

Kansas City Woodworkers' Guild, Inc.

How to make a simple sign or carve the top of a box

1. **Bring your own bit:**
 - a. Go to Woodcraft or elsewhere and buy a 60 degree, 0.5" diameter pointed sign lettering router bit. One quarter inch shank is fine; one half inch shank is OK too. If they do not have this bit, you may use up to a 90 degree pointed bit; up to 1" in diameter.
2. If you have never used the CNC machine before, find a member with at least some experience to help you over any problems you may encounter.

Creating Your Project in Aspire

3. Turn on the computer (button on front) and monitor (button on front).
4. After the computer boots, double-click to select the Aspire Short Cut. It has a little curved arrow to indicate that this icon is a short cut.
5. When the program opens, select create new file – a job setup screen will appear.
6. Enter the width, height (front to back), and thickness of your material.
7. Select your XY datum. This is the starting point and reference point that the software will use. If you want your carving in the center of your material, use the center point.
8. Inches should already be selected.
9. Click OK – should go to the drawing screen.
10. Click on the "Text in a box" icon. It is a capital "T" with a dotted line around the "T". It is under "create vectors"; 2nd icon in the third row.
11. Enter your text (including line breaks) in the box at the upper left corner of the screen.
 - a. The computer will make your text as large as it can in the space that you specify below.
 - b. If you enter a single long string of text, the letters will be very short.
12. Select a font from the dropdown menu. Simple, clean fonts will be easiest to read. Aerial is a good choice. Select Bold or italics if you wish.
13. Select text alignment as center.
14. Enter the length and width of the box for the text. Consider one inch less than the size of your material.
15. Change the margin size to none.
16. Click Apply – lettering will appear on the screen.
17. If you want to make changes to your text:
 - a. Make changes to the text, line breaks, or font

- b. Click Apply again.
18. Click Close on the auto layout text screen – return to drawing screen.
19. At the top right corner of the drawing panel (across from the word 'drawing'), there is a little box that is supposed to look like a push pin. Click the Pin to un-pin the drawing panel and close this panel.
20. Open the Tool Path screen by clicking the tool path tab that is sideways in the upper right corner of the screen.
21. Click Set to open material setup screen. No changes are required on this screen.
 - a. This screen will already be set with your thickness and XY datum (center).
 - b. Rapid Z gaps: leave at 0.2 clearance & plunge.
 - c. Home / start: leave at (0, 0, 0.2)(x, y, z).
 - d. Click OK to return to tool paths.
22. Under "tool path operations" select the V-carve icon. It has a big "V" on it – screen will open
23. Set start depth(D) at zero (0).
24. Set flat depth (F) to 0.2. This is how deep your letters will be. A greater depth will make larger letters.
25. Tool: select V-bit (60 degree, 0.5") unless you have a different bit.
 - a. You can also select a tool from the tool data base.
 - b. If the tool data base is open, click apply, then click OK.
26. At the bottom of the screen, the program will give you a default tool path name of V-Carve 1. Change the name to your initials and an identifier (e.g. **JG_001**)
27. Still on the V-Carve / Engraving Tool Path Screen: Make sure your vectors are selected (pink color) then click calculate at the bottom of the screen. The computer will calculate a tool path for the vectors selected (only one set in this case) then the computer will go to the preview tool path screen.
28. Make sure Tool path list and your tool path name are checked. Click preview selected tool path.
29. Click close.
30. You should be on the Tool Paths screen. Select (check mark in box) your tool path if not already selected
31. Click Save Tool Path icon. It is on the bottom row, third icon, it has a picture of a diskette – this will bring up the Save Tool Paths Screen. About 2/3 down the screen, click "Save Tool Paths to File"
32. A "save as" screen will pop up; your file name should already be entered. Click on Save.
33. Click on close to close the Save Tool Paths Screen.

34. In the upper right corner, minimize the whole program. This will take you out of Aspire.

Cutting Project with Mach 3 / Legacy

35. Make sure the Legacy CNC System (the “CNC”) is turned off. The gantry and the router position (X & Y) may be moved manually when the CNC is off. When the CNC is on, only move them with the keyboard.
36. Secure your material to the router table:
 - a. Move the router out of the way.
 - b. Make sure your material is flat on both sides and even in thickness.
 - c. Draw a light “X” on the face of your material or otherwise locate the center point.
 - d. Using four pieces of attachment hardware, secure your material “square” with the table.
 - e. Keep your attachment profile as low as possible. Keep attachment hardware clear of areas to be routed.
37. Put the router bit in the router; insert about 3/4th of the way; make sure the collet is snug.
38. The computer should be at the Microsoft desktop. Double-click on the Legacy Explorer Short Cut.
39. Session profile is Explorer: click OK.
40. Make sure the CNC is off.
41. Using both hands, pull the red gantry all the way to the front. Push the router all the way to the left.
42. Turn on the CNC.
43. On the screen (upper middle) click on Reset.
44. Click Yes to reference the machine. This tells the CNC that it is at the front left corner.
45. Using the arrow keys, page up, and page down, move the router so it is centered over your material (at the “X” that you made). You may have to tap or peck at the keys to get a response.
 - a. The tip of the router bit should just not be touching the material. Slide a piece of paper under the router tip to check the clearance.
46. Click on the X, Y, & Z buttons on the screen to set these parameters to zero. This will tell the machine that the router is at the X & Y datum point and on the surface of the material.
47. Click cycle stop to stop the processor so it can load new code. Ignore the error message
48. Click load code; select your tool path (xx_001.txt); click open.
49. Sometimes the program “stalls” at this point. If you get a pop up box that says “Please wait . . . generating path”. Do the following:

- a. Click “Cancel” in the box.
- b. Click Cycle Stop, wait two seconds.
- c. Click Unload Code.
- d. Click Reset.
- e. Go back to the previous step (#48).

50. You should see an image of the tool path in the upper right corner of the screen. If you do not see an image of your tool path, do not proceed.
51. Double check to make sure X, Y, & Z on the screen are set to zero.
52. Click Cycle Start – the apparatus will grumble a bit and then the lines of code will quit moving in the small upper left window. The program is giving you an opportunity to change the bit. You have already installed the proper bit.
53. Click Cycle Stop, **wait two seconds**.
54. Click Cycle Start, you will get a box telling you to turn on the router.
55. Turn on the router (switch on side of router) and click OK.
56. Watch in amazement.
 - a. The G-code will fly past in the upper left corner of the screen.
 - b. The X, Y, & Z coordinates will show in the middle of the screen. The Z coordinate is negative when it is cutting.
57. When the router is done, it will return to (0, 0, 0.2).

Shut Down, Store Tools, Cleanup

58. Turn off the router.
59. Turn off the CNC so you can use both hands to push the gantry back out of the way.
60. Remove your material. Remove your bit.
61. Close Legacy Explorer Program (the one that you are currently using). When prompted for Fixture Save?, Click NO.
62. Open the Aspire program from the menu bar at the bottom of the screen. Click on the X (upper right) to close the program. When prompted for Save changes to new?, click NO. When prompted for End Session, click YES. When prompted for Fixture Save, click NO.
63. Use the button in the lower left corner of screen to turn off computer. Use the button on front (lower right) of screen to turn off screen.
64. Return securing hardware to where you found it. Vacuum up your chips on the CNC and the table top.