



# MADA QUICK START GUIDE CNC FLAT BED ROUTER



## 1200MM X 2400MM FLAT BED CNC ROUTER BREIF INSTRUCTIONS FOR FILE PREPARATION.

THIS DOCUMENT IS LIMITED IN SCOPE TO COMMON PROCESSES TYPICAL OF FLAT BOARDS OR SHEET MATERIAL ON A 3 AXIS CNC ROUTER. SEE ALSO QUICK START GUIDE FOR 3D SHAPES.

### In this document:

#### How CNC works.

2 and 2.5D machine strategies used on a 3 axis CNC router

Holding down the Material

Profile cutting from flat board or sheet

Pocket milling on a flat board or sheet material (2.5D)

Dog Bone Flyouts

Tabs (Bridges)

Drill and Pre-milling

Setting up your drawing file so we can use it

Default assumptions we place on your drawing

File structure and naming

Drawing element structure

Tips

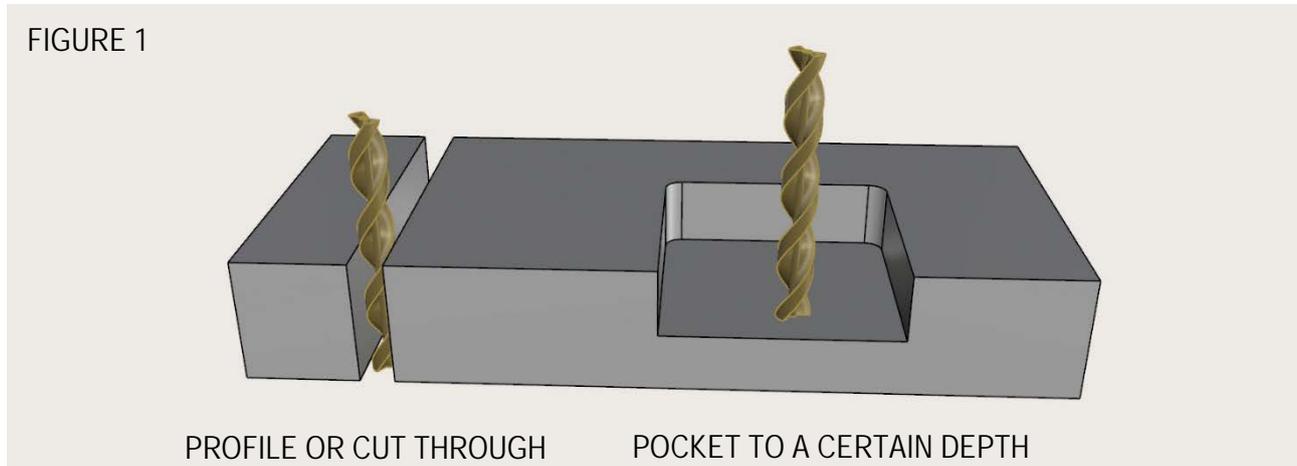
### How **COMPUTER NUMERIC CONTROL** works:

3 Axis CNC routing uses the same XYZ Cartesian coordinate system you find in CAD packages. If you imagine a horizontal construction plane in your cad software as the CNC table surface, then the vertical axis 'Z' will be elevation off the table surface, while the Y axis usually runs along the table, (and does so in the case of the MADA machine). The following three stages are required in order to command a CNC machine to perform an operation.

- The part must be drawn in a cad program as vectors, surfaces or solids.
- These CAD drawing elements are then imported into CAM software (Computer Aided Manufacture) This takes information from the CAD file and combines it with user defined parameters such as the type of cutting tool and how fast to cut. Once the tool path has been calculated as vectors it is then converted into Gcode. This process breaks down all tool path vectors into segments (tessellation) and produces a large amount of sequential straight lines, or in some cases arcs as well. Each of these segments has a start and end point in 3D space, referred to in XYZ coordinates. A list of these sequential coordinates (Gcode) instructs the machine to move from one point in 3D space into another, then another and so on.
- The Gcode is loaded into the machine controller, the controller then reads the Gcode and converts it into machine movement via electrical pulses.

## 2 and 2.5D machine strategies used on a 3 axis CNC router.

While a CNC router can make complicated 3D parts we are only concerned here with either profile cutting 2D or pocket milling 2.5D. (Fig.1.) The part must be drawn in a cad program as vectors, surfaces or solids.



### Holding down the Material.

There are two common ways to hold the material on the router table.

1. Clamp, screw or double sided tape the material down. Leave sufficient peripheral fixing points.
2. Suck away the air underneath the material so its held by Vacuum.

The first methods are used for solid timber as a typical board does not have enough surface area to be held by vacuum. Where there is a large surface area with large parts to be cut a vacuum works well. The largest sheet size possible is 2400 (Y axis) x 1200 mm.( X axis).

### Profile cutting from flat board or sheet material (2D):

Understanding the process and its limitations.

Profile cutting will release a shape from the parent material; cut all the way through. Key limitations:

- Because a router bit has a diameter that is typically 6 to 12mm in diameter it cannot cut a sharp right angle inside corner. Material will remain on any inside corner that equals the radius of the cutter. This is particularly a problem where a square hole is required to neatly house a square peg such as mortice and Tenon work. SEE DOGBONE FLYOUTS
- When the part is released from the parent sheet it may be dislodged or sucked off the table by either the spinning router cutter, or the dust extraction port. In any case a milling strategy has to make sure the pieces stay on the machine table. SEE TABS OR BRIDGES
- Small details that cannot be cut with a typical profile cutter of 10mm diameter. SEE PRE-MILLING

### Pocket milling on a flat board or sheet material (2.5D):

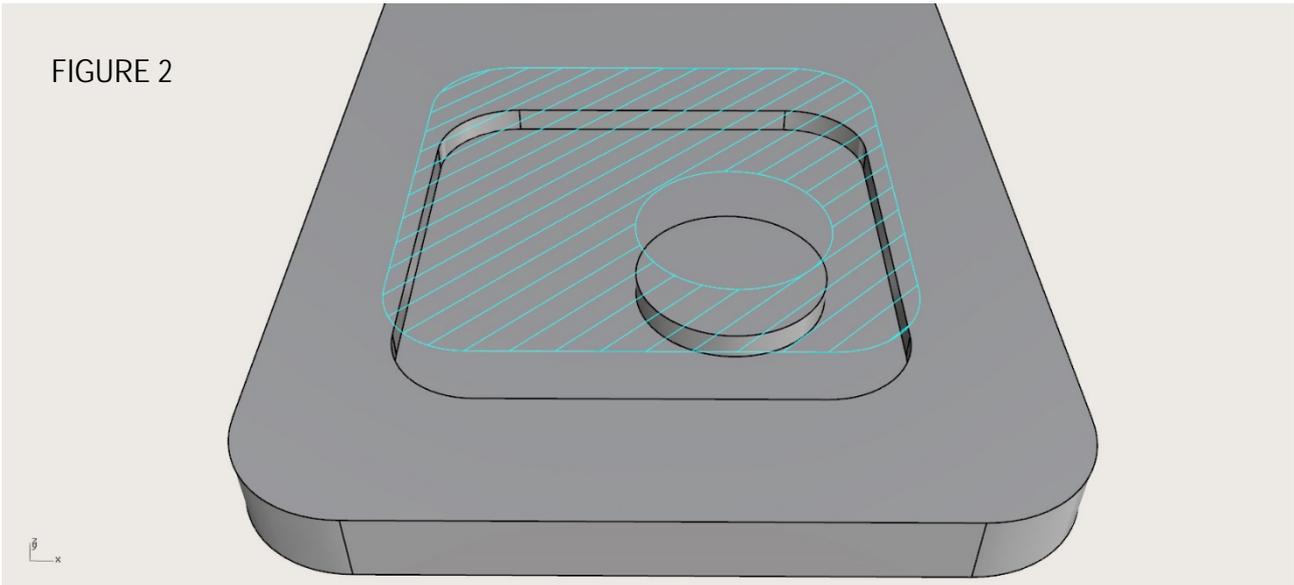
Understanding the process and its limitations.

Profile cutting will release a shape from the parent material; cut all the way through. Key limitations:

Pocket Milling will create a recess with vertical side walls and a flat bottom face within an area defined by a curve; (region curve). The hatched area will be milled. (Fig 2.)

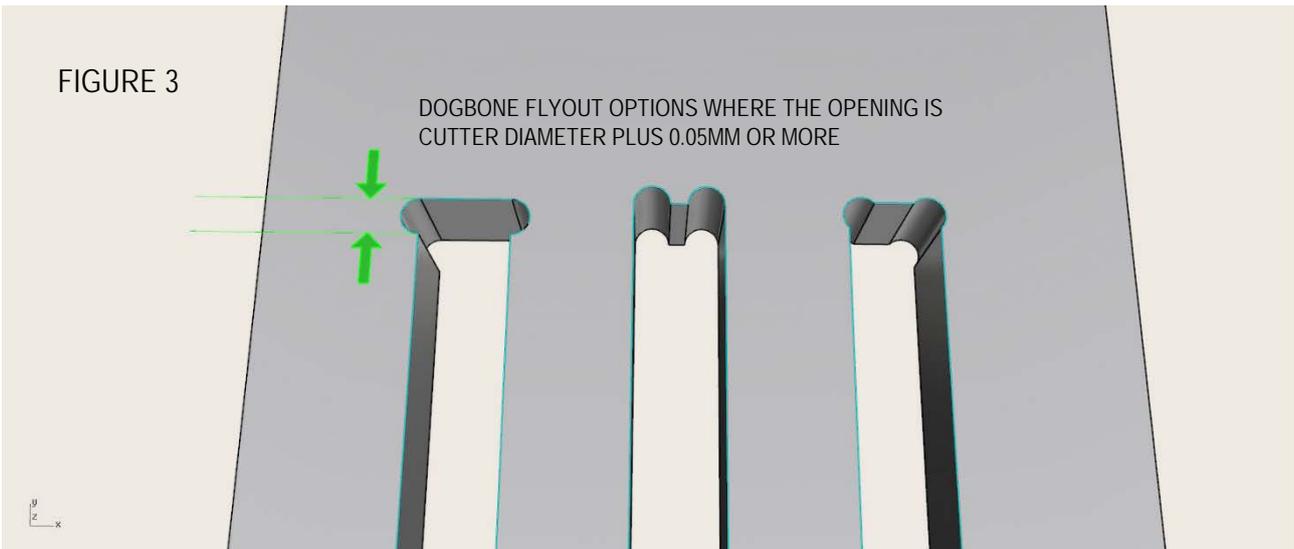
- Same inside corner issue as profile cutting, SEE DOGBONE FLYOUTS
- Small details that cannot be cut with a typical profile cutter of 10mm diameter. SEE PRE-MILLING

FIGURE 2



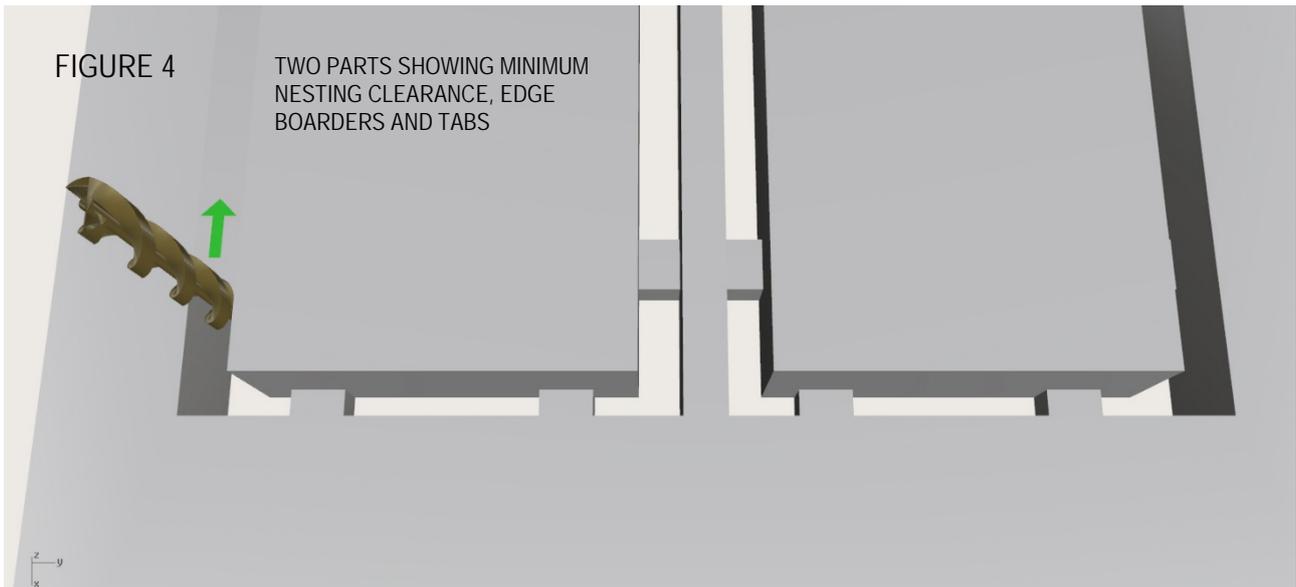
### Dog Bone Flyouts:

FIGURE 3



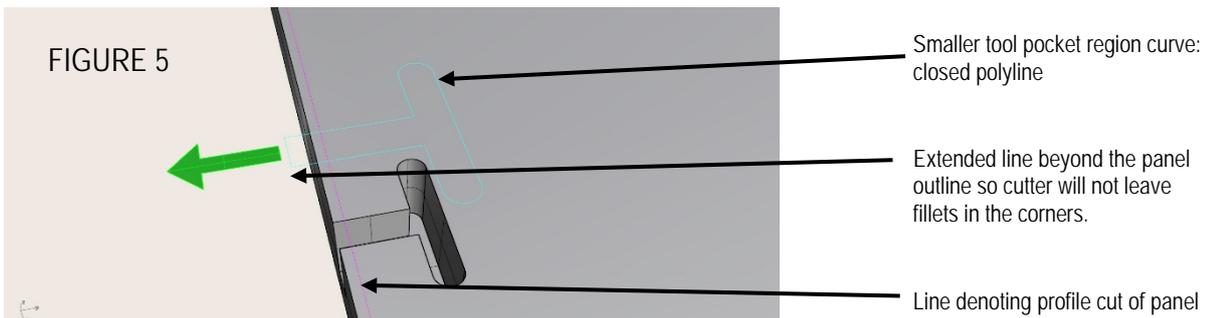
### Tabs (Bridges)

Profile cutting usually releases parts from sheet material. Anything with a surface area less than 1/4 of a full 1200 x 2400 sheet or less than 100mm width will need to be TABED. Tabbing is simply small bridges of material that are not cut and hold smaller parts into the parent sheet. These will be inserted for you when the job is processed but you will need to manually cut them through and smooth off the edge surface on your parts. For tabs to work on nested shapes leave at least three times the cutter diameter between all parts and the periphery of the sheet. Four times is preferred. ( Fig. 4)



### Drill and Pre-milling:

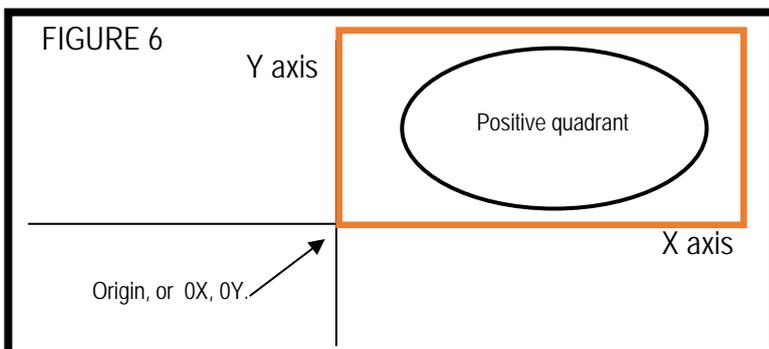
Small details and drilling holes with smaller diameter tools are normally performed before the process of pocket milling, then profile cutting. Where a smaller tool is required you can isolate that area with its own closed polyline, (region curve) or for drilling use a circle or point. Where a detail moves beyond a neighboring cut extend the region curve beyond (green arrow) so that fillets are not left at the intersection. (fig.5.)



### Setting up your drawing file so we can use it:

While the technical team can work with most common file types it would help us if you can send your work as either: **DXF, 3dm, Ai**. Please consult technical team if you must submit other file types.

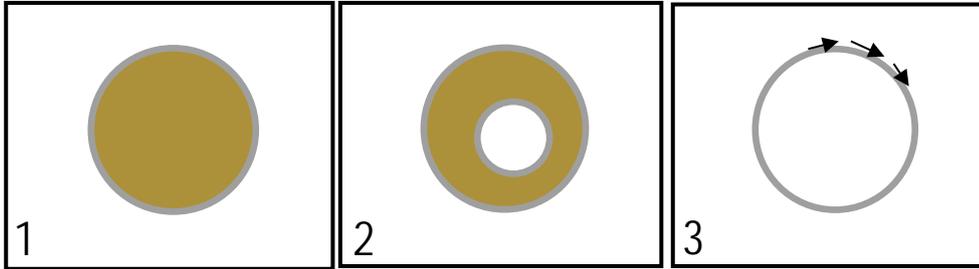
In the case of CNC routing it is preferable to work in the positive quadrant of the XY plane as this corresponds directly to the cutting table.



You will need to understand the dimensions of your material before you begin. Draw its size as a rectangle in the positive quadrant with the lower left corner at coordinate X0,Y0,Z0. If the long edge exceeds 1200mm it must orientated parallel to the Y axis. (fig.6)

### Default assumptions we place on your drawing:

1. Pocket milling will be inside a line unless stated otherwise.
2. Pocket milling will be between two lines if they are concentric.
3. Profile cutting will be around the outside of a line unless stated otherwise.



### File structure and naming:

A new file for each new piece of material or sheet named with this convention:

**First name\_project\_version\_sheet number.**

Version numbers: Sequential for submitted files. Sheet number: When more than one sheet in a job.

File Layers: please include a separate layer for following information:

1. All the outline curves for parts to be profiled, nested appropriately. Named: 'Profile'
2. The region curves for pocketing. Named 'Pocketing'
3. Drill points or circles Named 'drill'
4. Any regions for pre machining with a smaller tool. Named, 'Pre mach'
5. Annotations layer, including at least one critical dimension, pock depths, any text information you wish to convey, especially if the conventions above do not hold for your work.

### Drawing element structure:

1. All curves constructed from closed polylines on the X,Y, Cplane.
2. Be nested with appropriate tolerances as stated in Tabs and bridges.
3. Allow 0.05 to 0.1mm clearance for the cutting tool between tight curves. *Example: a 10mm pocket slot cut with a 10mm tool. The slot pocket will need to drawn at 10.05mm or wider.*
4. Design to the correct tool from our tool library. (see separate document). A default sheet processing tool is 10mm diameter or 3/8" (9.53mm).

### Tips:

When you want to cut intersecting parts from plywood it is useful to know the thickness of the ply exactly and model your parts accordingly. Measure your sheet don't rely on catalogue specification. Most reconstituted timber products will require at least 0.10mm to be taken from one face of the internal part for a push fit. It's a good starting place, but it is advisable to tolerance test materials first as they vary.