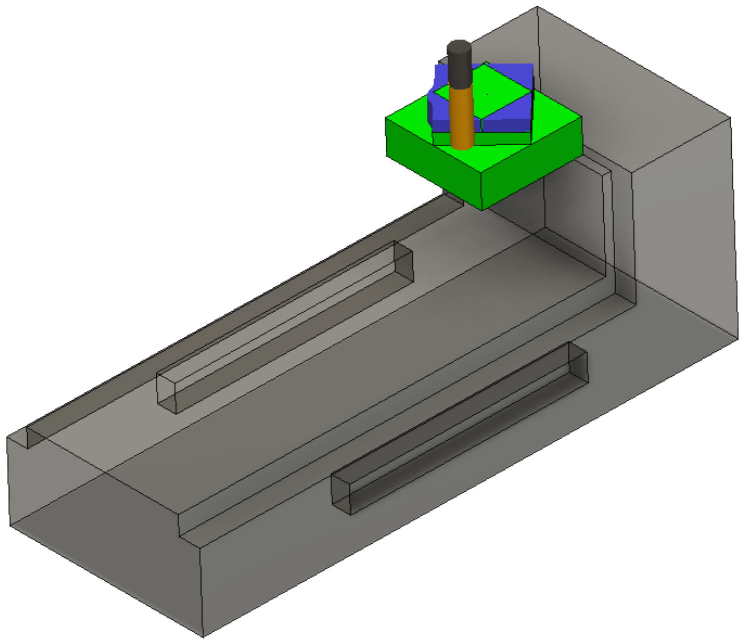
Desktop CNC Milling Machine: Standard Operating Procedure
I: General Guidelines



GENERAL SAFETY AND OPERATIONAL INSTRUCTIONS:

- Never operate the machine tool without all safety mechanisms (enclosure, door switch, emergency stop, limit switches) connected, complete, and fully operational.

- Never place hands, head, or other body parts inside the machine tool enclosure while spindle is powered and door switch / emergency stop are disengaged.

- Never handle sharp tools in the spindle without the emergency stop engaged.

- Always watch and listen to EVERY milling operation closely. Keep a hand resting over the emergency stop button: quick reactions are necessary in mitigating damage to the machine and tools.

- During ANY milling operation, if the cutting tool shows excessive wear or excessive chip build up, or if the spindle's RPM changes noticeably (this is often audible), quickly feed hold or emergency stop the machine (depending on severity) to inspect and determine if it is safe to continue.

- Always allow the spindle to reach its target speed before entering a cut.

- Always double-check setup steps and part programming to minimize machine accidents.

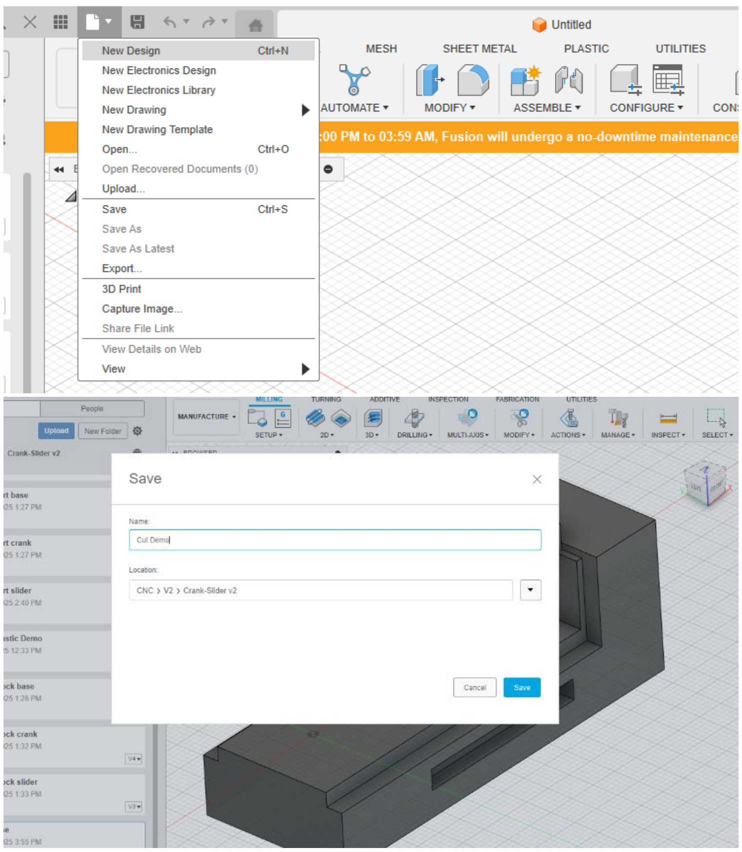
- Jogging any axis while the emergency stop is engaged will offset a probed coordinate system.

II. Part Programming (Autodesk Fusion)

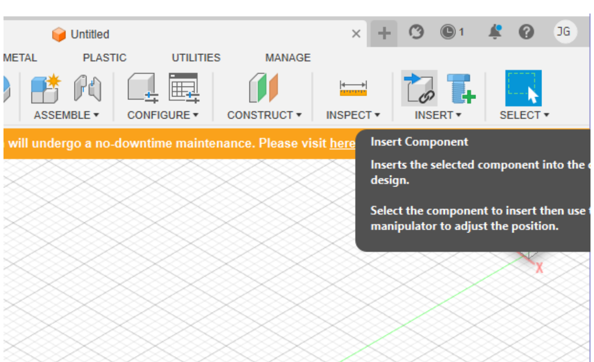
The following instructions are based on the programming for a simple machining test that will validate machine tool performance after construction. The part stock is a 36 x 36 x 20mm block, 3D Printed in ABS plastic with 100% infill. This part will be safer to cut than metals as a first machining test. This method of coordinate placement should be used for every part, however.

NOTE: This section of the SOP is geared toward machine validation and new part programming. Students should not have to place coordinates in this way when programming parts for the crank-slider demonstrator.

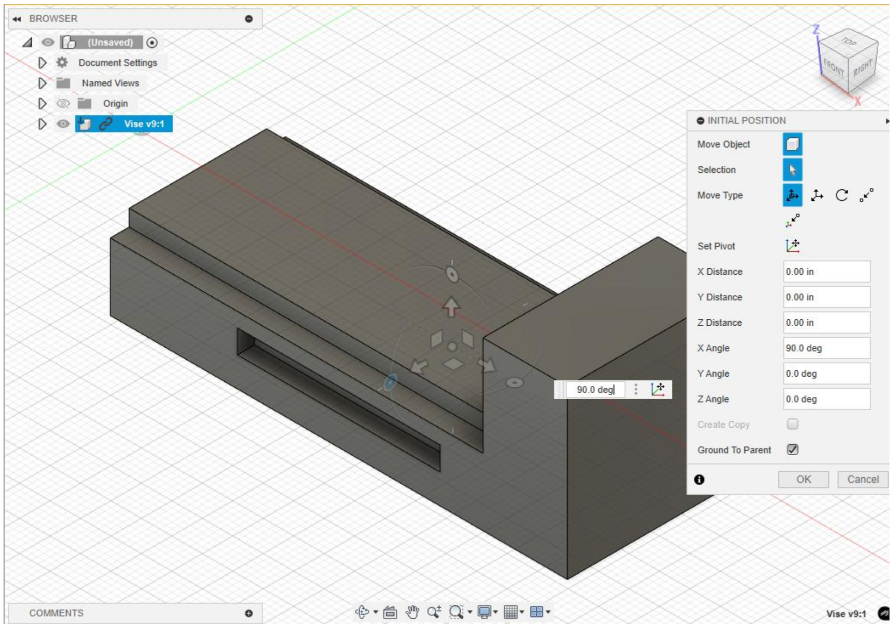
1. Start Fusion and open a new design. Save the file and name it.



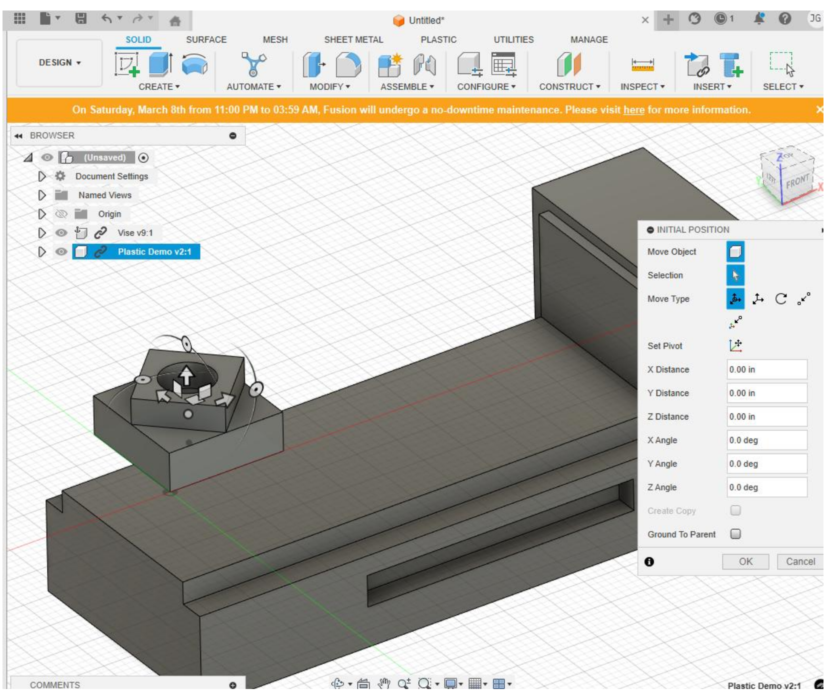
2. Ensure that the vise and square demo part are uploaded to Fusion. Select "Insert Component" and choose the "Vise" file.



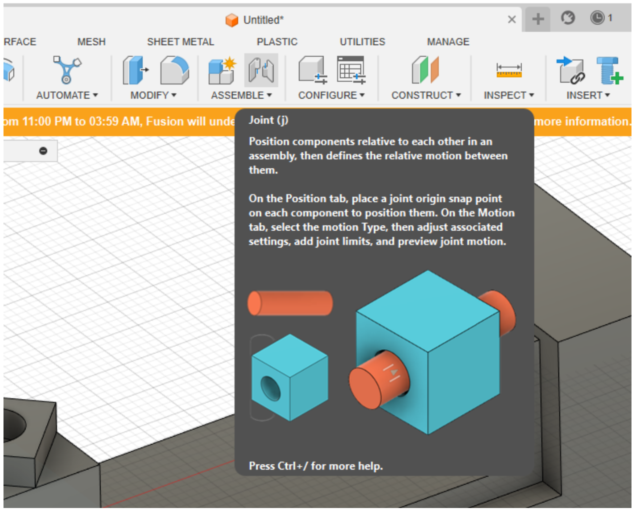
3. Orient the vise so that it is flat on the X-Y plane (any orientation on the planes will work, however, as the rest of the program will be built relative to the vise). Press "OK".



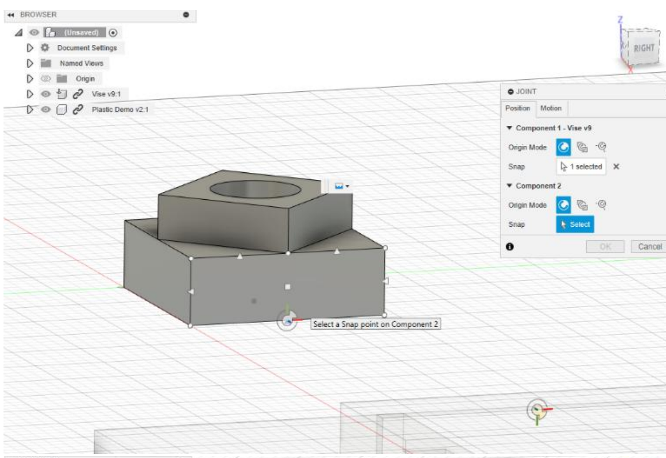
4. Insert the "Demo part" as a component. Using the sliders or number inputs, orient the part on the X-Y plane like the vise. Because this part is symmetrical, any face can be parallel to the vise jaw. The part should always be oriented in the way it will be placed in the vise for cutting.

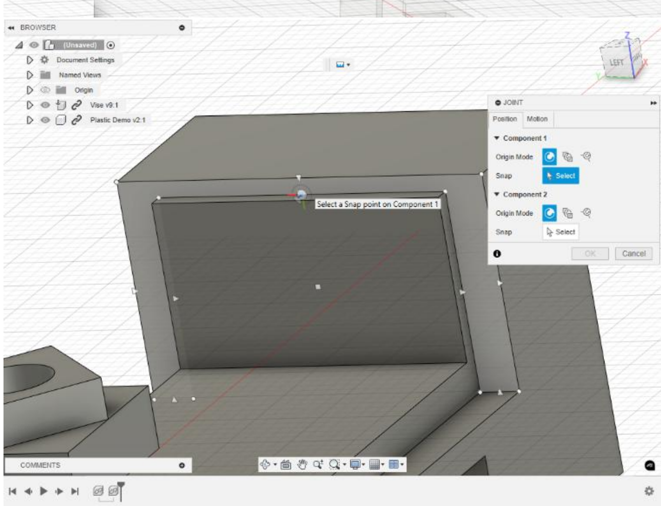


5. We will now use joints to finish orienting the parts. Select “Joint” from the top ribbon.

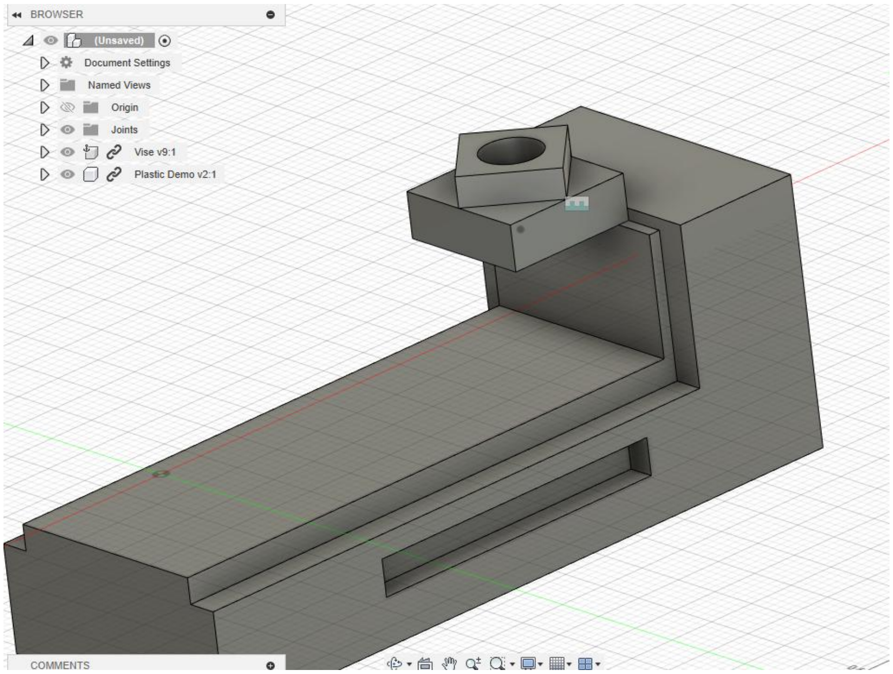


6. Join the part so that it rests on the “parallels” modeled on the vise. This is managed by selecting a coincident point on coincident faces of each component as seen below.

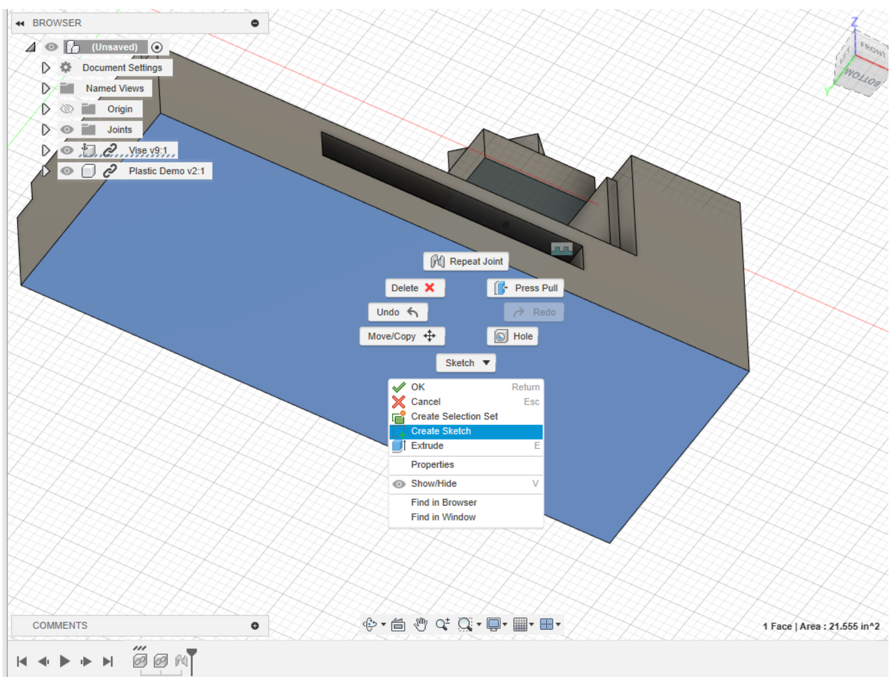




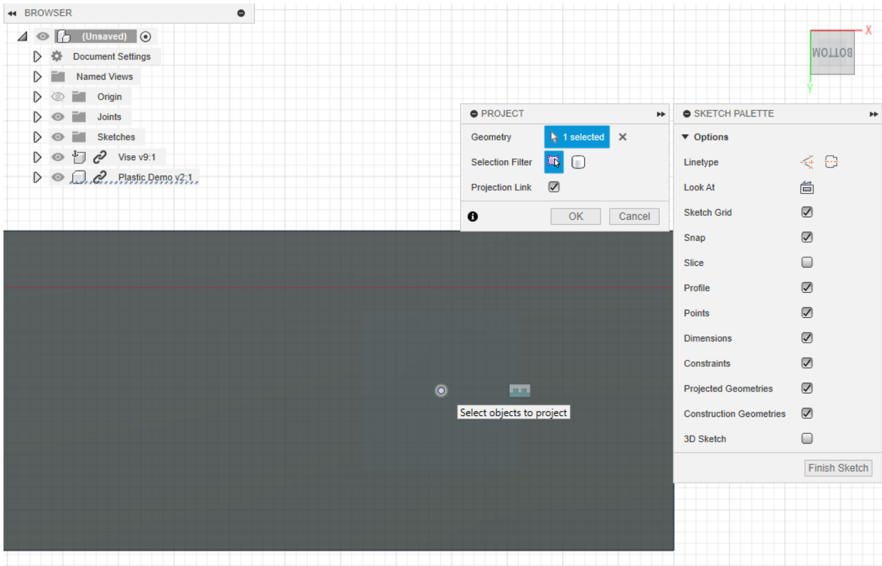
The result is shown below.

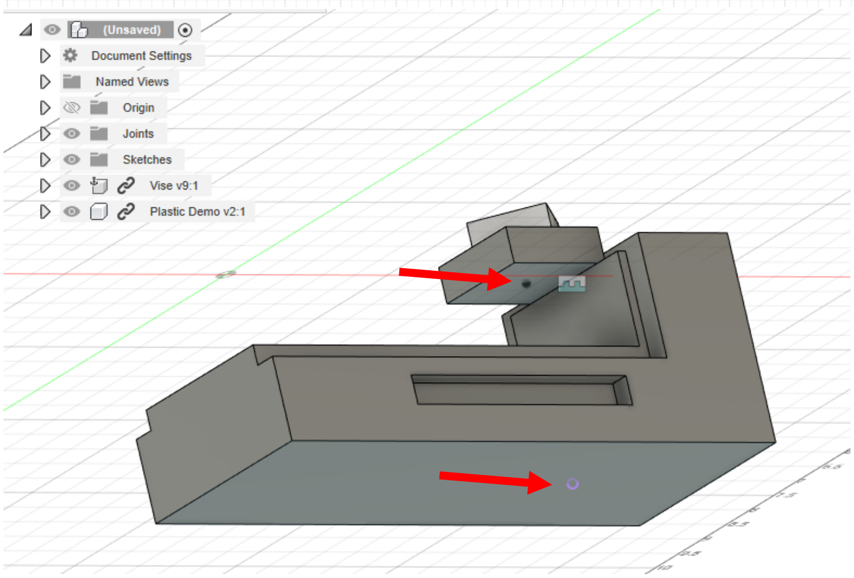


7. Create a new sketch on the bottom of the vise.

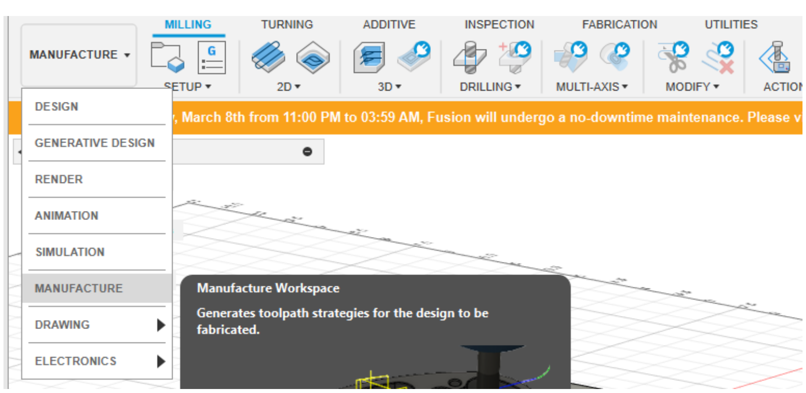


8. The demo part should have a center point sketch. If this is not visible, be sure that sketch visibility is enabled for this component. Use the project tool to project this point onto the bottom of the vise.

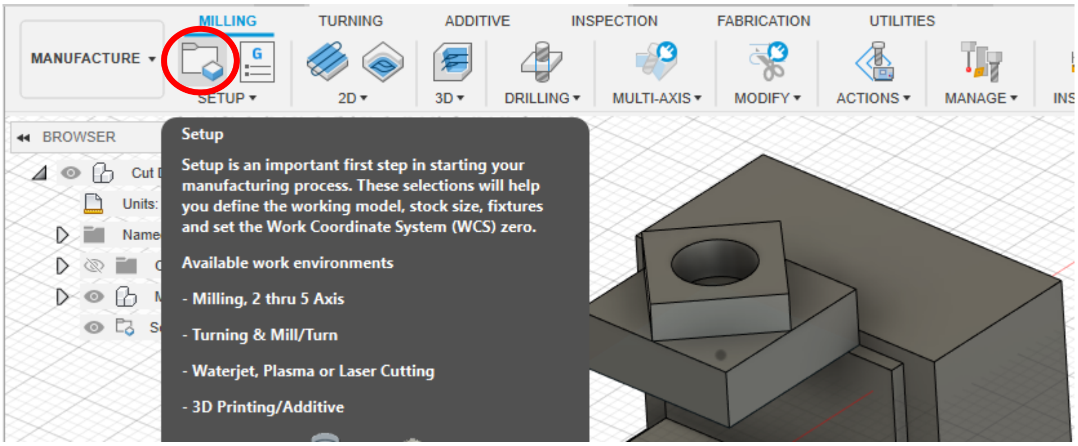




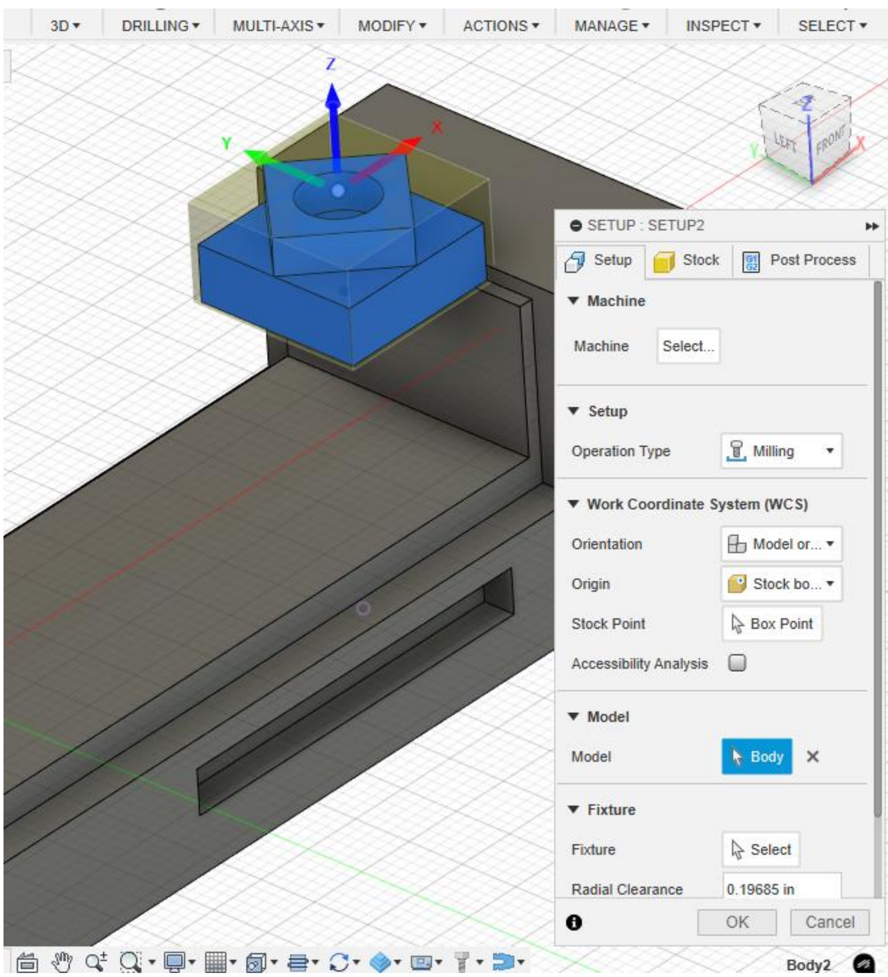
9. Change to the "Manufacture" workspace in Fusion.



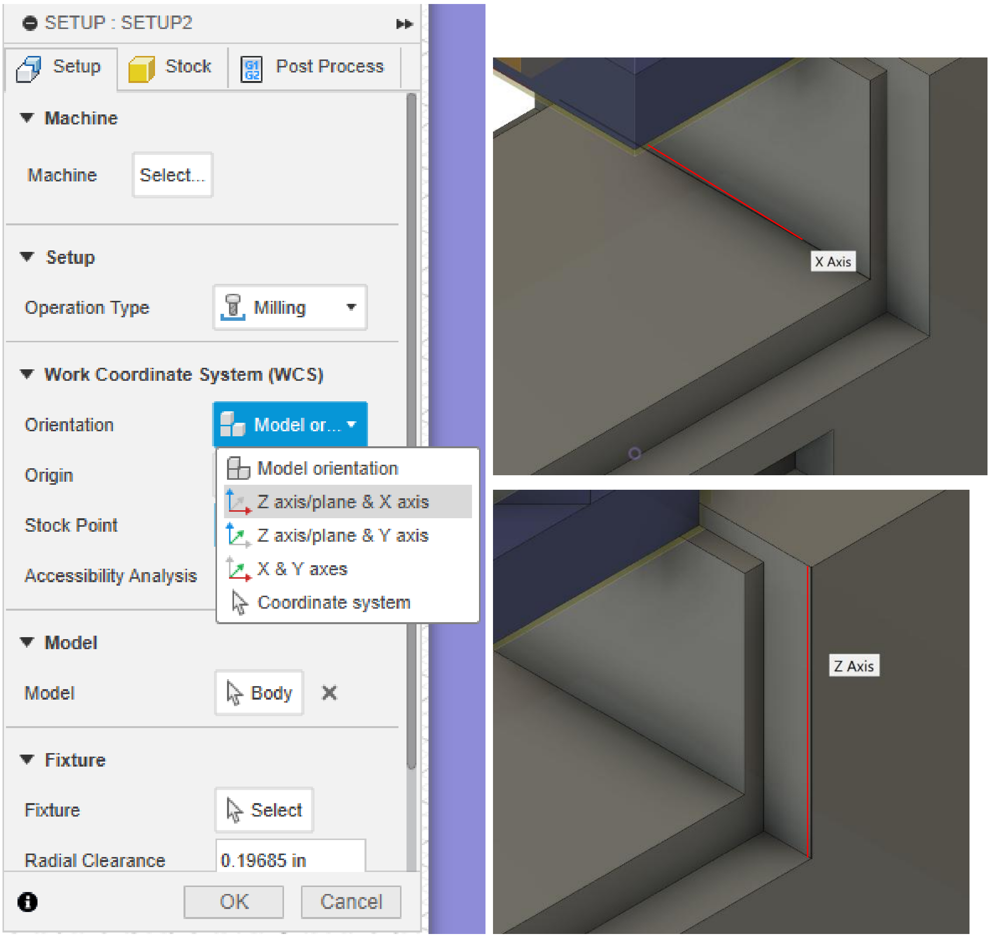
10. Once in the “Manufacture” workspace, create a new setup with the “Setup” button under “Milling”.



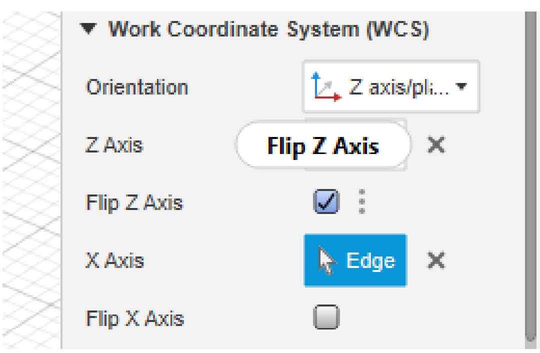
11. In the “Setup” tab, select the demo part for “Model”.



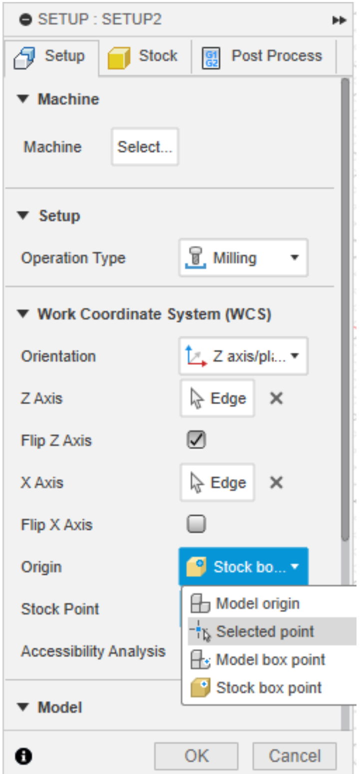
12. Under “WCS” and “Orientation”, select “Z axis/plane & X axis”.



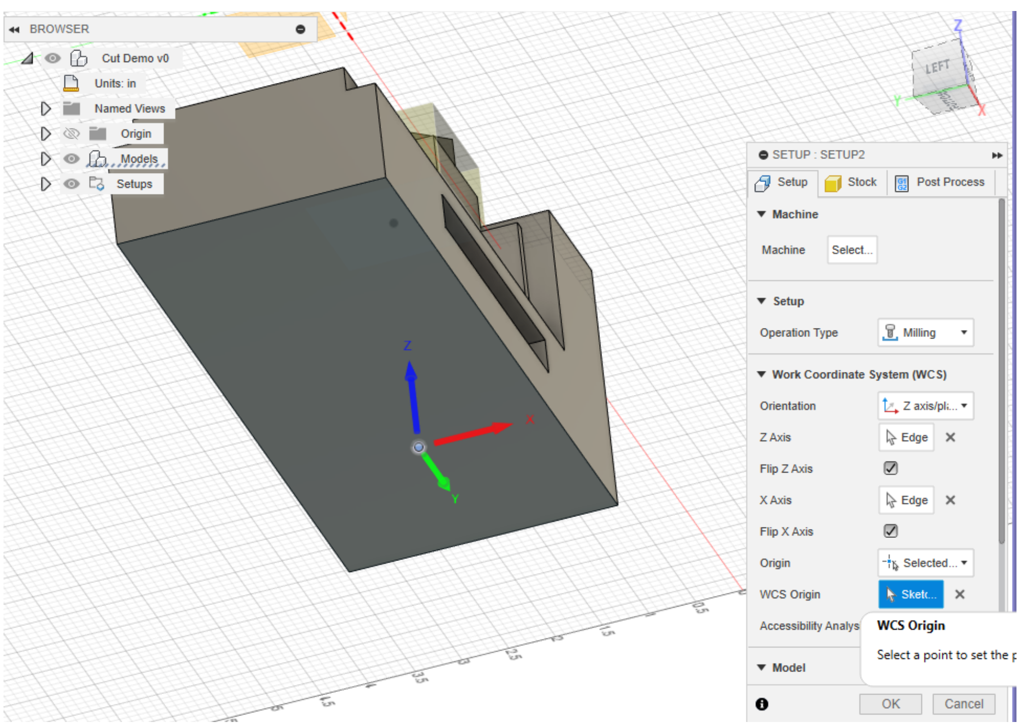
Choose an ‘up and down’ edge of the vise for the Z-axis and an edge across the vise jaw for the X-axis. If necessary, select “Flip Z Axis” to make the Z-axis point upward.



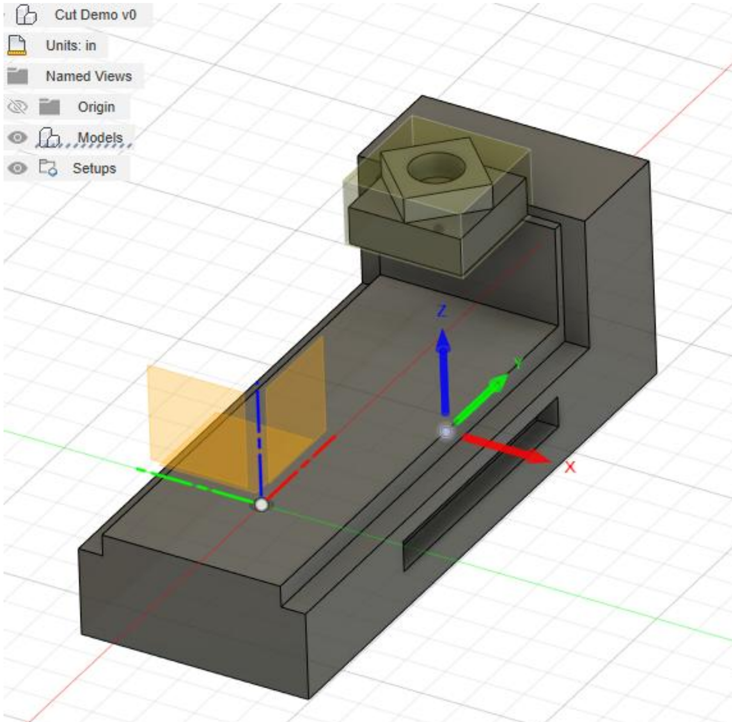
13. In the drop-down menu for “Origin”, choose “Selected point”.



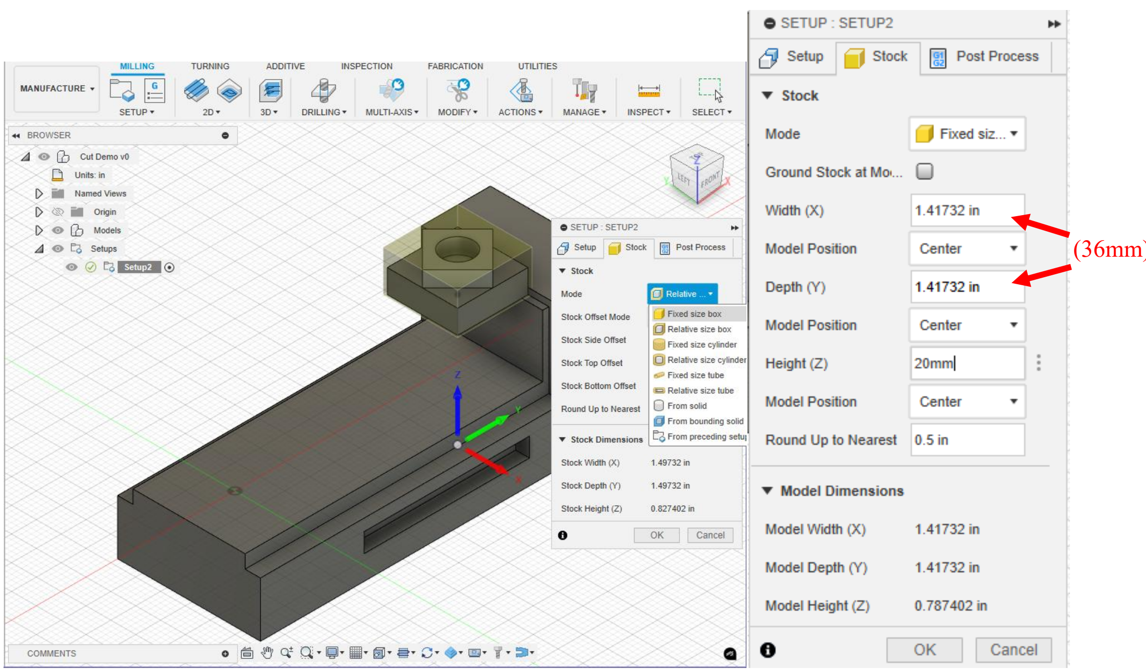
Select the point we sketched at the bottom of the vise. Again, sketch visibility must be enabled.



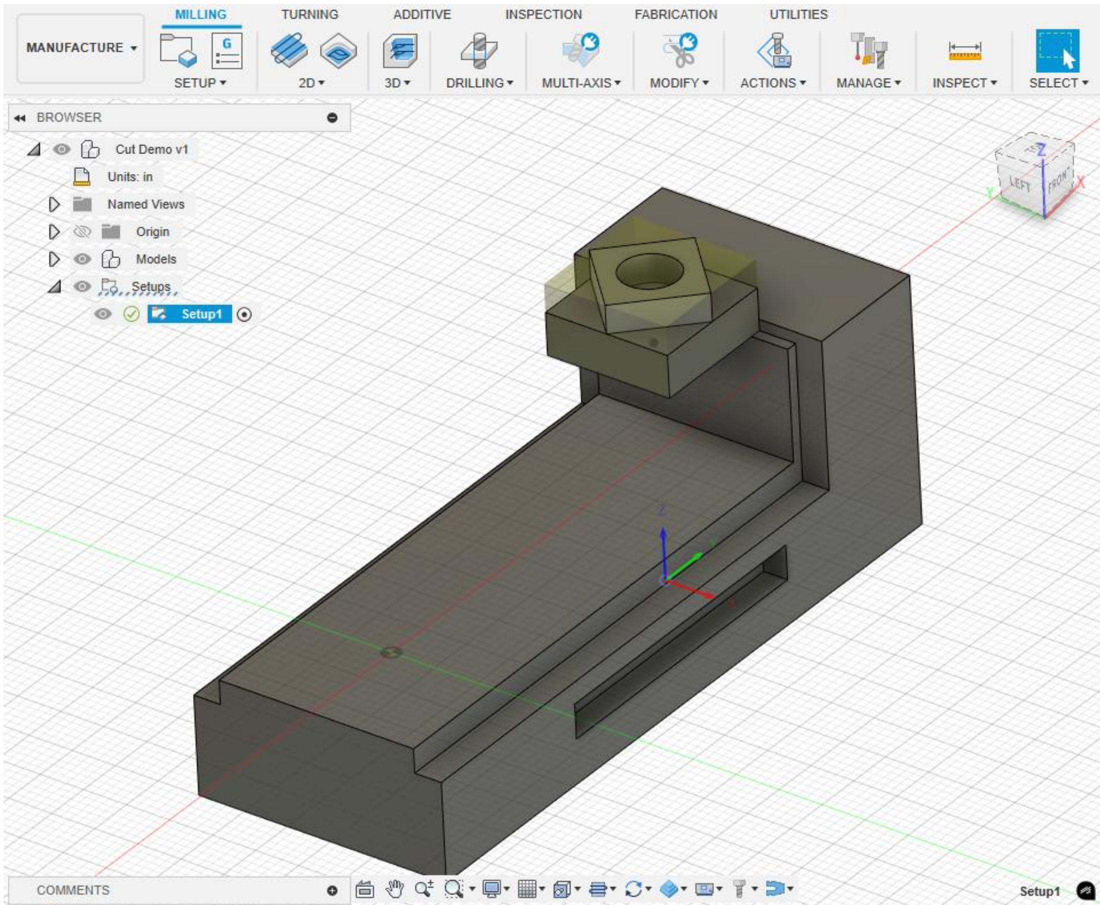
The result is shown below.



14. Move to the “Stock” tab. Choose “Fixed size box”. The dimensions of this stock, as mentioned, are 36x36x20mm. Be sure to input the correct units, as measured values may differ from active document units.

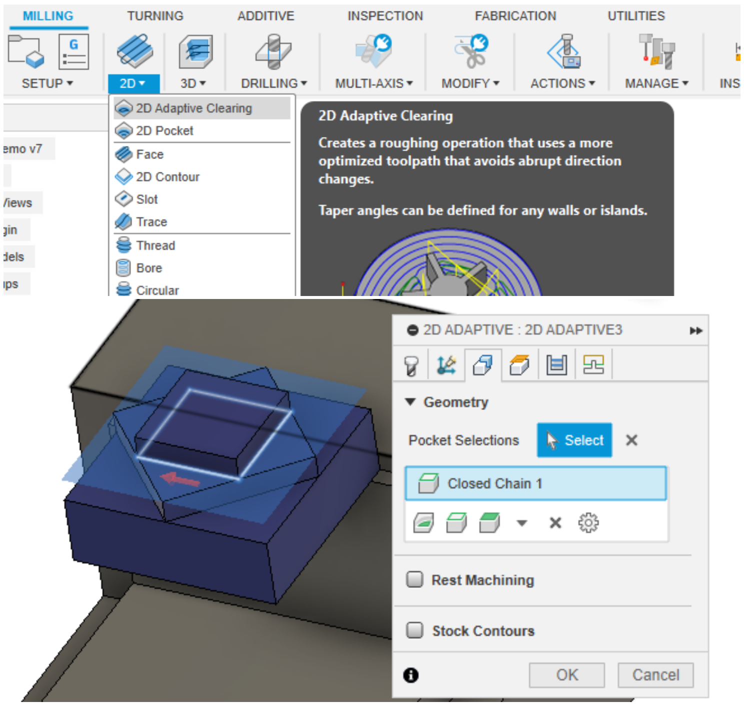


The result is shown below. The part is ready for milling tool paths.

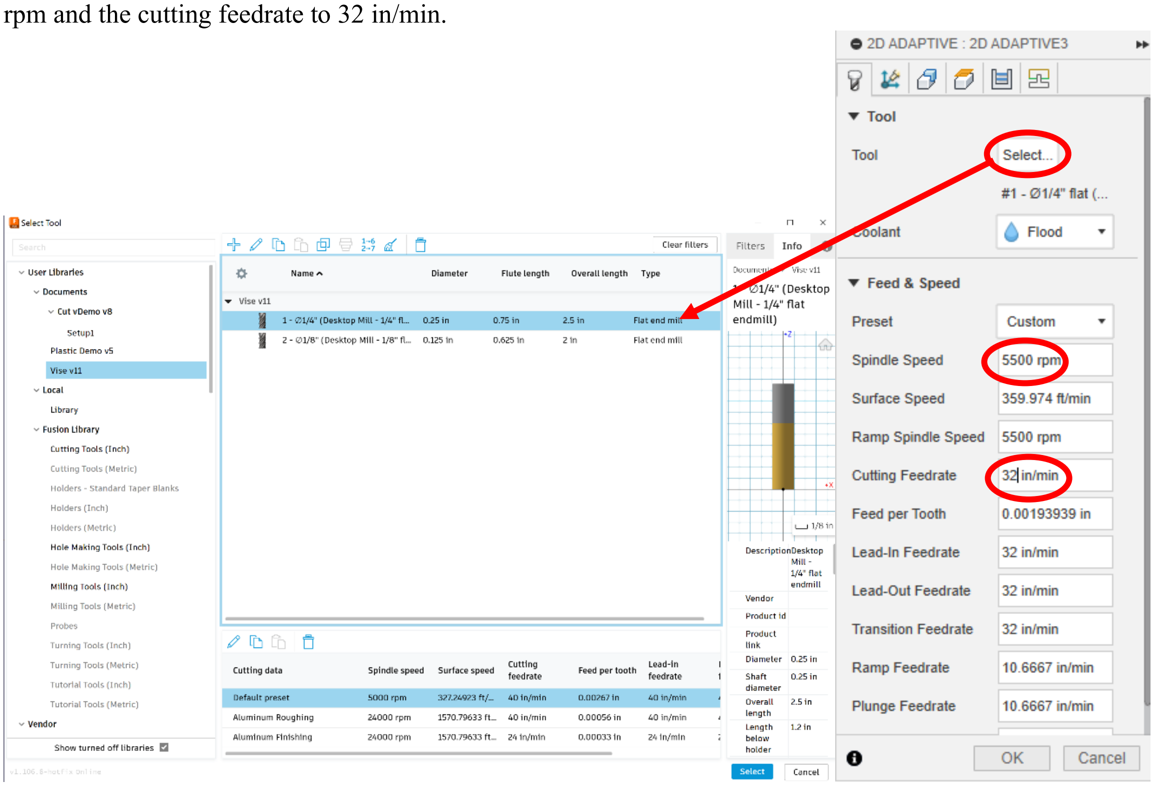


The following instructions are not related to coordinate placement and simply serve to complete the programming and manufacture of the square plastic demo part.

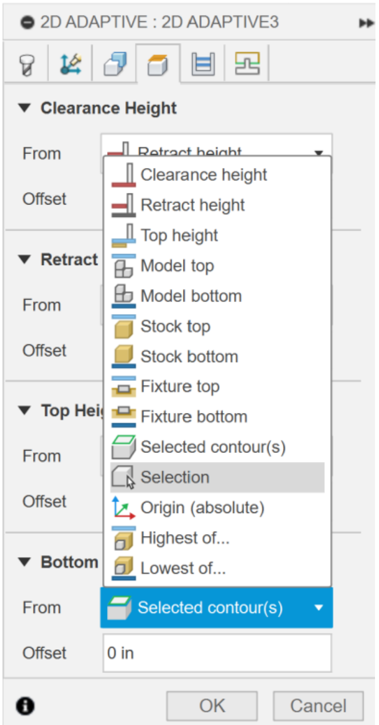
15. Create a 2D adaptive toolpath and select the bottom edge of the small square for "Geometry".

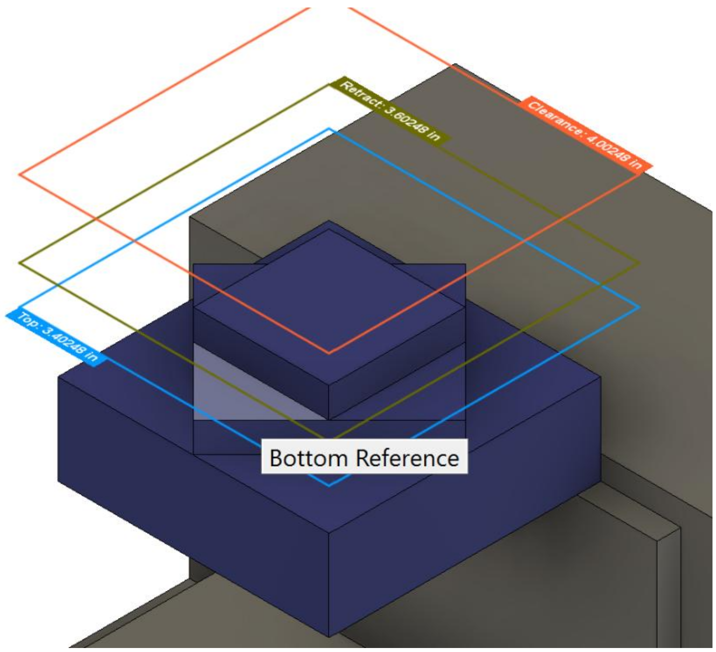


16. In the "Tool" tab, select the 1/4" flat endmill from the vise tool library. Change the spindle speed to 5500 rpm and the cutting feedrate to 32 in/min.



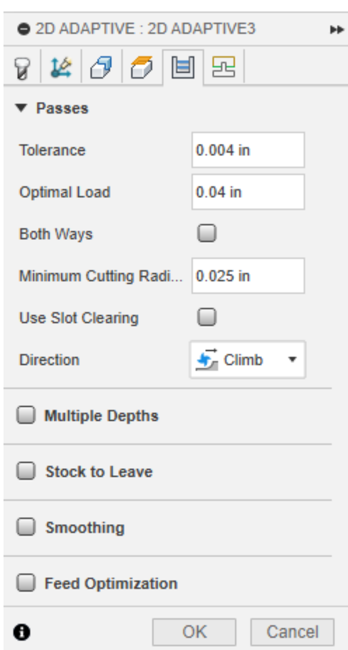
17. In the “Heights” tab, move to “Bottom Height” and change to “Selection”. Select the top face of the diamond as the bottom as seen.



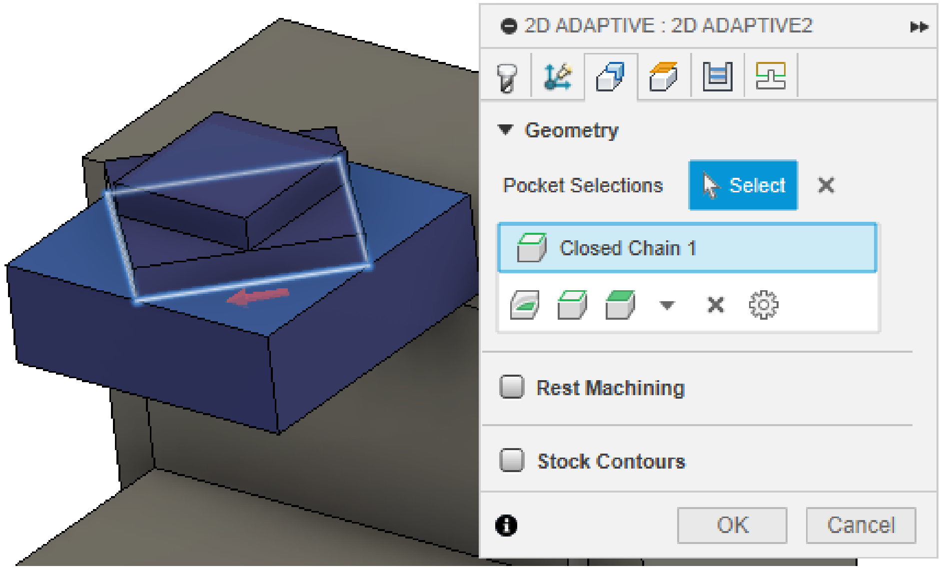


18. In the “Passes” tab, set the optimal load to 0.04 in. Turn off “Stock to Leave”.

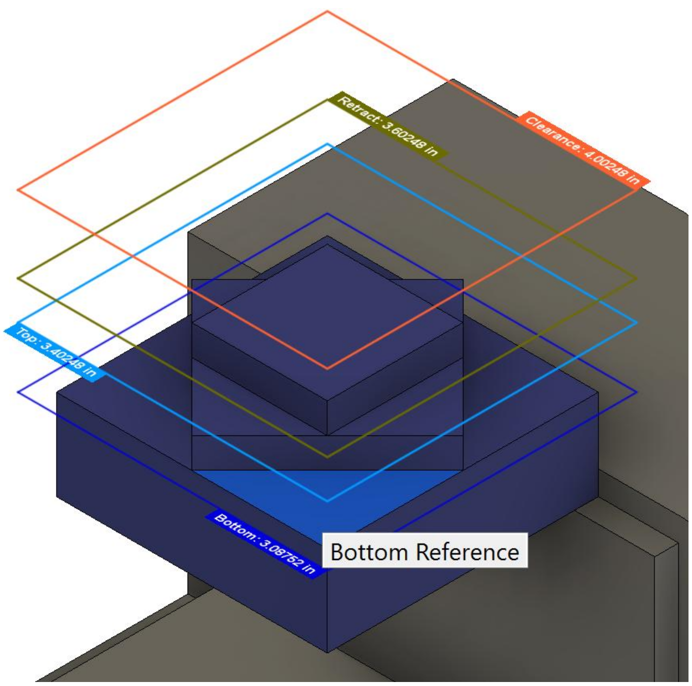
NOTE: These cutting parameters are for ABS plastic ONLY. Do NOT attempt to cut metal parts with these parameters.



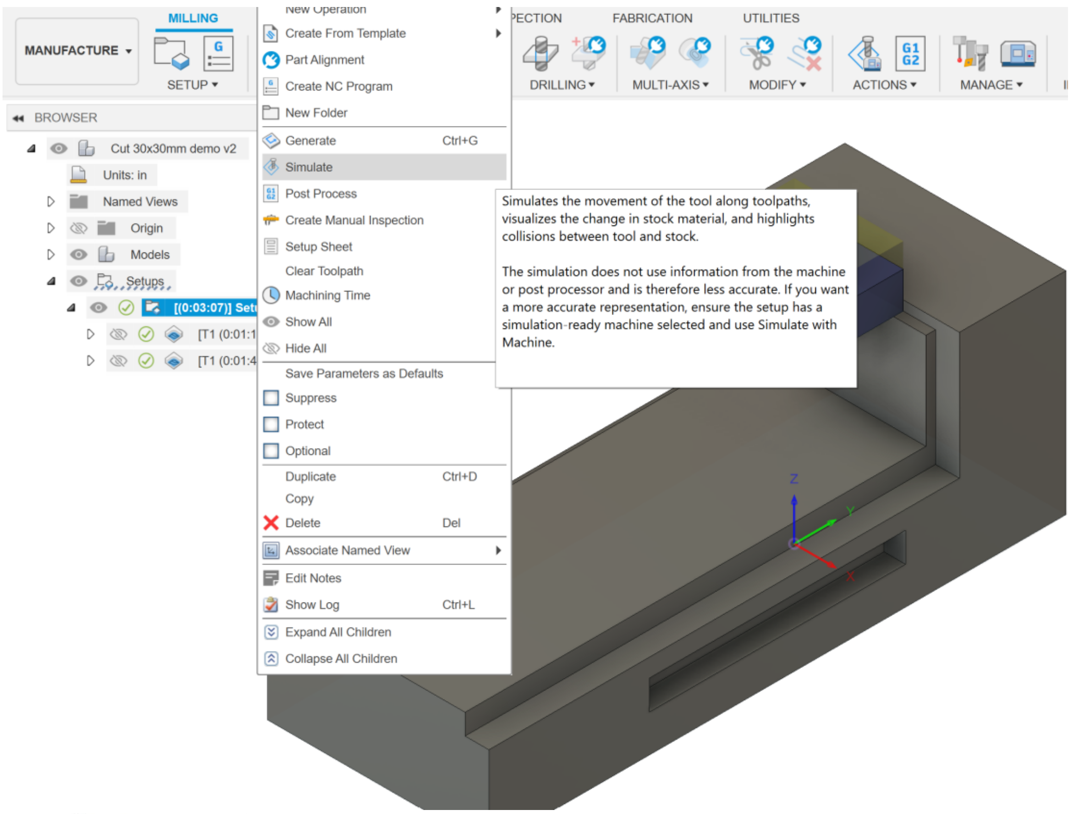
19. Copy and paste the toolpath, then change the geometry to the bottom edge of the diamond as seen below.

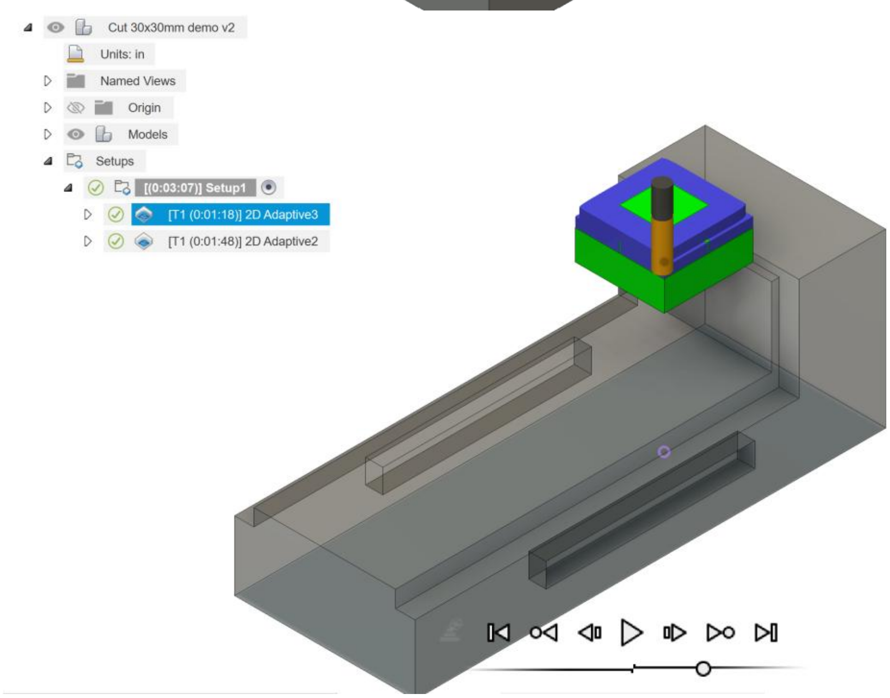


20. In the “Heights” tab, change the bottom surface selection to the bottom surface of the diamond as seen below.



21. Simulate the entire operation to be sure that no errors were made in programming.



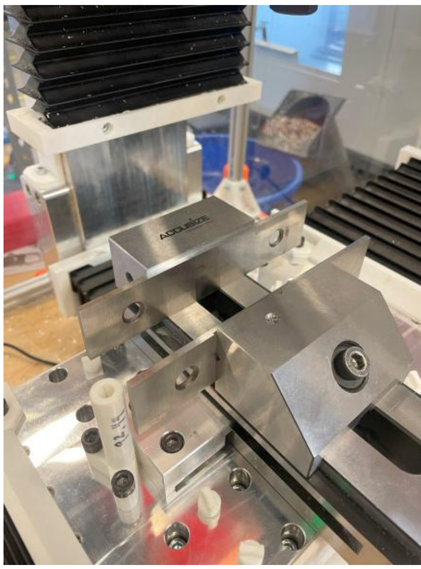
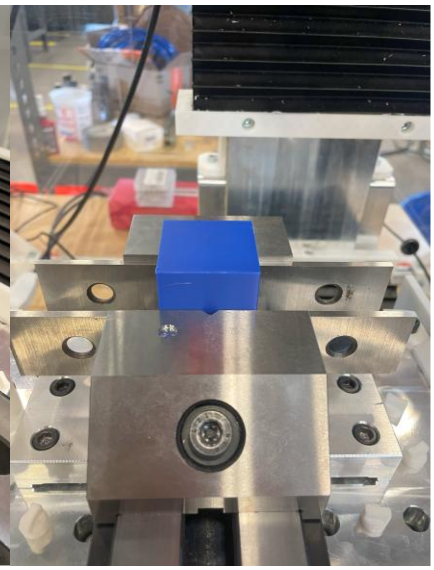


The part is ready for milling.

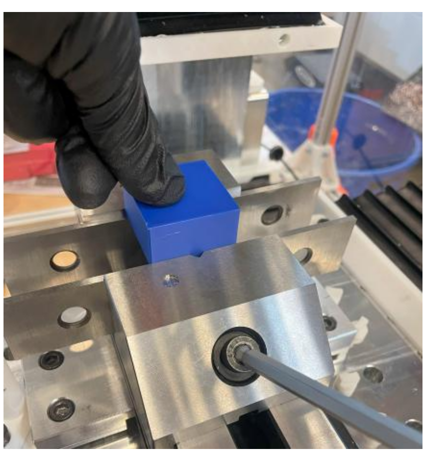
III: Setup and Probing

Due to the nature of this machine and the software used to run it, all toolpaths must be programmed with the Z-coordinate placed at the bottom of the vise (the top surface of the X-carriage). This will be covered in steps 1-13. The following steps show how to hold and probe a workpiece for milling operations.

22. Place the material stock in the center of the vice jaws on top of the necessary parallels. In the following pictures, a 36 x 36 x 20mm plastic block is placed on 1-1/4” parallels. This procedure will work for probing any **rectangular block**.

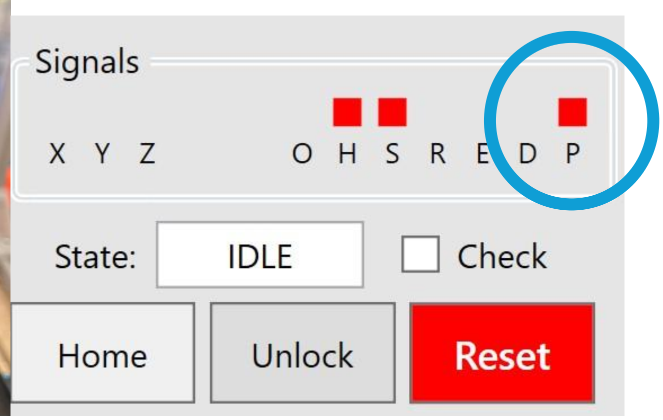



23. Use a 6mm Allen key to tighten the bolt in the center of the front vise jaw (if it is a different vise style, follow manufacturer instructions to clamp the part). This should close on the part and clamp tightly. As you tighten the vise, apply force down on the part. This will make sure that the part is evenly seated on the parallels. Check this by gently shaking the parallel; if it shakes noticeably against the part, try again (NOTE: an uneven surface may not seat perfectly against a flat parallel).
In certain cases, this vise style may struggle to clamp a part – if this occurs, see Troubleshooting *at the end of this document.*



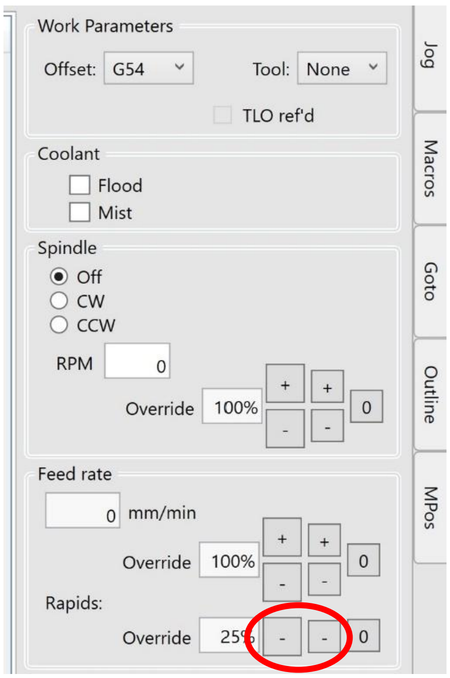
24. Gather the probe and attach the probing tip. Slide the probe into the spindle with a ¼” collet and tighten by hand. This should hold the probe safely. Be sure that the spindle is turned off by the emergency stop or door switch.

25. With the electrical box on and connected to both the computer and machine tool, plug the probe into the electrical box and GENTLY move the probe tip. The light should change from green to red and the computer software should show a “P” alarm while the probe tip senses contact. Both conditions must be met to ensure probe operation.

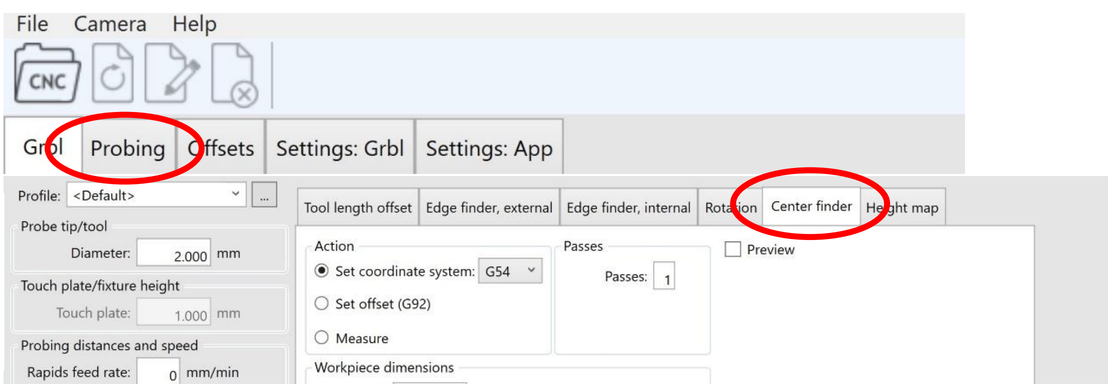


26. Carefully jog the probe over the part. Do not allow the probe tip to touch any surface while jogging, as excessive force or deflection can permanently damage the probe tip. Carefully jog the probe until it is as centered as visually possible over the part and 5-7 mm from the top surface of the part.

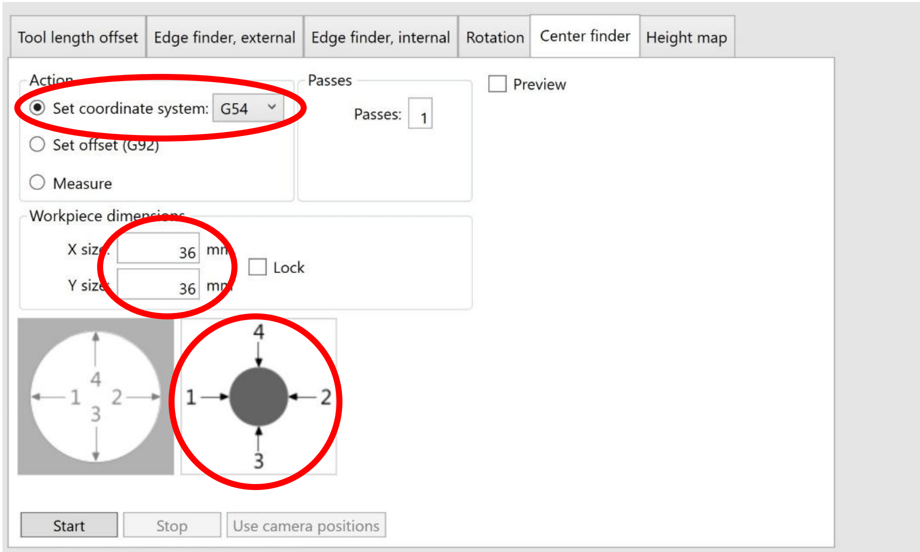
27. Before probing, reduce the rapids rate to 25%. Rapid movement rate can be changed with the minus sign buttons in the "Rapids" section shown below.
NOTE: this is an important step. This is done so that any errors in probing can be safely caught before any damage is done.



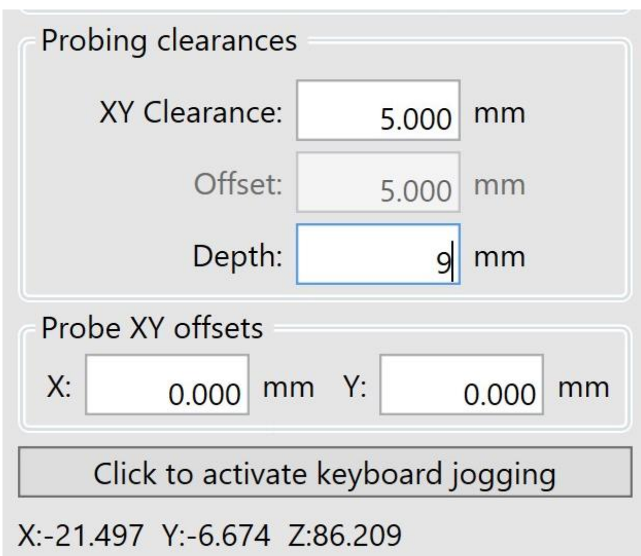
28. Move to the "Probing" tab in the ioSender software. From there, move to the "Center finder" option.



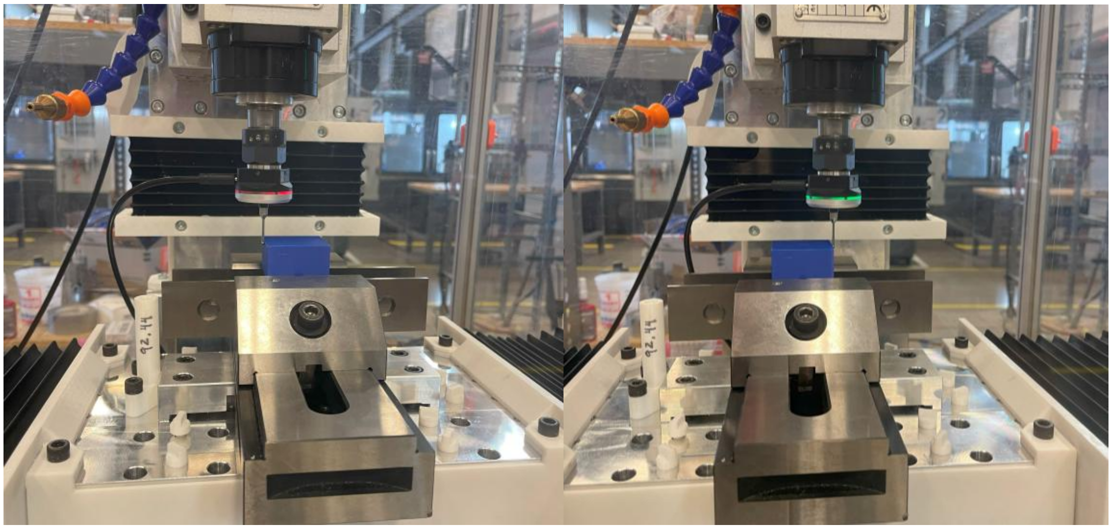
29. Type in the workpiece dimensions **in millimeters**, then select the external option. Select the external bore option for probing the outside surfaces of the stock. NOTE: The "Lock" box will need to be unchecked for a rectangular part.



30. In the "Probing clearances" section, set the probing depth. This value will determine how far down the probe travels to touch the workpiece. It is equal to the distance between the current probe tip position and the place on the part that will be probed- it may be necessary to roughly measure this distance before probing to ensure correct positioning and mitigate errors.

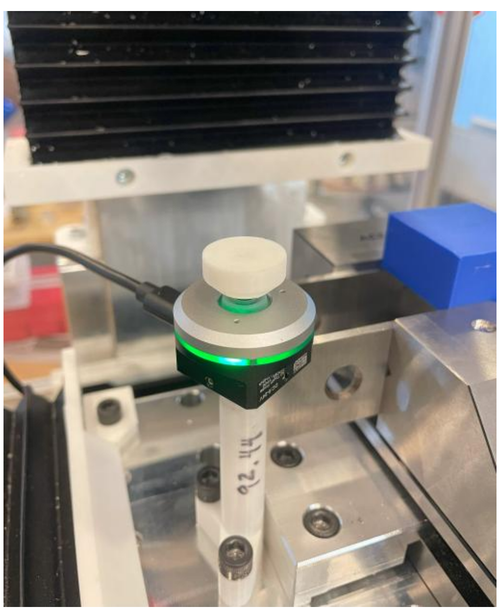


31. Double check inputs. Errors are much more difficult to catch quickly in operation. Press “Start” and watch the probing cycle very closely. Emergency stop the machine if anything unexpected occurs and safely try again. The machine should move across the part to touch each side once. This will set the X and Y offsets for a coordinate system centered on the rectangular block.

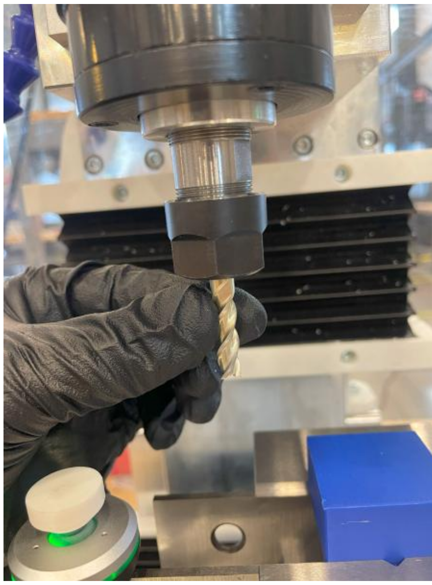
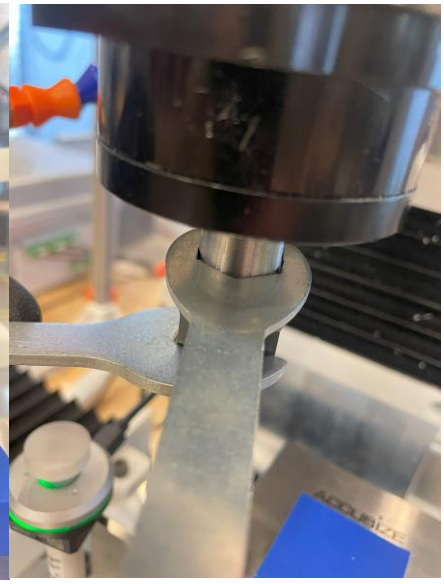


32. We will now set the Z height for the coordinate system. This is based on the surface of the X-carriage and will include the tool offset. Jog the probe safely away from the part and remove it from the spindle.

33. Gently replace the probe tip with the probe touch plate. Insert the shaft of the probe into the holder on the left side of the vise. If the signal cable was disconnected, plug it into the probe and test again to be sure the signal is working.

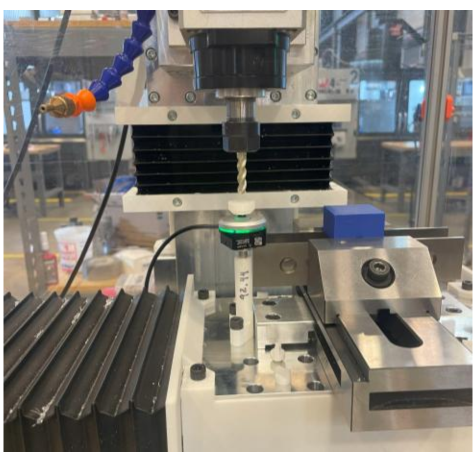


34. Press the emergency stop. **Never handle sharp tools in the spindle without the emergency stop engaged.**
    Insert the tool into the spindle at the desired length and secure tightly using the provided wrenches. This may require replacing the collet if the tool is not the same diameter as the probe shaft. In this case, a ¼” endmill is used.

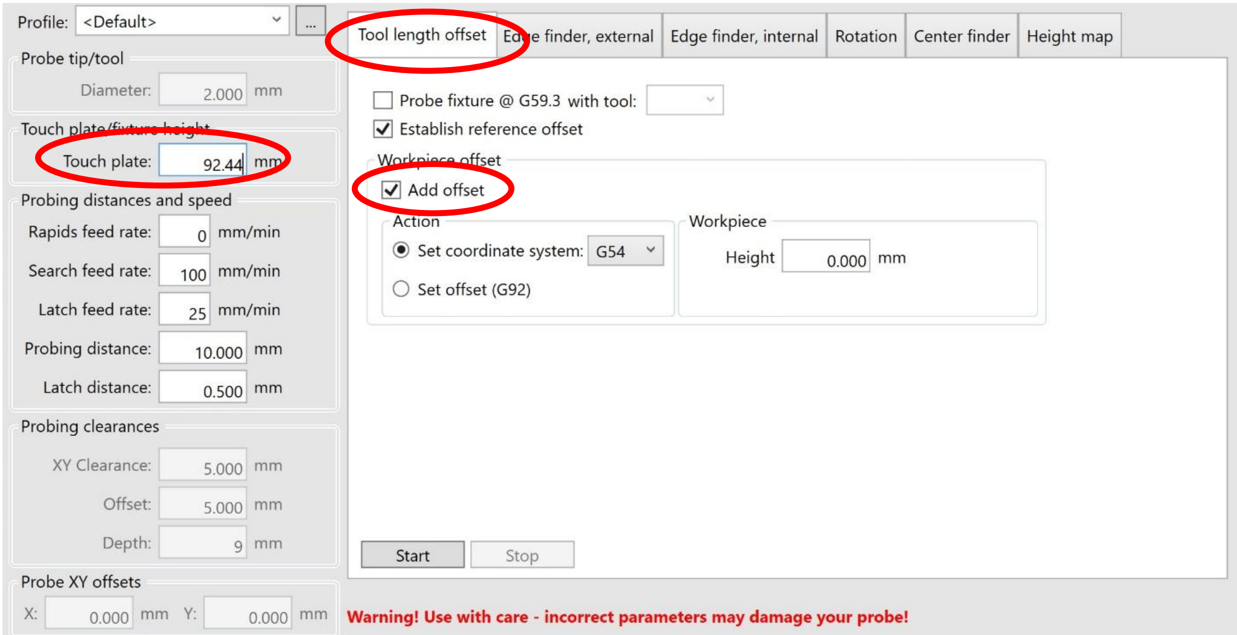
 

35. NOTE: Trying to jog any axis while the emergency stop is engaged will move the coordinate system without moving the machine. Do not jog until the emergency stop is disengaged.

    Carefully jog the spindle until the tool rests approximately 5mm above and centered on the touch plate.

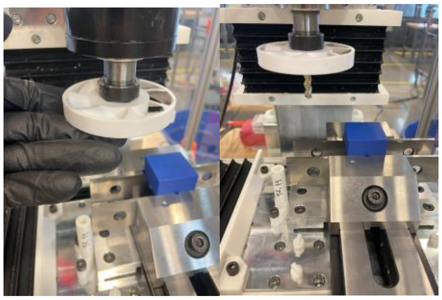


36. Move back to the “Probing” tab and select “Tool length offset”. Check the “Add offset” box and set the touch plate height to 92.44 mm (or the otherwise determined accurate distance from the touch plate to the X-carriage surface – see *Troubleshooting*).



37. Double check all parameters and press start. The tool should slowly approach the touch plate and gently press it twice. This will set the Z-coordinate at the bottom of the vise / top of the X-carriage.

38. Jog the tool out of the way and remove the probe completely from the enclosure. If the machine tool is not equipped with compressed air, slide the plastic chip fan onto the spindle over the tool as seen below.



39. Jog the tool so that it starts the operation above the part.
BEFORE RUNNING ANY PROGRAM THE ENDMILL MUST BE ABOVE THE PART.

Be ready to feed hold the machine IMMEDIATELY after pressing cycle start. This will allow the spindle to reach the desired speed before milling begins (this usually takes no longer than 5 seconds – the spindle should be making a constant sound and not "ramping up").
ALWAYS ALLOW THE SPINDLE TO GET UP TO SPEED BEFORE ENTERING A CUT.

After pressing feed hold and letting the spindle reach its target RPM, you can resume the operation with cycle start.

The machine tool should now be ready to run a program.

IV: Operator Checklist

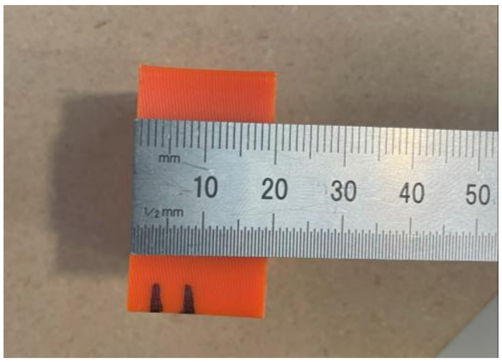
OP10	OP20	Check
☐	☐	Entire operation simulated in Fusion 360
☐	☐	Workpiece is properly secured
☐	☐	X and Y offsets established
☐	☐	Tool offset established
☐	☐	NC program is the correct version

V: Cutting and Checking Demo Part

NOTE: Read through this entire section before starting the machining operation. Several steps require quick reactions to machine movements and familiarity with the following step.

While these parameters should work for ABS plastic, it is possible that any material (*especially* plastics) can “gum up” the tool when milled. Follow all safety guidelines CLOSELY.

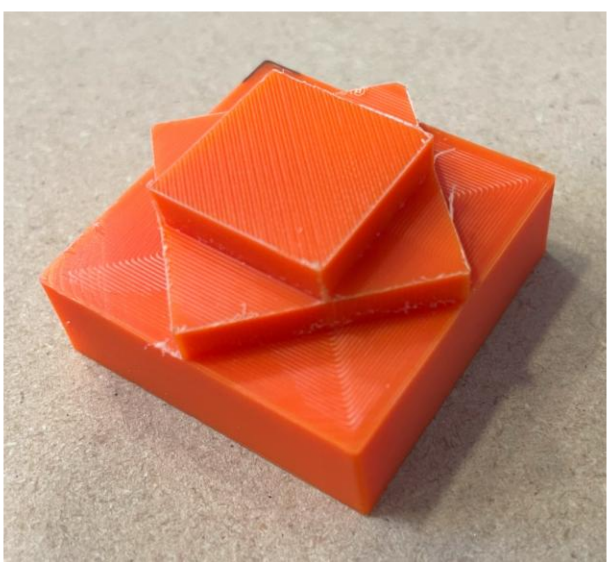
40. A good way to double-check probed offsets on this part is with physical marks on the part. Draw two lines on the part at roughly 4mm step downs.



41. After completing steps 1-37, start and feed hold the machine to allow the spindle to get up to speed. Reduce rapids to 25% for safety. If everything is correct, resume the operation and be prepared to feed hold.

42. Feed hold the machine as it approaches the first cutting pass. The tool tip should be very close to the marked line. Do this again as it approaches its second pass.

43. Allow the machine to safely finish the operation.



44. To check machine performance, measure the machined sides of the demo part. Nominally, the smaller square is 18 mm wide, and the diamond is $\sqrt{648}$ mm (about 25.46 mm) wide. The measured values should be quite close, as seen below. The depth of each stepdown should also be measured. This value is nominally 4mm.

VI: Troubleshooting

a) <u>Vise will not clamp part tightly after tightening.</u>

The style of vise shown in the instruction documents does not use a linear screw for clamping; instead, the moveable jaw clamps using a series of round openings and moveable pin. This creates “dead” spots in vice clamping range. One way to overcome the issue is to choose a workpiece or orientation with a slightly bigger or smaller width. This is done for the crank-slider parts using soft jaws.

Another solution (that may take some practice to get right and should not be necessary for educational parts) is to loosen the moveable jaw until it is free, press it against the workpiece, then “lean” the pin inside further backwards using the Allen key on the bolt. The goal of this method is to reach the next round opening. **This method will work but takes careful observation and understanding of the vise mechanism. It is easy to jam the vise or back the bolt out completely. This is not necessarily bad for the vise but is likely out of the scope of a basic CNC educational program.**

b) <u>Milled parts are “off” by a consistent amount in the Z-direction.</u>

First, check programming files to be sure that the coordinate system is exactly on the intended plane.

It is possible (and in many ways likely) that the provided touch plate height of 92.44mm is incorrect for your machine setup. This can be fixed. Note that this solution does NOT apply if the measured value is incorrect relative to another cut; that is, if the machine makes a stepdown within the same program (no changes in probing/coordinates/setup) and the stepdown is noticeably wrong, there is a different problem. In this scenario, the entire coordinate system is “shifted” up or down by an incorrect Z offset, and parts are thus *consistently* off by a single value.

The simplest way to correct this is by probing a test part and making a facing pass of known depth z. Measure this depth after the cut, compare to the nominal value, and adjust the nominal touch plate height by small increments as necessary. Test again to be sure that the offset increased milling accuracy.