Bobs Simple Cost Effective Designs

Bobs CNC E3 Controller setup for the EstlCAM controller with DRV8825 stepper motor drivers.

EstICAM is a great place to start and will work for the seasoned user. It has a lot of features that allow for custom g-code files to be create. The author has created tutorials that are intuitive and will have you creating g-code quickly. The video tutorials can be viewed here:

https://www.estlcam.de/

EstlCAM can be used to create g-code files to run on most CNC machines. However, included with EstlCAM is a g-code sender that will work with controller on the E3 CNC router. If you want to try it out you will need to change firmware and adjust the hardware. The firmware will need to be re-flashed from Grbl to the EstlCAM firmware. If you decide to go back to Grbl, the software has a "Restore Controller" button to return you to the default grbl firmware. You will also need to change the micro-stepping jumper settings on the controller itself.

If you are ready to give it a try, this instruction will help guide you thru the process. If you have any questions. Please email us at <u>helpdesk@bobscnc.com</u>

The first step is to change the firmware to the EstICAM controller. In EstICAM, please click the menu SETUP >> CNC CONTROLLER. This will bring up the controller setup menu. There is a pull-down text box that will have the "BobsCNC E3" listed. Select the "BobsCNC E3" and EstICAM will update the values for the E3 controller.



Next, will click the "Program Controller" button and follow the prompts to finish installing the EstICAM firmware

File View Setup Help						
			То	ol list		
1				File View Sort		
Select			No	. Name Ø	<mark>⁻₋</mark> Z+ ∑α(z+)	F(xy) F(z)
				1 End mill 1/8 0.125	" 0.050" 90.00°	25ipm 8ipm 2
Part			_	2 End mill 3/16 0.188	" 0.080" 90.00°	60ipm 35ipm 2
				3 End mill 5/16 0.313 4 End mill 3/8 0.375	0.100" 90.00"	70ipm 35ipm 1
				5 Engraving bit 0.750	0.400" 90.00	50ipm 50ipm 2
Hole	CNC controller settings					×
J	Basic settings Spindle Inputs Outputs Homing	Tool length sensor Touch plate Keyboa	ard and Mouse Gamep	ad Pendant Analog	inputs Misc	
Engraving	Controller hardware:	Bobs CNC E3	Overview Installation	7		
1	USB / COM port:	COM12	,			
Carve	¥.	Y. 7.				
	Steps per revolution: 1600	1600 🚔 800 🚔	+			
	Distance per revolution: 1.575"	1.575" 🔶 0.056" 🔶	;			
Drill	Maximum feedrate: 300ipm 🚔	300ipm 🔶 15ipm 🔶	+			
ABC	Inertia: 75.0%	75.0% 🔷 75.0% 🔶	+			
Text	Reverse direction:					
	Acceleration distance: 0.118" 🔶					
+	Start feedrate: 2ipm 🚔					
Zero	Step pulse length: Automatic 🔶	Restore Controller				
	Invert stepper enable signal 🛛	Open settings				
Preview	Program Controller	Save settings				
	CNC controller state:					
	CNC Controller disabled					
					V OF	c

Once the firmware is changed you will not be able to connect using a grbl g-code sender like UGS Platform. If you need to restore the grbl firmware, EstICAM has a restore controller button that will put the default firmware back to the controller.

The final step is to change the micro-stepping resolution to match the values in EstICAM.

Unplug the power supply before making hardware changes to the Controller or the control may be damaged.

The stepper motor drivers resolution (step size) selector inputs (MODE0, MODE1, and MODE2) will need to be changed on the controller.

The X1, X2, and Y step size will be a 1/8 step and require a jumper on MODE0 and MODE1

The Z step size will be a 1/4 step and require a jumper on MODE1

To change the jumper settings: watch a how-to video here: LINK

To change the jumper settings:

- 1. Unplug the power supply
- 2. Remove the stepper moto driver. Note orientation.
- 3. Add/change the jumper setting to match the picture. Note that the jumpers are shown in orange. Do not remove the jumpers in red. You will need to find three extra jumper or purchase them.
- 4. Replace the driver with the same orientation

