



# 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.

1. 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

2. To get to the parameters. Click on Setup F1, Config F3, enter password "137," Enter, Parms F3. Now courser 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	Tool		Х
Orient	Туре	Approach	Offset
OD	Turn	Front	0.0000
OD	Turn	Rear	0.0000
OD	Cutoff	Front	-3.0897
FFace	Groove	Front	0.0000

# FIGURE 2

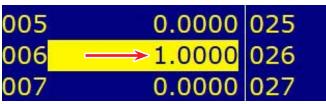
**For Front Orientation P1=1**. On our lathe and chucker lathe, the direction of travel may need to be reversed. In this case add (2) as stated on the parameter page and change this parameter to 3.0000 (see Figure 3).



# 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).





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 crosslide 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)

Axis	Return #1 (G28)	Return #2 (G30)	F				
z	-8.8503	0.0000					
X	1.8311	0.0000					
N	0.0000	0.0000					
N	0.0000	0.0000					
N	0.0000	0.0000					
N	0.0000	0.0000					
N	0.0000	0.0000					
Ν	0.0000	0.0000	0.0000				
/	Teach						
Esc	F2						

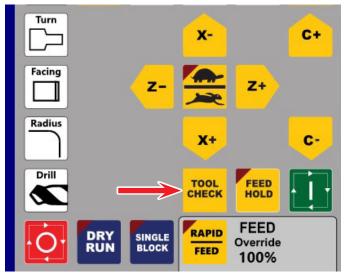
## FIGURE 6

- 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 Figure 7).

X Z		Current Positi	-40.0 -410.0	000	Job Name: NO_JOB_LOADED.cnc Tool: T0909 Feedrate: 100% 0.0 mm/m Part Cnt: 0 Spindle: 0 A Part #↑: 1 407 X+ limit (#50001) tripped 340 X+ limit (#50001) cleared 340 X+ limit (#50001) cleared 340 X+ limit (#26) 340 X+ limit (#26) 340 X+ limit (#26) 360 Job finished 302 Moving
Axis Z	Return #1 (G28)	Return #2 (G30) 0.000	Return #3 (G30 P3) 0.000	Return (G30 F	
x	-40.000	0.000	0.000		000
	0.000	0.000	0.000		000
N	0.000	0.000	0.000		000
N	0.000	0.000	0.000		000
N	0.000	0.000	0.000		000
N	0.000	0.000	0.000		000
N	0.000	0.000	0.000	0.0	000
X Fsc					Sav

FIGURE 7—Return #1 (G28) table.

 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 Gang Tooling Setup for Sherline Lathes with Acorn, Pg. 2 OF 14 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.

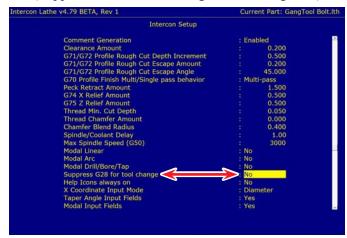
Now you have to decide if you want to use G28 for tool change or not.

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).



#### **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**

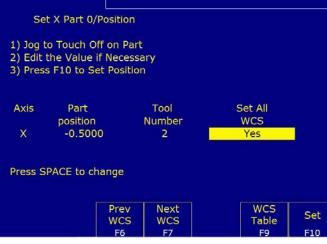
 First go to the Tool Wear and set all of the X and Z tool wear values to (0.0000). To do this click on Setup F1, Tool Offset F2, Tool Wear F9. Double Click on any X or Z value that is not (0.0000) and change it to (0.0000). Click on Save F10 (see Figure 10).

Tool Wear Adjustment											
Tool	Х	Z		(Des	cription)						
T01	0.0000	0.0000	55 I	Degree tu	rning 🗎						
T02	0.0000	0.0000	80	Degree tu	rning						
T03	0.0000	0.0000	.12	5 wide cut	off						
T04	0.0000	0.0000	.06	25 wide O	D Grve						
T05	0.0000	0.0000	OD	Threading	Tool						
T06	0.0000	0.0000	#3	Center Dr	ill						
T07	0.0000	0.0000	Dril	Drill							
T08	0.0000	0.0000	Тар								
T09	0.0000	0.0000	.12	5 wide OD	Grve						
T10	0.0000	0.0000	9/1	6 drill	-						
64 Tu				Entr	y mode:						
×	Clear All	AT	C	Abs Inc	+.001						
ESC	F1	F	3	F4	F5						

- 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).



- 6. Change the "Tool Number" to the reference tool number (in our case Tool #2)
- 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.

WCS #1 (G54) X 🔶	Current Position (Inche -0.50		T0002	_LOADED.cr 0.0 ipm 0 A	iC	
Z	+0.00		te: 100%			
		306 Job f 304 MD1				* 
	1) Jog to 1 2) Edit the	Part 0/Position Fouch Off on Part Value if Necessa 10 to Set Position	ary			
wcs	Axis z #1 (G54)	Part position 0,0000	Tool Number 2			
X Esc		Prev WCS F6	Next WCS F7	Set X F8	WCS Table F9	Set F10

# FIGURE 13

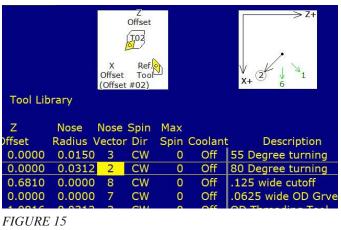
# Setting up Tool Lib (Approach and Nose Vector)

- 1. Define the tools you are using. It is very important to have the approach and nose vector correct.
- 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).

		X Offset (Offse	Z Offset T01 Ref. Tool- t #01)			>Z+ 3 ↑ 4 V X+
Tool Lit	orary					
z	Nose	Nose	Spin	Max		General Salation of
Offset	Radius \	/ector	r Dir	Spin	Coolant	Description
0.0000	0.0150	3	CW	0	Off	55 Degree turning
0.0000	0.0312	2	CW	0	Off	80 Degree turning
0.6810	0.0000	8	CW	0	Off	.125 wide cutoff
0.0000	0.0000	7	CW	0	Off	.0625 wide OD Grve
1 0916	0.0212	2	CW	0	Off	OD Throading Tool

## 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).



## **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).

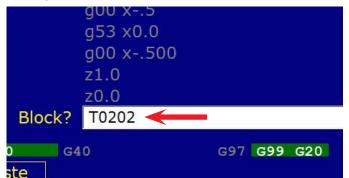


FIGURE 16

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).

X	cs #	1 (G54)	Cu	rrent Pos		5000	Tool: Feedr	ate: lle:	T000	% 0.0 i 0 /
L					>+0.0	0000	Rapid	Rate	: 100	%
						Tool Lib	⊐ rary	X Offset (Offse		3
Off.	Tool	Tool	Tool		x	z	Nose	Nose	Spin	Max
#	Loc	Orient	Туре	Approach	Offset	Offset	Radius '	Vecto	r Dir	Spin C
01	T01	OD	Turn	Front	0.0000	0.0000	0.0150	3	CW	0
02	T02	OD	Turn	Rear	0.470	-0.0000	0.0312	2	CW	0
03	T03	OD	Cutoff	Front	-1.5750	0.681	0.0000	8	CW	0
04	T04	FFace	Groove	Front	0.0000	0.0000	0.0000	7	CW	0
05	T05	OD	Thread	Front	-1.3878	1.0816	0.0312	3	CW	0
06	T06	FFace	Drill	Front	-2.8414	0.0000	0.0000	7	CW	0
	T07	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
	T08	FFace	Tap	Front	-1.2878	1.0816	0.0156		CW	0
09		OD	Groove	Front	0.0000	0.0000	0.0000		CW	0
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
		0.5000 -0.0000			Er	ntry mode:	absolute	2		
X	-	Ref		ATC	Abs Inc	Measure offset X	Measu		G	
Esc		F1		F3	F4	F5	F6			

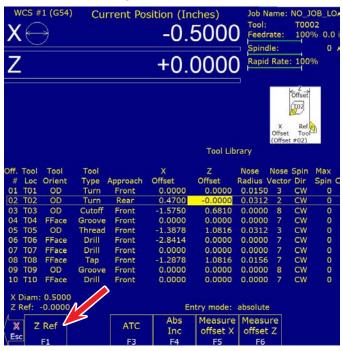
FIGURE 17

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

	*		(C.Tat	et #02)			
C	Offset Z Mea	surement:					
Press F10 t	o measure a	t the currer	nt pos	sition	Max Spin (	Coolan	t Description
	F10 = Mea	sure here			0	Off	55 Degree turning
	The child	Sare nere	_		0	Off	80 Degree turning
-1.5750	0.6810	0.0000	8	CW	0	Off	.125 wide cutoff
0.0000	0.0000	0.0000	7	CW	0	Off	.0625 wide OD Grv
-1.3878	1.0816	0.0312	3	CW	0	Off	OD Threading Tool
-2.8414	0.0000	0.0000	7	CW	0	Off	#3 Center Drill
0.0000	0.0000	0.0000	7	CW	0	Off	Drill
0.0000	0.0000	0.0156	7	CW	0	Off	Тар
0.0000	0.0000	0.0000	8	CW	0	Off	.125 wide OD Grve
0.0000	0.0000	0.0000	7	CW	0	Off	9/16 drill
En	try mode:	absolute					

# 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 courser 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).

WCS #1 (G54)	Current		<sup>ches)</sup> 5000 0000	Spindle:	T00 100	02 % 0.0 i 0 /
Off. Tool Tool # Loc Orient 01 T01 OD 02 T02 OD 03 T03 OD 04 T04 FFace 05 T05 OD 06 T06 FFace 07 T07 FFace	Tool Type Approa Turn Front Turn Rear Cutoff Front Groove Front Thread Front Drill Front Drill Front	ch Of - 1) Loat 2) Jog 3) Pres 1) Type 2) Pres - 1.3875 - 2.8414	OR the reference	e Tool	ir W	Max Spin C 0 0 0 0 0
08 T08 FFace 09 T09 OD 10 T10 FFace	Tap Front Groove Front Drill Front	-1.2878 0.0000	1.0816 0.0000 0.0000	0.0156 7 0.0000 8 0.0000 7	CW CW CW	000
X Diam: 0.5000 Z Ref: -0.0000 X Z Ref Esc F1	ATI F3	C Abs Inc	ntry mode: Measure offset X F5	absolute Measure offset Z F6		

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. Courser 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).

Off.	Tool	Tool	Tool		Х
#	Loc	Orient	Туре	Approach	Offset
01	T01	OD	Turn	Front	0.0000
02	T02	OD	Turn	Rear	0.5000
03	T03	OD	Cutoff	Front	0.0000
04	T04	FFace	Groove	Front	0.0000
05	T05	OD	Thread	Front	-1.4281
06	T06	FFace	Drill	Front	0.0000
07	T07	FFace	Drill	Front	0.0000
08	T08	FFace	Тар	Front	0.0000
09	T09	OD	Groove	Front	0.0000
10	T10	FFace	Drill	Front	0.0000
		0 500/	_		
-		-0.5000	<u>·</u>		
	Ref:	0.0000			Ent
		Diam			Abs
/ Χ	X	Diam			Inc

FIGURE 22

D. Click on "Measure Offset X F5", then "Measure Here F10" (see Figure 23).

		<b>₩11</b> 19		X Offse (Offse	Ref. t Tool et #02)			X+ 2 K 1 1
		Offset X Mea	surement:					
h	Press F10	to measure a	t the currer	nt po	sition		Coolan	
		F10 = Mea	sure here		-	0	Off Off	55 Degree turning   80 Degree turning
	-1.5750	0.6810	0.0000	8	CW	0	Off	.125 wide cutoff
	0.0000	0.0000	0.0000	7	cw	ŏ	Off	.0625 wide OD Grve
	-1.3878	1.0816	0.0312	3	CW	ŏ	Off	OD Threading Tool
	-2.8414	0.0000	0.0000	7	CW	ō	Off	#3 Center Drill
	0.0000	0.0000	0.0000	7	CW	0	Off	Drill
	0.0000	0.0000	0.0156	7	CW	0	Off	Тар
	0.0000	0.0000	0.0000	8	CW	0	Off	.125 wide OD Grve
	0.0000	0.0000	0.0000	7	CW	0	Off	9/16 drill .
	E	ntry mode:	absolute				<b></b>	Measure

- 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. Courser 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).

WCS #1 (G54)	Curi	rent Pos	ition (Ind +0.	<sup>ches)</sup>	Job Na Tool: Feedra Spindl	ite:	TOOO	)5 % 0.0	
Z			+0.(	0000	]	Z Offse	et .	76	
Off. Tool Tool # Loc Orient 01 T01 OD 02 T02 OD 03 T03 OD 04 T04 FFace	Tool Type A Turn Turn Cutoff Groove	pproach Front Rear Front Front	1) Load 2) Jog t 3) Press Of 1) Type	stablish the X the Referenc the Referenc F10 to Save OR the reference F10 to Save .5000	Diameter e Tool ce Position the Refer e location t	ence below	Ref. Toof D5) Din Vir IW IW IW	Max Spin 0 0 0 0	Coe
05 T05 OD	Thread	Front	-1.3878	1.0816	0.0312	3	CW	ő	c
06 T06 FFace	Drill	Front	-2.8414	0.0000	0.0000	7	CW	0	C
07 T07 FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0	c
08 T08 FFace	Тар	Front	-1.2878	1.0816	0.0156	7	CW	0	c
09 T09 OD	Groove	Front	0.0000	0.0000	0.0000	8	CW	0	c
10 T10 FFace X Diam: 0.5000 Z Ref: -0.0000	Drill	Front		0.0000 try mode:		7	CW	0	c
X Diam		ATC	Abs Inc	Measure offset X	Measur offset				
Esc F1		F3	F4	F5	F6				

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

						То
Off.	Tool	Tool	Tool		х	z
#	Loc	Orient	Туре	Approach	Offset	Offse
01	T01	OD	Turn	Front	0.0000	0.0
02	T02	OD	Turn	Rear	0.5000	0.0
03	T03	OD	Cutoff	Front	0.0000	0.0
04	T04	FFace	Groove	Front	0.0000	0.0
05	T05	OD	Thread	Front	-1.4281	1.0
06	T06	FFace	Drill	Front	0.0000	0.0
07	T07	FFace	Drill	Front	0.0000	0.0
08	T08	FFace	Тар	Front	0.0000	0.0
09	T09	OD	Groove	Front	0.0000	0.0
10	T10	FFace	Drill	Front	0.0000	0.0
L		0.5000			E.	ntry m
		0.0000				
	X	Diam			Abs	Mea
		Diam			Inc	offo

# 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).

			X Offse (Offs	Ref. et Tool et #02)	<u>ð</u>		X+ 2 4 1
	Offset X Mea	surement:					
Press F10	to measure a	t the currer	nt no	sition	Max		
-	to measure a	it are surrer		onuon		Coolan	
	F10 = Mea	sure here		-	0	Off Off	55 Degree turning 80 Degree turning
-1.5750	0.6810	0.0000	8	CW	0	Off	.125 wide cutoff
0.0000	0.0000	0.0000	7	CW	ŏ	Off	.0625 wide OD Grve
-1.3878	1.0816	0.0312	3	CW	ō	Off	OD Threading Tool
-2.8414	0.0000	0.0000	7	CW	0	Off	#3 Center Drill
0.0000	0.0000	0.0000	7	CW	0	Off	Drill
0.0000	0.0000	0.0156	7	CW	0	Off	Тар
0.0000	0.0000	0.0000	8	CW	0	Off	.125 wide OD Grve
0.0000	0.0000	0.0000	7	CW	0	Off	9/16 drill
E	ntry mode:	absolute				~	Measure here F10

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

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

		₩111K		X Offse (Offse	Ref. t Tool t #02)			X+ 2 K 1 1
	C	offset Z Mean	surement:					
h	Press F10 t	o measure a F10 = Mea		nt po:	sition	Max Spin ( 0	Coolan Off Off	t Description 55 Degree turning 80 Degree turning
	-1.5750	0.6810	0.0000	8	CW	0	Off	.125 wide cutoff
	0.0000	0.0000	0.0000	7	CW	0	Off	.0625 wide OD Grve
	-1.3878	1.0816	0.0312	3	CW	0	Off	OD Threading Tool
	-2.8414	0.0000	0.0000	7	CW	0	Off	#3 Center Drill
	0.0000	0.0000	0.0000	7	CW	0	Off	Drill
	0.0000	0.0000	0.0156	7	CW	0	Off	Тар
	0.0000	0.0000	0.0000	8	CW	0	Off	.125 wide OD Grve
	0.0000	0.0000	0.0000	7	CW	0	Off	9/16 drill
	En	try mode:	absolute					Measure here F10

- 10. 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.
- 11. 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).

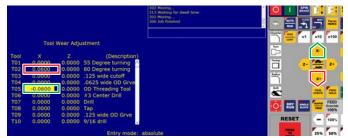


FIGURE 28—The red outlines indicate a positive value and the green outlines indicate a negative value.

# Measure Tools with Tool Touch Off Probe

1. 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: <u>The latest V6 anti-roll 3D Touch</u> <u>Probe edge finder finds the center of the desktop CNC</u> <u>probe compatible with mach3 and grbl (aliexpress.us)</u>

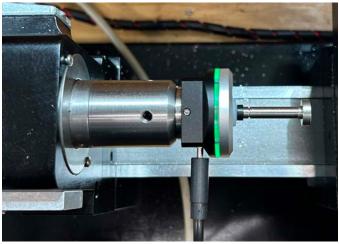
2. The touch probe is held in our 6mm end mill holder (P/N 3076) (see Figure 30).

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

Amazon Link: <u>Flat Contact Points for Dial Digital</u> <u>Indicator Depth Gauge Thickness Gauge 4-48 UNF</u> <u>Thread 15mm Diameter 5mm Length</u> 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)



FIGURE 29



# FIGURE 30

- 3. 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)

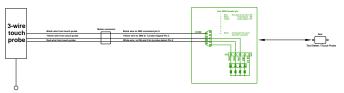


FIGURE 31—The complete schematic file is acorn\_mill\_ connection\_back\_panelconfig\_touch\_probe.pdf.

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)

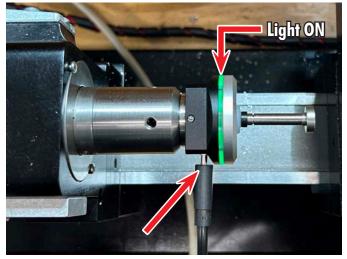


FIGURE 32

- 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).

Primary System	Input Type: Lathe	Acorn In	tegrated Inputs 1-8
Output Definitions	Axis1DriveOk Axis2DriveOk Axis3DriveOk Axis4DriveOk		Definition SecondAxisHomeOk
Axis Configuration Homing and Travel Axes Pairing Advanced	ChucksClosed ChucksClosed ChucksClosed CutoffisDown CutoffisDown	2 IN2 3 IN3 4 IN4 5 IN5	FirstAxisHomeOk
Spindle Spindle #1 Rigid Tapping PWM Setup	CycleStart2 DriveOk FeedHold2 FirstAxisHomeLimitOk FirstAxisMinusLimitOk	6 IN6 7 IN7 8 IN8	ProbeDetect ProbeTripped EStopOk
Touch Devices Probe Tool Touch Off	FirstAxisPlusLimitOk FourthAxisHomeLimitOk FourthAxisHomeOk FourthAxisMinusLimitOk	to the Input nu function to an i	
Control Peripheral Input Devices Wireless MPG	FourthAxisPlusLimitOk HomeAll LimitAll LubeOk PartChuteIsIn	state from NC t	number circle to toggle the input to NO. Note: Probe Input states are the Probe setup menus.
DB25 Connector L Mapping	PressureLowMessage PressureLowStop		

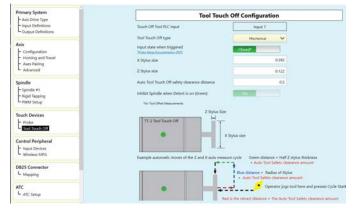
FIGURE 33

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



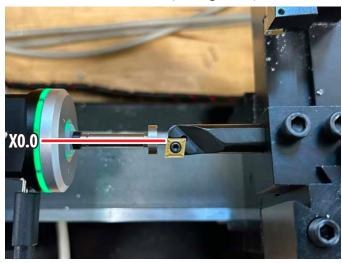
FIGURE 34

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)





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).

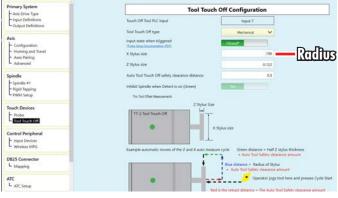
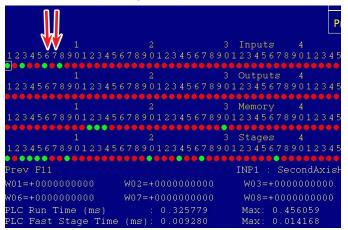


FIGURE 37

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



#### FIGURE 37

- A. With the Input screen up, if the touch probe is connected and there is power to the probe, Number 6 should be Green.
- B. When you activate the touch probe by pushing on the tip, Number 7 should change from Red to Green.
- C. 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).

WCS #1 (G54)	Current Position (Inches) -0.5000	Job Name: NO_JOB_LOADED.cnc Tool: T0002 Feedrate: 100% 0.0 ipm Soindle: 0 A
Z	+0.0000	
		313 Waiting for dwell time (M26)

# FIGURE 38

- 7. Now we are ready to touch off the tools.
  - A. 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).

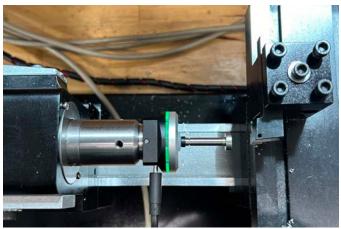


FIGURE 39

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

						Tool Lib	rary	X Offset (Offset	T02 T02 Ref. Tool #02)	<b>)</b>
Off.	Tool	Tool	Tool		x	z	Nose	Nose	Spin	Max
#	Loc	Orient	Туре	Approach	Offset	Offset	Radius 1			Spin C
01	T01	OD	Turn	Front	0.0000	0.0000	0.0150	3	CW	0
02	T02	OD	Turn	Rear	0.3310	-0.1130	0.0312	2	CW	0
03	T03	OD	Cutoff	Front	-1.5750		0.0000	8	CW	0
04	T04	FFace	Groove	Front	0.0000	0.0000	0.0000	7	CW	0
05	T05	OD	Thread	Front	-1.3878	1.0816	0.0312	3	CW	0
06	T06	FFace	Bore	Front	-3.0789	1.2598	0.0000	7	CW	0
07	T07	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
08	T08	FFace	Tap	Front	-1.2878	1.0816	0.0156	7	CW	0
09	T09	OD	Groove	Front	0.0000	0.0000	0.0000	8	CW	0
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
	Ref:	-0.0684 -0.1800 Ref F1			E Abs Inc F4	ntry mode: Measure offset X F5		ire M	leasu and	

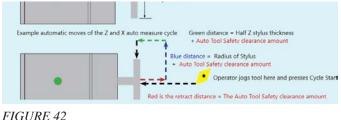
# FIGURE 40

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).

								* X Offset	Offset	5
Off. T	ool	Tool	Tool		Z and X A Press CYCLE	uto Tool Off START to st				1ax
	Loc	Orient		Approad		X Directio	n X+			pin
	01	OD	Turn	Front		A Directic				0
	02	OD	Turn	Rear		F1 = X O	rient			0
03 T	03	OD	Cutoff	Front				~		0
04 T	04	FFace	Groove	Front	0.0000	0.0000	0.0000	7	CW	0
05 T	05	OD	Thread	Front	-1.3878	1.0816	0.0312	3	CW	0
06 T	06	FFace	Bore	Front	-3.0789	1.2598	0.0000	7	CW	0
07 T	07	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
08 T	08	FFace	Тар	Front	-1.2878	1.0816	0.0156	7	CW	0
09 T	09	OD	Groove	Front	0.0000	0.0000	0.0000	8	CW	0
10 T	10	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
Z Re	ef: Cha	-0.0684 -0.1800 nge X ection F1	<b>Ç</b>		En	try mode:	absolute			

## 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).



- 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).

						0.940 - 19930		Offset (Offset
						Tool Lib	rary	
Off.	Tool	Tool	Tool		х	Save cha	nges?	Nose
#	Loc	Orient	Туре	Approach	Offset	F1 = Yes	F2 = No	/ector
01	T01	OD	Turn	Front	0.0000	0.0000	0.0150	3
02	T02	OD	Turn	Rear	0.7952	0.0004	0.0312	2
03	T03	OD	Cutoff	Front	-1.5750	0.6810	0.0000	8
04	T04	FFace	Groove	Front	0.0000	0.0000	0.0000	7
05	T05	OD	Thread	Front	-1.3878	1.0816	0.0312	3
06	T06	FFace	Bore	Front	-3.0789	1.2598	0.0000	7
07	T07	FFace	Drill	Front	0.0000	0.0000	0.0000	7
08	T08	FFace	Тар	Front	-1.2878	1.0816	0.0156	7
09	T09	OD	Groove	Front	0.0000	0.0000	0.0000	8
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000	7
10 M		-0.068 -0.1800			e de la dela de la dela E	ntry mode:	absolute	
		Yes	No					
Es		F1	F2					

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

						Tool Lit	orary
Off.	Tool	Tool	Tool		Х	Z	No
#	Loc	Orient	Туре	Approach	Offset	Offset	Rad
01	T01	OD	Turn	Front	0.0000	0 0000	0.
02	T02	OD	Turn	Rear	0.7952	0.0004	0.
03	T03	OD	Cutoff	Front	-1.5750	0.6810	0.
04	T04	FFace	Groove	Front	0.0000	0.0000	0.
05	T05	OD	Thread	Front	-1.3878	1.0816	0.
06	T06	FFace	Bore	Front	-3.0789	1.2598	0.
07	T07	FFace	Drill	Front	0.0000	0.0000	0.
08	T08	FFace	Тар	Front	-1.2878	1.0816	0.
09	T09	OD	Groove	Front	0.0000	0.0000	0.
10	T10	FFace	Drill	Front	0.0000	0.0000	0.
		-0.0684 -0.1800			Er	ntry mode:	abs
Esc	Z	Ref	1easure Tool		Abs Inc	+.001	
V LOC		F1	F2		F4	F5	

FIGURE 44

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

						2014	(0	X ffset	Ref. Tool #02)	
						Tool Li		i i		
~~~		-	-			Z Refe	rence			
	Tool	Tool	Tool	Annach	X				Spin	1
# 01	Loc T01	Orient OD	Type Turn	Approach Front	Offset 0.000	F1 = Auto	F10 = Manual		Dir	-
02	T02	OD	Turn	Rear	0.7952			5	CW	
03	T03	OD	Cutoff	Front	-1.5750	0.6810		8	CW	-
04	T04	FFace	Groove		0.0000	0.0000		7	CW	
05	T05	OD	Thread		-1.3878			3	CW	
06	T06	FFace	Bore	Front	-3.0789	1.2598	0.0000	7	CW	
07	T07	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	
08	T08	FFace	Тар	Front	-1.2878	1.0816	0.0156	7	CW	
09	T09	OD	Groove	Front	0.0000	0.0000	0.0000	8	CW	
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	
	Aut	-0.0684 -0.1800 o Tool ch Off F1			ł	Entry mode:	absolute			

- 9. Now we are going to set the Part Zero in Z and X-axis.
  - A. Mount your part in the chuck or collet with it extended away from the collet by the correct distance.
  - B. Again we will be using our Reference Tool to set the part zero (tool #2 for these instructions).
  - C. Jog the tool up to the front of the part until it touches.
  - D. Click on Setup F1, Part F1.
  - E. 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).

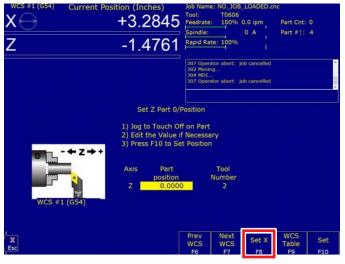


FIGURE 47

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).

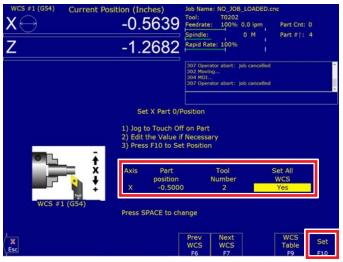


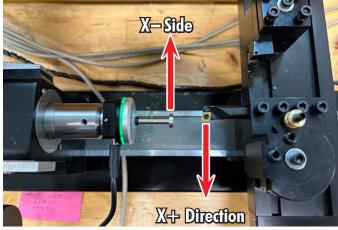
FIGURE 48

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).



Off.	Tool	Tool	Tool		x	Z	Nose N
#	Loc	Orient	Туре	Approach	Offset	Offset	Radius Ve
01	T01	OD	Turn	Front	0.0000	0.0000	0.0150
02	T02	OD	Turn	Rear	0.4034	-0.0011	0.0312
03	T03	OD	Cutoff	Front	-1.5750	0.6810	0.0000
04	T04	FFace	Groove	Front	0.0000	0.0000	0.0000
05	T05	OD	Thread	Front	-1.3878	1.0816	0.0312
06	T06	FFace	Bore	Front	-2.9783	1.5664	0.0000
07	T07	FFace	Drill	Front	0.0000	0.0000	0.0000
08	T08	FFace	Тар	Front	-1.2878	1.0816	0.0156
09	T09	OD	Groove	Front	0.0000	0.0000	0.0000
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000
		-0.004	5				
ZI	Ref:	-0.0780			Er	ntry mode:	absolute
			Measure		Abs	Export	
×			Tool		Inc	Lib	
Ese	C		F2		F4	F5	

FIGURE 50—Boring tool front.

					Z and X A Press CYCLE	uto Tool Off START to st				
off. #	Tool Loc	Tool Orient	Tool Type	Approach		X Directi	on X-			Max 5pin
01 02	T01 T02	OD OD	Turn Turn	Front Rear		F1 = X 0	rient			0
02	T02	OD	Cutoff	Front	-1.5750	0.6810	0.0000	8	CW	0
04	T04	FFace	Groove		0.0000	0.0000	0.0000	7	CW	ŏ
05	T05	OD	Thread		-1.3878	1.0816	0.0312	3	CW	Ō
06	T06	FFace	Bore	Front	-2.9783	1.5664	0.0000	7	CW	0
07	T07	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
08	T08	FFace	Тар	Front	-1.2878	1.0816	0.0156	7	CW	0
09	T09	OD	Groove	Front	0.0000	0.0000	0.0000	8	CW	0
10	T10	FFace	Drill	Front	0.0000	0.0000	0.0000	7	CW	0
08 09 10 X [	T08 T09 T10 Diam: Ref: Cha	FFace OD	Tap Groove Drill	Front Front	-1.2878 0.0000 0.0000	1.0816 0.0000	0.0156 0.0000 0.0000	7 8	CW CW	

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).

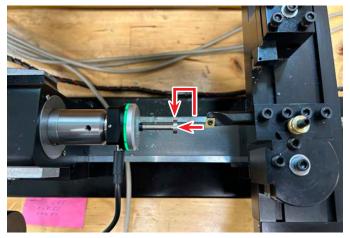


FIGURE 52

## 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).

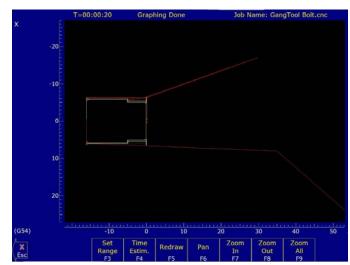


FIGURE 54

# 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).

WCS #1 (G54) Curr	rent Position (mm) +74.34	Job Name: GangTool Bolt.cnc Tool: T0400 Feedrate: 100% 0.0 mm/m Part Cnt: 0
Z	+74.34 +49.33	Spindle: 0 A Part #1: 1
		302 X+ limt (#5001) cleared     302 Moving     313 Waiting for dwall time (M26)     365 Job Finished     302 Moving     366 Job Finished
		Press CYCLE START to start job
	5.9	un
Single Block Mode: Optional Stops:	On	Run-Time Graphics: On
Block Skips:	Off	
Job Repeat:	Off	
Part Count: 0		
X Search Esc F2	Repeat/SkipsBlocOnOnF3F3F4F5	K Stops Graph Rapid RTG On/Off On/Off F6 F8 F9 F10

- 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
```