

# AXIS Overview

Tutorial



# Table of Contents

## AXIS OVERVIEW

RUNNING A GCODE FILE

BUTTONS IN THE AXIS INTERFACE

MDI INTERFACE

PREVIEW MODE VS DRO MODE

## AXIS OVERVIEW

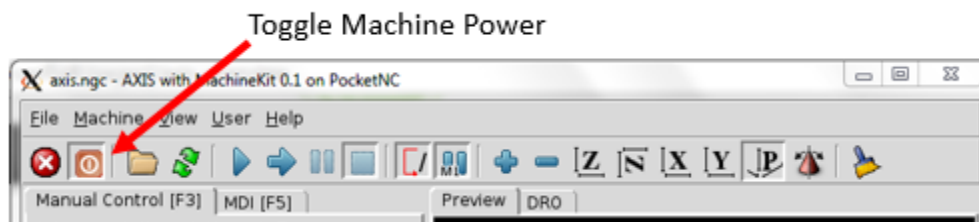
AXIS is the LinuxCNC GUI used to control the Pocket NC. This tutorial will go over the basic use of AXIS with a Pocket NC.

For general information on AXIS, [click here](#).

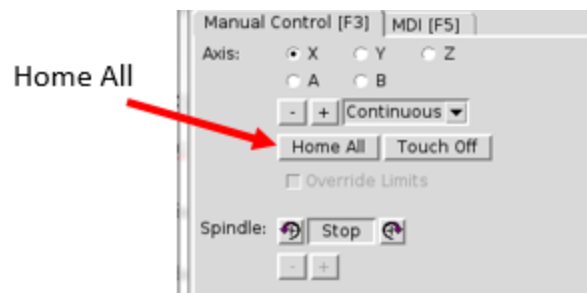
## RUNNING A GCODE FILE

Once you have launched AXIS using the instructions in the [Software Setup Tutorial](#), the first step is to turn the machine on. You will do this using the orange "Toggle Machine Power" button shown below.

When the machine is turned on, you will hear the motors engage but nothing will move yet.

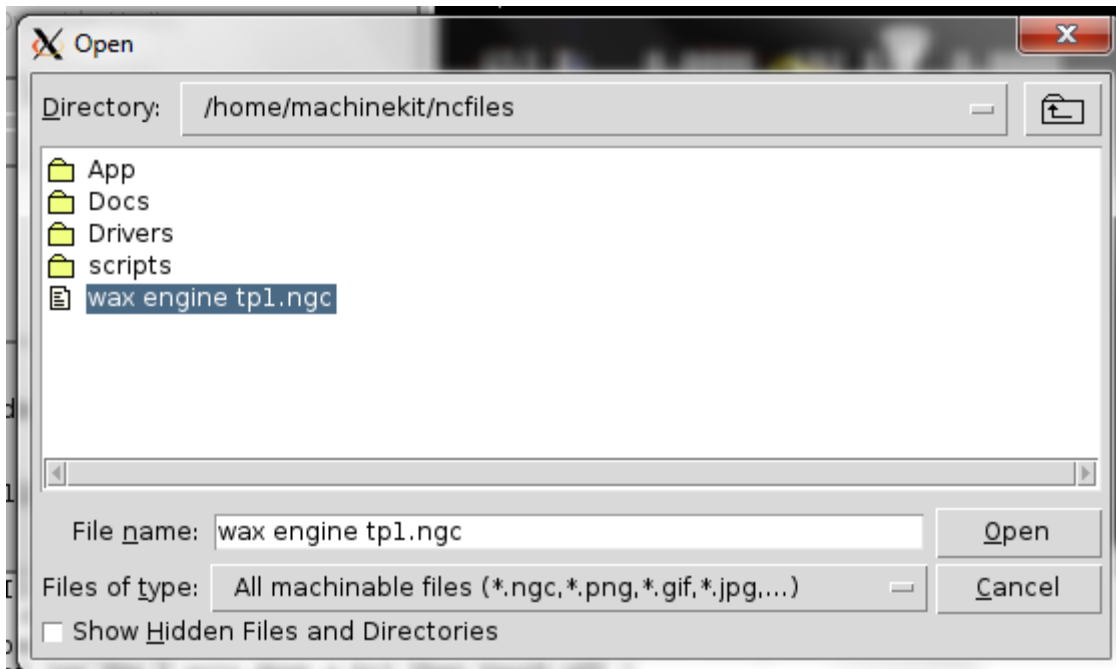


The next step is to home all 5 axes using the "Home All" button shown below. After pressing the button, the axes will move into their home positions. This will take around 40 seconds to complete.

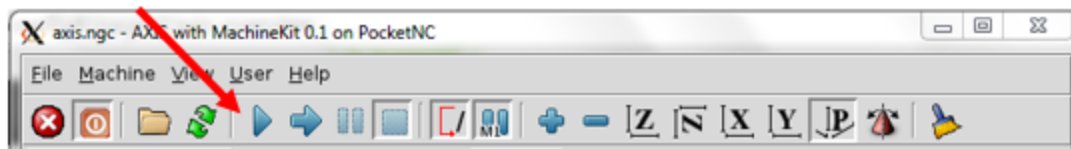


At this point in time, you will want to set your tool length offsets. If you don't know how to do that, check out our Tool Length Offsets tutorial.

Once your offsets are set, load your Gcode file using File>Open as shown below. The mouse will turn to a watch icon for a few seconds as the Gcode loads. Then press the play button to begin executing the current file.



### Begin Executing Current File



At this point in time, your Gcode may pause if it has run into an "M1" command which tells the machine to pause and wait for the user to load the next tool. If you already have the correct tool loaded, you can click that button to resume the execution. Otherwise, load the correct tool and proceed. This will also be the process for other tool changes throughout a Gcode file. You can also use this button to pause the machine at any point during your program. It will pause all 5 axes, but leave the spindle running. The yellow button on the machine has the same functionality. If you are running a program for the first time or unsure of your toolpath, we recommend keeping a finger ready to press the pause button as that will help to avoid plunging into the material or crashing the machine if you notice something is not right.

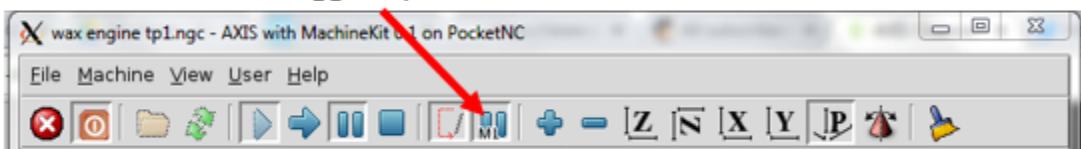
## Pause/Resume Execution



## BUTTONS IN THE AXIS INTERFACE

Another feature that should also be pointed out is the "Toggle Optional Pause" button shown below. When it is engaged, this tells the machine to stop and wait for manual tool changes as explained above. For the Pocket NC, this button is active by default, and should remain engaged, otherwise the machine will not wait for the manual tool change and will continue machining with the current tool.

## Toggle Optional Pause



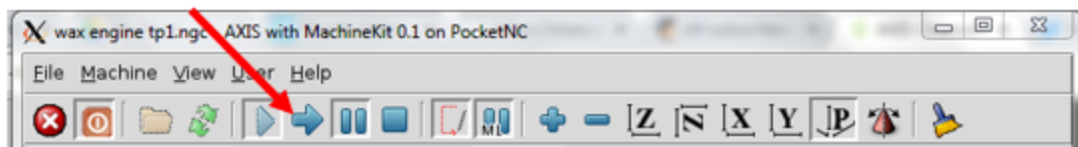
The Toggle Skip Lines with "/" will skip all the lines in your Gcode that begin with "/". This can be useful for leaving comments in your file for later reference or to remove lines of Gcode that you don't want to be used.

## Toggle Skip Lines with “/”



The Execute Next Line button will execute the next line of Gcode and then wait for user input before continuing. This can be helpful if you have a stretch of Gcode that you aren't sure about and need to proof out, but you probably won't want to run your whole part this way.

## Execute Next Line

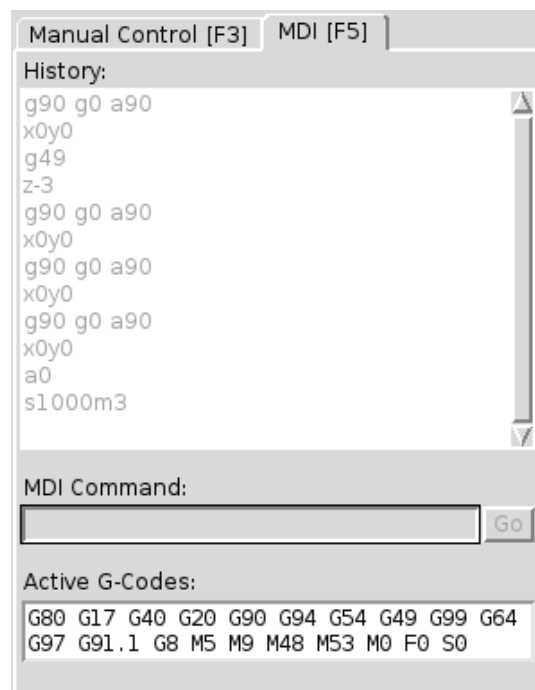


## MDI INTERFACE

AXIS includes a program called mdi which allows text-mode entry of MDI commands to a running LinuxCNC session.

When a blank line is entered, the machine's current position is shown. When a command is entered, it is sent to LinuxCNC to be executed.

This is a sample session of MDI.



## PREVIEW MODE VS DRO MODE

AXIS defaults to the preview mode to preview a tool path. This mode can slow down the BeagleBone and its connection to your machine (especially when using an HDMI screen). If this is happening, switch to DRO mode to improve processing speed.

