



## UCCNC – Axiom CNC Quick Start Guide



BG Precision PTY LTD

Email: [info@bgprecision.com.au](mailto:info@bgprecision.com.au)

Website: [www.bgprecision.com.au](http://www.bgprecision.com.au)

BG Precision


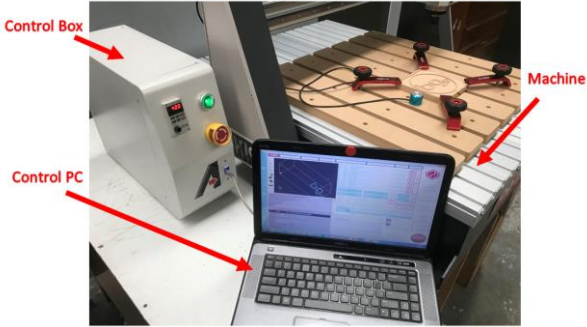

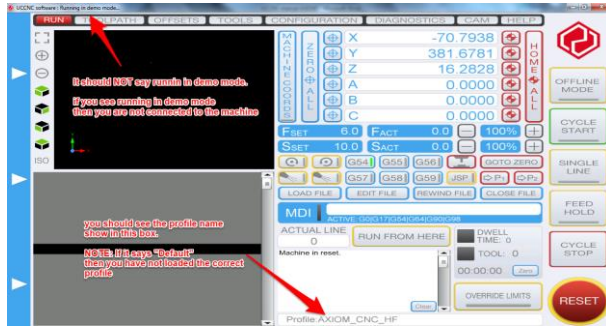

Version 1.1




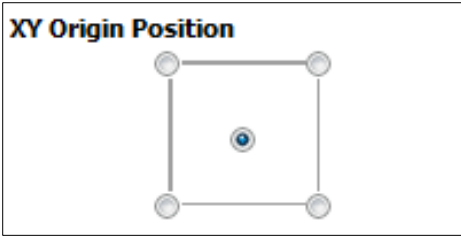

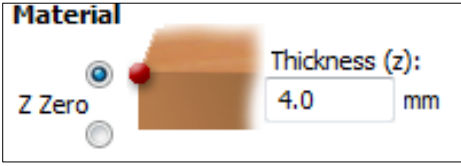

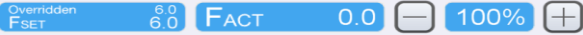

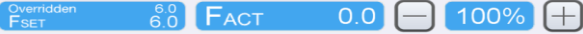
Date: 15 Jan 2019



**! WARNING**

Read & understand operators manual before using this machine. Failure to follow operating instructions could result in death or serious injury

	<p>This quick start guide assumes you have read and understood the “<b>Axiom CNC Router User’s Manual</b>” in its entirety. It also requires of you to have completed the <b>AXIOM SETUP AND CONFIGURATION</b>.</p>	
<p>The purpose of this guide is to remind you of the basic steps to follow prior to starting your job. These steps should be followed every time you start the machine and/or software.</p>	<ol style="list-style-type: none"> <li>1) Securely located your machine on your workbench or AXIOM supplied stand</li> <li>2) Connect your control box to your Axiom machine</li> <li>3) Turn on the control box</li> <li>4) Ensure you have a windows-based laptop running beside the machine</li> <li>5) Connect the machine to the control computer using CAT5 Ethernet cable provided</li> </ol>	
	<ul style="list-style-type: none"> <li>• When UCCNC and Profile installs correctly you will have a machine specific profile on your desktop</li> <li>• NOTE!!!! When UCCNC installs for the first time there is a generic shortcut always loaded to the desktop.</li> <li>• VERY IMPORTANT you either delete the generic shortcut or understand that it won’t be the correct profile.</li> </ul>	
<p>When you have established a successful connection between your Axiom CNC and Control PC and run the “Machine Specific” profile, you should note the following:</p> <ul style="list-style-type: none"> <li>• top left-hand side. You <u>SHOULD NOT</u> see the text “Running in Demo mode”. It should just say UCCNC software after the icon.</li> <li>• In the profile summary box (next to red reset button) you should see the text “AXIOM_AR4” or similar, showing you have loaded the correct profile.</li> </ul>		
 <p>NOTE – Whenever you are not directly interacting / controlling the machine you must click the reset button, so the <b>button flashes red and yellow</b>. This is a safety procedure to make sure you don’t accidentally turn on the spindle system.</p>	<ul style="list-style-type: none"> <li>• Right arrow key – jog X right / positive</li> <li>• Left arrow key – jog X left / negative</li> <li>• Up arrow key – Jog Y up / positive</li> <li>• Down arrow key – Jog Y down / negative</li> <li>• Page up – Z axis up / positive</li> <li>• Page down – Z axis down / negative</li> <li>• Holding the “Shift” key while pressing the directions of movement keys will move the machine in rapid mode.</li> </ul>	

<p>1. <b><u>Home All</u></b></p> 	<p>This function aligns the software limits (or softlimits) for your machine with the actual working boundary on your table. For the softlimits to work properly you must home the machine every time you turn the machine on.</p>
<p>2. <b><u>Load your G-Code</u></b></p> 	<p>The G-code file is generated from Cut2D/Vcarve and from there loaded into the UCCNC software</p>
<p>3. <b><u>Clamp Your Work Piece</u></b></p>	
<p>4. <b><u>Load your Tool</u></b></p>	<p><b>DO NOT TOUCH</b> the spindle unless the reset button is flashing red and yellow on UCCNC control interface on your PC</p>
<p>5. <b><u>Set X and Y working origin</u></b></p> 	<p>Jog the machine such that the centre of the spindle is in line with the defined origin position you have set in the “Job Setup” in Vectric. When in position you can select the set axis origin button for the applicable axis as below:</p> 
<p>6. <b><u>Set Z working origin</u></b></p> 	<p>To set the z working origin, plug in the Axiom Touch off puck provided with your machine into the back of the left side upright. Place the puck either on the material surface or the table surface as per your defined origin position you have set in the “Job Setup” in Vectric. Jog the machine over the puck and run the auto touch off command:</p> 
<p>7. <b><u>Reduce Feed rate %</u></b></p> 	<p>Prior to starting the cycle you can override and reduce the feed rate set in your G-Code by clicking the “-“ button on the Fset line. This will slow the machine down giving you more reaction time if your set-up is incorrect for machining</p>
<p>8. <b><u>Start cycle</u></b></p> 	<ul style="list-style-type: none"> <li>• Verify the working area is sufficient for machining</li> <li>• Start Cycle</li> <li>• <b><u>Observe and listen</u></b></li> </ul>
<p>9. <b><u>Increase Feed Rate %</u></b></p> 	<p>After you have observed the start of your cycle and believe you set-up is correct and the machine is performing appropriately, you can increase the feed rate set in your G-Code by clicking the “+“ button on the Fset line.</p>