With the tool in front of the part, you can click on “Tool Check” to move to the clearance position that you set for G28 at the beginning.
- Follow this procedure for each tool until all of your tools are set.

Wear Offsets for Front and Rear Tools

It’s important to understand which direction your wear offsets must be for both the front a rear tools. In order to offset a rear tool toward the center of the part, you will use a positive value. In order to offset a front-side part toward the center of the part, you will use a negative value. The picture below shows tools #2 (rear) and tool #5 (front) both offset .060 toward the center of the part (see Figure 28).

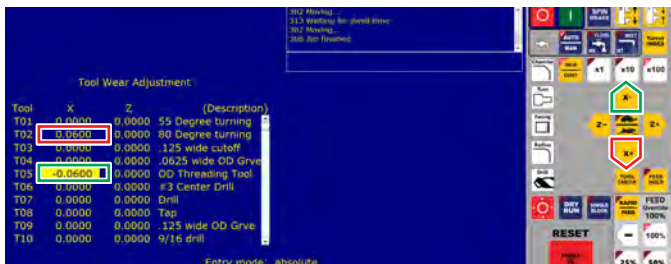


FIGURE 28—The red outlines indicate a positive value and the green outlines indicate a negative value.

Measure Tools with Tool Touch Off Probe

- For the tool touch off, we are using a 3D Touch Probe Model: TP06, DC: 5-24V, with three wires (see Figure 29). AliExpress Link: [The latest V6 anti-roll 3D Touch Probe edge finder finds the center of the desktop CNC probe compatible with mach3 and grbl \(aliexpress.us\)](https://www.aliexpress.us/item/32961711111.html)
- The touch probe is held in our 6mm end mill holder (P/N 3076) (see Figure 30).

We replaced the probe tip with dial indicator extension and a flat contact point for a dial indicator.

Amazon Link: [Flat Contact Points for Dial Digital Indicator Depth Gauge Thickness Gauge 4-48 UNF Thread 15mm Diameter 5mm Length](https://www.amazon.com/dp/B07K111111)

Amazon Link: [Dial Digital Indicator Extension Stem Rod 4.5mm Diameter 4-48 UNF Thread 10mm to 105mm Long by Assembly \(4-48 UNF Thread\)](https://www.amazon.com/dp/B07K111111)



FIGURE 29

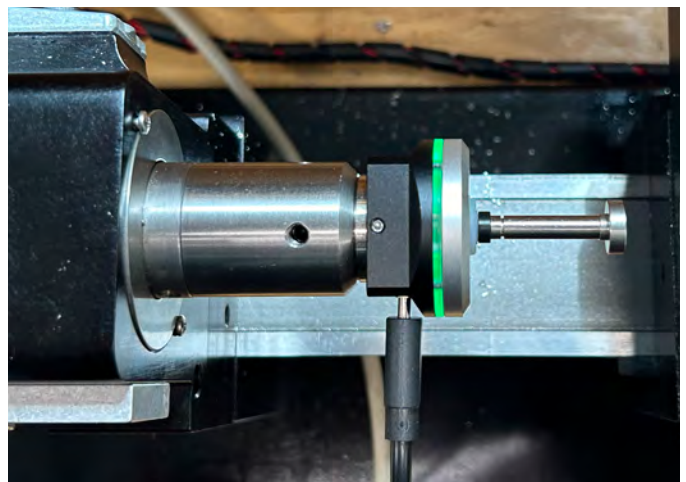


FIGURE 30

- You connect the three wires from the touch sensor to the 9400 controller box using the DB9 connector that goes into the Aux Tool Setter / Touch Probe DB9 (see Figure 31).

A. Probe Wires to the DB9 connector as follows:

Touch Probe Wires ---To --- DB9 Wires

Red ---To --- White Wire on Pin # 2 (which goes to IN6 on Acorn board)

Yellow ---To --- Yellow Wire on Pin # 3 (which goes to IN7 on Acorn board)

Black ---To --- Black Wire on Pin # 9 (which goes to Common Ground)

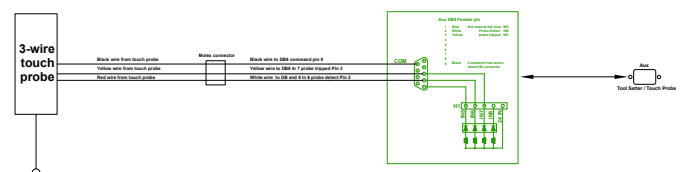


FIGURE 31—The complete schematic file is acorn_mill_connection_back_panelconfig_touch_probe.pdf.

B. Plug the touch probe wire into the side of the touch probe. The green light on the touch probe should come on if it is wired correctly (see Figure 32)

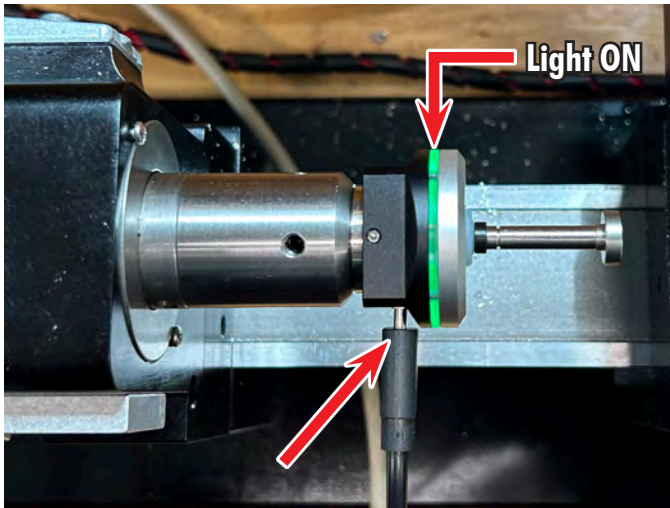


FIGURE 32

4. Once you verify that you have power to the touch probe, you need to check the setting in the Acorn Wizard. Click on Utility F7, Acorn Wizard.

A. Click on “Input Definitions”. Then from the Lathe list drag “Probe Detect” to IN6 and “Probe Tripped” to IN7 (see Figure 33).

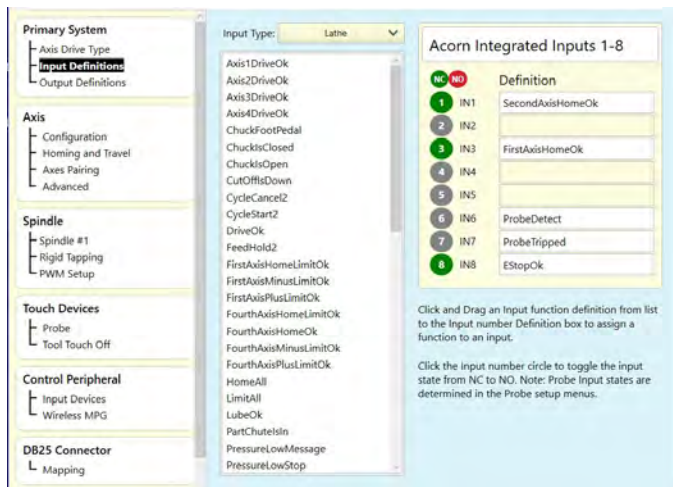


FIGURE 33

B. Click on Probe and input the values shown below (see Figure 34).

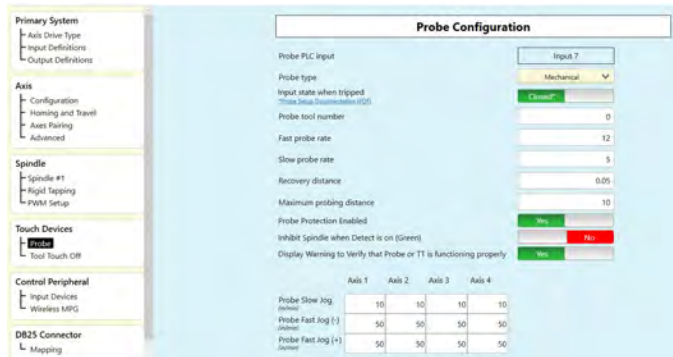


FIGURE 34

C. Click on “Tool Touch Off” and input your probe information. The picture below represents our probe dimensions (see Figure 35).

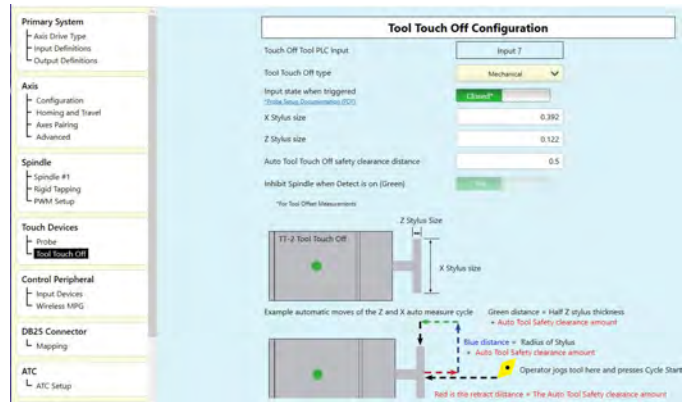


FIGURE 35

NOTE: Our actual X Stylus Size was .392. When we entered the stylus diameter of .392, the tools went past the part centerline (X0.0) when commanded in MDI (see Figure 36)

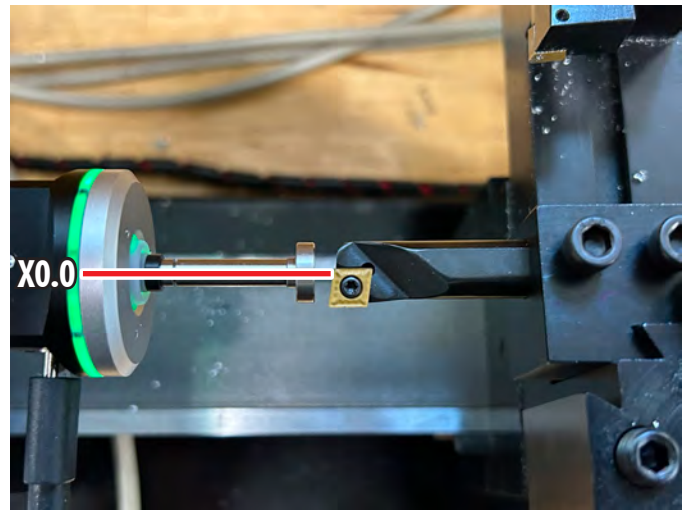


FIGURE 36

To fix this, we entered the stylus Radius instead of the Diameter and that fixed the problem. We also have parameter 55 set to (0) for Diameter (see Figure 37).

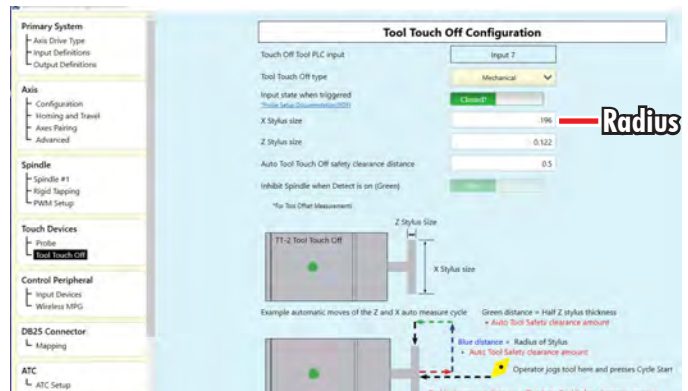


FIGURE 37

5. Now we are going to check to see if the control is receiving a signal from the touch probe. On the keypad, press (Alt I) and the Input screen will show on the control (see Figure 37).

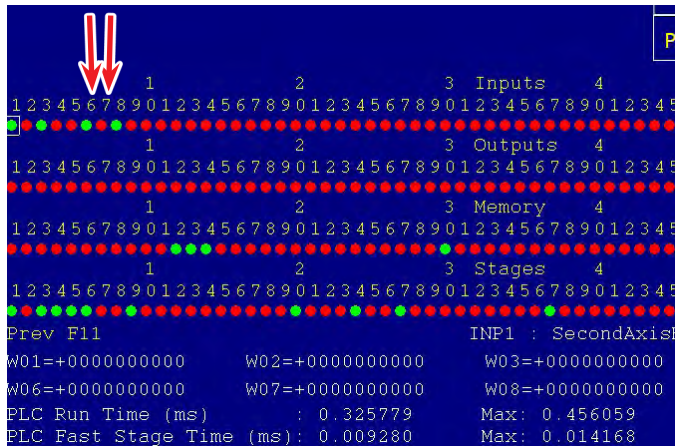


FIGURE 37

- With the Input screen up, if the touch probe is connected and there is power to the probe, Number 6 should be Green.
 - When you activate the touch probe by pushing on the tip, Number 7 should change from Red to Green.
 - To turn off the Input Screen, type (Alt I) again.
6. As stated earlier in these instructions, I like to tell the control which tool I am currently using. Click on MDI F3. Enter the tool that you are using (in our case "T0202"), Click on Cycle Start. The control screen should show the tool that you entered (see Figure 38).



FIGURE 38

7. Now we are ready to touch off the tools.
- Jog the tool so the cutting tip is approximately at the center of the stylus in the X-axis. Then Jog to a clearance point in the Z-axis (about 1/2" in front of the stylus) (see Figure 39).

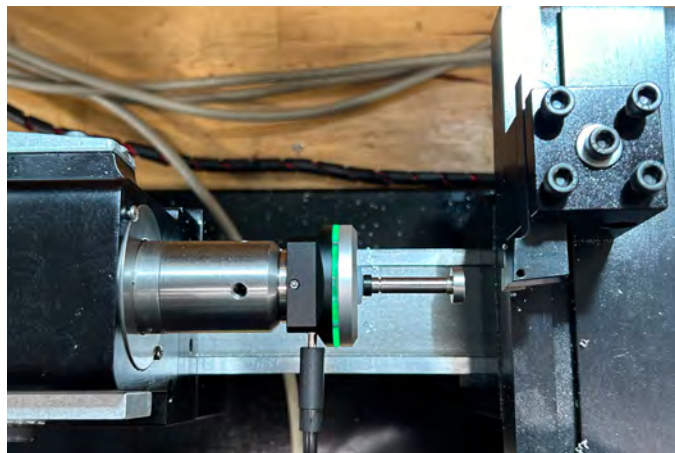


FIGURE 39

B. Click on Setup F1, Tool Offset F2, Highlight the tool / Axis Z, Measure Tool F2, Measure Z & X F7 (see Figure 40).

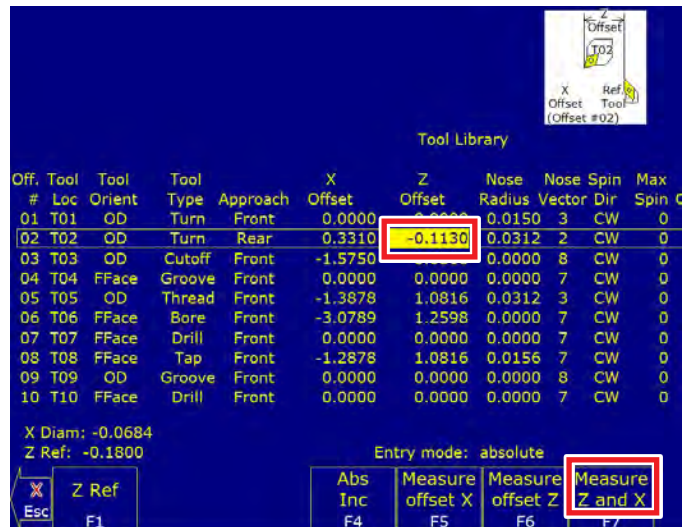


FIGURE 40

C. At this point a box will come up and ask you if need to "Change the X Direction". This is the direction that the tool is going to move in order to touch the probe. If the direction is correct, you click on Cycle Start to begin the tool touch off program. If you need to change the X direction, click on "Change X Direction F1. This will change the X direction. Then click on Cycle Start (see Figure 41).



FIGURE 41

D. After you click on Cycle Start, the tool will move toward the probe and touch off in the Z-axis first, and then the X-axis as shown on the probe setting page (see Figure 42).

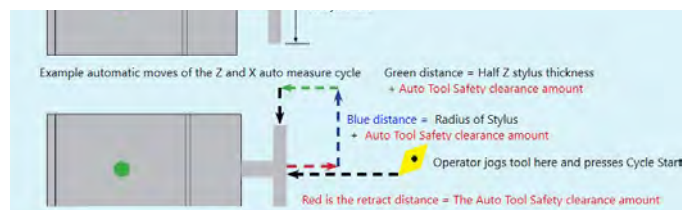


FIGURE 42

E. After the tool has touched off on the probe click on “Save F10” to save the setting to the tool offset page.

If you try to leave the tool offset page the screen will ask you if you would like to save the changes, click F1=Yes (see Figure 43).

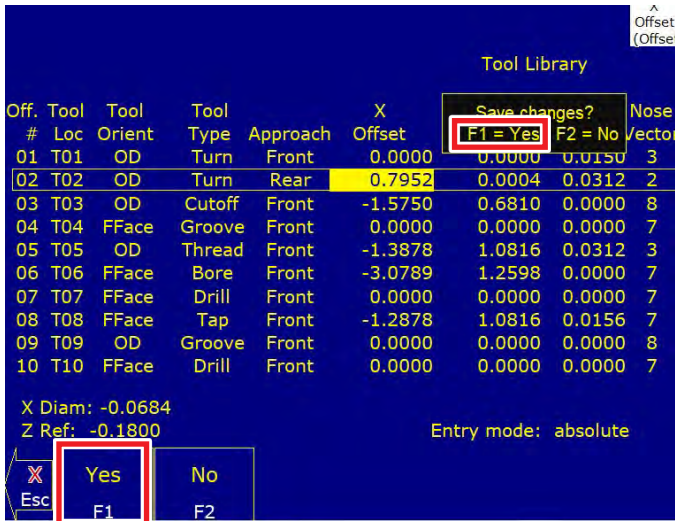


FIGURE 43

8. Now we are going to use our reference tool (tool #2 for these instructions) to set the “Z Ref” on the tool touch probe.

- Click on Setup F1, Tool Offset F2, then highlight the tool / axis (Z).
- Click Z Ref F1 (see Figure 44).



FIGURE 44

C. Click on Z Ref Auto F1. Then click on Cycle Start (see Figure 45).



FIGURE 45

9. Now we are going to set the Part Zero in Z and X-axis.

- Mount your part in the chuck or collet with it extended away from the collet by the correct distance.
- Again we will be using our Reference Tool to set the part zero (tool #2 for these instructions).
- Jog the tool up to the front of the part until it touches.
- Click on Setup F1, Part F1.
- Set the “Part Position” to (0.0000). Set the tool # to tool 2 (Ref Tool). Then click on Set F10 (see Figure 46).

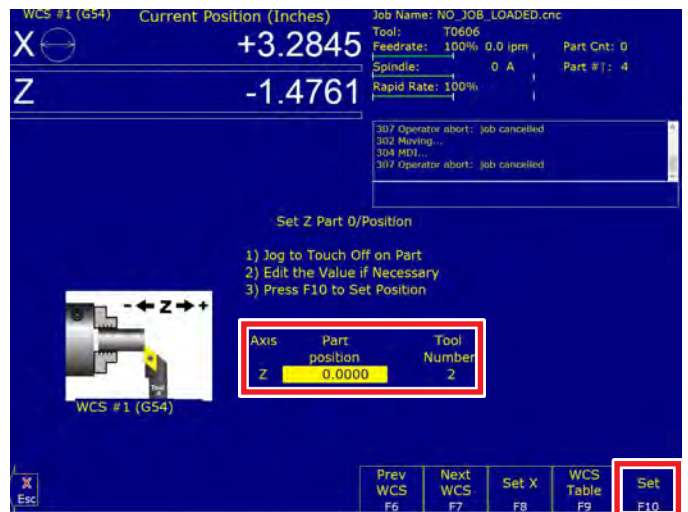


FIGURE 46

F. Once the Z is set we will now set the X Zero. Click on “Set X F8” (see Figure 47).

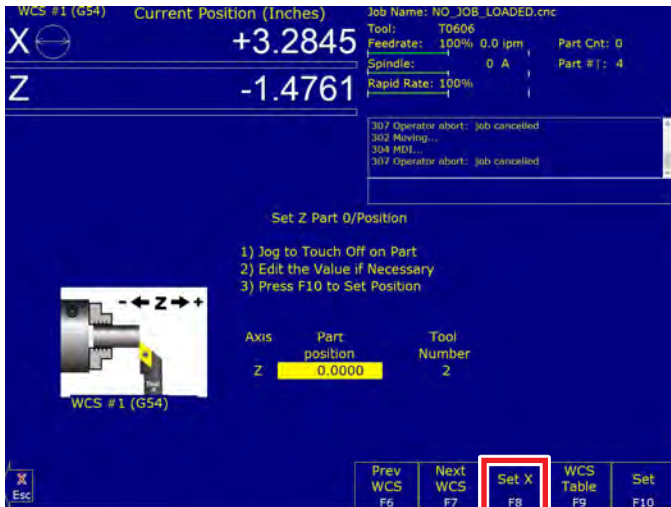


FIGURE 47

G. Jog the tool into position to touch the diameter of the part. Our part is .500” diameter. Our tool is on the backside of the part. For our tool on this part we will be setting the X-axis at (-0.5000) Part Position, Tool Number 2, and “Yes” for Set All WCS. Then click on Set F10 (see Figure 48).

NOTE: If our tool was a front side tool we would have set the Part Position at (0.5000).

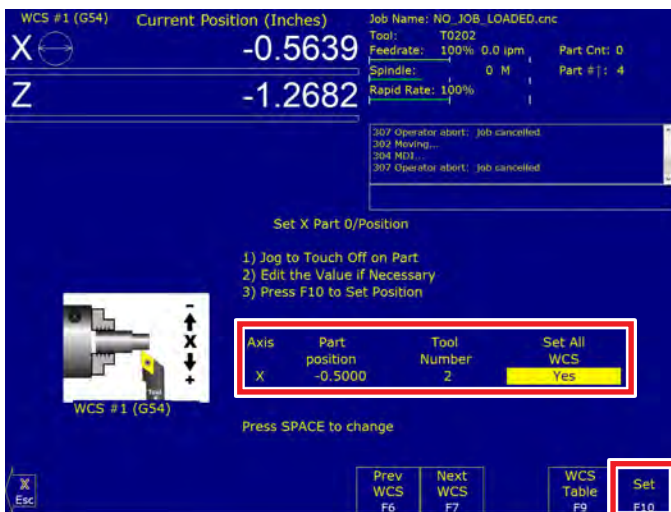


FIGURE 48

10. Now your reference tool is set. Your reference tool is set to the tool touch probe. Your Z and X Reference are set, and your Part Z and X zero are set.

Follow the instruction in #7 (on page 10) to set the rest of your tools using the tool touch probe.

11. Touching off a boring bar.

A. The boring bar in the picture below will be described as a “front side” tool on the tool offset page. When it is cutting it will be cutting on the X+ side of the

part centerline. However, in order to touch off this tool on the tool touch probe, you will need to touch off on the X- side of the touch probe and tell the control to move this tool in the (X+) direction when it touches off (see Figures 49, 50, and 51).

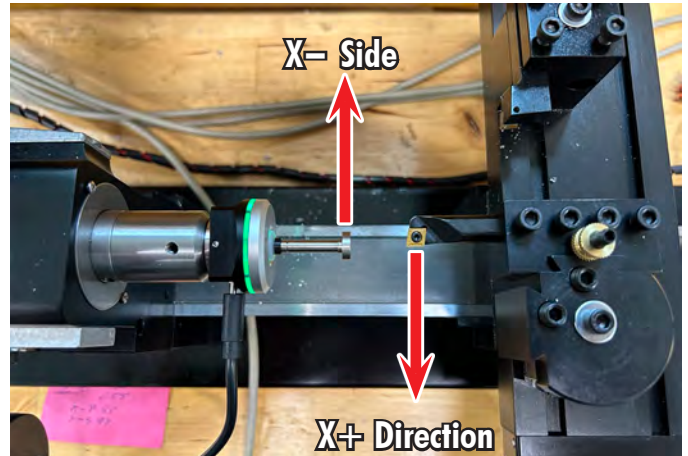


FIGURE 49

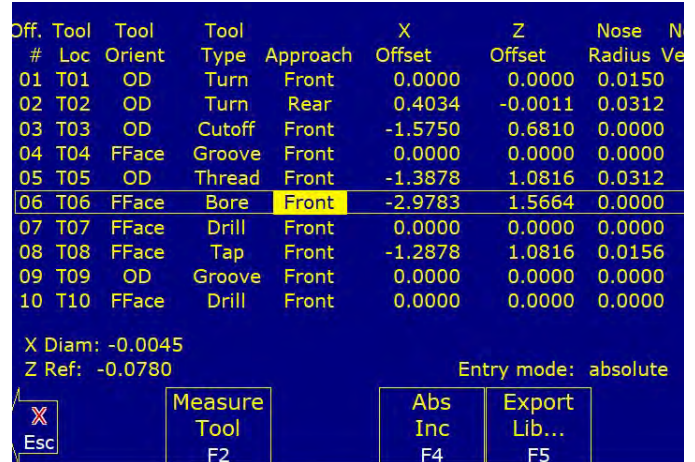


FIGURE 50—Boring tool front.



FIGURE 51—Boring Tool, Change Direction to X+.

B. The boring tool will now touch off on the touch probe as shown below (see Figure 52).

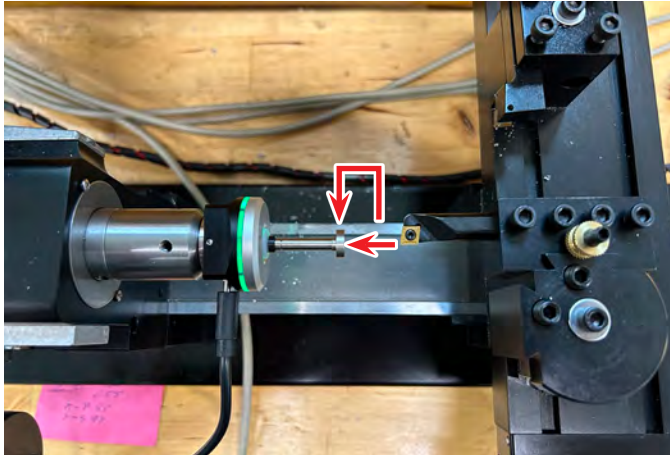


FIGURE 52

Intercon with Gang Tooling

Now, we will make a simple bolt with just two tools: one OD Turn tool, which will approach from the rear, and a cut-off tool, which will approach from the front.

Keep in mind that if the primary orientation is front, a tool with an orientation from the rear has to be programmed with negative X values. So, it is obvious that you cannot use two tools in one cycle with different orientations. The roughing and finishing tools must either be front or rear.

NOTE: You can manually change the X coordinates in the intercom program of a rear side tool to Negative values. Then you can use both front and rear side tools as rough and finish tools.

Here, I used Pre/Post Cycle Pos. for approach and retract (see Figure 53).

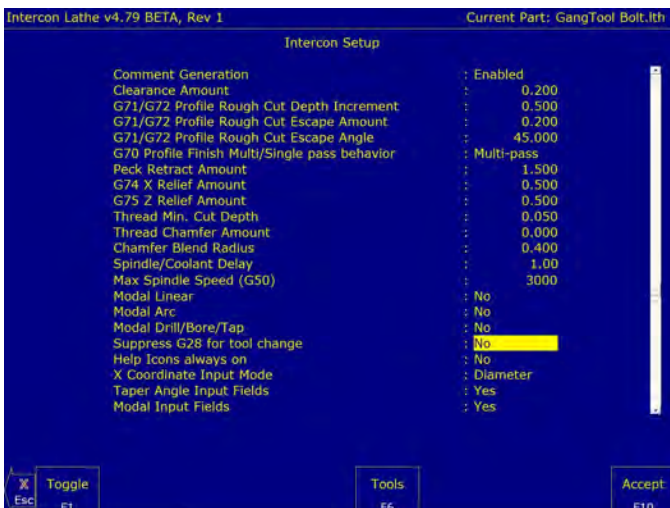


FIGURE 53

You can see in the rapid moves that they are not crossing the part; there should be enough space for the tools while rapidly moving to the part. Make sure that the longer tools do not move into the chuck while turning with other tools (see Figure 54).

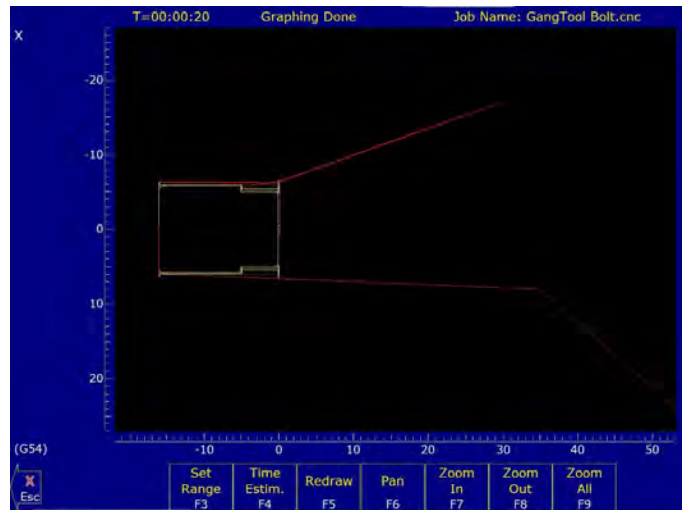


FIGURE 54

Run Program

If the graph looks OK and you are confident, set the feed to 100% and hit Cycle Start.

If you are a careful person, set the feed to 25% / F4 Run / F5 Block. This turns on Single Block Mode, which means you have to press Cycle Start for each line of your g-code for the first part just to make sure everything moves the way it should (see Figure 55).



FIGURE 55

G-code of this Part

```

; ICN_PATH = C:\icn_lath\GangTool Bolt.lth
; CNC code generated by Intercon v4.79 BETA, Rev 1
; Description: Gang Tool Bolt
; Programmer: Uwe
; Date: 10-Jun-2022
; --- Header ---
N0001 G21 ; millimeter measurements
::: --- Stock Dimensions ---
::: X- = -6.0, X+ = 6.0
::: Z- = -16.0, Z+ = 0.0

```

.... ---

G50 S3000 ; max CSS spindle speed G40 ; Cutter Comp Off

M5 M9 G28

; --- Rapid ---

N0002 T0900 ;GangOD CCGT06 M9

G96 S90.0 M3 G4 P1.0

G0 X-13.0 Z0.2 T0909

; --- Facing --- N0003 X-13.0 Z0.2

X-13.0 Z0.2

G94 X1.0 Z0.0 G99 F0.1

; --- Profile --- N0004 G0 X-12.5 Z0.0

G71 U1.0 R0.2

G71 P5 Q9 U0.4 W0.03 F0.2

; --- Linear ---

N0005 G1 X-9.5 Z0.0 G99 F0.1

; --- Linear --- N0006 X-10.0 Z-0.5

; --- Linear --- N0007 X-10.0 Z-5.0

; --- Linear --- N0008 X-11.5 Z-5.0

; --- Linear --- N0009 X-11.5 Z-16.0

; --- Finish Pass --- N0010 G0 X-12.5 Z0.0

G70 P5 Q9 U0.4 W0.03

; --- Profile End --- N0011

G28 T0900

G50 S3000 ; max CSS spindle speed

; --- Cutoff Cycle --- N0012 T0400 ;MGGN200 AL

M9

G96 S60.0 M3 G4 P1.0

G0 X16.0 Z35.0 T0404 X12.0 Z-16.0

G1 X11.6 Z-16.0 G99 F0.05 X10.3915 Z-16.0

G0 X12.0 Z-16.0 X12.0 Z-15.3957

G1 X11.6 Z-15.3957 X11.6 Z-15.4101

G3 X11.3657 Z-15.6929 I-0.4 G1 X10.9859 Z-15.8828

G3 X10.4202 Z-16.0 I-0.2828 K0.2828 G1 X10.3915 Z-16.0

G0 X12.0 Z-16.0 G1 X11.6 Z-16.0 G75 R1.5

G75 X-1.0 Z-16.0 P11.6 Q0.0 R0.0 G0 X12.0 Z-16.0

X16.0 Z35.0

; --- End of Program --- N0013 M5

G28 T0400 M9

G40

; End of Program