

PROTOTRAK DPM2 QUICK START GUIDE

TABLE OF CONTENTS

1. USING THE WCS (World Coordinate System) WITH FUSION 360
2. POSTING TOOLPATHS FROM FUSION 360
3. EDGE FINDING WITH THE PROTOTRAK
4. ATTACHING STOCK MATERIAL TO FIXTURE
5. SETTING THE Z AXIS
6. BASIC OPERATION
7. TIPS, TRICKS, AND BEST PRACTICES
8. TOOL LIBRARY
9. WCS LOCATION CHEAT-SHEET

USING THE WCS (World Coordinate System) WITH FUSION 360

WCS Offset: This is where the Machine Offset number will be entered. Values of 0 and 1 refer to G54, a value of 2 refers to G55, a value of 3 refers to G56, and so on. For example, if you are creating toolpaths for a machining operation utilizing G55 as its Machine Offset, enter a “2” in the WCS offset box for the posted g-code to reference G55 instead of G54 (the default) (See **WCS LOCATION CHEAT SHEET**).

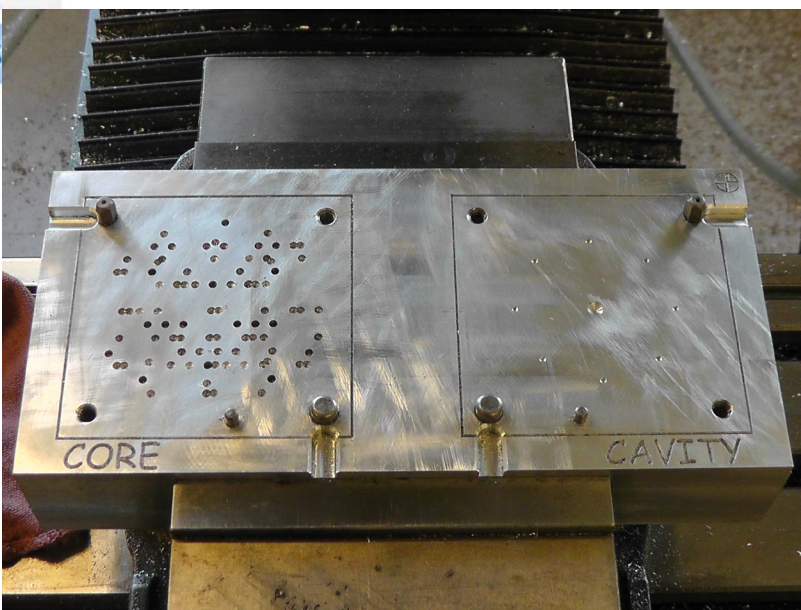
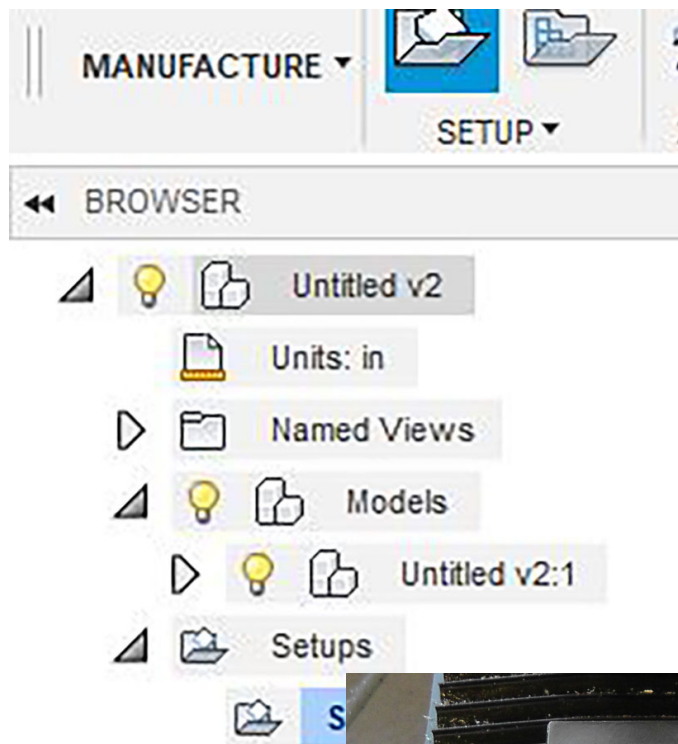
In 2.008 terms, what this boils down to is that the WCS that you set in the CAM part of Fusion 360 tells the Prototrak where to do its work. The three most important things to remember is:

- **WCS 0 or 1 = G54 = X, Y, Z HOME.** The upper-right corner of the fixture (You will reference spot this while performing the edge-finding process). This is the HOME X, Y, Z Zero position that the Prototrak uses for referencing the CORE and CAVITY locations.
- **WCS 2 = G55 = CORE.** Assigning your Stock Setup a WCS 2 in Fusion 360 will tell the Prototrak to mill a part on the fixture on the CORE side
- **WCS 3 = G56 = CAVITY.** Assigning your Stock Setup a WCS 3 in Fusion 360 will tell the Prototrak to mill a part on the fixture on the CAVITY side

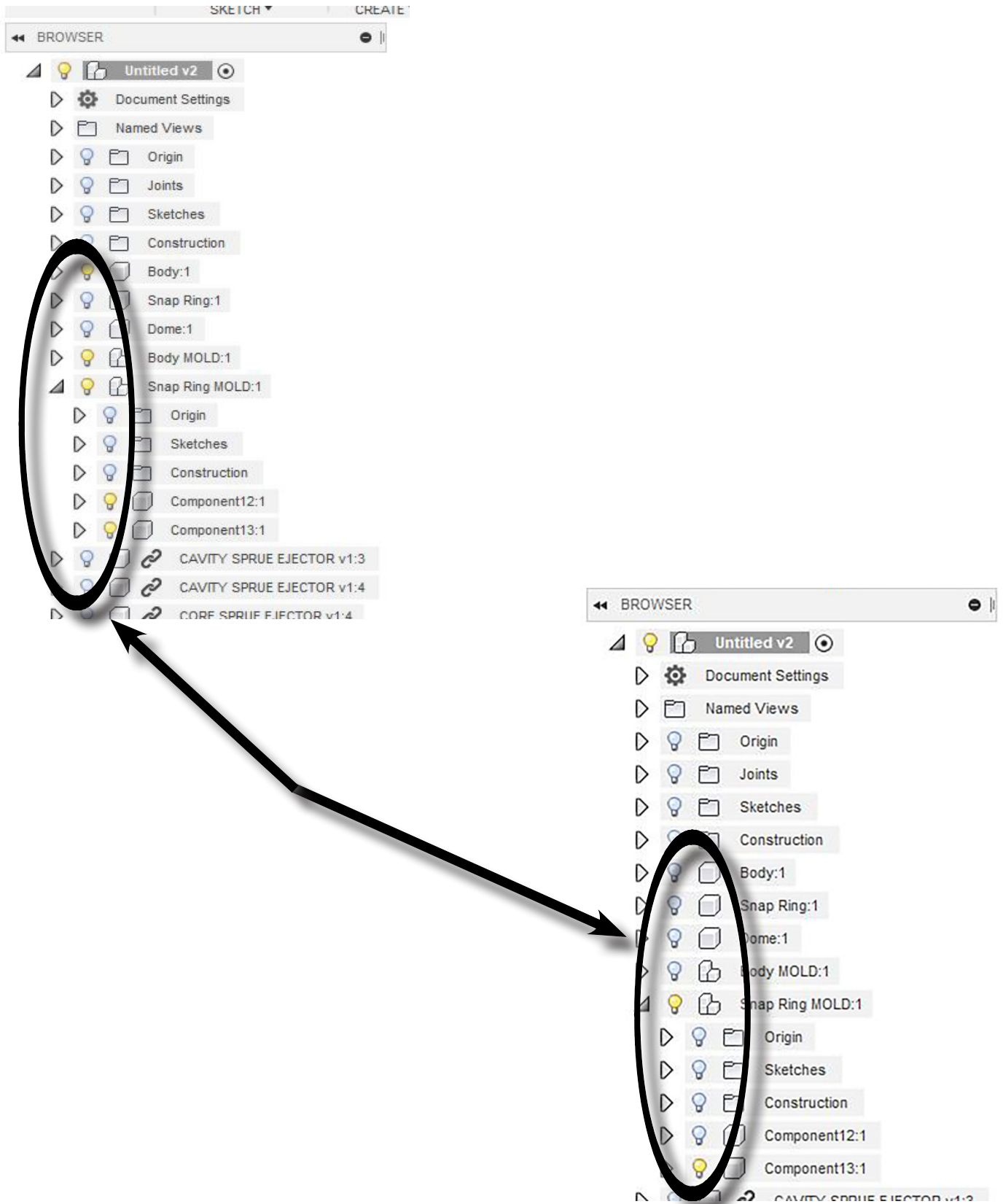
It's very important to assign the correct WCS number to the right part (CORE/CAVITY) If you do not assign the correct WCS number, the Prototrak will attempt to mill in the incorrect location, which could result in the machine crashing and/or breaking a tool.

PLEASE NOTE: *This section only covers setting up the WCS Fusion. It does not cover toolpath creation, edge finding, or other setup information.*

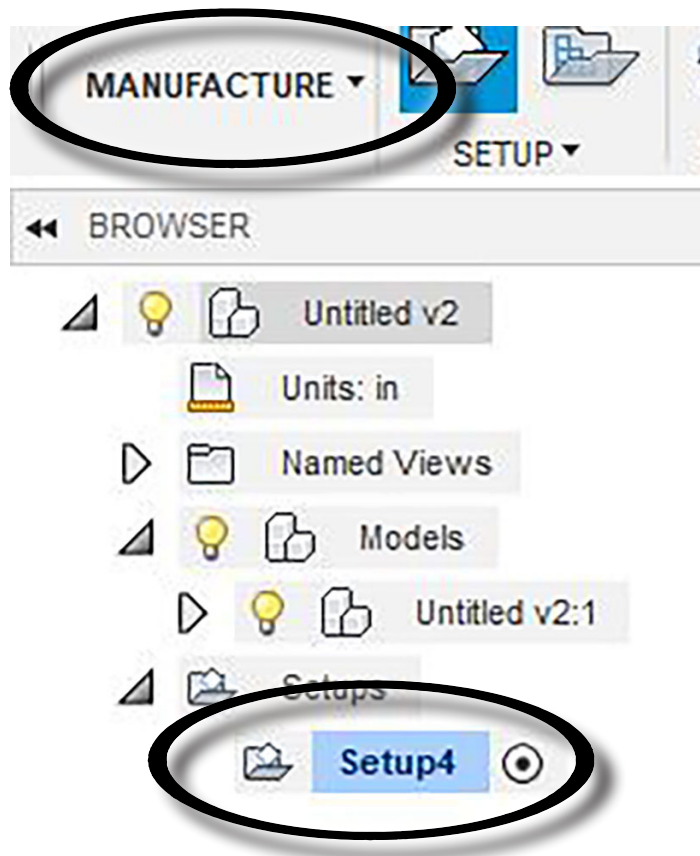
How to set the WCS In Fusion 360 If you are starting a new set of toolpaths



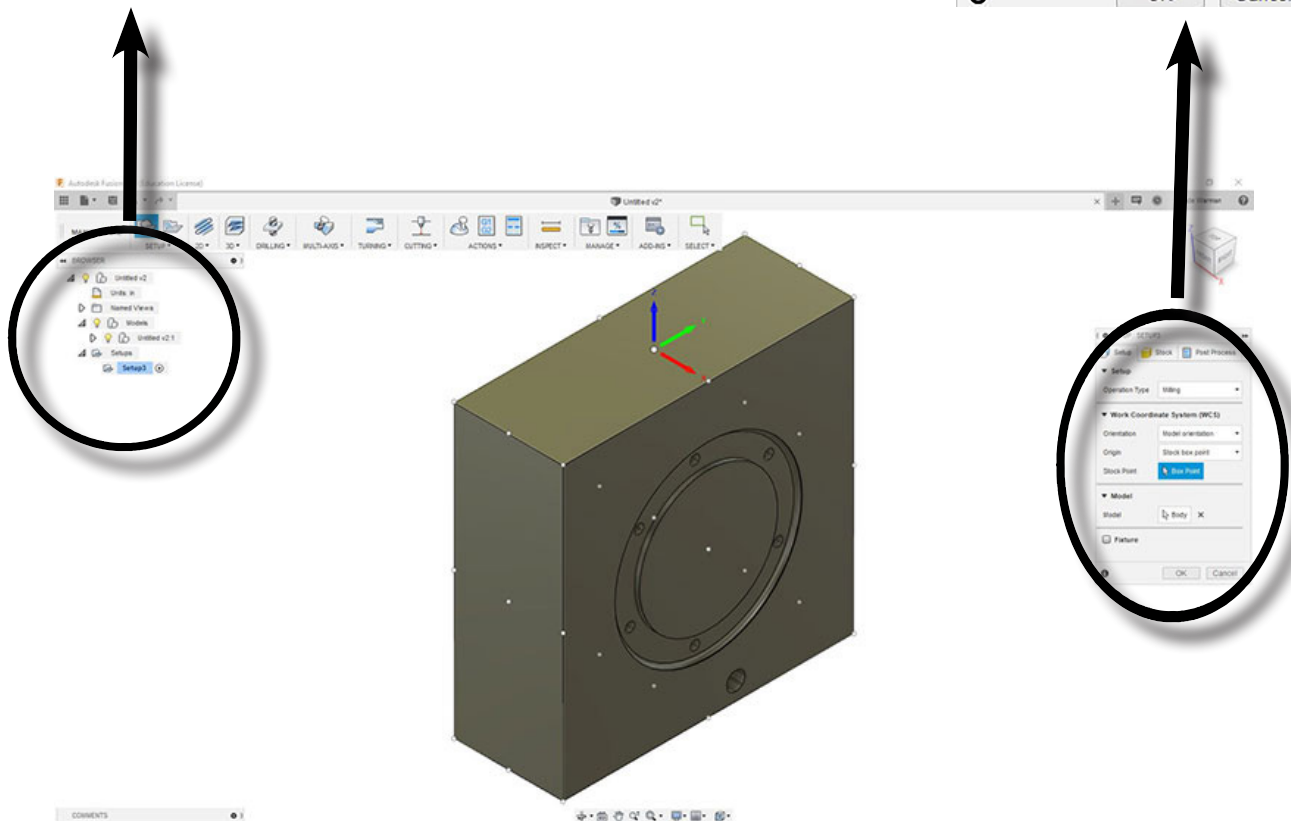
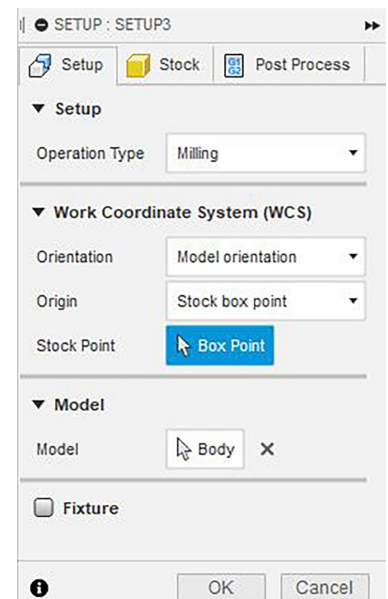
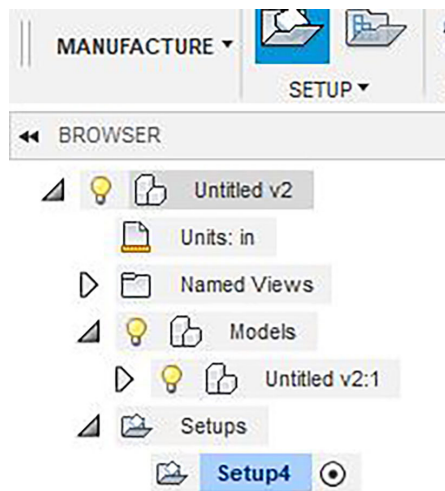
1. In the MODEL Workspace mode, hide all other parts except for the part you want to use for the CAM Setup



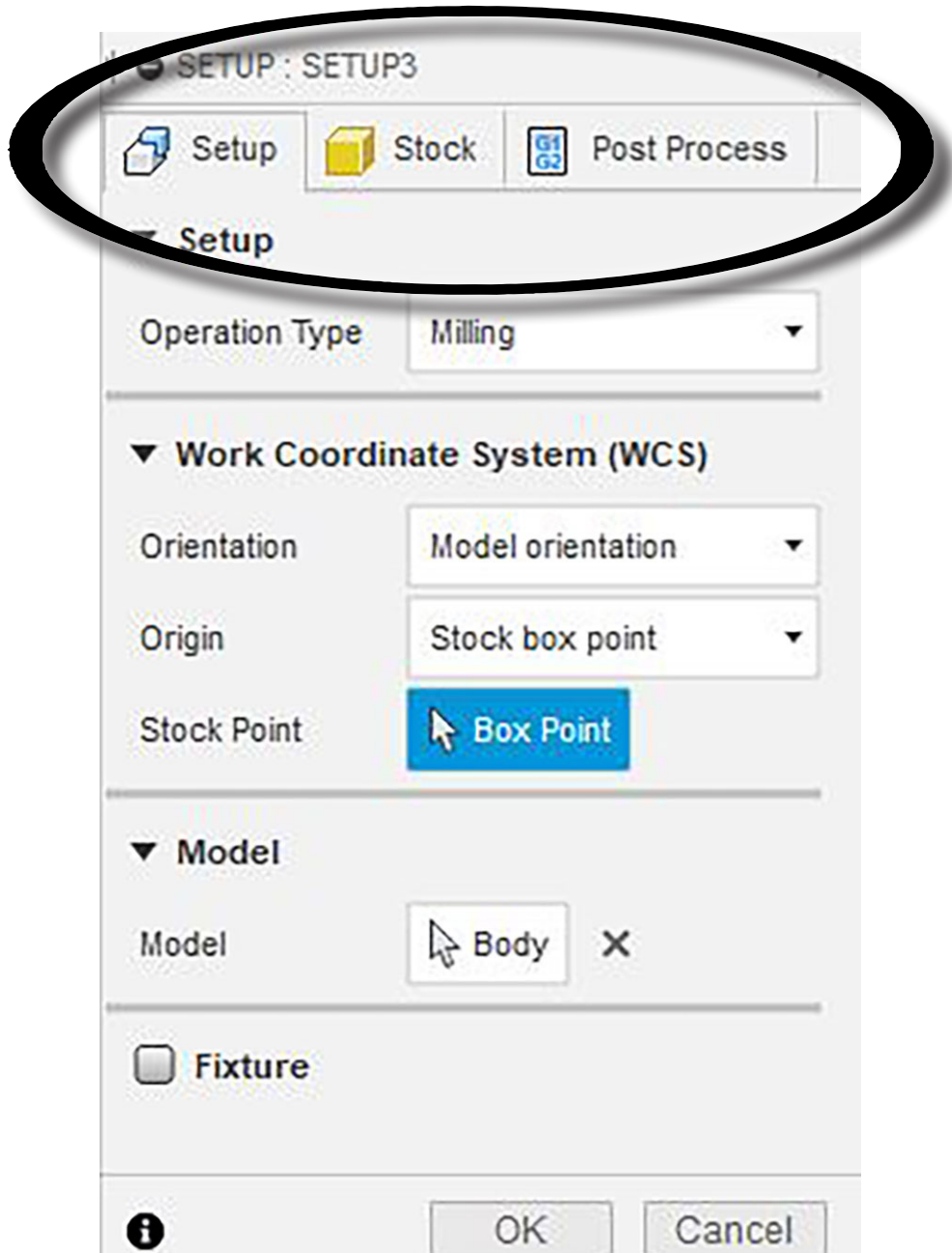
2. Go to MANUFACTURE workspace mode. (You will see a new empty folder at the bottom of your Browser Tree named SETUPS)



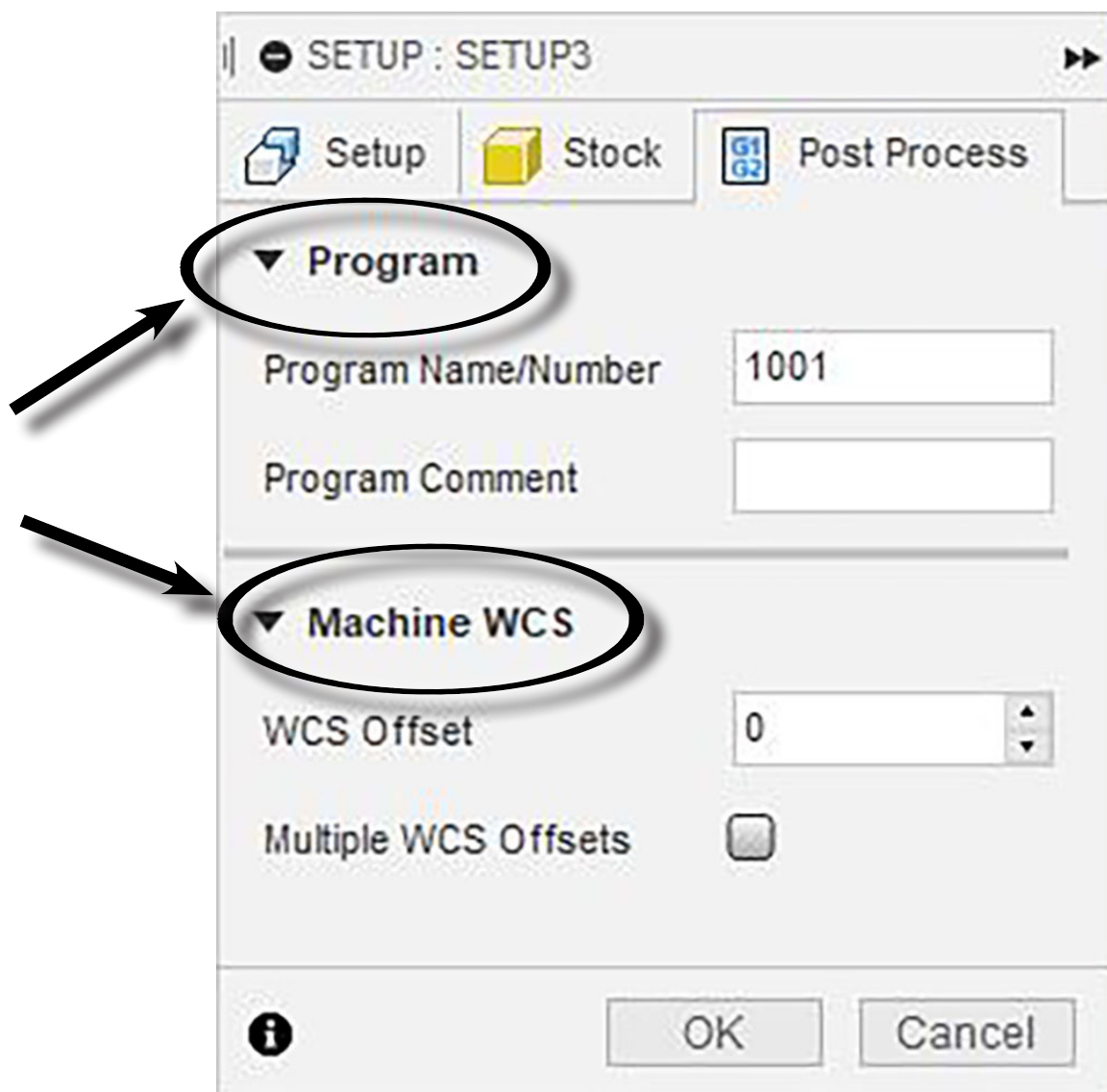
3. Click the SETUP drop-down and select "New Setup".
 - a. A pop-up window will appear titled SETUP (Usually in the right side of your screen) and the part you will be using will now have a series of dots as well as a new X, Y, Z, Origin.
 - b. Additionally, in your Browser Tree you will now have a sub-folder under SETUPS. (If this is the first setup, it will be named Setup1 by default)



4. This pop-up window has 3 tabs
 - a. Setup
 - b. Stock
 - c. Postprocess



5. Click the POST PROCESS tab. In the Post Process tab, you will see PROGRAM and MACHINE WCS



7. In the WCS Field, enter the appropriate number and click OK to Exit

|| ● SETUP : SETUP3 ▶▶

Setup | Stock | **G1 G2** Post Process

▼ Program

Program Name/Number

Program Comment

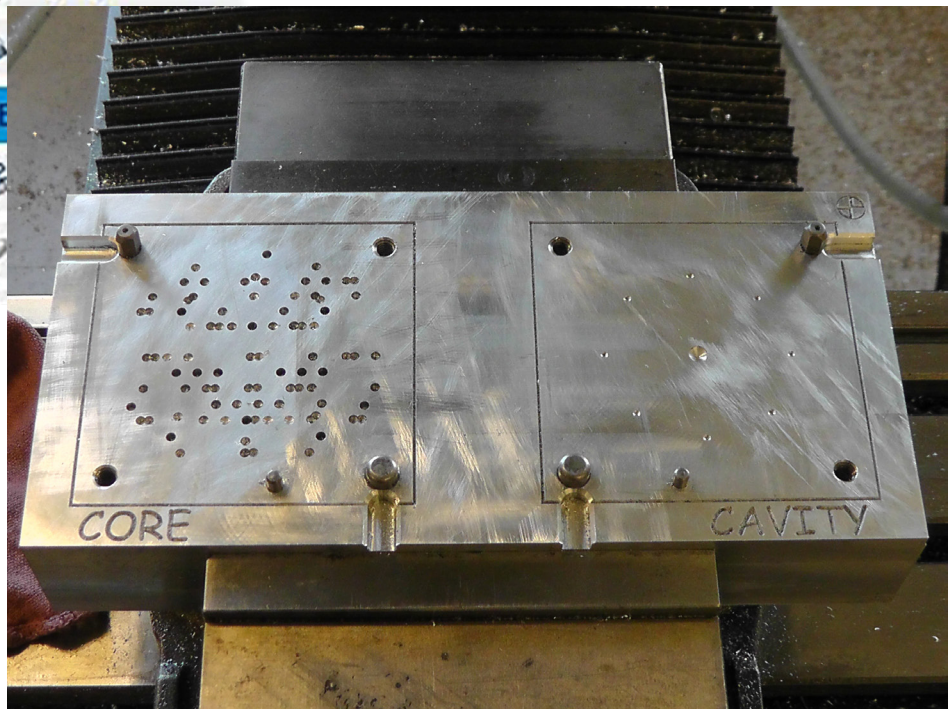
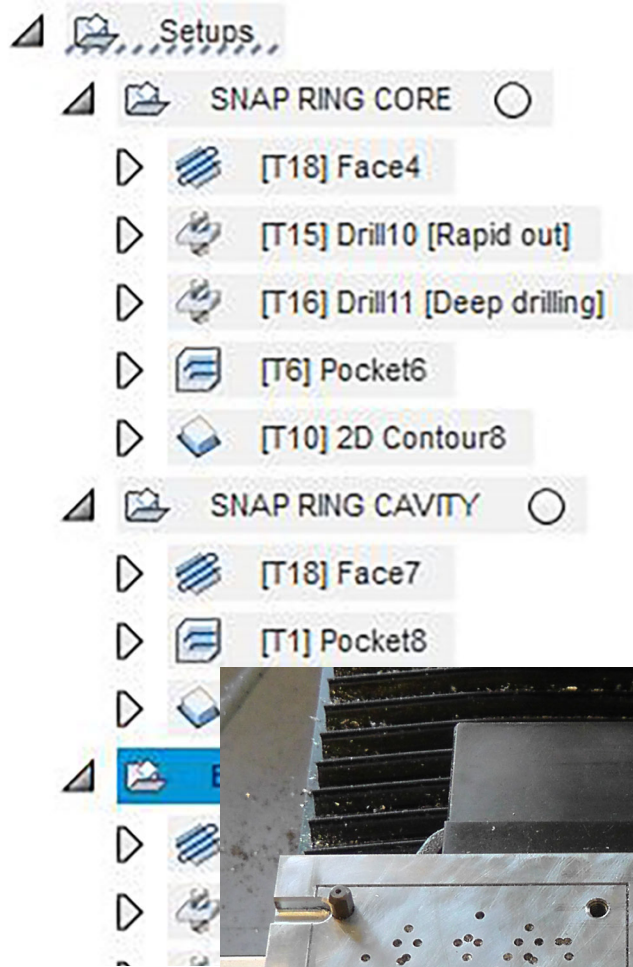
▼ Machine WCS

WCS Offset

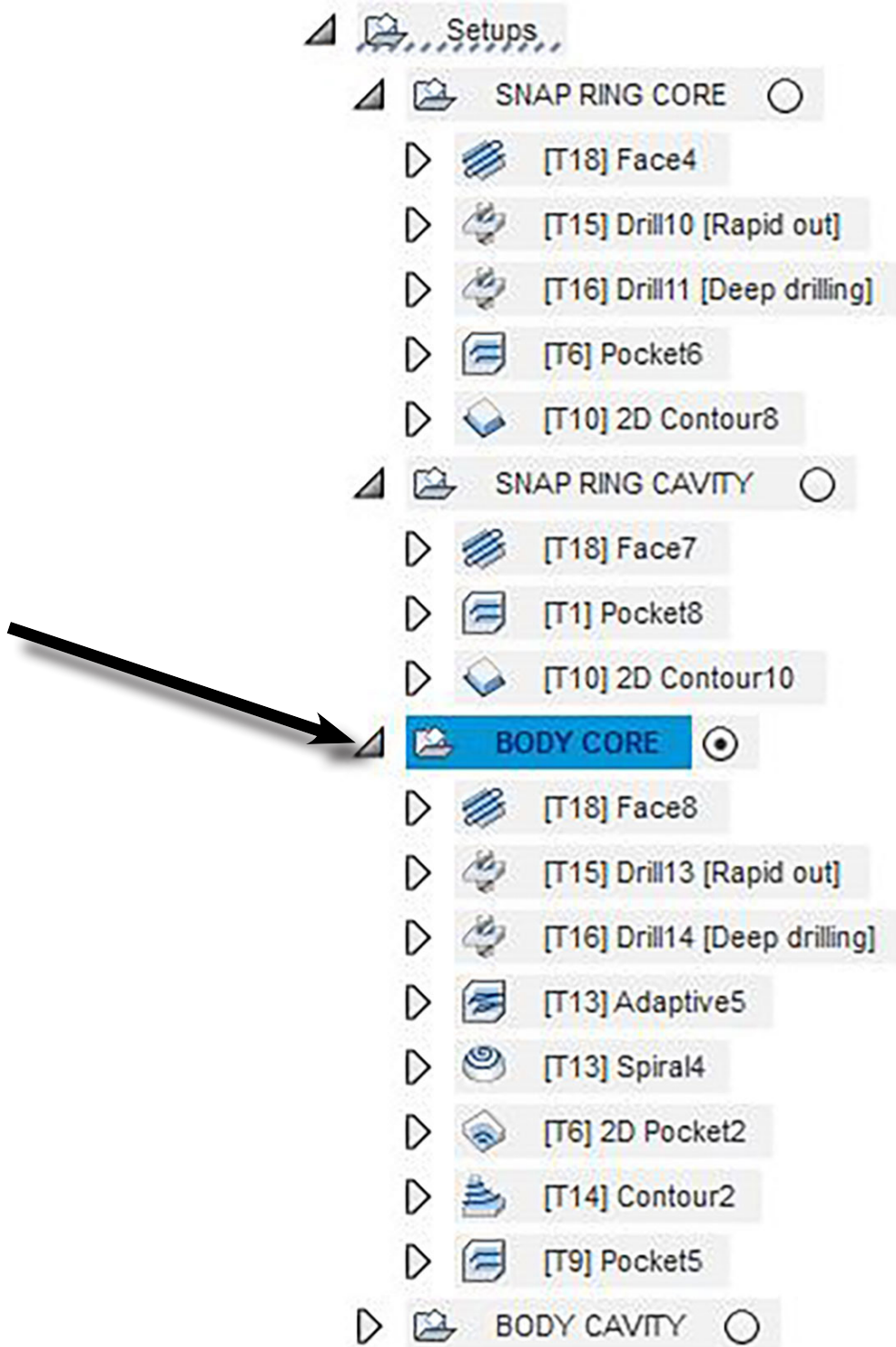
Multiple WCS Offsets

i

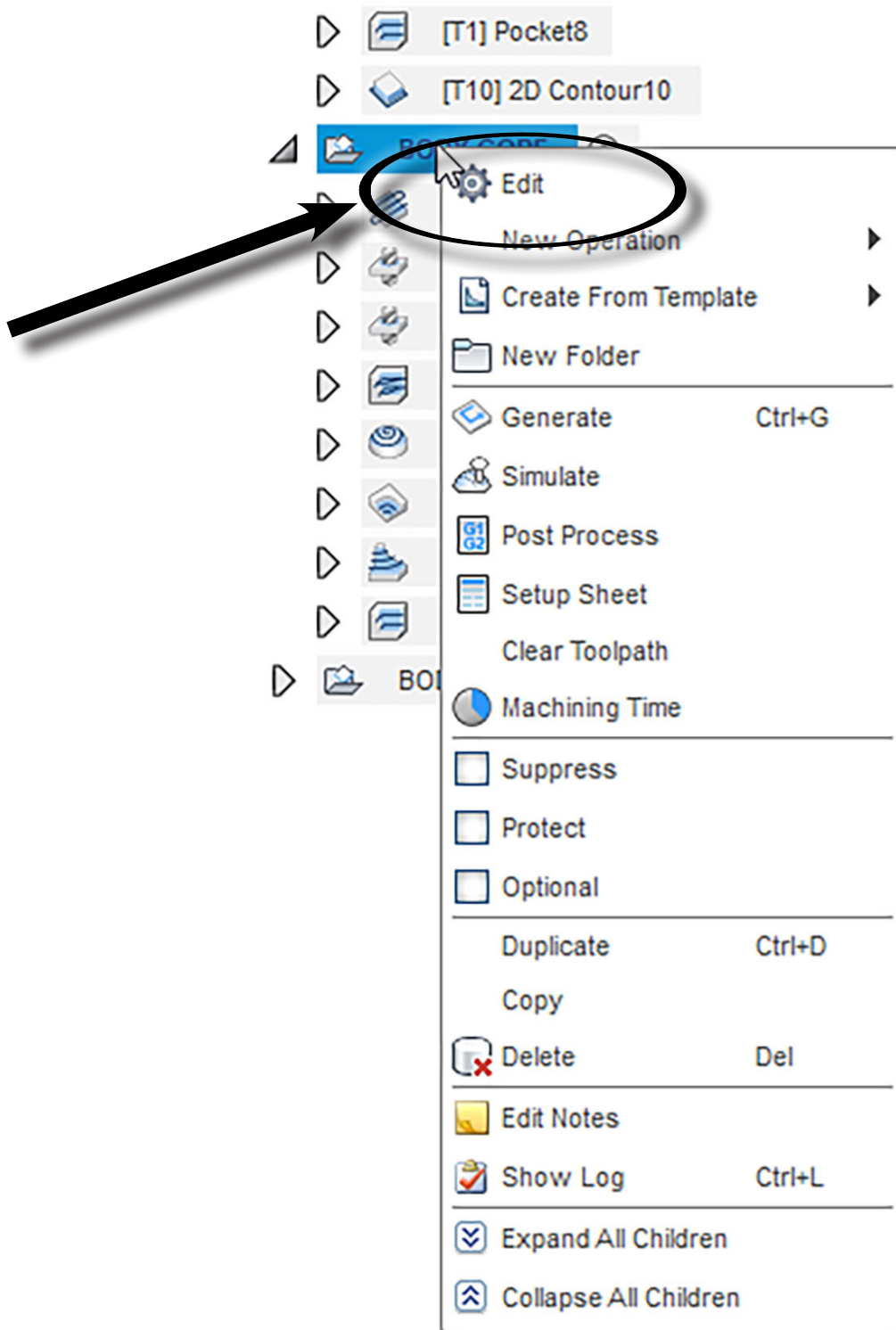
How to set the WCS IN Fusion 360 if you have already created a Setup and Toolpaths



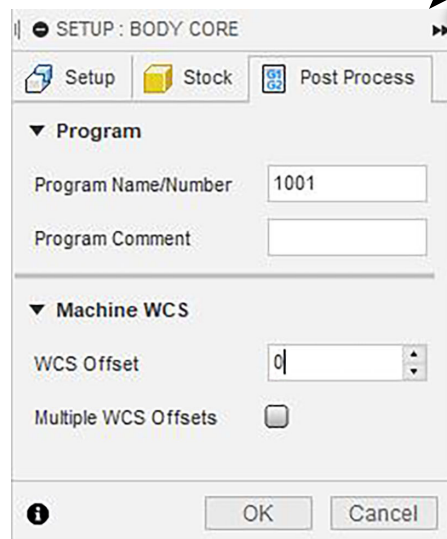
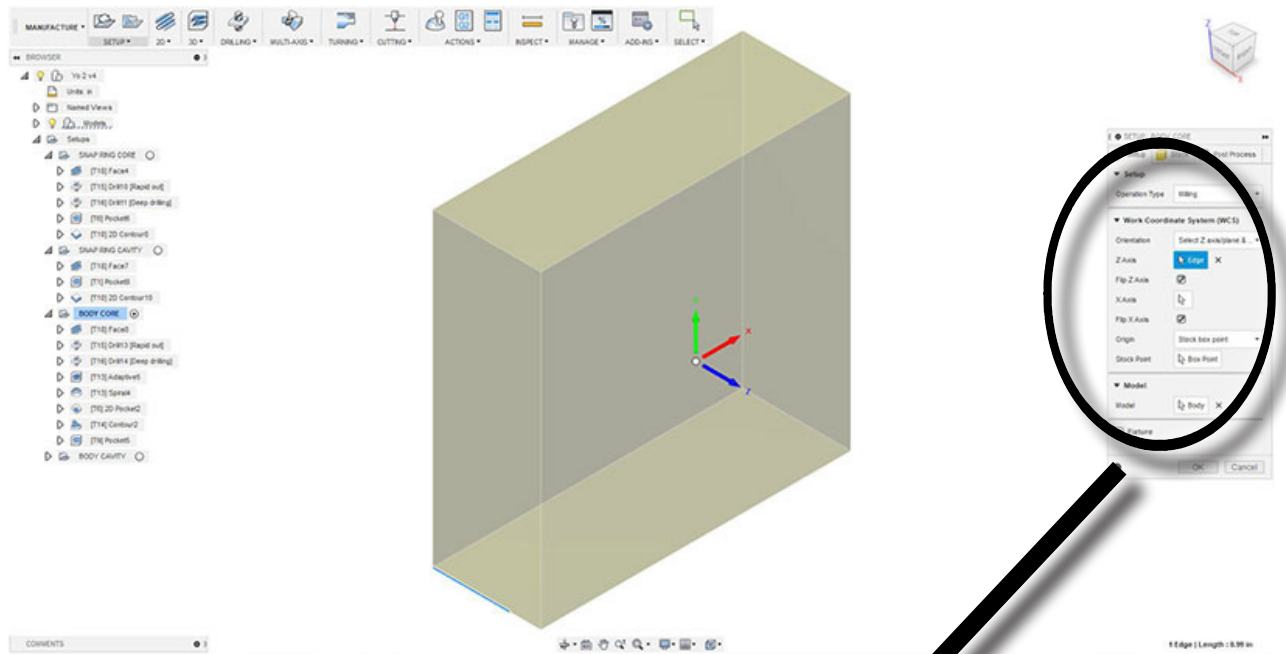
1. Click to highlight the specific setup you want to use



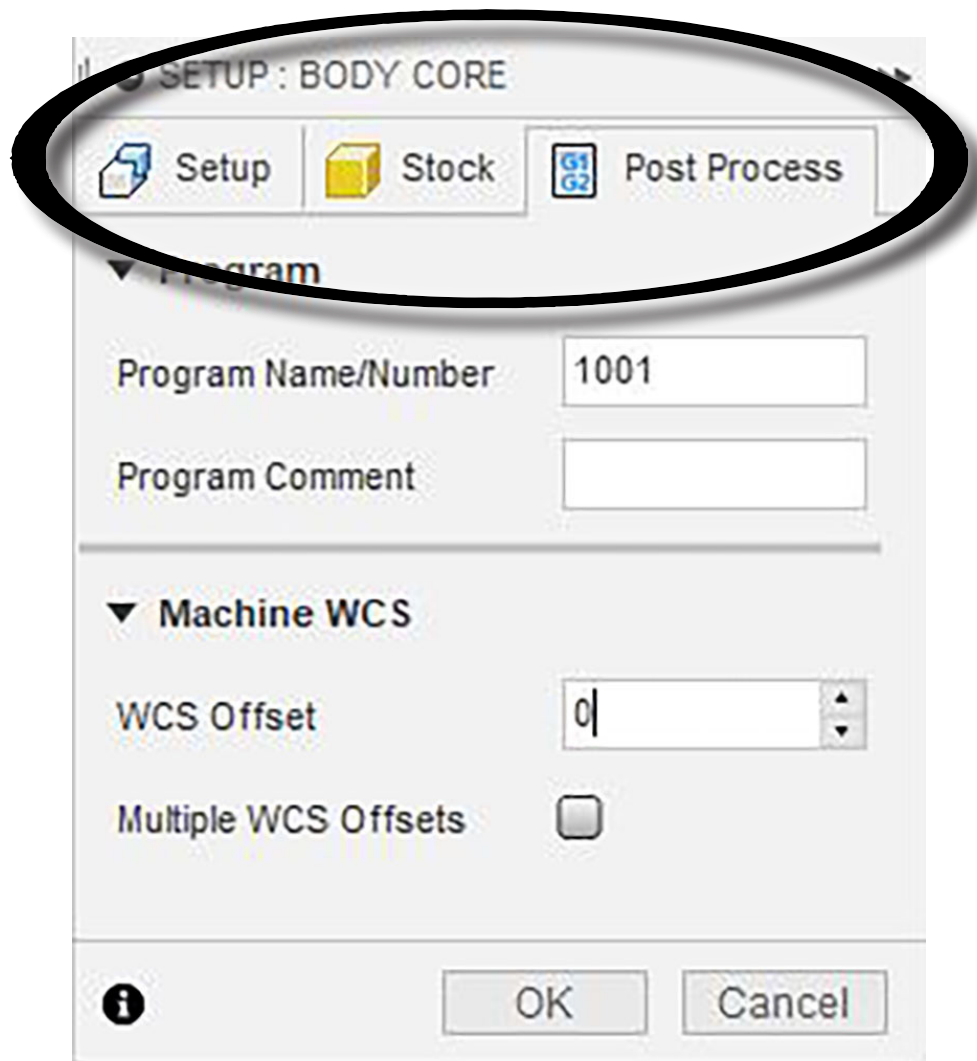
2. Right-click that setup and select EDIT (top selection in the pop-up)



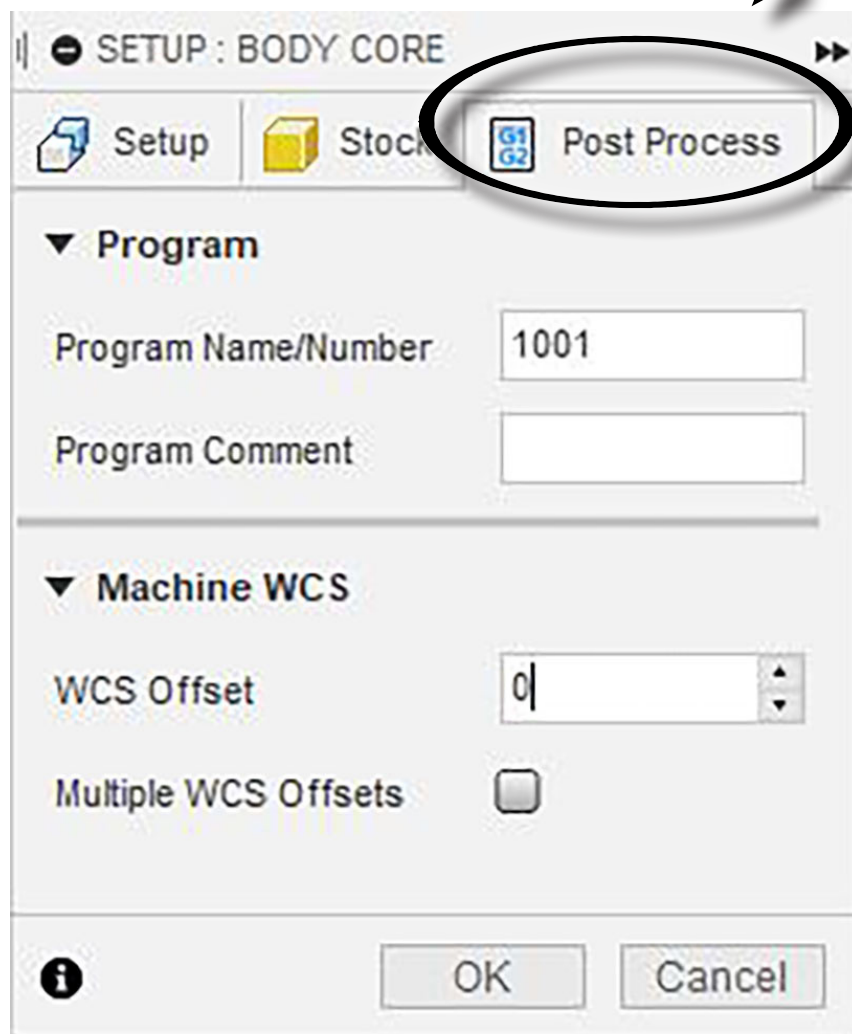
3. A new pop-up will appear (usually on the right side of your screen)



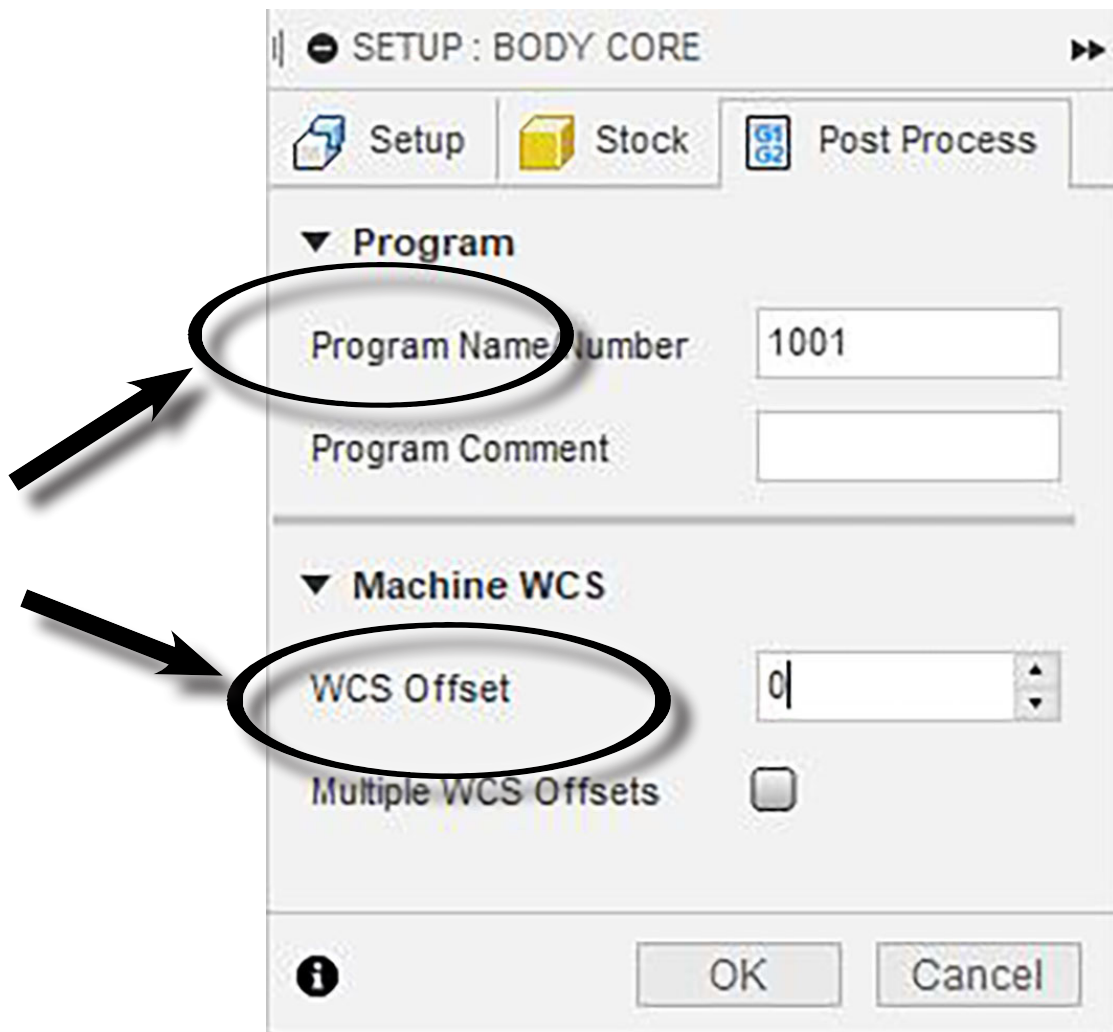
4. This pop-up window has 3 tabs
 - a. Setup
 - b. Stock
 - c. Postprocess



5. Click the POST PROCESS tab



6. In the Post Process tab, you will see PROGRAM and MACHINE WCS



7. In the WCS Field, enter the appropriate number and click OK to Exit

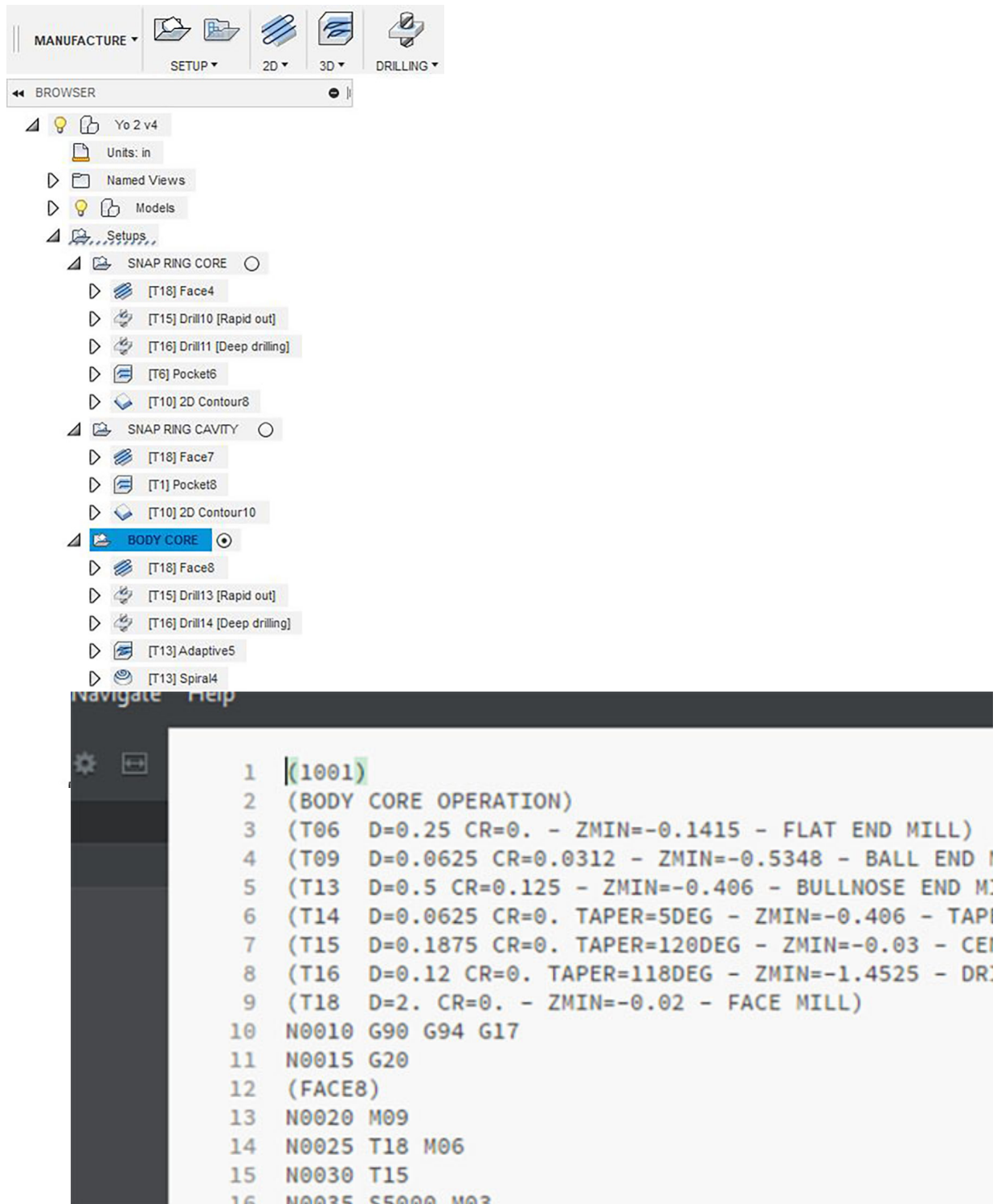
The image shows a software dialog box titled "SETUP : BODY CORE". At the top, there are three tabs: "Setup" (with a blue cube icon), "Stock" (with a yellow cube icon), and "Post Process" (with a blue cube icon containing "G1" and "G2"). Below the tabs, there are two main sections. The first section is "Program", which contains a "Program Name/Number" field with the value "1001" and a "Program Comment" field. The second section is "Machine WCS", which contains a "WCS Offset" field with the value "2" (circled in black with an arrow pointing to it from the right) and a "Multiple WCS Offsets" checkbox which is unchecked. At the bottom of the dialog, there is an information icon (i) on the left and "OK" and "Cancel" buttons on the right.

POSTING TOOLPATHS FROM FUSION 360

In Fusion 360, you can “post” (save the Toolpath G Code to use with the Prototrak) toolpaths either as a group, (i.e. the complete set of toolpaths for a CORE or a CAVITY), or just a single toolpath (i.e. Face8, Drill 13, or Drill14, etc.). This section describes how to “post” an entire toolpath operation as a group, as well as how to “post” a single toolpath by itself.

NOTE: To load your G code onto the Prototrak, see *BASIC PROTOTRAK OPERATION* section

TO POST THE ENTIRE TOOLPATH OPERATION



The image shows a CAD software interface with a manufacturing setup tree on the left and a post-processor output window on the right.

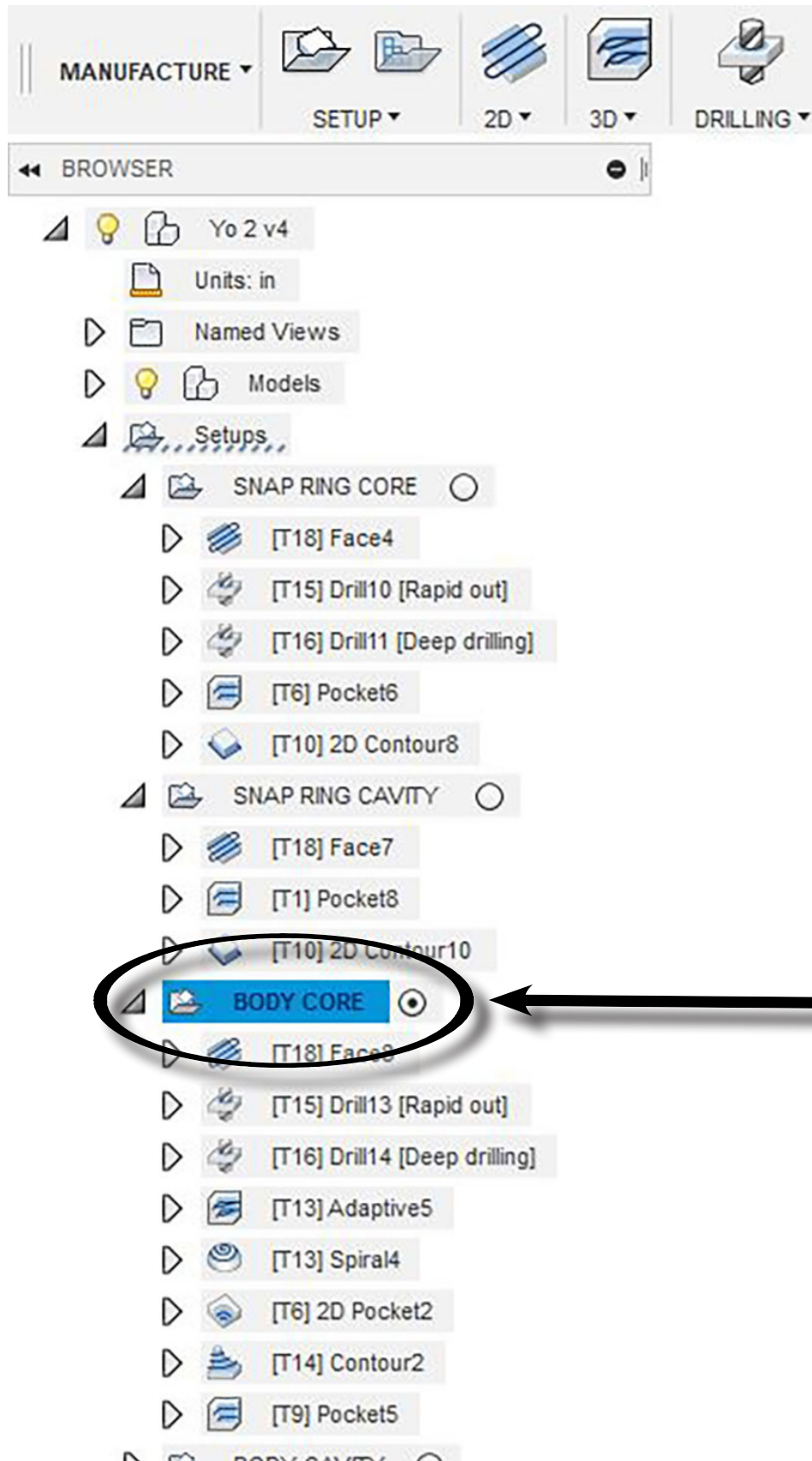
Manufacturing Setup Tree:

- MANUFACTURE ▾
 - SETUP ▾
 - 2D ▾
 - 3D ▾
 - DRILLING ▾
- BROWSER
- Yo 2 v4
 - Units: in
 - Named Views
 - Models
 - Setups
 - SNAP RING CORE ○
 - [T18] Face4
 - [T15] Drill10 [Rapid out]
 - [T16] Drill11 [Deep drilling]
 - [T6] Pocket6
 - [T10] 2D Contour8
 - SNAP RING CAVITY ○
 - [T18] Face7
 - [T1] Pocket8
 - [T10] 2D Contour10
 - BODY CORE** ●
 - [T18] Face8
 - [T15] Drill13 [Rapid out]
 - [T16] Drill14 [Deep drilling]
 - [T13] Adaptive5
 - [T13] Spira4

Post-processor Output Window:

```
1 (1001)
2 (BODY CORE OPERATION)
3 (T06 D=0.25 CR=0. - ZMIN=-0.1415 - FLAT END MILL)
4 (T09 D=0.0625 CR=0.0312 - ZMIN=-0.5348 - BALL END I
5 (T13 D=0.5 CR=0.125 - ZMIN=-0.406 - BULLNOSE END M
6 (T14 D=0.0625 CR=0. TAPER=5DEG - ZMIN=-0.406 - TAPI
7 (T15 D=0.1875 CR=0. TAPER=120DEG - ZMIN=-0.03 - CEI
8 (T16 D=0.12 CR=0. TAPER=118DEG - ZMIN=-1.4525 - DR
9 (T18 D=2. CR=0. - ZMIN=-0.02 - FACE MILL)
10 N0010 G90 G94 G17
11 N0015 G20
12 (FACE8)
13 N0020 M09
14 N0025 T18 M06
15 N0030 T15
16 N0035 S5000 M03
```

1. Highlight the name of the setup by clicking the name of the Setup you wish to post (i.e. BODY CORE)



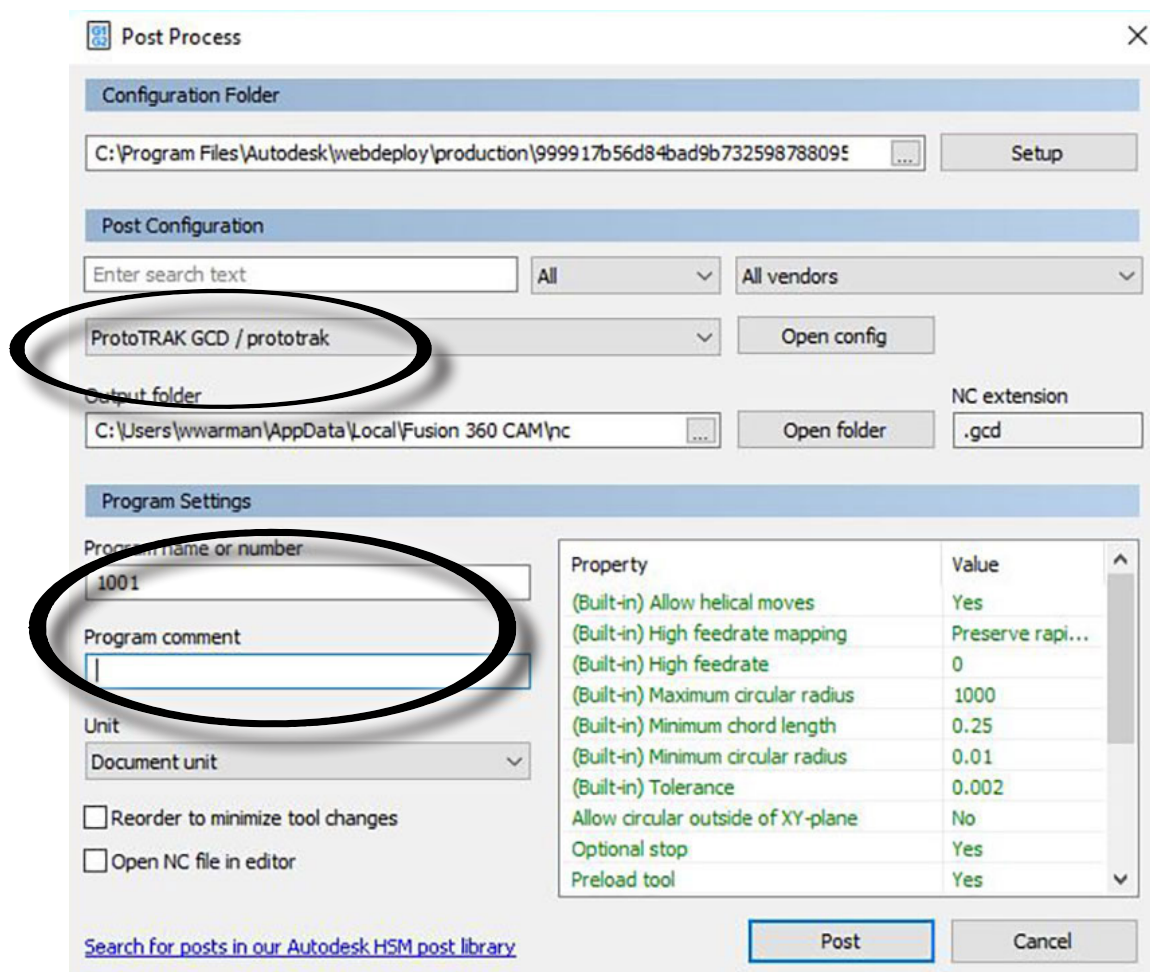
2. In the top toolbar, click the POST PROCESS button (looks like a snippet of G Code)



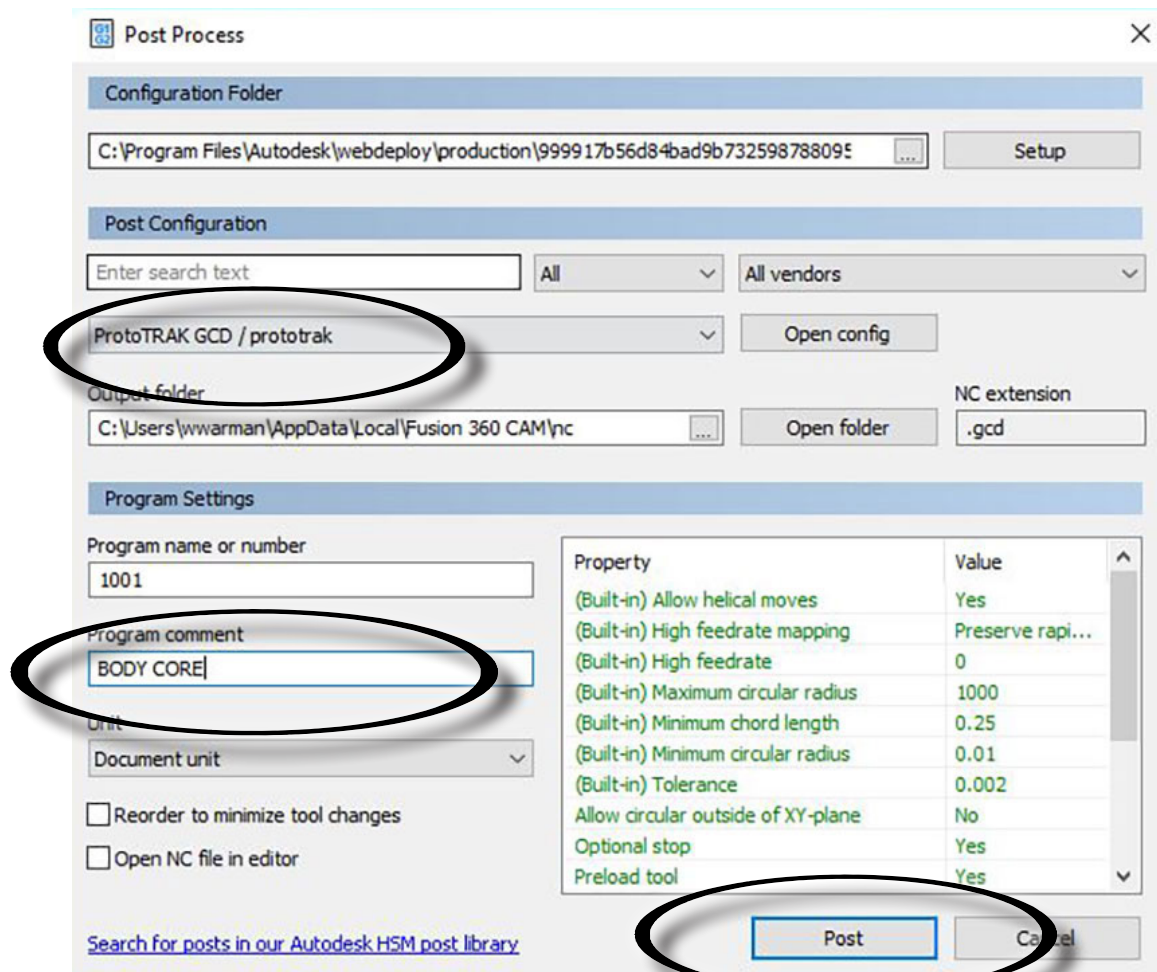
3. You will get a pop-up window. In this window, there are a few important things to adjust:

a. In the section labeled “post Configuration”, make sure that you have chosen “PROTOTRAK GCD / Prototrak”. This tells Fusion that you are using a Prototrak machine instead of a different type of CNC.

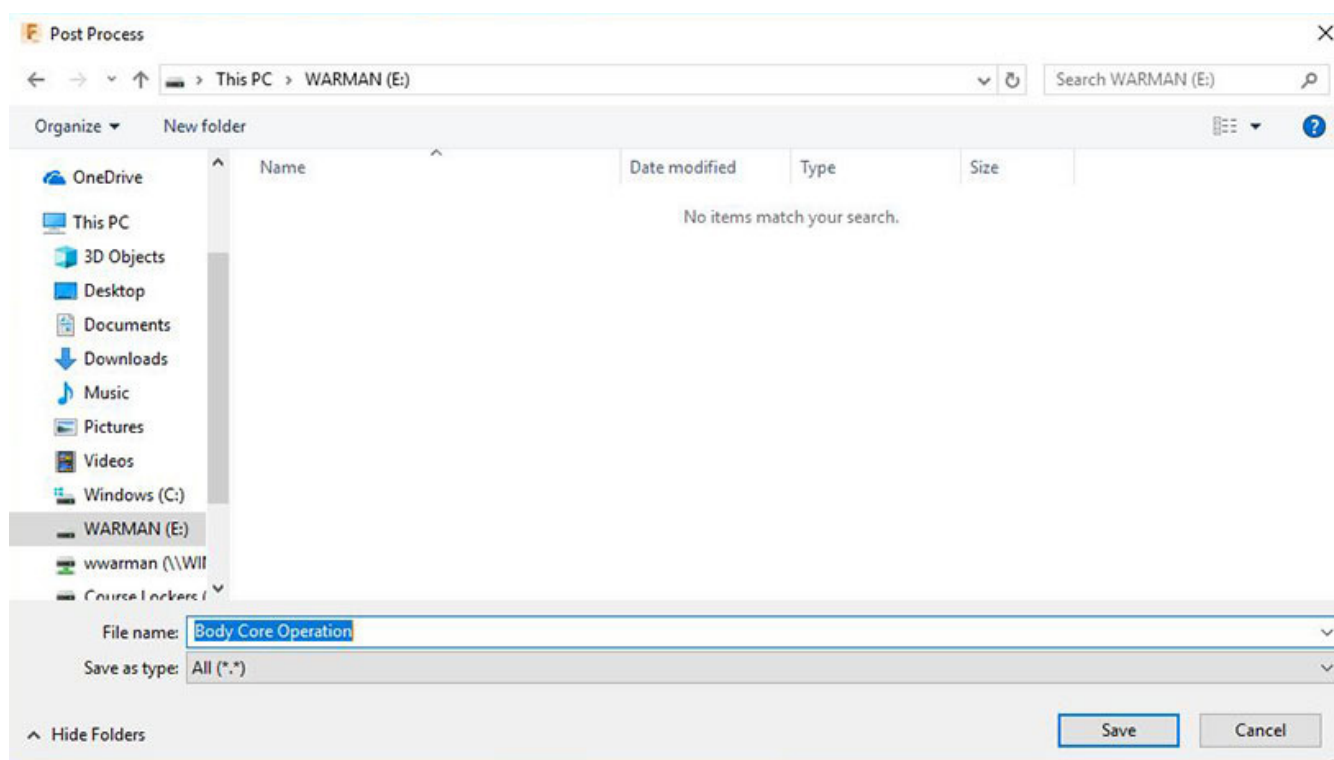
b. In the section of the window labeled “Program Settings” you will see a number, 1001. Beneath this is a field where you can add a comment. It is extremely helpful to add a comment as this can help you see if your toolpath was successfully loaded into the Prototrak later.



4. After you have ensured that you are posting to the Prototrak and have written your comment, click the POST button in the lower right-hand corner of the screen.



5. Name your file something unique and save it to a thumb drive.



6. As soon as you click the SAVE button, a new window will pop up with your G Code. It is very helpful to look at the G Code and pay attention to a few key details:
- On line 2 of your G Code, it should contain the comment, (In parenthesis) you made while processing.
 - After your comment, (again, in parenthesis), you should see a list of all the tools that you will be using (i.e. T06, T09, T13, etc.)
 - Below your tool list you will see the first operation in parenthesis, (once again, in parenthesis). This should match the first operation that you chose to post from Fusion.
 - NOTE: If any of these do not look accurate, go back to Fusion to determine what went wrong.

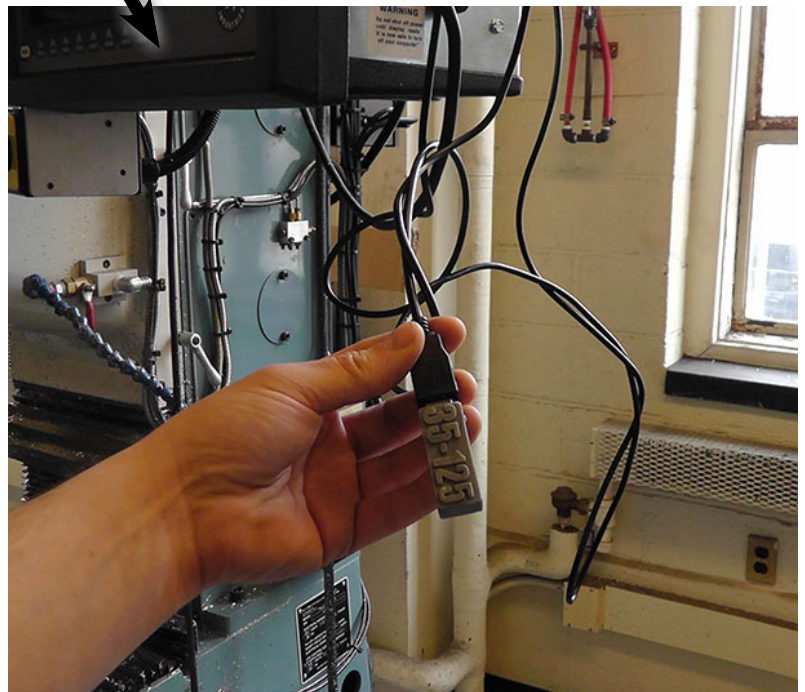
The image shows a screenshot of a G-code output window. The G-code is as follows:

```
1 |(1001)|
2 (BODY CORE OPERATION)
3 (T06 D=0.25 CR=0. - ZMIN=-0.1
4 (T09 D=0.0625 CR=0.0312 - ZMI
5 (T13 D=0.5 CR=0.125 - ZMIN=-0
6 (T14 D=0.0625 CR=0. TAPER=5DE
7 (T15 D=0.1875 CR=0. TAPER=120
8 (T16 D=0.12 CR=0. TAPER=118DE
9 (T18 D=2. CR=0. - ZMIN=-0.02
10 N0010 G90 G94 G17
11 N0015 G20
12 (FACE8)
13 N0020 M09
14 N0025 T18 M06
15 N0030 T15
16 N0035 S5000 M03
```

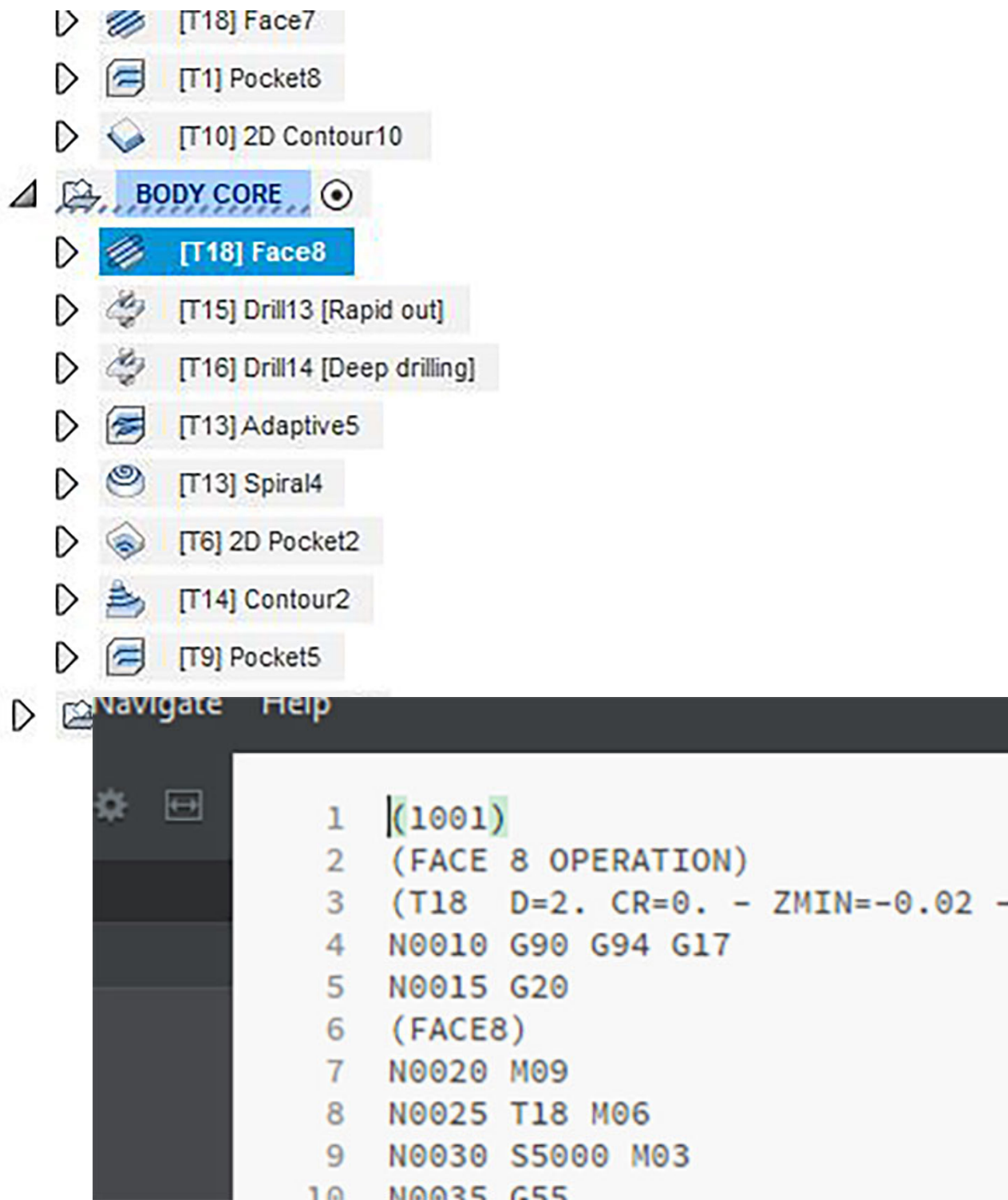
Three callout boxes with arrows point to specific lines:

- The first box, "IS YOUR COMMENT CORRECT?", points to line 1: |(1001)|
- The second box, "ARE THESE THE TOOLS YOU CHOSE?", points to line 6: (T14 D=0.0625 CR=0. TAPER=5DE
- The third box, "DOES THIS MATCH THE FIRST OPERATION IN FUSION 360?", points to line 12: (FACE8)

7. Take your thumb drive out of your computer, and install it into the USB dongle connected to the Prototrak



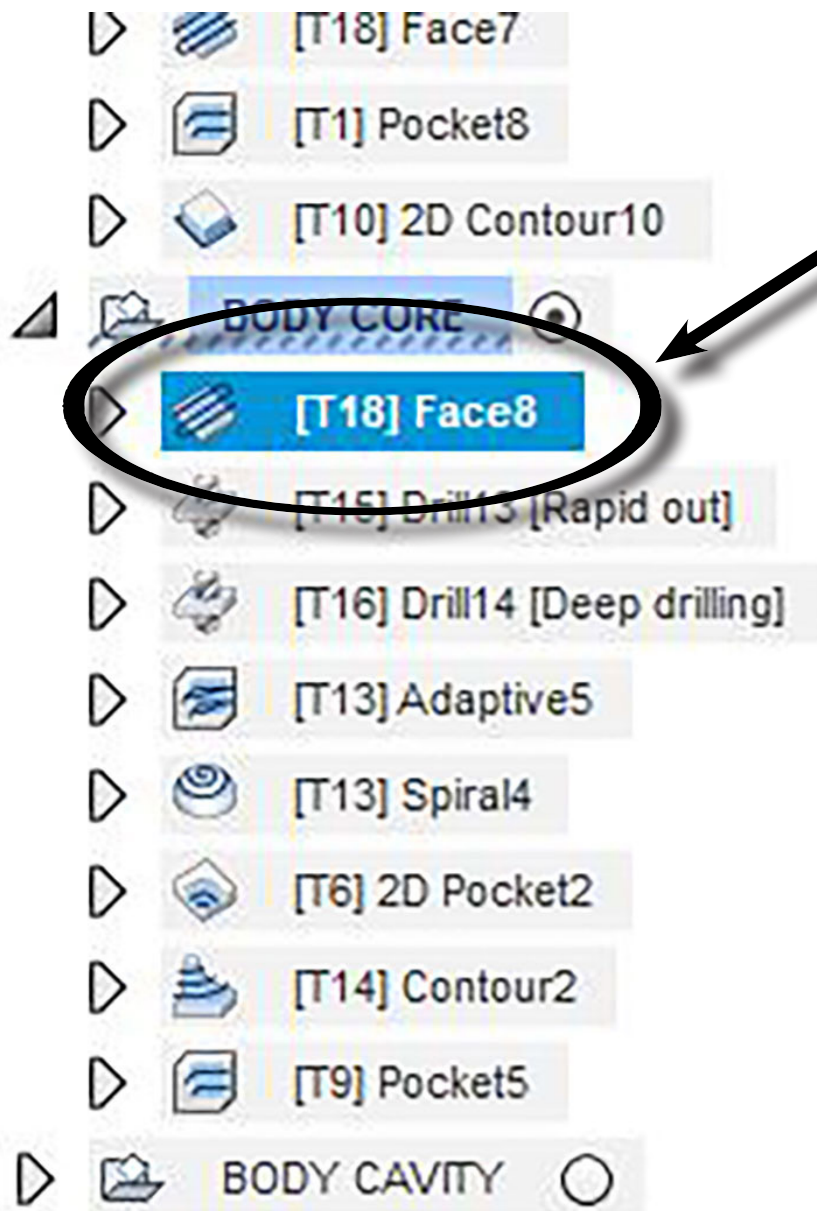
TO POST A SINGLE TOOLPATH OPERATION



The image shows a CAD software interface with a tree view of operations. The 'BODY CORE' folder is expanded, and '[T18] Face8' is selected. Below the tree view, a post-processor output window is open, displaying the following code:

```
1 |(1001)
2 (FACE 8 OPERATION)
3 (T18 D=2. CR=0. - ZMIN=-0.02 -
4 N0010 G90 G94 G17
5 N0015 G20
6 (FACE8)
7 N0020 M09
8 N0025 T18 M06
9 N0030 S5000 M03
10 N0035 G55
```

1. Highlight the name of the toolpath you want to use by clicking on it (i.e. Face8)



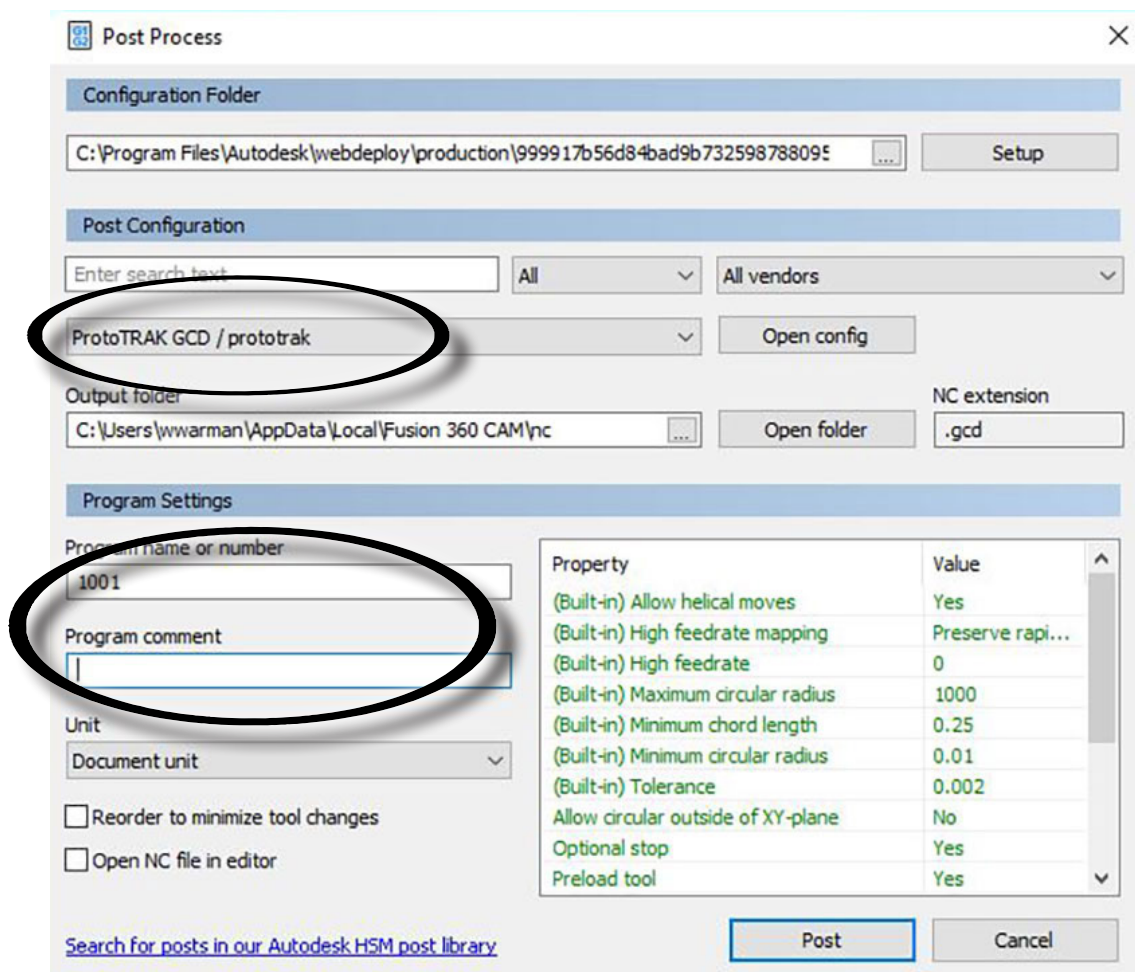
2. In the top toolbar, click the POST PROCESS button (looks like a snippet of G Code)



3. You will get a pop-up window. In this window, there are a few important things to adjust:

a. In the section labeled “post Configuration”, make sure that you have chosen “PROTOTRAK GCD / Prototrak”. This tells Fusion that you are using a Prototrak machine instead of a different type of CNC.

b. In the section of the window labeled “Program Settings” you will see a number, 1001. Beneath this is a field where you can add a comment. It is extremely helpful to add a comment as this can help you see if your toolpath was successfully loaded into the Prototrak later.



4. After you have ensured that you are posting to the Prototrak and have written your comment, click the POST button in the lower right-hand corner of the screen.

Post Process

Configuration Folder

C:\Program Files\Autodesk\webdeploy\production\999917b56d84bad9b732598788095

Setup

Post Configuration

Enter search text

All

All vendors

ProtoTRAK GCD / prototrak

Open config

Output folder

C:\Users\wwarman\AppData\Local\Fusion 360 CAM\nc

Open folder

NC extension

.gcd

Program Settings

Program name or number

1001

Program comment

Face 8 Operation

Unit

Document unit

Reorder to minimize tool changes

Open NC file in editor

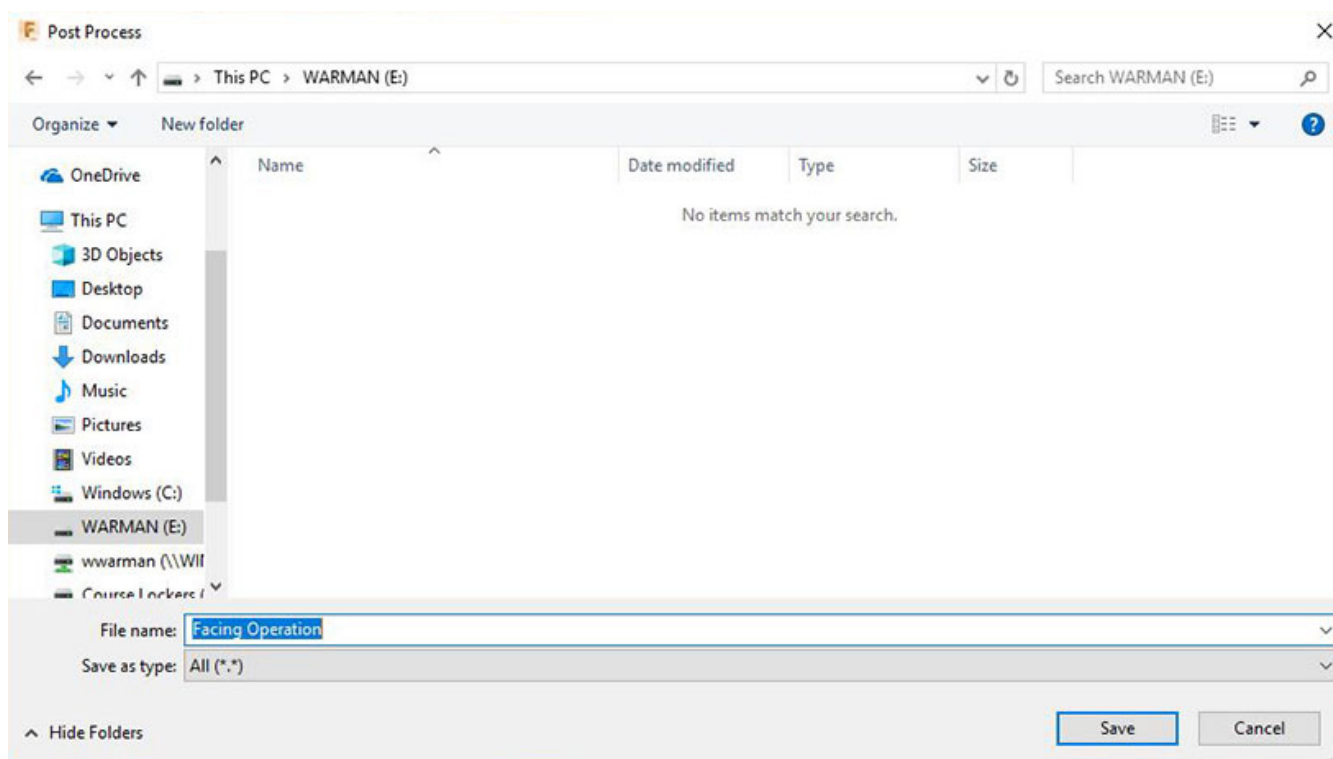
Property	Value
(Built-in) Allow helical moves	Yes
(Built-in) High feedrate mapping	Preserve rapi...
(Built-in) High feedrate	0
(Built-in) Maximum circular radius	1000
(Built-in) Minimum chord length	0.25
(Built-in) Minimum circular radius	0.01
(Built-in) Tolerance	0.002
Allow circular outside of XY-plane	No
Optional stop	Yes
Preload tool	Yes

[Search for posts in our Autodesk HSM post library](#)

Post

Cancel

5. Name your file something unique and save it to a thumb drive.



6. As soon as you click the SAVE button, a new window will pop up with your G Code. It is very helpful to look at the G Code and pay attention to a few key details:
- On line 2 of your G Code, it should contain the comment, (In parenthesis) you made while processing.
 - After your comment, (again, in parenthesis), you should see the tool that you will be using (i.e. T18)
 - Below your tool list you will see the name of the operation, (once again, in parenthesis). This should match the operation that you chose to post from Fusion.
 - NOTE: If any of these do not look accurate, go back to Fusion to determine what went wrong.

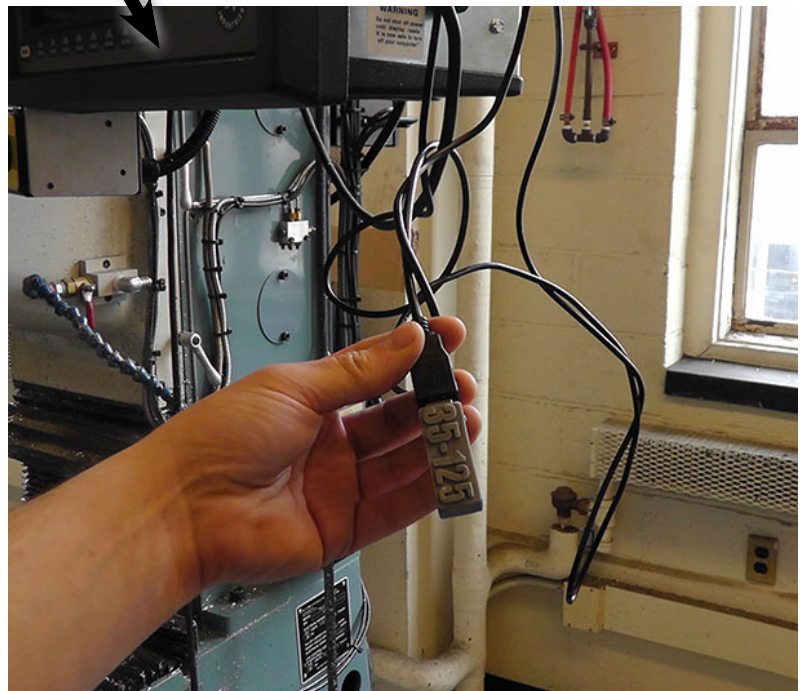
IS YOUR COMMENT
CORRECT?

IS THIS THE TOOL
YOU CHOSE?

DOES THIS MATCH THE
OPERATION IN FUSION
360?

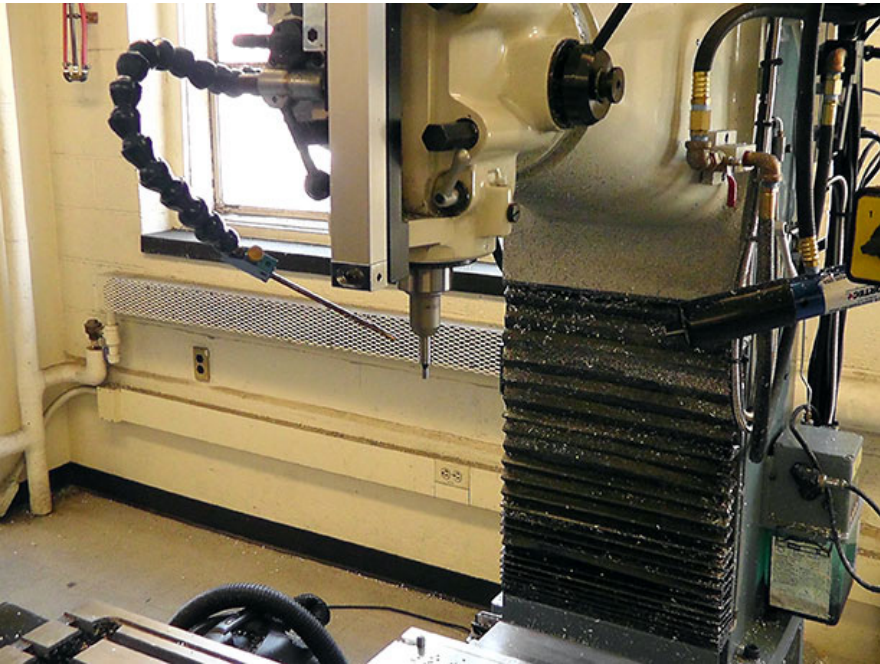
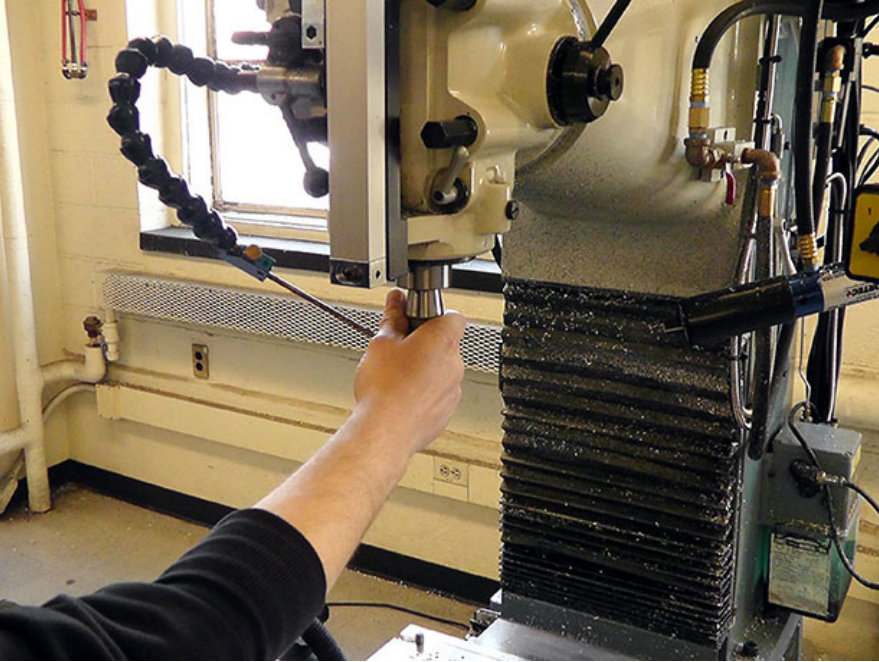
```
ate Help
1 |(1001)
2 (FACE 8 OPERATION)
3 (T18 D=2. CR=0. - ZM
4 N0010 G90 G94 G17
5 N0015 G20
6 (FACE8)
7 N0020 M09
8 N0025 T18 M06
9 N0030 S5000 M03
10 N0035 G55
```


7. Take your thumb drive out of your computer, and install it into the USB dongle connected to the Prototrak

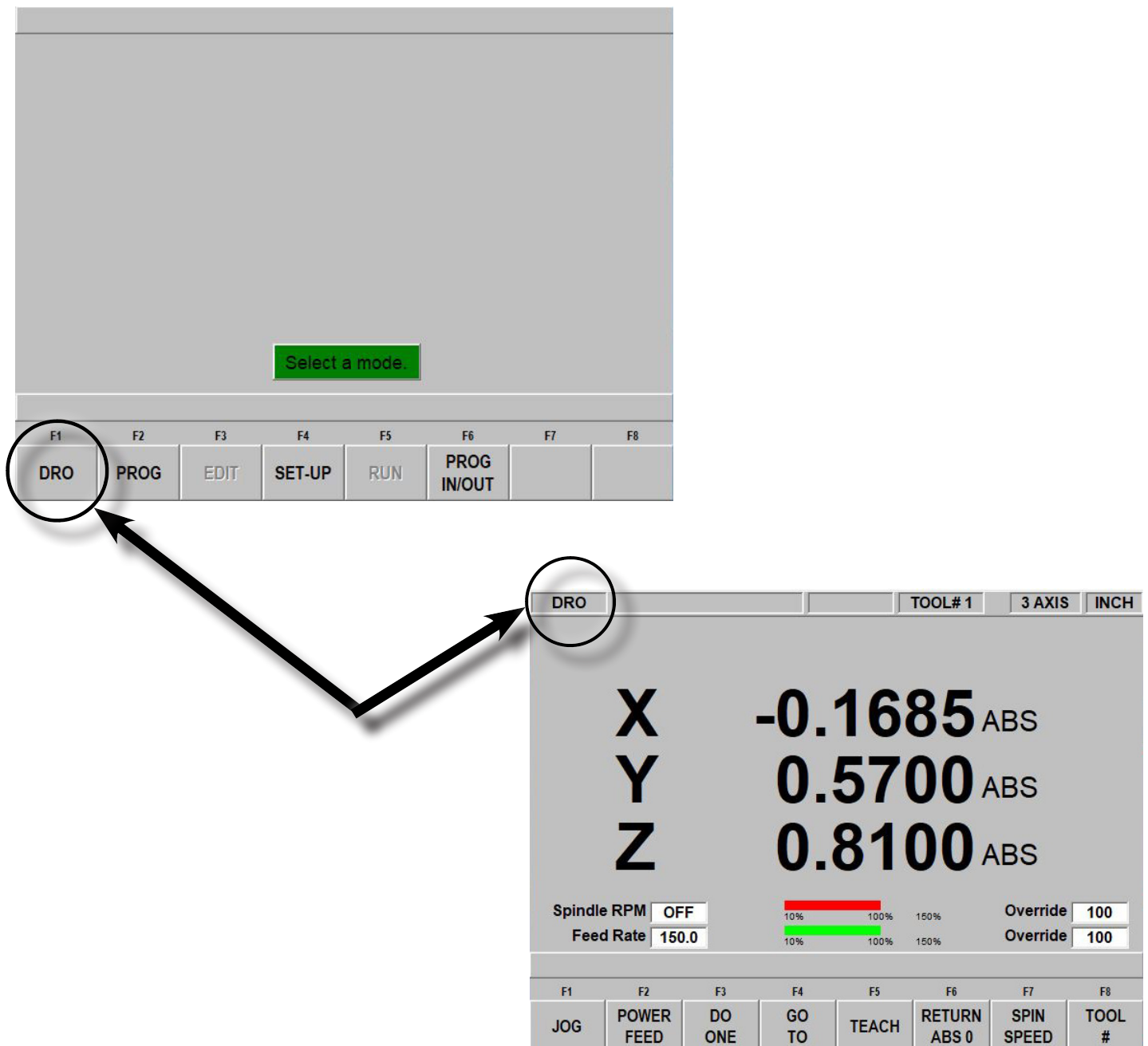


EDGE FINDING WITH THE PROTOTRAK

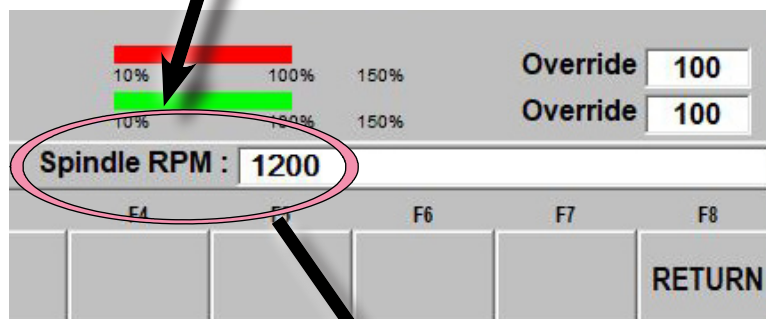
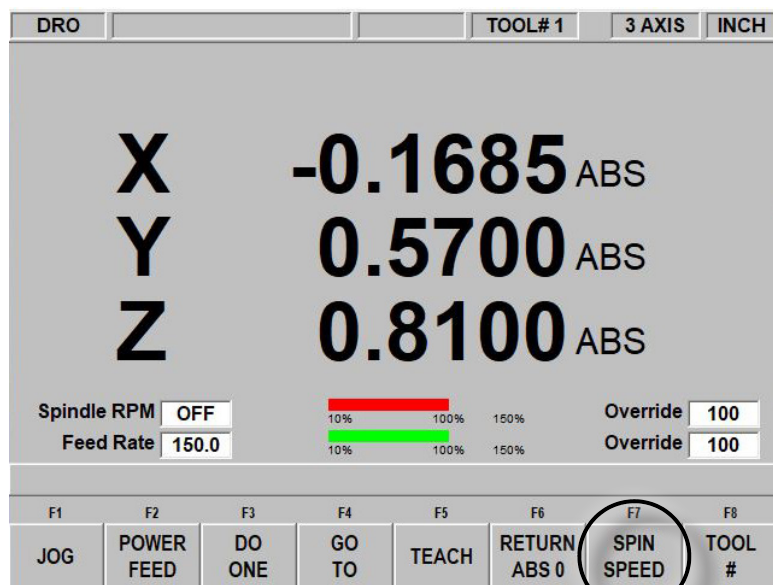
1. Install the edge finding tool



2. Make sure you are in the "DRO" Screen of the Prototrak controller

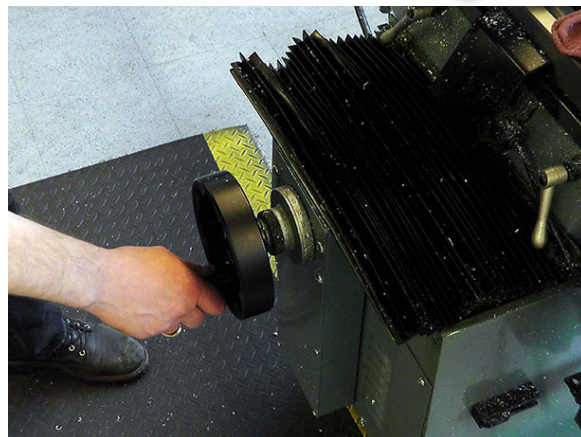
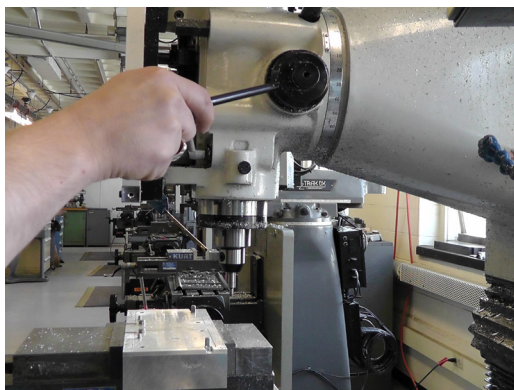
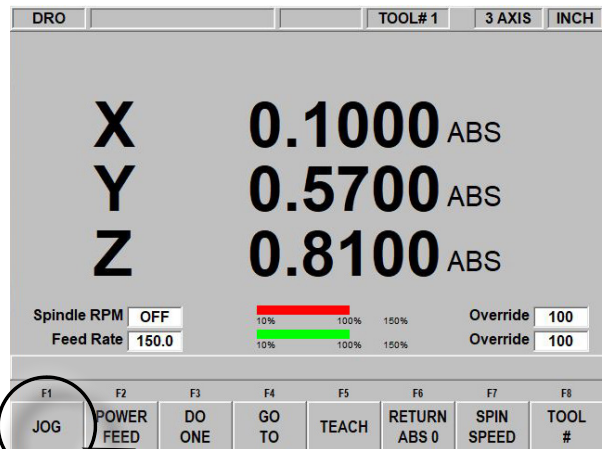
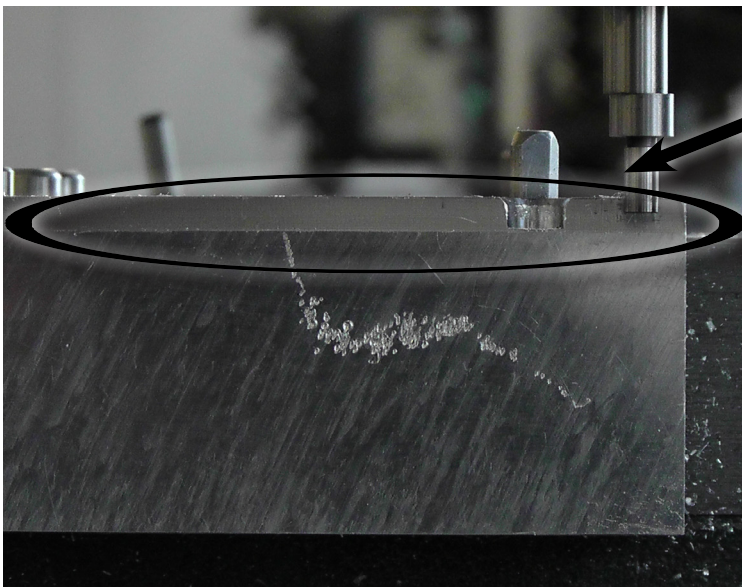


3. Change spindle speed to 1200 rpm on the pendant controller

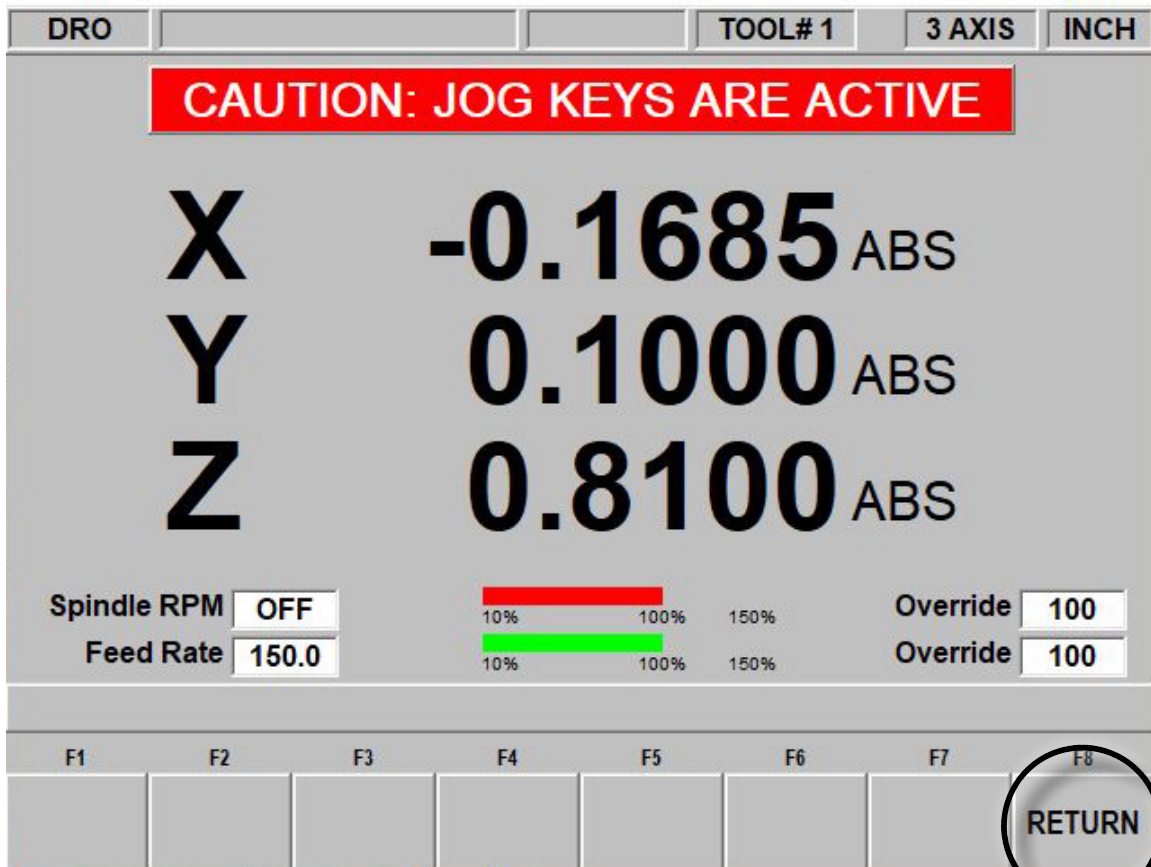


4. Jog the edge finding tool using both automatic (JOG Softkey) and manual (handwheels) so that the tip is positioned close to the **finished** edge of the fixture.

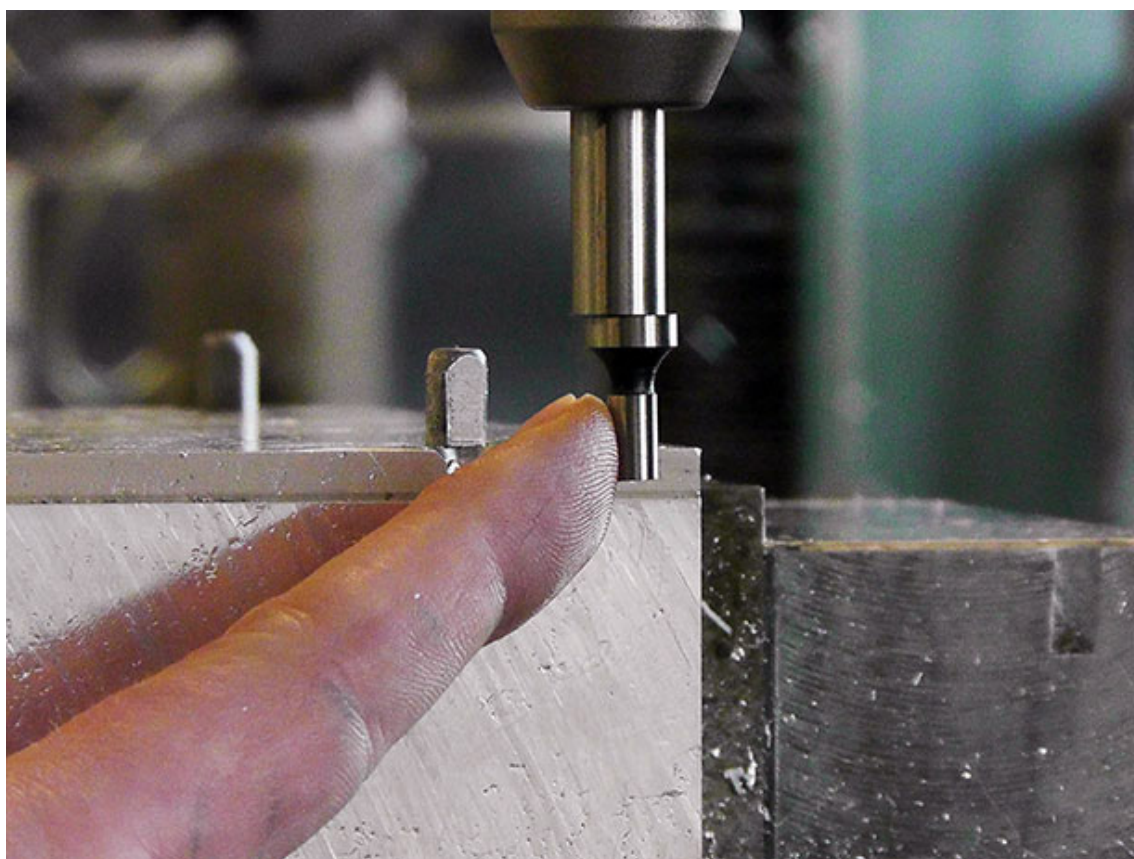
a. NOTE: For the Z-Axis, it is a good idea to use the automatic JOG button on the pendant controller to get close, and finish using the manual Quill handle. Don't forget to lock the Quill in place when you have it set at the right height!!!)



5. Return to the DRO by pressing the RETURN Softkey



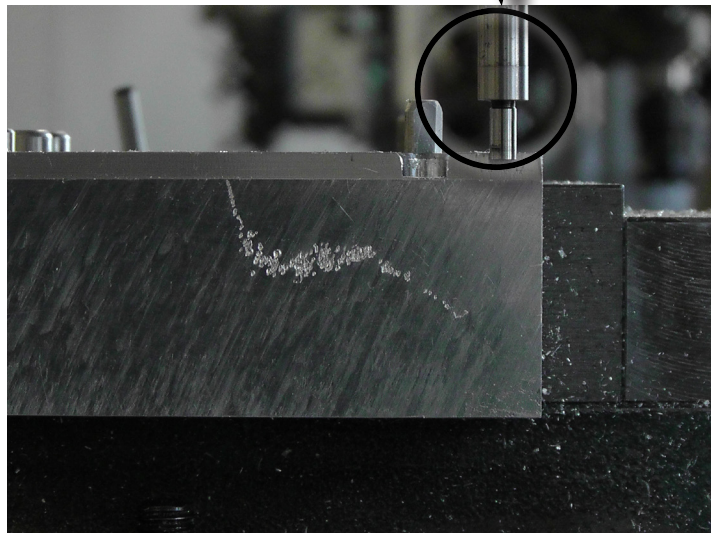
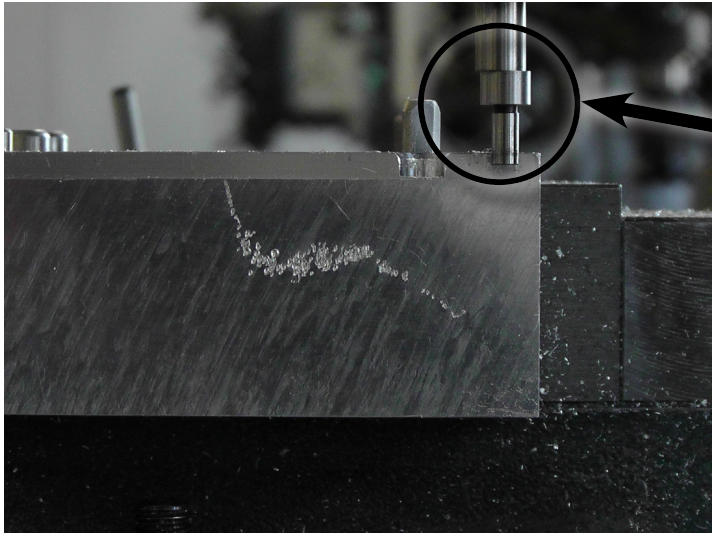
6. Make sure that the tip is off center BEFORE you start the spindle. (You can do this by gently touching the tip)



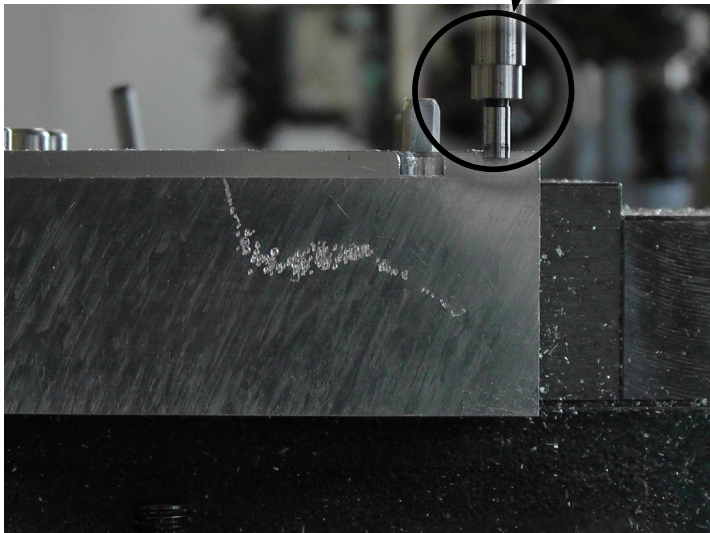
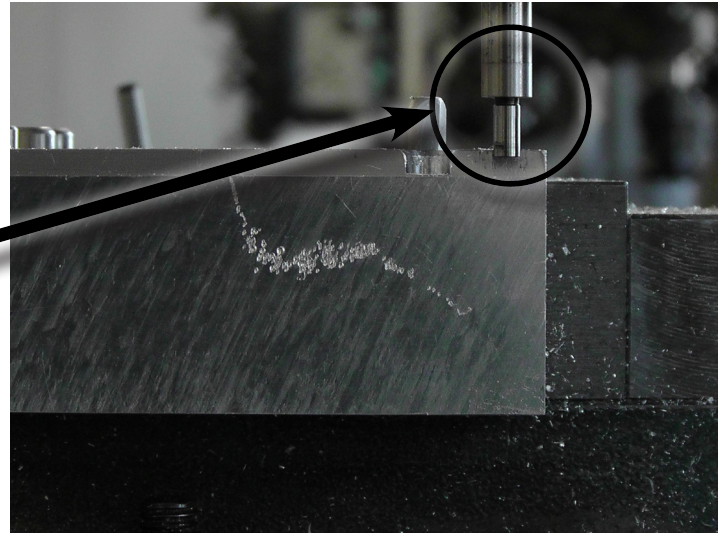
7. Turn on the spindle by turning the motor switch to the 2 position



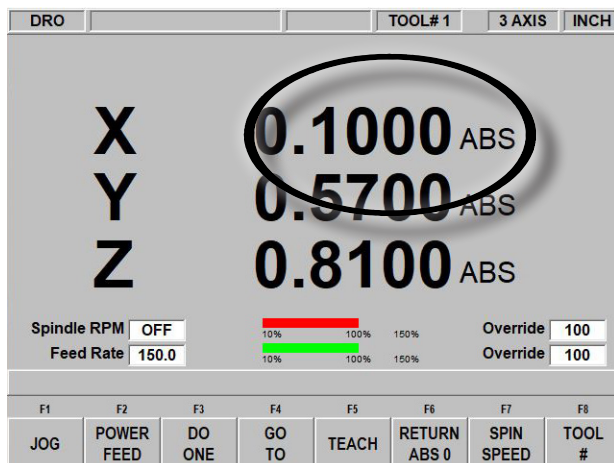
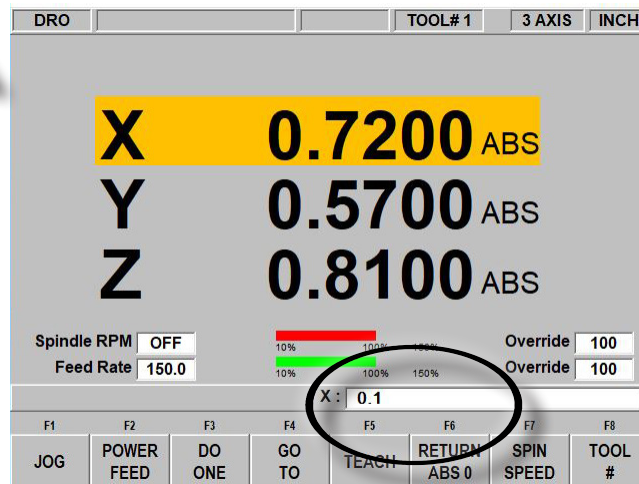
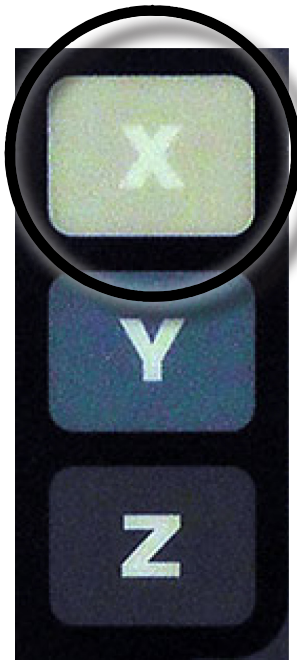
8. Slowly jog toward the edge of the fixture using the handwheels in the X axis edge until the tip of the edge finder touches the fixture edge. This will re-center the edge finder tool.



9. Very slowly continue to jog the edge finder tool until it moves the tip of the edge finder tool off center once again.



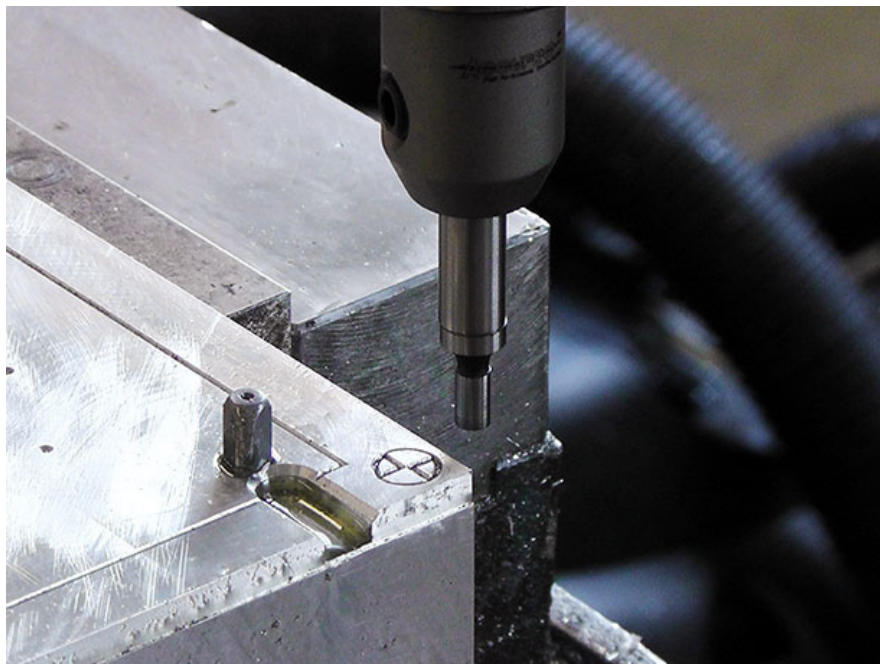
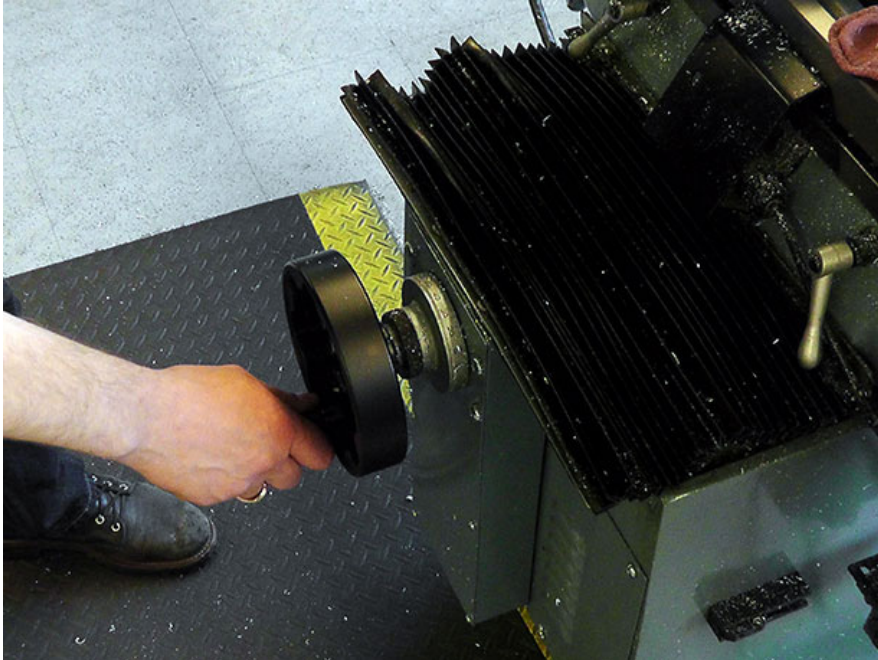
10. On the controller pendant in the DRO (Digital Read Out) screen, press the X button and then type in 0.1 using the number pad, then press the ABS SET button. The X axis is now zeroed.



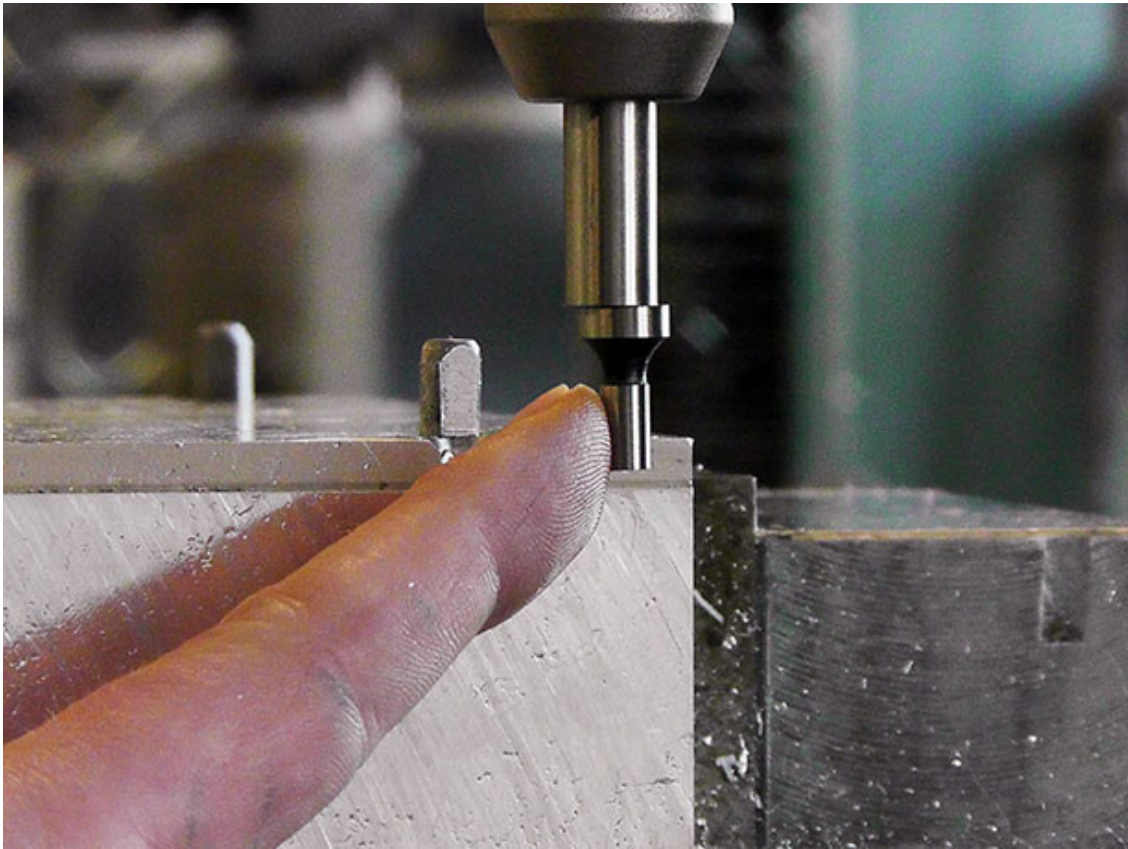
11. Turn off the spindle



12. Using ONLY the handwheels, move the table so that the Edge finder Tool is now on the Y-Axis side of the fixture



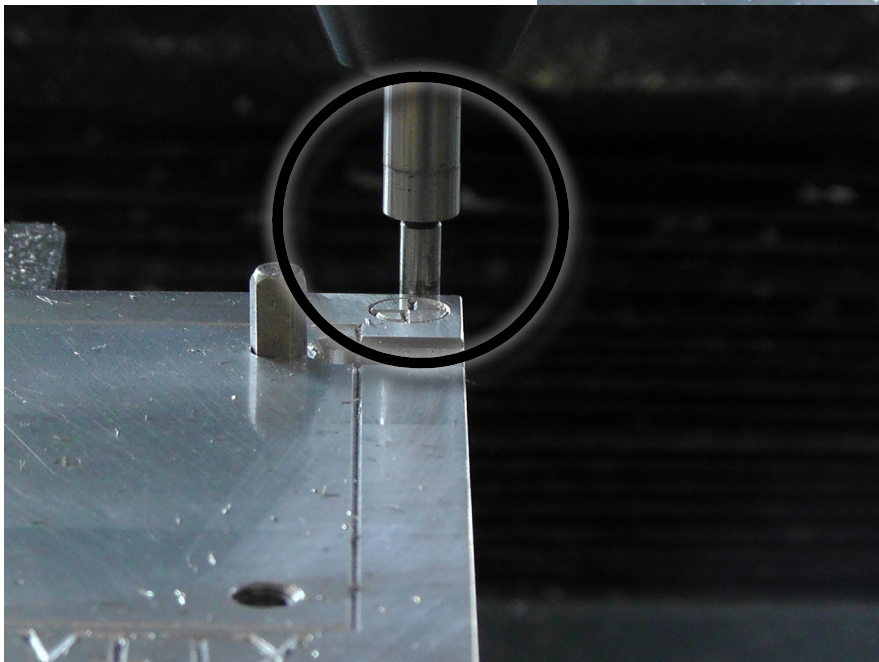
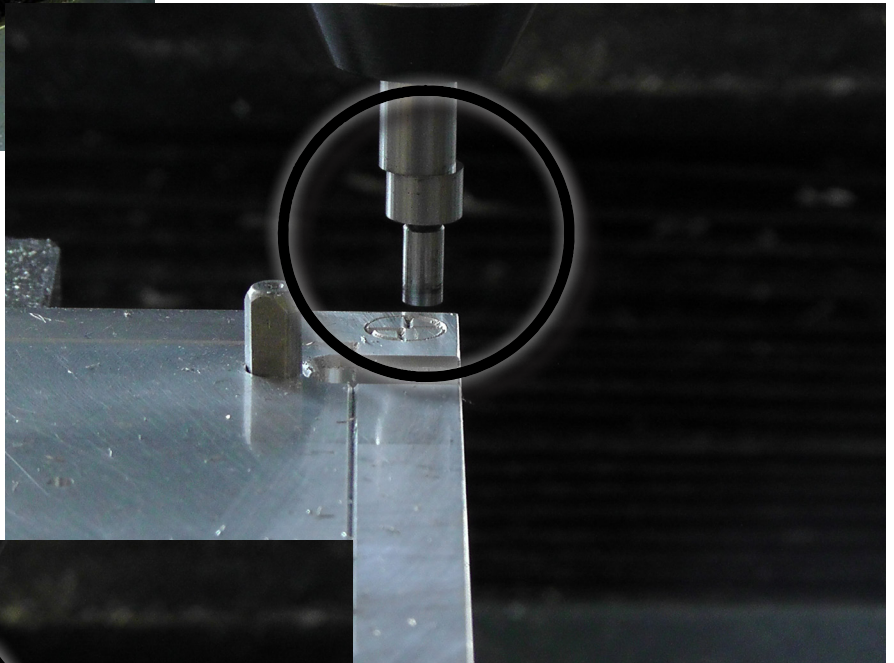
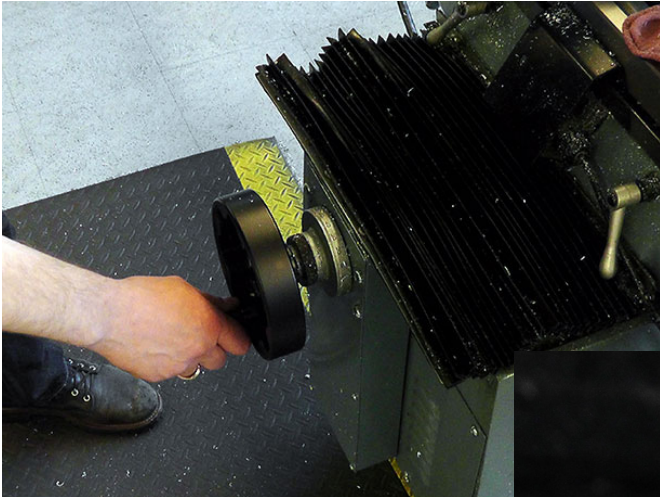
13. Make sure that the tip is off center again. (You can do this by gently touching the tip)



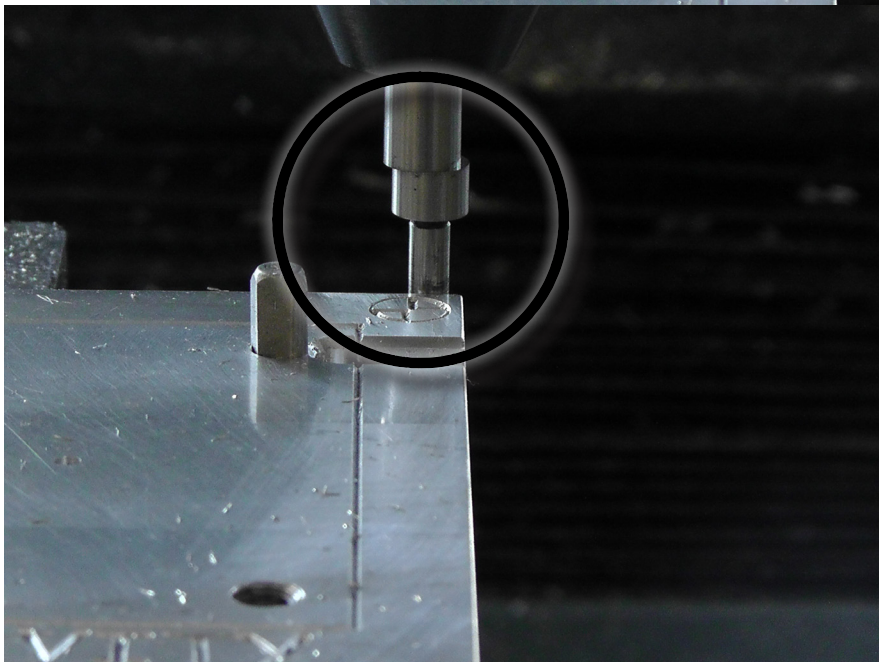
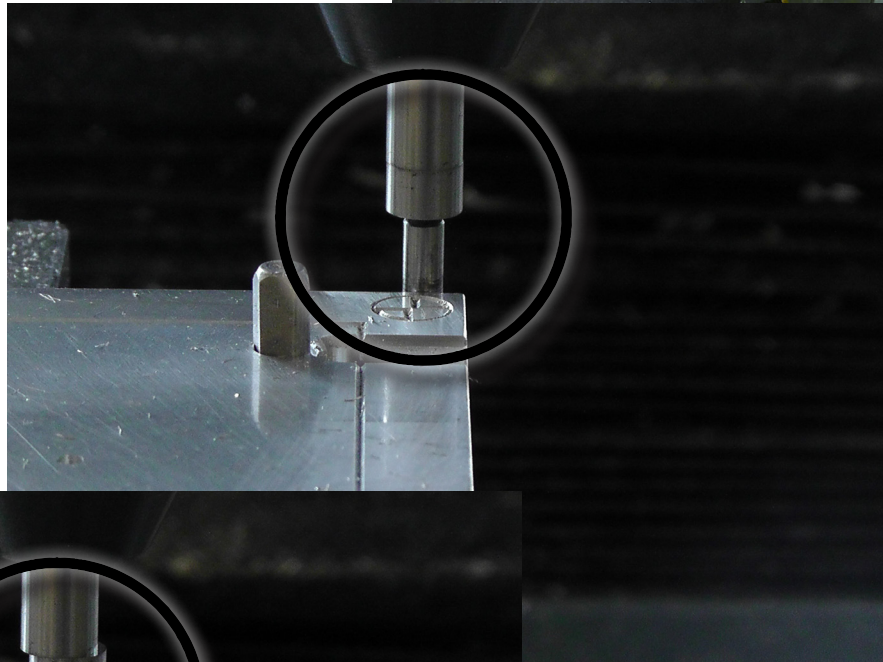
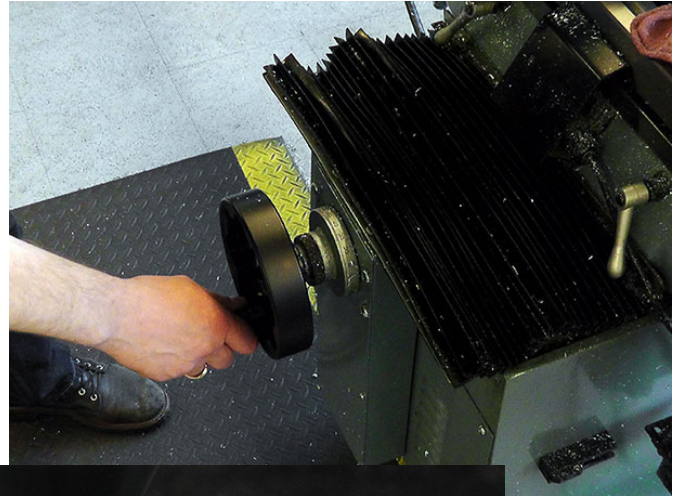
14. Turn on the spindle again by turning the motor switch to the 2 position



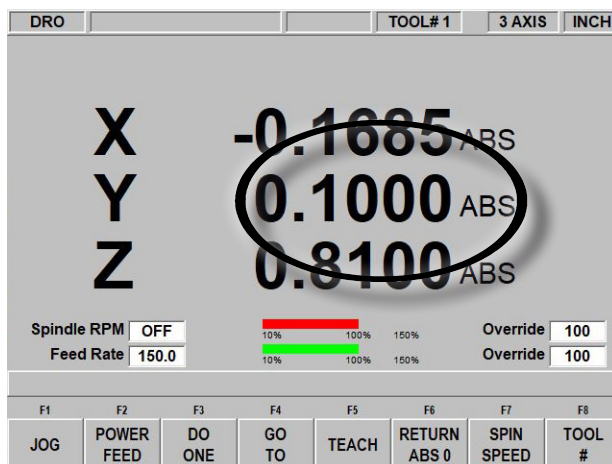
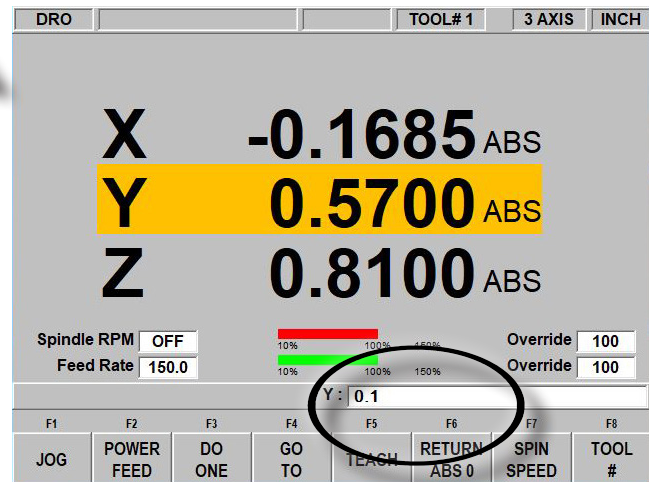
15. Using the handwheels, slowly jog toward the edge of the fixture in the Y axis edge of the fixture until the tip of the edge finder touches the fixture edge. This will re-center the edge finder tool.



16. Very slowly continue to jog the edge finder tool until it moves the tip of the edge finder tool off center once again.



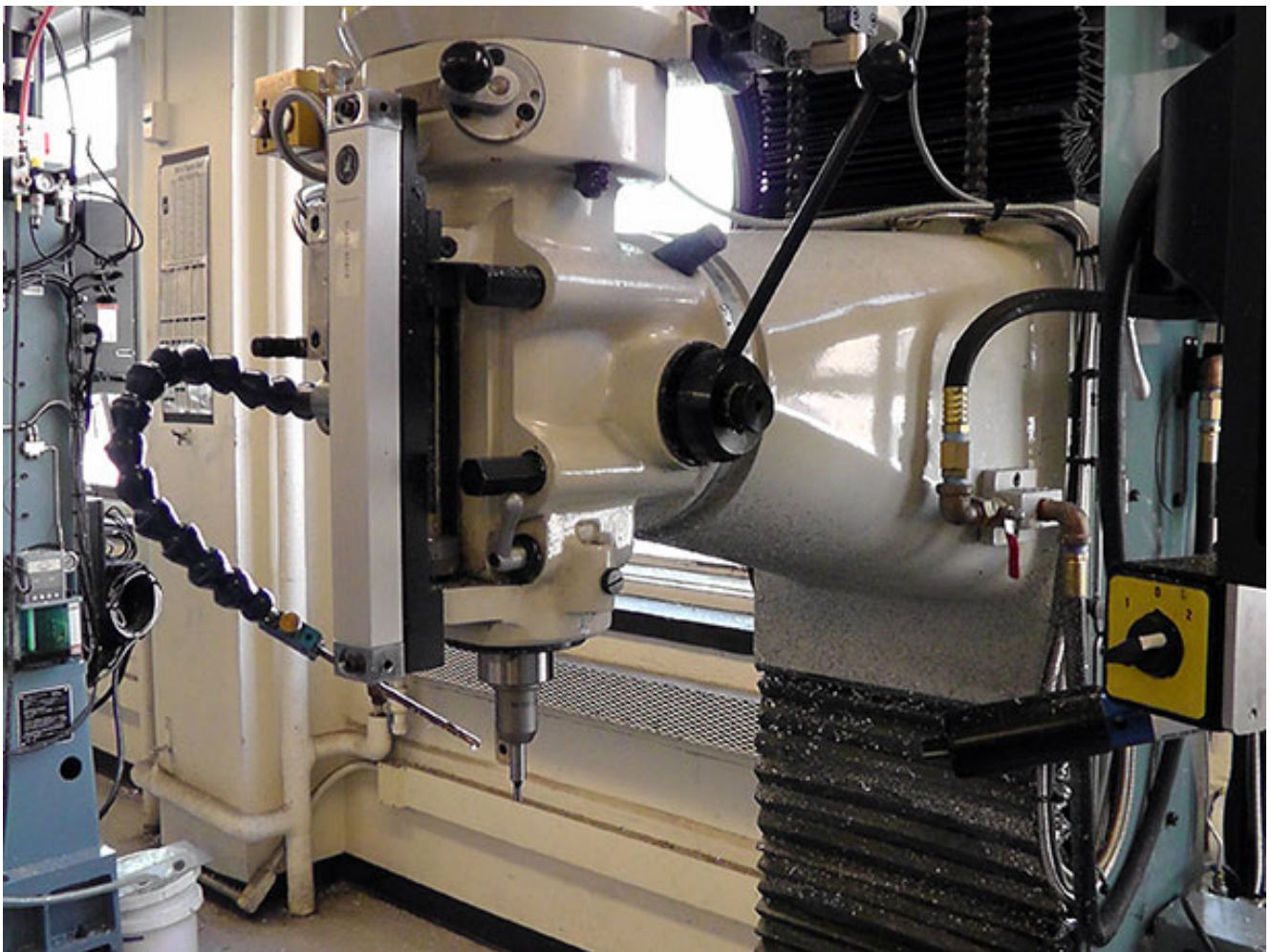
17. On the controller pendant in the DRO (Digital Read Out) screen, press the Y button and then type in 0.1 then press the ABS SET button. The Y axis is now zeroed.



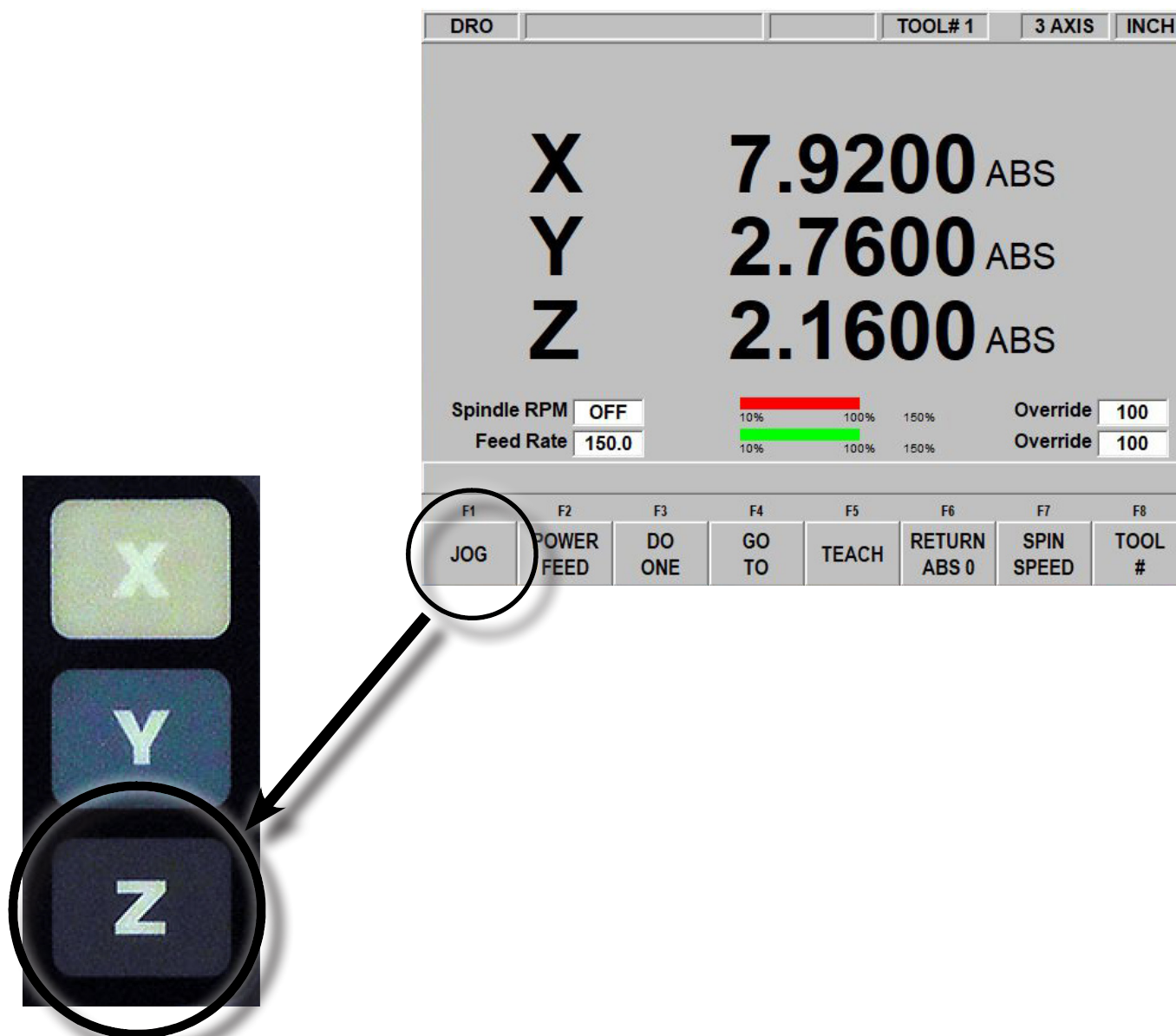
18. Turn off the spindle by turning the switch to the 0 position



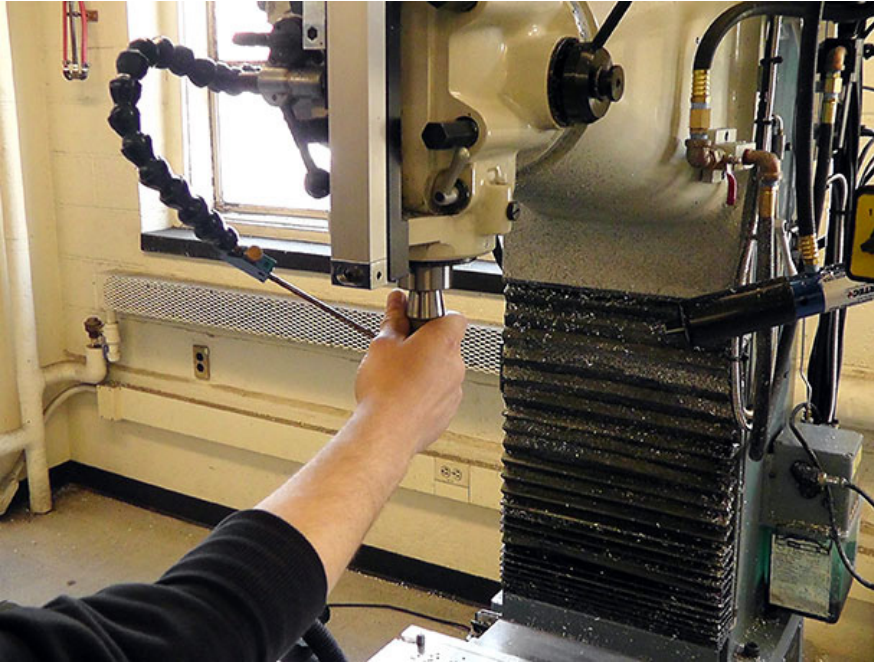
19. Unlock the Quill and use the manual Quill handle to retract the quill safely. (Make sure to re-lock the quill after it is fully retracted.)



20. If needed, press the JOG softkey and retract the spindle by pressing the Z button

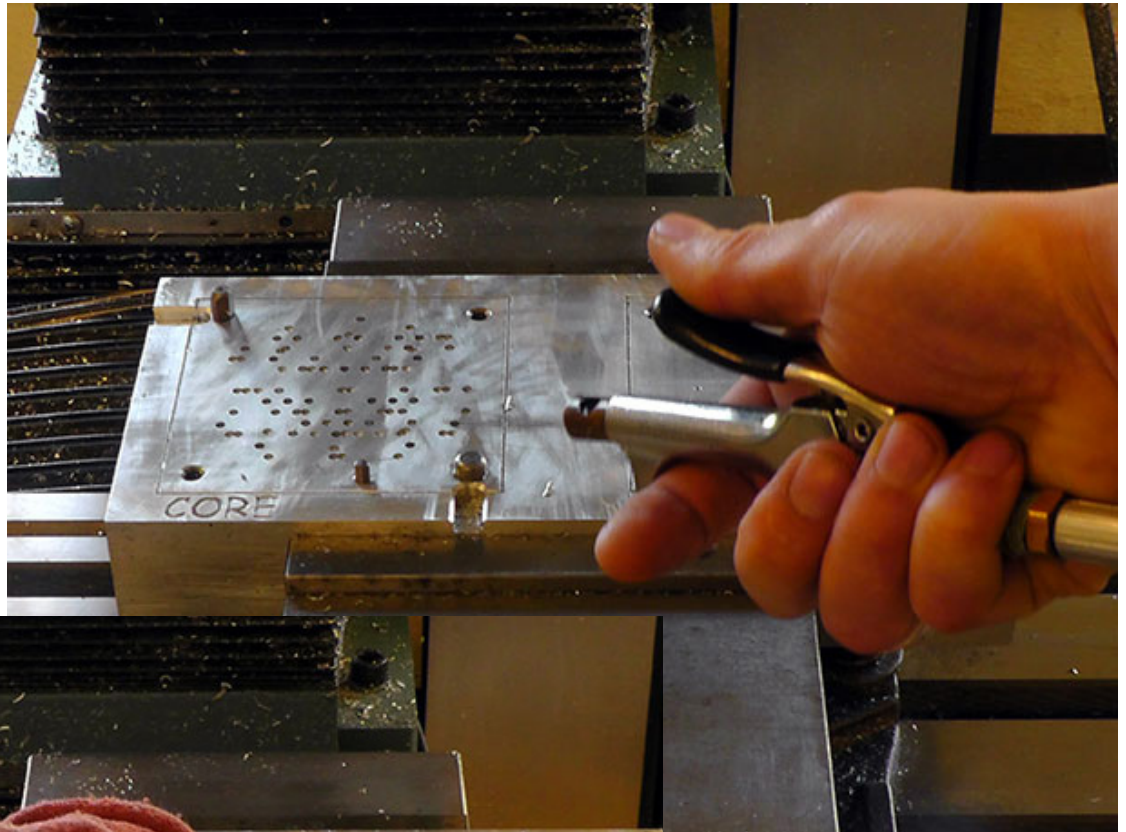


21. Remove the edge finder tool and return it to the tool rack on the bench

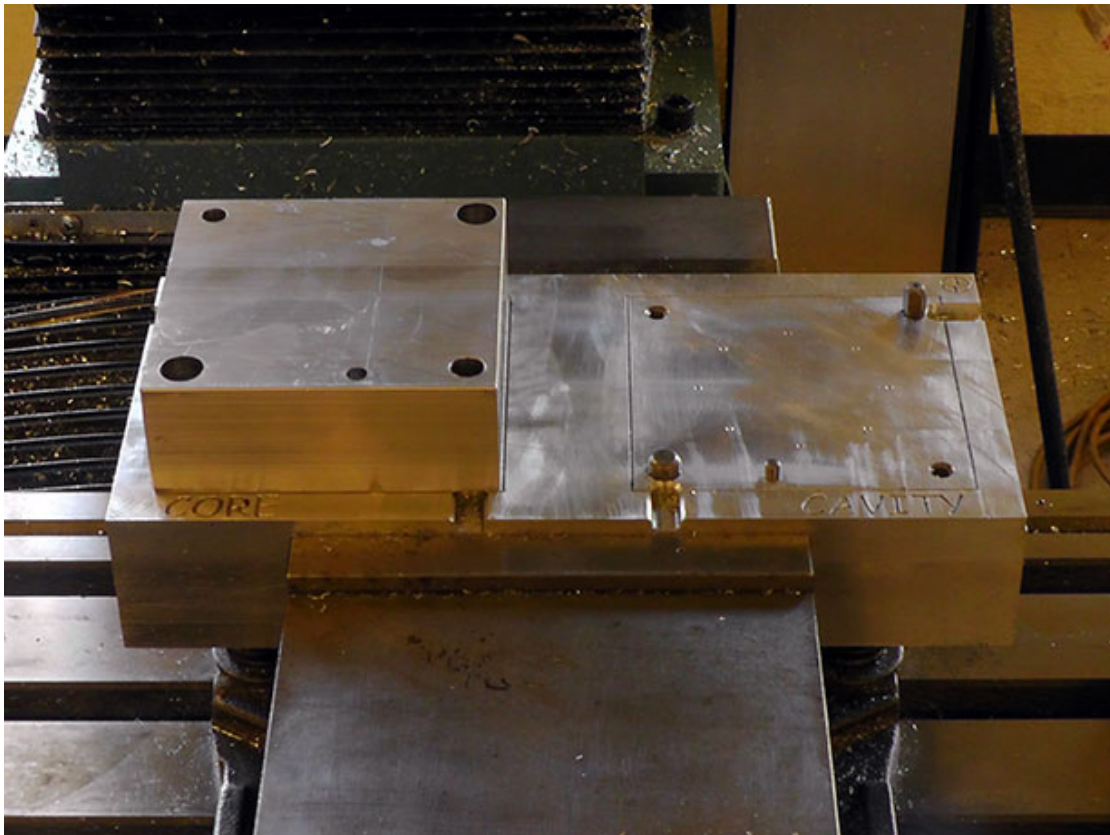


ATTACHING STOCK MATERIAL TO FIXTURE

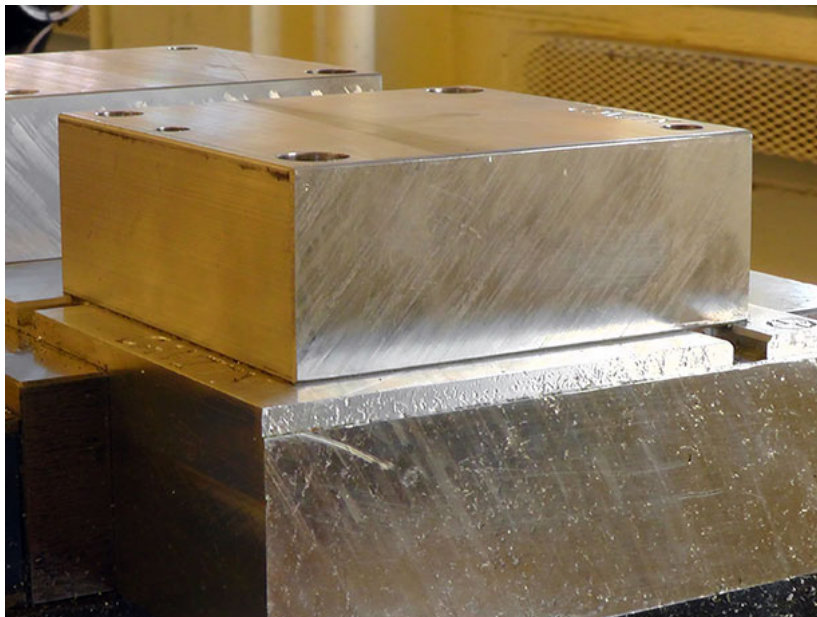
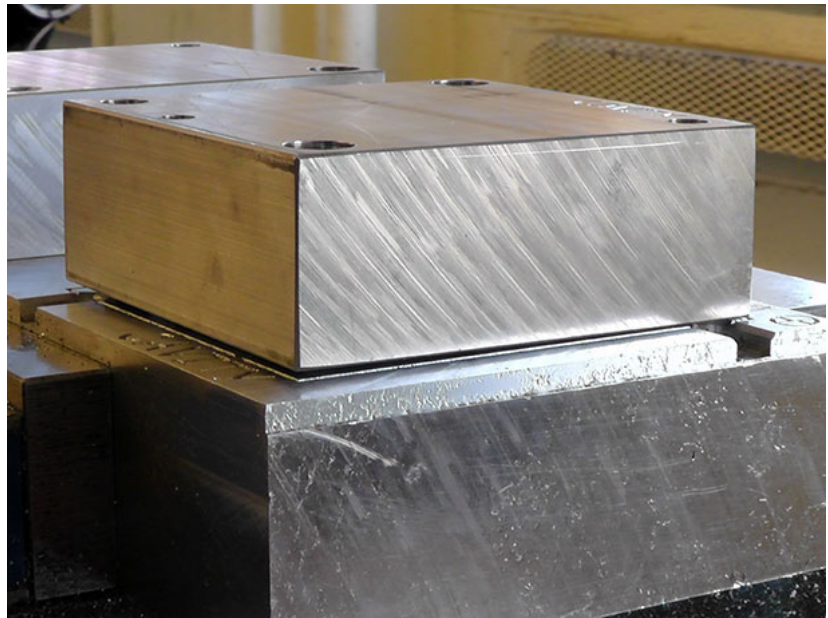
1. Make sure that the fixture is very clean and free of any left-over materials. If needed, clean thoroughly with a brush or a rag and compressed air until no chips or particles are left



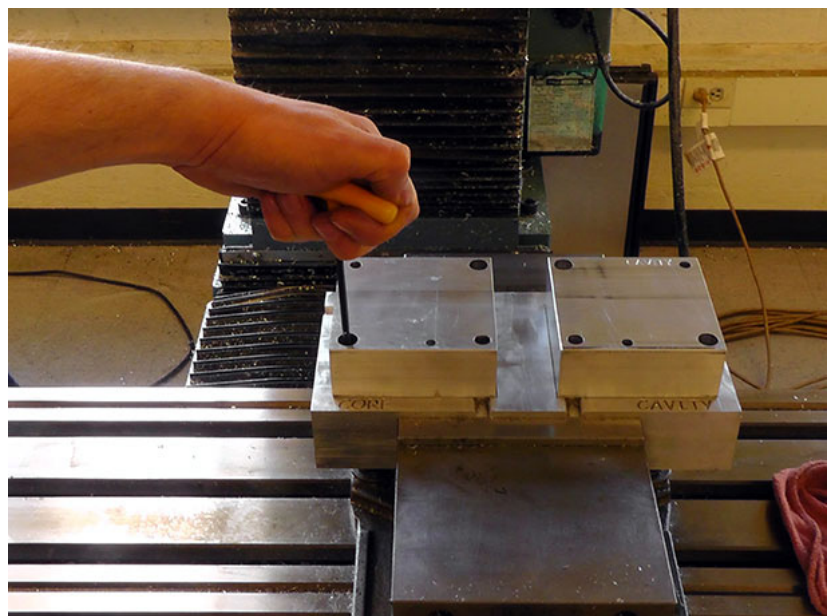
2. Place your Core/Cavity block in the appropriate location. (See section: ***USING THE WCS (World Coordinate System) WITH FUSION 360***)



3. Your material should sit flush with the fixture. If it does not, it could bind and get stuck and therefore not mill properly. If the material does not sit flush with the fixture, remove it, clean all surfaces (posts, holes, surfaces, etc.), and re-attach the part to the fixture

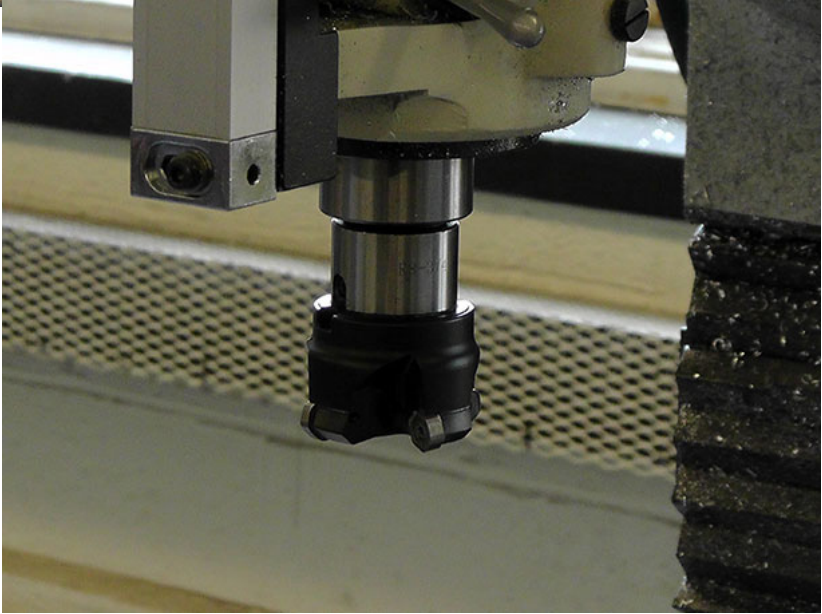
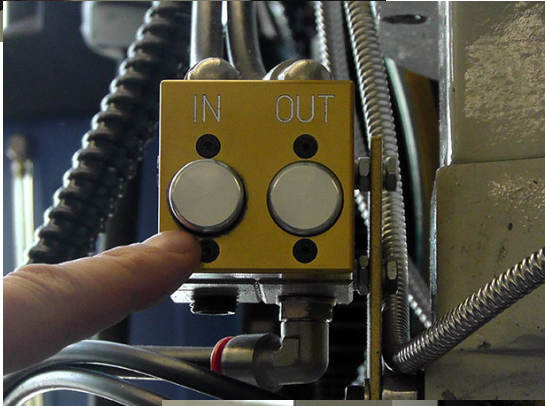


4. Using the Flat-Head Cap-screws and the 5/32 hex wrench, fasten the material to the fixture. Make sure the part is tightly snug onto the fixture.



SETTING THE Z AXIS

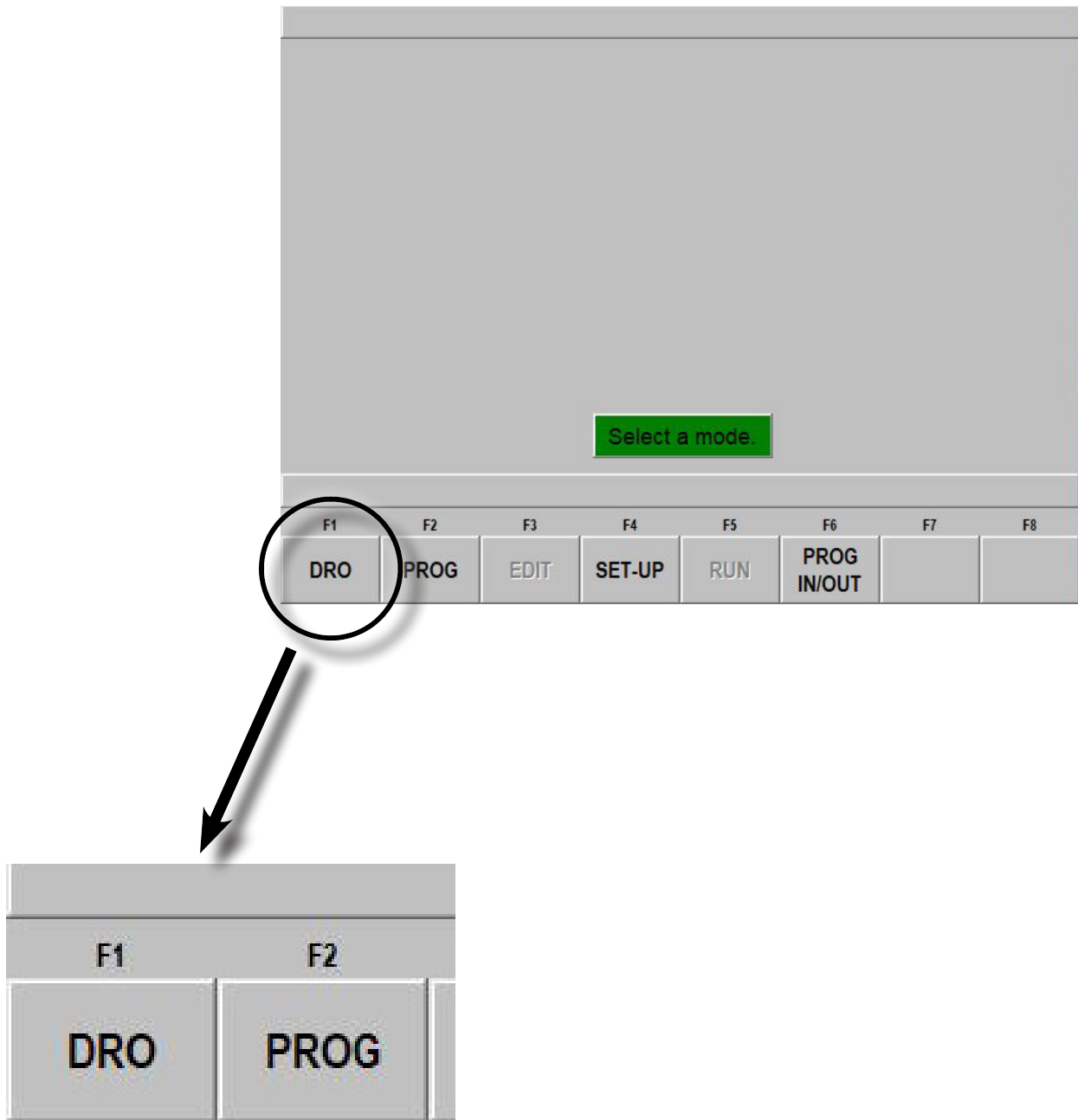
1. Install the first tool of your operation into the spindle



2. Using the pendant controller, press the MODE button

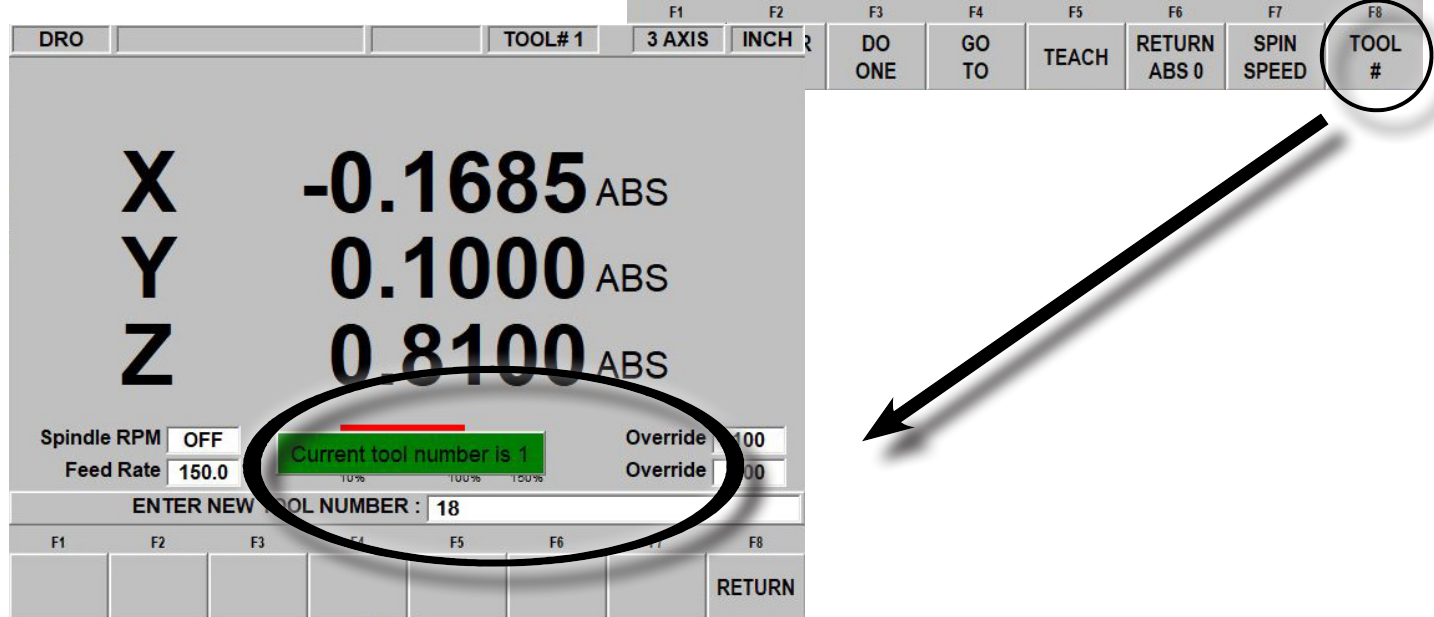
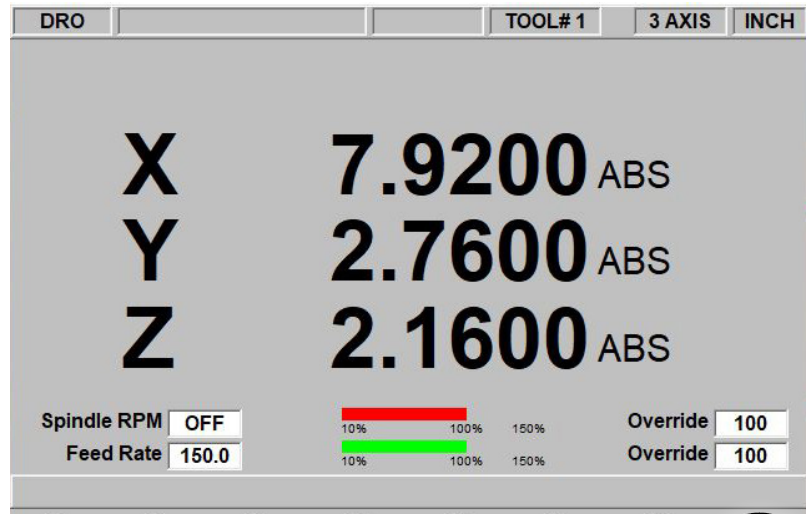


3. Press DRO Softkey

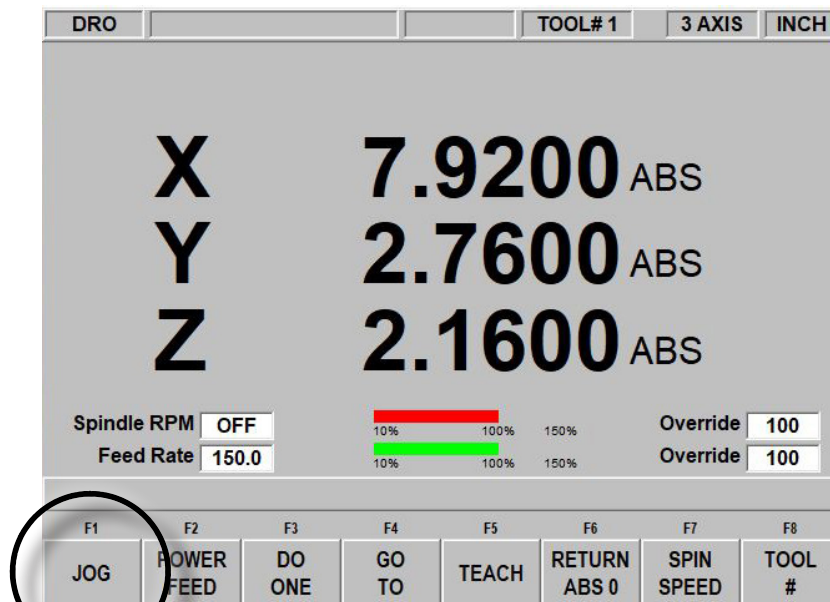


4. Make sure that the tool that is in the spindle matches the tool that is selected on the Prototrak screen. To change your tool:

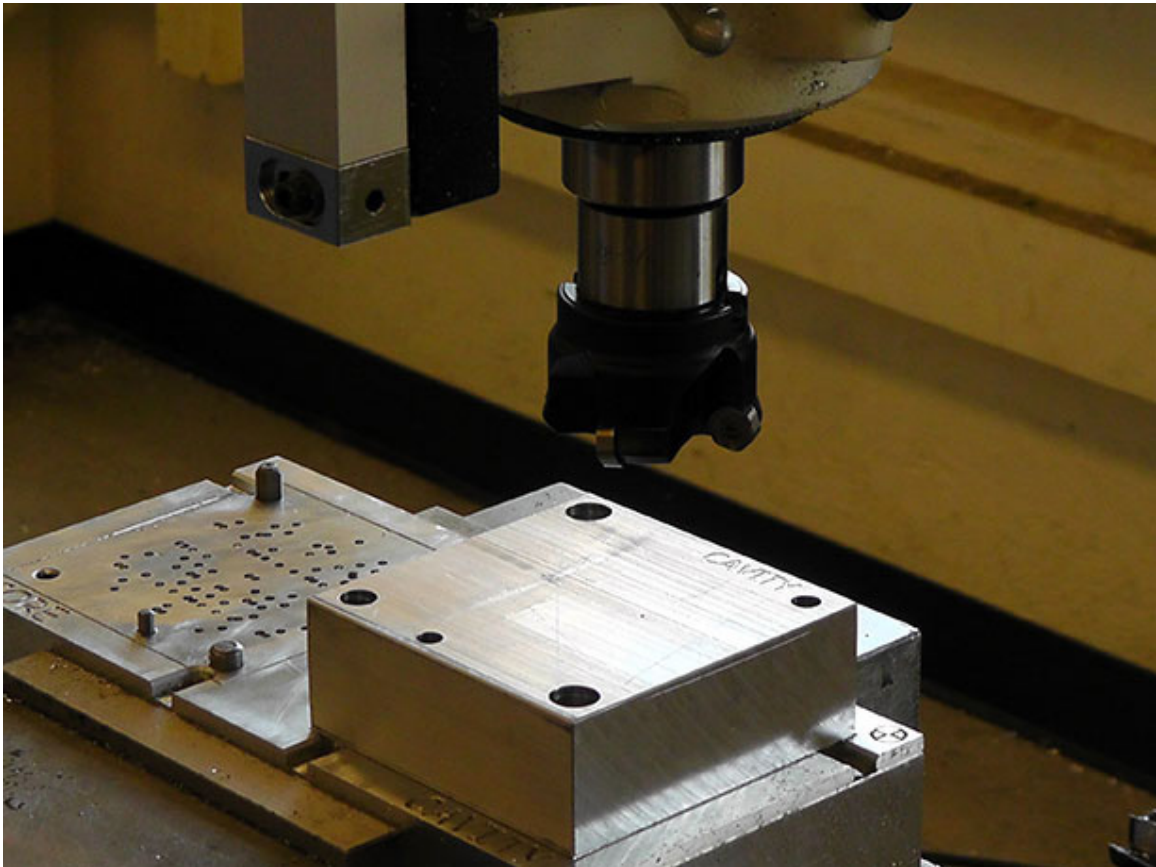
- a. Press TOOL # Softkey
- b. Enter the correct tool number
- c. Press ABS SET button on the pendant controller



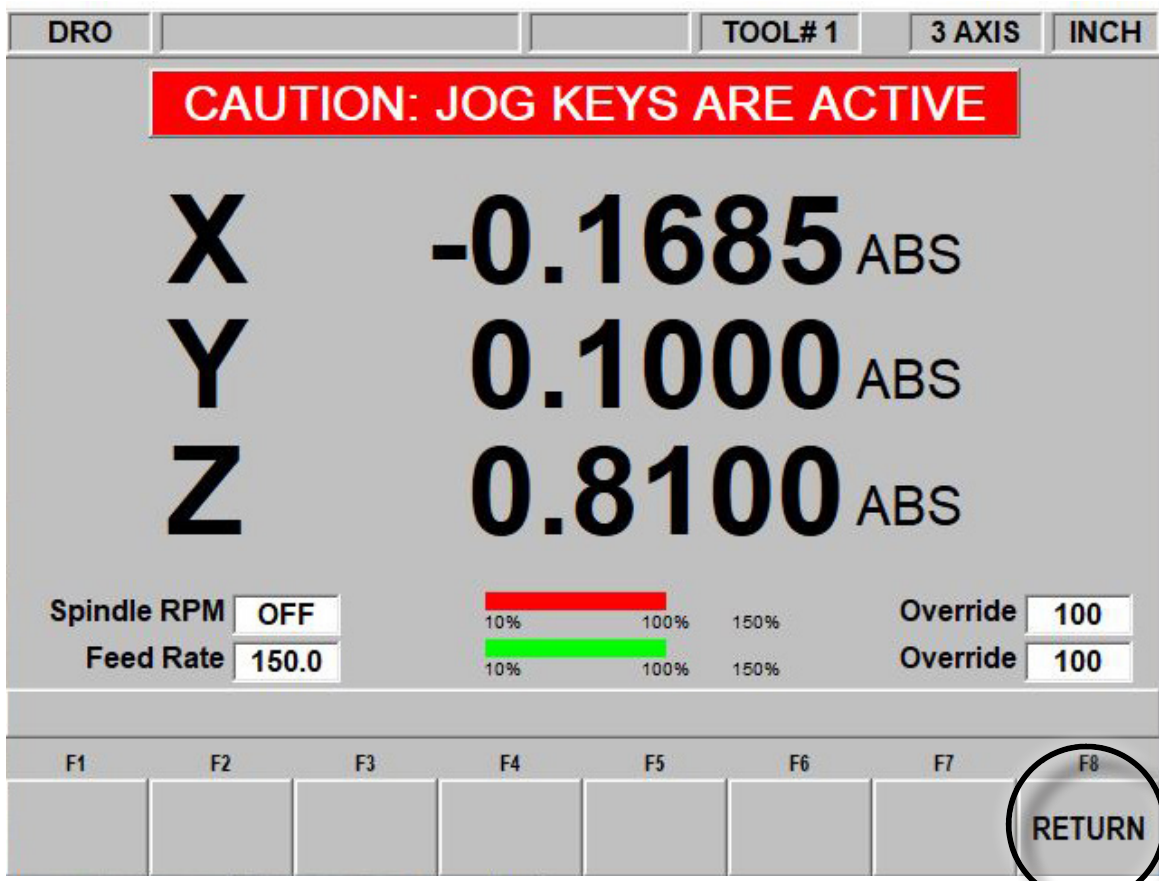
5. Press JOG Softkey



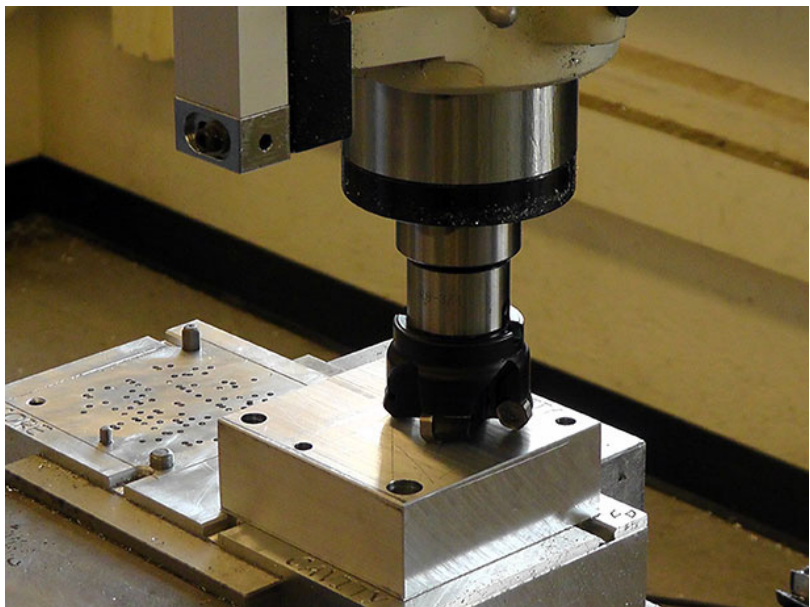
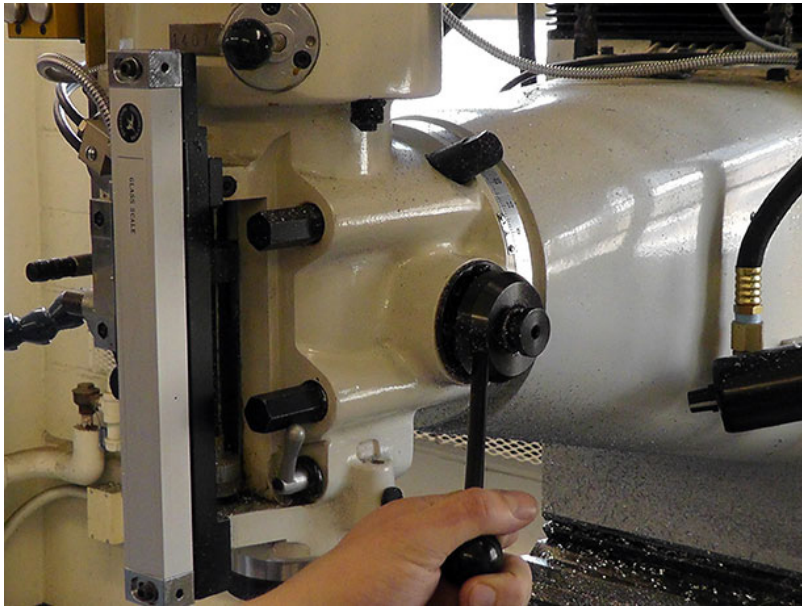
6. Jog the spindle until the tool is directly above your material by a few inches but not touching



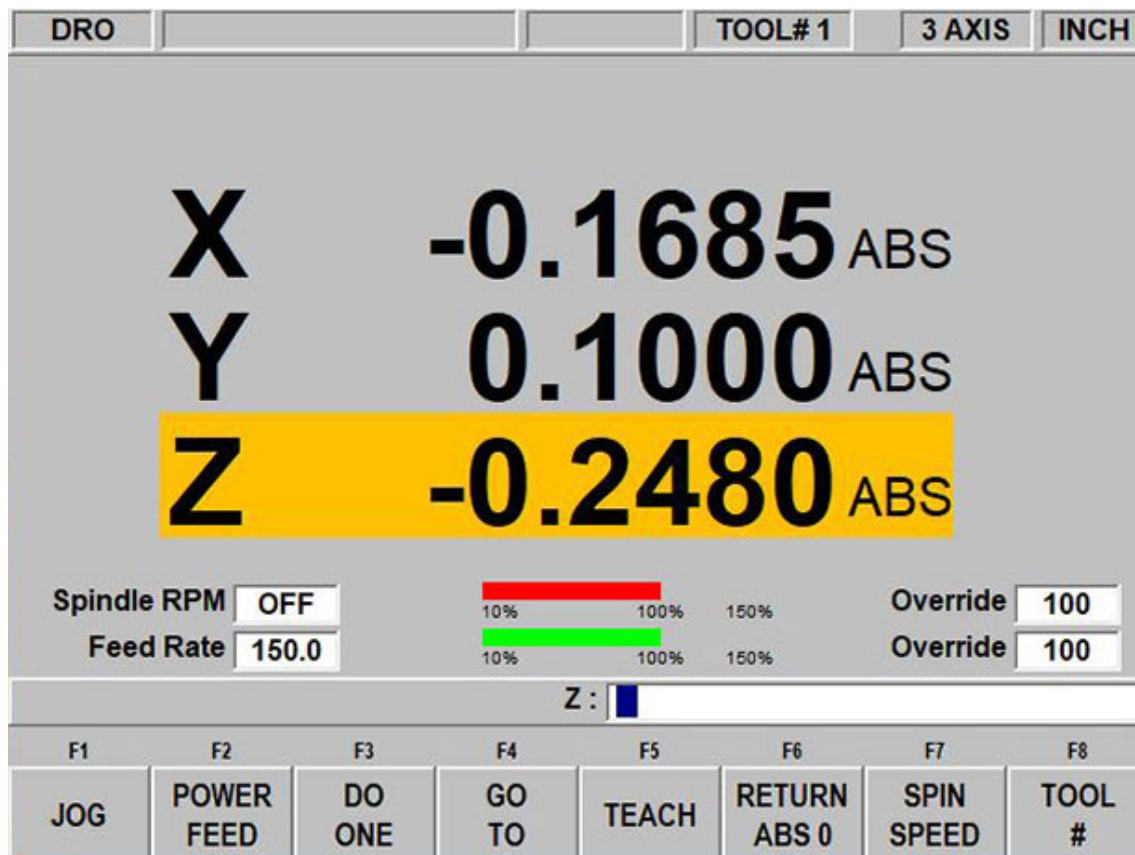
7. On the control pendant, exit the JOG mode and return to the DRO mode by pressing the RETURN Softkey



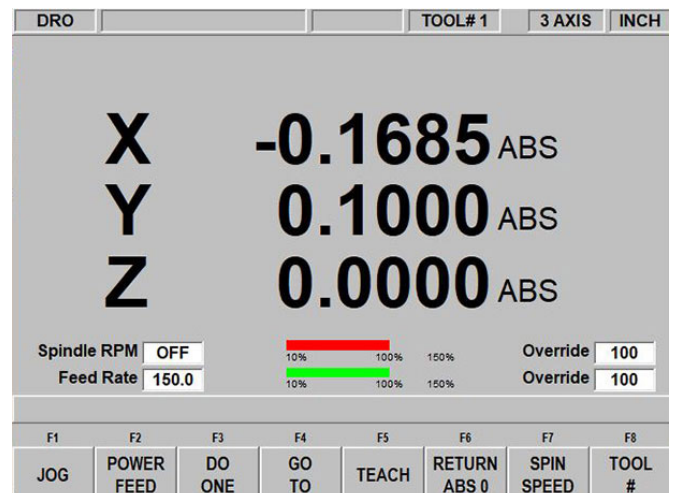
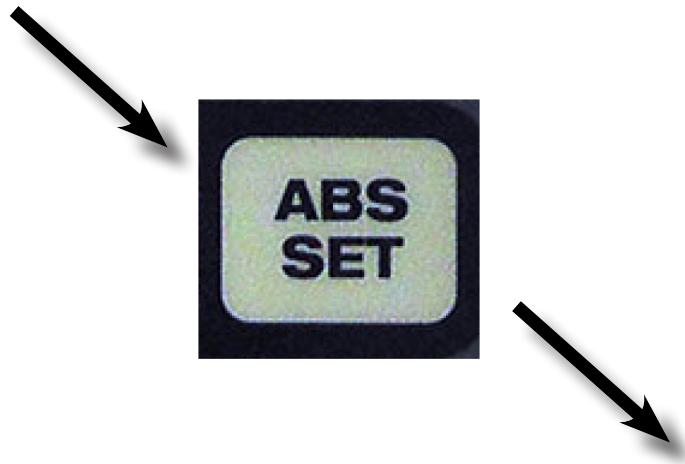
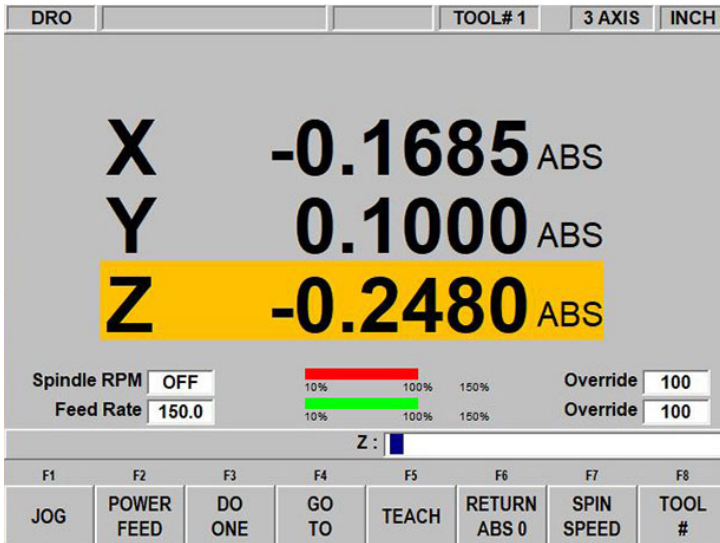
8. Unlock the quill and lower the tool with the handle until it is touching the surface of the material



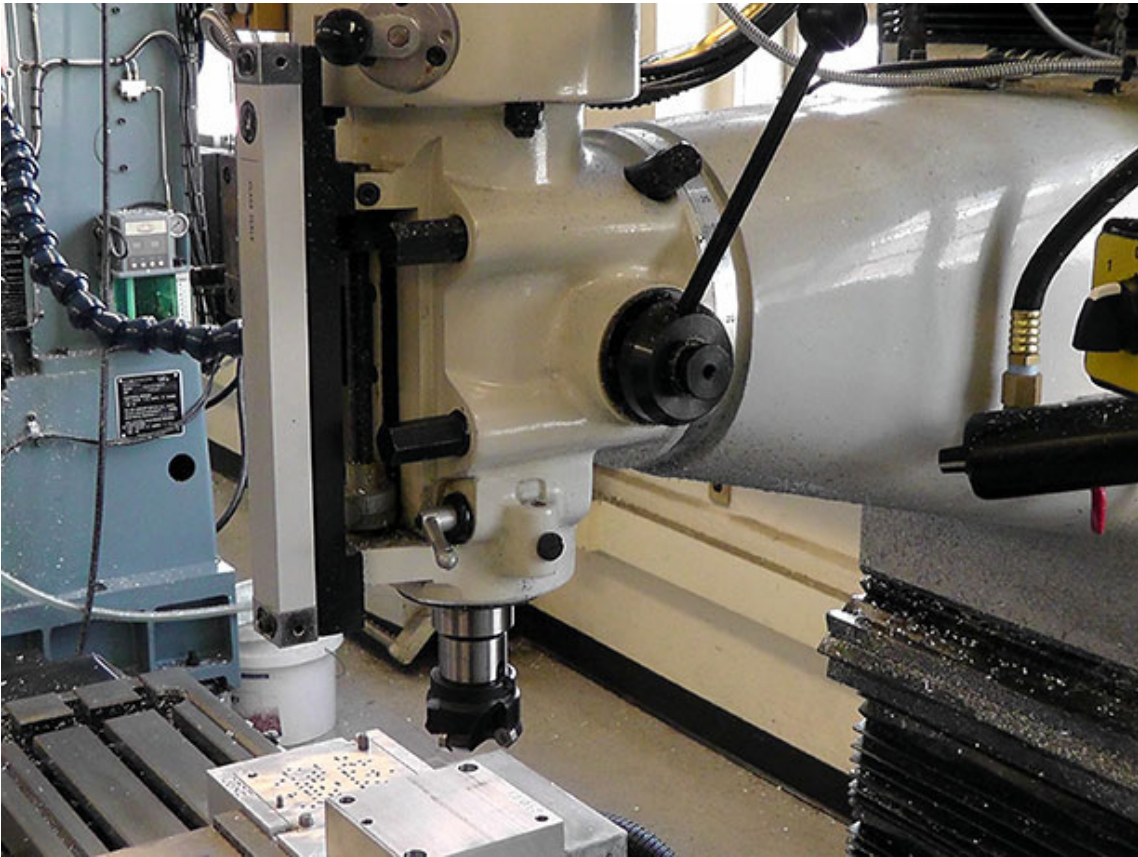
9. Hold the tool on the material surface and then select the Z Axis on the pendant controller. (The Z axis read-out will be highlighted)



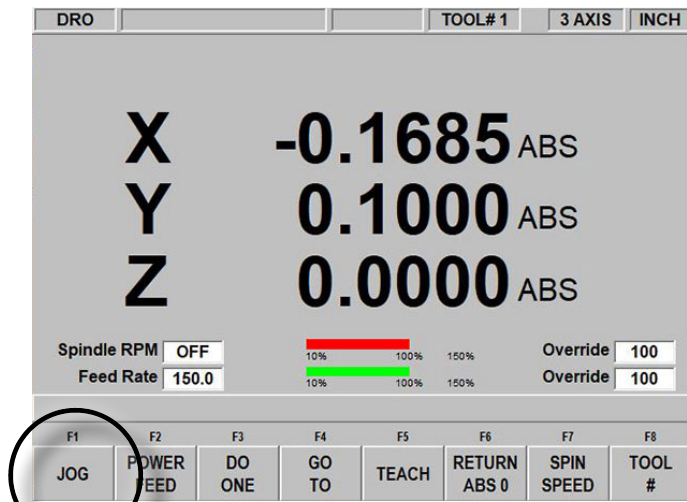
10. While the tool is still touching the material surface and the Z axis read-out is highlighted, press the ABS SET button on the pendant controller. The read-out on the screen will now read 0.000”



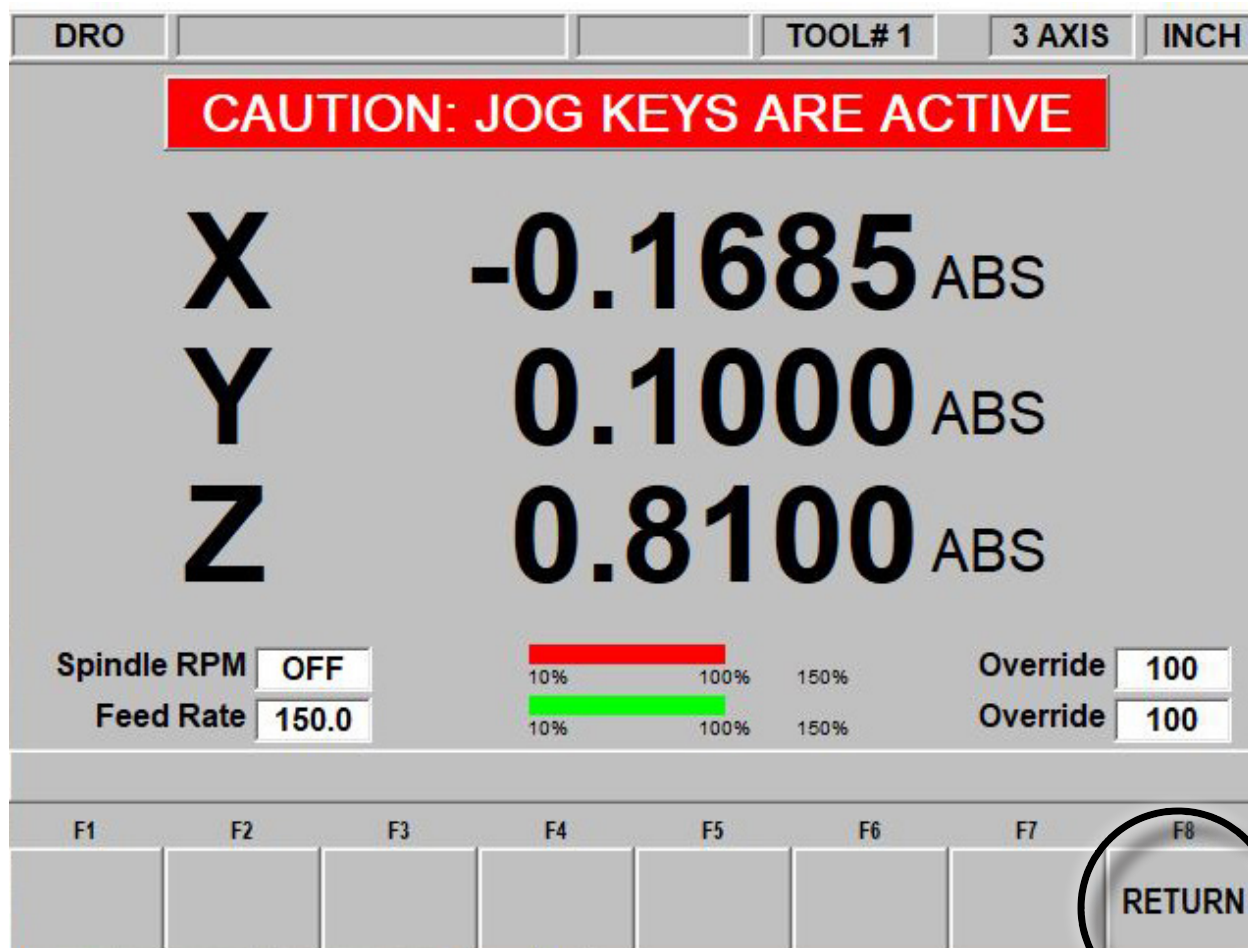
11. Raise the quill using the handle and lock the quill



12. Press the JOG Softkey and then press the Z button on the pendant controller to retract the tool to a safe distance.



13. When the spindle reaches a safe height, press the RETURN Softkey to exit JOG mode

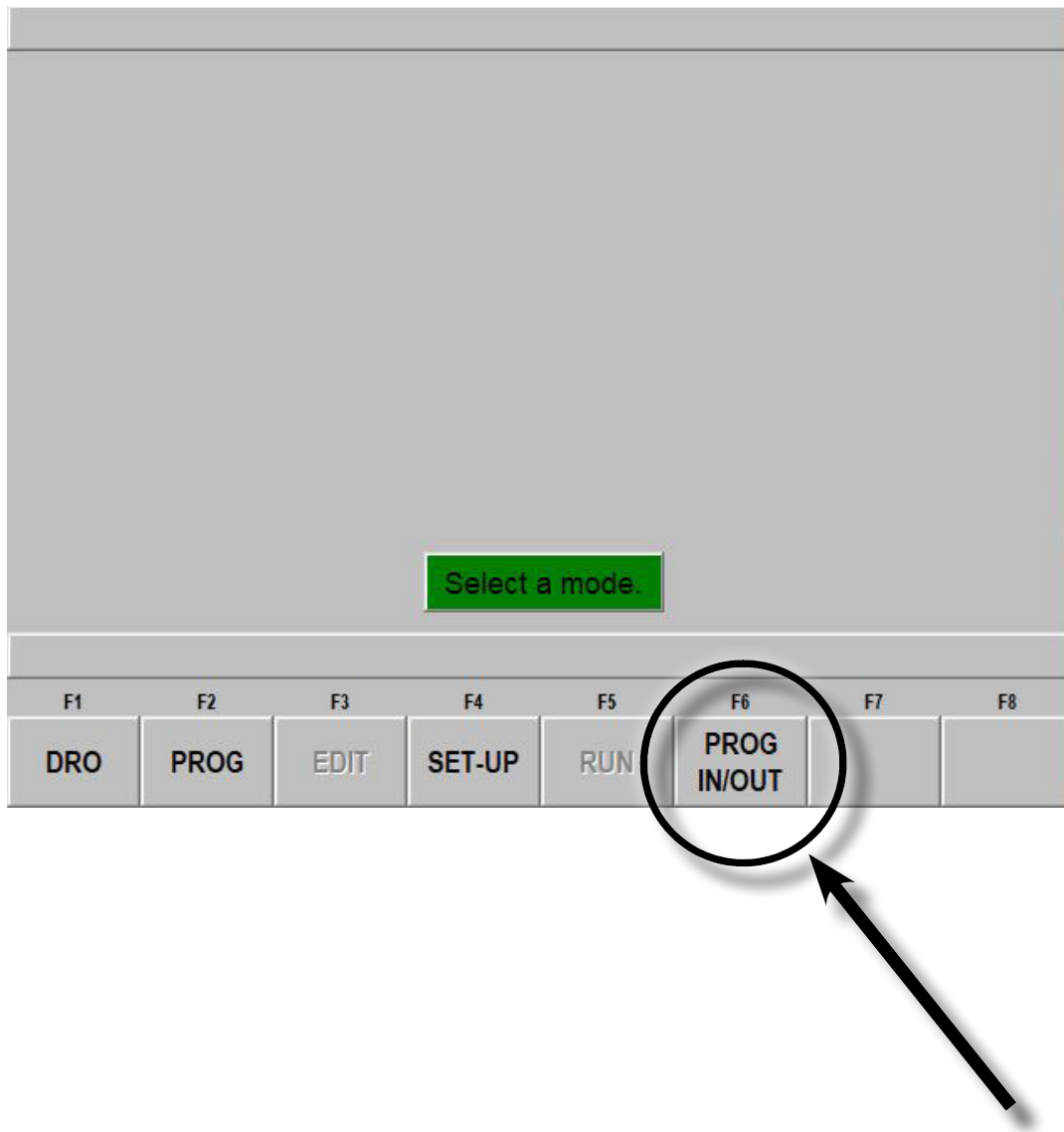


BASIC OPERATION

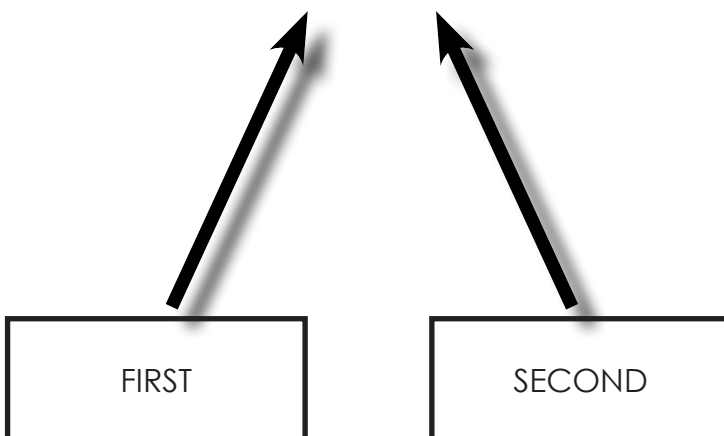
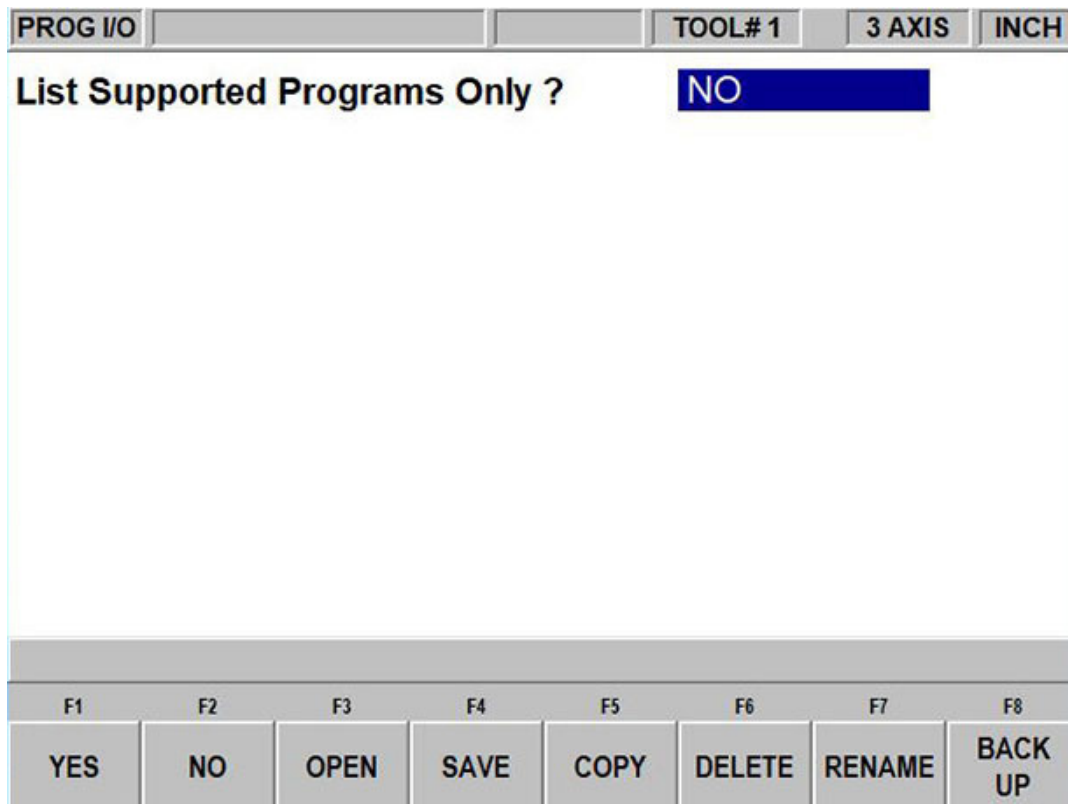
1. Press the MODE button on the pendant controller



2. Using the softkeys, press the “PROG IN/OUT” button

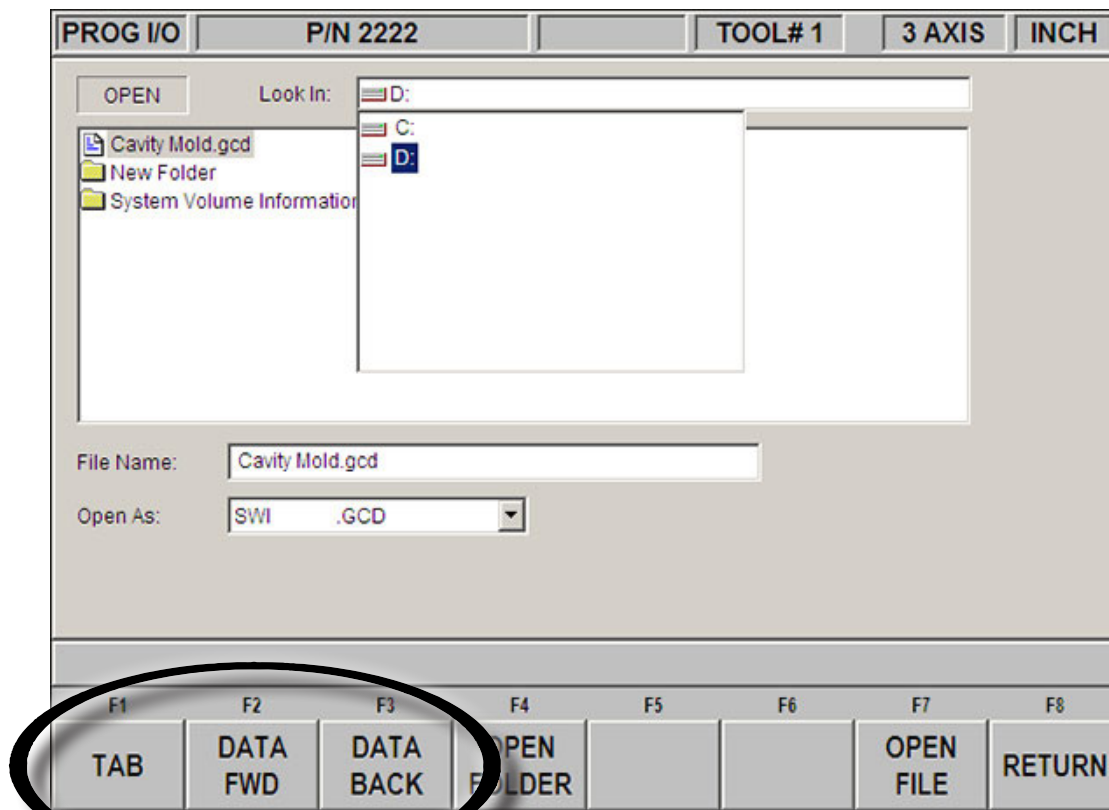


3. Press the Softkey NO, then OPEN

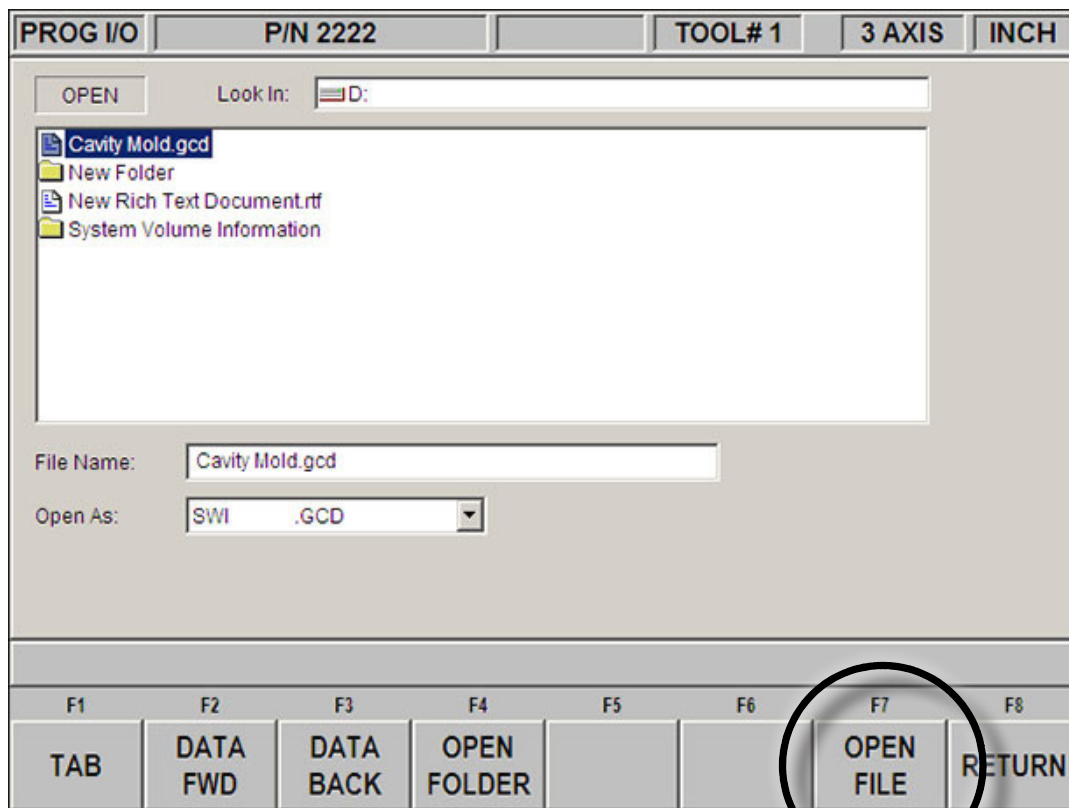


4. Locate your file using the Softkeys (TAB, DATA FWD, DATA BACK, etc.)

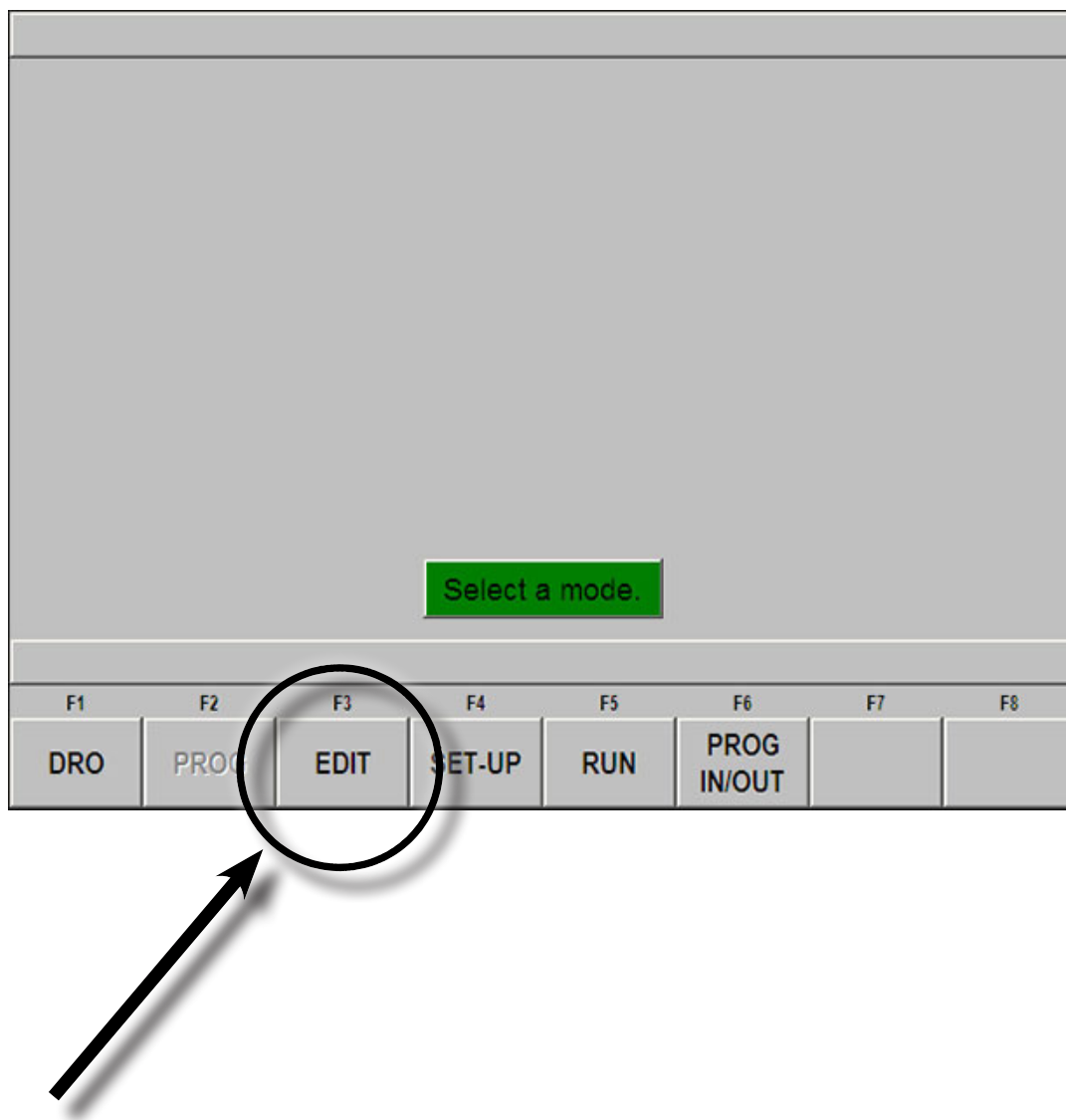
a. You may need to open a specific drive in the area labeled "Look In"



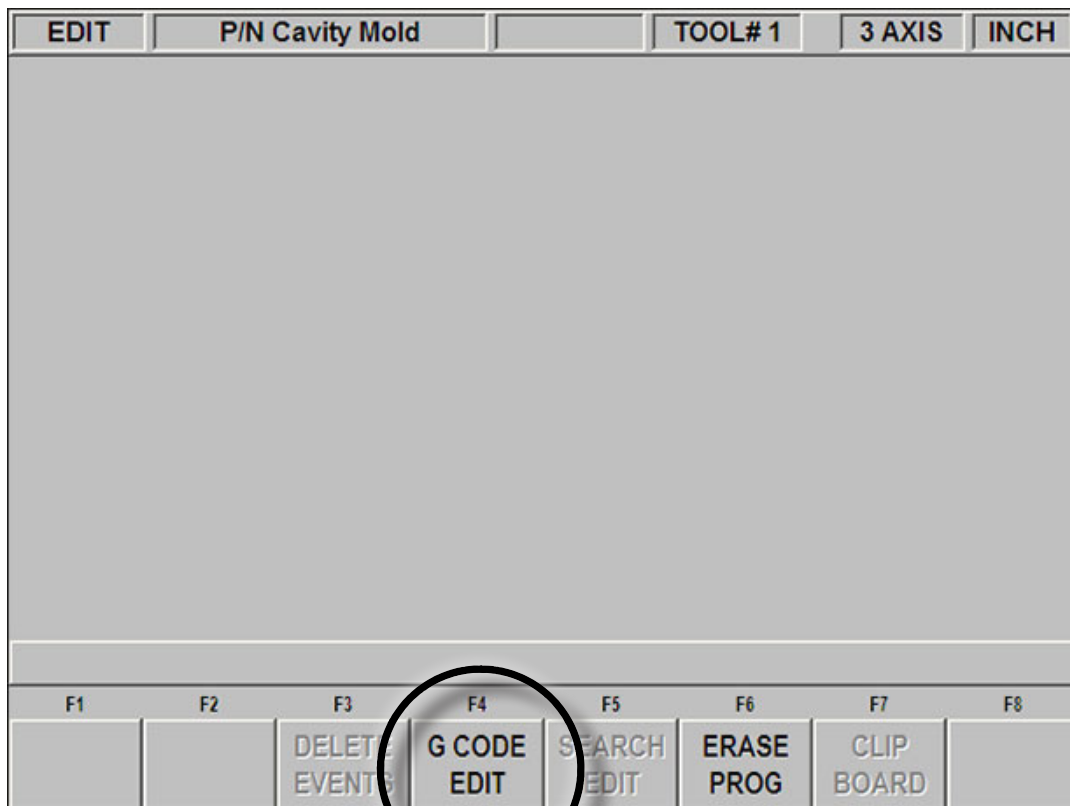
5. When you have located your file, press the OPEN FILE Softkey



6. Take a moment to confirm that you loaded the correct program. Press the EDIT Softkey



7. Press G CODE EDIT Softkey



8. You will see the G Code that has just been loaded. Make sure this is correct (Does your comment from Fusion look right? Are the tools the ones you're using? Does the first operation in the G Code match the first operation you assigned in Fusion? Etc, etc.)

If everything looks good, press the RETURN Softkey

The screenshot shows a G Code Editor window with the following content:

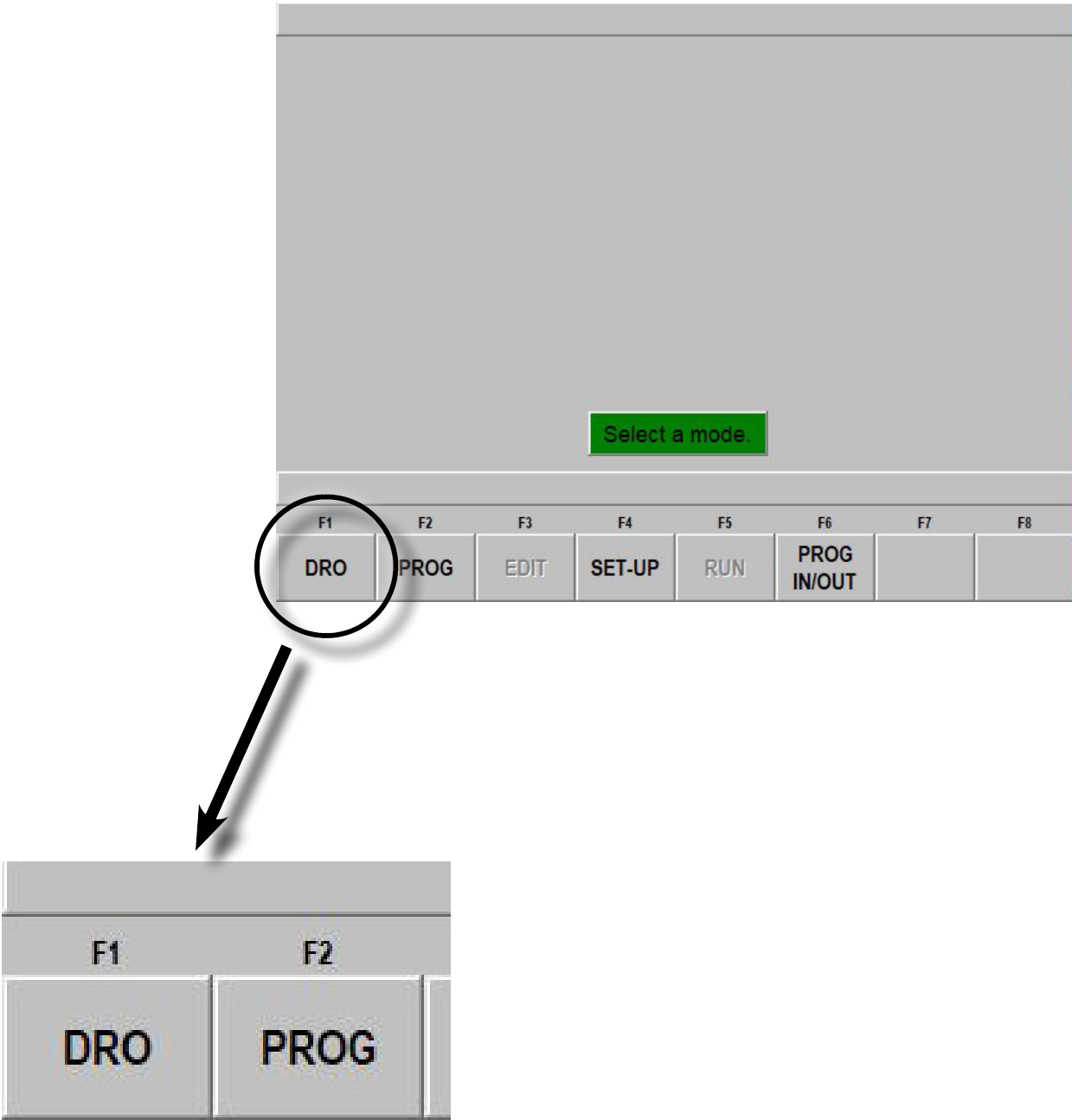
```
EDIT P/N Cavity Mold TOOL# 1 3 AXIS INCH
G Code Editor
(1001)
[CAVITY]
(T04 D=0.125 CR=0. - ZMIN=-0.5906 - FLAT END MILL)
(T09 D=0.0625 CR=0.0312 - ZMIN=-0.5059 - BALL END MILL)
(T10 D=0.125 CR=0.0625 - ZMIN=-0.6679 - BALL END MILL)
(T11 D=0.25 CR=0.125 - ZMIN=-0.5906 - BALL END MILL)
(T14 D=0.0625 CR=0. TAPER=5DEG - ZMIN=-0.49 - TAPERED MILL)
(T18 D=2. CR=0. - ZMIN=-0.01 - FACE MILL)
N0010 G90 G94 G17
N0015 G20
[FACE2]
N0020 M05
N0025 T18 M06
N0030 T11
N0035 S5000 M03
N0040 G56
N0045 G00 X3.3 Y-.6437
N0050 Z0.6
N0055 G00 Z0.2
N0060 G01 Z0.19 F30.
```

At the bottom of the window, there is a control panel with function keys F1 through F8. The F1 key is labeled 'SEARCH' and the F8 key is labeled 'RETURN'. A black box highlights the first few lines of the G code, and an arrow points from a box containing the text 'CHECK YOUR G CODE!!!' to this highlighted area.

9. If your G Code is correct, press the MODE button on the pendant controller

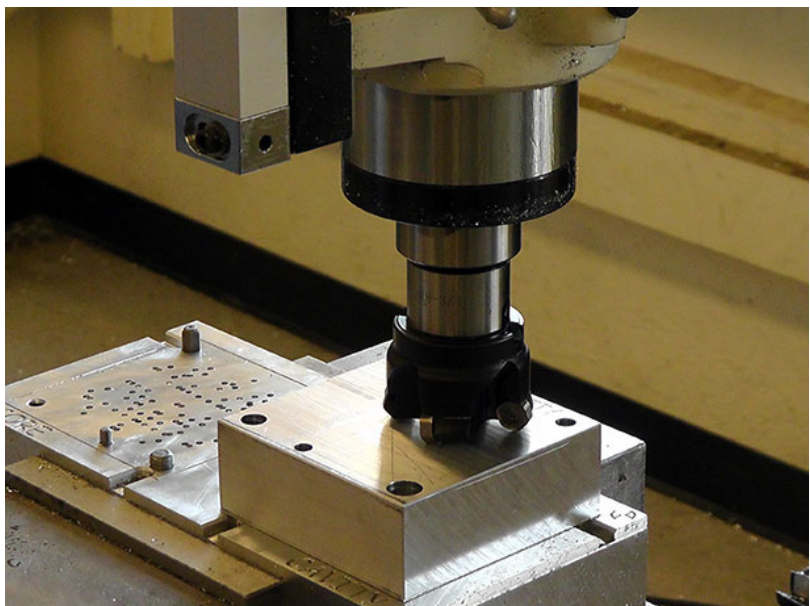
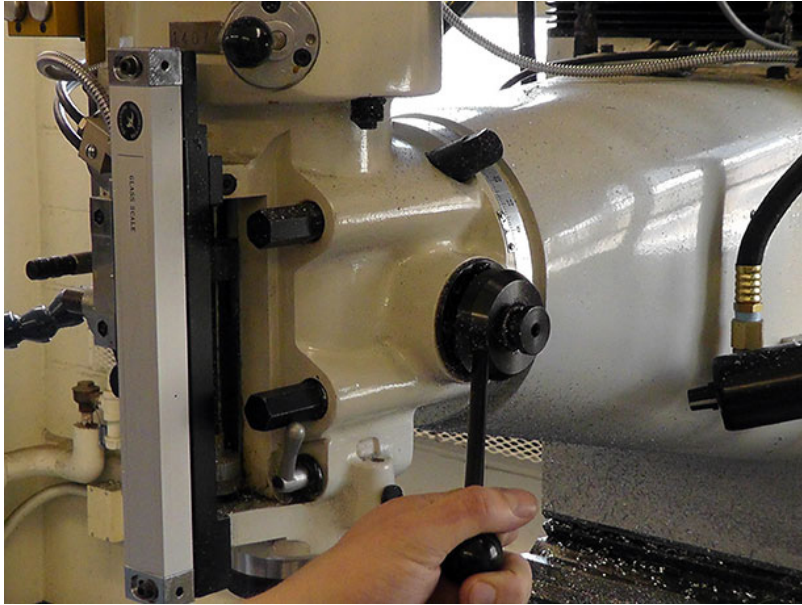


10. Press Softkey DRO



11. Touch off the tool to the top surface of the material
(See: **SETTING Z-AXIS** section)

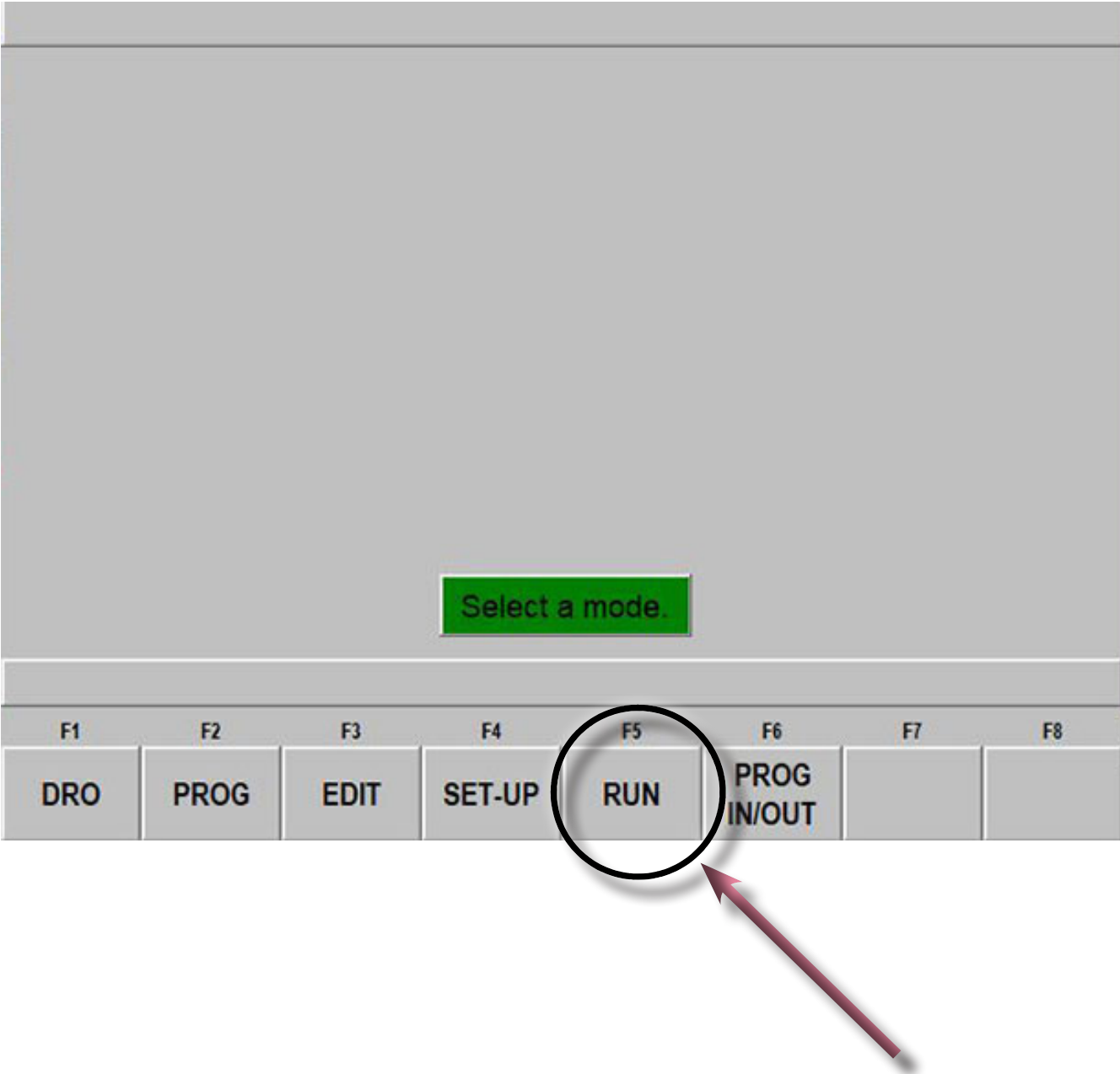
a. You may need to move the Prototrak using both manual (handwheels) and auto (Jog buttons)



12. After you have set your Z reference, press the MODE button on the pendant controller





13. Press RUN Softkey

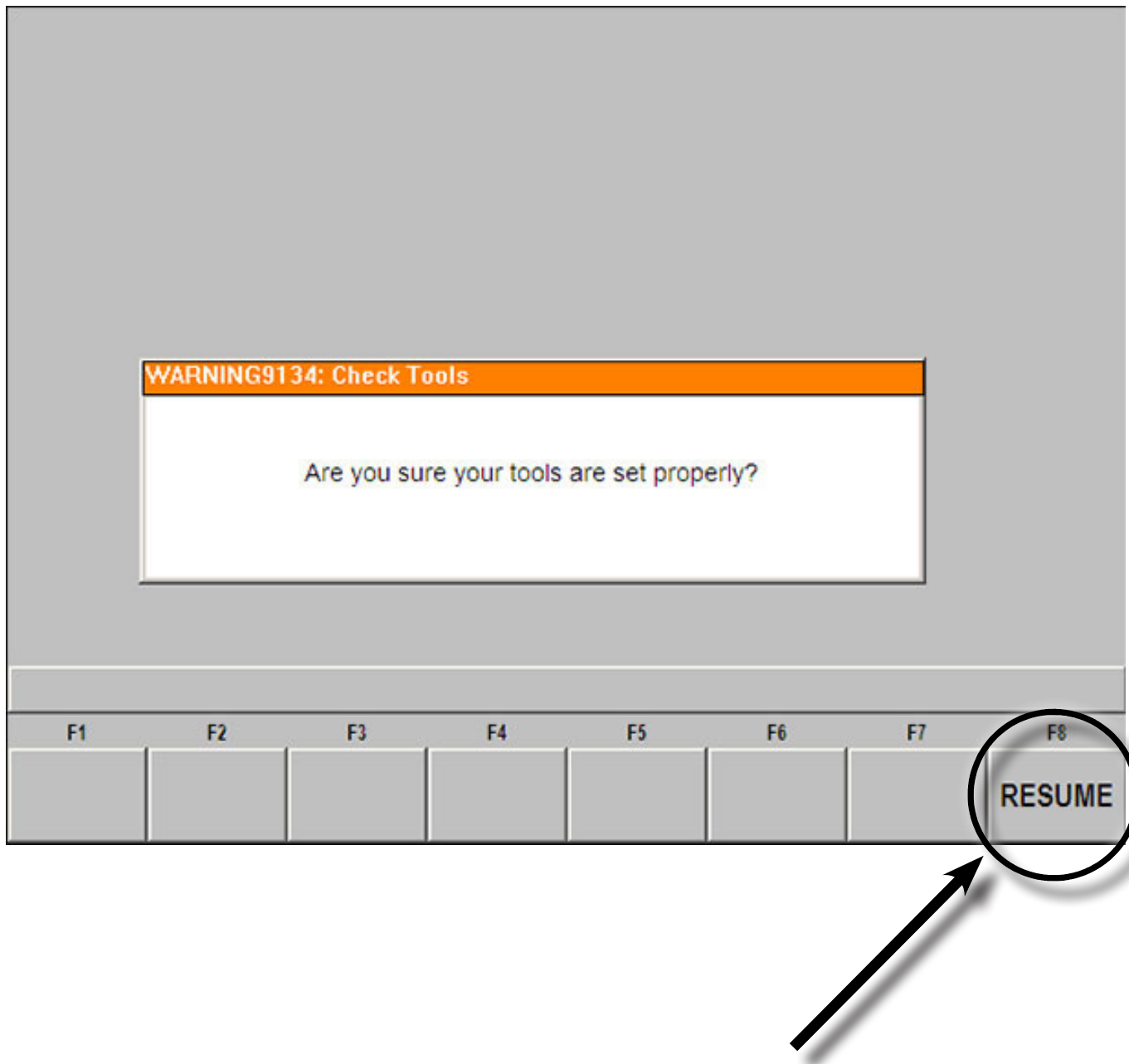


14. Press START Softkey

The screenshot displays a CNC control interface with the following elements:

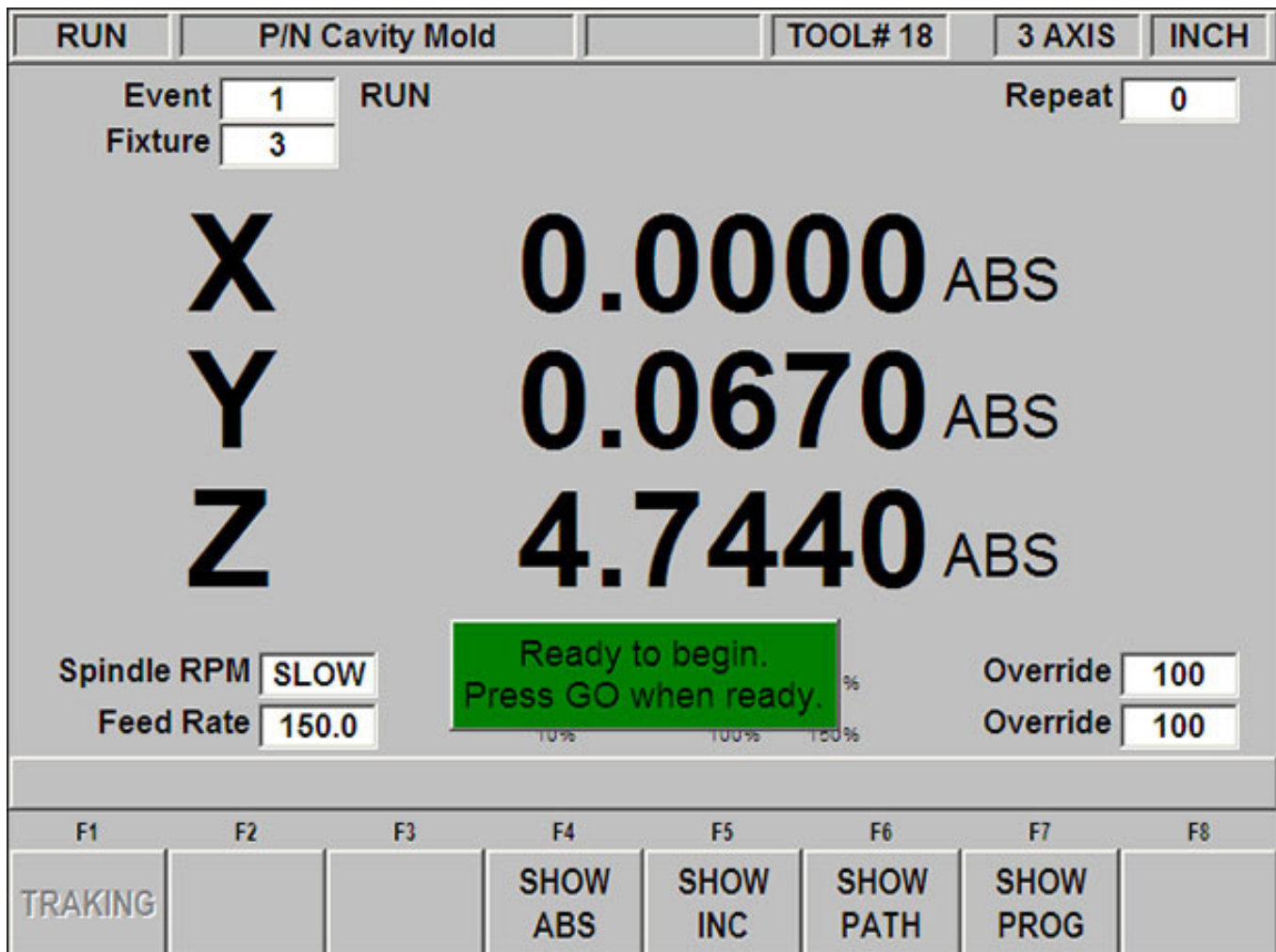
- Top Bar:** RUN | P/N Cavity Mold | TOOL# 1 | 3 AXIS | INCH
- Event/Fixture:** Event | Fixture | RUN | Repeat
- Axis Positions:**
 - X: 0.0000 ABS
 - Y: 0.0670 ABS
 - Z: 4.6360 ABS
- Spindle RPM:** |  | 10% | 100% | 150% | Override
- Feed Rate:** |  | 10% | 100% | 150% | Override
- Function Keys (F1-F8):**
 - F1: START (circled with a black circle and pointed to by a red arrow)
 - F2: START EVENT #
 - F3: TRIAL RUN
 - F4: (empty)
 - F5: (empty)
 - F6: (empty)
 - F7: (empty)
 - F8: (empty)

15. You will get a warning message asking if tools are set properly. If this is true, press RESUME Softkey



16. You will see a small green pop-up window saying "Ready to Begin, Press GO when ready"

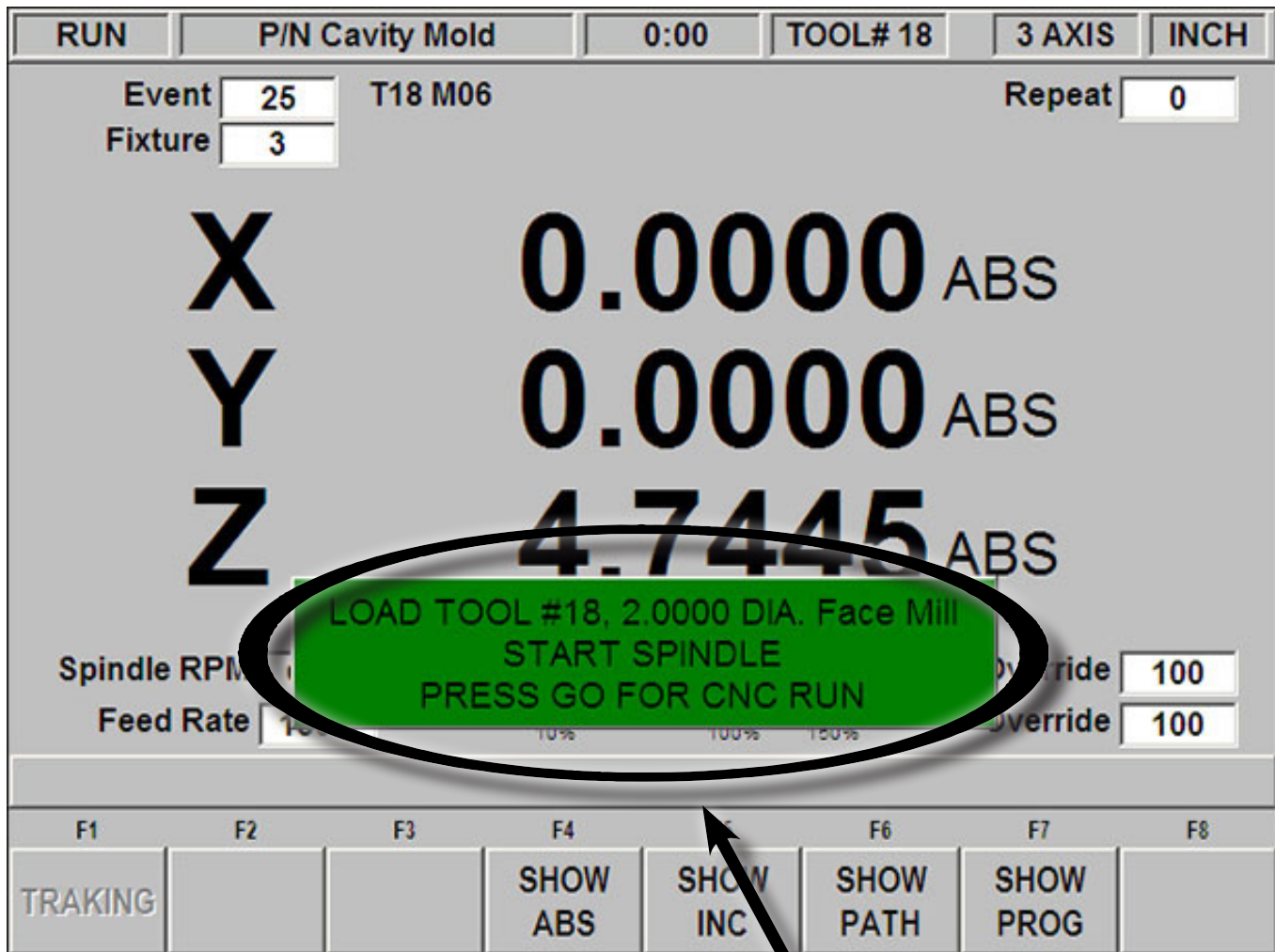
a. NOTE: You will still have time to change/adjust things before the Prototrak begins milling



17. When you are ready, use the hand-clicker to proceed by pressing the button



18. After pressing the button, a new green pop-up window will appear with important information and directions. Pay close attention to what tool number it calls for and follow the directions!

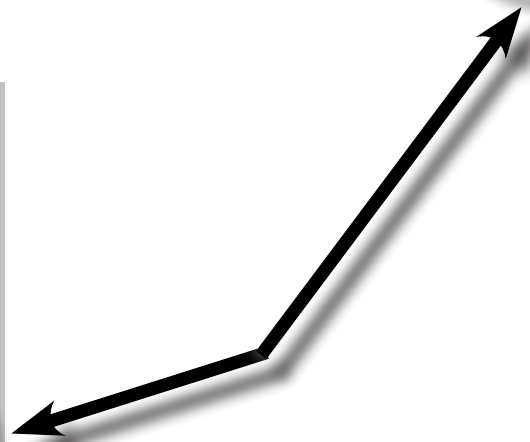


19. Turn on the spindle by turning the switch to the 2 position

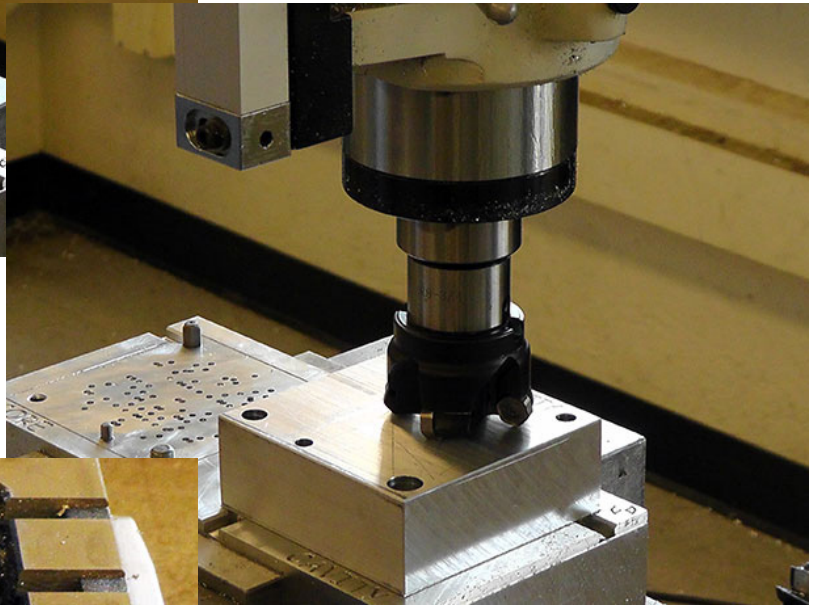
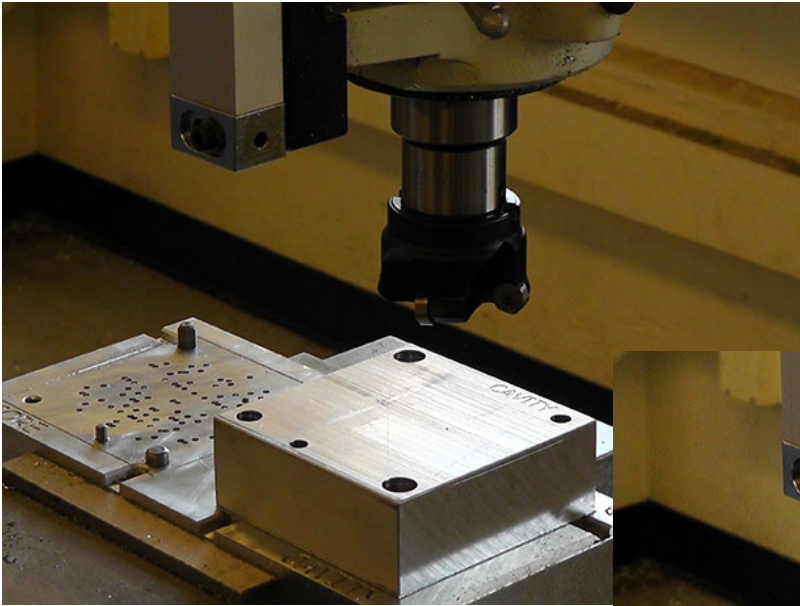


20. It is HIGHLY recommended that if this is your first time running your code that you manually lower the federate!!!! You can do this by pressing the ↓ button on the controller pendant. (30% is a good speed to start with)

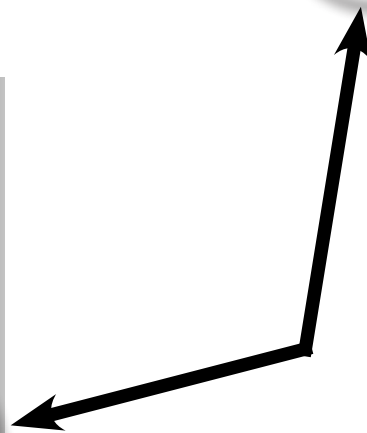
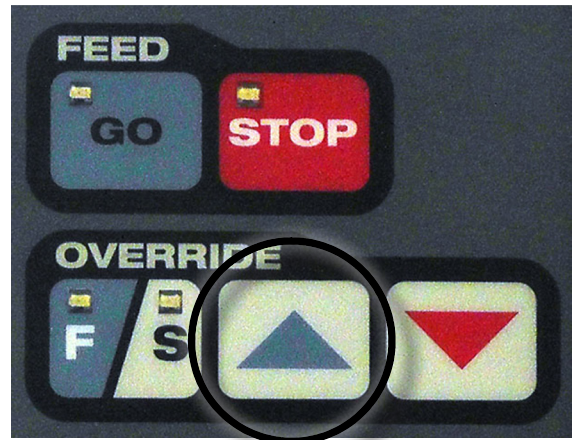
a. This will slow down the Prototrak and allow you more reaction time to stop the machine in case something is wrong



21. If you have confirmed that everything is correct and are ready to begin milling, press the clicker button. Your job will start. Once the Prototrak begins, pay extra close attention to where it is moving to!!! If it appears to be moving too fast, or to an incorrect location, press the clicker button to pause the machine. If necessary, turn off the spindle



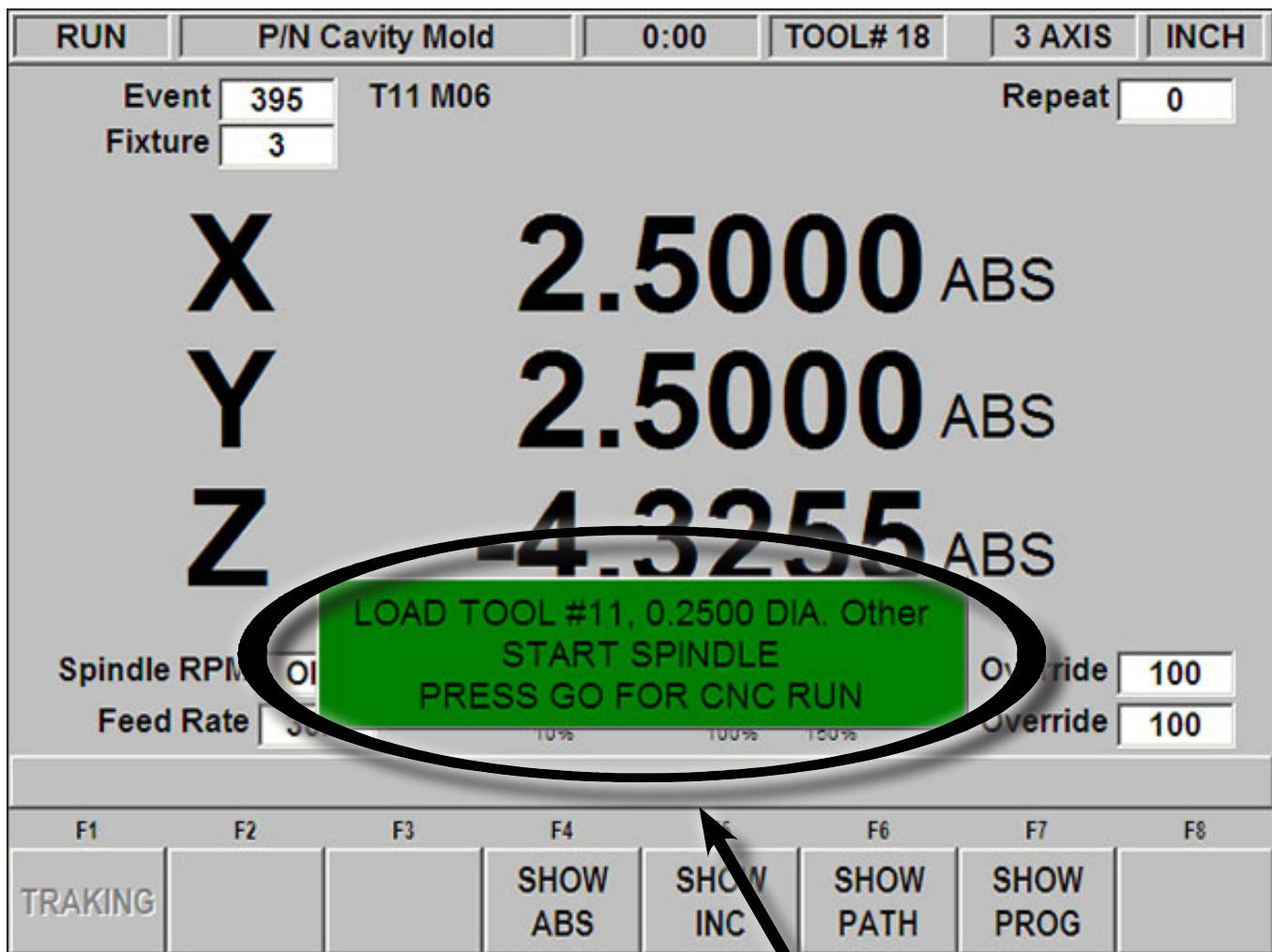
22. If you can observe that the Prototrak is milling your part correctly, you can then return the federate to 100% in most cases (See: **TIPS, TRICKS, AND BEST PRACTICES**)



23. As the Prototrak mills the part, be sure to always keep the clicker in your hand!!

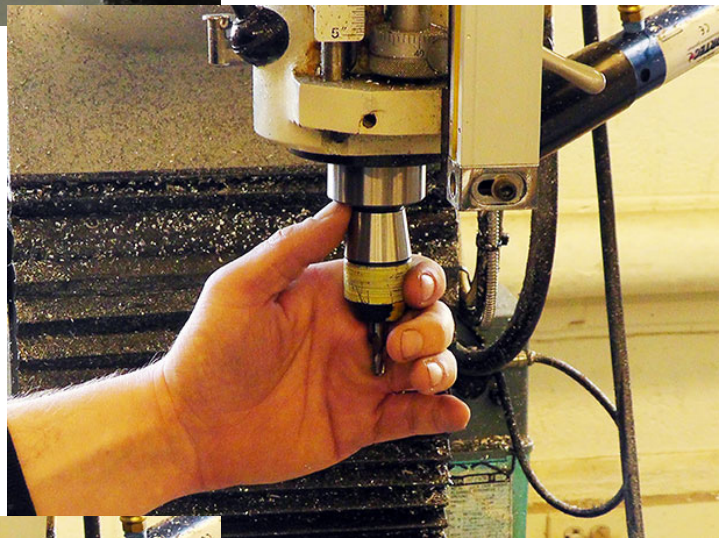


24. After the toolpath has completed, the spindle will automatically retract and the spindle will stop, followed by the green pop-up window telling you to change tools and re-start the spindle if available. (If it is the last toolpath, the pop-up window will simply say "Run Over")



NOTE THE NEW
TOOL TO BE USED!

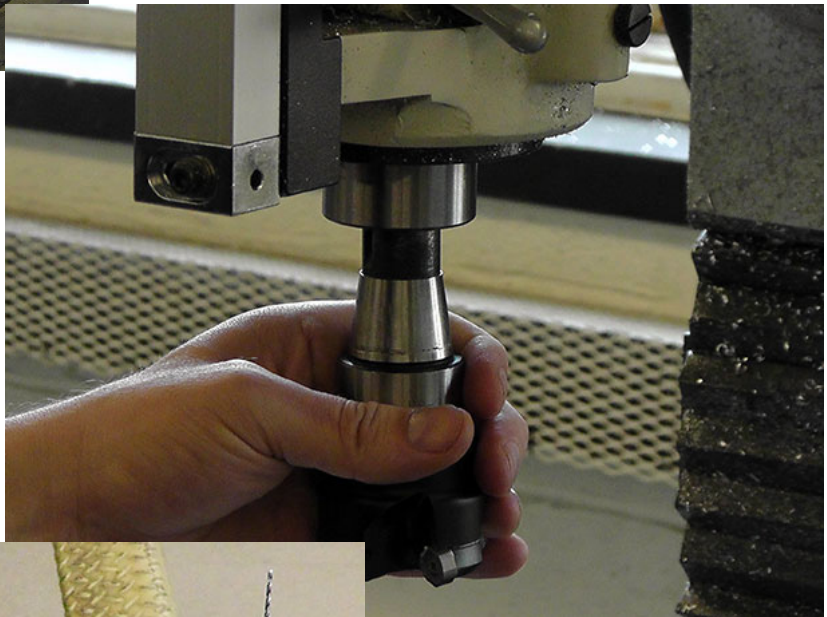
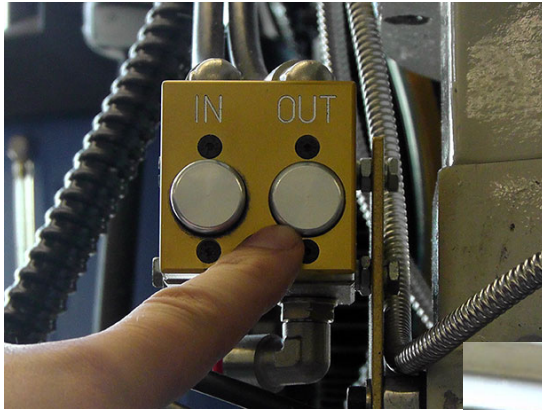
25. If there are more toolpaths to run, remove the finished tool and install the correct tool



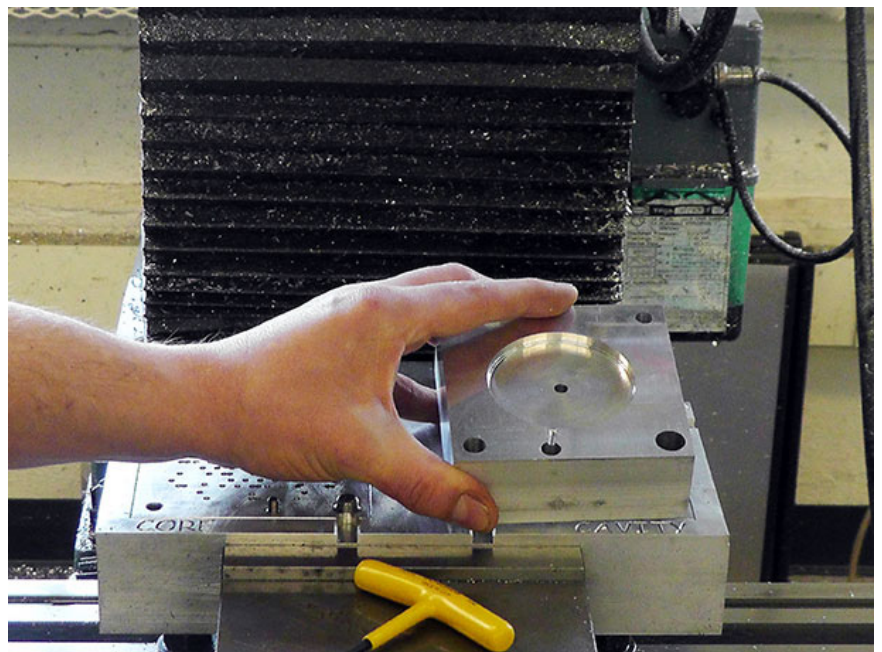
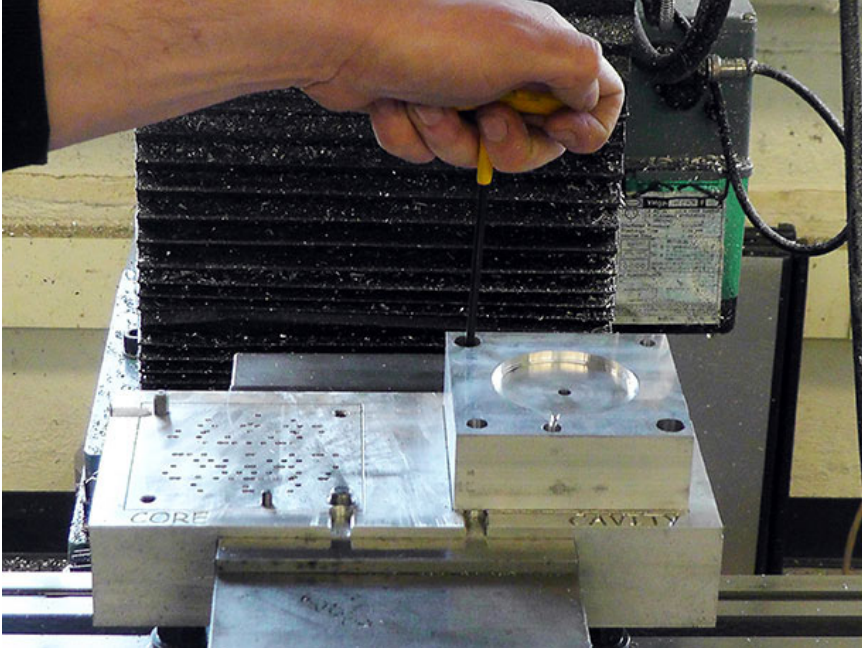
26. Turn on the spindle by turning the switch to 0, then back to 2 position (REMEMBER TO MANUALLY TURN DOWN THE FEED-RATE IF NEEDED!!!)



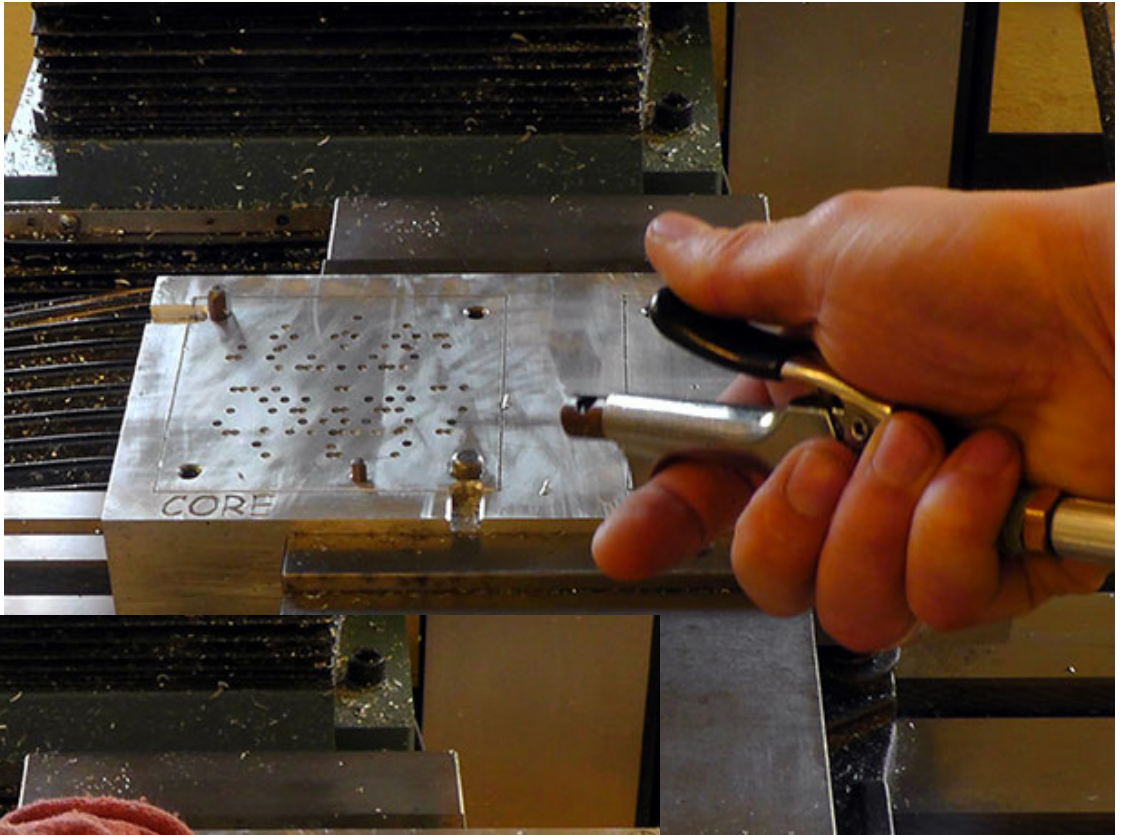
27. After your job is totally completed, remove the tool from the spindle and return it to the tool rack on the bench



28. Remove your part



29. Clean the Prototrak, the bench, and the surrounding area



TIPS, TRICKS, AND BEST PRACTICES

- When starting any operation for the first time, always manually lower the federate on the Prototrak for the first few moments. This slows the machine and allows you more time to react in case there is an issue such as incorrect WCS assignment, wrong Z height, incorrect tool, etc.
- When using a center drill, it is not necessary to drill very deeply. The purpose of the center drill is to create a recess to be used by another drill that prevents the second drill from “traveling”.
- While performing a “deep drilling” operation, it is ok to manually lower the federate on the Prototrak (~30%), but it is very important to raise the federate back up to 100% as soon as you know that the operation is correct. If you leave the federate lower than the intended federate, the drill will rub on the material instead of cutting, which can easily break the tool.
- When performing a “deep drilling” operation, be sure to use both the Fog-Buster spray coolant as well as a brush with oil. Using the brush not only helps with applying coolant, but also removes chips that might be stuck in the drill’s flutes.
- When using a very small tool, such as a 1/16” endmill, it is often most advantageous to manually lower the federate to ~30% for the entire duration of the toolpath operation. While this will make the operation take longer, it will save time in the long run because the risk of breaking the tool will be far lower as well.
- Going too slow or too fast with your feeds and speeds is dangerous and can easily destroy both the tool you are using and the part you are milling. If you aren’t sure, ask one of the instructor’s opinions!
- Try to strategize the order of your toolpaths. If you have two or more different operations that use the same tool, try to organize them sequentially in Fusion 360 because the Prototrak will not require a tool change if sequential operations use the same tool.

TOOL LIBRARY

Select Tool

Operation

Type

Dimensions

Search

Tool Library **LMP Voyo**

Name	Cutting diameter	Corner radius	Overall length	Flute length	Shaft diameter	Surface speed	Cutting feed rate	Spindle speed	feedrate per rev	Description
R8 Face Mill Holder										R8 Face Mill Holder
R8 Tool Holder										R8 Tool Holder
R8 Tool Holder										R8 Tool Holder
R8 Tool Holder										R8 Tool Holder
1 - Ø1/16" - flat (End Mill SHORT)	0.062500 "	0"	1.00000 "	0.20000 "	0.18750 "	81.812 Ft/min	5.5000 inch/min	5000.0 rpm	0.0011000 inch/...	End Mill SHORT
2 - Ø1/16" - flat (End Mill LONG)	0.062500 "	0"	1.00000 "	0.20000 "	0.18750 "	81.812 Ft/min	5.5000 inch/min	5000.0 rpm	0.0011000 inch/...	End Mill LONG
3 - Ø3/32" - flat (End Mill)	0.093750 "	0"	1.00000 "	0.28000 "	0.18750 "	122.72 Ft/min	8.3000 inch/min	5000.0 rpm	0.0016600 inch/...	End Mill
4 - Ø1/8" - flat (End Mill)	0.12500 "	0"	1.5000 "	0.38000 "	0.37500 "	163.62 Ft/min	11.100 inch/min	5000.0 rpm	0.0022200 inch/...	End Mill
5 - Ø3/16" - flat (End Mill)	0.18750 "	0"	2.0000 "	0.45000 "	0.37500 "	245.44 Ft/min	16.700 inch/min	5000.0 rpm	0.0033400 inch/...	End Mill
6 - Ø1/4" - flat (End Mill)	0.25000 "	0"	2.5000 "	0.62000 "	0.37500 "	327.25 Ft/min	22.300 inch/min	5000.0 rpm	0.0044600 inch/...	End Mill
7 - Ø3/8" - flat (Endmill)	0.37500 "	0"	3.0000 "	0.75000 "	0.37500 "	490.87 Ft/min	50.000 inch/min	5000.0 rpm	0.0100000 inch/...	Endmill
8 - Ø1/16" - ball (End Mill SHORT)	0.062500 "	0.031250 "	2.0000 "	0.18750 "	0.18750 "	81.812 Ft/min	7.5000 inch/min	5000.0 rpm	0.0015000 inch/...	End Mill SHORT
9 - Ø1/16" - ball (End Mill LONG)	0.062500 "	0.031250 "	2.0000 "	0.18750 "	0.18750 "	81.812 Ft/min	7.5000 inch/min	5000.0 rpm	0.0015000 inch/...	End Mill LONG
10 - Ø1/8" - ball (End Mill)	0.12500 "	0.062500 "	1.5000 "	0.39000 "	0.37500 "	163.62 Ft/min	15.100 inch/min	5000.0 rpm	0.0030200 inch/...	End Mill
11 - Ø1/4" - ball (End Mill)	0.25000 "	0.12500 "	2.0000 "	0.54000 "	0.37500 "	327.25 Ft/min	30.300 inch/min	5000.0 rpm	0.0060600 inch/...	End Mill
12 - Ø3/8 RO.03" - bull nose (End...)	0.37500 "	0.030000 "	3.0000 "	0.75000 "	0.37500 "	490.87 Ft/min	75.000 inch/min	5000.0 rpm	0.015000 inch/...	End Mill
13 - Ø1/2 RO.125" - bull nose (En...)	0.50000 "	0.12500 "	3.0000 "	0.86000 "	0.50000 "	654.50 Ft/min	90.000 inch/min	5000.0 rpm	0.018000 inch/...	End Mill
14 - Ø1/16" 5° - tapered mill (En...)	0.062500 "	0"	2.0000 "	0.57700 "	0.37500 "	81.812 Ft/min	5.5000 inch/min	5000.0 rpm	0.0011000 inch/...	End Mill
15 - Ø3/16" 120° 60° - center drill...	0.18750 "	0"	1.5000 "	0.43000 "	0.18600 "	98.175 Ft/min	8.0000 inch/min	2000.0 rpm	0.0040000 inch/...	#2
16 - Ø0.12" 118° - drill (#31)	0.12000 "	0"	4.0000 "	1.4800 "	0.50000 "	150.00 Ft/min	19.099 inch/min	4774.6 rpm	0.0004000 inch/...	#31
17 - Ø0.234" 118° - drill (A)	0.23400 "	0"	3.0420 "	2.3400 "	0.23400 "	150.00 Ft/min	6.0000 inch/min	2448.5 rpm	0.0024504 inch/...	A
18 - Ø2" - face mill	2.0000 "	0"	1.5500 "	0.43000 "	1.9000 "	2618.0 Ft/min	60.000 inch/min	5000.0 rpm	0.012000 inch/...	

OK
Cancel

WCS LOCATION CHEAT-SHEET

