

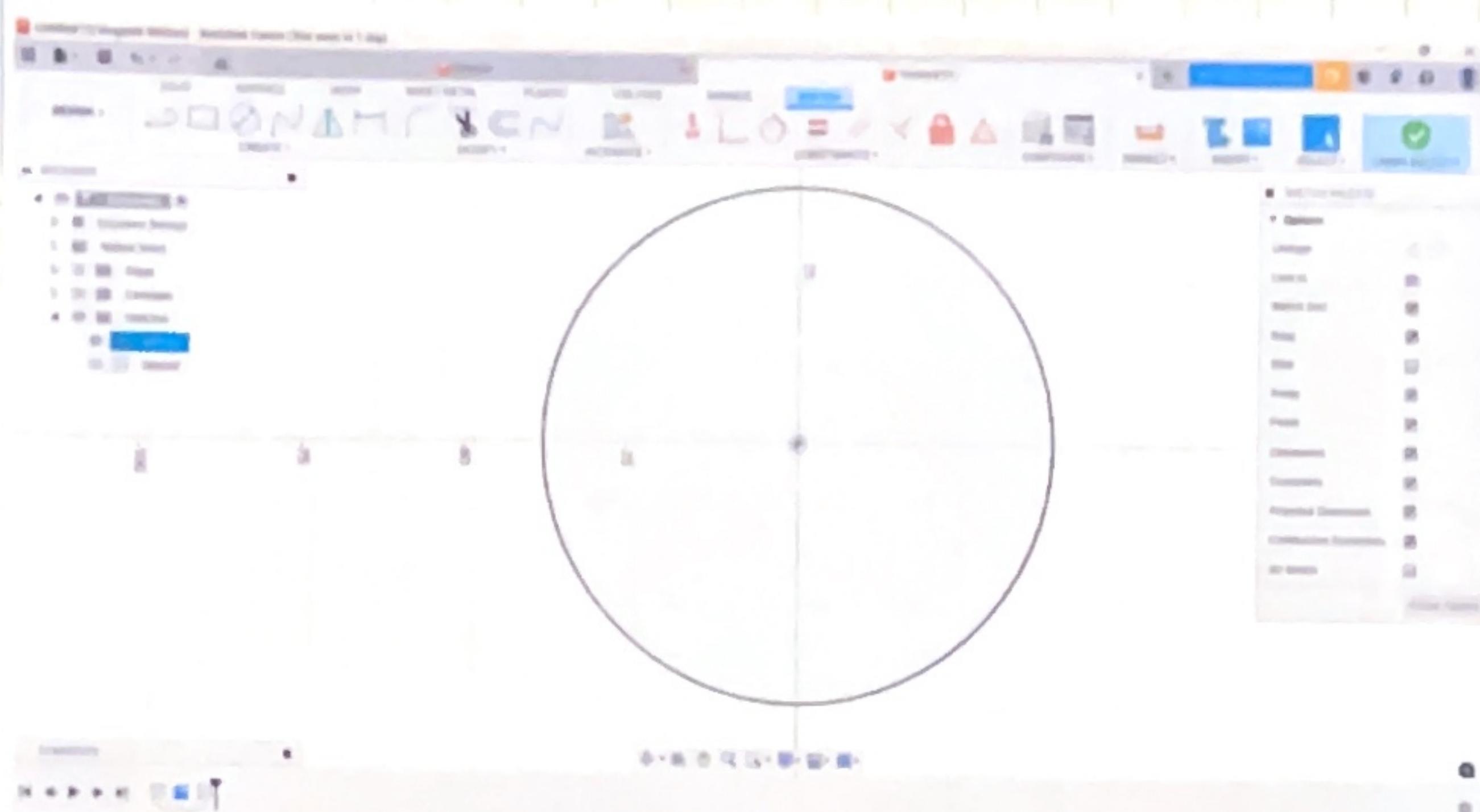
07/01/25
Wesley
code

CNC Basics; V-carve

129

In fusion 360:

Design



Start with a sketch of an outline you want to continue your design to.

I imported the utilities logo. On a new sketch Insert → Canvas → Choose file from computer.

Use the sketch tools to trace the design.

I found a font similar to the font. The spline tool was useful for curves.

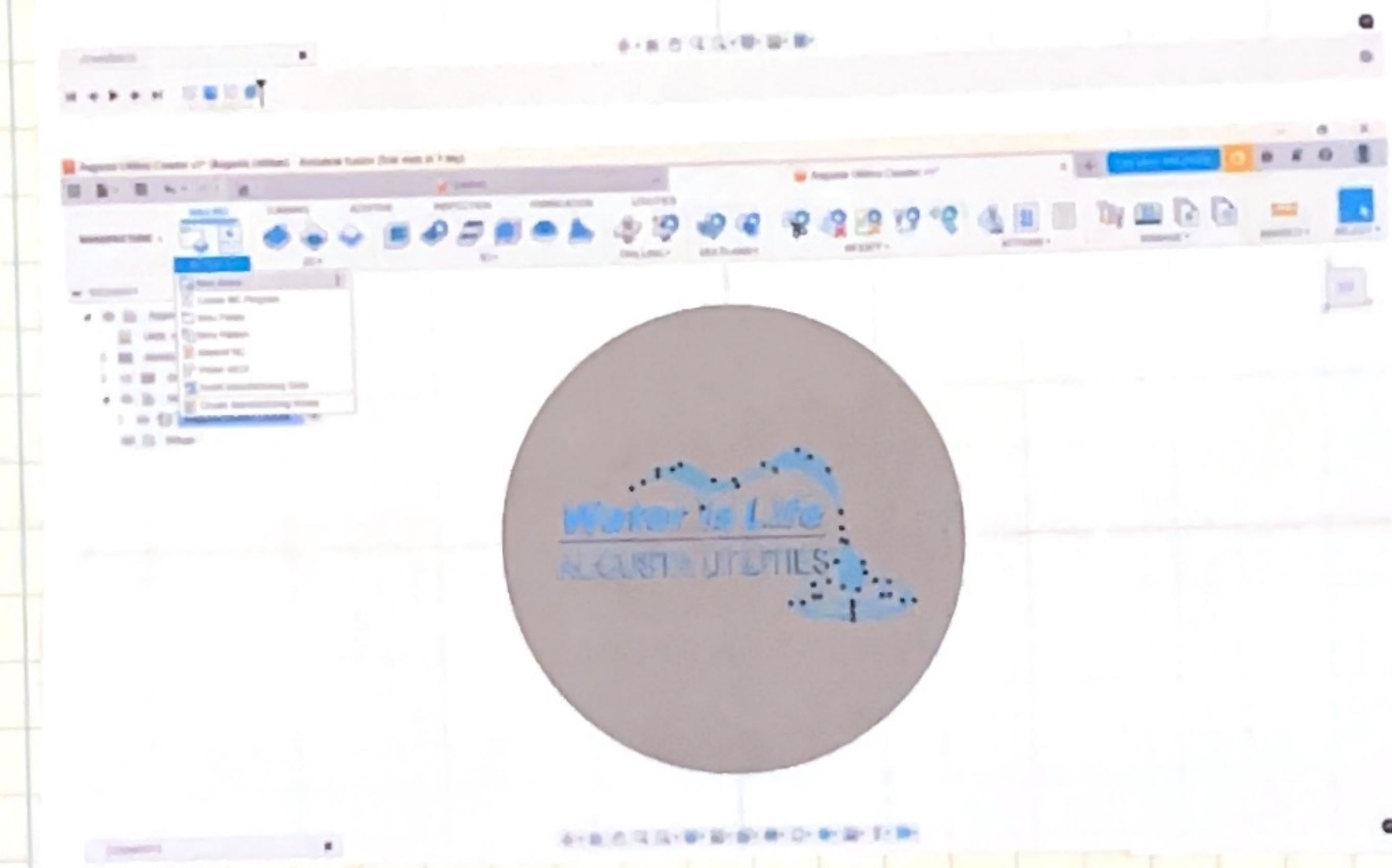


I extruded the outer ring downward so that the logo sketch is still on top.



Now we are ready to get a program from fusion.

Switch to manufacture.



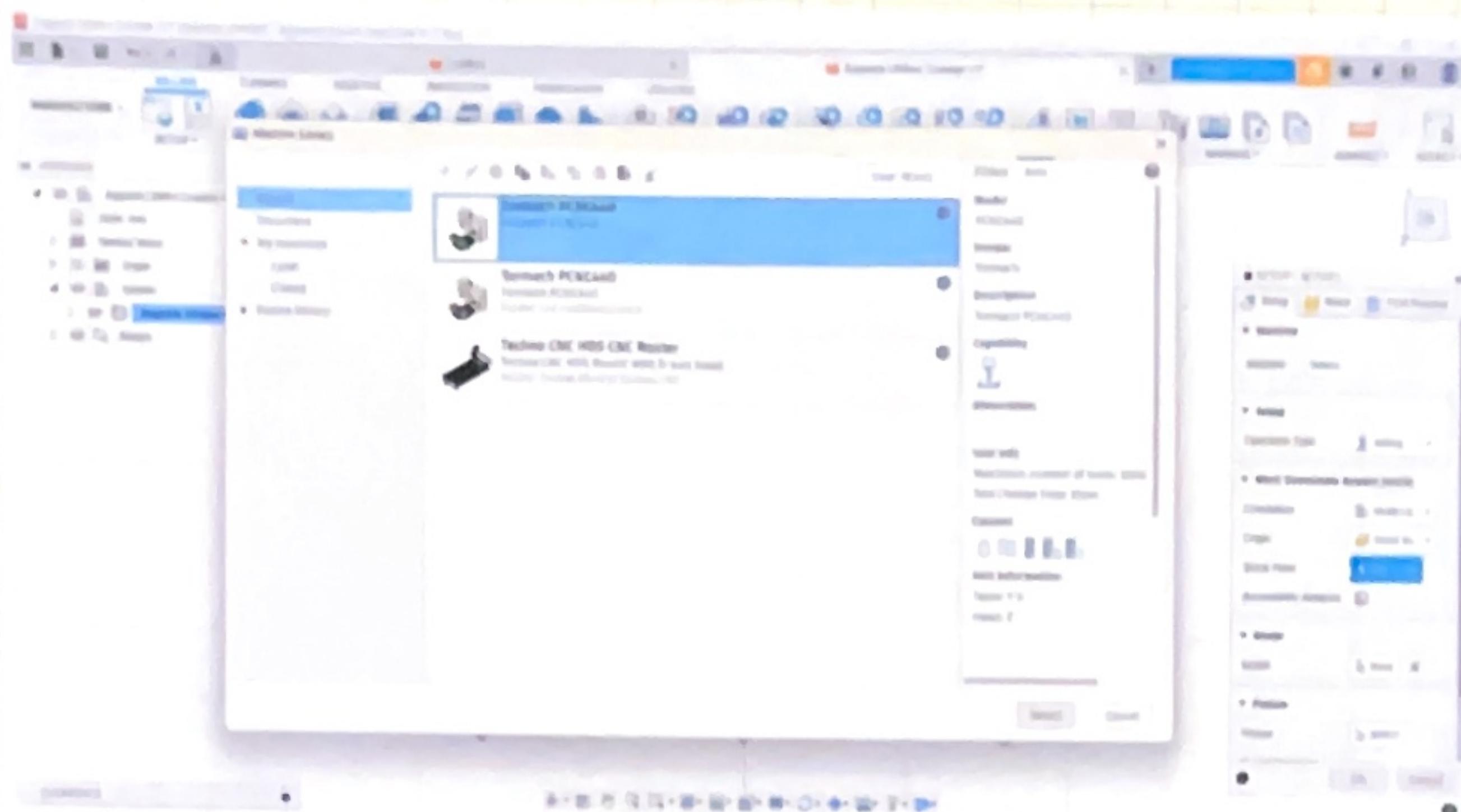
Add a new setup.

07/01/25
Wesley
Cook

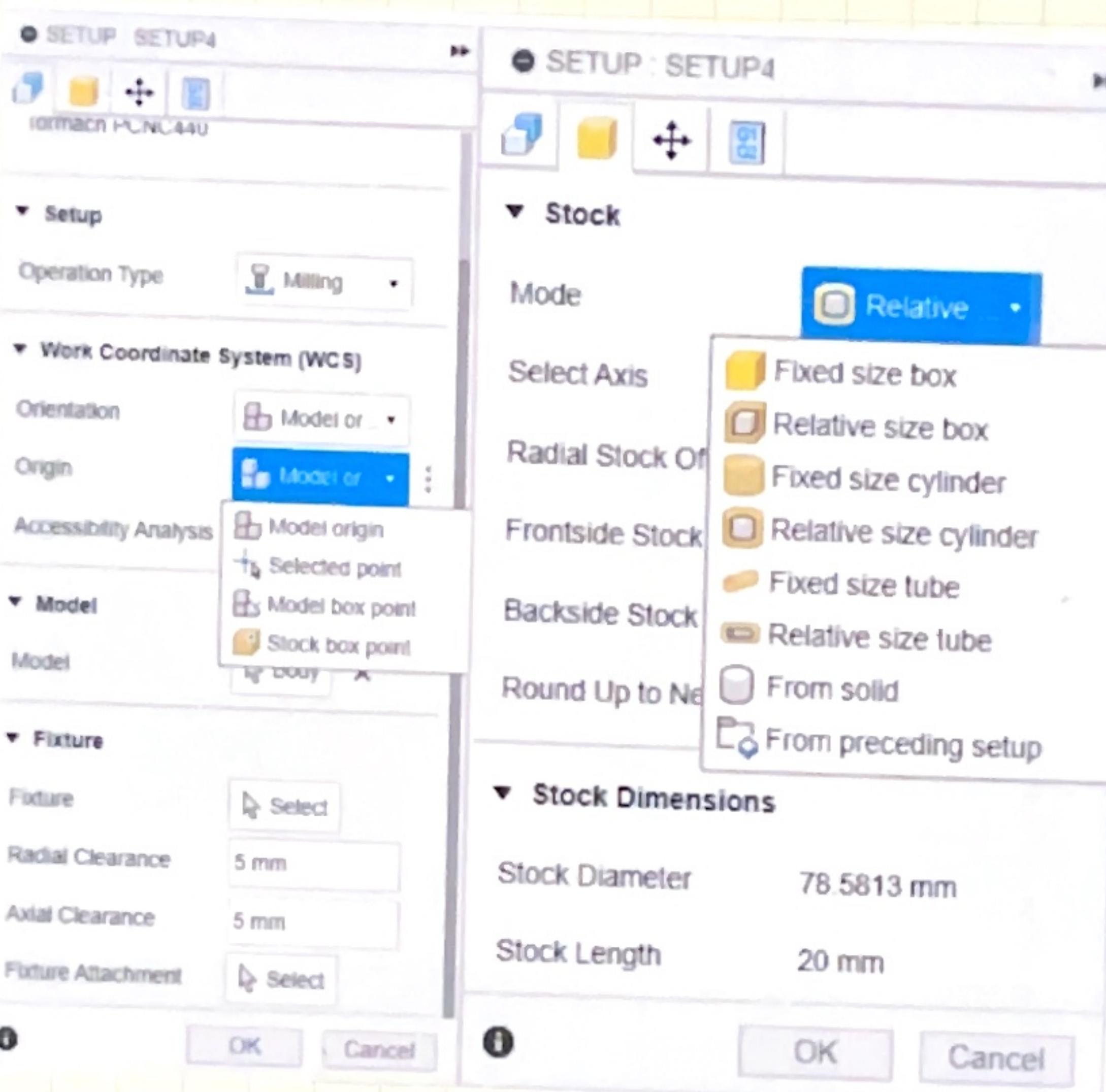
Manufacture

CNC Basics: V-Carve

131



Search and
Select the
Tormach L40



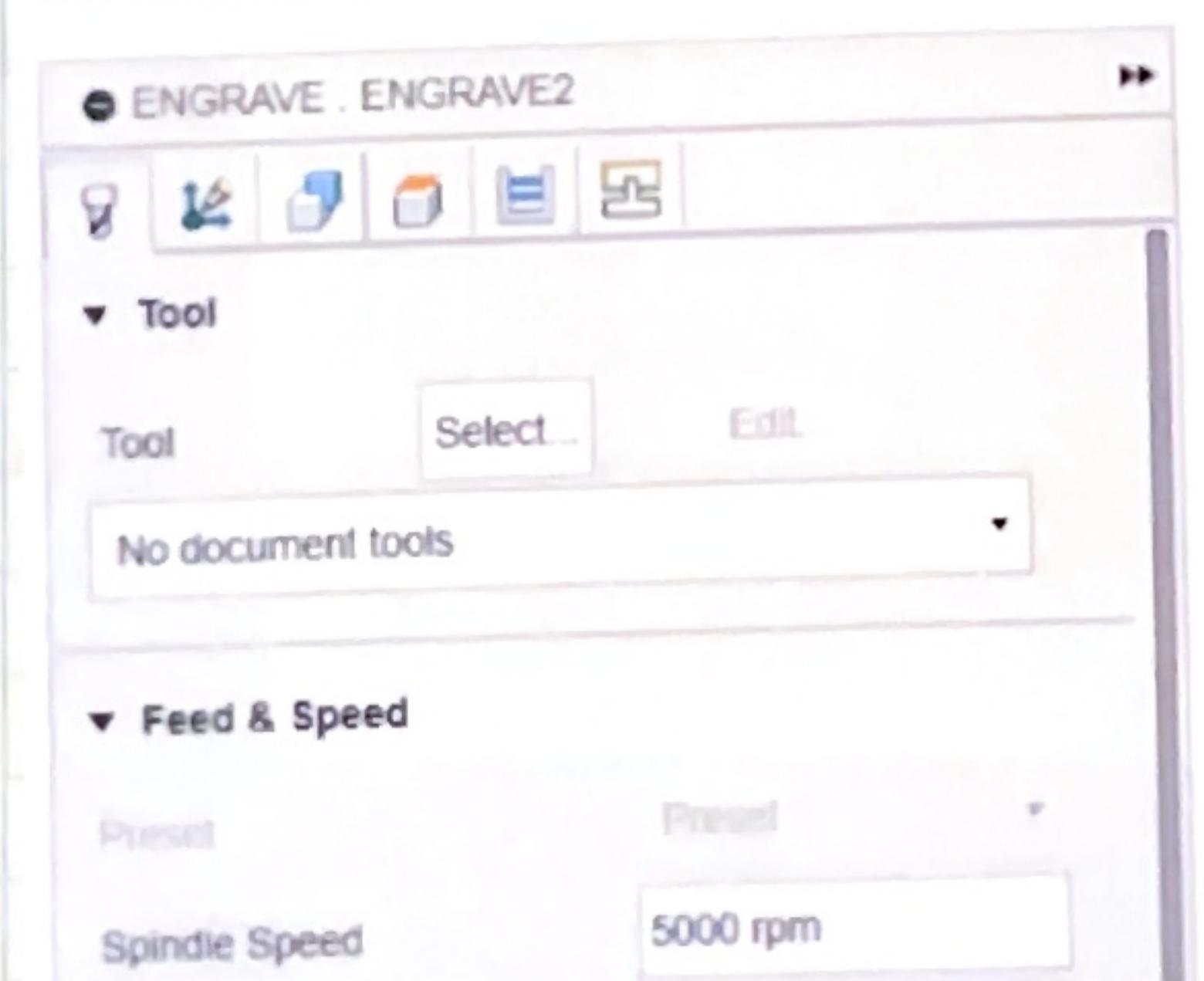
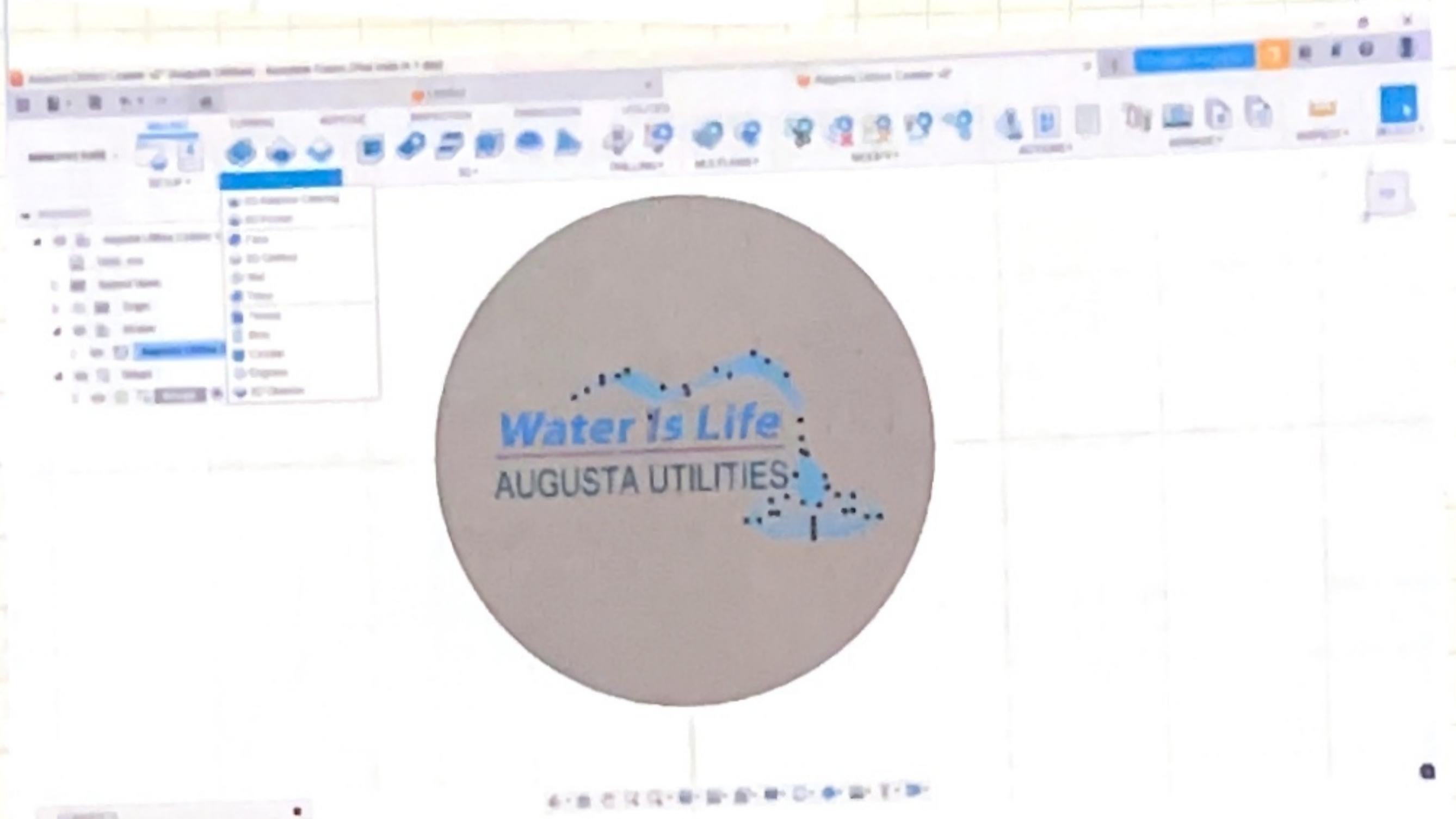
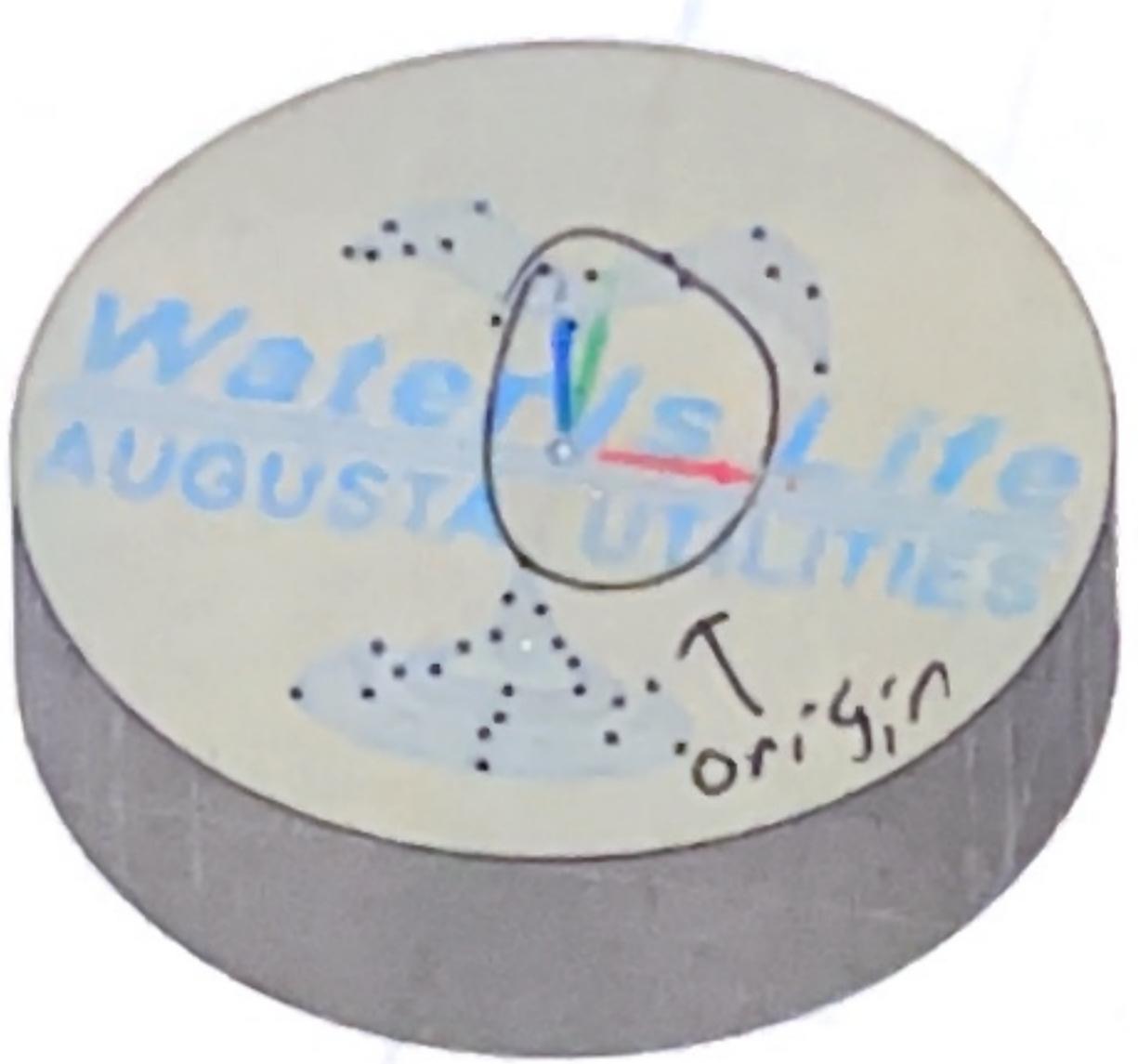
Examine the setup.

I Set my stock to
be the size of
the outline I
modeled. You may
want some offset
depending on what
you are doing.

Additionally, I set
the origin to the
middle of my
circle.

You can use model
origin if you want
to design based on
that, or just
select point and
tell it exactly
where.

For square starting shapes,
I would use one of the corners
as the origin.



Next Select
the engrave
tool from the
"2D" dropdown

Select the sketch
face that you
traced.

Select a tool

07/01/25
wesley
oactu

CNC Basics: V-Carve

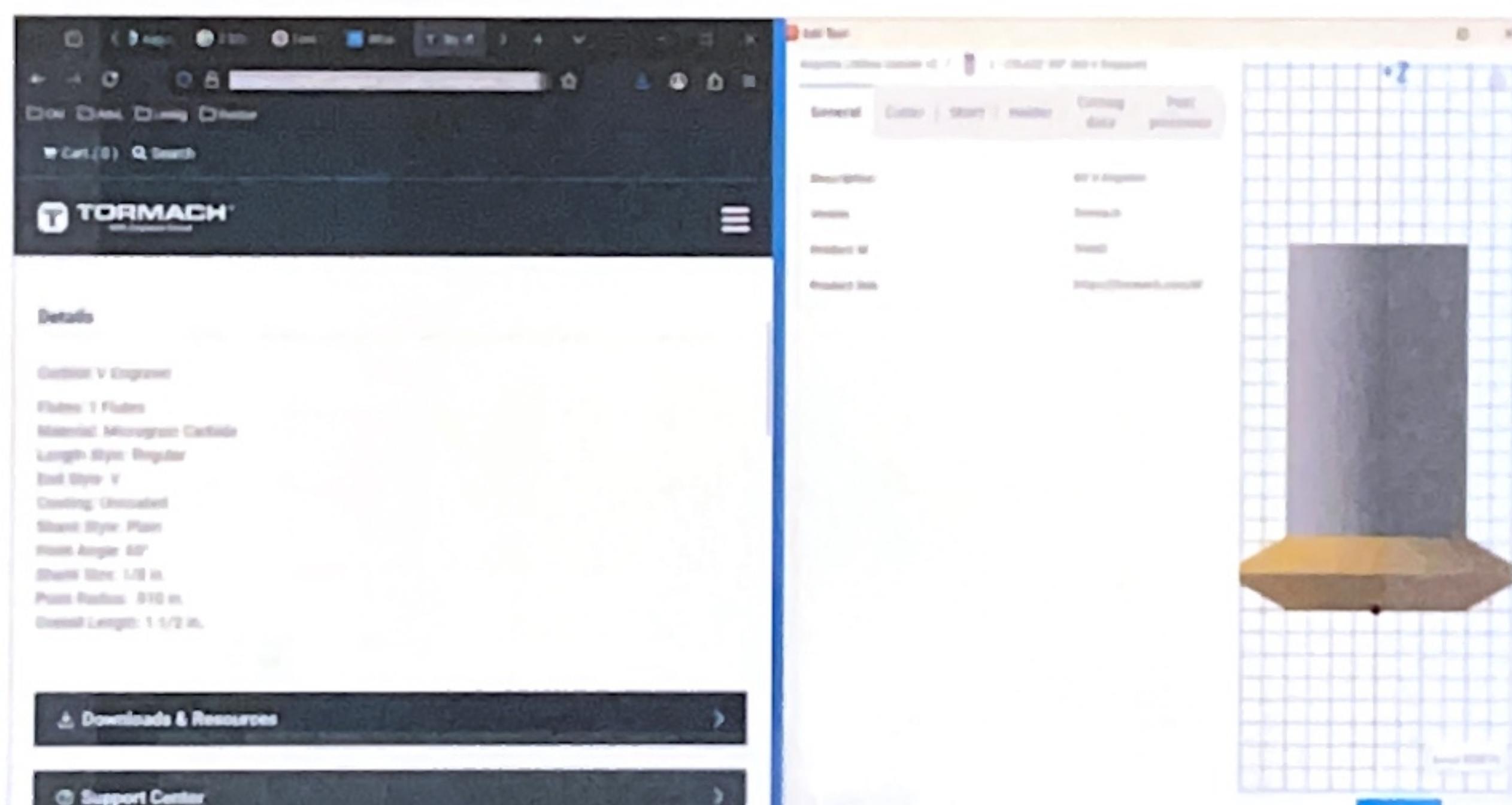
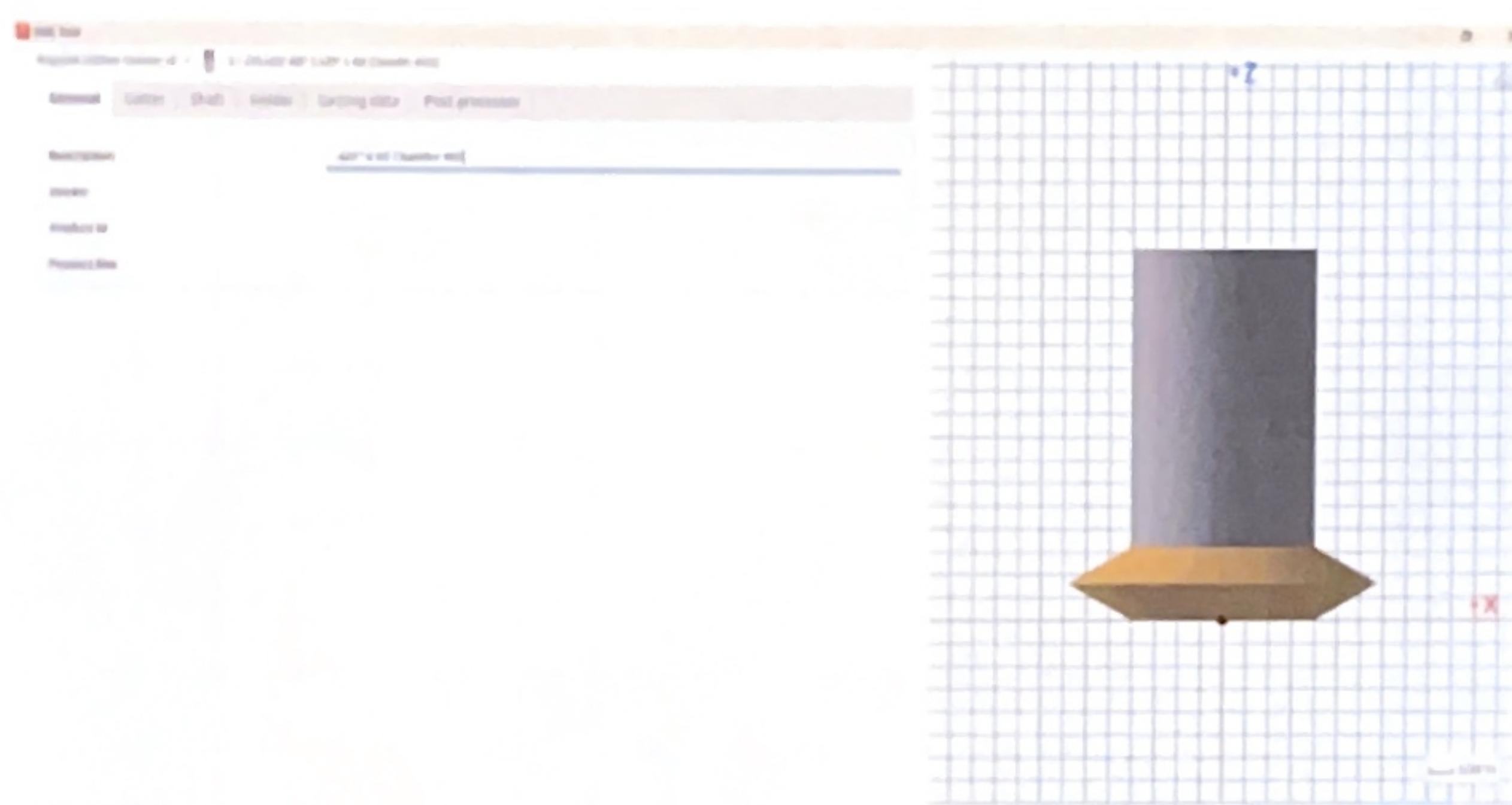
133

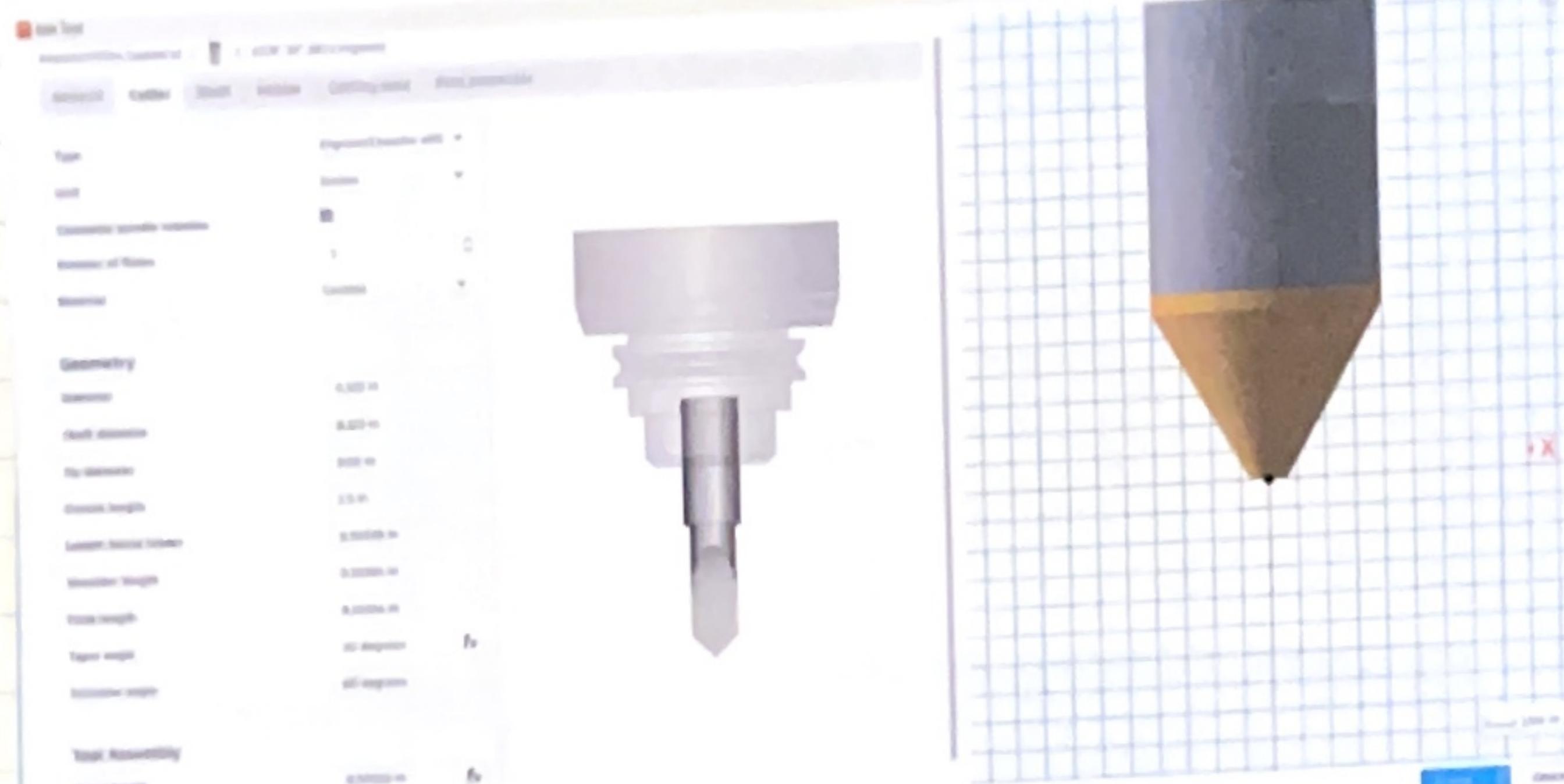
Tools

If the tool is common, you might find it by searching for it.

OR you can edit a copy or one.

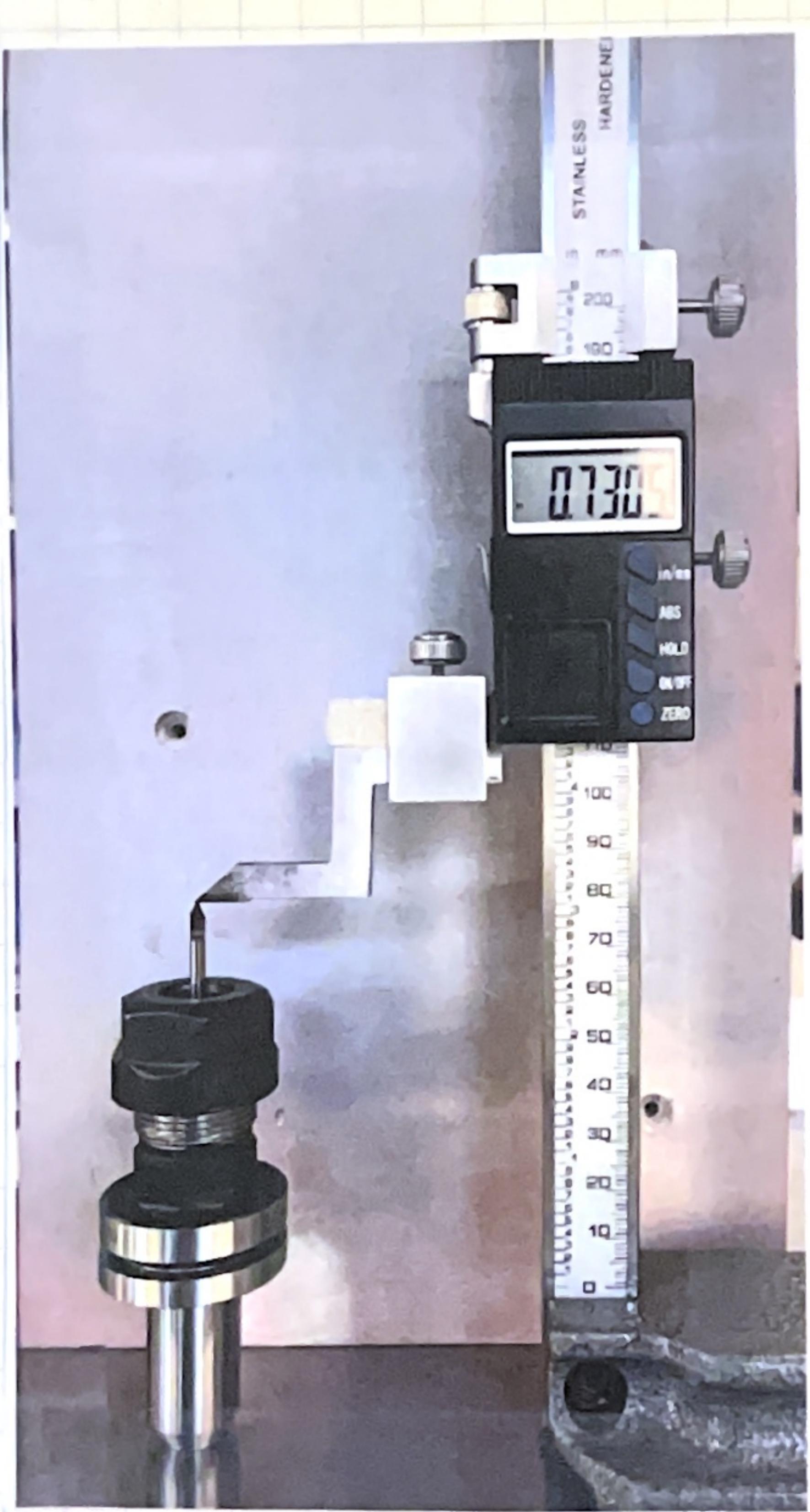
Pull up the manufacturer's details on the bit and fill in what you can.



07/01/28
wesley
Cesta

Fusion 360 gives you hints for what each value is using blue arrows.

If the manufacturer doesn't list the value, you can measure it



Using the calipers, you can measure values that you can't find from the manufacturer.

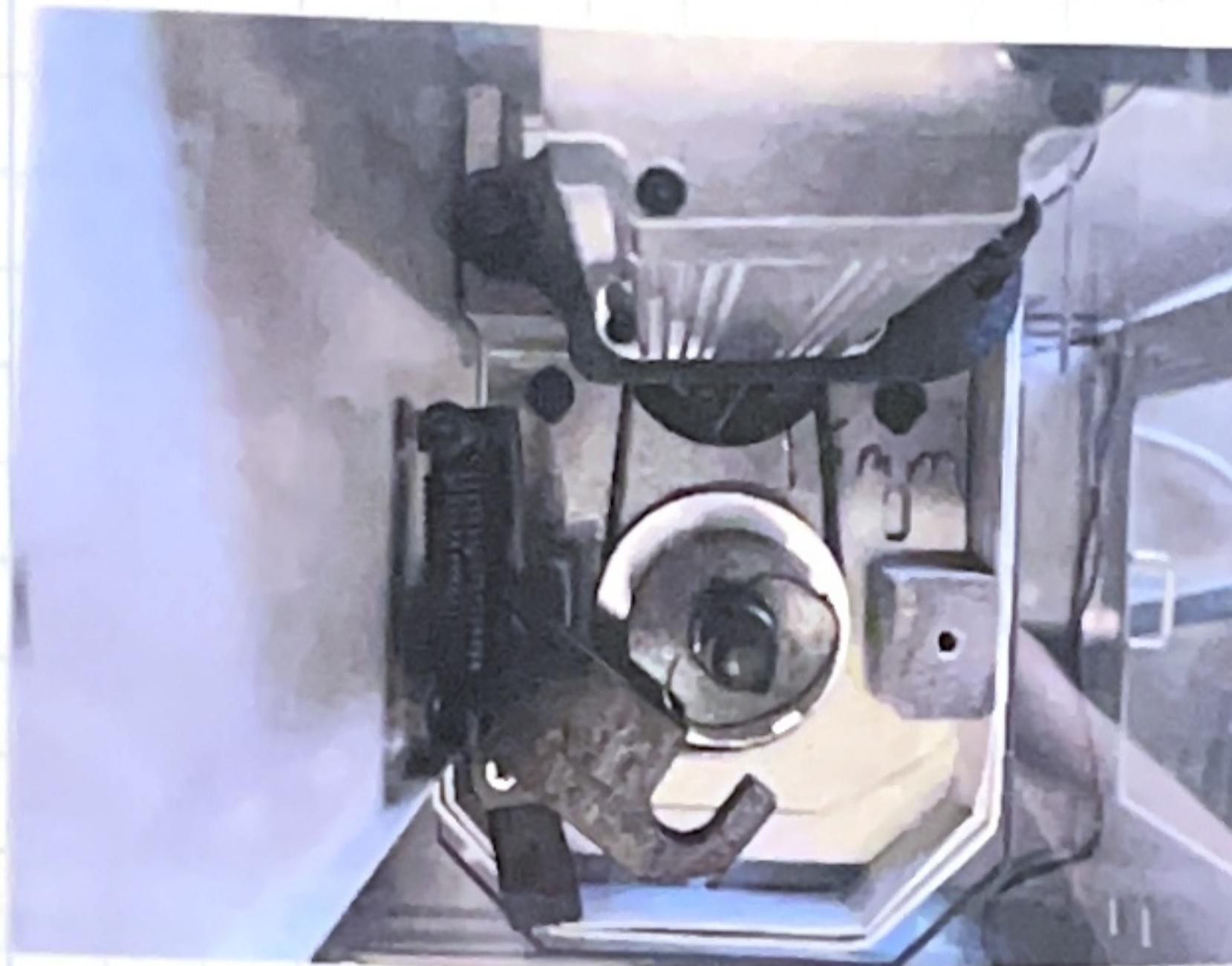


Marking the center of the circle for the carve.

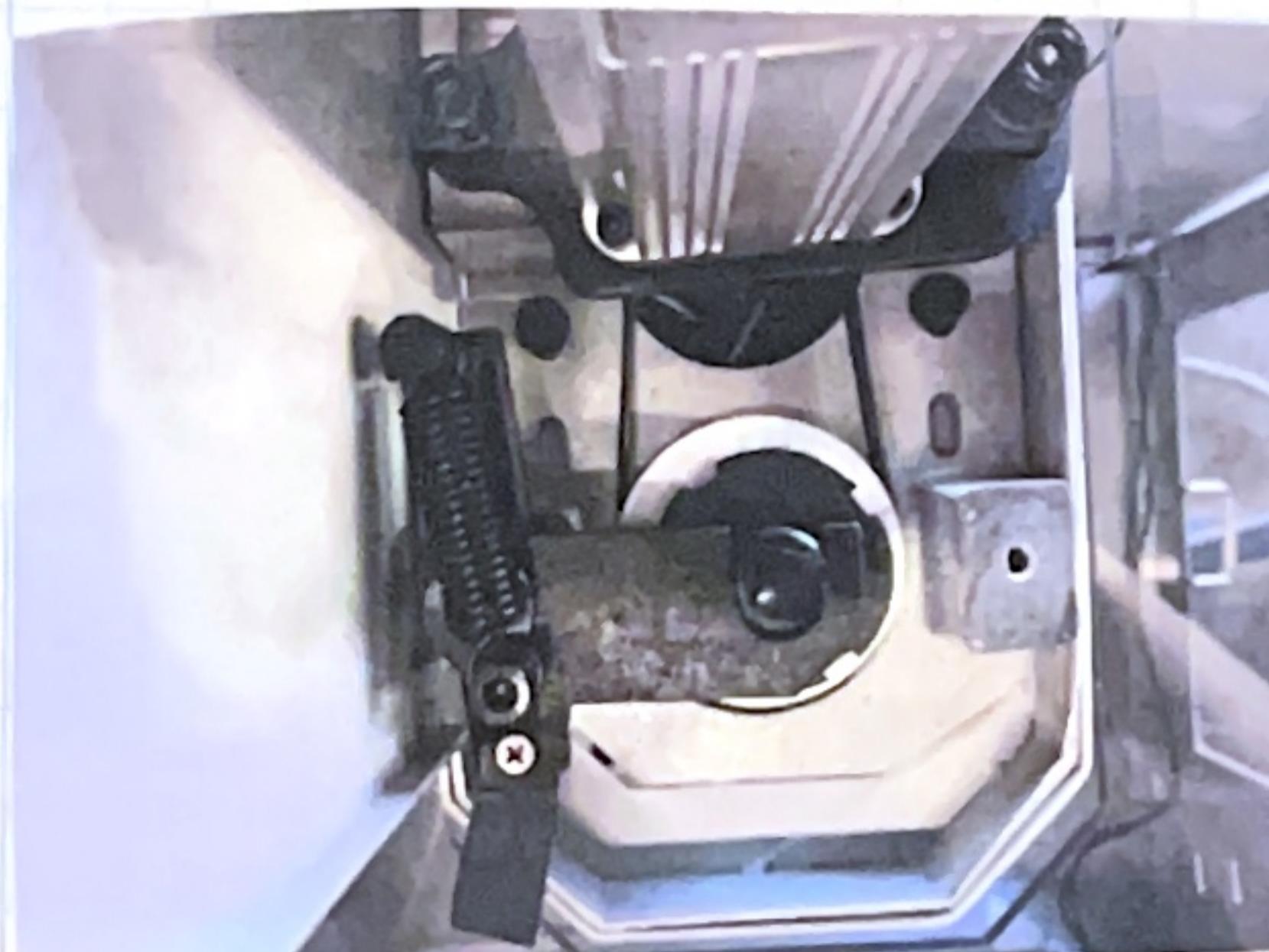
Changing tools:



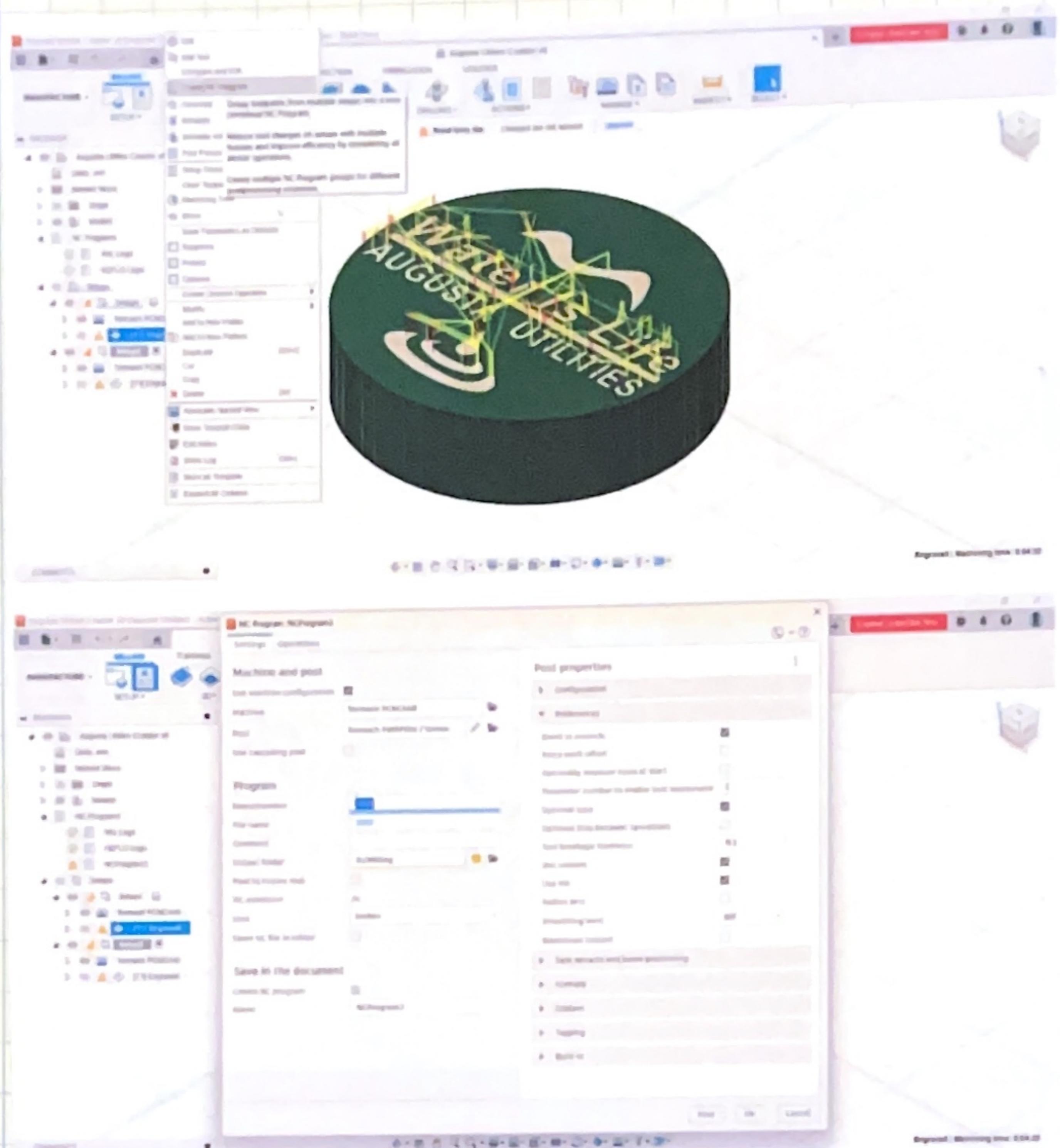
Set the Z height
to allow the machine
to open the top cage



Engage the holder
and use a 13 mm
wrench to loosen
the draw bar.

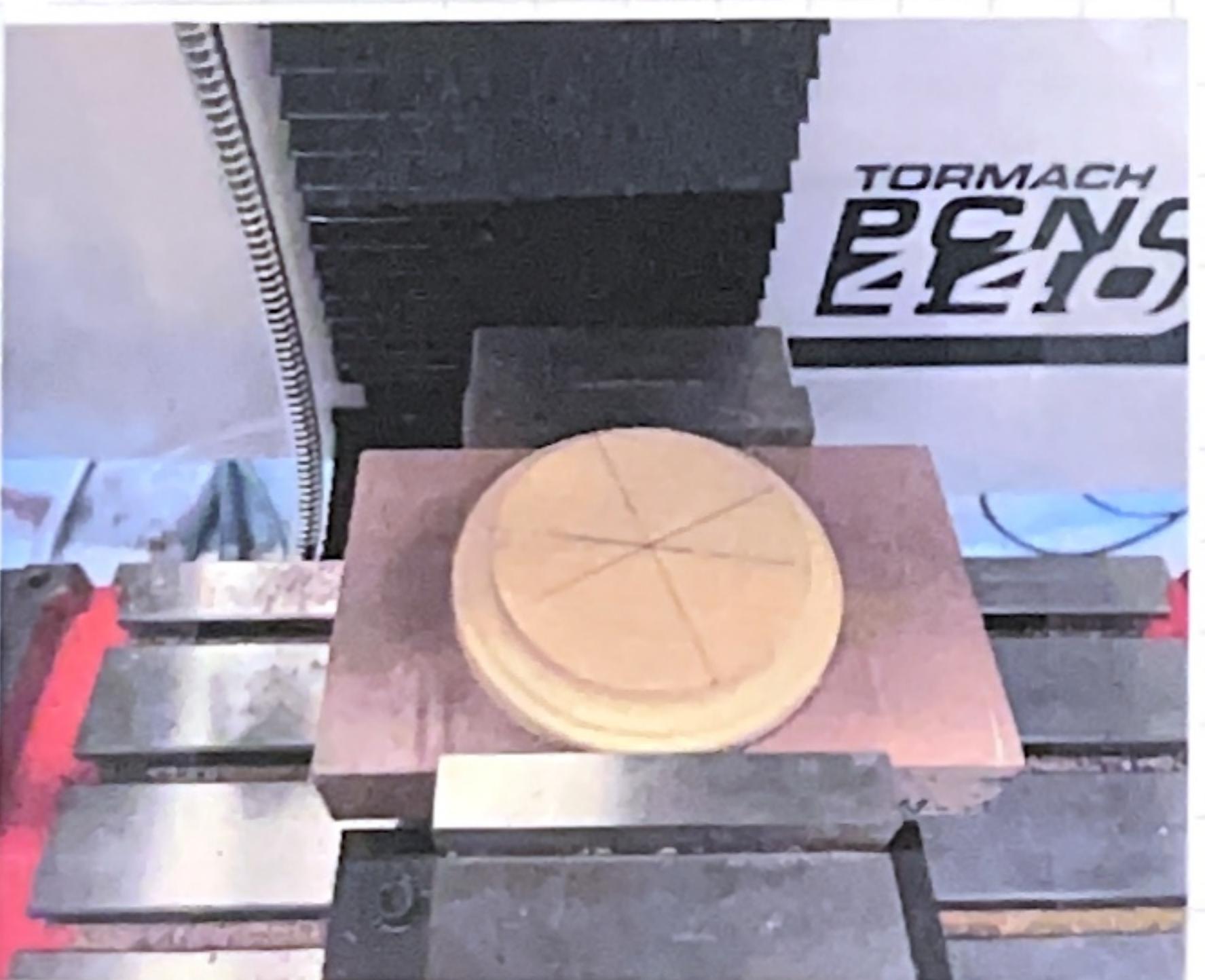


Make sure to tighten it
and disengage the holder
after installing the tool.

Create a NC program:

Right click the path you want a program for.

Click Create NC program. Tell it where to save to.

Clamp part:

Make sure the part is clamped good enough so that it doesn't move.

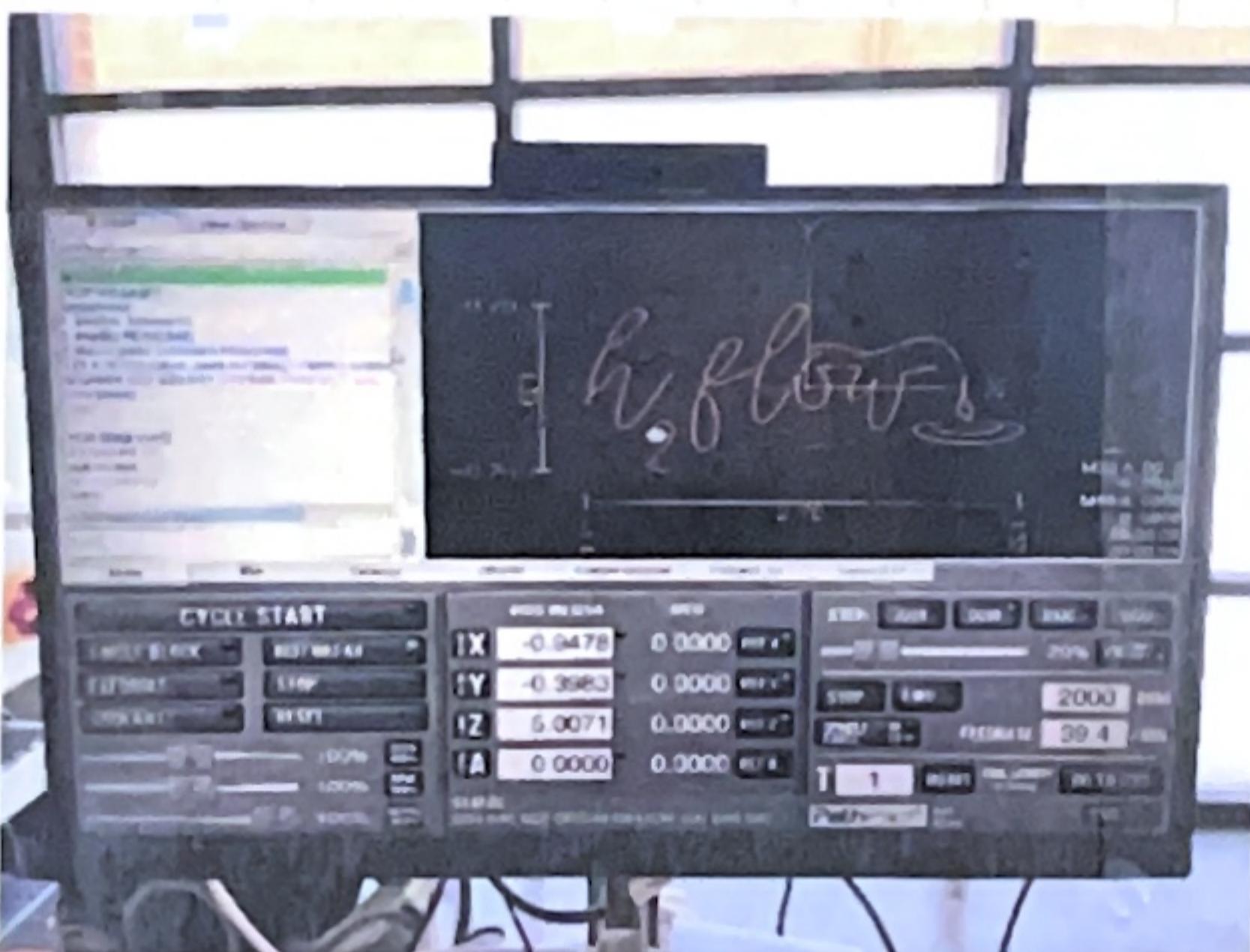
If you're unsure about where the tool head is going, try to elevate the part with a flat material. This makes sure the vice isn't damaged if you mess up your origin.

07/01/25
Wally
Ceda

CNC Basics: V-Carve

137

Path Pilot



Reference X Y Z
by clicking REF X, Y, Z

This will take the CNC to the limit switches.

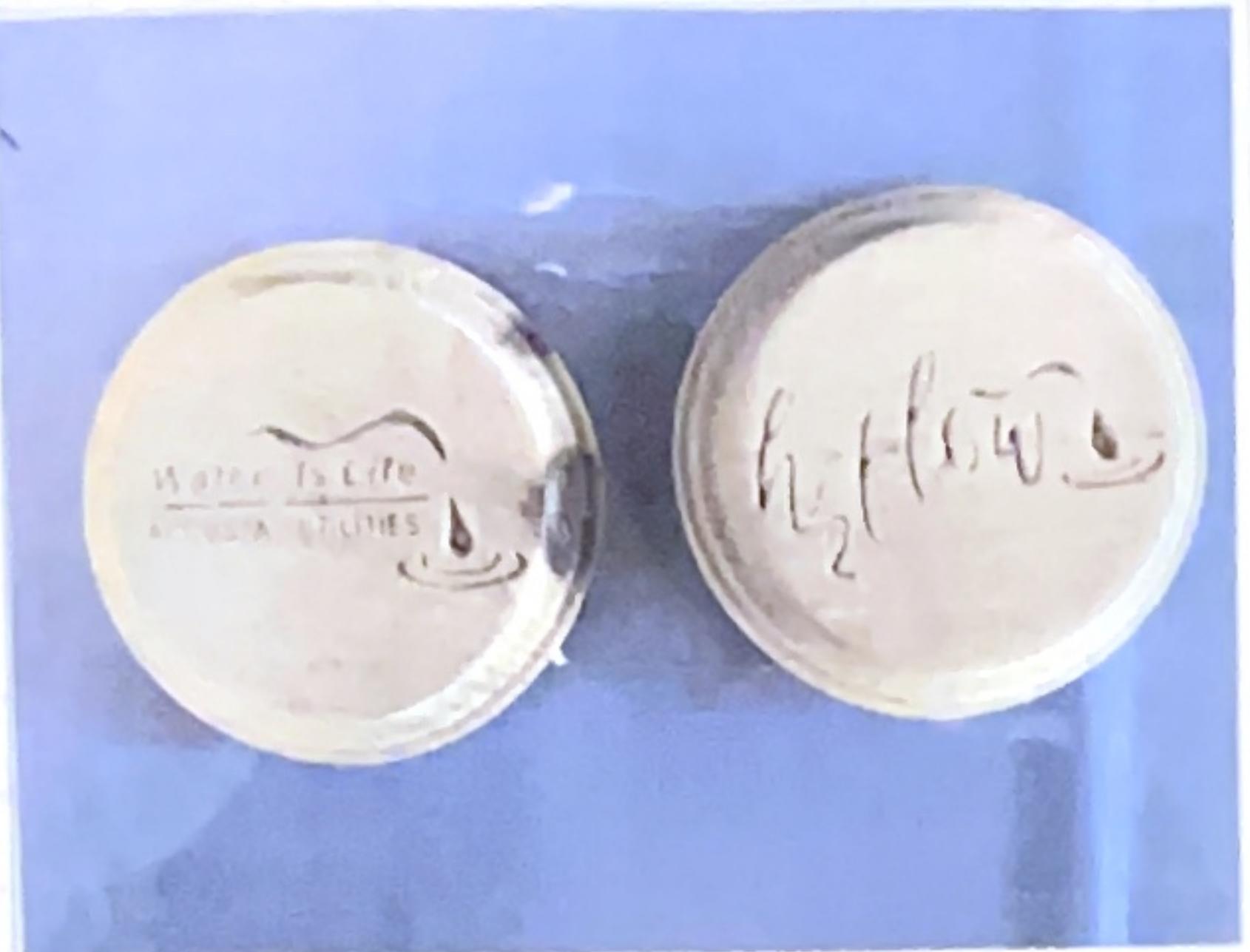
Use the round controller to navigate the tool head to 0 X, 0 Y, ~~0 Z~~ of the origin you chose on Fusion. For me this was the center of my part. Click zero X and zero Y when satisfied.

Place a sheet of paper on top of the part. Lower Z until the paper can't be moved.

Click OFF Sets and click touch Z. Raise Z and remove the paper.

Load your program and click cycle start.

The display will show the path and the current location of the tool.

Results:

→ Linseed oil finish



→ Paint then linseed oil