

JointCAM



Reference Guide

Version 1.03

Copyright © G-Force CNC, LLC, 2014. All Rights Reserved.

Disclaimer

All CNC machines are potentially dangerous. Because G-Force CNC, LLC cannot control how the software described in this manual will be used, G-Force CNC, LLC can not be responsible for any loss or damage to workpiece, machine or individual caused by misuse of the software or CNC machine.

The output from the software should be thoroughly checked before using it with a CNC machine.

The information in this manual may be subject to change without notice. The software described in this manual is supplied under the terms and conditions of the End User License Agreement and may only be used in accordance with this agreement.

Table of Contents

Introduction.....	7
Requirements.....	8
CNC Machine Requirements.....	8
PC Requirements.....	9
Supported Operating Systems:.....	9
Installation.....	10
Updating.....	10
Configuration.....	11
G-Code Options.....	11
G-Code or Shopbot .sbp.....	11
Separate Rough/Finish Code– G-Code Only.....	11
File Extension.....	11
Decimal Places.....	11
Use Line Numbers– G-Code Only.....	11
Inch/mm Radio Buttons.....	11
Use Mach3 G91.1– G-Code Only.....	11
Include Tool and Material Comments.....	11
Startup Codes 1-3.....	11
Shopbot Start/Stop Spindle Yes/No.....	12
Initial Tool Change/ Spindle Start – G-Code Only.....	12
G43 Length Offset Yes/No– G-Code Only.....	12
Tool Change– G-Code Only.....	12
M7 Yes/No– G-Code Only.....	12
M8 Yes/No– G-Code Only.....	12
Ending Codes 1-3.....	12
M2 Yes/No (Shopbot END).....	13
M30 Yes/No – G-Code Only.....	13
Final End Code – G-Code Only.....	13
Go to Park Position at End of Cycle.....	13
Include File Name in G-Code File.....	13
General Settings.....	13
Machine Setup.....	13
Clearance Plane.....	13
First Rapid Height.....	13
Machine Width.....	13
Dovetail/Box Joint Axis.....	13
Machine Width = Stock Width.....	14
File Paths.....	14
Specify and Use Default G-Code folder.....	14
Specify and Use Default Project Folder.....	14
Show/Hide Tooltips.....	14
Plunge Rate Percentage.....	15
Scale Feedrate Yes/No.....	15
Scale Factor.....	15
Disable Rabbeted Dovetail Warning Messages.....	15
Dovetails.....	16
The Half Blind Interface(s).....	17
Dovetail Parameters.....	17
Width of Stock.....	18
Thickness of Stock.....	18

Depth of Cut.....	18
Start Distance.....	18
Dovetail Type.....	18
One Pass or Multi-Pass.....	18
Tail Width.....	18
Pin Width.....	18
Pins = Tails.....	18
Full Tails Only.....	19
# of Tails.....	19
Pin/Tail Offset =.....	19
Scoring Pass.....	19
Scoring Depth.....	20
Clearance.....	20
Use Roughing Tool.....	20
Roughing Allowance.....	20
Roughing Depth Allowance.....	21
Roughing Stepover %.....	21
Tail Socket Overcut.....	21
Calculate / Preview.....	22
Tail/Pin Data (Half Blind Custom only).....	22
#1 Tail Width through #11 Tail Width.....	22
#1 Pin Width through #10 Pin Width.....	22
Use Minimum Pin Width.....	22
Symmetrical.....	22
Center Last Tail.....	22
Center Pin Width (read only).....	22
Clear All.....	22
Tool Parameters.....	24
Tool #.....	24
Diameter.....	24
Length.....	24
Angle (Dovetail tools only).....	24
Depth/Pass (straight tools only).....	24
Feedrate.....	24
RPM.....	24
Select Tool Button.....	24
Machining Options.....	25
Pin & Tail Boards.....	25
Pin Boards Only.....	25
Tail Boards Only.....	25
Left & Right Joints.....	25
Left Joints Only.....	25
Right Joints Only.....	25
Load Project.....	25
Save Project.....	26
Write G-Code File.....	26
Save Settings.....	26
Exit.....	26
Creating a half blind dovetail joint.....	27
Stock and Machine Setup.....	27
Pins and Tails.....	28
Tail Boards Only.....	29

Pin Boards Only.....	30
The Through Dovetail Interface(s).....	31
Dovetail Parameters.....	31
Width of Stock.....	31
Thickness of Stock.....	31
Tail Height.....	31
Tail Width.....	31
Pin Width.....	31
Pin/Tail Extension.....	31
Pins /Tails = Tool.....	32
Pins = Tails.....	32
Start Distance.....	32
Configuration (Pin First or Tail first).....	32
Roughing Options.....	32
Clearance.....	32
Calculate / Preview.....	32
Tail/Pin Data (Through Custom only).....	33
#1 Tail Width through #11 Tail Width.....	33
#1 Pin Width through #10 Pin Width.....	33
Use Minimum Pin Width.....	33
Symmetrical.....	33
Center Last Tail.....	33
Center Pin Width (read only).....	33
Clear All.....	33
Tool Parameters.....	36
Machining Options.....	36
Pin Boards Only.....	36
Tail Boards Only.....	36
Left & Right Joints.....	36
Left Joints Only.....	36
Right Joints Only.....	36
Multiple Tails.....	36
Load Project.....	37
Save Project.....	37
Write G-Code File.....	37
Save Settings.....	37
Exit.....	37
Creating a through dovetail joint.....	38
Pin Boards.....	38
Tail Boards.....	39
Box Joints.....	40
The Box Joint Interface.....	41
Box Joint Parameters.....	41
Width of Stock.....	41
Thickness of Stock.....	41
Finger #1 Width & Finger #2 Width.....	41
Finger Extension.....	42
Start Distance.....	42
Layout Type.....	42
Finger Width Data.....	42
Finger and Socket width entry boxes.....	42
Sockets = Fingers.....	43

Sockets = Tool Diameter.....	43
Symmetrical.....	43
Center Finger/Socket Width.....	43
Tool Parameters.....	43
Tool #.....	43
Diameter.....	43
Length.....	43
Depth/Pass (straight tools only).....	43
Feedrate.....	43
RPM.....	43
Select Tool Button.....	44
Machining Options.....	44
Finger Board “A”.....	44
Finger Board “B”.....	44
“A” and “B” Boards.....	44
Multiple Boards.....	44
Left & Right Joints.....	44
Left Joints Only.....	44
Right Joints Only.....	44
Load Project.....	45
Save Project.....	45
Write G-Code File.....	45
Save Settings.....	45
Exit.....	45
The JointCAM Tool Table.....	46
Adding Tools to the tool table.....	46
Editing Tools in the tool table.....	46
Deleting Tools from the tool table.....	46
Selecting a tool for use.....	47
Exit.....	47

Introduction

JointCAM is a program for the layout of woodworking joints and for producing NC code that allows CNC machines to be able to cut those joints. In the past, cutting woodworking joints on a CNC router required time consuming and often very complex CAD and CAM work to design and tool path them. JointCAM can do in minutes what would take hours to do with traditional CAD/CAM software. JointCAM offers considerable flexibility, to allow the user to create joints for a wide variety of projects.

Requirements

CNC Machine Requirements

Most woodworking joints require the workpiece to be clamped in a vertical orientation, in order to machine the end. JointCAM is no exception. The user's CNC machine must have a jig or clamping fixture to mount a vertical workpieces. A vertical fence is ideal to quickly position the workpiece for clamping. The spindle needs to be able to reach at least 1x the tool diameter past the vertically mounted workpiece.

JointCAM provides the ability to cut pairs of joints, so a fixture with two vertical fences to cut pairs of joints together is preferred. A horizontal fence for each vertical fence should also be incorporated into the jig/fixture, and should be aligned with the vertical fence(s). JointCAM uses the edge of the fence as either X or Y zero, depending on the orientation of the user's machine. See the individual joint sections of this manual for more information on program origin locations.

Some tool paths created by JointCAM will start behind the vertical fence. To prevent cutting into the horizontal fence, it should be located at least 2x the diameter of the largest used tool behind the vertical fence.

Illustration 1 depicts a typical gantry style router, with fences mounted for use with JointCAM.

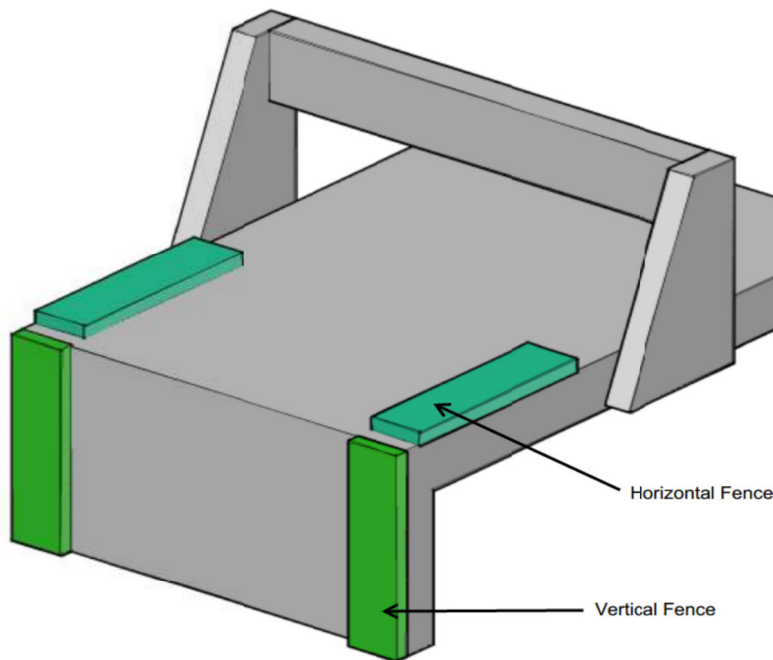


Illustration 1: Typical fence locations

Dovetails and other woodworking joints require a very high degree of precision in order to achieve good results. High precision is dependent on a few different factors.

- 1) Accuracy of stock. It's important that the stock (workpiece) used is accurately sized, with the ends cut square, and flat. If the stock is not accurate, the fit of the joints will be adversely affected.
- 2) Z axis zero location. Z zero is set to the top surface of the stock. The fit of the dovetail joint is controlled by the depth of cut. Deeper cuts result in a tighter fit, while shallower cuts will result in a looser fit. Because of this, the Z zero position is critical in achieving consistent, tight fitting dovetail joints. An Auto Zero Setting device is highly recommended when using JointCAM.
- 3) Machine rigidity. Cutting dovetails can create high forces on the tool and machine, often causing deflection, resulting in poor fitting joints. The reason is that dovetails must be cut at full depth, and the full width of the bit.

Many DIY and small hobby machines may run into rigidity issues when cutting dovetails. With very hard woods, and/or large dovetail bits, the issue may be even worse. On less rigid machines, JointCAM has the ability to remove the majority of material using multiple roughing passes with a straight tool, prior to cutting the actual dovetails. This can greatly reduce the cutting forces, and give much better results if rigidity is an issue on your machine. Using roughing passes will, however, result in much longer cut times.

Using slower feed rates can also help with rigidity issues. But again, longer cut times will result, along with possibly shorter tool life.

PC Requirements

JointCAM has very minimal PC requirements. The following are the minimum supported system requirements:

1.5Ghz or faster Intel or AMD processor
512Mb RAM
1Gb free storage space

Supported Operating Systems:

Note: JointCAM currently does not support Windows font size settings other than “Normal” or 96dpi. If larger font sizes are used, slipping or overlapping of text labels may occur.

JointCAM is a 32 bit application, and will run on the following operating systems:

Windows XP SP3 with .NET 4.0
Windows 7 with .Net 4.0 (x86 and x64)
Windows 8 (x86 and x64)
Windows 8.1 (x86 and x64)

.NET 4.0 can be installed from <http://www.microsoft.com/en-us/download/details.aspx?id=17851>

Installation

Double Click (or Right Click and choose “Install”) the JointCAM_Setup.exe file to begin the installation.

During the installation, you will be asked to choose either inch or metric default settings. This will install default settings files with the appropriate units.

JointCAM is installed in the appropriate “Program Files” folder for the version of Windows being used. Settings files are stored in the application data folder for the version of Windows being used. The “Settings Files” button on the Configuration tab will open the location in Explorer.

Updating

During the installation of updates, the saved user settings will not be overwritten unless the update contains a change to the format of the settings files.

Configuration

The Configuration tab contains settings for general program operation and configuring the g-code output.

G-Code Options

G-Code or Shopbot .sbp

Select “G-Code” for standard g-code output (Mach3, LinuxCNC, WinCNC) or “Shopbot .sbp” to output native Shopbot format code.

Separate Rough/Finish Code– G-Code Only

When “Use Roughing Tool” is selected for the current joint type, select this option to export roughing toolpath code only. When not checked, both roughing and finishing toolpaths will be in a single file, with a toolchange command between them. To export finish toolpaths only, disable roughing passes in the settings for the individual joint type.

(This option is not available when exporting .sbp files for Shopbots)

File Extension

Select preferred g-code file extension from drop down list, or type in additional extension.

Decimal Places

Number of decimal places to be used for coordinates in g-code file. Typical values are 4 for imperial (inches), and 2 for metric (mm).

Use Line Numbers– G-Code Only

Check box to enable line numbers in g-code file. Select line number increment from drop down list, or enter user defined increment. (Integer values only, not available when exporting .sbp files for Shopbots)

Inch/mm Radio Buttons

Select Inch (G20) for inch output or mm(G21) for metric output.

Note: This option adds a G20 or G21 to the g-code. It DOES NOT convert from inches to mm's or mm's to inches. All units used in JointCAM should match the setting chosen here.

Use Mach3 G91.1– G-Code Only

Selecting this option will add a G91.1 to the g-code, forcing Incremental IJ mode in Mach3/Mach4. This will ensure that Mach3 will follow the arcs in the toolpaths correctly.

Include Tool and Material Comments

Select this option to have tool and workpiece information added to the g-code as comments.

Startup Codes 1-3

Up to three lines of user defined g-code are available. Lines will be added prior to the toolpath code, before the spindle and coolant commands. Text will be added to the g-code file exactly as entered in the three input boxes. If adding comments, add the appropriate comment characters for your specific control.

Shopbot Start/Stop Spindle Yes/No

When Shopbot output is selected, SO 1, 1 followed by a Pause 3 will be added for spindle start, and SO 1, 0 will be added for spindle stop.

Initial Tool Change/ Spindle Start – G-Code Only

This section offers up to seven (7) user defined lines of g-code to execute a tool change, start the spindle, specify spindle speed, add comments, or any other use prior to any machine movement.

All entered text will be placed in the g-code file exactly as typed, with two special exceptions explained below.

If adding comments, add the appropriate comment characters for your specific control.

For Tool Change commands, tool number and spindle speed can be passed to the g-code by entering [T] for tool number, and [S] for spindle speed. Entering the following two lines of code:

```
T[T] M6  
S[S] M3
```

will result in the following two lines of g-code, if the current tool is tool number 3, with an rpm of 12500:

```
T3 M6  
S12500 M3
```

Any input lines left blank will be ignored.

G43 Length Offset Yes/No– G-Code Only

Select this option to include tool length offset command in G-code. Format will be G43 Hxx where xx is the tool number. G49 will be added to turn off length offset at the end of the file.

Tool Change– G-Code Only

This section offers up to seven (7) user defined lines of g-code to execute when a tool change is required. These lines are inserted in the g-code when a change of tools is required. Be sure to add the appropriate command to turn off the spindle prior to any tool change commands, if required by the control. (See “Initial Tool Change/ Spindle Start” section above for more information)

M7 Yes/No– G-Code Only

Select this option to add an M7 (mist coolant on) to the g-code. An M9 will be added to the end of the file to turn off the mist coolant.

M8 Yes/No– G-Code Only

Select this option to add an M8 (flood coolant on) to the g-code. An M9 will be added to the end of the file to turn off the flood coolant.

Ending Codes 1-3

Up to three lines of user defined g-code are available. Lines will be added after the toolpaths and coolant are turned off. Text will be added to the g-code file exactly as entered in the three input boxes.

If starting the spindle in the “Toolchange” sections, use one of these lines to turn it off. If adding comments, add the appropriate comment characters for your specific control.

M2 Yes/No (Shopbot END)

Selecting this option will add an M2 (file end) command to g-code or and END command to shopbot files.

M30 Yes/No – G-Code Only

Selecting this option will add an M30 (file end and rewind) to the end of the g-code file. Not available with Shopbot output)

Final End Code – G-Code Only

Add a user defined end code at the end of the file. A “%” is typically used as with most machine controls, but may or may not be required.

Go to Park Position at End of Cycle

Selecting this option will send the machine (at rapid rate) to a user defined park position. The Z axis will move first, followed by a coordinated XY move.

Include File Name in G-Code File

Selecting this option will add the full file name (with path) at the beginning of the g-code file as a comment.
Note: This line will not have a line number, even if line numbers are enabled.

General Settings

Machine Setup

Clearance Plane

Z height of rapid moves above workpiece during joint cutting.

Note: Some rapid moves are at cut level, retracing previously cut areas to reduce cycle times.

First Rapid Height

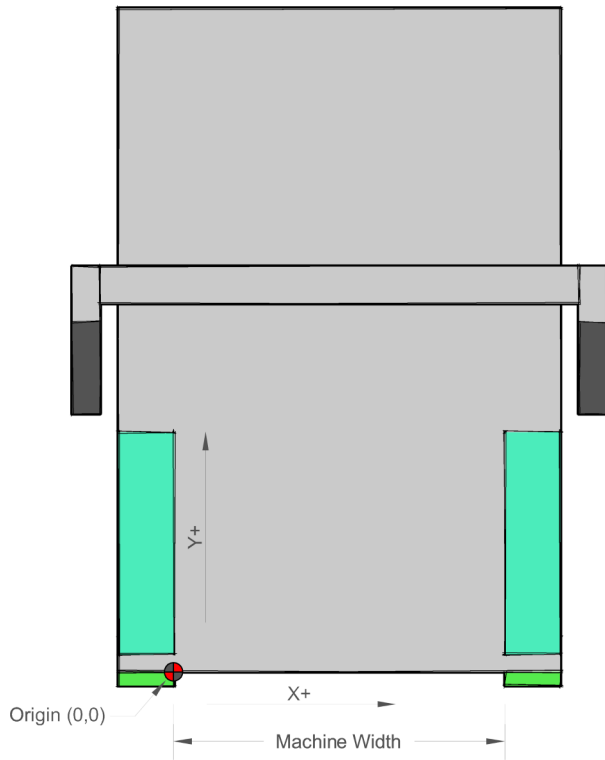
Z height of first move prior to cutting. Set to a value higher than any clamps or other objects between the starting position and the workpiece(s). “First Rapid Height” is also used after all tool changes, to protect against crashing into any clamps or holding devices after a tool change.

Machine Width

Width of machinable are between left and right vertical fences when cutting dovetails and box joints.
See Illustrations 2 and 3.

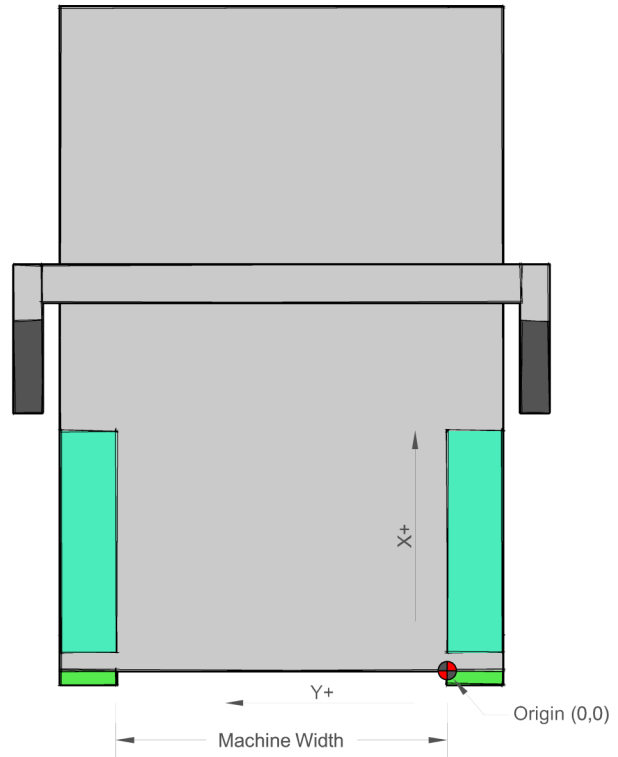
Dovetail/Box Joint Axis

Select axis along which the workpiece is mounted for dovetails and/or box joints.
See Illustrations 2 and 3.



X Axis Dovetails/Box Joints
(Top View)

Illustration 2: Dovetail/Box Joint Axis set to X axis



Y Axis Dovetails/Box Joints
(Top View)

Illustration 3: Dovetail/Box Joint Axis set to Y axis

Machine Width = Stock Width

Selecting this option allows cutting both left and right joints using a single fence located at the origin. This is useful for machines that are not wide enough for two vertical boards, or where only one fence is available. When this option is selected, pins and tails must be cut separately when cutting half blind dovetails.

File Paths

Specify and Use Default G-Code folder

Select this option to select a default output folder for g-code files. Use the Browse button to select the folder. If not selected, the file location should default to the last folder used (Windows default)

Specify and Use Default Project Folder

Select this option to select a default output folder for project files. Use the Browse button to select the folder. If not selected, the file location should default to the last folder used (Windows default)

Show/Hide Tooltips

Select this option to show tooltips in JointCAM. JointCAM must be closed and restarted for this option to take effect

Plunge Rate Percentage

Feedrate for plunge moves. Enter a percentage of the full feedrate. This applies to straight tools only. JointCAM does not plunge into the workpiece itself. For dovetail and box joints, all plunge moves outside of the workpiece are done at rapid rate. Plunge moves behind the workpiece (over the machine table) are done at the plunge rate. The user will often want to use a backer board to prevent tearout on the backside of the workpiece. The only places the tool will actually be plunging into material would be into a backer board.

Plunge rate will also be used as a ramp feedrate in future releases of JointCAM, mainly when cutting mortises.

Scale Feedrate Yes/No

This option allows you to scale the feedrate in the exported g-code. Primarily for ShopBot users, this allows you to specify the feedrate in JointCAM in units/minute, and have the feedrate written as units/second in the g-code or .sbp code.

To use, check the box, and set an appropriate scale factor.

Scale Factor

Feedrate scale factor. For shopbot users wanting to specify feedrate in units/minute, and export using units/second, use 0.1667 (1/60).

Disable Rabbeted Dovetail Warning Messages

When cutting through dovetails, the length of the tails must be at least as long as the stock thickness. JointCAM can optionally cut a rabbet in the pin board, resulting in tails shorter than the stock thickness. If the user specifies a tail length less than the stock thickness, JointCAM will display a warning message. This gives the user 3 options:

- 1) Cut a rabbeted joint with the current settings
- 2) Increase the tail height for a full joint (If the tool is long enough)
- 3) Cancel the calculation to allow the settings to be changed

This warning message is displayed whenever the Calculate/Preview button is pressed, as well as when the g-code is written.

Selecting this option disables the warning message, and automatically defaults to a rabbeted joint.

Dovetails

Dovetails are one of the most common types of woodworking joints. They are commonly found on all types of furniture and cabinetry, from mass produced drawers and other types of boxes, to carcass construction on fine furniture. JointCAM offers four different options for creating dovetails:

- 1) Half Blind EQ – Half Blind dovetail joints where pins and tails are equally spaced, similar to dovetails cut with a standard dovetail jig. With equal sized pins and tails, both pin and tail boards can be cut simultaneously, which makes for a very fast joint to machine.
- 2) Half Blind Custom – Half Blind dovetail joints where pins and tails can be different sizes, with different spacing between them. A virtually unlimited number of joint styles can be created by specifying the sizes of each pin and tail in the joint.
- 3) Through EQ - Through dovetail joints where both pins and tails are equally spaced. Most hand cut dovetails are through dovetails. Through dovetails are visible on both faces of the joint.
- 4) Through Custom – Through dovetail joints where pins and tails can be different sizes, with different spacing between them. A virtually unlimited number of joint styles can be created by specifying the sizes of each pin and tail in the joint.

For a typical drawer box, the pin boards are the front and back of the drawer, with the tail boards being the drawer box sides.

Illustration 4 below shows a half blind dovetail box with the locations of left and right hand joints as JointCAM refers to them.

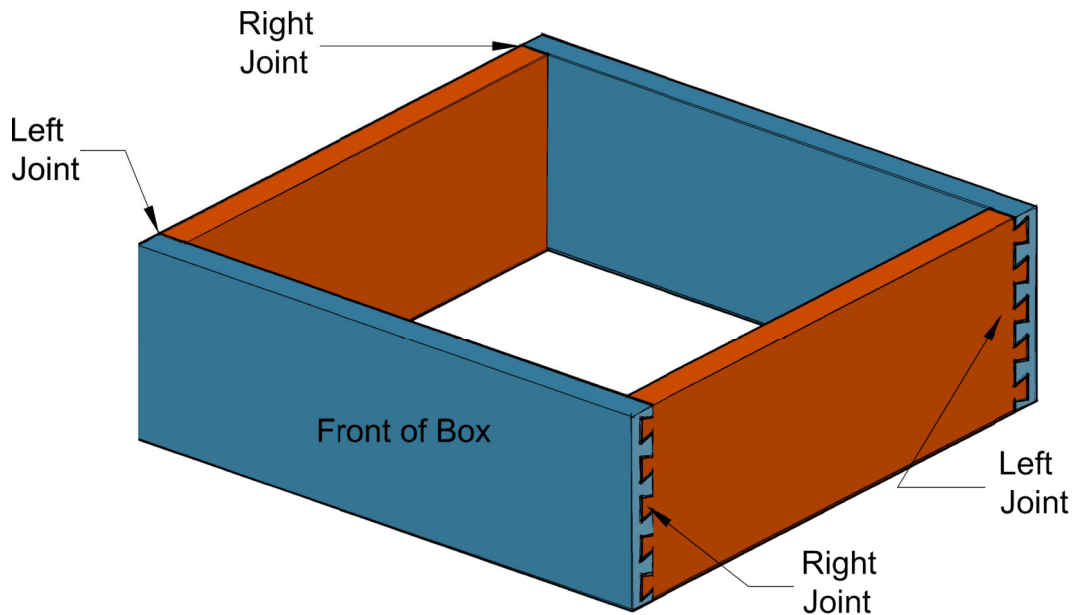


Illustration 4: Typical box with half blind dovetails

Illustration 5 below shows a through dovetail box with the locations of left and right hand joints as JointCAM refers to them.

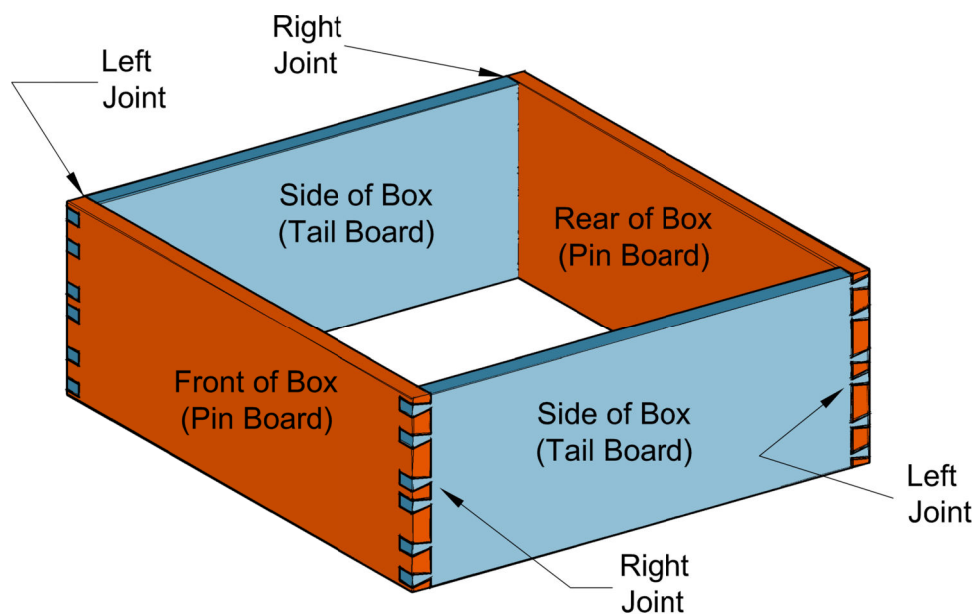


Illustration 5: Typical box with through dovetails

The Half Blind Interface(s)

The following descriptions cover both the Half Blind Equal and Half Blind Custom tabs in JointCAM. Some options are not available on both tabs.

Dovetail Parameters

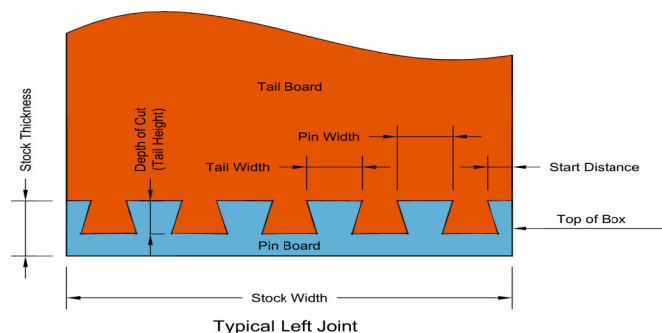


Illustration 6: Half blind dovetail parameters

Width of Stock

The width of the material being used. For a drawer, this is typically the drawer height.

Thickness of Stock

The thickness of the material being used for the tail boards (mounted vertically).

Note: The pin board can be any thickness greater than the depth of cut. If depth of cut is greater than the pin board thickness, JointCAM will be unaware of this situation, and will not give an error.

Depth of Cut

The depth of cut setting is the depth of cut into the pin board, and the height of the dovetails.

Start Distance

The Start Distance is the distance from the top edge of the stock to the edge of the first cut. Three options are available:

½ Tail Width – Start distance will be equal to ½ the width of the tails. At their widest point..

Centered – Dovetails will be centered on the stock. Start distance will be equal at the top and bottom of the stock. As many tails as possible will be used. Use this option for a symmetrical joint, where both left hand and right hand joints will be exactly the same.

(Not available for Half Blind Custom dovetails, as the “Symmetrical” option duplicates this functionality.)

Custom – Enter a user defined Start Distance. An error message will be displayed if the value is less than the minimum required amount.

Dovetail Type

One Pass or Multi-Pass

One pass dovetails use the diameter of the dovetail bit as the width of the dovetails. These are the fastest dovetail joints to cut. Both pin and tail boards can be cut simultaneously.

Multi-Pass dovetails have either pins or tails that are wider than the dovetail bit, requiring multiple passes to cut. If pins and tails are different widths, pin boards and tail boards must be cut in separate operations.

(Not available for Half Blind Custom dovetails.)

Tail Width

Width of Tails for Multi-Pass dovetails – See Illustration 5 above.

(Not available for Half Blind Custom dovetails.)

Pin Width

Width of Pins for Multi-Pass dovetails – See Illustration 5 above.

(Not available for Half Blind Custom dovetails.)

Pins = Tails

When this option is selected, pin width will automatically be set to the tail width. (Multi-Pass only)

(Not available for Half Blind Custom dovetails.)

Full Tails Only

When this option is selected, JointCAM will not allow a partial tail at the bottom edge of the stock.
(Not available for Half Blind Custom dovetails.)

of Tails

of tails in the current layout (read only, updated with calculation)

Pin/Tail Offset =

When simultaneously cutting both pin boards and tail boards, the pin boards must be offset away from X zero or Y zero. A spacer of the correct width is the typical method used.

Pin/Tail Offset varies with pin width and depth of cut.

See Illustration 6.

(Not used for Half Blind Custom dovetails.)

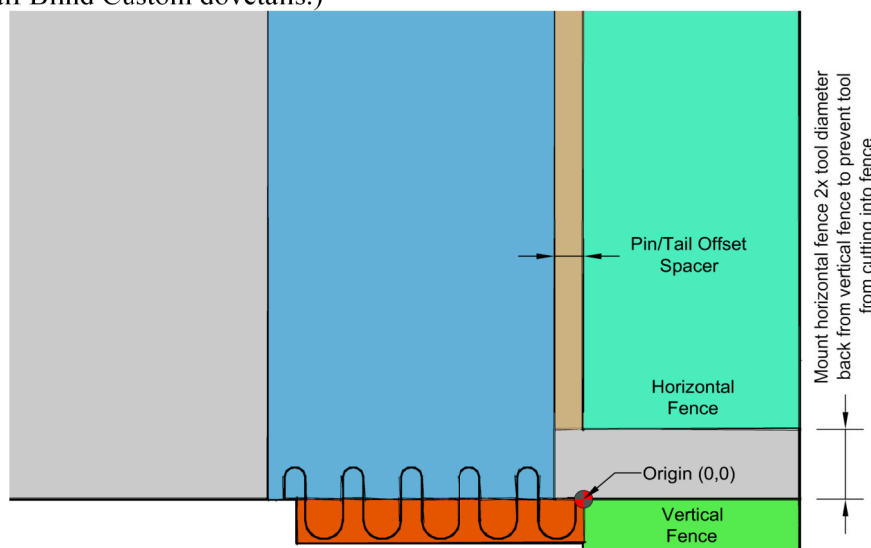


Illustration 7: Top view of Pin/Tail Offset for Y axis Dovetails (Left)

Scoring Pass

Select this option to make a scoring cut prior to cutting the dovetails. The scoring pass should eliminate any tear out or chipping in the tail board.

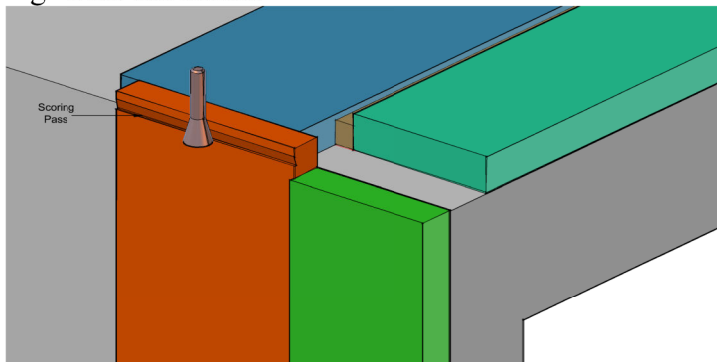


Illustration 8: Typical scoring pass

Scoring Depth

Depth of the scoring pass from face of tailboard.

Clearance

The Clearance value is used to adjust the fit of the dovetail joint. The Clearance is the actual space on each side of the tails, so the actual clearance space will be double the Clearance value. Positive values will increase the clearance, giving a looser fit. Negative values will decrease the clearance, giving a tighter fit. Very small clearance values can have a large effect on the fit. Start with small values if more clearance is needed (.001-.002 inches, or .02-.05 mm's)

For half blind dovetails, JointCAM adjusts the clearance by varying the cut depth. When a Clearance value is used, the actual depth of cut will vary from the entered Depth of Cut.

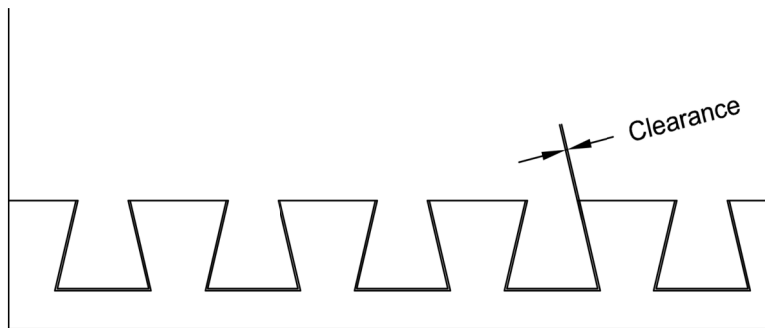


Illustration 9: Dovetail Clearance

Use Roughing Tool

Selecting this option will have JointCAM create roughing passes with a straight bit. Roughing passes can result in better fitting joints on machines where flex is an issue, as roughing will result in lower cutting forces when cutting with the dovetail bit.

Roughing Allowance

Amount of material left to remove after roughing passes.

Note: Roughing Allowance is not used for Single Pass dovetails. With Single Pass dovetails, the roughing toolpaths are single pass as well. JointCAM will give a warning if the selected straight roughing tool is too large. See Illustration 10.

Roughing Depth Allowance

Amount of material left at the bottom of the tail and tail socket roughing cuts. See Illustration 9

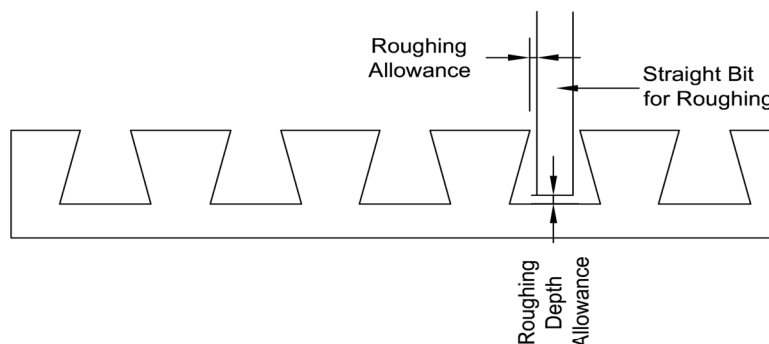


Illustration 10: Roughing Allowance and Depth Allowance

Roughing Stepover %

Percentage of straight tool diameter for roughing pass stepover. Roughing passes are equally spaced, with the value entered being a maximum value.

Note: The first roughing pass between tails will always be a full width pass.

Tail Socket Overcut

The Overcut setting serves two purposes.

- 1) JointCAM's half blind toolpaths cut into the pin board, then reverse direction 180 degrees to move out of the pin board. On some machine controls, constant velocity settings can sometimes result in the tool reversing direction before reaching the full depth of cut. Adding a small overcut value can let the user adjust the socket depth to achieve the desired fit.
- 2) With dovetails, it's often desirable to cut the joints slightly deeper, then sand or trim the protruding edges of the pin boards flush to the tail board surface.

Note: Actual amount of overcut may be different from the entered value. Test cuts may be required to find the correct value to achieve the desired results. Be sure to add 2 times the actual overcut to the length of the pin boards to finish with accurately sized boxes. See Illustrations 11 and 12.

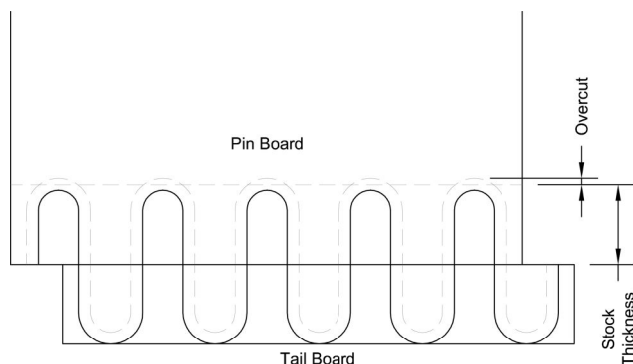


Illustration 11: Tail socket Overcut

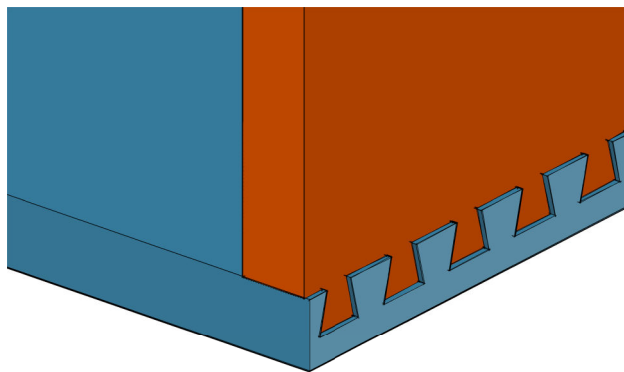


Illustration 12: Effect of Overcut

Calculate / Preview

This button calculates the joint layout and displays a preview of the joint. The preview is a view of the edge of the pin board for a left joint. Joints must be Calculated/Previewed before g-code can be exported.

Note: Changes to any parameters after a preview is created WILL be reflected in the exported g-code, but the preview will NOT be updated to reflect these changes, unless the Calculate/Preview button is clicked again.

Tail/Pin Data (Half Blind Custom only)

See Illustrations 13-15 on next page for examples.

#1 Tail Width through #11 Tail Width

Width of each tail. To create a tail, enter a value greater than or equal to the dovetail tool diameter. Tails must be entered in order. For example, if a value is entered for Tail #1, #2, and #4, but tail #3 is left at zero, an error will occur.

#1 Pin Width through #10 Pin Width

Width of each pin. A pin width must be entered for each tail width entered, except the last tail. If a value is entered for Tail #1, #2, and #3, then pin width values must be entered for Pin #1 and #2.

Use Minimum Pin Width

When checked, all pin widths will be equal to the dovetail tool diameter. All pin width values can be left at 0. Any entered pin width values will be ignored.

Symmetrical

When checked, for each tail created, a duplicate, mirrored tail will be created at the opposite side of the board. This allows for custom joints with up to 22 tails.

Center Last Tail

When checked, the last tail will be centered in the stock. Total # of tails will be (Tails x 2) -1. If values are entered for 3 tails, a total of 5 tails will be created. ((3x2)-1=5)

Center Pin Width (read only)

Displays the width of the center pin for Symmetrical dovetails. When “Center Last Tail” is selected, the Center Pin Width will be the width of the two pins on each side of the center tail.

Clear All

Sets the value of all Tails and Pins to zero.

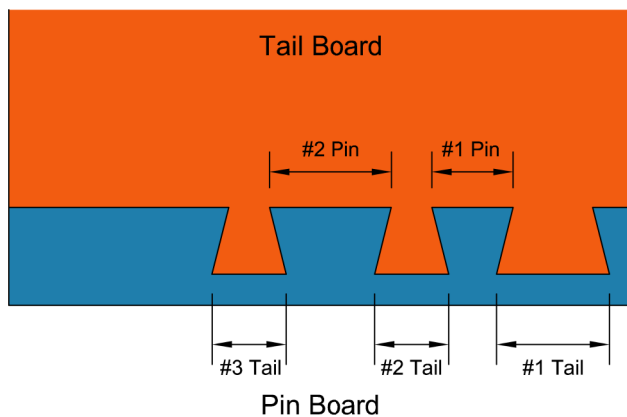


Illustration 13: Non Symmetrical Half Blind Dovetails

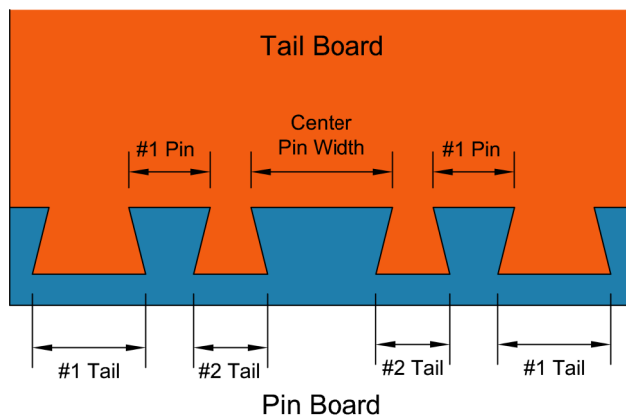


Illustration 14: Symmetrical Half Blind Dovetails - Last Tail NOT Centered

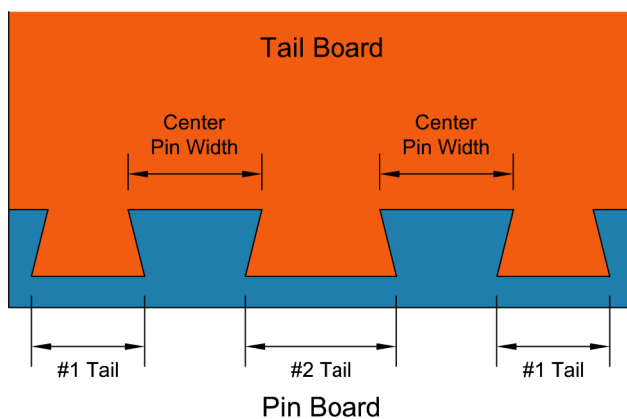


Illustration 15: Symmetrical Half Blind Dovetails - Last Tail Centered

Tool Parameters

The Tool Parameter section provides access to JointCAM's tool table, and also allows direct entry of tool parameters.

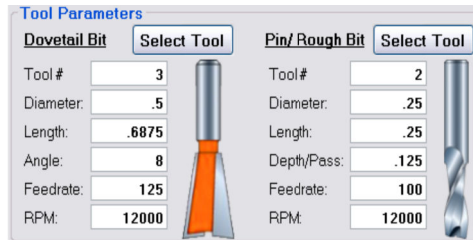


Illustration 16: Tool Parameters

Tool #

Each tool should have a unique tool number. The tool number is only used with tool change commands.

Diameter

Tool diameter at cutting edges. For dovetail bits, the large (bottom) diameter is used. The dovetail diameter is critical for accurate joints. In some cases, metric dovetail bits are actually imperial sized bits, and the stated values are rounded. Be sure to enter an accurate diameter.

Length

Flute (cutting) length of tool.

Angle (Dovetail tools only)

Flute angle of the dovetail bit. Commonly available bits range between 6 and 18 degrees.

Depth/Pass (straight tools only)

Maximum depth of cut per pass for straight tools used in roughing operations. Actual depth of cut may be less than the entered value, as all passes are made at equal depths.

Feedrate

Cutting feedrate for selected tool.

Note: Plunge moves for dovetail bits are done at rapid rate, outside of the workpiece.

Plunge moves for straight bits outside of the workpiece are done at rapid rate. Plunge rates behind the workpiece (into possible backer boards) are done at a percentage of the feedrate specified on the Configuration page.

RPM

RPM for the selected tool.

Select Tool Button

Select Tool opens the Tool table, allowing you to add/edit tools to JointCAM, and select them for use. See the Tool Table section later in this manual for more information.

Machining Options

JointCAM has several options available for cutting half blind dovetails. Individual boards, complete joints, or pairs of joints can all be output into a single g-code file.

Pin & Tail Boards

Select this option to export g-code for complete joints (both pin and tail boards together). When mounting the pin board to the machine, it must be offset away from the fence. The offset distance is the Pin/Tail Offset, displayed in the “Dovetail Type” section after Calculation.

A wood spacer should be used to insure that the pin boards are accurately located relative to the tailboard and fence.

If “Include Tool and Material Comments” is selected in Configuration, the Pin/Tail Offset will be included in the g-code as a comment.

Note: This option is not available if pin and tail widths are not equal. For unequal pins and tail, pin and tail boards must be machined separately.

Pin Boards Only

Select this option to export g-code for pin boards only. When cutting pin boards only, no Pin/Tail Offset spacer is required. The pin board should mount directly against the horizontal fence, at the X or Y zero position.

Tail Boards Only

Select this option to export g-code for tail boards only.

Left & Right Joints

Select this option to export g-code for both left and right joints in one file.

Half blind dovetails are cut with the inside of the joint facing out when mounted on the machine for machining. Because of this, left joints are actually machined on the right side of the machine, and right joints are machined on the left. Refer to Illustration 4 for reference. The machining options graphic displays the stock orientation as it will be mounted on the machine, based on the selected machining options.

Left Joints Only

Select this option to export g-code for Left Joints only.

Right Joints Only

Select this option to export g-code for right joints only.

Load Project

Load a previously saved Project (.jnt file).

JointCAM allows the users to save the current parameters for all joint types into a JointCAM Project file. See “Save Project” below for more information. If the Project file contains a joint type other than

the currently selected joint, JointCAM will switch to the appropriate joint type and load the data for that joint.

Save Project

Save all options for the current joint type to a JointCAM Project file (.jnt file). This allows you to design a particular joint , and recall the joint at a later time to create g-code.

The .jnt file is a simple text file. Do not attempt to edit .jnt files outside of JointCAM.

Write G-Code File

Click this button to export a g-code file with the selected parameters. The “Write G-Code File” button is disabled until a “Calculation/Preview” is performed. If an error is encountered during the calculation, the “Write G-Code File” button will remain disabled.

Note: Changes to any parameters after a preview is created WILL be reflected in the exported g-code, but the preview will NOT be updated to reflect these changes, unless the Calculate/Preview button is clicked again.

Save Settings

Saves all available parameters for the current joint type as default settings that will be loaded each time that JointCAM is run.

Note: Tool parameters will be stored as shown, and may differ from parameters stored for the tool in the tool table. To restore default tool parameters from the tool table, use the “Select Tool” button and re-select the tool from the tool table.

Exit

The Exit button will close JointCAM, with no opportunity to save any current work.

Creating a Half blind dovetail joint

Creating half blind dovetails in JointCAM is a quick and easy process.

- 1) Enter the width and thickness of the stock you'll be using.
- 2) Enter tool information, or select tool(s) from tool table.
- 3) Select the desired options.
- 4) Click the Calculate / Preview button to preview the joint. Adjust parameters if needed until desired joint is achieved. Calculate Preview to see the result of any changes.
- 5) Select Machining Options.
- 6) Save Project (Optional).
- 7) Write G-Code file.
- 8) Mount stock on machine.
- 9) Set X, Y, and Z zero positions.
- 10) Machine a test joint to verify fit of joint. Adjust Clearance and/or Tail Socket Overcut if needed.
- 11) Start Machining dovetails.

On machines that have rigidity issues resulting in flex at the tool, using a straight bit to remove the bulk of material (roughing passes) prior to using the dovetail bit can result in much better joints, as the cutting forces with the dovetail bit will be much lower. An alternative to using roughing passes would be to reduce the feedrate. However, lowering the feedrate may result in reduced tool life due to excessive heat.

Stock and Machine Setup

Prior to machining dovetails, all stock should be prepared to a uniform thickness, and must be straight, flat and square to achieve best results.

If the machine fences or jig/fixture to be used can be located in a repeatable location, then the user may find it desirable to set up a work offset to use exclusively when cutting joints with JointCAM. The user can simply add the work coordinate offset (typically G54-G59) as one of the Startup Codes on the Configuration page of JointCAM. This can greatly reduce setup time, by possibly eliminating the task of setting X and Y zero each time you machine joints with JointCAM.

It's strongly recommended to make as many test cuts as needed to be sure that the machine is cutting accurate joints.

Pins and Tails

When machining both pins and tails, both the pin boards and tailboards must be mounted to the machine very precisely. The top end of the tailboard must be flush to the top surface of the pin board, and the end of the pin board must be tight to the rear face of the tailboard. Illustration 17 depicts a typical left joint setup on a machine setup to cut dovetails along the Y axis.

The tail board mounts with the top edge against the vertical fence, which is the Y zero position. The pin board is offset away from the horizontal fence by the Pin/Tail Offset amount. A spacer block is shown. (See Illustration 7 for more information)

The X zero position is the rear face of the tail board, which is also the location where the pin and tail boards meet. In JointCAM, the tailboard is located in negative X coordinates, while the pin board is located in positive X coordinates.

The Z zero position is located at the top face of the pin board, which is in the same plane as the end of the tail board.

Right joints are mounted against the opposite fence, in a mirror image orientation from that shown in Illustration 17.

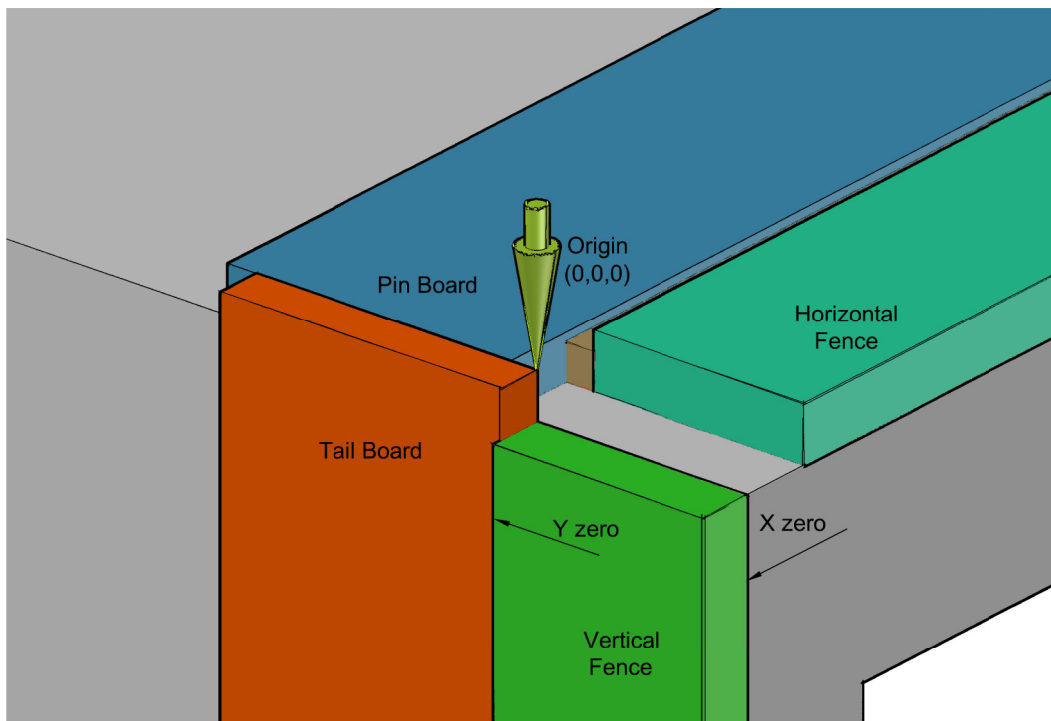


Illustration 17: Left hand half blind joint setup – Pin & Tail boards

Tail Boards Only

When machining tail boards only, the boards are mounted in the same locations as when machining both pins and tails. The top edge of the tail board is mounted against the vertical fence (Y zero in the illustration). Illustration 18 depicts a tail board setup for a typical left joint on a machine setup to cut dovetails along the Y axis.

While the location of the end of the board (mounting height) is not critical, it must be higher than the table top by a distance at least equal to the depth of cut, to prevent cutting into the table. Using a scrap “backer board” equal in thickness to the pin board, will help both to reduce tearout on the backside of the tail board, and also provide a reference height for mounting the tail board.

Z zero is set to the top end of the tail board.

Right joints are mounted against the opposite fence, in a mirror image orientation from that shown in Illustration 18.

Note: To cut both left and right joints using a single fence located at the origin, Select “Machine Width = Stock Width” in the JointCAM Configuration page.

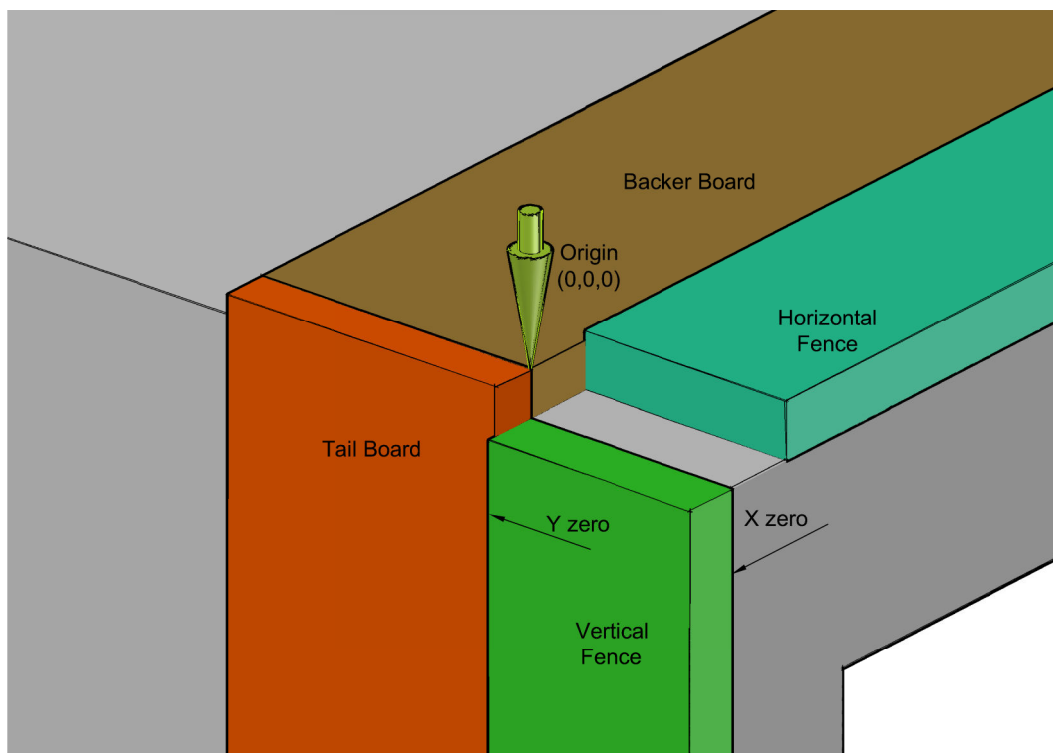


Illustration 18: Left half blind joint setup - Tail boards only

Pin Boards Only

When machining pin boards only, the edge of the pin board is mounted against the edge of the horizontal fence (Y zero in the illustration). The Pin/Tail Offset is not used, and no spacer is required. The end of the pin board is located at the edge of the vertical mounting surface (X zero in the illustration). Illustration 19 depicts a pin board setup for a typical left joint on a machine setup to cut dovetails along the Y axis.

Z zero is set to the top face of the pin board.

Right joints are mounted against the opposite fence, in a mirror image orientation from that shown in Illustration 19.

Note: To cut both left and right joints using a single fence located at the origin, Select “Machine Width = Stock Width” in the JointCAM Configuration page.

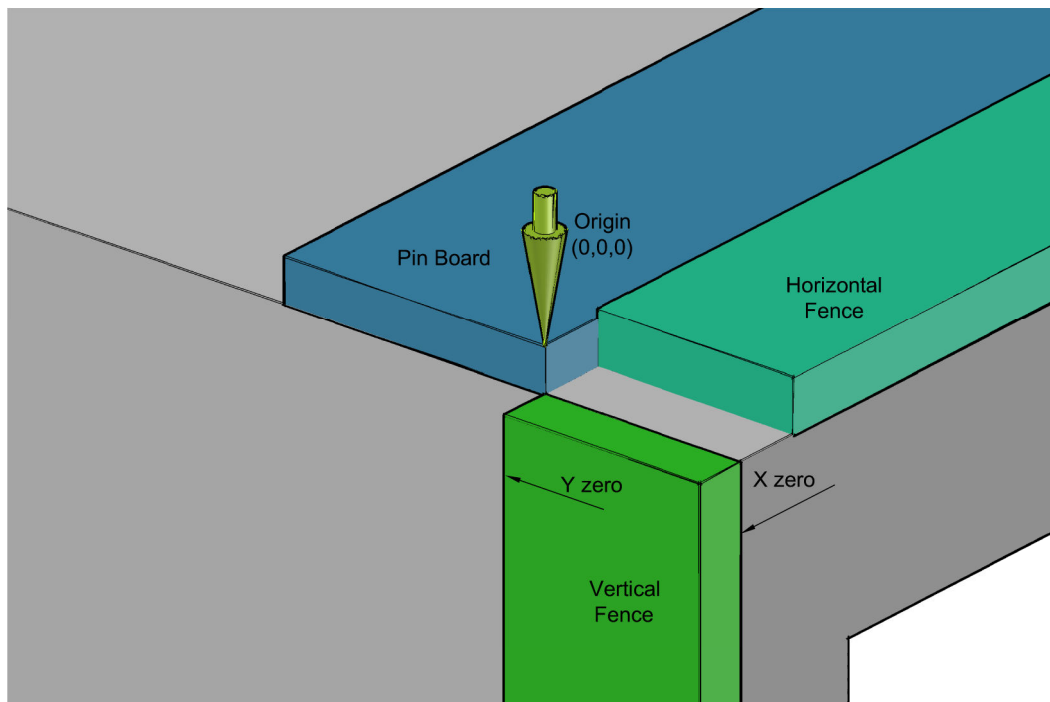


Illustration 19: Left half blind joint setup - Pin boards only

The Through Dovetail Interface(s)

The following descriptions cover both the Through Equal and Through Custom tabs in JointCAM. Some options are not available on both tabs.

Dovetail Parameters

Width of Stock

The width of the material being used. For a drawer, this is typically the drawer height.

Thickness of Stock

The thickness of the material being used for both the pin and tail boards. Currently both pin and tail boards must be the same thickness.

Tail Height

With through dovetails, the tail height is normally equal to the stock thickness. However, JointCAM offers an alternate through dovetail with a tail height less than the stock thickness. This is accomplished by cutting a rabbet on the inside of the pin board, which reduces the tail height. This allows you to cut a through dovetail when available tools are shorter than the stock thickness, or to achieve a specific look.

Tail Width

Width of Tails.
(Not available for Through Custom dovetails.)

Pin Width

Width of Pins.
(Not available for Through Custom dovetails.)

Pin/Tail Extension

Distance that both pins and tails will extend past the corner of the assembled joint. Cutting the pins and tails slightly long allows the extra material to be trimmed away after assembly, giving more consistent quality joints.

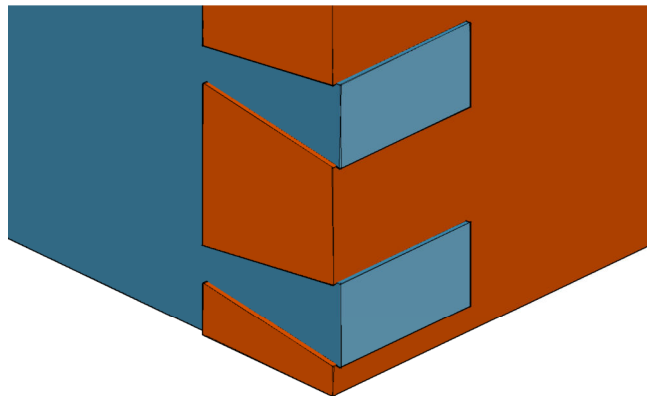


Illustration 20: Pin/Tail Extension

Pins /Tails = Tool

When this option is selected, both tail and pin widths will automatically be set to diameter of the currently selected tool.

Note: Changing the tool after selecting this option will not update the pin and tail widths.
(Not available for Through Custom dovetails.)

Pins = Tails

When this option is selected, pin width will automatically be set to the tail width.
(Not available for Through Custom dovetails.)

Start Distance

The Start Distance for through dovetails is either the distance from the top edge of the stock to the edge of the first tail (when Tails First is selected) or the width of the first pin (when Pins First is selected).
(See Configuration option below)

Three Start Distance options are available:

½ Tail Width – Start distance will be equal to ½ the width of the tails at their widest point (top).

Centered – Dovetails will be centered on the stock. Start distance will be equal at the top and bottom of the stock. As many tails as possible will be used. Use this option for a symmetrical joint, where both left hand and right hand joints will be exactly the same.
(Not available for Through Custom dovetails, as the “Symmetrical” option duplicates this functionality.)

Custom – Enter a user defined Start Distance. When Tails First is selected, an error message will be displayed if the value is less than the minimum required amount.

Configuration (Pin First or Tail first)

Unlike half blind dovetails, which always start with a pin, through dovetails give you the option to start with either a pin or tail.

Roughing Options

The roughing options for through dovetails are the same as for half blind dovetails. Please see the half blind section for more information.

Clearance

See the half blind Clearance section for more information.

For through dovetails, JointCAM adjusts the clearance by varying the width of the pins. Tail size and spacing remains as specified.

Calculate / Preview

This button calculates the joint layout and displays a preview of the joint. The preview for through dovetails displays a side view of the end of the tail board for a *right* joint. Joints must be Calculated/Previewed before g-code can be exported.

Note: Changes to any parameters after a preview is created WILL be reflected in the exported g-code, but the preview will NOT be updated to reflect these changes, unless the Calculate/Preview button is clicked again.

Tail/Pin Data (Through Custom only)

See Illustrations 21-26 for examples.

#1 Tail Width through #11 Tail Width

Width of each tail. To create a tail, enter a value greater than or equal to the dovetail tool diameter. Tails must be entered in order. For example, if a value is entered for Tail #1, #2, and #4, but tail #3 is left at zero, an error will occur.

#1 Pin Width through #10 Pin Width

Width of each pin. A pin width must be entered for each tail width entered, except the last tail. If a value is entered for Tail #1, #2, and #3, then pin width values must be entered for Pin #1 and #2.

Note: The location of the Pin and Tail width input boxes (text boxes) will change to reflect the setting of the Pin First / Tail First Configuration.

Use Minimum Pin Width

When checked, all pin widths will be equal to the dovetail tool diameter. All pin width values can be left at 0. Any entered pin width values will be ignored.

Symmetrical

When checked, for each tail created, a duplicate, mirrored tail will be created at the opposite side of the board. This allows for custom joints with up to 22 tails.

Center Last Tail

When checked, the last tail will be centered in the stock. Total # of tails will be $(\text{Tails} \times 2) - 1$. If values are entered for 3 tails, a total of 5 tails will be created. $((3 \times 2) - 1 = 5)$

Note: When Tails First is selected, 2 additional partial tails will be added to the quantities mentioned above.

Center Pin Width (read only)

Displays the width of the center pin for Symmetrical dovetails. When “Center Last Tail” is selected, the Center Pin Width will be the width of the two pins on each side of the center tail.

Clear All

Sets the value of all Tails and Pins to zero.

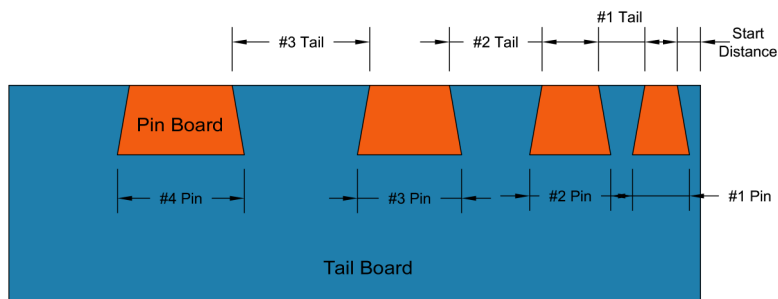


Illustration 21: Non Symmetrical Through Dovetails - Tails First

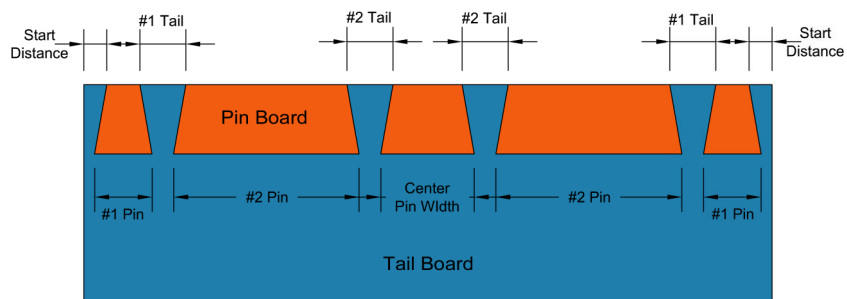


Illustration 22: Symmetrical Through Dovetails - Last Tail NOT Centered - Tails First

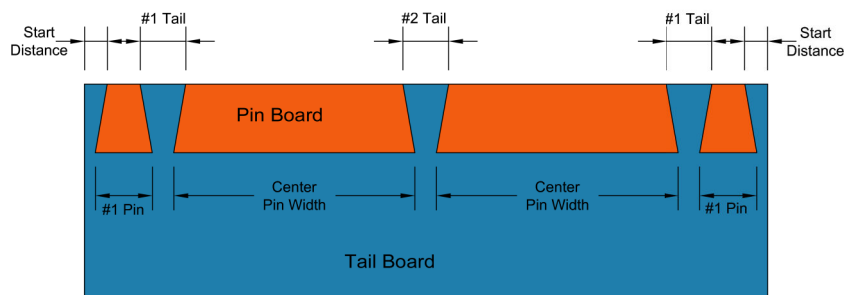


Illustration 23: Symmetrical Through Dovetails - Last Tail Centered - Tails First

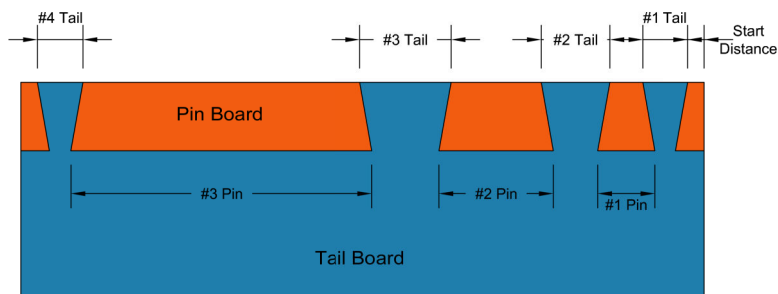


Illustration 24: Non-Symmetrical Through Dovetail - Pins First

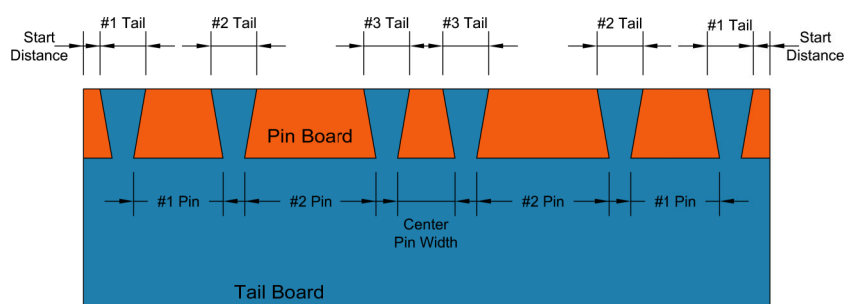


Illustration 25: Symmetrical Through Dovetails - Last Tail NOT Centered - Pins First

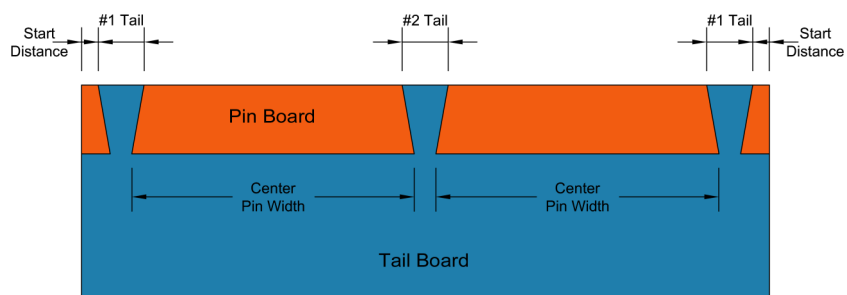


Illustration 26: Symmetrical Through Dovetails - Last Tail Centered - Pins First

Tool Parameters

The tool parameters for through dovetails are the same as explained in the half blind dovetail section. However, when machining through dovetails, the pin boards are cut with a straight bit only.

With through dovetails, the minimum tail width is limited by the diameter of the selected dovetail bit, while the minimum tail width is limited by the diameter of the selected straight bit.

Machining Options

JointCAM has several options available for machining through dovetails. Individual boards, or pairs of joints can be output into a single g-code file. Both boards are machined vertically, and must be machined in separate operations.

Pin Boards Only

Select this option to export g-code for pin boards only.

Tail Boards Only

Select this option to export g-code for tail boards only.

Left & Right Joints

Select this option to export g-code for both left and right joints in one file.

The mounting orientations for through dovetails are different than for half blind dovetails. Pin boards are machined with the inside of the joint facing out when mounted on the machine. Tail boards are machined with the outside face facing out. Right joints are machined on the right side of the machine, and left joints are machined on the left. Refer to Illustration 4 for reference. The machining options graphic displays the stock orientation as it will be mounted on the machine, based on the selected machining options.

Left Joints Only

Select this option to export g-code for Left Joints only.

Right Joints Only

Select this option to export g-code for right joints only.

Multiple Tails

Selecting this option allows you to cut multiple tail boards in one operation. Enter the quantity of boards to be cut simultaneously. The toolpath length is calculated by multiplying the stock thickness x quantity entered.

This option can also be used if a backer board is to be mounted outside of the stock, to have the toolpath move straight through the backer board.

Load Project

Load a previously saved Project (.jnt file).

JointCAM allows the users to save the current parameters for all joint types into a JointCAM Project file. See “Save Project” below for more information. If the Project file contains a joint type other than the currently selected joint, JointCAM will switch to the appropriate joint type and load the data for that joint.

Save Project

Save all options for the current joint type to a JointCAM Project file (.jnt file). This allows you to design a particular joint , and recall the joint at a later time to create g-code.

The .jnt file is a simple text file. Do not attempt to edit .jnt files outside of JointCAM.

Write G-Code File

Click this button to export a g-code file with the selected parameters. The “Write G-Code File” button is disabled until a “Calculation/Preview” is performed. If an error is encountered during the calculation, the “Write G-Code File” button will remain disabled.

Note: Changes to any parameters after a preview is created WILL be reflected in the exported g-code, but the preview will NOT be updated to reflect these changes, unless the Calculate/Preview button is clicked again.

Save Settings

Saves all available parameters for the current joint type as default settings that will be loaded each time that JointCAM is run.

Note: Tool parameters will be stored as shown, and may differ from parameters stored for the tool in the tool table. To restore default tool parameters from the tool table, use the “Select Tool” button and re-select the tool from the tool table.

Exit

The Exit button will close JointCAM, with no opportunity to save any current work.

Creating a through dovetail joint

Creating through dovetails in JointCAM is very similar to creating half blind dovetails. The main difference is that pins and tails must be cut separately, and both boards are mounted vertically.

On machines that have rigidity issues resulting in flex at the tool, using a straight bit to remove the bulk of material (roughing passes) prior to using the dovetail bit can result in much better joints, as the cutting forces with the dovetail bit will be much lower. An alternative to using roughing passes would be to reduce the feedrate. However, lowering the feedrate may result in reduced tool life due to excessive heat.

Pin Boards

When machining pin boards, the TOP edge of the pin board is mounted against the edge of the vertical fence (Y zero in the illustration).

Z zero is set to the top face of the pin board.

Right joints are mounted against the right side fence, as shown in Illustration 27.

Left joints are mounted against the left side fence, in a mirror image orientation from that shown in Illustration 27.

Note: To cut both left and right joints using a single fence located at the origin, Select “Machine Width = Stock Width” in the JointCAM Configuration page.

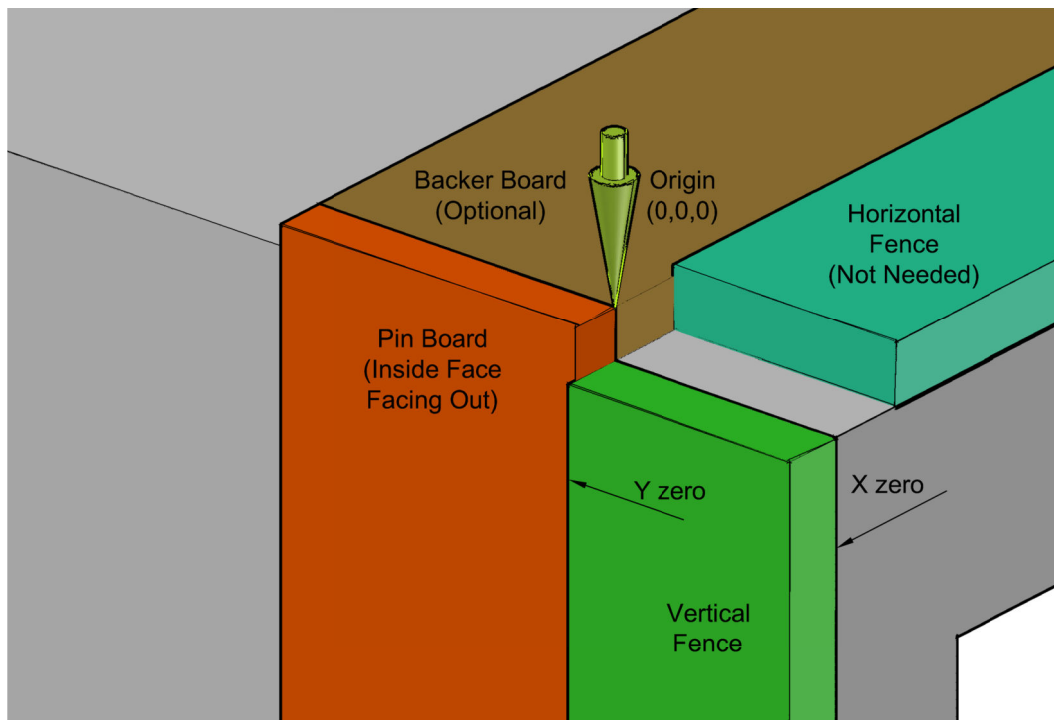


Illustration 27: Right Hand Through Dovetail Pin Board Setup

Tail Boards

When machining tail boards, the TOP edge of the tail board is mounted against the edge of the vertical fence (Y zero in the illustration).

Z zero is set to the top face of the pin board.

Right joints are mounted against the right side fence, with the outside face facing out, as shown in Illustration 28.

Left joints are mounted against the left side fence, in a mirror image orientation from that shown in Illustration 28.

Note: To cut both left and right joints using a single fence located at the origin, Select “Machine Width = Stock Width” in the JointCAM Configuration page. Only one side can be cut at a time.

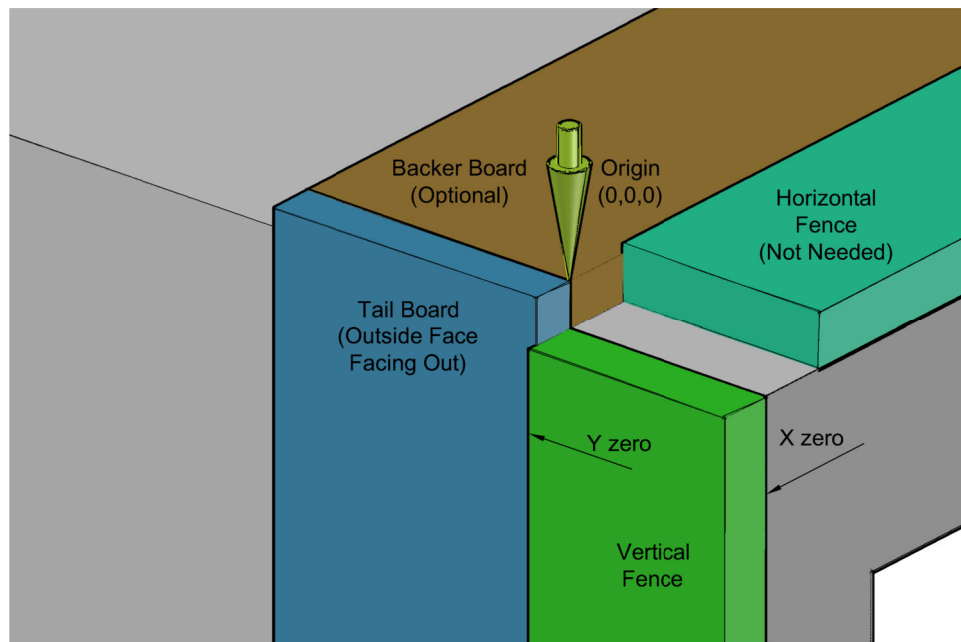


Illustration 28: Right Hand Through Dovetail Tail Board Setup

Box Joints

Box joints (also known as finger joints) are a simpler alternative to dovetail joints. They are very strong, due to the large glue surface area, and very simple to cut on a CNC router. Box joints have traditionally been cut with table saw or router table jigs.

JointCAM can create tool paths and g-code for 3 types of box joint:

- 1) Standard box joints – Box joints with standard rectangular shaped interlocking fingers. JointCAM gives you several layout options, allowing you to create an unlimited variety of joint designs.
- 2) Coming Soon!
- 3) Coming Soon!

Illustration 29 below depicts a typical box using box joints, with the joints and boards labeled as JointCAM refers to them. Note that board “A” always starts with a finger at the top of the joint. Both left and right joints can be cut in each others machine location, by keeping the top edge against the vertical fence.

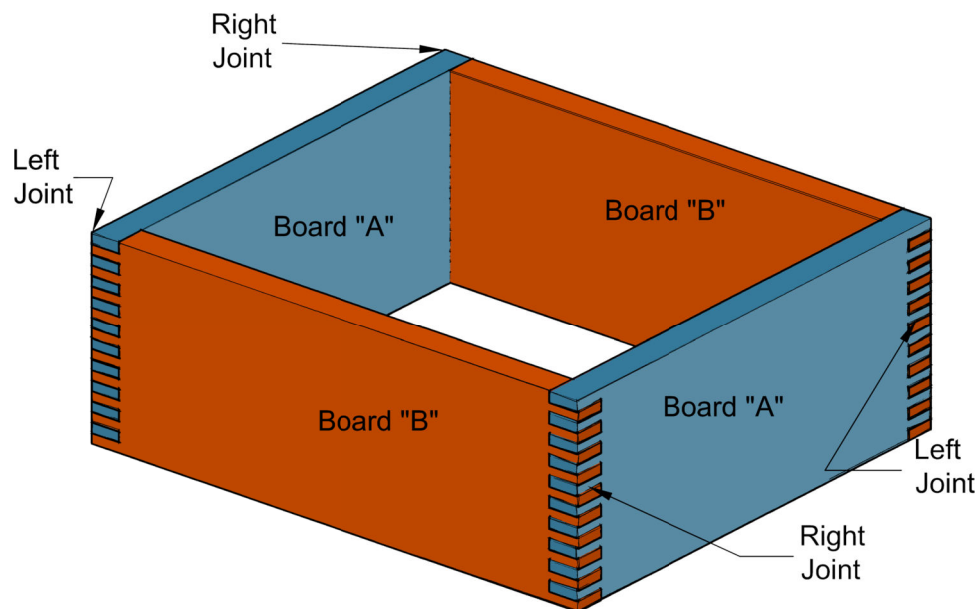


Illustration 29: Typical Box with Box Joints

The Box Joint Interface

The following descriptions cover both the Half Blind Equal and Half Blind Custom tabs in JointCAM. Some options are not available on both tabs.

Box Joint Parameters

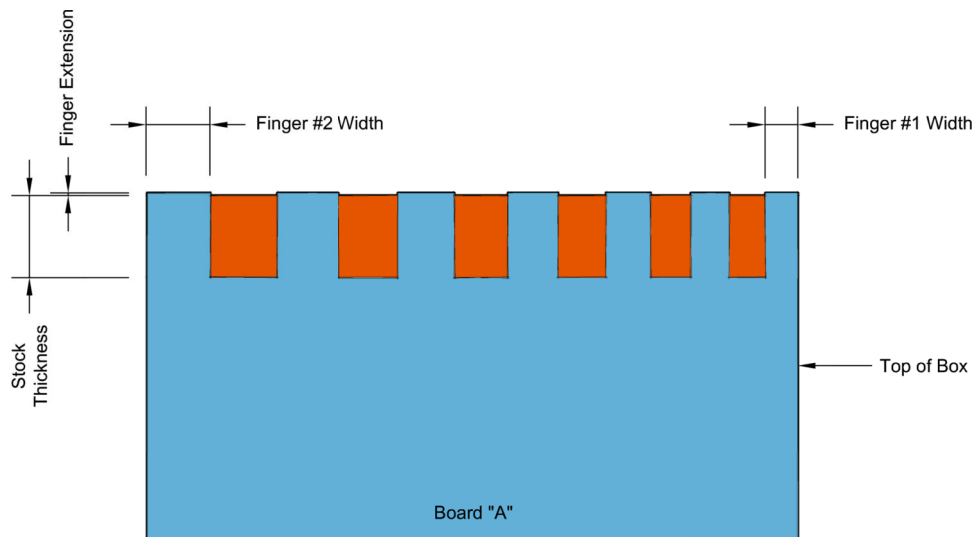


Illustration 30: Parameters for a Right Graduated Box Joint

Width of Stock

The width of the material being used. For a drawer, this is typically the drawer height.

Thickness of Stock

The thickness of the material being used for the box joints.
The thickness is also the depth of cut.

Finger #1 Width & Finger #2 Width

How these values are used depends on the Layout Type chosen by the user. Three different Layout Types use these values:

- 1) For an Equal layout, the Finger #1 Width is the width of all fingers in both "A" and "B" boards. Finger #2 Width is not used with Equal Layouts. (See Start Distance below for changing the top finger width of the "A" board.
- 2) For a Graduated Layout, the Finger #1 Width is the width of the first (top) finger of the "A" board. Finger #2 Width is the width of the last (bottom) finger of the "A" board. JointCAM will gradually increase or decrease the width of the fingers from the first finger (#1) to the last finger (#2), with the increase (or decrease) in size being uniform for each finger in the joint. The smaller of the two values will be the actual size entered. The larger of the two values will most often be an approximate value, and may be slightly larger or smaller than the specified value, in order to provide a uniform increase in finger widths.

- 3) For a Graduated-Mirrored Layout, the Finger #1 width is the width of both the first (top) and last (bottom) fingers of the joint. The Finger #2 Width is the width of the center finger in the joint. JointCAM will gradually increase or decrease the width of the fingers from the first finger (#1), to the center finger (#2) and back to the last finger (#1 again), with the increase in size being uniform for each finger in the joint. As with the Graduated Layout, the smaller of the two values will be the actual size entered. The larger of the two values will most often be an approximate value, and may be slightly larger or smaller than the specified value, in order to provide a uniform increase in finger widths.

Finger Extension

An allowance to the depth of cut, to allow for trimming the assembled joint flush.

Start Distance

The Start Distance sets the width of the first finger at top edge of the “A” board. Start Distance is only used for Equal Layouts. Three options are available:

Full Finger – First finger will be the same size as all other fingers, as specified in Finger #1 Width setting.

Centered – First and last fingers will be equal, with all other fingers being equal to Finger #1 Width. The first and last finger widths will be less than or equal to the Finger #1 width.

Custom – Enter a user defined width for the first finger. All other fingers will be equal to the Finger #1 width.

Layout Type

JointCAM has four options for laying out your box joints:

Equal – All fingers will be equal in width, as specified in Finger #1 Width setting. There are two exceptions. The first finger width can be modified with the Start Distance settings, and the last finger width is dictated by the stock width.

Graduated – Fingers will gradually increase or decrease in width across the joint.

Graduated-Mirrored – Fingers will gradually increase or decrease in width across the joint from the edge to the center, and then again to the opposite edge in a mirrored pattern.

Custom – Allows the user to enter a user defined width for each finger. The space between the fingers are labeled as “Sockets”, and are the widths of the fingers of the “B” board.

See Finger #1 Width & Finger #2 Width settings above for more information on the Graduated and Graduated-Mirrored Layout Types.

Finger Width Data

Parameters for Custom Layout Type box joints.

Finger and Socket width entry boxes

Each Finger and socket (“B” finger) width can be entered for a truly custom appearance.

Sockets = Fingers

When this option is selected, each socket ("B" finger) width will be equal to the preceding finger width. This also doubles the available number of finger width entry boxes.

Sockets = Tool Diameter

When this option is selected, each socket ("B" finger) width will be equal to the tool diameter. This also doubles the available number of finger width entry boxes.

Symmetrical

When this option is selected, each finger and socket will be mirrored at the opposite edge of the board.

Center Finger/Socket Width

A read-only display of the width of the center finger or socket in a Symmetrical joint. Useful for fine tuning finger widths.

Tool Parameters

The Tool Parameter section provides access to JointCAM's tool table, and also allows direct entry of tool parameters. Box joints only require a single straight tool. The available parameters are described below.

Tool #

Each tool should have a unique tool number. The tool number is only used with tool change commands.

Diameter

Tool diameter at cutting edges. For dovetail bits, the large (bottom) diameter is used. The dovetail diameter is critical for accurate joints. In some cases, metric dovetail bits are actually imperial sized bits, and the stated values are rounded. Be sure to enter an accurate diameter.

Length

Flute (cutting) length of tool.

Depth/Pass (straight tools only)

Maximum depth of cut per pass for straight tools used in roughing operations. Actual depth of cut may be less than the entered value, as all passes are made at equal depths.

Feedrate

Cutting feedrate for selected tool.

Plunge moves for straight bits outside of the workpiece are done at rapid rate. Plunge rates behind the workpiece (into possible backer boards) are done at a percentage of the feedrate specified on the Configuration page.

RPM

RPM for the selected tool.

Select Tool Button

Select Tool opens the Tool table, allowing you to add/edit tools to JointCAM, and select them for use. See the Tool Table section later in this manual for more information.

Machining Options

JointCAM has several options available for cutting box joints. Individual boards, complete joints, or pairs of joints can all be output into a single g-code file. Box joints are cut with both boards in the vertical position.

Finger Board “A”

Select this option to export g-code for “A” boards only. The board should be mounted directly against the vertical fence, at the X or Y zero position (or the opposite side fence). The exterior face of the “A” board should be facing out when mounted.

Finger Board “B”

Select this option to export g-code for “B” boards only. The board should be mounted directly against the vertical fence, at the X or Y zero position (or the opposite side fence). The interior face of the “B” board should be facing out when mounted.

“A” and “B” Boards

Select this option to export g-code for complete joints (both “A” and “B” boards together). The “A” board is positioned against the vertical mounting surface, with the edge against the vertical fence. The “B” board is placed in front of the “A” board, offset from the vertical fence by the width of the first finger.

Note: This option is only available for Equal layouts, with the first finger set to “full finger”. For all other layout types, both “A” and “B” boards must be machined separately.

Multiple Boards

When this option is selected, multiple identical “A” or “B” boards can be cut simultaneously. Enter the number of boards to be cut, and JointCAM will create a tool path to accommodate the chosen number of boards. Be sure the machine has the capability to reach over the multiple boards.

Left & Right Joints

Select this option to export g-code for both left and right joints in one file.

Left Joints Only

Select this option to export g-code for Left Joints only.

Right Joints Only

Select this option to export g-code for right joints only.

Load Project

Load a previously saved Project (.jnt file).

JointCAM allows the users to save the current parameters for all joint types into a JointCAM Project file. See “Save Project” below for more information. If the Project file contains a joint type other than the currently selected joint, JointCAM will switch to the appropriate joint type and load the data for that joint.

Save Project

Save all options for the current joint type to a JointCAM Project file (.jnt file). This allows you to design a particular joint , and recall the joint at a later time to create g-code.

The .jnt file is a simple text file. Do not attempt to edit .jnt files outside of JointCAM.

Write G-Code File

Click this button to export a g-code file with the selected parameters. The “Write G-Code File” button is disabled until a “Calculation/Preview” is performed. If an error is encountered during the calculation, the “Write G-Code File” button will remain disabled.

Note: Changes to any parameters after a preview is created WILL be reflected in the exported g-code, but the preview will NOT be updated to reflect these changes, unless the Calculate/Preview button is clicked again.

Save Settings

Saves all available parameters for the current joint type as default settings that will be loaded each time that JointCAM is run.

Note: Tool parameters will be stored as shown, and may differ from parameters stored for the tool in the tool table. To restore default tool parameters from the tool table, use the “Select Tool” button and re-select the tool from the tool table.

Exit

The Exit button will close JointCAM, with no opportunity to save any current work.

The JointCAM Tool Table

JointCAM has a tool table where the user can store and edit tool information, and quickly import the tool parameters into a project.

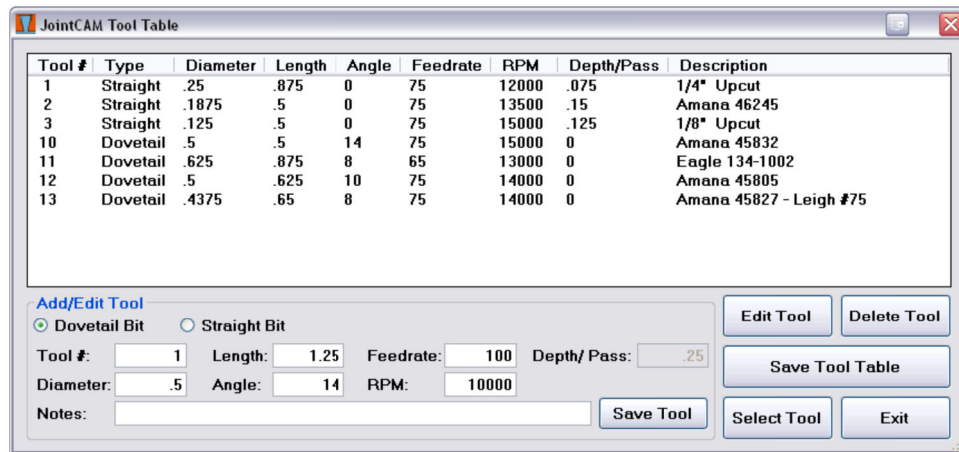


Illustration 31: The JointCAM Tool Table

Adding Tools to the tool table

To add tools to the tool table, enter the tool information in the Add/Edit tool section, and click the Save Tool button.

Editing Tools in the tool table

To edit tool information, select a tool in the table by clicking on it to highlight the row that it's on. Once highlighted, click the Edit Tool button to load the tool into the Add Edit tool area. Note: This will remove the tool from the tool table during editing. Once any changes are made, click Save Tool to save the tool back to the tool table. If the Save Tool button is not clicked, the tool will not be restored to the table.

Note: After any changes have been made to the tool table, the Save Tool Table button must be clicked to save the changes. If the Save Tool Table button is not clicked, all changes will be lost. A message will warn the user if the table has not been saved.

Deleting Tools from the tool table

To delete a tool from the tool table, select it by clicking on the tool to highlight the row that it's on, and click the Delete Tool button.

Selecting a tool for use

To select a tool for use in JointCAM, click on the tool to highlight the row that it's on, and click the Select Tool button. This will close the Tool table, and import the selected tool parameters into JointCAM.

Exit

The Exit button will close the tool table and return to JointCAM without importing any tool data.