Tim Savas
MIT Media Lab
PartWorks3D-to-Trak Mill Protocol

Overview

This protocol describes the process of fabricating a component on the MIT Media Lab's Trak K3 mill with a G-code-based toolpath derived from ShopBot PartWorks3D software.

The MIT Media Lab Trak K3 mill has advanced automation features capable of reducing fabrication time and human error while operating. One of these features in particular is its ability to import and execute g-code toolpaths created from external software. While the Trak's manual controls are powerful on their own, their ability to be automated make it capable of precise, multi-axial subtractive manufacturing executions with hands-off operation.

ShopBot PartWorks3D File Creation

File Selection

The MIT Media Lab shop's computer system is outfitted with ShopBot PartWorks3D toolpath design software. This portion of the protocol applies specifically to this software, though it is likely transposable to similar applications.

Start designing your toolpath by first opening ShopBot PartWorks3D.

Select "File > Open..." and choose the STL file of interest. Note that your component should already be properly modeled and rendered as an STL file. Take care to consider if your component is appropriately sized for the Trak mill's vice before proceeding with your project.

Orientate and Size Model

The "Orientate and Size Model" menu determines the model's global orientation with relation to its software and subsequent hardware. Select the "Top Surface" according to how you intend to orient the component in the mill's vice.

Next check that your Model Size is correctly proportioned. X, y, and z values should be identical to the according extents of your original model, whether rendered from Solidworks, Rhino, or other CAD suite. Not the case? Try selecting "mm" or "inches" from the Units pane. Otherwise check your original model's export parameters or consider scaling the component in PartWorks.

Lastly, select "1-Top" under the Sides to the Machine pane. Select "Apply."

Material Size and Margins

The "Material Size and Margins" menu provides parameters for setting the dimensions of your project's stock material and cutting tool margins. When looking at the object in PartWorks, note the following colored components:

- turquoise: the STL/component to be created
- wireframe: the material from which the component will be milled
- gray: the tooling margin, or the allowable area through which the tool can travel

Under the menu, populate the Material Size dimensions according to the dimensions of your stock. Take care to determine the order of your X and Y axes as they relate their future position in the mill's table.

Set the Z Zero to the top of the material.

Set the XY Origin position to X: 0.0 and Y: 0.0.

Set the Machining Margins Around Model according to your end mill diameter and the constraints of your stock. Fair practice is to set the margin to around 1/8” greater than the diameter of the end mill. The tool should not need any more margin space beyond its diameter to cut the component.

Set the Depth of Model below Surface to 0.0.

Adding tabs may be a critical step to producing your toolpath. If your component is to be extracted from your stock from a profile cut, and thus detached from the vice from the tool’s last passes, your toolpath requires tabs.

Roughing Toolpath

The project’s roughing toolpath optimizes for removing initial bulk material from the stock. If possible, design the toolpath around the largest end mill diameter possible for the cut, typically around 1/4” for my own projects.

The project’s finishing toolpath follows the roughing toolpath, and typically cuts with higher step resolution for a finer finish. Design the toolpath around the largest end mill diameter possible for these final passes, typically around 1/8” for my own projects.

Setting these toolpath parameters are common across most all G-code generative software suites. In short, different combinations of materials and cutting tools incur corresponding chip loads. These chip loads can be accounted for by adjusting the feed rate, pass depth, spindle speed, and other cutting parameters.

After determining your material (e.g. polycarbonate, brass, aluminum) and end mill, use the standard “speeds and feeds” chip load calculation to derive these tool parameters. Excellent guides and rubrics sorted by material can be found here on Onsrud.

Preview Machining

Selecting "Calculate" at the end of each of the previous toolpath menus will render a blue line preview of the two toolpaths. This is the virtual path to be travelled by the end mill. How does the toolpath look? Do margins and stock material sizes look correct? Are tabs placed correctly?

For another look, a full virtual preview of both the roughing and finishing paths can be played on this menu. Select Roughing Toolpath Preview, then select Finishing Toolpath Preview. This is how your component will appear at the end of the mill operation. How does it look? Are you thrilled with it? Great.

Output ToolPaths

This step must be completed on either shop administrator Tom Lutz's computer or another machine with the proper post-processor. The post-processor in question is:

> shopbot g-code arcs herco (inch) (*nc)

Export your roughing and finishing toolpaths as two separate files. This is my preference, as my roughing and finishing toolpaths often require their own end mills, and therefore should not occur synchronously from the same toolpath file.

After exporting, you must change the file names' extensions from ".NC" to ".CAM." If this step is not taken, it will not be recognized by the mill.

Place the files on a USB stick.

Trak Setup

Start the file upload by inserting your USB stick into the mill's computer.

Select the SYS button on the lefthand side of the mill's computer interface. In the menu tab, ensure that the second tab reads "Go to 3 Axis." If it reads "Go to 2 Axis," press the tab to toggle it to "Go to 3 Axis mode." Video instruction of this step can be found here.

Next select the PROG tab on the mill's main menu. Follow the file tree to your USB stick and highlight your project file. With it highlighted, select the "Open File" tab on the menu. A load bar should appear indicating that the mill is verifying the file. This compiling process should conclude after just a few seconds, with its completion being indicated on screen by a green bar reading "Select a mode."

Press the MODE button on the computer's physical interface, and select the "DRO" tab. Here is where you will set your x, y, and z axis "zero" positions. With your material placed appropriately in the vice and necessary edge finding tools, select your zero positions. This step is not described in full detail in this protocol as any operator of the mill should already be well acquainted with it.

After setting your zeroes, push the MODE button, and then select the RUN tab followed by the START tab.

Video instructions for the above steps can be found here.

Press the GO button on the computer's physical interface.

Z-Axis Error Troubleshooting

It's possible that you may receive the error:

> Critical error 109: Servo Error

If this is the case, it is likely that the machine is sensing that its quill is too close to the table along the z-axis direction. To resolve this issue, start by selecting RETURN. Select SETUP followed by the REF POSN tab. Then physically move the quill upward roughly one inch along its z-axis and select ABS SET to reset its zero position.

Once again select the MODE, RUN, and START tabs, followed by the GO button. The error should be cleared. If the error is not cleared, attempt these steps again but withdraw the quill along its z-axis 2-3 inches.

Video instructions for the above steps can be found here.