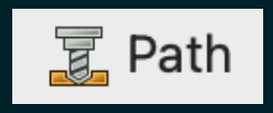
# CNC Router

## Quick start guide

# PART 1

# Generating the code

**This guide assumes that you have just finished modelling a part in FreeCAD's Part Design workbench, and are reading to use the CNC. Next steps:**

1. Head to the  workbench

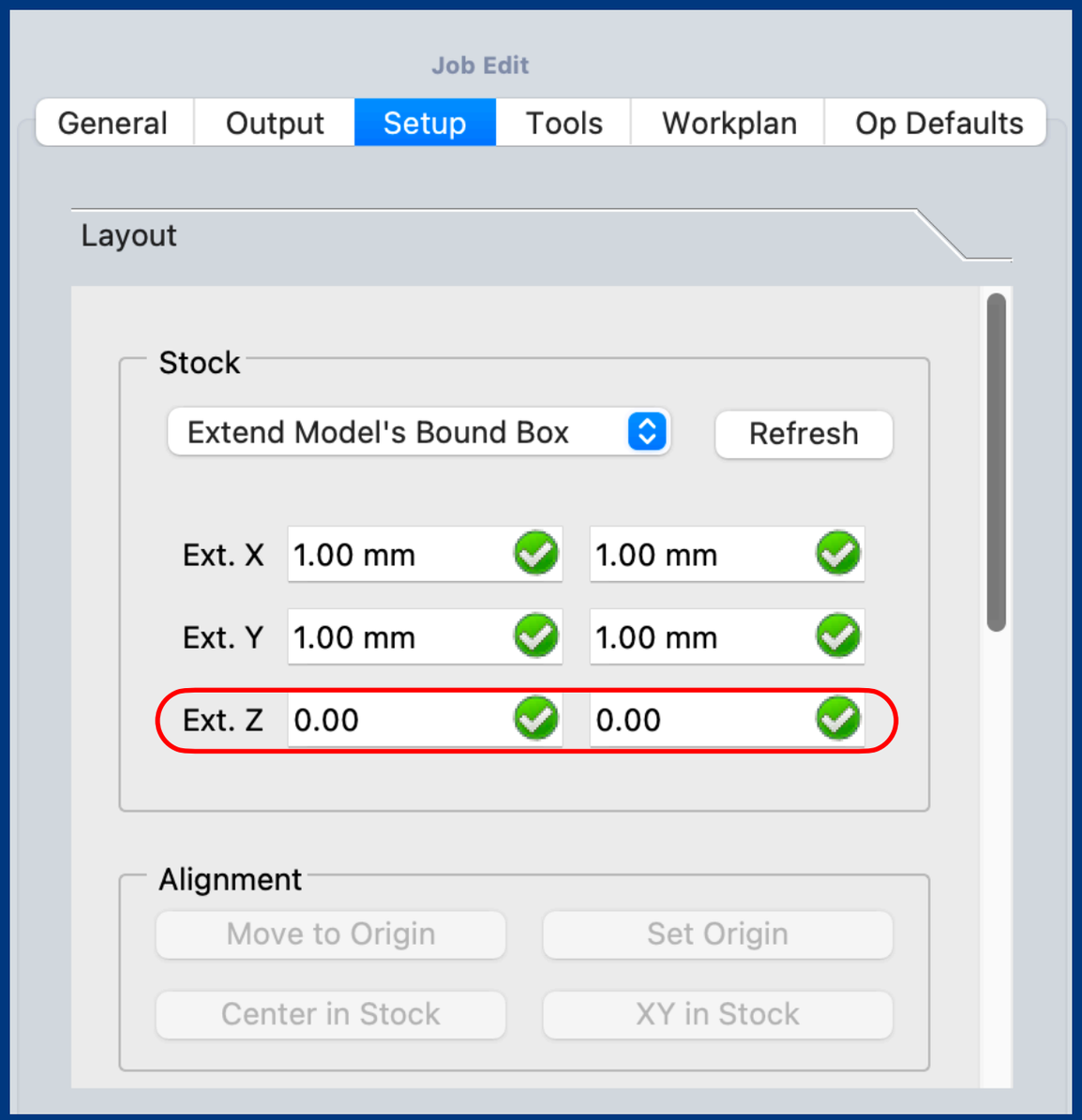
2. Select "Job"  from the toolbar

3. Select all relevant bodies from the dropdown and click "OK"

**FreeCAD will then take you directly to the Job editor.**

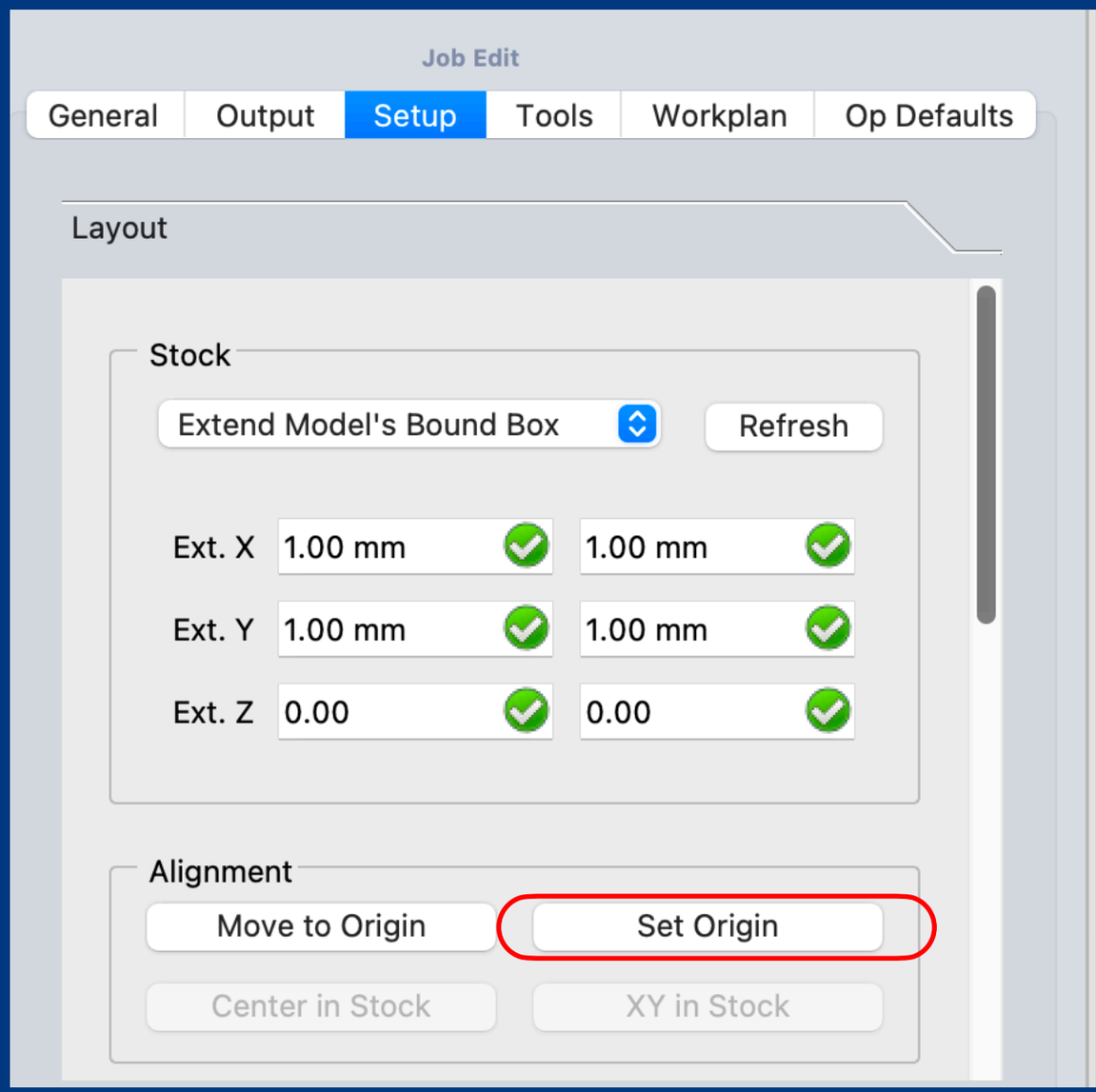
# The default tab is the Setup tab. We will make two changes:

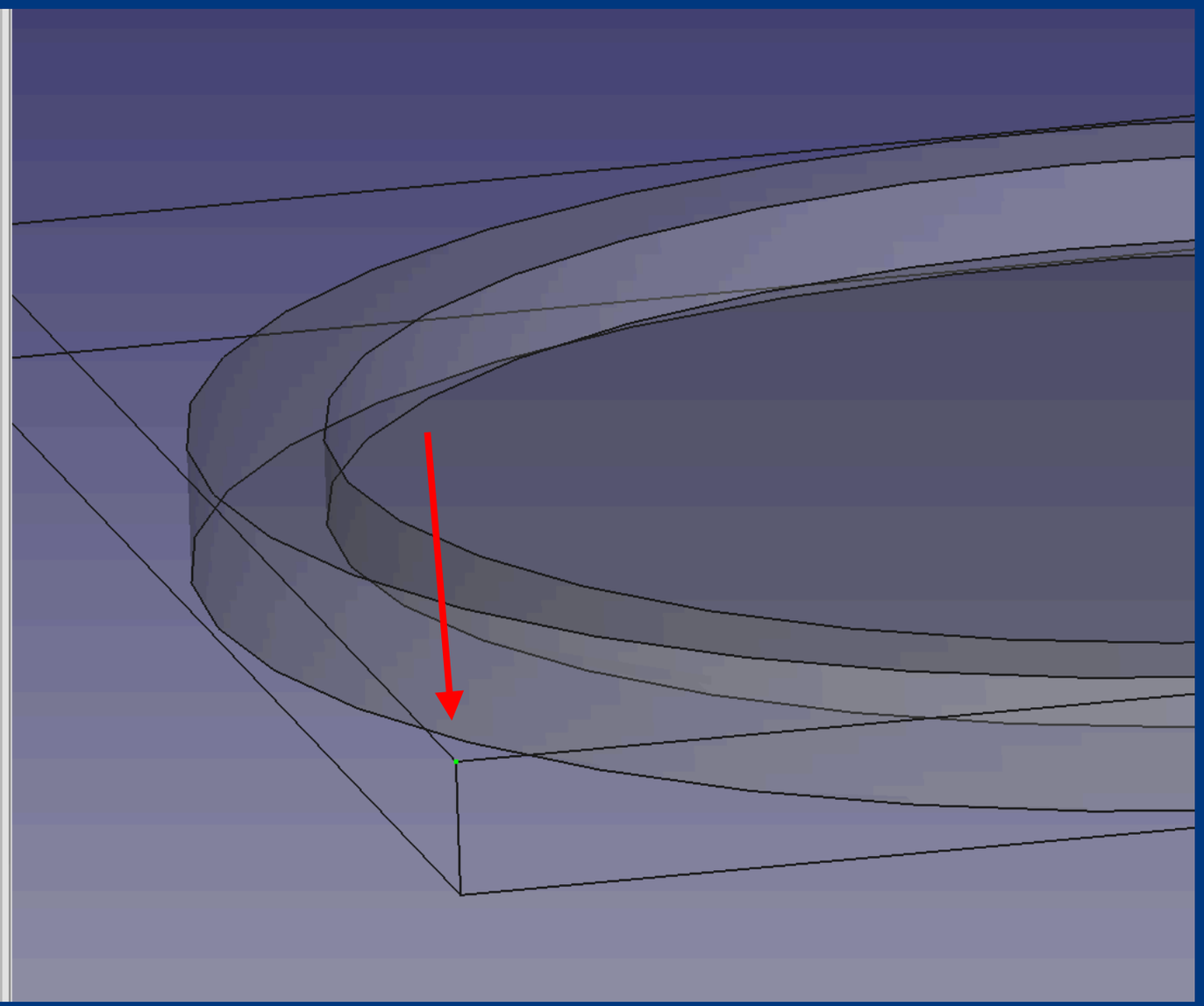
## 1) Change the Ext. Z to 0 mm



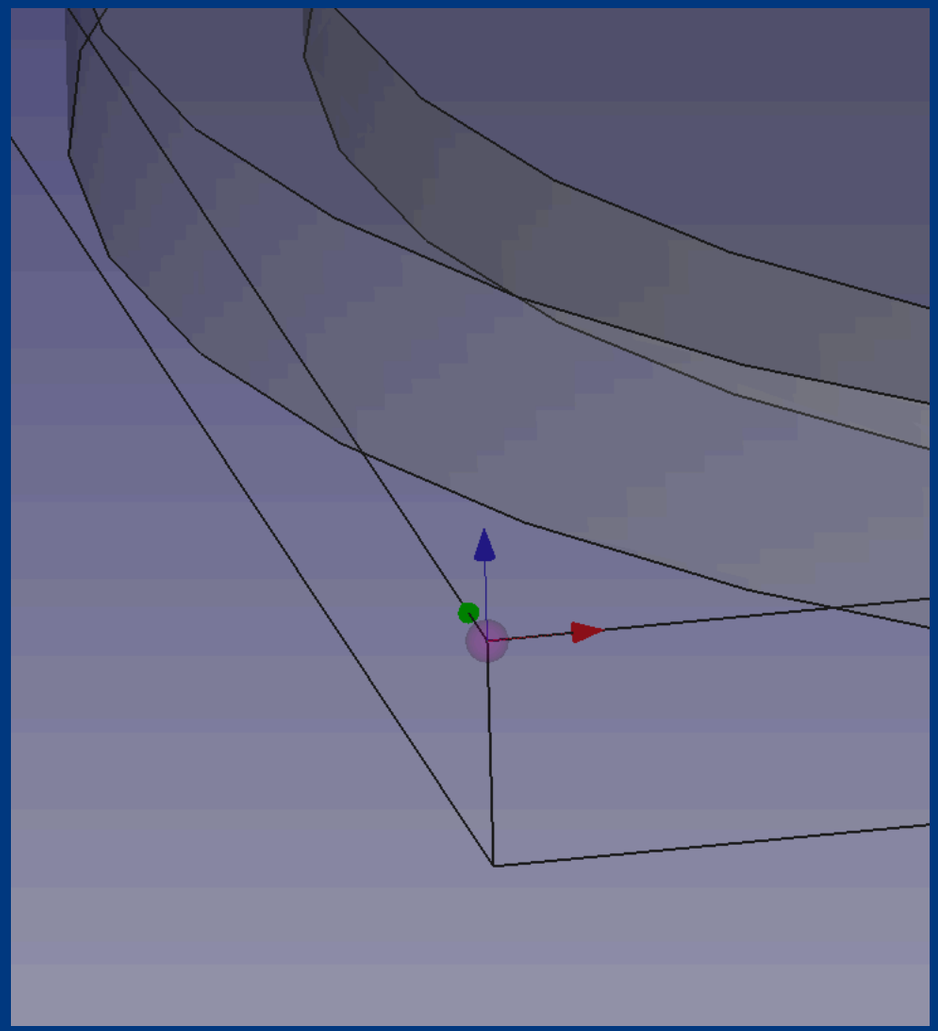
# The default tab is the Setup tab. We will make two changes:

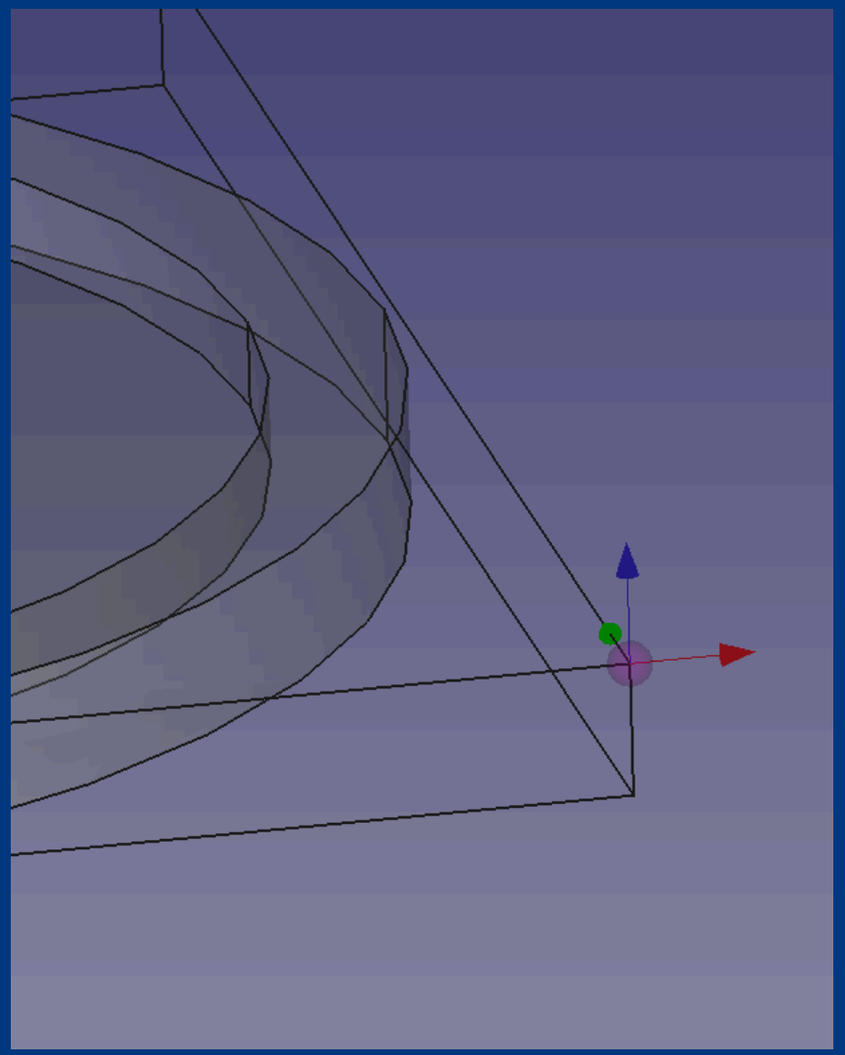
## 2) Select the corner of the stock material and select “Set Origin”



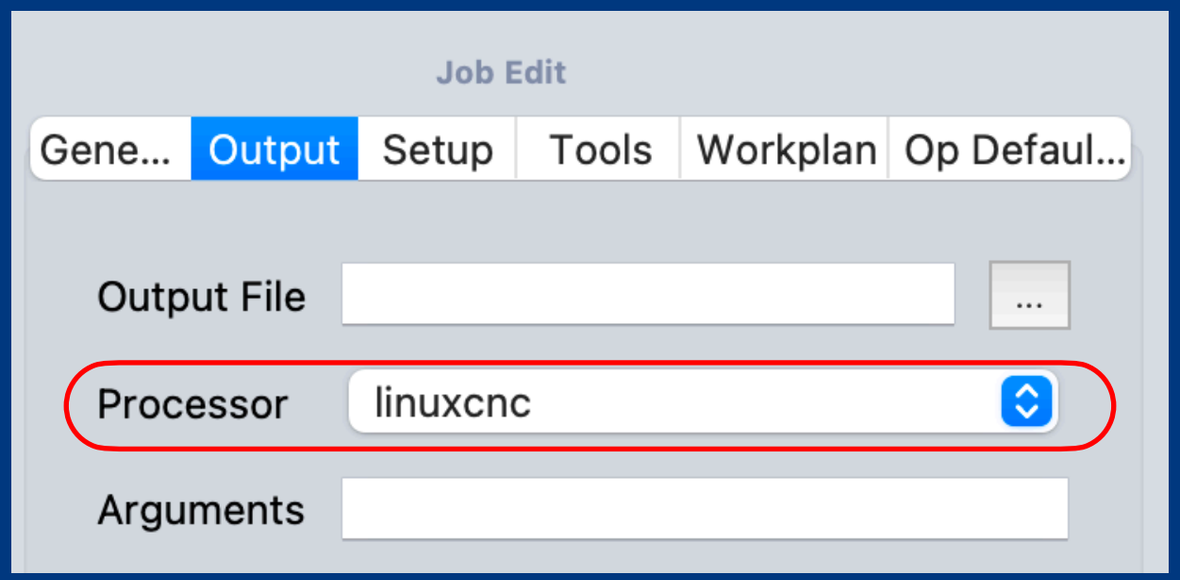



# Always ensure that the coordinate axes are pointing along the edges of the stock material!





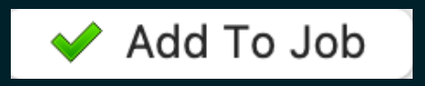
**Next, go to the Output tab and select** linuxcnc **in the Processor dropdown:**




**Then click OK to exit the editor.**

# Now we have to specify what milling bit(s) we plan to use.

1. Open the ToolBit Dock 

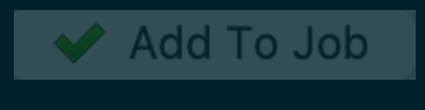
2a. Select a tool by clicking 

, or

2b. Create a custom tool by clicking the  icon

# Next we have to specify what milling bit(s) we plan to use.

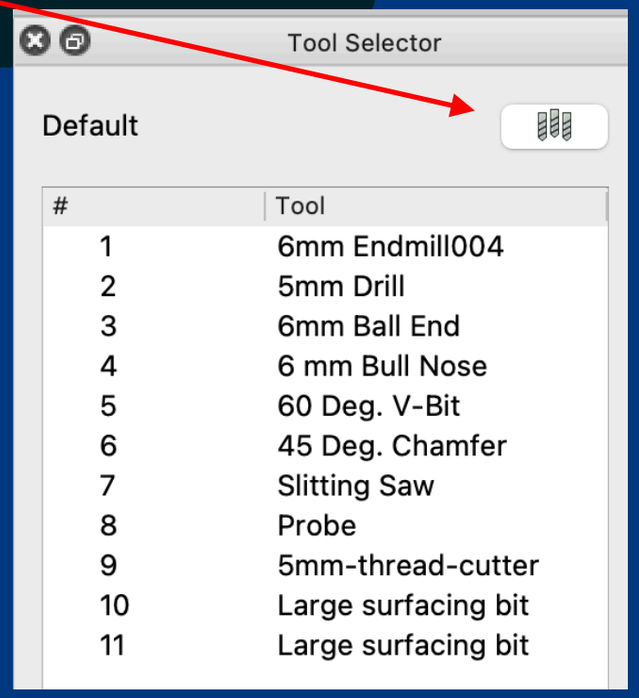
1. Open the ToolBit Dock 

2a. Select a tool by clicking 

, or

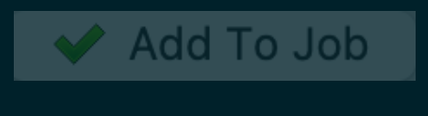
2b. Create a custom tool by clicking the  icon

Same icon, different function



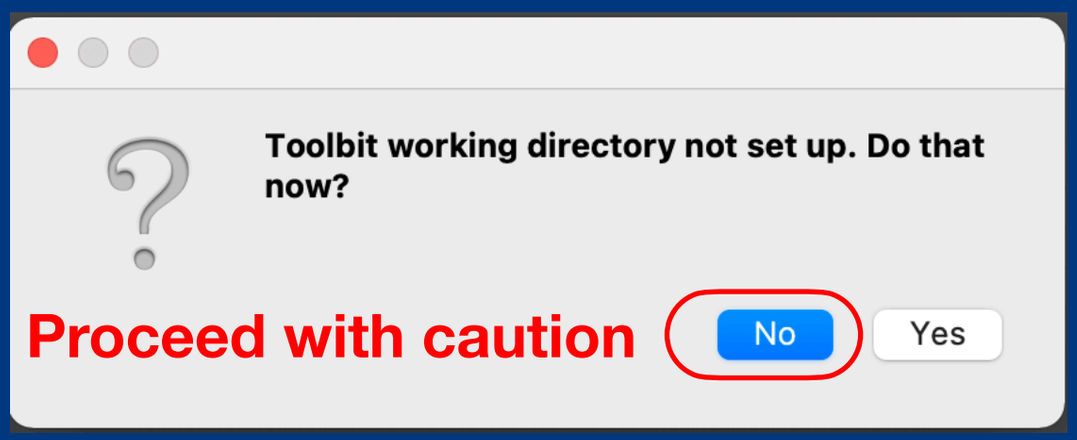
# Next we have to specify what milling bit(s) we plan to use.

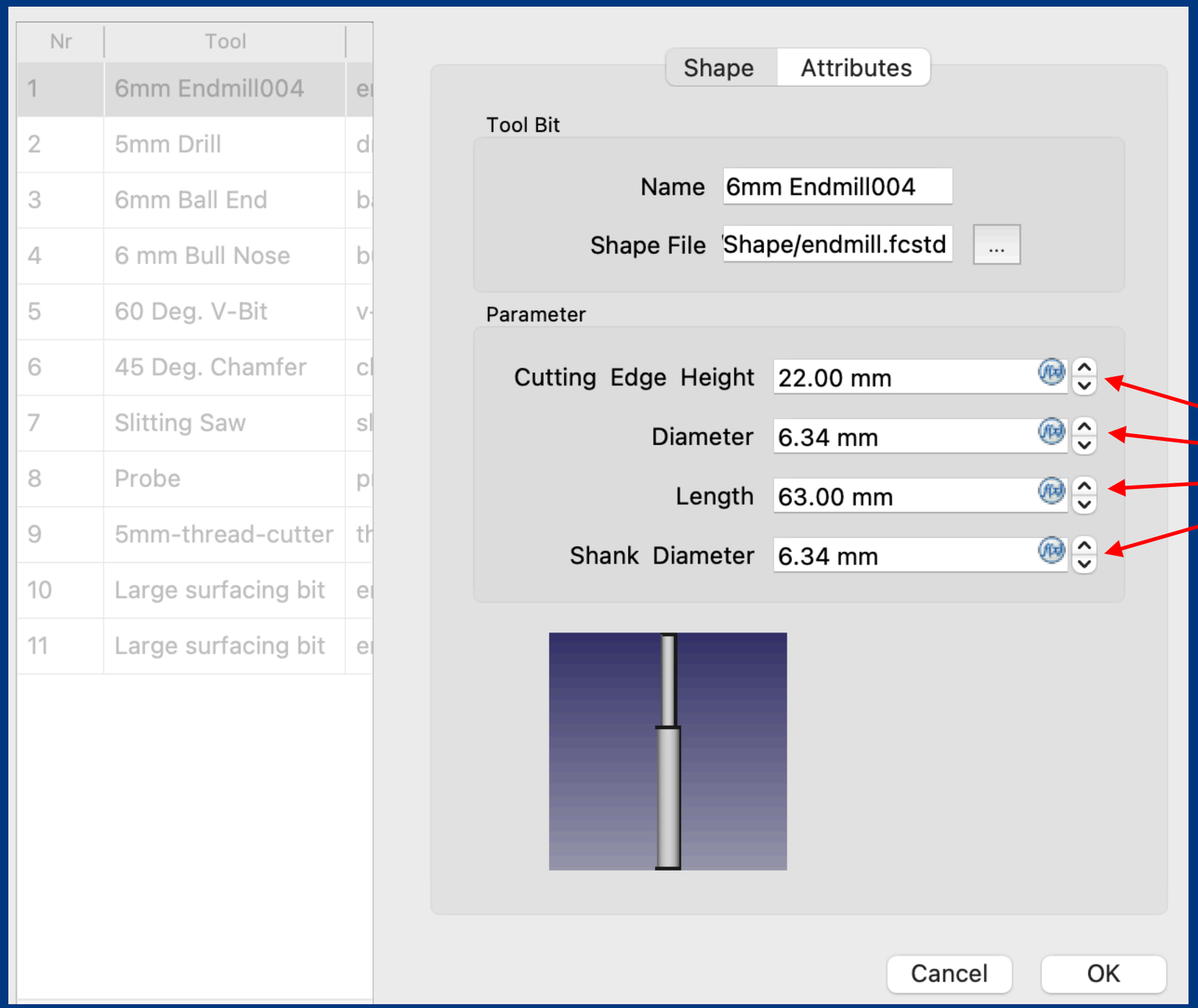
1. Open the ToolBit Dock 

2a. Select a tool by clicking 

, or

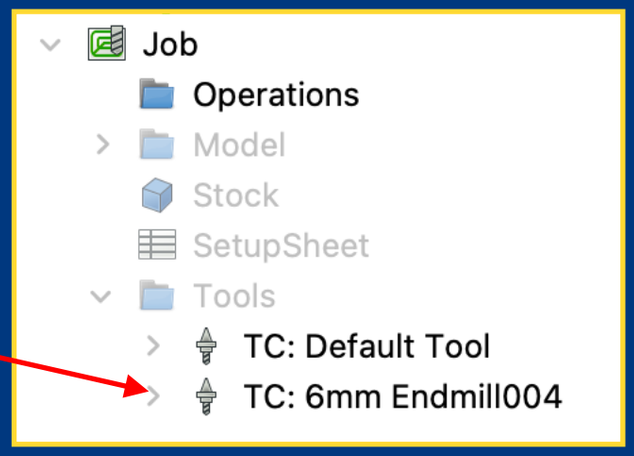
2b. Create a custom tool by clicking the  icon



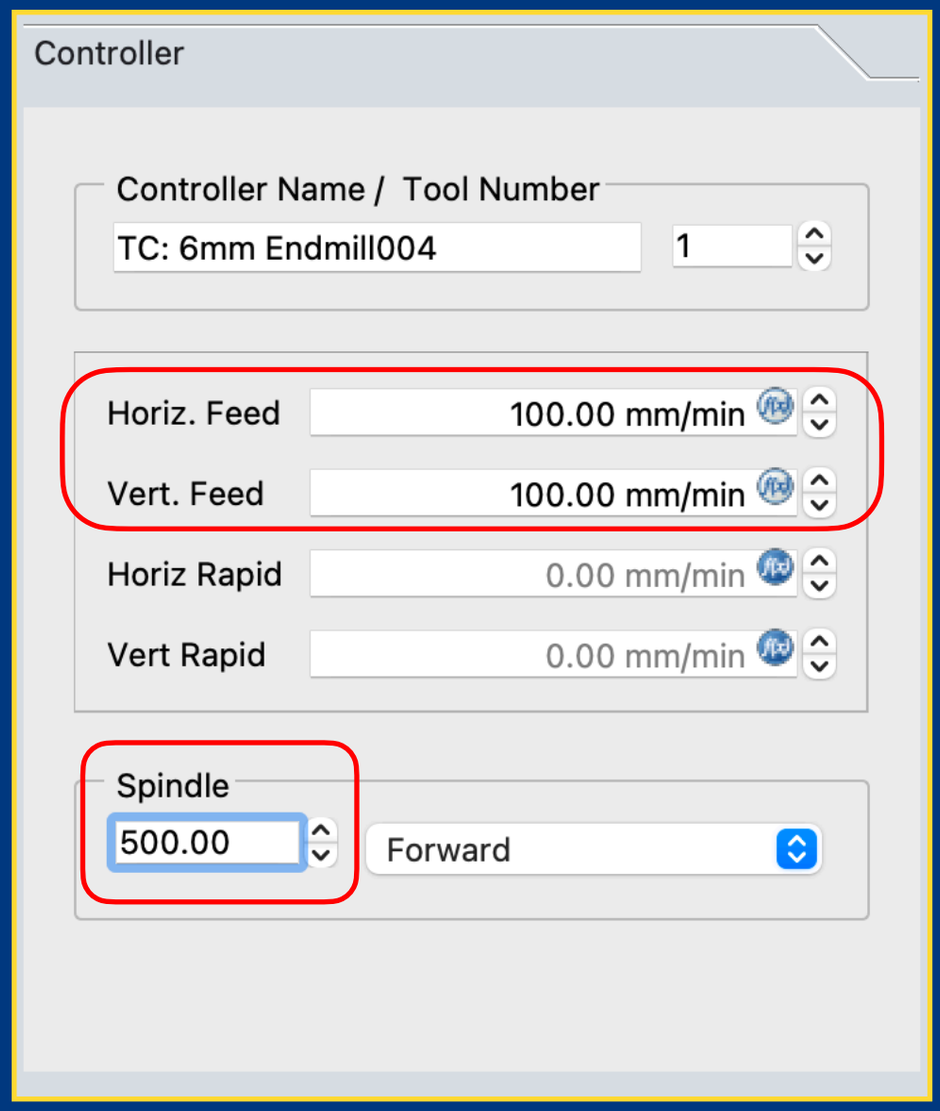



- **Double-click** an existing tool to edit
- **Use callipers to measure bits accurately**
- **Give your tool a useful name**

**3. Double-click the tool in the Tool folder of your job**




**4. Enter some values for**
Horiz. Feed
Vert. Feed
Spindle

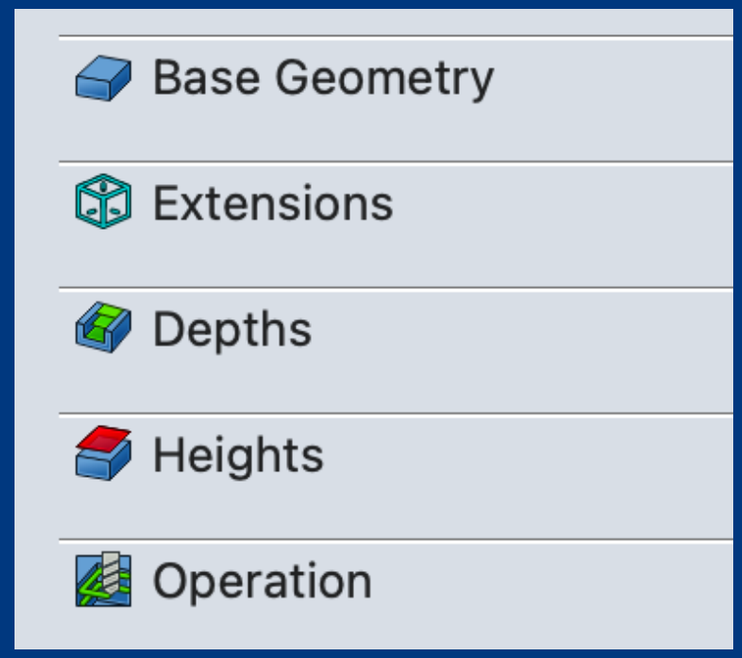



**Note for the OMNI:**
Feeds and speeds will be set manually during hardware setup, but the software still requires nonzero entries initially.

# PATH GENERATION

## General steps

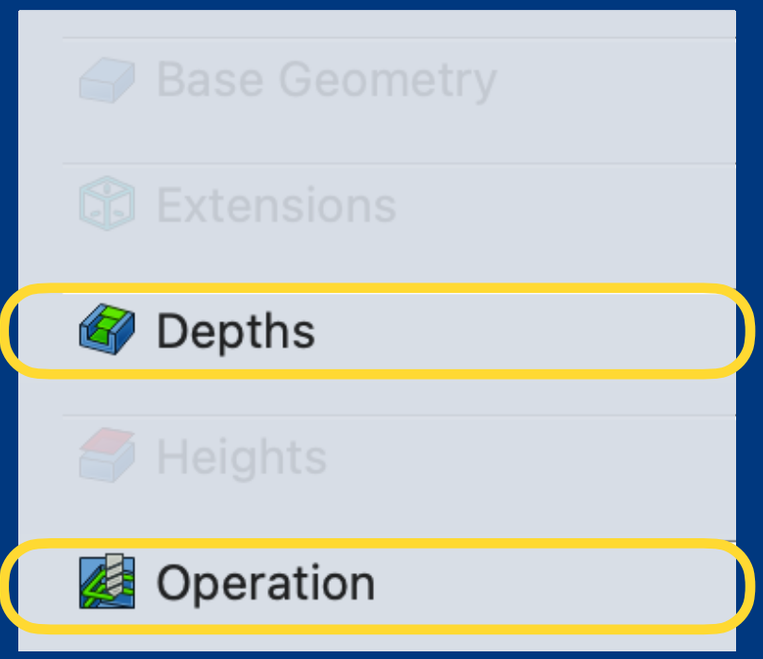
1. Select a lines or a face
2. Select an operation
3. Choose a tool
4. Adjust the settings in the tabs



# PATH GENERATION

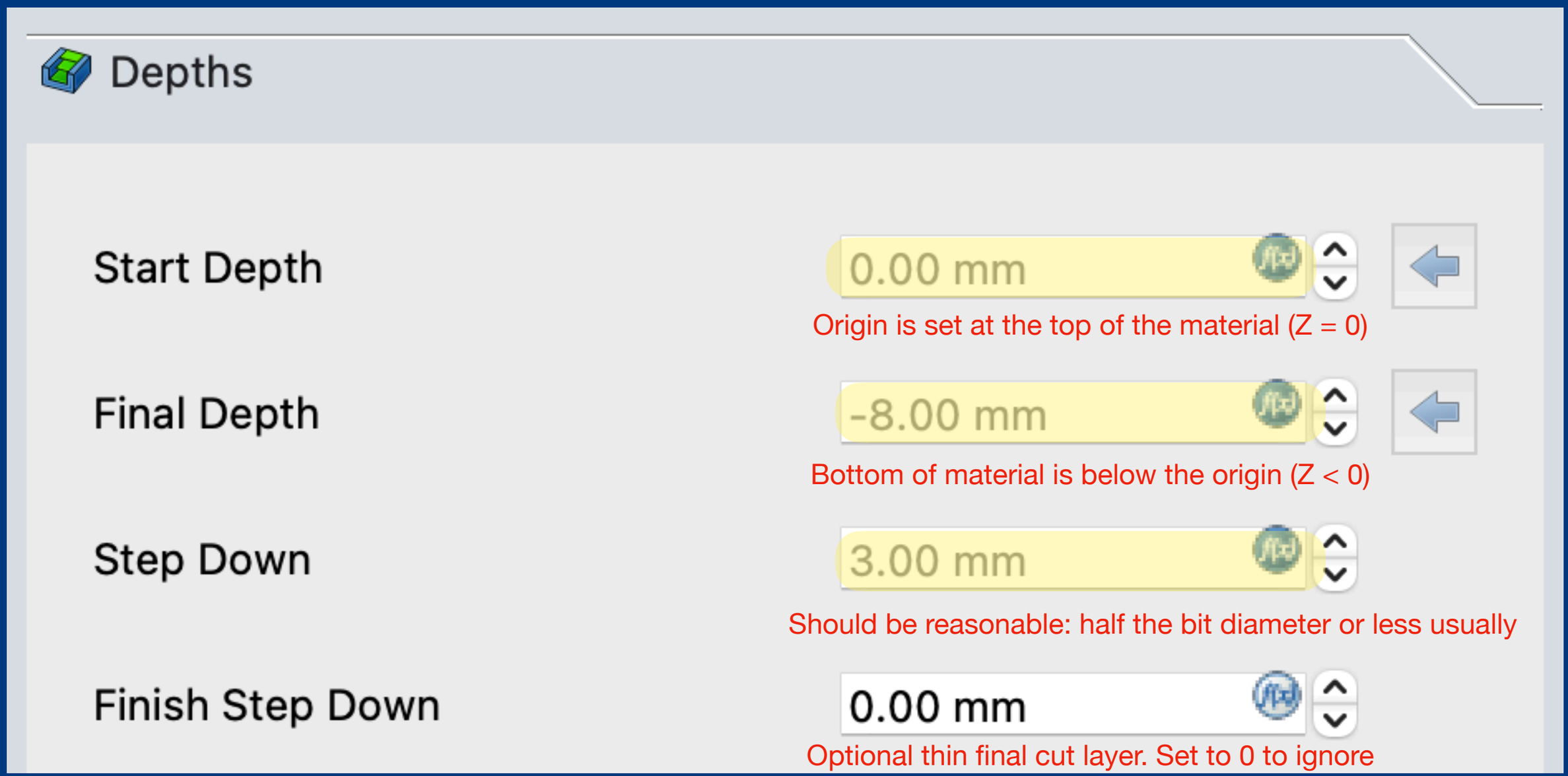
## General steps

1. Select a face
2. Select an operation
3. Choose a tool
4. Adjust the settings in the tabs

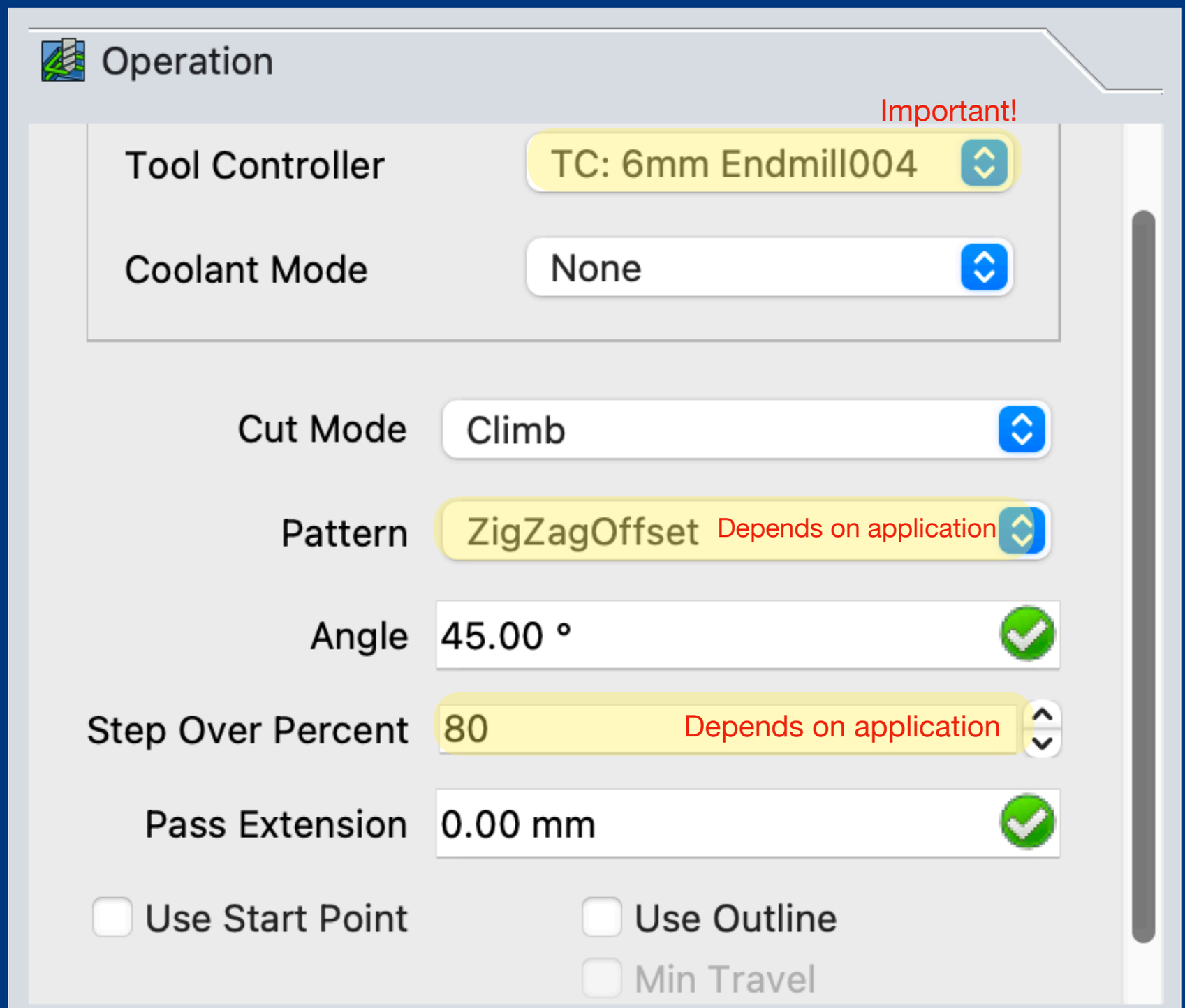



Most important

# Depths


Depths
Start Depth
0.00 mm
Origin is set at the top of the material (Z = 0)
Final Depth
-8.00 mm
Bottom of material is below the origin (Z < 0)
Step Down
3.00 mm
Should be reasonable: half the bit diameter or less usually
Finish Step Down
0.00 mm
Optional thin final cut layer. Set to 0 to ignore

# Operation

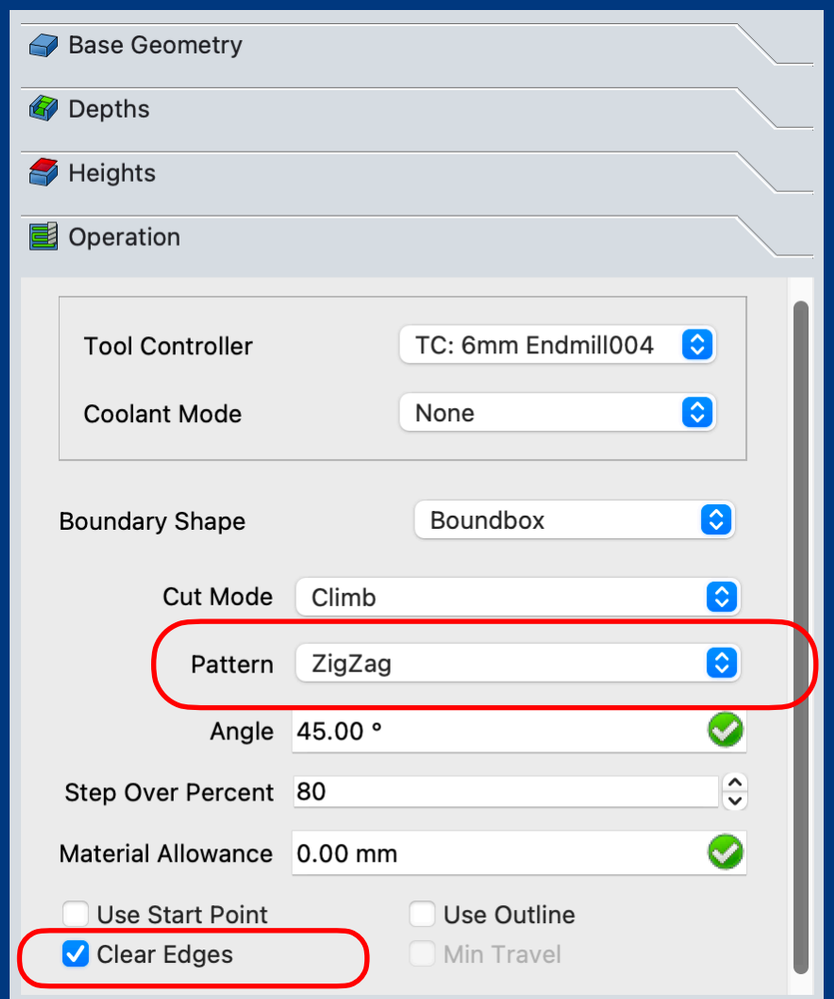

Operation
Important!
Tool Controller
TC: 6mm Endmill004
Coolant Mode
None
Cut Mode
Climb
Pattern
ZigZagOffset
Depends on application
Angle
45.00 °
Step Over Percent
80
Depends on application
Pass Extension
0.00 mm
Use Start Point
Use Outline
Min Travel

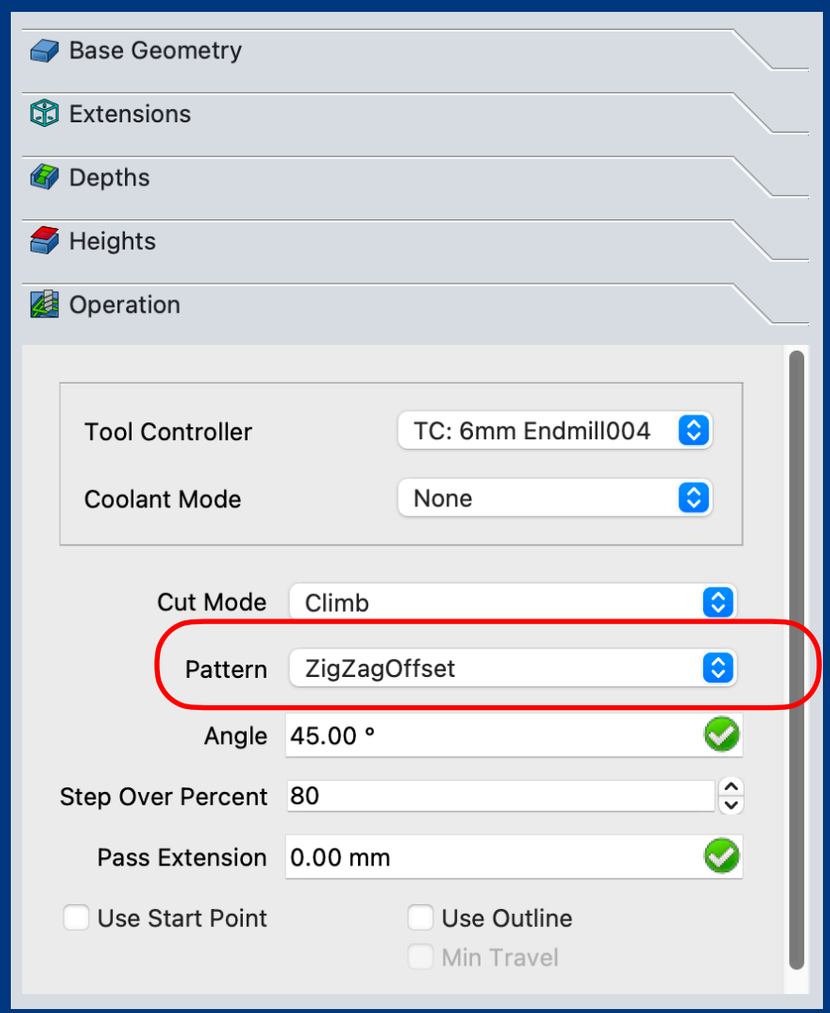
# Recommended settings

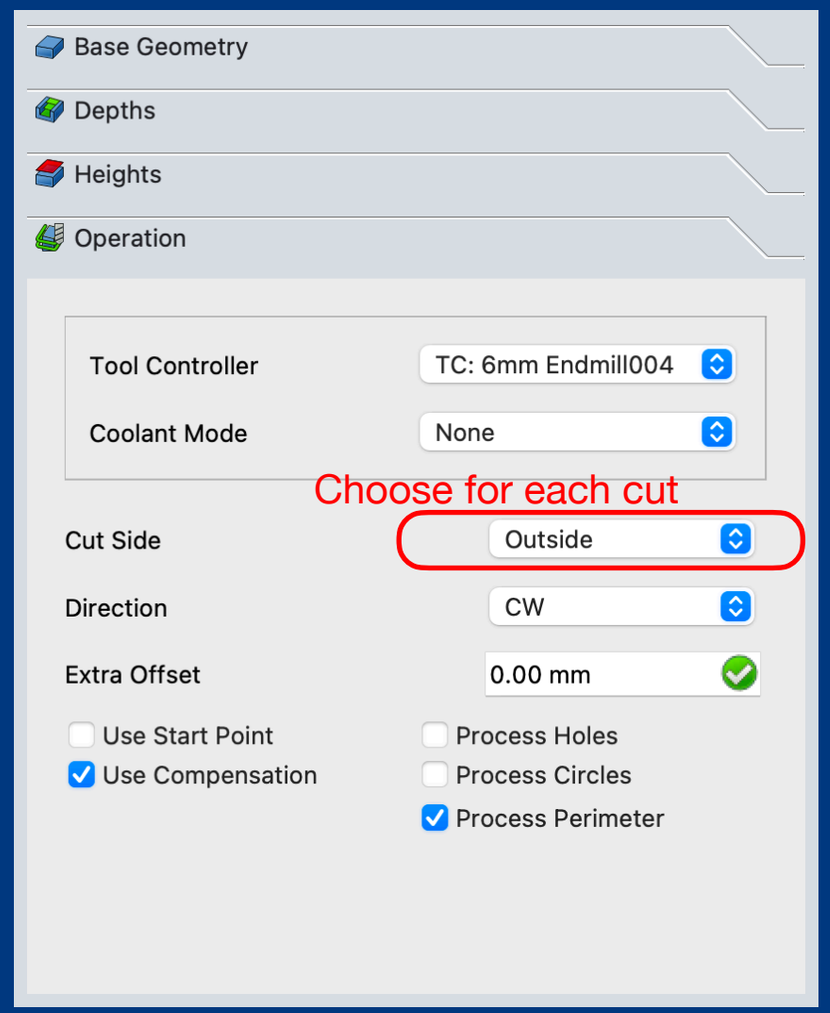
## Facing

## Pockets

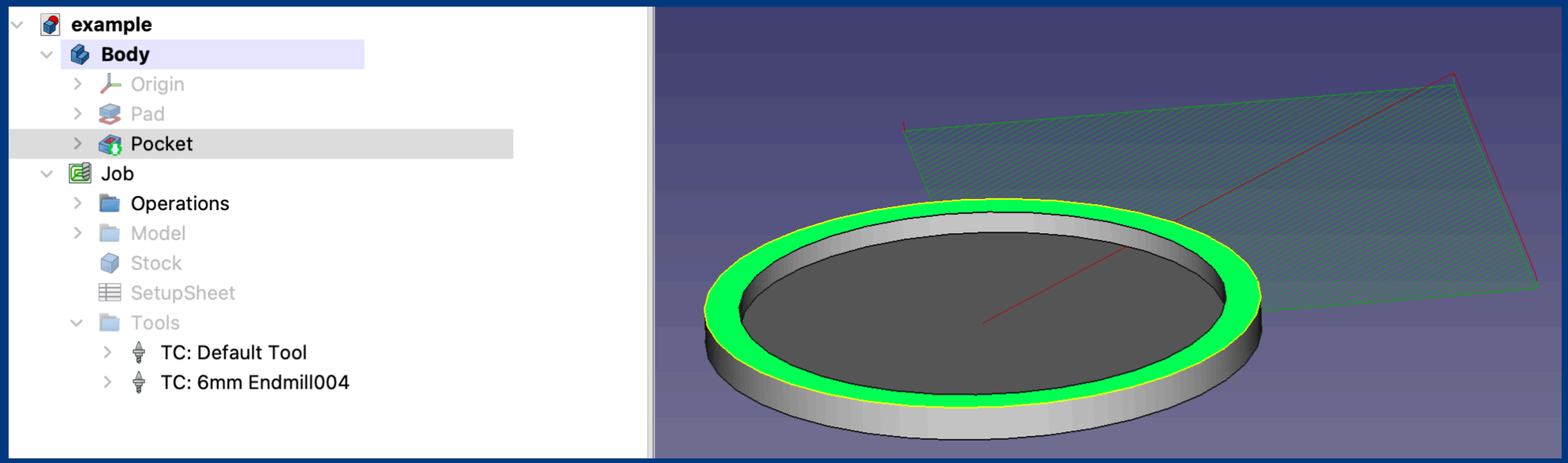
## Profiles

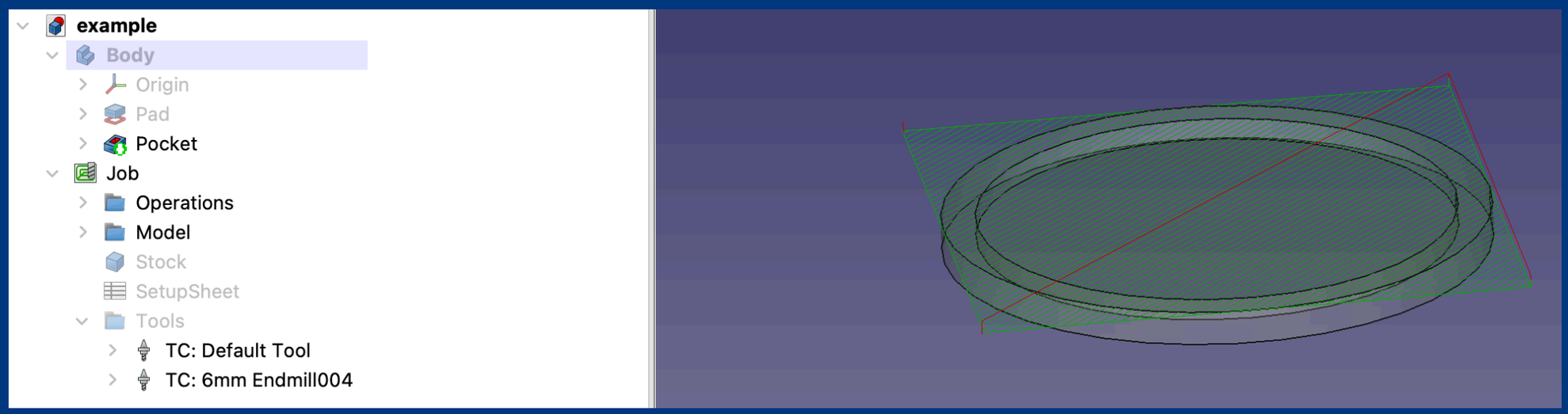


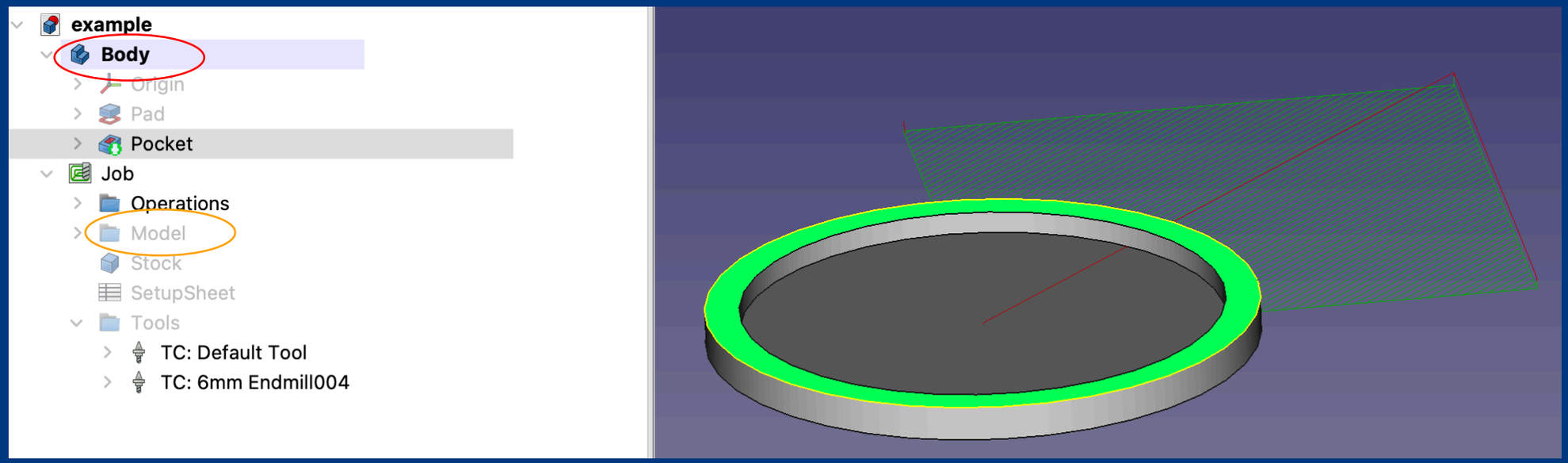



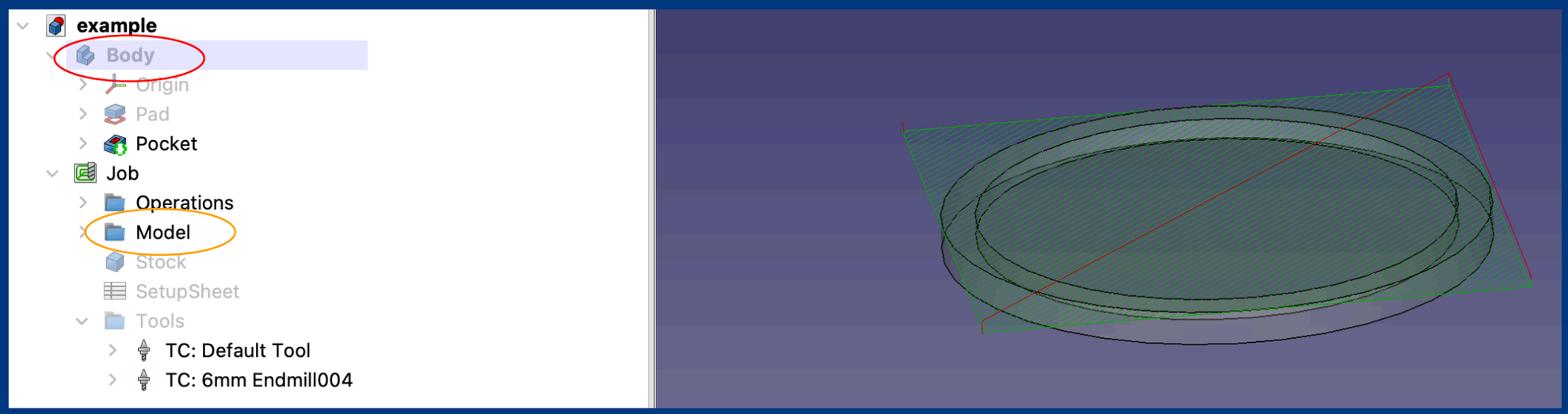
# MINIGAME:
# Spot the difference!



# MINIGAME:
# Spot the difference!



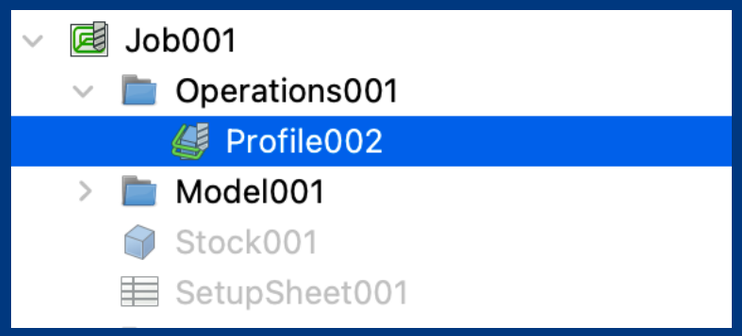




The main body is hidden and model body is made visible

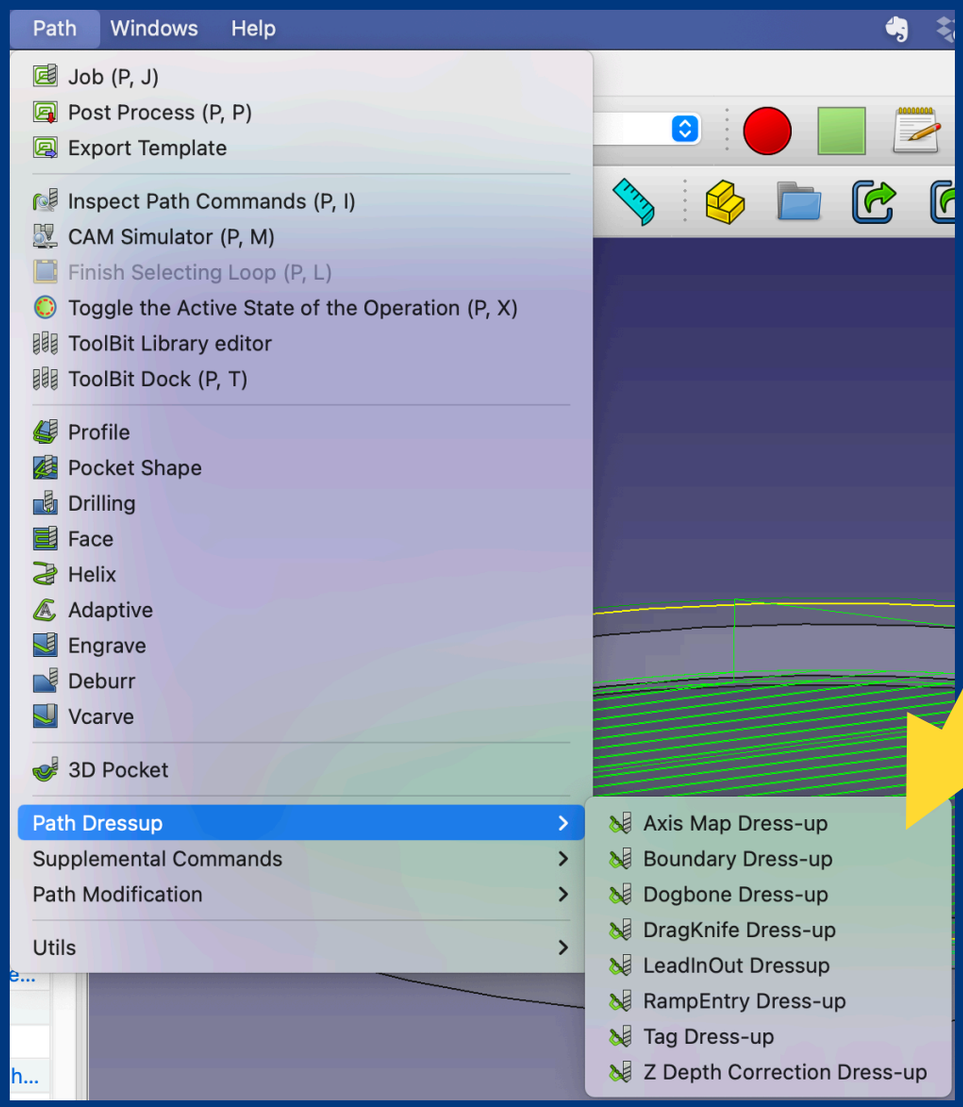
# Path Dressups

**Sometimes you may need to modify the paths that are generated by default:**

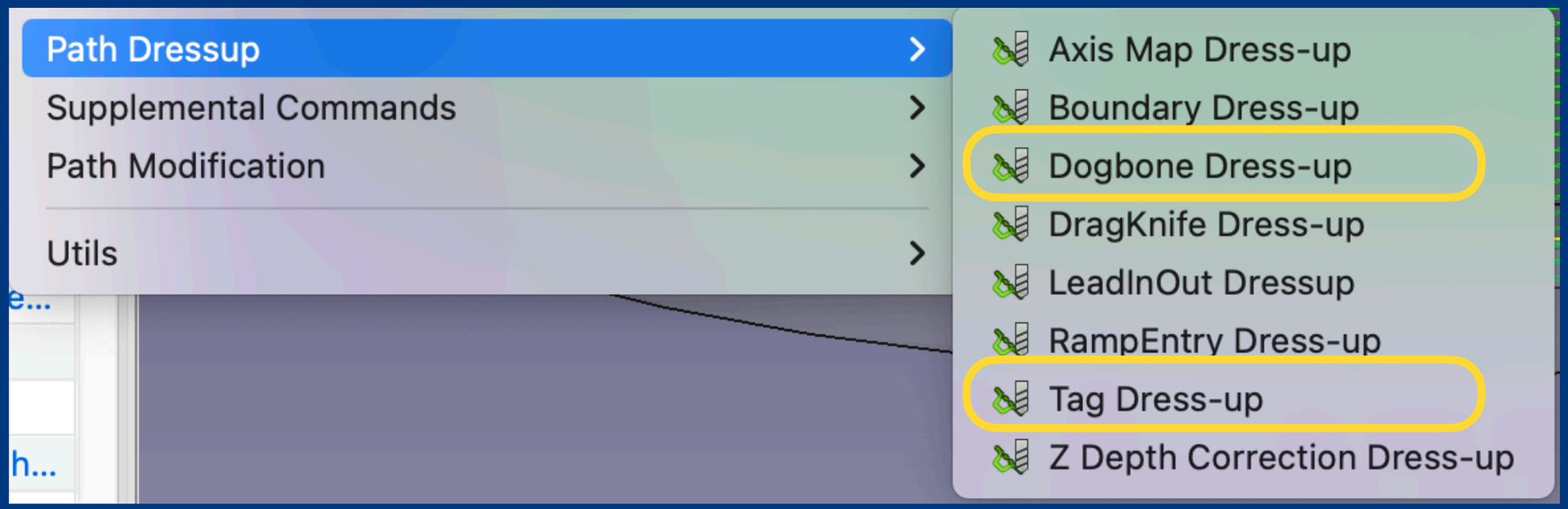
## 1. Select an operation




## 2. Select a dressup



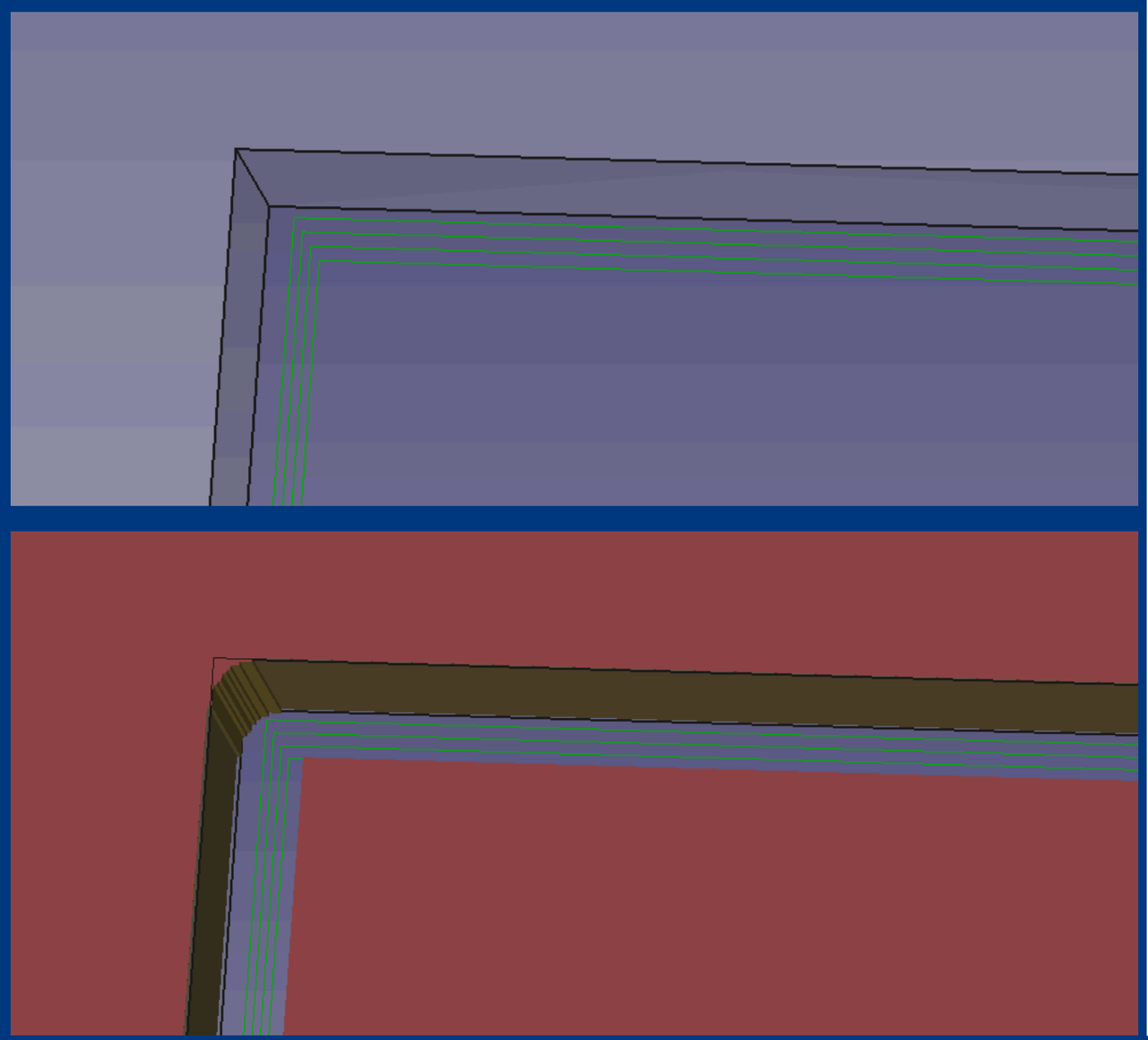
# Path Dressups



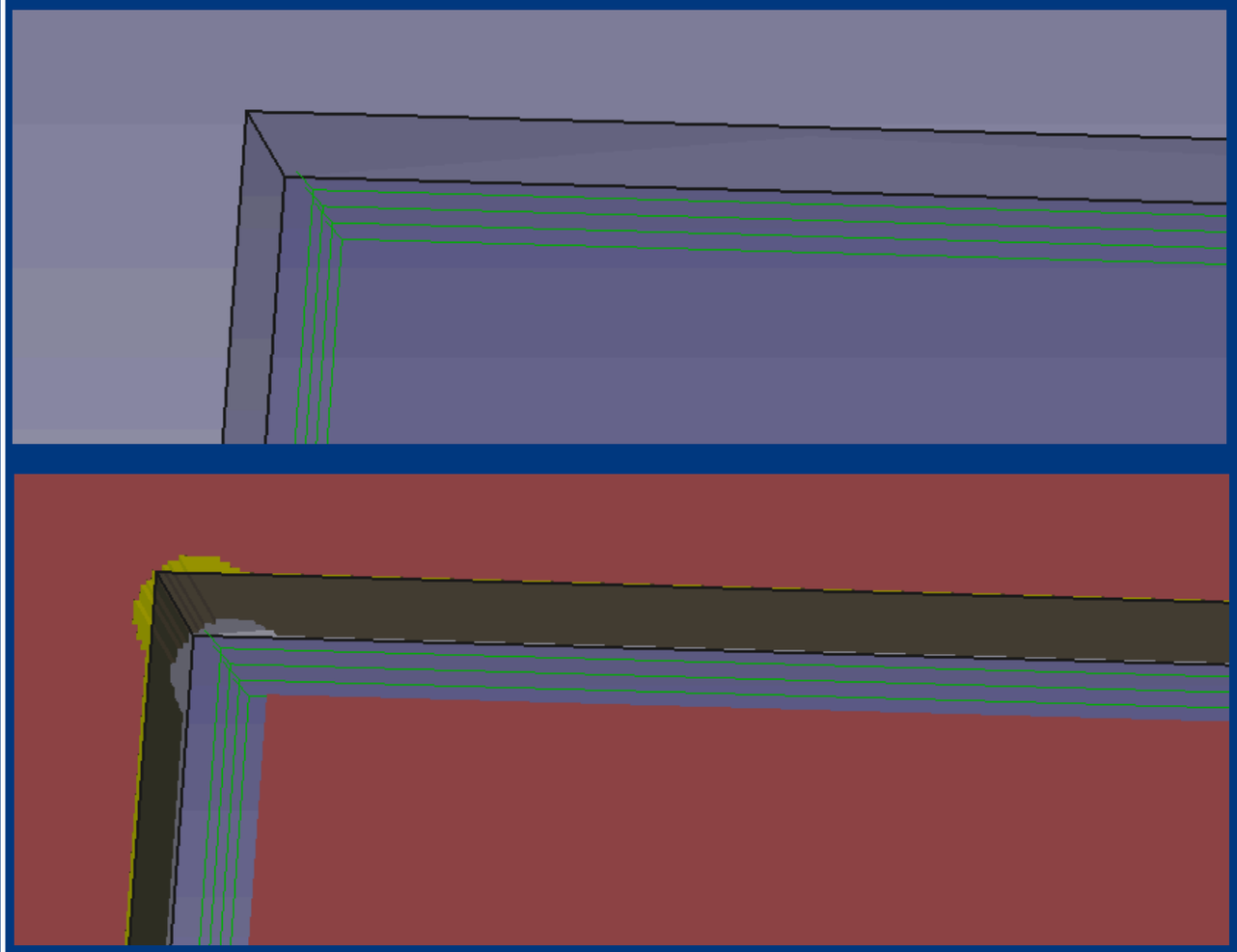

**Most common**

# Dogbone dress-up

## Before



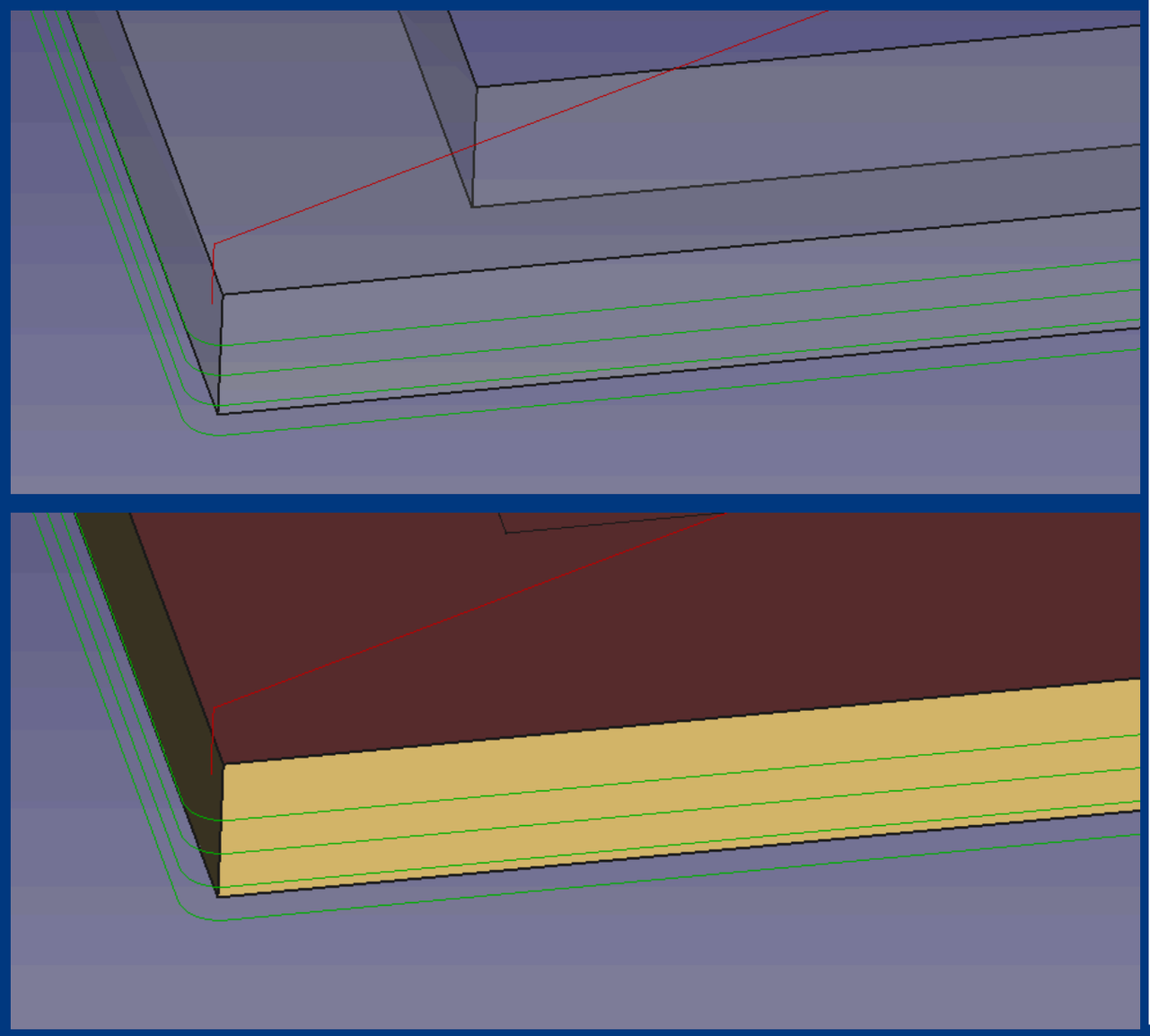
## After



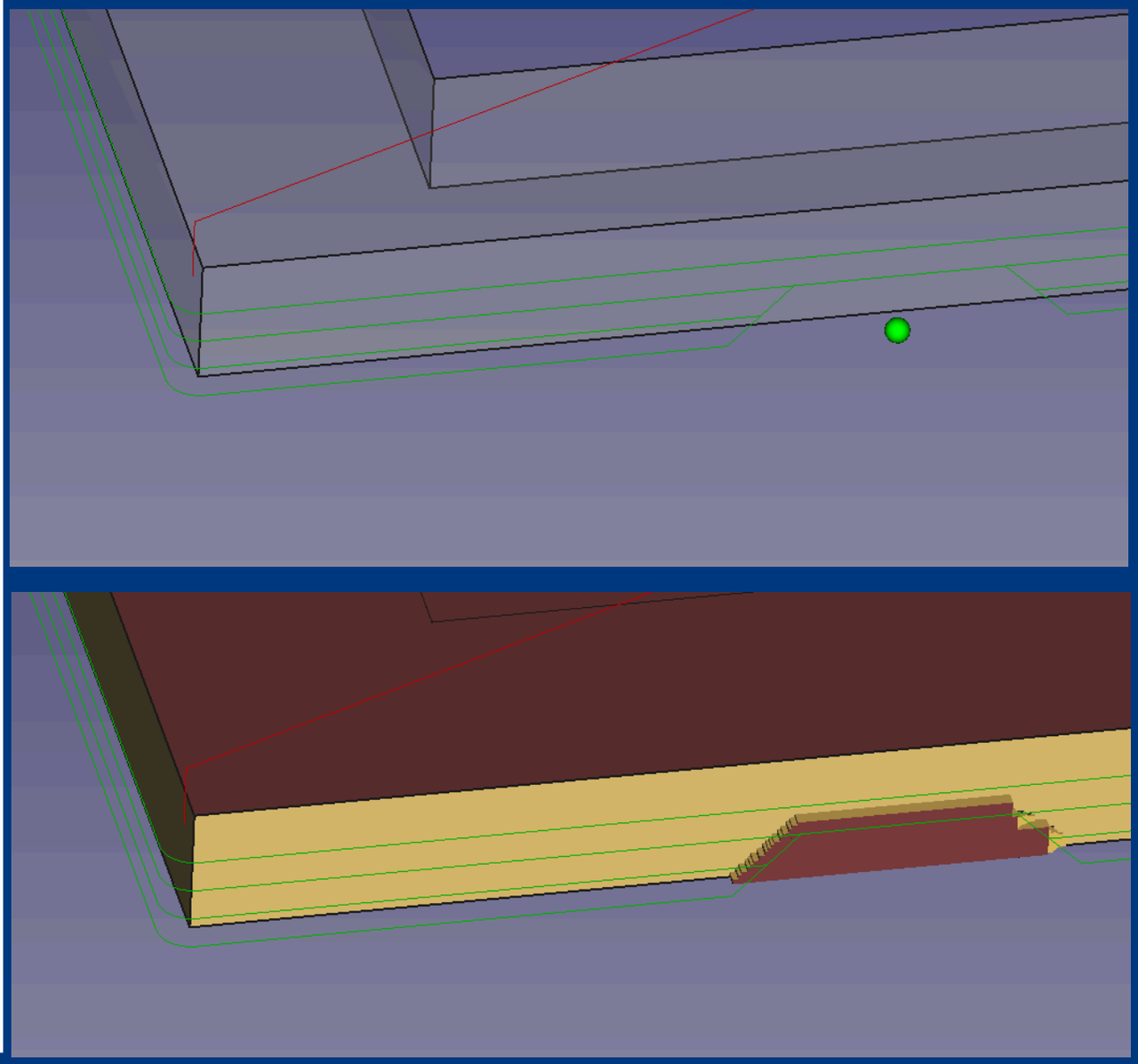
**Dogbones clear the corners of an interior cut**

# Tag dress-up

## Before



## After



**Tags stop cut material from moving around**

# CAM Simulator

Cancel OK

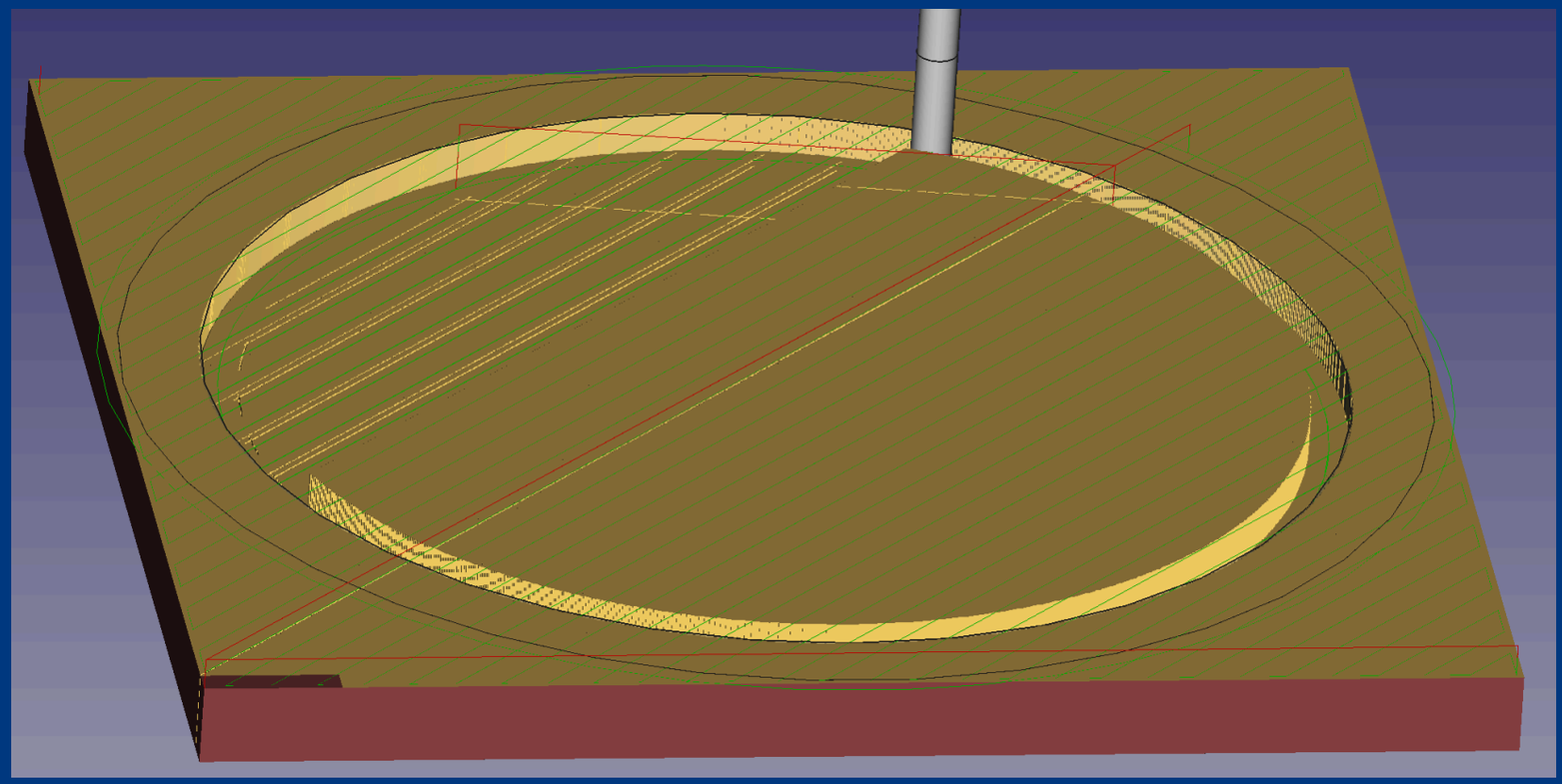
Path Simulator

Speed: 50 G/s

Accuracy: 0.1%

Job: Job

- MillFace
- Pocket_Shape
- Profile



This tool shows how a simulation of how the job will go

# Multi-bit jobs

**Sometimes you may want to use different bits for different paths.**

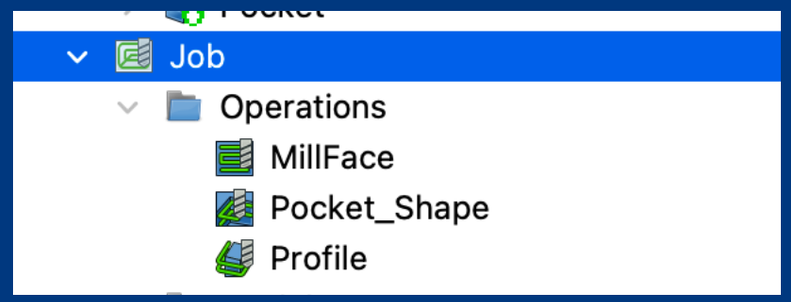
**To do this, we can selectively hide certain operations and export multiple gcodes — one for each bit**

**This icon is used to hide/reactivate operations:**

**WARNING: Path dress-ups may be lost upon deactivation**

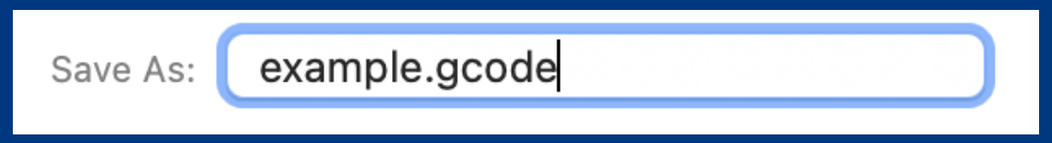
# Post processing

1. Select the whole job




2. Hit the post process button

3. Save the file with an **explicit .gcode extension**



# PART 2

# Operating the machine

# STEP 1:

# Operating the machine is a two step process.

# OPERATION OVERVIEW

## 1. Preparing the bed

- ☐ Secure bit in chuck
- ☐ Place spoilboard under material
- ☐ Check material level
- ☐ Secure material along XY
- ☐ Secure material along Z

## 2. Running the file

- ☐ Zero the machine
- ☐ Select a file
- ☐ Adjust speeds/feeds
- ☐ Run a ‘ghost pass’
- ☐ Run the file

# STEP 1:

# PREPARING THE BED

# BED PREP CHECKLIST

- ☐ Secure bit in chuck
- ☐ Place spoilboard under material
- ☐ Check material level
- ☐ Secure material along XY
- ☐ Secure material along Z

# BED PREP CHECKLIST

- ☐ Secure bit in chuck
- ☐ Place spoilboard under material
- ☐ Check material level
- ☐ Secure material along XY
- ☐ Secure material along Z

**NOTE ON BRAD NAILS/FASTENERS:**

**- Double check that they are away from the machine path**

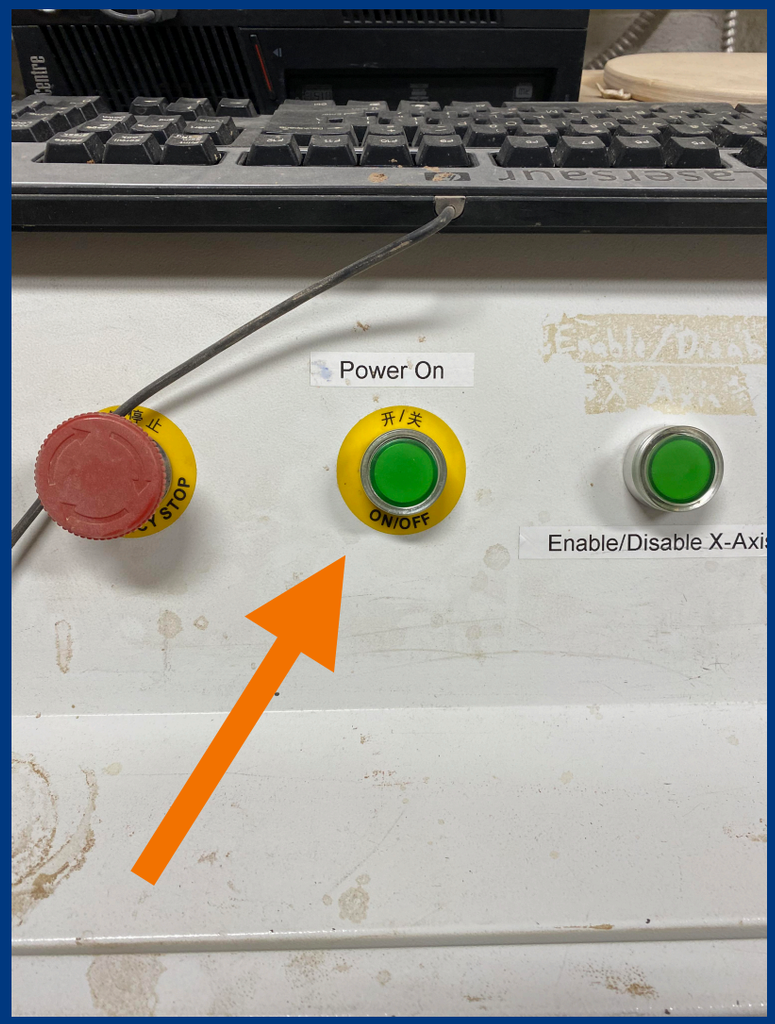
**- Do not penetrate the base CNC bed**

# STEP 2:

# RUNNING A FILE

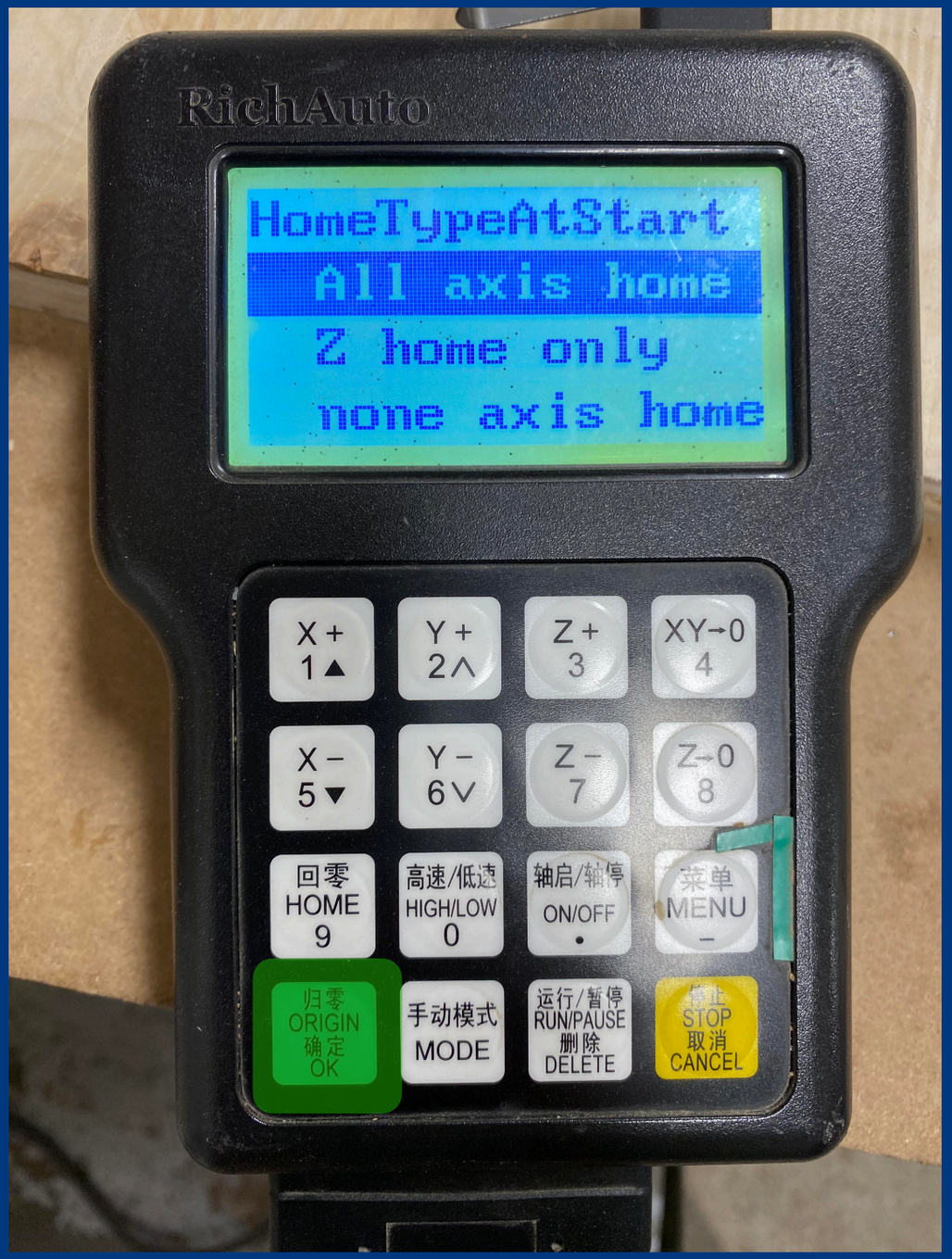
# 0. Turn the machine on

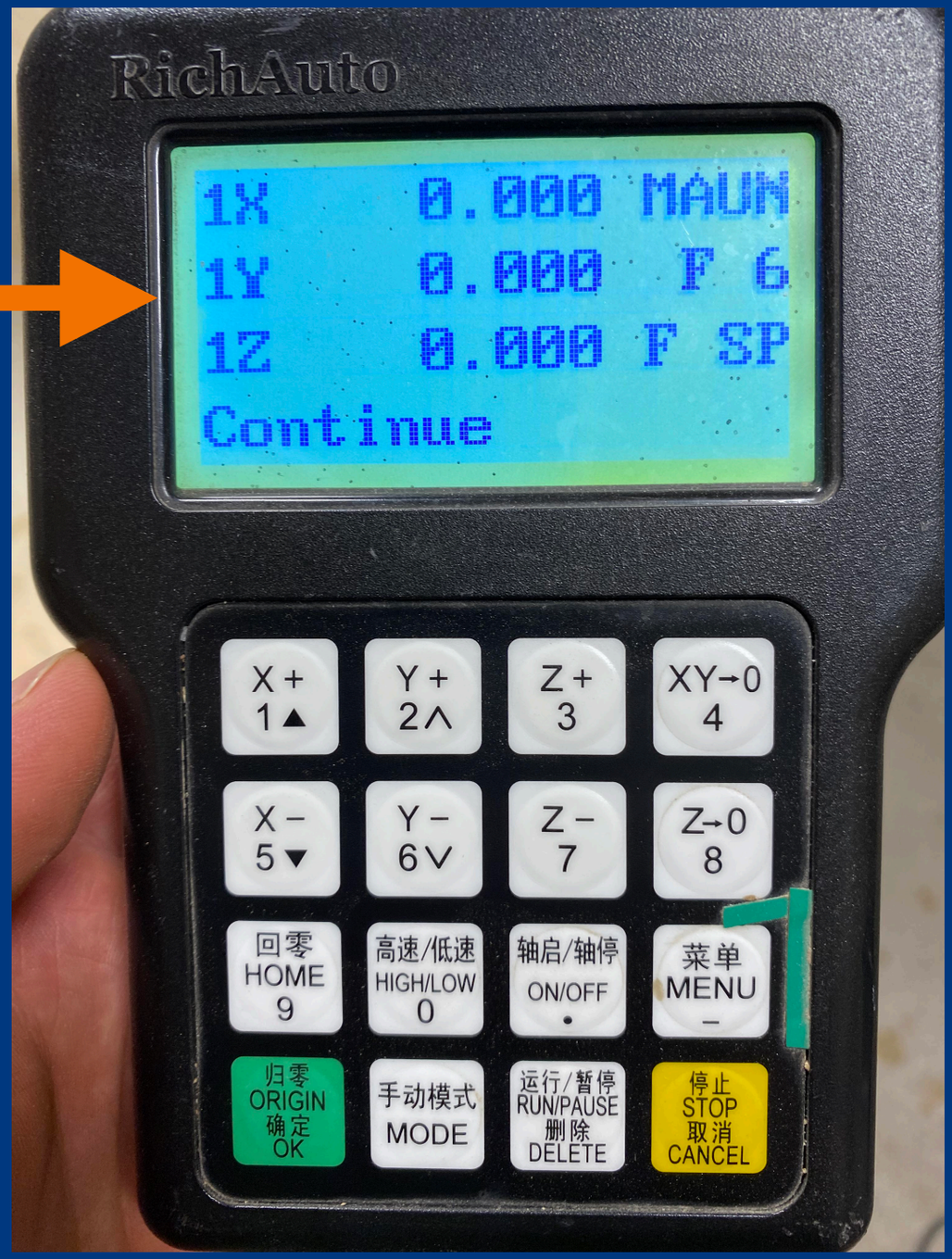

1




2

# 0. Turn the machine on




Press **OK** to select ‘All axis home’

# 1. Zero the machine

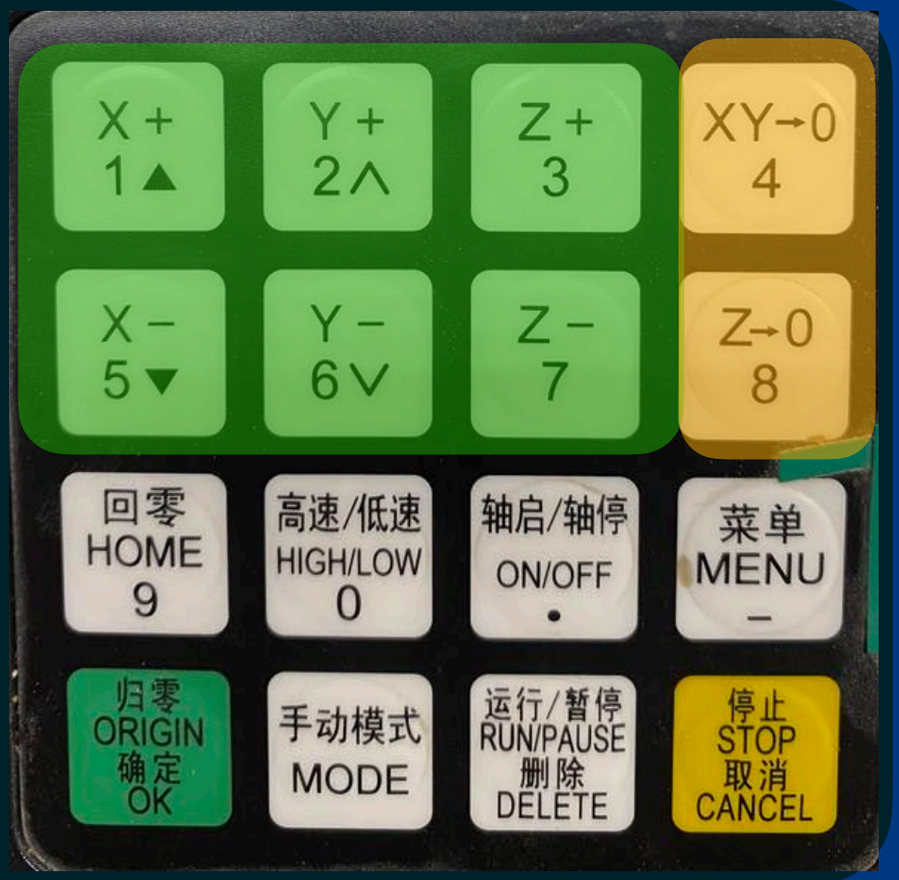
Use the **X+, X-, Y+, Y-, Z+, Z-** keys to move the milling bit around

Use the **XY->0, Z->0** to set the origin coordinates

# 1. Zero the machine

Use the **X+, X-, Y+, Y-, Z+, Z-** keys to move the milling bit around

Use the **XY->0, Z->0** to set the origin coordinates

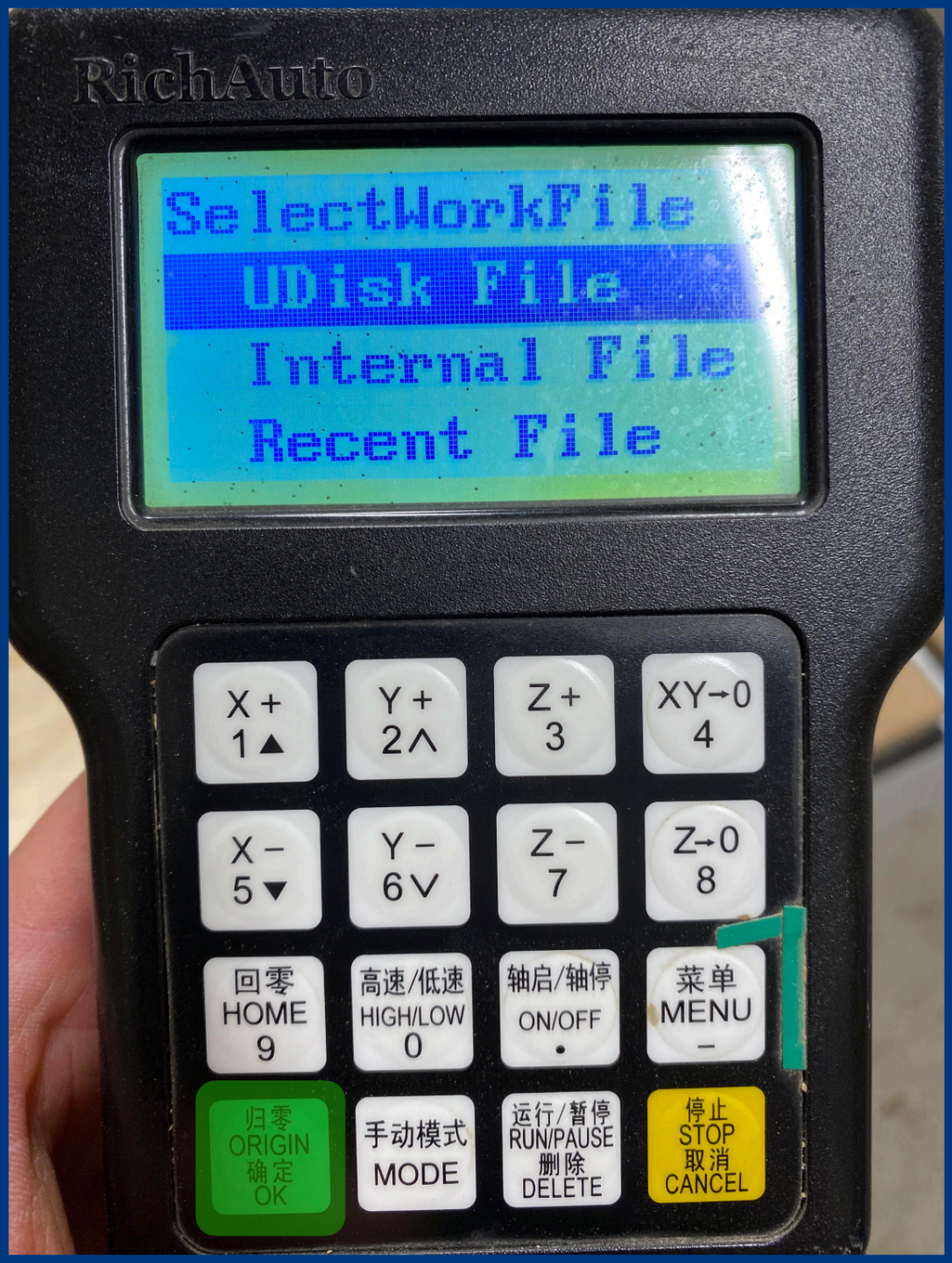



**Recommended: set Z origin above material to perform ‘ghost pass’ (see later)**

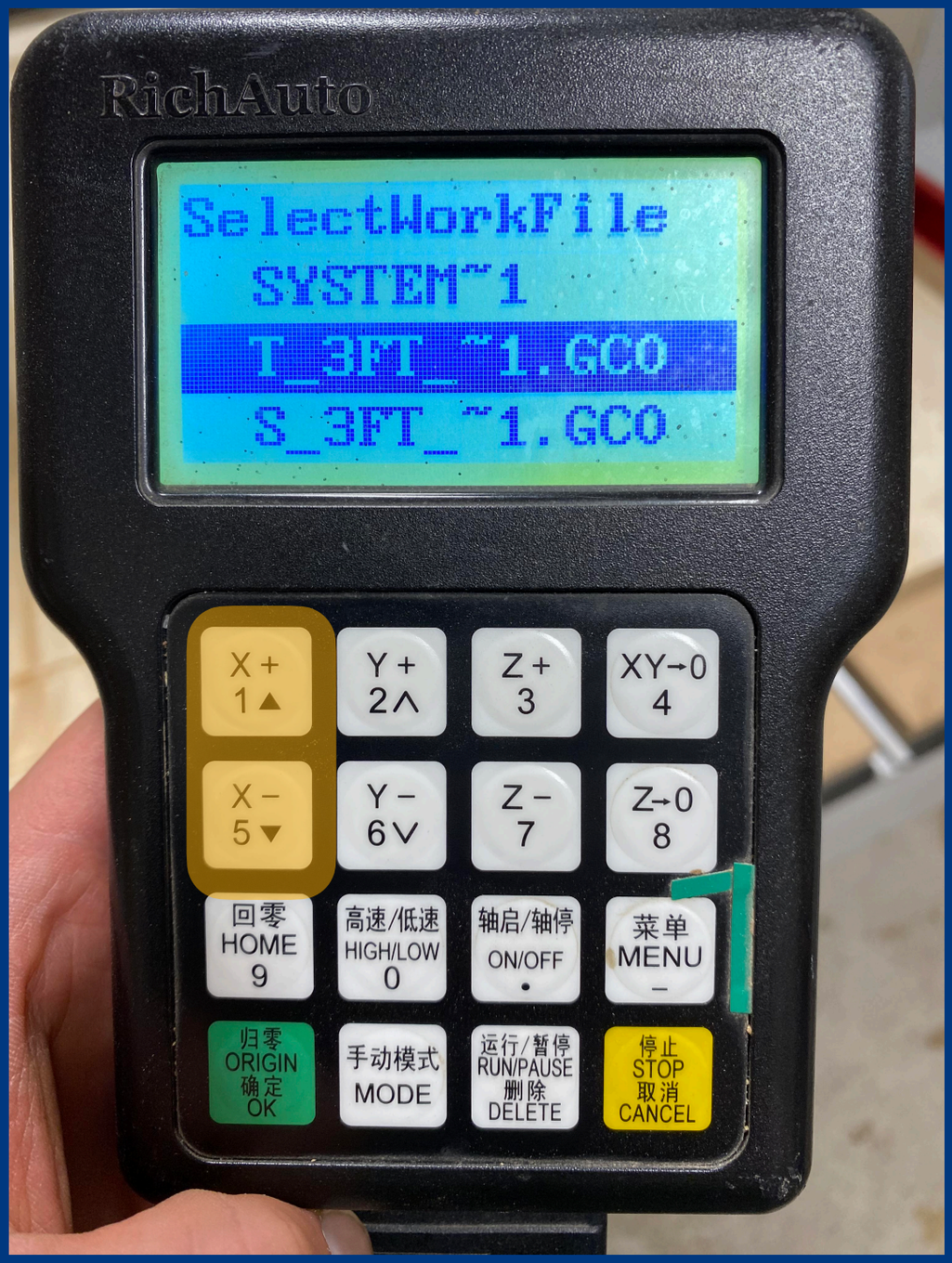
# 2. Select a file

Select **RUN/PAUSE** to begin loading your file

# 2. Select a file




Press **OK** to select ‘UDisk File’




Use **arrow keys** to highlight gcode. Click **OK** to select

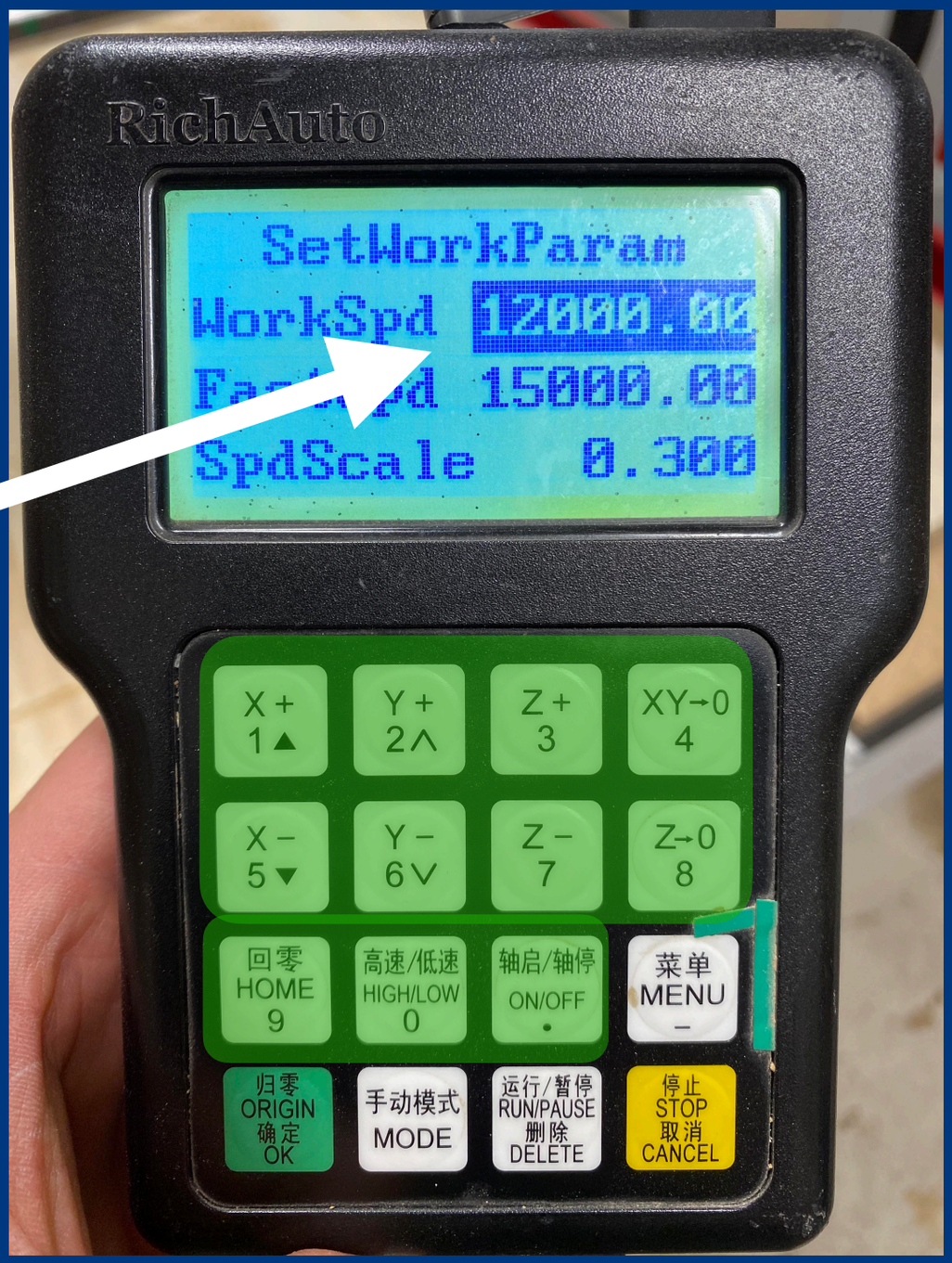
# 3. Adjust speeds/feeds


Press **DELETE** to edit an entry


Use **numpad** to type new entry

# 3. Adjust speeds/feeds

Entry can be edited when **only numbers** are highlighted




Use **numpad** to type new entry

# 3. Adjust speeds/feeds


Press **OK** to save new entry


Press **OK** again to start!

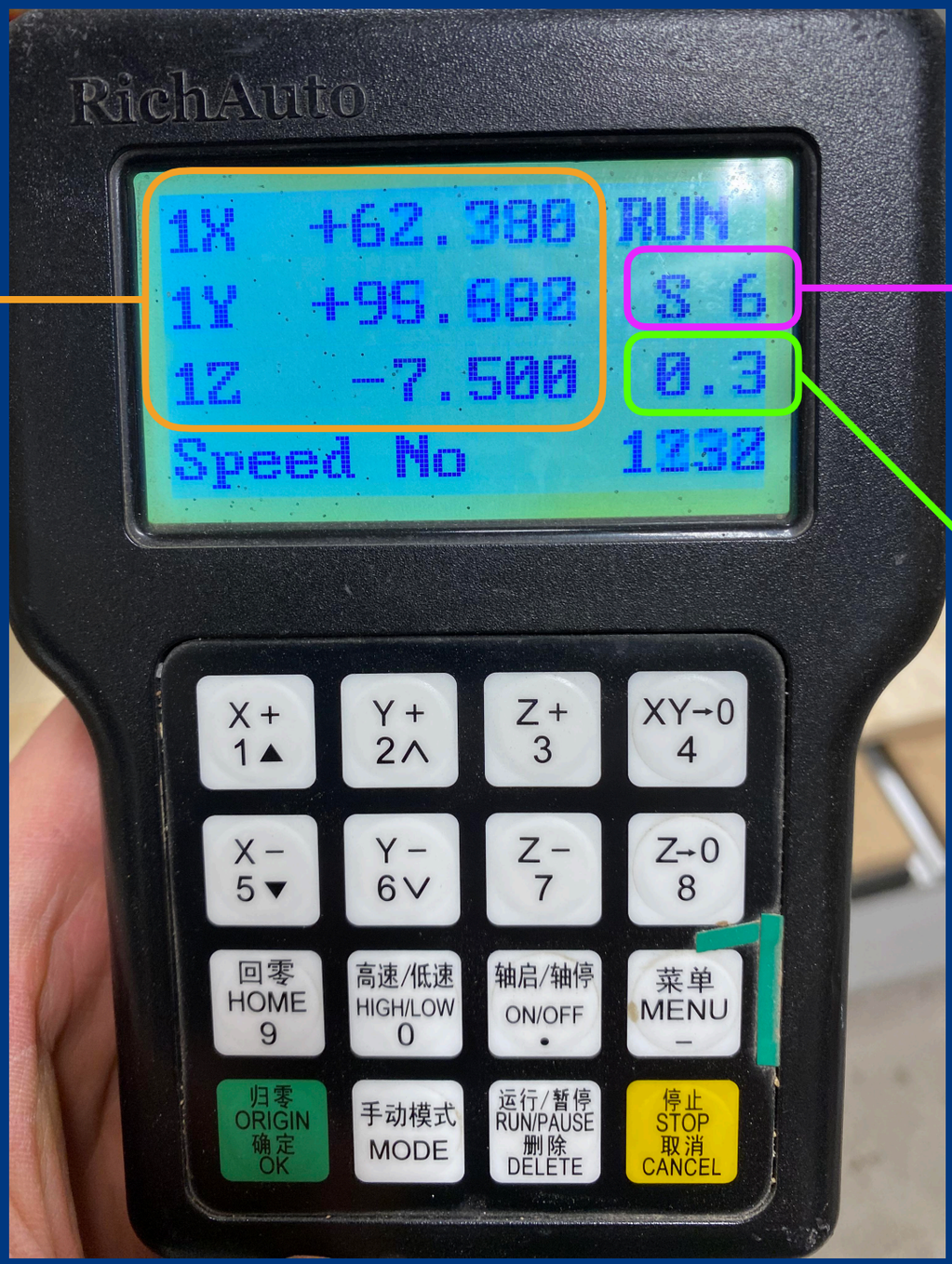
# 4. Ghost pass

Running files will display the following information:

# 4. Ghost pass

Running files will display the following information:

Coordinates of milling bit

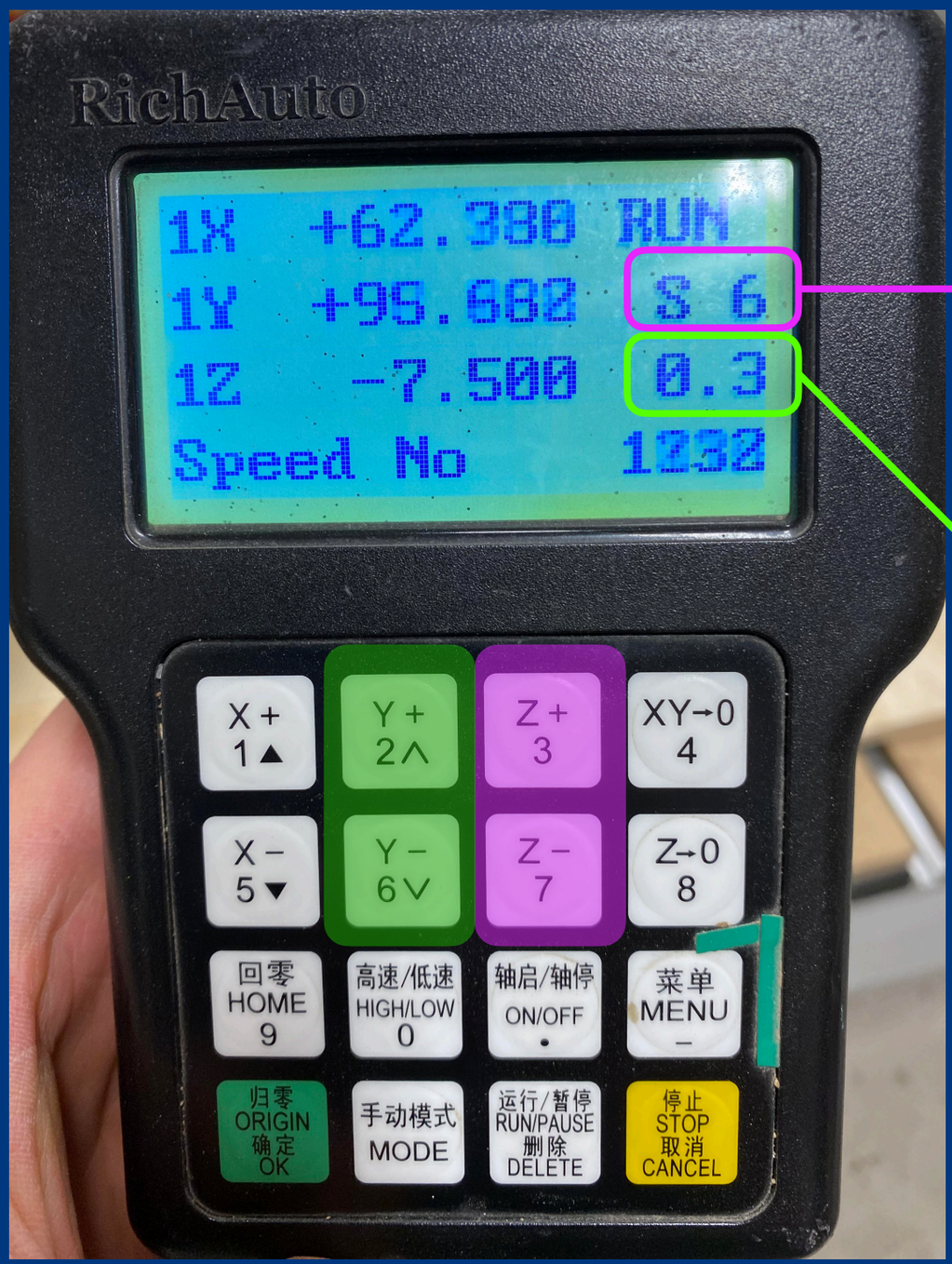



Spindle speed (on scale from 1-8)

Feed rate (translational speed from 0.1 to 1)

# 4. Ghost pass

**Adjust** the speeds and feeds during operation with the **Y+,Y-**, **Z+, Z-** keys




Spindle speed (on scale from 1-8)

Feed rate (translational speed from 0.1 to 1)

## 4. Ghost pass

Operation info will SAVE for the next run.

## 4. Ghost pass

Operation info will SAVE for the next run.

This is one of the reasons the ghost pass is recommended:

**Operators can tune the speeds and feeds before actually cutting any material**

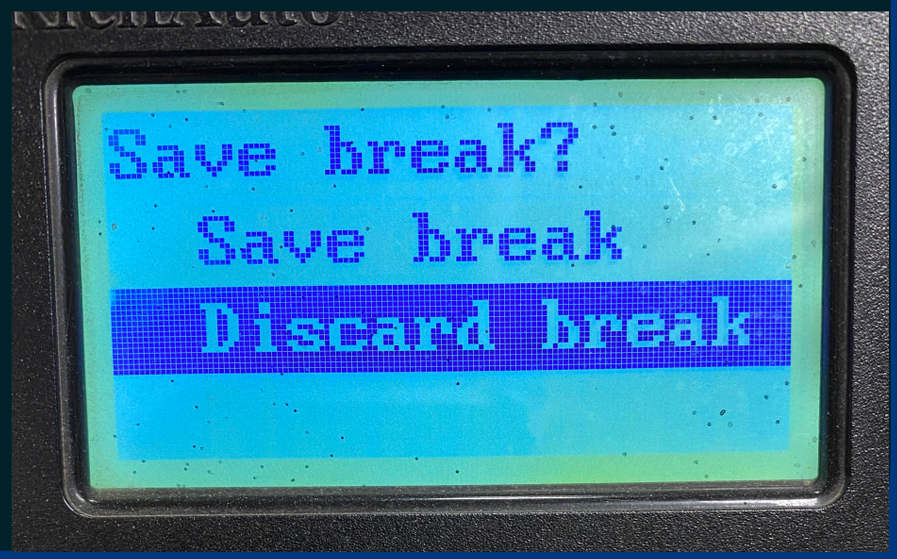
# 5. Run the file

1. When ready, press **STOP/ CANCEL** to cancel the ghost pass

2. Discard the **break** point


3. Re-zero the Z axis to the top of the material

4. Run the file (following steps from ghost pass)



# Manual operation

You can also run the mill manually.

Pressing the **ON/OFF** button will stop/start the milling bit


This can be useful for manual planing or experimentation.