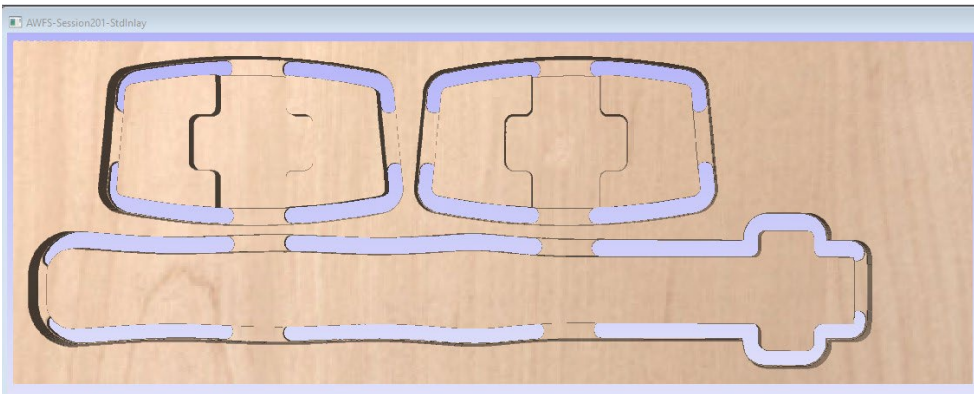


SWM CNC Teacher Training Session

Session 201/202: Making a wooden mallet customized with initials using the Vectric Inlay Toolpath (Standard Inlay)

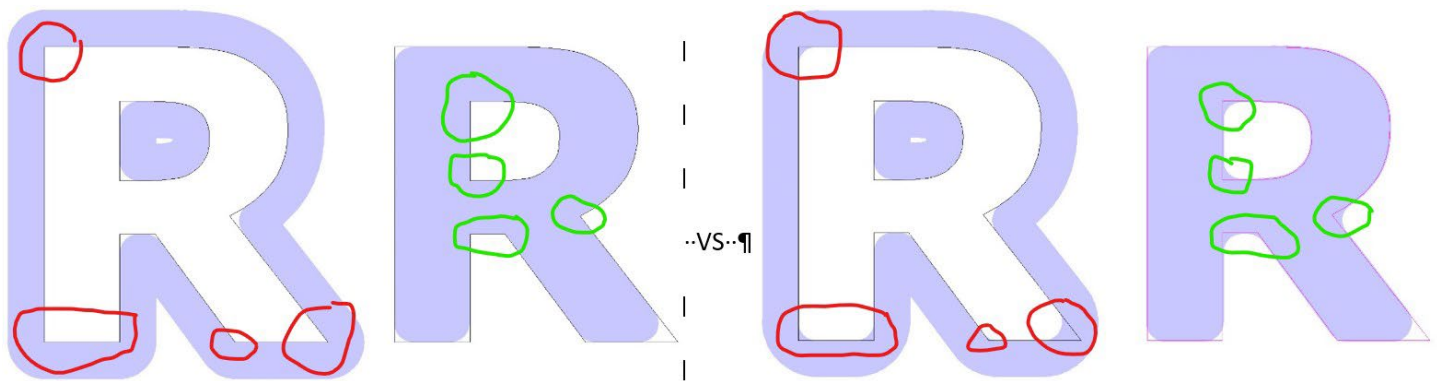
This session is about spicing up your projects by adding an inlay using the Inlay Toolpath function in Vectric software. To be able to complete this project during the time allotted for this training session, premade blanks were cut of the mallet head and handle in 3/4" Baltic Birch using the Pocket and Profile toolpaths. In addition, a piece of Cherry was milled to fit a pocket that was cut in the premade blanks. This Cherry will be used to make the male inlay pieces.

Once it is cutting time please pick up one of the 3/4" Birch blanks and one of the 2"x3.5"x0.25" pieces of Cherry. The Birch Ply blank should look like this.



The Vectric INLAY TOOL PATH is a very powerful and easy way to automatically modify your geometry so that the male inlay parts fit perfectly into the female pockets every time. Remember that unlike VCarve inlays, standard inlays are dependent on tool diameter. While the INLAY POCKET tool path looks very similar to a regular pocket toolpath and the INLAY INSERT toolpath looks very similar to the profile toolpath, there is one HUGE difference. Using the "Create Inlay Toolpath" function eliminates the need to modify your vectors to accommodate your tool diameter as this gets done automatically. In this session I'll explain this in greater detail.

Here is what happens when using the regular pocket and profile toolpaths to make a standard inlay vs. using the auto inlay toolpath functionality...



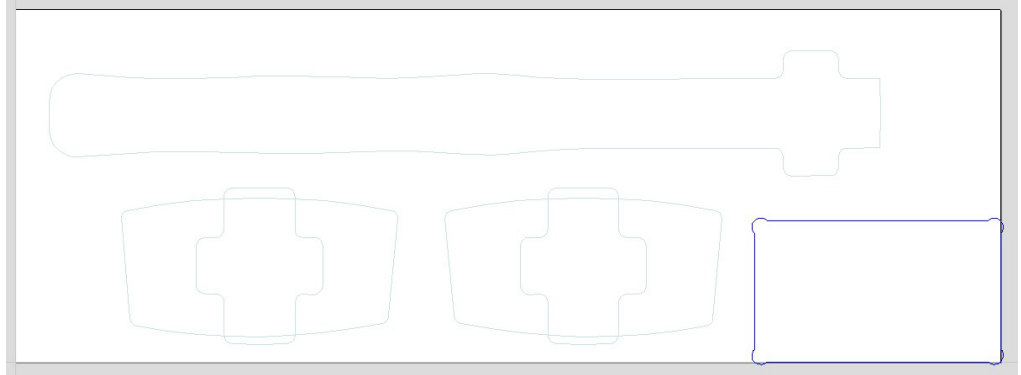
Standard Profile / Pocket

Inlay Toolpath Insert / Inlay Toolpath Pocket

Let's get started. Part one is designing your project. In this case we have started the project design for you.

1) Load the supplied file and verify the JOB SETUP parameters as:


- a. Double Sided
- b. 14W x 5H x 0.710 Thick
- c. Material Surface
- d. Lower Left Corner
- e. Flip bottom to top
- f. Std Resolution

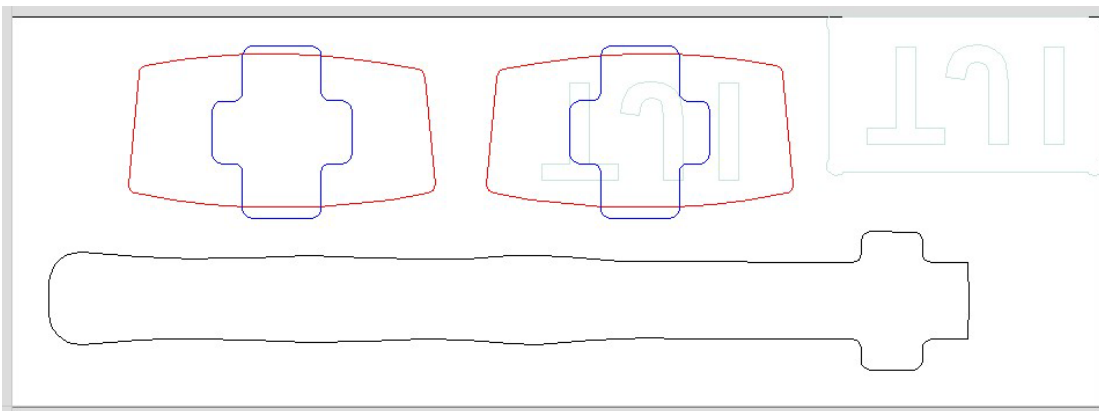


Once you have the file open you should see this image to the right.

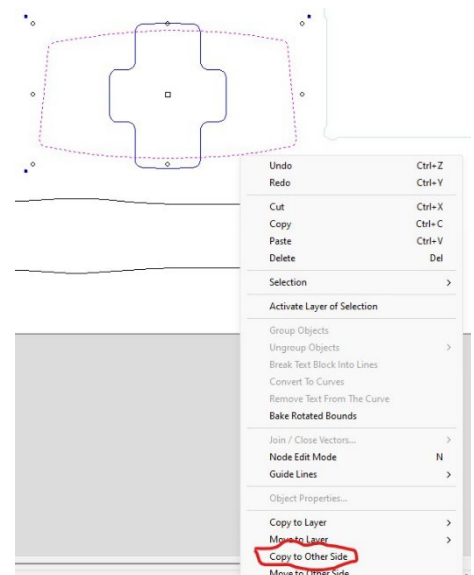
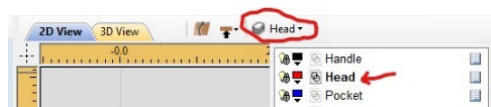
You are looking at the top side showing a rectangular vector that was used to create a pocket where the Cherry wood will be secured prior to cutting. We will be putting text vectors in this area to cut the male inlays. Also you can see light colored vectors from the other side of this project.

This is a 2 sided project so if you click on this icon  the screen changes to the bottom view

 showing the vectors that were precut for this session. Note top side vectors are light colored.



While viewing this side, one of the Mallet Head vectors needs to be copied to the other side. That layer is currently locked so first it needs to be unlocked.

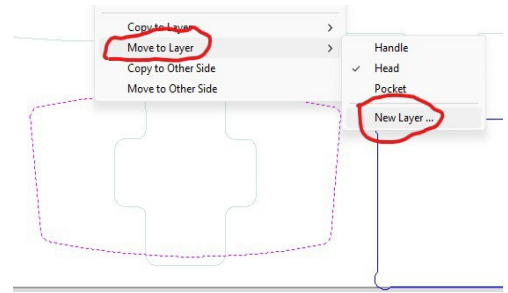
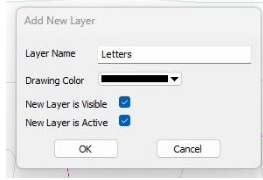


2) Go to Layers

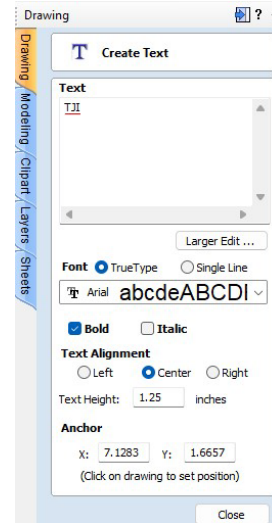
- a. Select Head layer
- b. Head layer is now active as show by the highlighted layer name and Head is shown at the top layer menu.
- c. Head Layer is locked as noted by the lock symbol over the lightbulb
 - i. Right click on head and choose unlock from the drop down menu
- d. While the Head layer is active and unlocked, select the right mallet head vector
- e. Right click on the right mallet head vector and select Copy to Other Side from the drop down menu (Shown on right).
- f. Change view to Top Side to see that the head vector transferred properly.

3) Create a Layer for the text (your initials) and add the head vector to that layer.


- a. Select the Head vector
- b. Right click and choose Move to Layer on drop down menu
- c. Choose New Layer and enter Letters
 - i. Check box New Layer is Visible
 - ii. Check box New Layer is Active
 - iii. Click OK
- d. Note Letters is currently the layer you are working on.
- e. Select the Head vector
- f. Right click, choose Move to Layer and pick Letters
- g. Relock the head layer the same as it was unlocked



- 4) Use "DRAW TEXT" and enter your 3 initials for the inlay
 - a. True Type Font
 - b. Select Ariel font
 - c. Select Bold
 - d. Select Center Alignment
 - e. Text Height 1.125" (some text combinations will allow 1.25")
 - f. Close the text box

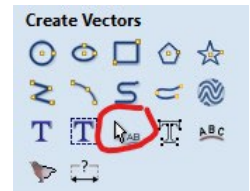


5) Center your text inside the mallet head vector.

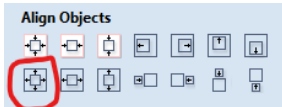
- a. Click on the text you just created
- b. Hold down the shift key
- c. Click on the Mallet head vector
- d. Select  to center your text inside the mallet head.

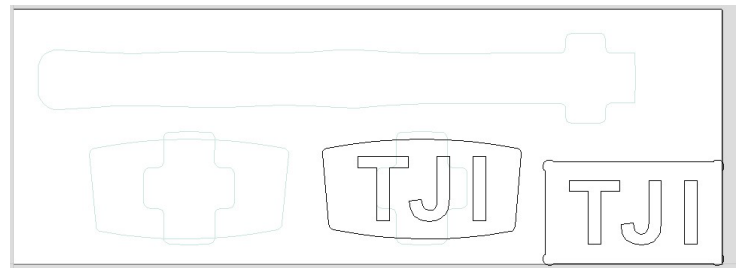
6) Adjust spacing as needed using the "Edit text spacing and curve" icon.

- a. Place cursor between letters and click to move them closer together.
- b. Place cursor between letters, hold down shift and click to move them further apart.



7) Create a copy of the text and align it the same way to the rectangle in the lower right hand corner.

- a. Click on the text you just created
- b. Hold down the control key and press "C"
- c. Hold down the control key and press "V"
- d. Click on the text
- e. Hold down the shift key
- f. Click on the lower right rectangle vector
- g. Select 
- h. Your file should look like this with your initials instead of TJI.

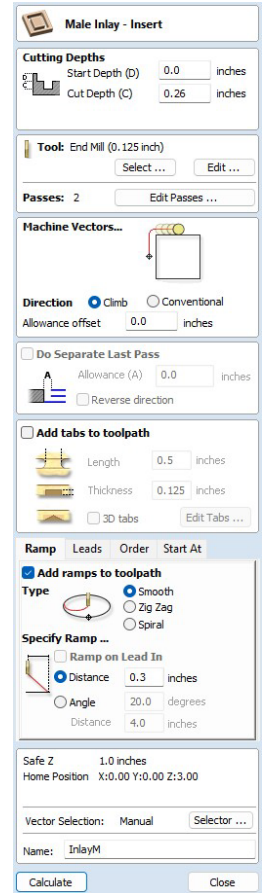
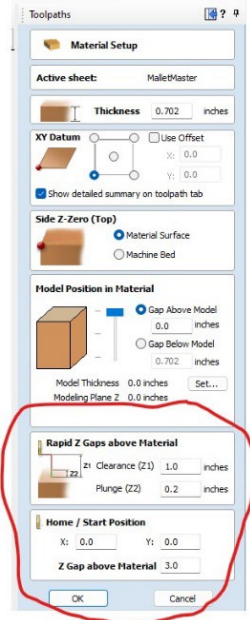
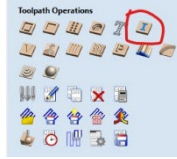


8) DONE, ready to make TOOL PATHS

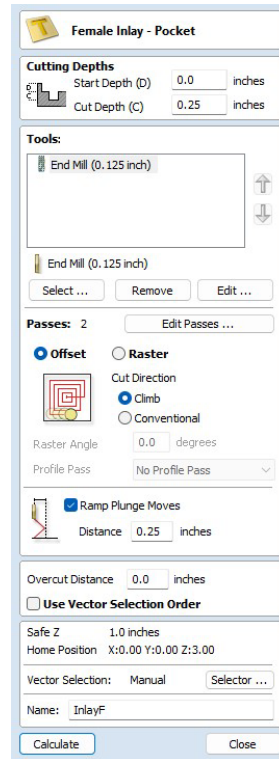
Part two is creating toolpaths for both the letters and the pockets.

VERY IMPORTANT you **MUST** use the exact same tool diameter for **BOTH** toolpaths.

- 1) Review Material Setup
 - a. Tool clearance to 1" or less if comfortable with less
- 2) INLAY Tool Path (always cut letters first)
 - a. Select text
 - b. Open Inlay Toolpath menu
 - c. Select Male Inlay (Straight)
 - d. Set cut depth to 0.26"
 - e. Select tool 0.125" End Mill
 - i. 50 inches/min
 - ii. Pass depth 0.125"
 - iii. Speed 16,000 RPM
 - f. Climb
 - g. Offset Allowance 0.01"
 - h. No tabs needed as CA glue will hold letters in place
 - i. Add Ramps to toolpath
 - i. Smooth
 - ii. Distance 0.25 (2XD min)
 - j. Change name to "InlayM"
 - k. Calculate
 - l. Preview toolpath to make sure everything looks good



- 3) INLAY Tool Path (cut pockets second, this allows running a revised toolpath if the letters don't fit before your remove your work piece)
 - a. Select text
 - b. Open Inlay Toolpath menu
 - c. Select Female Inlay (Pocket)
 - i. Set cut depth to 0.25"
 - ii. Select tool 0.125 End Mill
 1. 50 inches/min
 2. Pass depth 0.125"
 3. Speed 16,000 RPM
 - iii. Offset/Climb
 - iv. Ramp plunge Distance 0.25"
 - v. Change Name to "InlayF"
 - vi. Calculate
 - d. Preview toolpath to make sure everything looks good



- 4) Save above Tool Paths to USB drive
 - a. Select the correct post processor for your CNC machine from the drop down menu
 - b. Add tool description to file name if desired
 - c. Save GCode file
 - d. Save ENTIRE project file

5) DONE – ready to cut