TRAK CNC

TRAK TRL 1630RX Lathe
Guided Introduction

BU College of Engineering
Engineering Product Innovation Center

Welcome to the TRAK CNC Lathe Guided Introduction! This document is intended for users with a basic understanding of how a lathe works, and is a great place to develop your skills in CNC lathe programming and machining.

All users must consult an EPIC Lab Supervisor whenever indicated in this document.

Table of Contents

INTRODUCTION

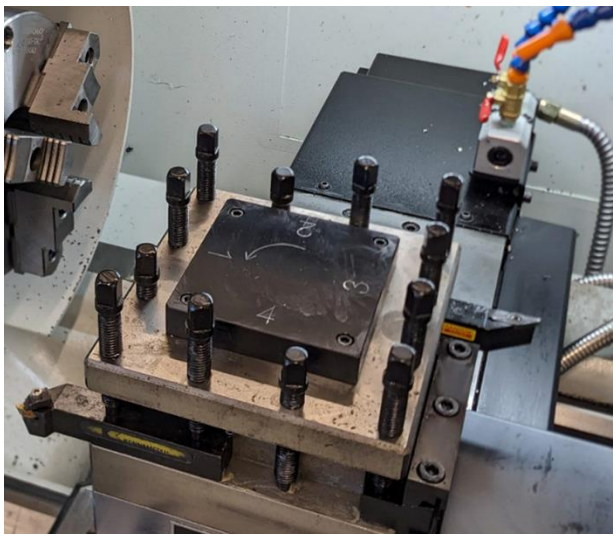
Before working through this guide, consult an EPIC Lab Supervisor regarding which TRAK CNC lathes to use:

TRAK 1 is outfitted with a four-station TOOL TURRET capable of switching tools automatically during a CNC program.

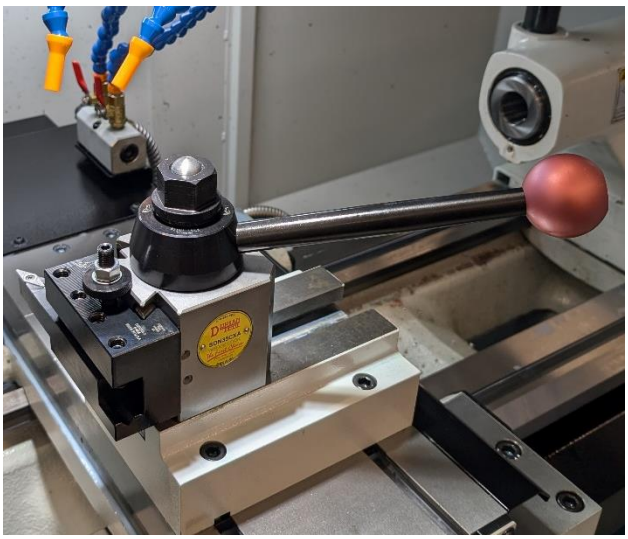
TRAK 2 is outfitted with a TOOL POST which requires the user to manually switch tools between operations.

This guide references TRAK 1 specifically, however, all machining and programming operations remain the same. If using TRAK 2, any steps that call for use of the TOOL TURRET are essentially omitted or replaced with the action of manually loading/unloading a tool.

EPIC staff will provide an overview of TOOL POST operations as needed.



TOOL TURRET



TOOL POST

A Note While Using This Guide

Try tapping the on-screen “EPA” button at any point to access the contextual help library. Some topics will automatically display based on which screen is active, while others you can be entered in the search field.

START UP PROCEDURE

Note: The term “soft key” refers to selectable icons or terms on the touchscreen, whereas “hard key” refers to the physical buttons directly outside the screen.

1. Rotate the main power switch at the rear of the machine to the ON position. The control screen will go through its boot sequence and eventually display the main control screen (Figure 1).

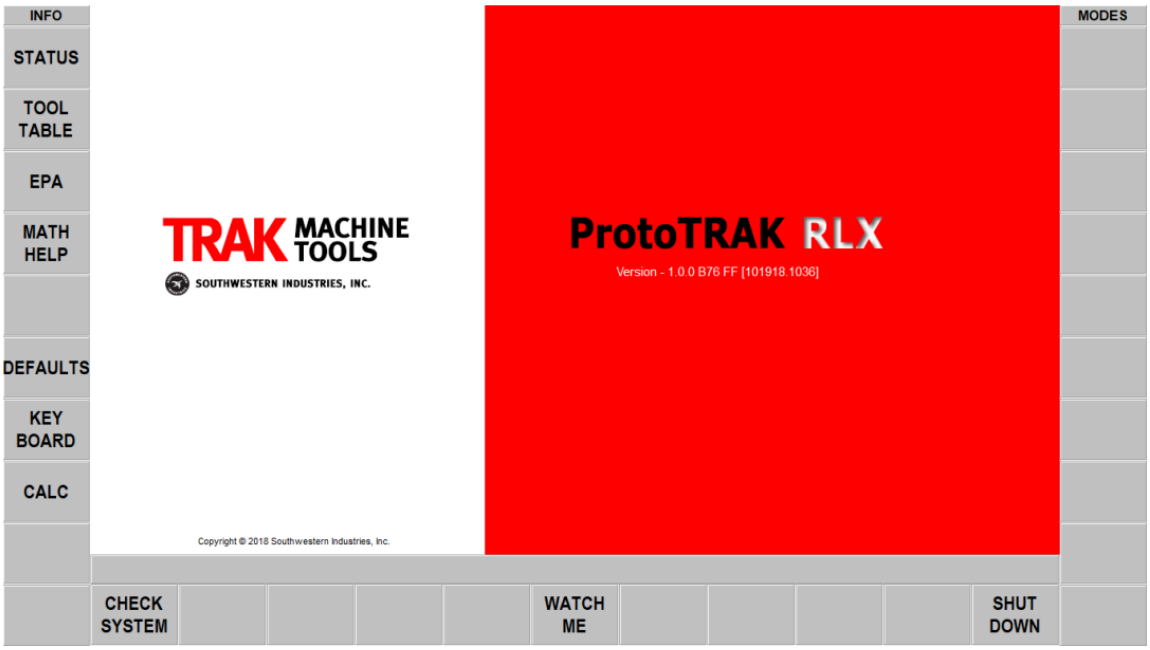


Figure 1

2. Locate two red emergency stop buttons and ensure they are disengaged.
3. Inspect the automatic oiler at the lower front side of the enclosure and ensure there is a sufficient amount of oil. The automatic oiler will turn on briefly during the startup procedure.
4. Tap the CHECK SYSTEM soft key, use the numeric hard keys to enter 2500 for the maximum spindle RPM, then tap the ABS SET hard key to set the value.
5. Press and hold RESET hard key until the white LED beside the key illuminates. This activates power to the motors and the automatic oiler.
6. Press the SET-UP and then HOME TURRET soft keys. Enter the current turret position number as it appears on top of the tool turret (Figure 2) and press ABS SET.
7. The machine is now fully powered on and ready to use.

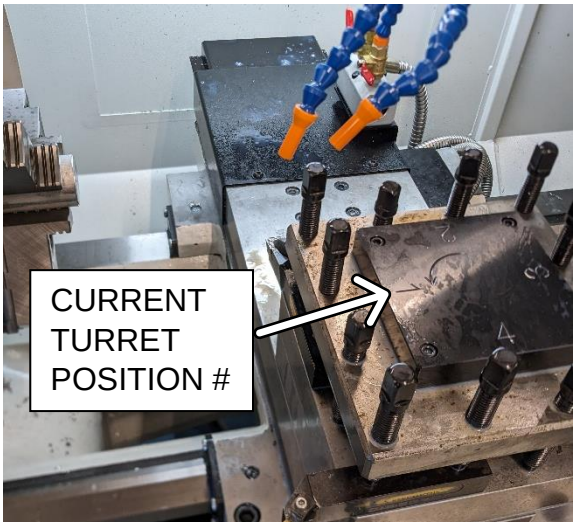


Figure 2

MANUAL OPERATION

Enter DRO mode (Figure 3) by pressing the DRO (digital read out) soft key. In this mode we can manually control the X and Z axis, set reference points, spindle RPM, and more. Manual movement of the X and Z axis can only be done in DRO mode.

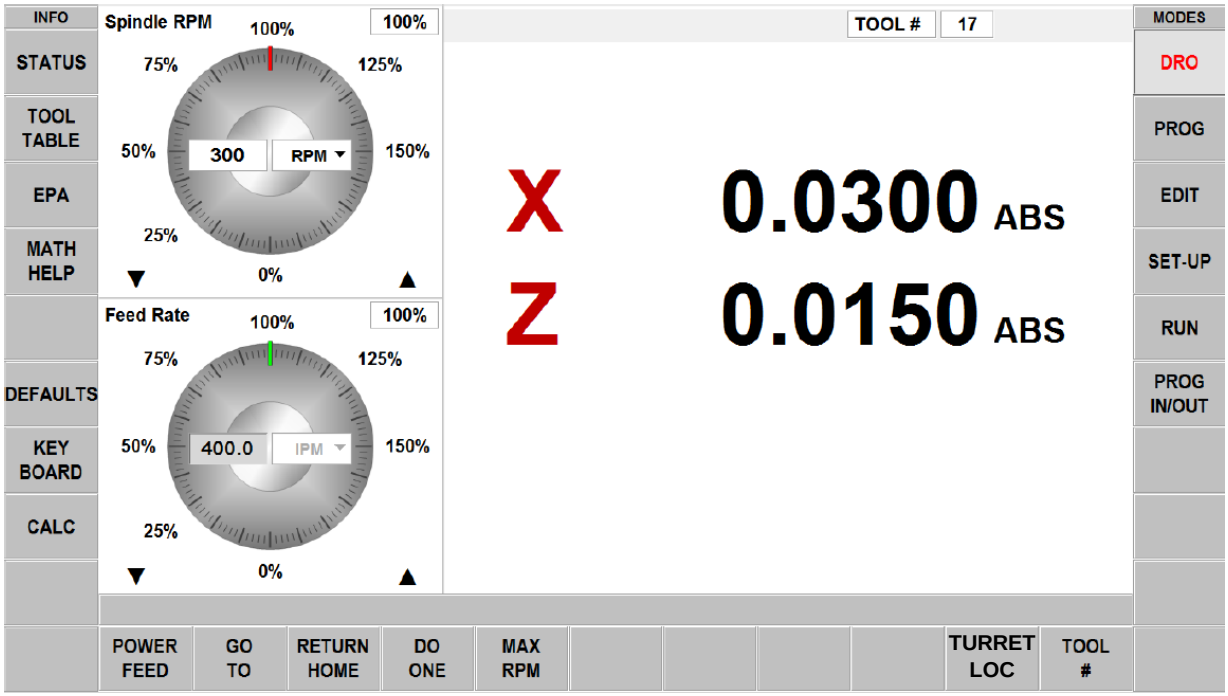


Figure 3

Review Figure 4 which illustrates machine motion as it corresponds to various controls.

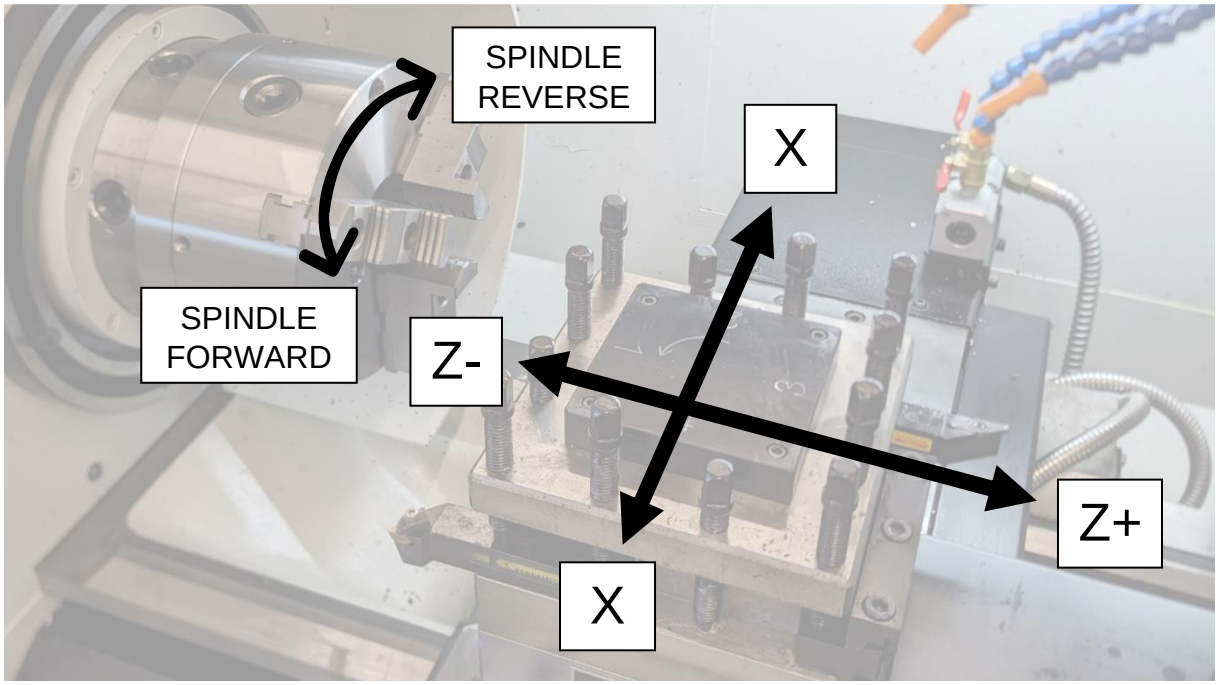


Figure 4

MANUAL CONTROLS

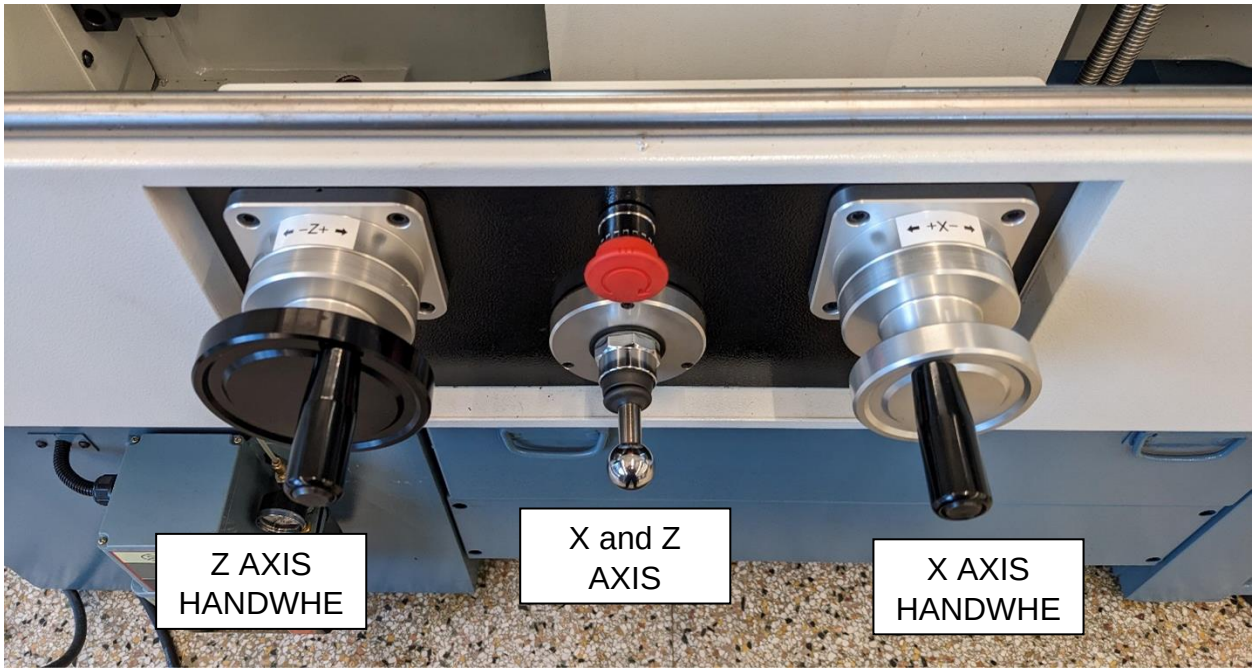


Figure 5

1. Locate the EHW (electronic handwheel) label on the control pendant and press the “C” hard key. This selects the COARSE, or faster, rate of motion for X and Z axis movement.
2. Slowly rotate each handwheel and observe the movement of the TOOL. TURRET. Also note how the X and Y values in the DRO will correspond to handwheel movement.
3. Now press the “F” (or FINE) hard key and note the difference in axis movement when the handwheels are turned.
4. The JOGSTICK, located between the handwheels, is intended to quickly move to a desired location along either the X or Z axis. Note the following corresponding directions and try using the JOGSTICK:
   Right = Z+ Left = Z- Up = X- Down = X+
5. In the SPINDLE RPM section of the screen, tap the numerical field and use the hard keys to enter 150, then press the ABS SET.
6. Press the FWD hard key and the spindle will turn on.
7. Now enter an RPM of 300 and press ABS SET. The spindle speed will increase.
8. Press the OFF hard key to stop the spindle.
9. To change the TOOL TURRET position, tap the TURRET LOC soft key, enter a number 1-4, and press ABS SET. ***IMPORTANT! The turret will automatically rotate during a tool change. Make sure the turret is in a safe location where the tools will not collide with any other objects!*** Slide the machine door into the closed position and press the GO hard key to change the turret location.

WORKHOLDING

Prepare a 1”x4” piece of aluminum rod before proceeding with this section.

The TRAK lathe will have either a 3-JAW CHUCK or COLLET CLOSER installed that is used to secure stock material. Properly securing stock material is crucial in order to safely and successfully machine parts. While the 3-jaw chuck provides a quick and easy way to load and unload stock, the collet closer provides better clamping force and accuracy.

LENGTH OF MACHINABLE STOCK

Tightly clamping stock material is only half of a successful workholding setup, the other being how far the stock sticks out from the clamping device. Clamping devices such as a chuck or collet will allow us to machine a certain length of material, however, if the stock sticks out too far it may become unstable under the constant load of the machining process. This can result in decreased accuracy, part warping, and potential damage to the tools and/or machine.

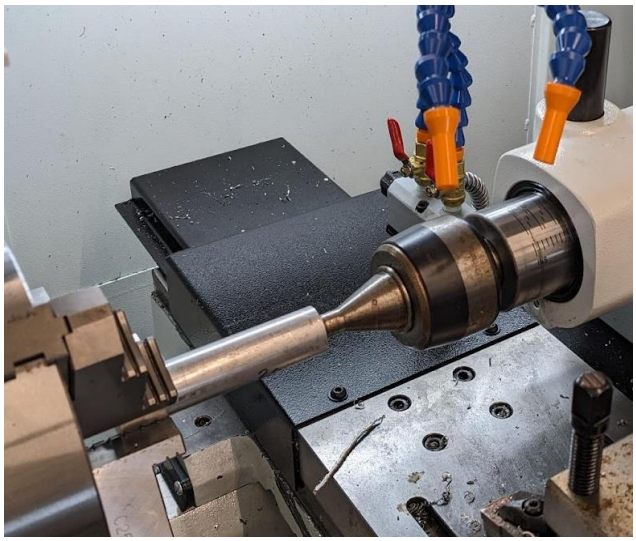


Figure 6

To avoid this instability a diameter to length ratio of 1:3 can be used to determine how much stock sticks out from the clamping device. For example, a 1” diameter rod of aluminum may extend up to 3” beyond the clamping device with at least 1” clamped in the chuck or collet. The stick-out ratio and length clamped will vary depending on several factors, such as: material strength, clamping method, and toolpath setup.
For parts that exceed the optimal stick-out length, a **tailstock** and **live center** (Figure 6) may be used to stabilize the unclamped end of the stock.

Please note: If the 3-JAW CHUCK is installed, please skip the COLLET CLOSER instructions. If your project calls for swapping the workholding devices, ask a Lab Supervisor to assist you.

COLLET CLOSER and COLLET

A collet works within a tube or shaft to apply clamping pressure to a tool or part. In the case of the TRAK COLLET CLOSER, the collet screws into a drawtube that, when further force is applied, causes the collet to clamp tightly onto a part.

Figure 7

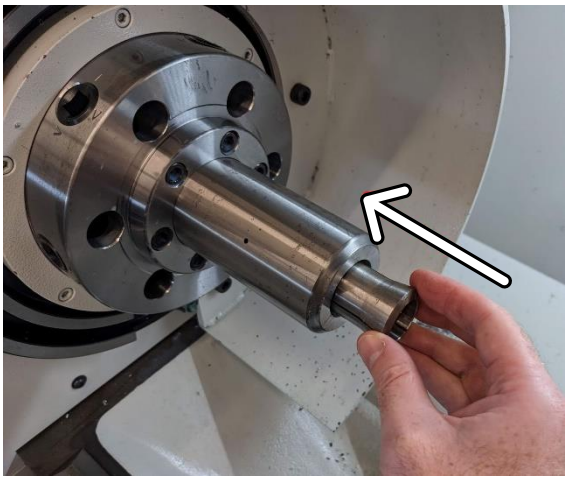


Figure 8

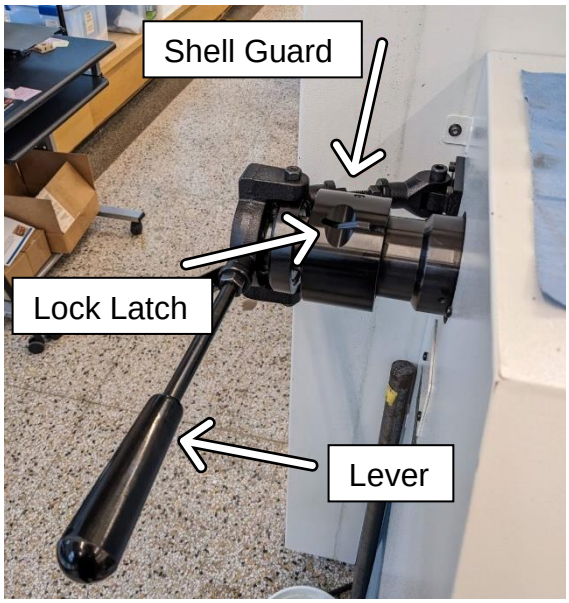


Figure 9

1. Locate a 1” 5C collet (Figure 7) from the lathe tool chest and check if the stock will fit. If not, try using a 1.1/64” collet.
2. Place only the collet into the drawtube, thread-side first (Figure 8).
3. Identify the shell guard at the opposite end of the collet closer (Figure 9) and ensure the lock latch is not engaged. The guard should spin freely in either direction when turned by hand.
4. While gently pressing the collet into the draw tube, turn the shell guard toward you until you feel the collet threading into the draw tube. Continue turning until the collet feels fully threaded, then unthread it by a few turns to make room for adjustment in the next step. Make sure the lever remains in the disengaged position by pulling it toward the machine.
5. Insert the stock into the collet using the 1:3 stick-out ratio explained in the prior section. You may need to further thread or unthread the collet a bit until the stock can be inserted.
6. Try to engage the collet closer drawtube by pushing the large lever (Figure 9) *away* from the machine. The lever should provide some resistance and then move into a locked position. If you do not feel the lock engaging, continue unthreading the collet in small increments until you can fully engage the drawtube and the stock is securely clamped in the collet. ***Never run the spindle without a collet loaded in the draw tube!***

3-JAW CHUCK

1. Insert the aluminum rod into the chuck so at least 1” will be clamped in the jaws with approximately 3” sticking out.
2. Use the chuck key to tightly clamp the jaws concentrically around the stock (Figure 10).

Figure 10

LATHE TOOLING

Lathe tooling typically consists of a tool holder and a single-point insert which performs the cutting operation. Understanding which tools are suitable for a given operation is critical to the process, as is the condition of the cutting inserts.

Upon close inspection of the cutting insert in Figure 11 we see that part of the radius has been chipped off and **should not be used**. With this symmetrical style of insert we can simply reorient the insert to use the undamaged side.

Always inspect each tool you intend to use before machining any material.

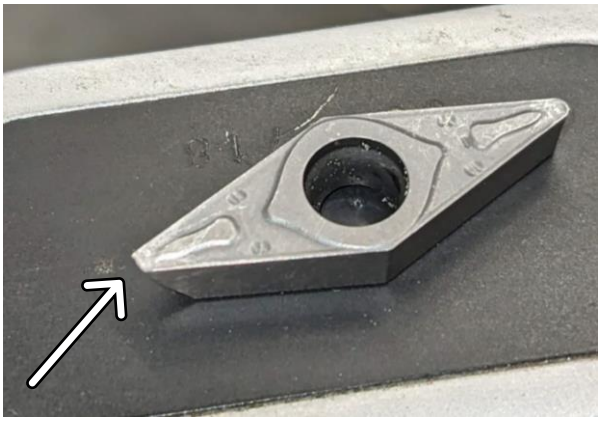


Figure 11

IMPORTANT! A Lab Supervisor must inspect your workholding and tool setup before moving on to the next section.

PART ZERO

To accurately machine part geometry we must determine and set PART ZERO, which is a TRAK term for *work offset.* A work offset typically defines a reference point, or *datum*, relative to machine motion along its axes. **Work offsets are typically defined in relation to a tool's cutting point.**

While a work offset can be set in any location in relation to the stock material, the most common reference point used is the center of the spindle along the Z axis, and the end of the part extending from the workholding along the X axis (Figure 12).

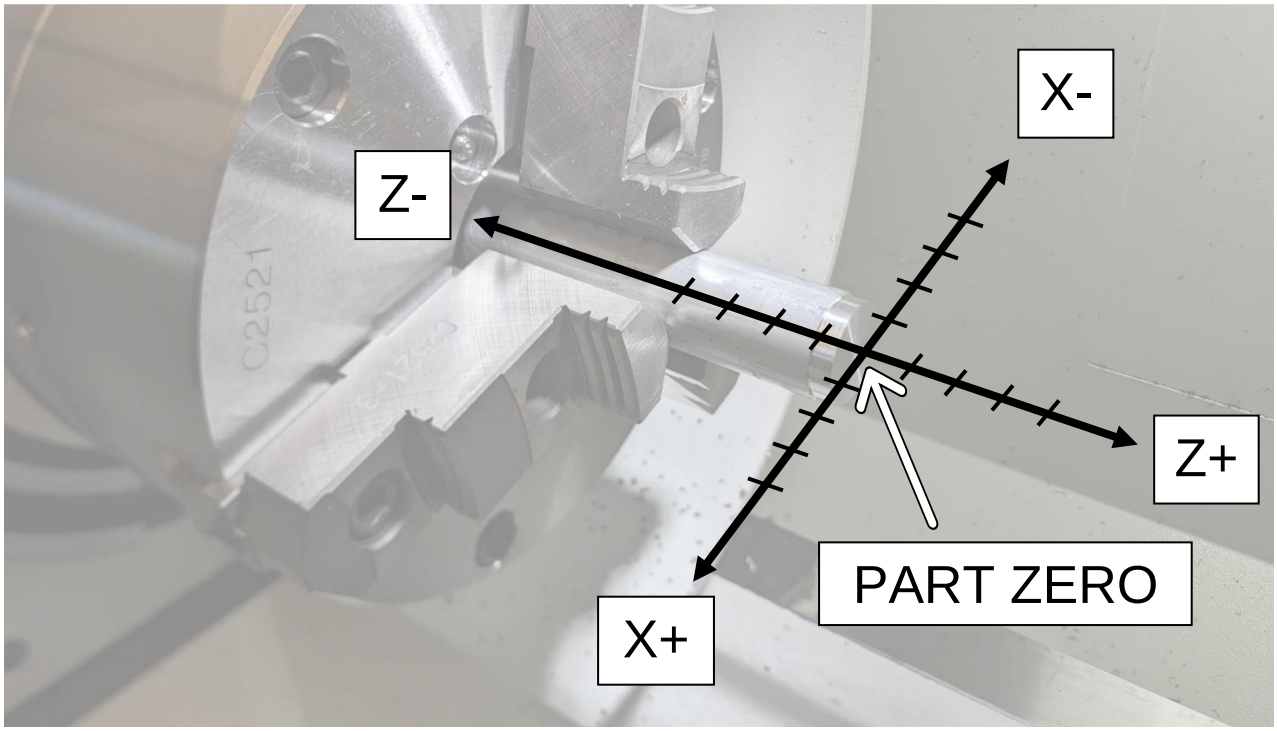


Figure 12

TOOL TABLE

Tap the TOOL TABLE soft key to open its pop-out window. The TOOL TABLE (Figure 13) is used to setup and store tool information. Several tools have already been setup in the TOOL LIBRARY by EPIC supervisors. TOOL SETUP involves assigning information such as tool type, tool offset, turret location, and library number. Later in this guide we will return to the tool table and assign LIBRARY TOOLS to a PROGRAM. Tap the LIB # soft key to arrange the tool list in numerical order.

Note that the X OFFSET and Z OFFSET columns beside each LIBRARY TOOL read "SET" in a green cell. This indicates that the offset for each tool has already been defined, whereas PART ZERO is labeled "NOT SET".

Tools listed in the TOOL LIBRARY section should not be edited without the consent or guidance from an EPIC supervisor.

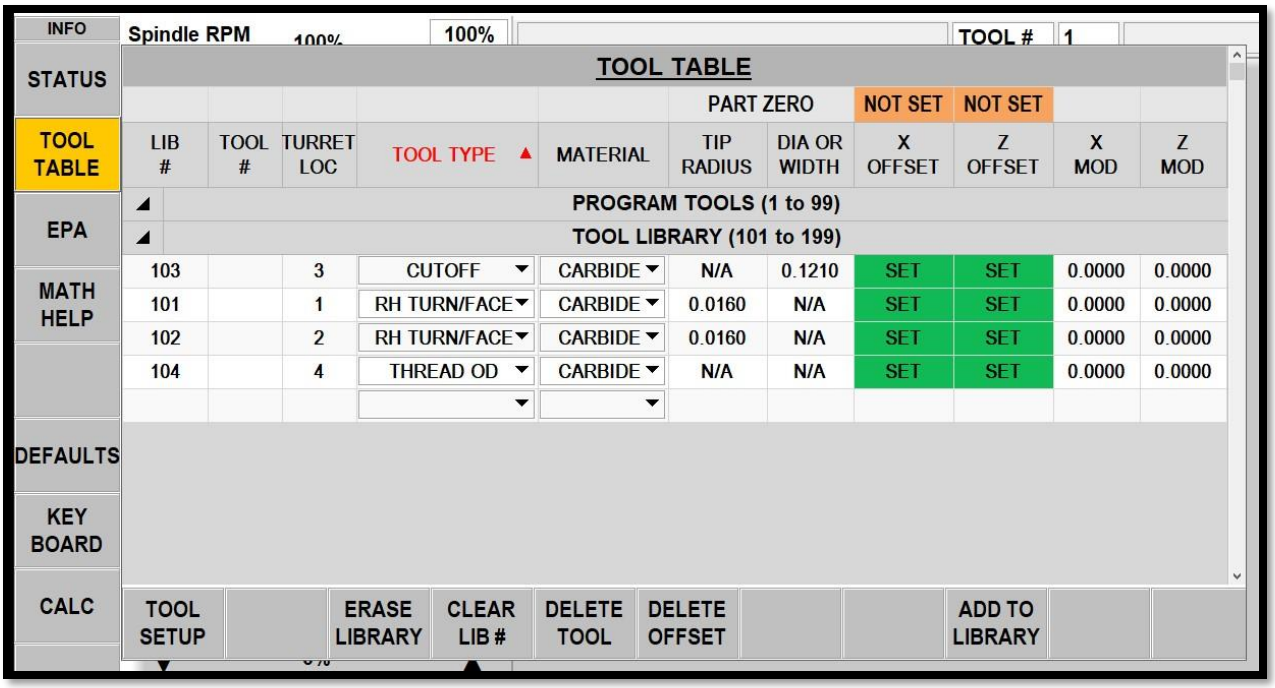


Figure 13

SET PART ZERO

To set PART ZERO you will use MANUAL OPERATION to remove some material from the stock. This is a great opportunity to *feel* and *hear* the machining process. Ask a Lab Supervisor about the terms *roughing, finishing,* and *chip breaking* and use this opportunity to practice manually machining a part on the lathe.

1. Enter DRO mode.
2. Confirm that the 80° RH TURN/FACE tool (Figure 14) is loaded in TURRET LOCATION 1. If not, ask a lab supervisor for assistance.
3. Move the TOOL TURRET to a safe location to complete a tool change.
4. Load LIBRARY TOOL 101 by pressing TOOL #, enter “101”, press ABS SET, then GO to complete the tool change. ***Note that loading a LIBRARY TOOL is different than simply changing the TURRET LOCATION. Several library tools can be assigned to the same turret location!***
5. Enter 500 for the SPINDLE RPM and press ABS SET.
6. Press the FWD hard key to start the spindle.
7. Press the “C” hard key to set the handwheel movement rate to COARSE.

IMPORTANT! When manually machining the stock using the handwheels DO NOT hold both handwheels at the same time. ***Practice moving only one axis at a time whenever you are cutting into material manually.*** *This will avoid accidental movements which may damage the cutting tool, stock, or machine.*

SET Z ZERO

8. Using the X and Z handwheels, align the cutting tip so it will remove less than 0.005” from the end of the stock (Figure 14). This does not need to be a precisely measured value. The goal is to remove a marginal amount of material so the end of the stock is consistently flat.
9. Use the X handwheel to steadily move along X- just past the center of the stock diameter, observing how the material is being removed, then retract along X+ away from the stock.
10. Now increase the spindle speed to 1000RPM and practice steps 7 and 8 again. **After making a final pass along the X axis *do not move the Z axis.***
11. Press the Z hard key and then ABS SET. This sets the Z axis PART ZERO reference point. The Z value in the DRO should now read “0”.

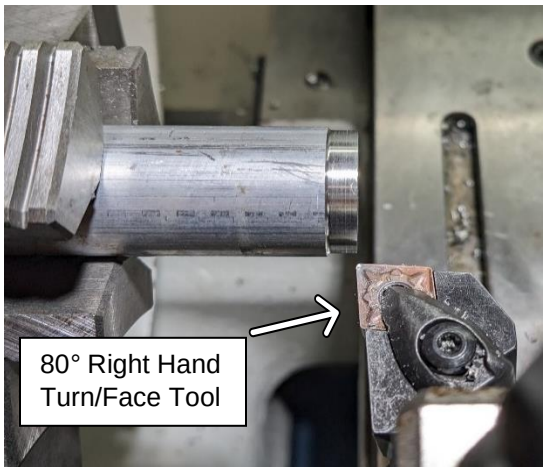


Figure 14

SET X ZERO

12. Using your new Z reference point, position the cutter tip at Z0.05 and align along the X axis to remove approximately 0.005” from the outer diameter of the stock.
13. Press the GO TO soft key, then Z hard key, and enter the value “-0.25” and press ABS SET. This will limit axis motion to a set value until the RETURN soft key is pressed. This can also be used to set an X axis “go to” limit.
14. Use the handwheel to steadily move in the Z- direction. Movement will automatically stop at the defined GO TO position. Now move away in Z+. Practice this a few more times in order to decrease the diameter at the end of the stock. Once comfortable with the process, make one final slow pass so the outer diameter of the turned section looks smooth and shiny (Figure 14). **After making a final pass do not move the X axis.** Move only in Z+ to position the tool a safe distance from the part.
15. Stop the spindle and press RETURN to exit GO TO mode.
16. Use calipers to measure and record the outer diameter of the newly turned section.
17. Press the X hard key, enter the recorded value from step 15, and then press ABS SET. The PART ZERO is now set for the X axis and the DRO should read the cutter’s current position.
18. Verify the work offset by jogging to **X0 Z0.05**. If done correctly, the tip of the cutter should be aligned with the center of the stock along the X axis, and 0.05” away from the end of the stock along Z (Figure 15).

VERIFY TOOL OFFSETS

You have already set and verified PART ZERO for TOOL 101. Follow the steps below to verify that PART ZERO is also accurate with tools 102 and 103, which will be used in the following section. If you suspect the TOOL OFFSET to be incorrect, ask an EPIC Lab Supervisor for assistance.

Figure 15

Note that this method is meant to approximately gauge whether tool offsets have been set correctly. Ask a lab supervisor to go into more detail on tool setup in general.

1. Ensure the turret is at a safe position to complete a tool change.
2. Use the TOOL # soft key to change to TOOL 102 and verify the turret is loaded with a 35° Right Hand Turn/Face tool (**Error! Reference source not found.**). Move to **X0 Z0.05** and verify the location of the tool's cutting point is centered just beyond the end of the stock (**Error! Reference source not found.**).
3. Move to a safe position and change to TOOL # 103 (**Error! Reference source not found.**). Move to **X0 Z0.18** and verify the position (**Error! Reference source not found.**). Note that the cutoff tool's reference point is at the right side of the cutter; it is this point that should in fact be 0.18" away from the end of the stock in the Z direction.

Figure 16

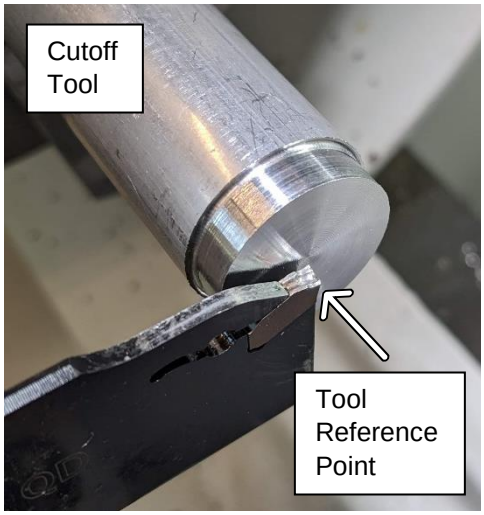


Figure 17

Ask an EPIC Lab Supervisor to verify PART ZERO and TOOL OFFSETS before continuing to the next section.

RUNNING A PROGRAM

In this section you will run a PROGRAM that will machine a chess pawn using the WORKHOLDING and PART ZERO setups from the previous sections. You will be introduced to the TRAK conversational programming interface, along with the “TRAKING” feature, which allows manual control of a CNC program’s federate using the handwheels.

The furthest toolpath extent for this part is -1.795” from Z0. Before proceeding ensure that the length of stock sticking out from the chuck or collet is no less than 2” to avoid crashing a tool into the workholding.

1. Press the PROG IN/OUT soft key to enter the file browser screen, select “DEMO PART – CHESS PAWN 1x2” then press OPEN. The screen will change to the default white and red TRAK graphic.
2. ***IMPORTANT!*** Tap the SET-UP soft key, then REF POSN (**Error! Reference source not found.**), and ensure the positions for X HOME are set to 2.0 and Z HOME 4.0. This is the position the turret will move to during a tool change, and at the beginning and end of a program. If this is not set properly it can result in damage to the machine, tool, and/or part. When running a program, ensure the axes are moving in the correct direction during reference position moves.

RX Offline

INFO | STATUS | TOOL TABLE | EPA | MATH HELP

REFERENCE POSITION TABLE

REF POSITION	POSITION		STATUS	
X HOME	2.0000	abs		
Z HOME	4.0000	abs		
X LOWER LIMIT			OFF	
X UPPER LIMIT			OFF	
Z LOWER LIMIT			OFF	
Z UPPER LIMIT			OFF	

Figure 18

3. Open the TOOL TABLE window (Figure 19) and note that there are now three tools listed under the PROGRAM TOOLS section and the X and Z OFFSET columns are labeled NOT SET. Once a program is opened, any tools called for in the program will be listed here.

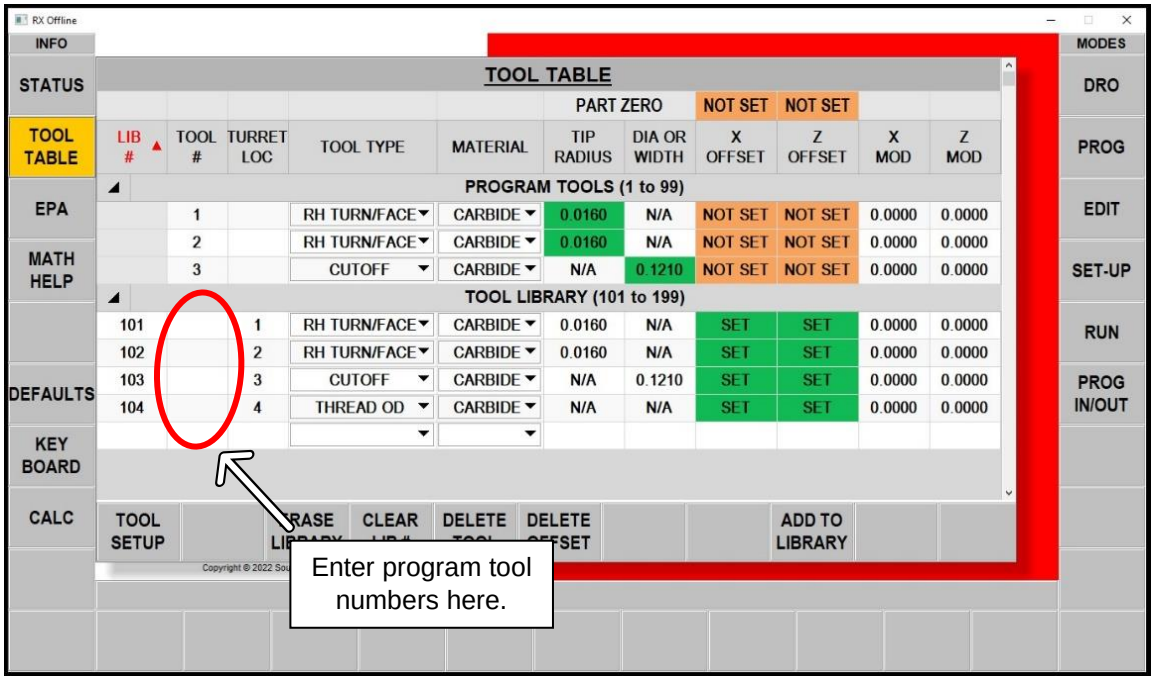


Figure 19

4. Now we must assign tools from the TOOL LIBRARY list to their corresponding PROGRAM TOOLS. In the TOOL # column (Figure 19), enter “1” beside LIBRARY TOOL 101 and press ABS SET. Note that tool 101 has been called into the PROGRAM TOOLS list under the LIB # column. Continue assigning LIBRARY TOOL 102 to PROGRAM TOOL 2, and 103 to 3. Close the TOOL TABLE window when done.
5. Press the PROG soft key to enter the programming interface. The first page will list the program name.
6. Swipe left on the touchscreen to view EVENT 1 at the right side of the screen and a graphic display of the program geometry on the left (Figure 21).
7. Utilize the touchscreen controls to navigate the graphical and event displays:
    1. Pan the graphical display with single finger motion.
    2. Zoom in/out with two finger pinching motion.
    3. Use the swipe motion to move through events. Note that as you swipe to a new event its corresponding geometry will highlight briefly in the graphic display.
    4. Expand or collapse multi-event view by dragging the divider bar (Figure 21) left or right.
8. Press the SET-UP then TOOL PATH soft keys to preview the programed toolpaths (Figure 20). Press the RETURN when done exploring this view.

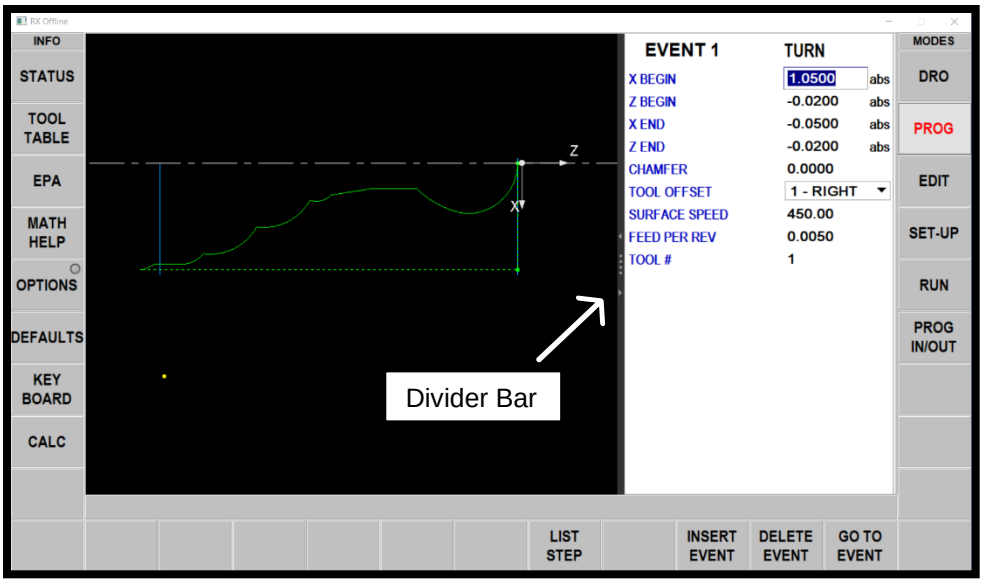


Figure 21

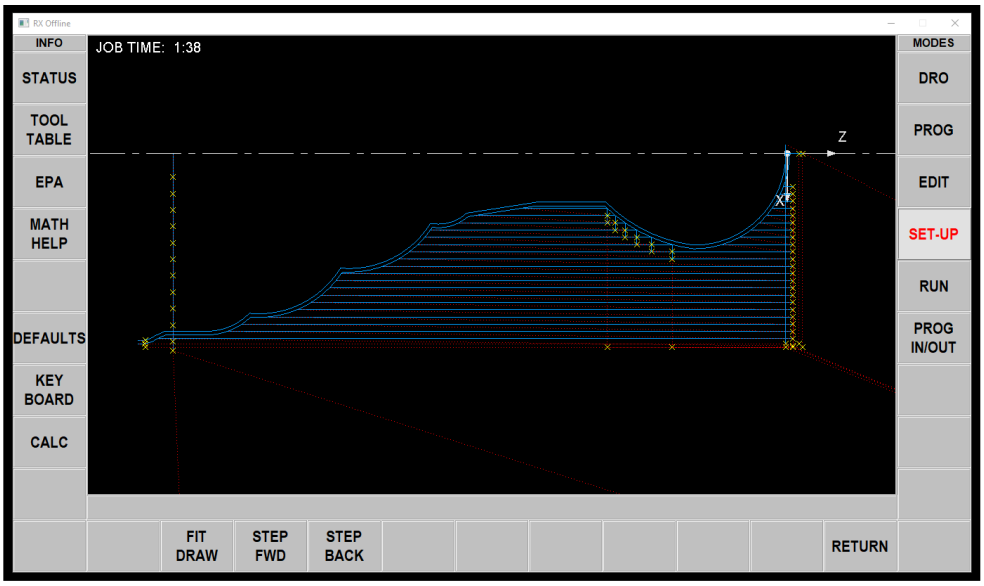


Figure 20

9. Press the VERIFY PART then MAKE PART soft keys to enter the SIMULATION screen. Press VERIFY PART to preview a simulation of the program operations (Figure 22). Press EXIT when done exploring this view.
10. Ensure the sliding door is in the closed position.

IMPORTANT! Ask an EPIC Lab Supervisor to monitor your progress for the remainder of this section.

11. Press the RUN then START soft keys. You will then be prompted to start the spindle by pressing the FWD hard key.

12. Press the TRAKING soft key. The machine is now in TRAKING mode which allows you to manually control a program's X and Z feedrate in both forward and reverse using the HANDWHEELS.

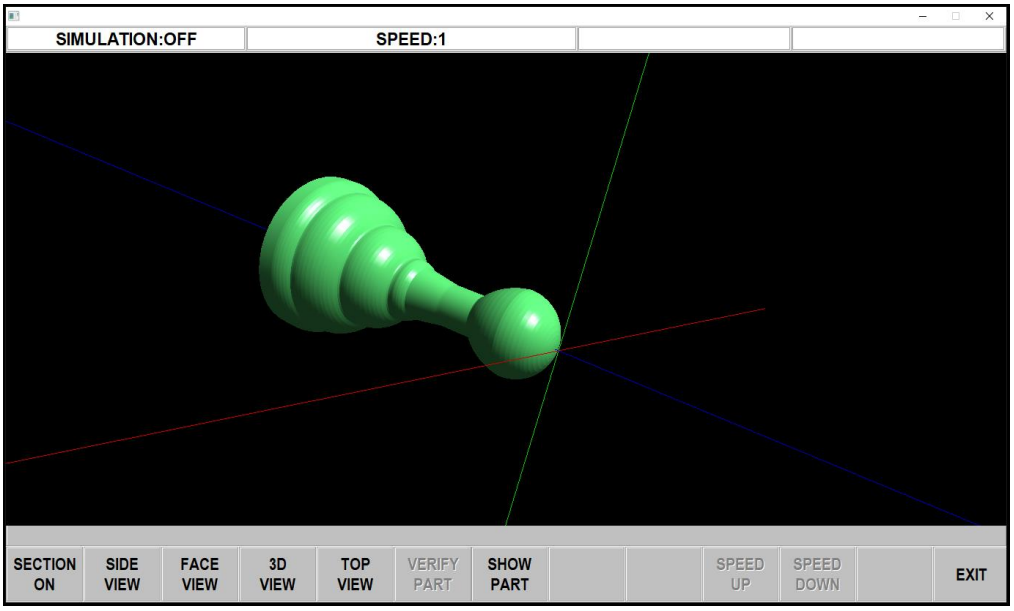


Figure 22

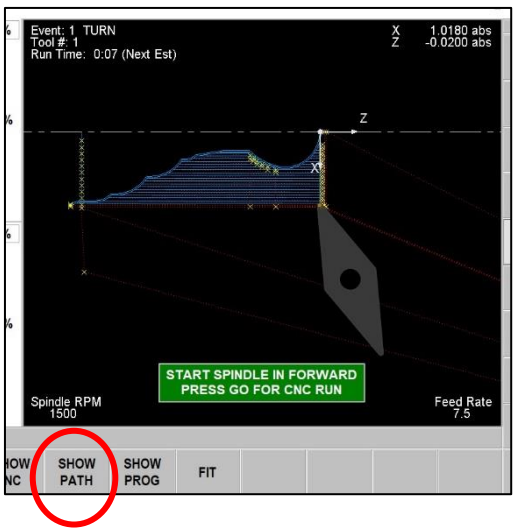
13. Begin turning the Z HANDWHEEL in the Z- direction. This will advance program operations forward, first moving to the HOME REFERENCE POSITION to initiate a tool change if needed. After the tool change completes continue turning the handwheel in Z- to advance the program. Jogging in Z+ will reverse the program's movements. The X HANDWHEEL also controls program tracking but at a finer rate, with X- advancing the program and X+ reversing.
14. Continue rotating the HANDWHEEL in Z- to approach the part, then closely observe as you drive the program forward to face 0.05" of material off the end of the stock. **It may help to advance the program with the X handwheel until you are comfortable moving at a faster rate.**
15. Continue moving the program forward. The program will change to TOOL 2 and then advance to begin turning the overall profile of the part. Observe the first pass in the Z- direction and ensure the tool will not crash into the workholding. After completing a profile pass, the tool will retract slightly away from the part and move back toward the end of the stock to begin the second pass.
16. This will confirm the furthest extent of TOOL 2's operation which can now be run in full CNC mode.

Figure 23

17. Turn on the flood coolant system by pressing the ACCESSORY hard key and direct the stream at the tool's cutting tip (Figure 23).
18. **Be prepared to stop in the event of a potential issue or emergency:** If you suspect either, note that in CNC mode the STOP hard key will cease X and Y axis movement while the spindle remains on and you will be able to continue CNC or TRAKING operations from where you left off. Pressing the red EMERGENCY STOP button near the handwheels or screen will cease all axis and spindle operations and the motors will need to be reset and the program reloaded.
19. Press the STOP hard key to exit TRAKING mode, then press the CNC RUN soft key.

Note: As the program runs in CNC mode, closely observe the small chips being produced by the turning process. Consistency of the size and shape of these chips play a large role in efficient machining.

20. Pressing the GO hard key will prompt the program to move forward automatically at predefined speeds. Press GO now to continue the program.
21. When the profiling operation finishes the turret will load TOOL 3. Press STOP then press TRAKING before continuing with the final CUTOFF EVENT.
22. As with step 12, use tracking to ensure TOOL 3 will not crash into the workholding. The CUTOFF TOOL will approach the base of the part, close to the workholding, and begin moving into the part in X+. This operation includes "pecking" which moves the cutter in and out of the part incrementally until the part falls off. Feel free to continue tracking this event or switch to CNC mode until the job is completed.
23. After a cutoff operation, a part will often fall into the large tray at the base of the machine. This tray, meant to catch chips and allow coolant to drain, can be pulled out like a drawer.
24. Upon inspecting the part, you will notice a small nub at the center of the bottom face. During the parting operation this feature is nearly unavoidable and must be removed either manually or, if possible, by clamping the part again and facing the nub off. *Ask an EPIC staff member for help finishing the part.*
25. Congratulations on running your first program!



NOTE: While in RUN mode, the SHOW PATH soft key will display and animate the tool at its current position in the program. This is a useful feature to estimate the physical tool's location relative to the programmed toolpath.

CONVERSATIONAL PROGRAMMING

In conversational programming a machine's control software prompts the user to enter parameters that define part geometry and machine motion. This method allows for potentially faster programming operations relative to the typical CAD-to-CAM (computer aided manufacturing) workflow, and is suitable for a variety of projects.

ABSOLUTE AND INCREMENTAL DIMENSIONS

While programming part geometry we will use both ABS SET (absolute) and INC SET (incremental) hard keys. If a value is entered with ABS SET, the dimension will be relative to X or Z ZERO. If entered with INC SET, it is relative to the previously entered dimension. You may find that using one mode versus the other is dependent on how a part's drawing is dimensioned. Ultimately, which mode is used is the programmer's choice.

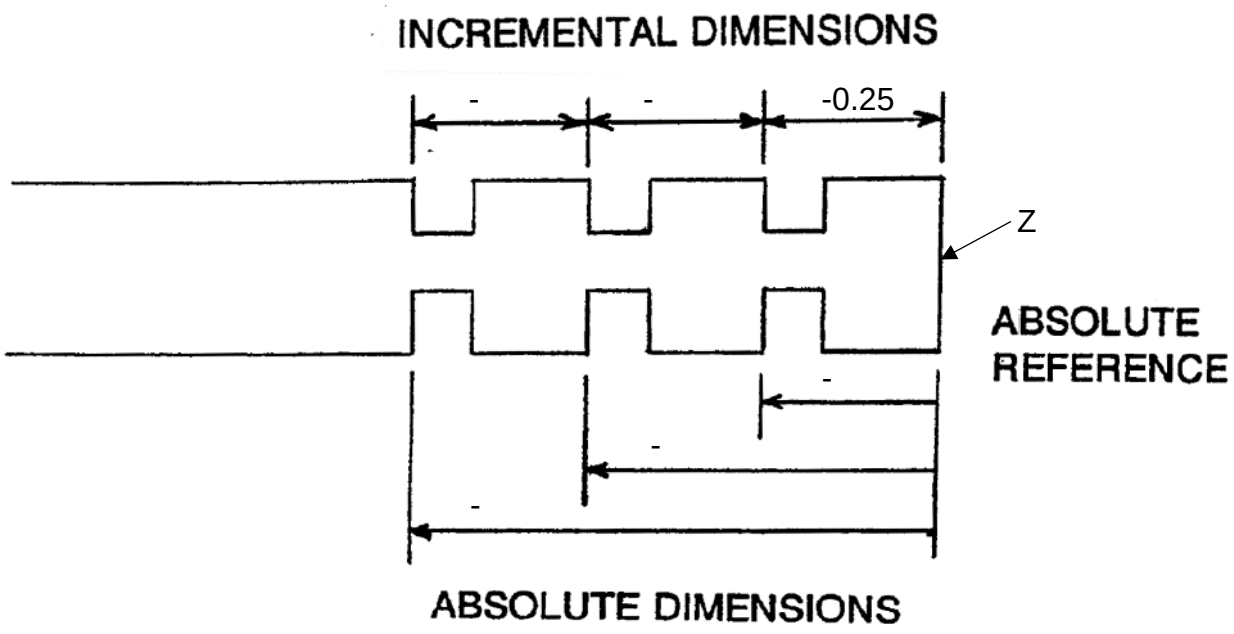


Figure 24

FEEDS and SPEEDS

The term *feeds and speeds* is associated with the rate at which a cutting tool machines a workpiece. Generally speaking, it refers to how quickly a tool feeds into a workpiece, and the rotational speed of the spindle.

During the conversational programming process, the values for the following terms will need to be defined in order to achieve optimal cutting conditions.

Depth per Cut: The thickness of material removed by the cutting tool in a single pass. Adjusting depth per cut influences factors such as machining time, surface finish, and tool wear.

Feed per Revolution: The axial distance a cutting tool travels along the workpiece for each complete rotation of the part or spindle. Higher FPR values decrease machining time but yield a rougher surface finish, while lower values increase machining time yet result in a smoother surface finish. These terms are often linked with "roughing" for quicker material removal and "finishing" for achieving smoother surfaces.

RPM: The spindle speed measured in *Rotations Per Minute.* When programming an event in this mode the spindle will spin at the set RPM and will not change unless programmed to

Surface Speed: Measured in *Surface Feet per Minute* (SFM). When programming in this mode the spindle RPM will vary based on the set SFM value, and will increase in speed as the cutter moves toward the spindle center.

NOTE: *The following part will be programmed using RPM mode for the spindle speed. Surface Speed mode may be assigned to any specific programming event by pressing the OPTIONS softkey and selecting the desired mode from the SPINDLE SPEED submenu. Consult a Lab Supervisor for more info on using Surface Speed mode.*

IMPORTANT! *The feeds and speeds referenced in this guide are meant to be used as starting values only.*

PROGRAMMING A SAMPLE PART

The programming in this section can be completed using the control software at the machine or the offline software at a lab computer. If working with the offline software, ask a Lab Supervisor to help with setup. **Review PROGRAM MANAGEMENT in the following subsection and be sure to save your program regularly.**

IMPORTANT! This program will use 35° RH TURN/FACE, THREAD OD (outer diameter), and CUTOFF tools. Ensure these tools are loaded and all offsets are correct before proceeding!

Prepare a 0.75”x3” piece of aluminum rod for the stock material.

1. Press the PROG soft key to enter the programming interface. *If a program is already open, tap SET-UP, ERASE PROG, and return to the program screen*. Title your program, including your name, and press ABS SET.
2. Swipe to EVENT 1 and tap the CYCLE soft key. A CYCLE is used to define a starting point and parameters that will be used across several events. For this CYCLE we are defining geometry so the RH TURN/FACE tool will machine only the outer profile of the part. This will not include the screw threads or the back face of the part.

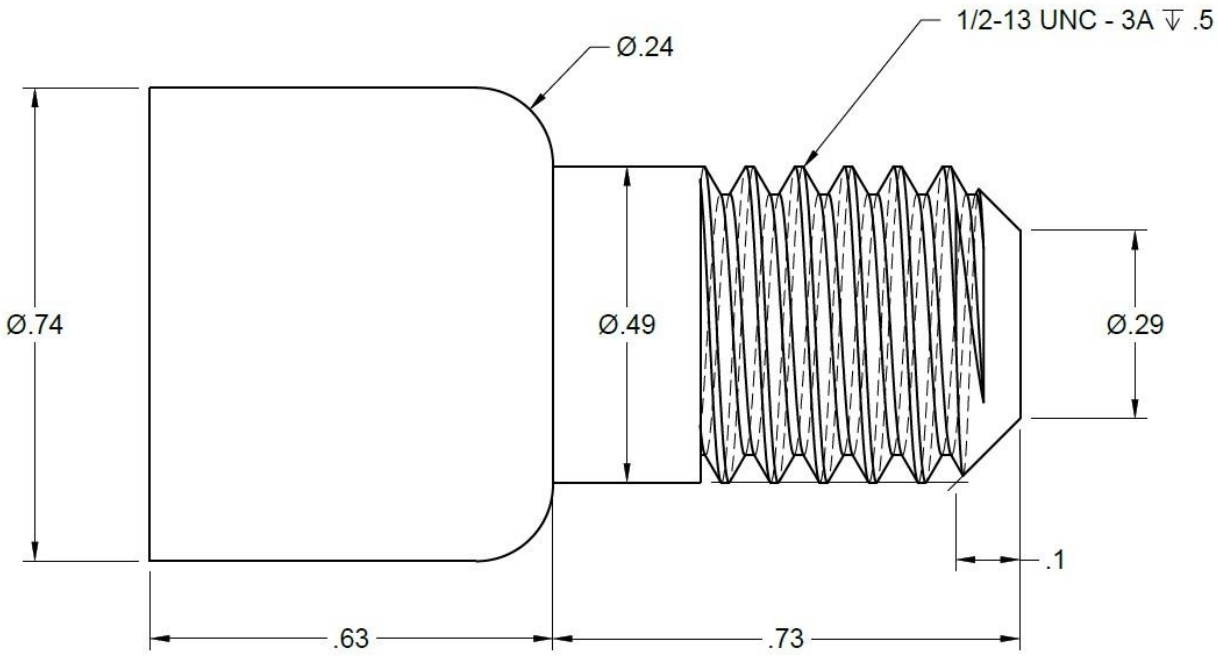


Figure 25

3. Reference the part drawing (Figure 25) as you proceed through the following steps.
4. Enter the data for EVENT 1 as seen in Figure 26. Be sure to press ABS SET after each entry. The values for X BEGIN and Z BEGIN define the first point of the part's geometry.
5. Swipe to EVENT 2 and tap the CYCLE TURN soft key. Enter 0.49 for X END and press ABS SET. Enter -0.1 for Z END and press **INC SET**. When the CHAMFER field is selected a pop-up will indicate ABS SET FOR CHAMFER, INC SET FOR CONRAD (corner radius). There is no chamfer or radius to enter here so simply press ABS SET to enter the value "0". Note that a green line is now drawn between the two coordinates indicated in EVENT 1 and EVENT 2, as seen in Figure 27.
6. Add another CYCLE TURN for EVENT 3 and populate the X and Z values based on the measurements detailed in the drawing. Try entering each Z value as *incremental* and each X value as *absolute.* Consider how it may be beneficial to choose one rather than the other when plotting the geometry of your part.
7. For EVENT 4 select CYCLE ARC and define the parameters according to the drawing.

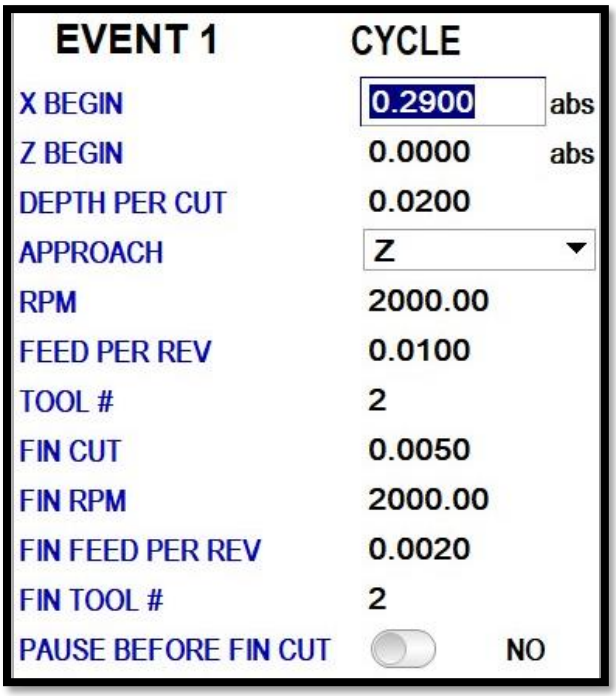


Figure 26

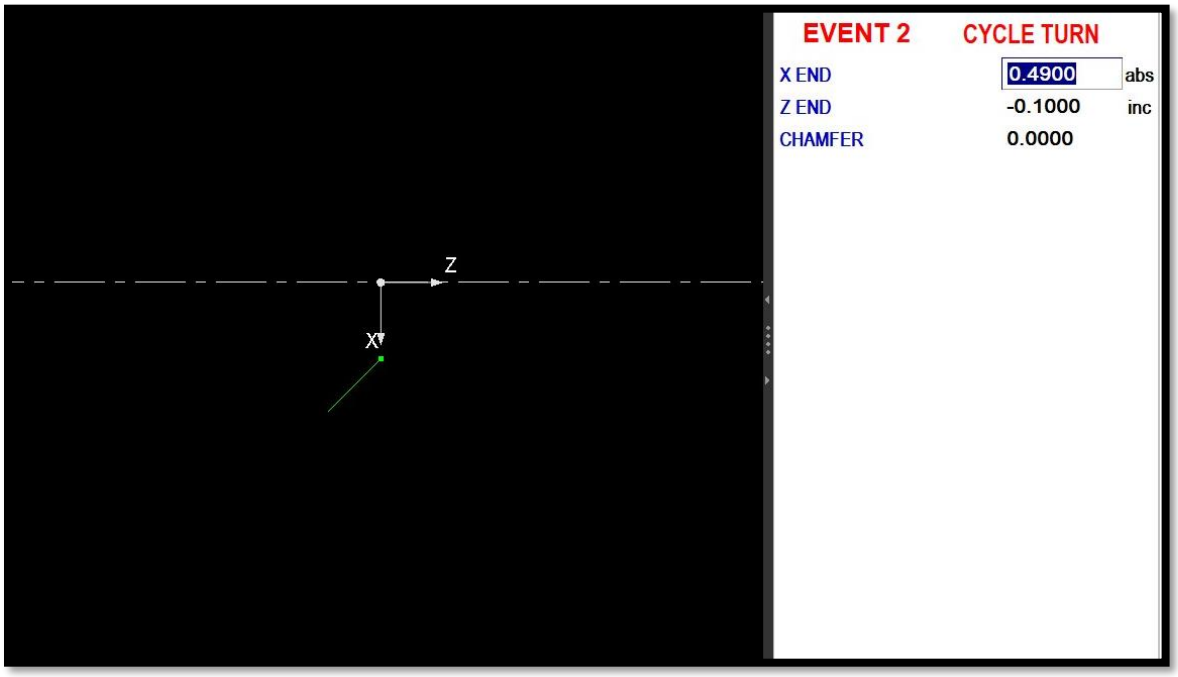


Figure 27

8. Continue defining the part geometry. Note that a final TURN or ARC event must extend to the diameter of the stock and be followed by CYCLE POSN (position) events that define the extent of the stock (see Figure 28). POSN events are represented as dashed lines.
9. Once the geometry resembles Figure 28, tap END CYCLE to move on.

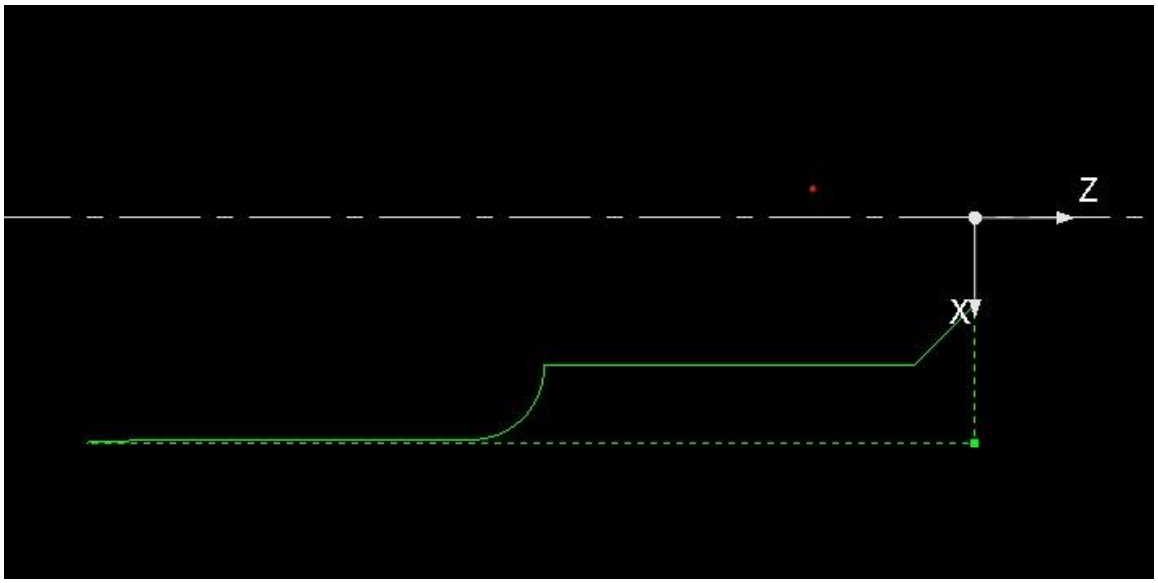


Figure 28

10. Add a THREAD event and enter the parameters as seen in Figure 29. Thread geometry is orange in the graphic display (Figure 30).
11. Tap the MORE soft key to access the CUTOFF event. This cutoff operation will move along the X axis to separate the part from the remaining stock. Use the drawing to determine the X and Z dimensions, then populate the remaining values as seen in Figure 30. Cutoff geometry is blue in the graphic display.
12. Assign LIBRARY TOOLS to their corresponding PROGRAM TOOLS using the TOOL TABLE.

EVENT 9	THREAD	
X BEGIN	0.4900	abs
Z BEGIN	0.0000	abs
X END	0.4900	abs
Z END	-0.5000	abs
PITCH	0.0769	
# OF PASSES	8	
# OF SPRING PASSES	2	
PLUNGE ANGLE	29.5000	
SIDE	OUTSIDE	
# OF STARTS	1	
RPM	150.00	
TOOL #	4	

Figure 29

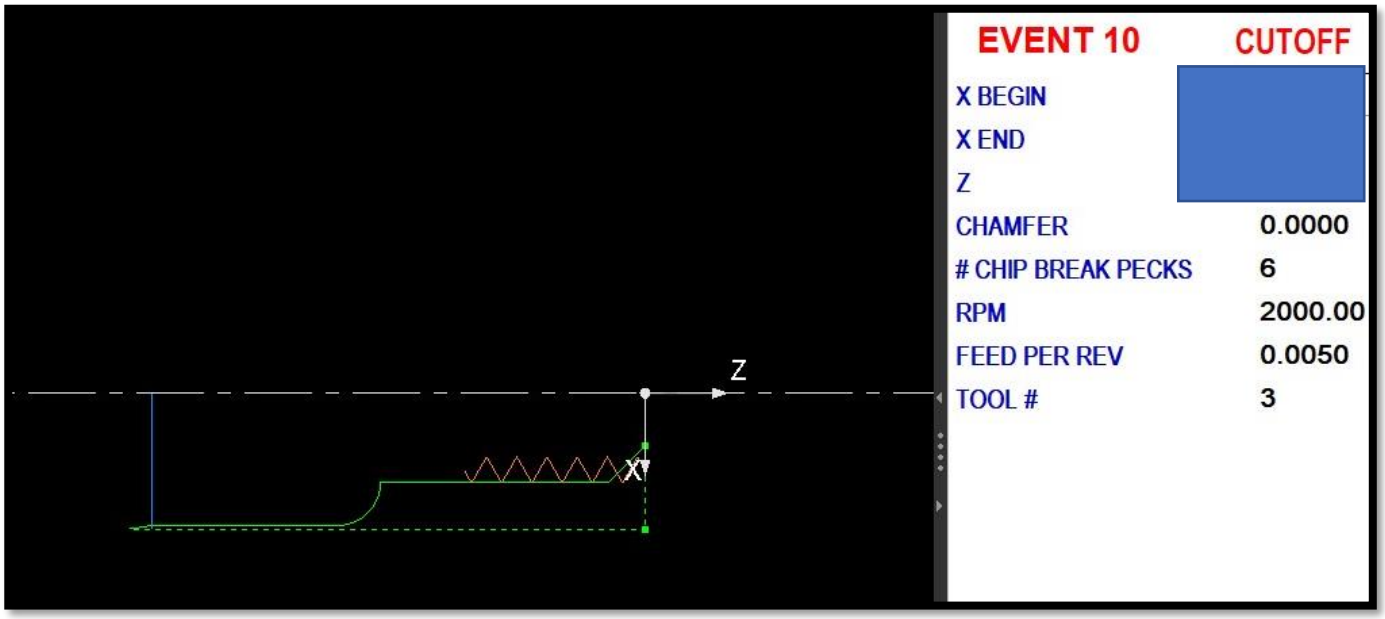


Figure 30

13. Review the program's TOOL PATH and run the VERIFY PART simulation. If there are errors in your program, they will be listed in the WARNING pop-out window. The errors must be resolved before moving on.
14. Verify that PART ZERO and all tools used in the program are labeled SET and in the green.
15. **Ask a Lab Supervisor to review your program and setup before continuing.**
16. Run the program and machine your part! When complete, be sure to inspect the part for dimensional accuracy relative to the drawing. You should be able to screw a 1/2-13 UNC nut onto the threads with ease.
17. If the part does not meet specification, consider which variables may have been overlooked and make the necessary adjustments before running the part again.

PROGRAM MANAGEMENT

- To save a program, tap the PROG IN/OUT soft key. Ensure the file name is correct then tap SAVE FILE.
- To erase a program and start a new one, tap the EDIT soft key then ERASE PROG. **This will not delete your program as long as it's been saved.**
- To preview any program's geometry, in PROG IN/OUT, tap LOOK ON then select a program. Tap LOOK OFF to exit this mode.
- In PROG IN/OUT, when SAVE TEMP is pressed, the current program, assigned tools, and PART ZERO will be stored even when the machine is turned off. When powered on again, pressing OPEN TEMP will restore all of these settings. This is useful if you will resume machining immediately the next day.

SHUT DOWN PROCEDURE

1. Move the turret to a safe position.
2. If there is stock clamped in the workholding, remove it and return it to storage.
3. If there is a collet seated in the draw tube, remove it and return it to storage.
4. Press the BACK hard key until the SHUT DOWN soft key appears. Tap SHUT DOWN and OK. Wait for the control pendant to shut down completely.
5. Turn of the main power switch at the back of the machine.
6. Use the air gun to blow chips down into the chip pan at the base of the machine and clean up any remaining debris and/or coolant with a shop cloth or brush.
7. Open the chip pan drawer and use a chip rake or other tool to remove all debris into a trash bin.
8. Leave the workspace clean and organized. 😊