

Pizza Peel File Instructions



Hello CNC'er,

Congrats on getting this pizza peel file. Now you can make a pizza peel without a much design work.

These instructions will tell you how to set up your CN router to make the peel with the provided g-code.

First...To get unannounced discounts, be sure to join the [CNC Insiders](#) email list. You will get discounts, freebies and special deals that ***are not announced*** on the [IDC Woodcraft YouTube channel](#).

To learn how to build a CNC business, join the [CNC Entrepreneurs Facebook](#) group.



Pizza Peel File Instructions

To watch the full step-by-step video to design & run this peel, [click here now](#).

This file package is provided to you as a gift to help you on your CNC journey. The provided g-code has been tested multiple times (I have a few pizza peels laying around...lol). On that note, this gift set is final. No modifications will be made to it by IDC Woodcraft.

If you have issues getting things running properly, it is likely caused by one of these issues:

- You have not set up your zero point(s) properly.
- Your machine does not recognize some codes in the provided gcode file.
 - If this happens, it will show up as an alarm in your control software. You will need to regenerate the g-code using your post processor
- Your machine controller does not recognize the file extension (txt, nc, gcode, etc)
 - The gcode files provided here are *gcode* extensions
 - If your system uses a different extension, change the extension to the appropriate extension and your controller will see it. You can [watch this video](#) to show you how to change the extension.



Pizza Peel File Instructions

PLEASE FOLLOW ALL INSTRUCTIONS IF YOU USE THE PROVIDED G-CODE

NOTE: It is your responsibility to verify these files work properly on your machine.

To confirm, be sure to dry-run each g-code file first without a bit in the machine!

You received several files:

- These instructions
- The Vectric (crv) design file for version 11 (earlier versions of Vectric may not open this file)
- SVG and DXF file formats (these should load into any design software)
 - To learn how to import these file types into Vectric, [watch this video](#)
- 4 G-code files titled
 - Pizza Peel_1-Surfacing 1su
 - Pizza Peel_2-Taper 150v
 - Pizza Peel_3-Round over 14R
 - Pizza Peel_4-5-Hole 14cmp

***Run the files in the order listed above. You do not have to run the surfacing program. If you do not, you may have slight steps in the project around the cut areas due to variations in material thickness.

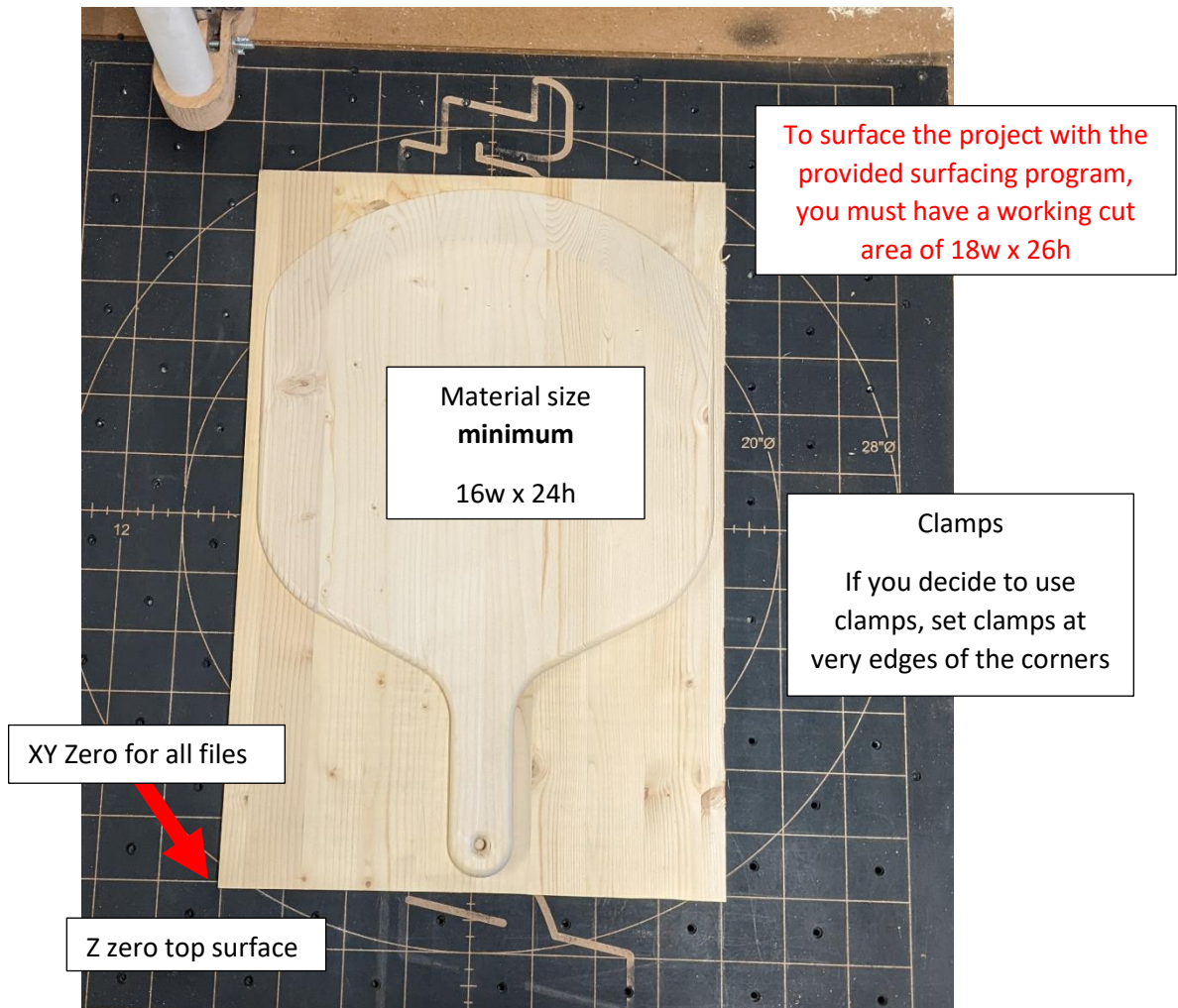
Bits required to run this gcode file:

- [IDC Woodcraft 1" surfacing bit](#)
- [IDC Woodcraft 150° x 2" V-bit](#)
- [IDC Woodcraft ¼" radius rounding bit](#)
- [IDC Woodcraft ¼" compression bit](#)

NOTE: Feeds & speeds are preset for these bits in the gcode file.

Pizza Peel File Instructions

Set Up Of Pizza Peel On Your CNC Router



This file is designed to use a minimum 16" x 24" x $\frac{3}{4}$ " material blank. If your material is significantly thinner than $\frac{3}{4}$ ", you may want to regenerate the toolpath after adjusting the thickness in the design file.

Pizza Peel File Instructions

Work Holding Of Pizza Peel On Your CNC Router

For work holding, I suggest using the CA glue hold down method. For a 10% discount on Starbond CA glue and accelerator, use the coupon code **IDCWOODCRAFT**.

Order your CA glue [using this link](#).



You will need:

- Medium clear CA glue
- Accelerator