Instructional Guide: Creating Finger Joints for CNC Machining

Author: Frank Currie , @you
Document Status: In progress
Last Updated: 26 Jan 2025

This guide will walk you through the process of designing and creating finger joints for your woodworking projects, specifically tailored for CNC machining. We'll use Inkscape for design, the "Quick Joint" extension to automate finger joint creation, and ESTLCAM to generate the G-code for your CNC machine.

Screencast/Recording:

To be added

Software You Will Need:

Software	Version	Download Link
Inkscape	1.4	https://inkscape.org/release/inkscape-1.4/
Quick Joint (Ext)	(Latest)	https://github.com/JarrettR/QuickJoint
ESTLCAM	11.245	https://www.estlcam.de/downloads/Estlcam_64_11245.exe
ESTLCAM Settings (Optional but I found helpful).	FluidNCv12	• Profile download - https://forum.v1e.com/uploads/short-url/2a0WOCQPui6K1UQ2XJ1Bjmcp4Xq.zip • Thread - https://forum.v1e.com/t/jackpot-lowrider-cnc-settings-config-for-estlcam-12-beta/41385 • NOTE: The settings file works for v11 • NOTE2: Download your ESTLCAM setting prior to applying the settings above

Example files:

File Type	Description	Download Link
Inkscape SVG	Example SVG file designed for finger joint box, created according to the guide's instructions. (Horizontal Orientation)	https://drive.google.com/file/d/1CiDSPi6AsGt4yxtVuX5Aoxrp1B3hguLd/view?usp=sharing (please request access)
ESTLCAM Project (.e10)	ESTLCAM project file corresponding to the example SVG. Includes rotated design, toolpaths, holding tabs, etc.	https://drive.google.com/file/d/1QFu-mrwi9-dNa2kbd_Fv9Lak7W8CHjOC/view?usp=sharing (please request access)

Part 1: Design in Inkscape

Step 1: Install the Quick Joint Extension

1. Download the Quick Joint extension from the link provided above.
2. Extract the contents of the downloaded ZIP file. You should see two files - a .inx file and a .py file
3. Open Inkscape.
4. Go to Edit > Preferences > System.
5. Find the "User extensions" path and open that folder location in your file explorer.
6. Copy the .inx and .py files from the Quick Joint extension into this extensions folder.
7. Restart Inkscape.

Step 2: Create Your Box Design

1. Open Inkscape and create a new document.
2. Use the rectangle tool (press R) to draw the individual sides of your box.
3. Set the dimensions of each rectangle to match the desired dimensions of your box. Ensure you create separate rectangles for each side.
    a. See "Flip Side" note below.

4. You may want to add a rectangle to test the sizes and fit (add in an extra to ensure you get it right the first time).
5. **CRITICAL** : Make sure to convert the rectangle using "**Object to Path**" or the BoxJoint plugin won't pick up anything.
6. Once you are happy with the size, it's time to add in the finger joints.

Step 3: Numbering Sides for Finger Joint Placement

It's crucial to understand how Inkscape and the Quick Joint extension number the sides of your rectangles. Here's the convention:

- **Top:** Side 0
- **Right:** Side 1
- **Bottom:** Side 2
- **Left:** Side 3

Important: We will start by adding finger joints to the **left** side (Side 3) and then move to the **right** side (Side 1). This order is recommended because as you add more finger joints, each segment of the joint is treated as a new side, making it difficult to keep track of the correct side numbers if you don't follow this sequence.

Step 4: Adding Finger Joints

1. Select the first rectangle
2. Go to Extensions > Modify Path > Quick Joint.
3. The Quick Joint dialog box will appear. Here are the settings:
    - **Side 1:** The side number of the first rectangle you selected (in our example, 3 for the left side).
    - **Side 2:** The side number of the second rectangle (e.g., 0 for top or 2 for bottom).
    - **Kerf:** Set this to 0 as we are working with a CNC machine, not a laser cutter.
    - **Material Thickness:** Enter the measured thickness of your material. If you measured 23.4mm, enter 23.5 (adding a 0.1mm fudge factor, which is approximately a 0.43% increase). The percentage calculation is (0.1/23.4) * 100. It's a good idea to test your joints with scrap material.
    - **Number of tabs:** Choose the number of tabs/fingers you want for the joint.
    - **Tab Width:** If you leave it blank it will automatically distribute the tabs across the distance.
    - **Flip side:** This option moves the tabs to the inside or outside of the rectangle. We will use the "outside" for this guide. To set this to outside, leave it unchecked. It's important to note that it will flip the joint on Side 2.
4. Click Apply.

5. The finger joints will be created.
6. Repeat this process for the right side (Side 1) of the first rectangle, joining it to the corresponding top or bottom of the next rectangle.

NOTE: "Flip Side" for Internal/External Dimensions

- By default (unchecked), the "Flip side" option places the tabs on the *outside* of the rectangle lines. This is suitable if the *internal* dimensions of your box are critical.
- If the *external* dimensions are more important, check the "Flip side" box to move the tabs to the *inside* of the rectangle lines.

Step 6: Finalize and Save

1. Once you've added all the finger joints, carefully review your design.
2. Create duplicates of each unique side. For a simple box, you'll likely need two of each side. Use Edit > Copy (Ctrl+C) and Edit > Paste (Ctrl+V) to create the copies.
3. Arrange the pieces efficiently to minimize material waste.
4. Save your design as an SVG file. Go to File > Save As and choose "Inkscape SVG" as the file type.

Part 2: ESTLCAM for G-Code Generation

Step 1: Import the SVG

1. Open ESTLCAM.
2. Go to File > Open and select the SVG file you saved from Inkscape.

Step 2: Rotate the Design for LowRider Orientation

1. Press Ctrl+A to select all objects in your design.
2. Go to Automatic functions > Rotate.
3. In the Rotate dialog, enter -270 degrees.
4. Ensure the "Rotate around object center" option is **not** selected. Click "OK".
5. Visually confirm that your design is now oriented vertically, as required for your LowRider CNC.

Step 3: Define the Machining Operations

1. Select all the objects in your design by pressing Ctrl+A or dragging a selection box around them.

2. In the left-hand toolbar, click the "Path" button (it looks like a squiggly line).
3. In the "Path" settings dialog:
    - **Function:** Choose "Outside" or "Inside" depending on whether you want to cut outside or inside of the lines. Since we want to cut out the shapes and our finger joints are on the outside, select "Outside".
    - **Tool:** Select the end mill you will be using (e.g., 3.18mm or 6.35mm).
    - **Depth per pass:** Set an appropriate depth of cut for your material and end mill.
    - **Feed rate:** Set the appropriate feed rate for your material and end mill. You can get a guide from your end mill supplier or the CNC manufacturer.

Step 4: Automatic Corner Overcuts

1. ESTLCAM can automatically add overcuts to inside corners to ensure the finger joints fit correctly. The size of the overcut depends on the radius of the bit.
2. After creating the "Path" operation, go to Automatic functions > Create corner overcuts.

Step 5: Holding Tabs

1. To prevent your parts from moving during the cutting process, you need to add holding tabs.
2. Click the "Holding tabs" button in the left-hand toolbar.
3. Click on the outlines of your parts where you want to place the tabs.
4. Adjust the "Tab width" and "Tab height" in the settings as needed.
5. Note: When you change from automatic to manual, make sure you click "OK" otherwise the holding tabs will disappear.

Step 6: Reset the zero point.

1. In the bottom left corner of your workpiece, click the "Zero point" button in the left hand toolbar to reset the X and Y zero.

Step 7: Generate G-Code and Set Material Thickness

1. Go to File > Save CNC program.
2. Enter the thickness of your material.
3. Choose a file name and location to save your G-code file.

Part 3: Machining with a V1 Engineering Jackpot Controller and FluidNC

Now that you have your G-code generated, it's time to cut out your project using your CNC machine equipped with a V1 Engineering Jackpot controller running FluidNC firmware.

Step 1: Load the G-Code File

1. **Homing (Optional but Recommended):** If your machine has homing switches, it's a good idea to home the machine first. Click the "Homing" button (usually represented by a house icon) in the FluidNC interface. This will move the machine to its home position, establishing a known reference point. You will need to set up your endstops to enable homing.
2. **Upload:** In the FluidNC web interface, go to the "Files" section.
3. **Select File:** Click the "Upload" button and select the G-code file you generated in ESTLCAM.
4. **Verify:** Once uploaded, the file should appear in the file list. You can click on it to preview the toolpath in the visualizer.

Step 2: Run the Job

1. **Spindle On:** Turn on your spindle. In FluidNC, you may need to manually send the spindle start command (e.g., M3 S10000 for 10,000 RPM, adjust the speed as needed) or configure a spindle control button.
2. **Start:** Click the "Start" or "Play" button (usually represented by a green triangle) in the FluidNC interface to begin cutting.
3. **Monitor:** Keep a close eye on the machine as it cuts, especially during the first run. Be prepared to stop the machine if anything goes wrong.

Step 3: Finishing Up

1. **Job Completion:** Once the job is complete, the machine will return to its home position (if homing was used) or remain at the end of the cutting path.
2. **Spindle Off:** Turn off your spindle.
3. **Power Down (Optional):** You can power down the Jackpot controller if you're finished.
4. **Remove Workpiece:** Carefully unclamp your workpiece from the machine.
5. **Clean Up:** Use a vacuum or brush to clean up any dust or chips from the machine and surrounding area.
6. **Remove Tabs:** Use a chisel, utility knife, or oscillating multi-tool to carefully remove the holding tabs that were connecting your parts to the main material.