Sheet CAM Manual (R_3)

Startup:
Applications/Graphics/SheetCAM

Options/Application Options – UNITS tab – Set Linear units to - inches or mm
Angular units = Degrees
Feed Rate = Inches per Minute
Thread Pitch = TPI (This is not really used on the router)
Time = Seconds
Close SheetCAM and re-start

Options/Machine/…….. Machine Type = Rotary Cutting
Post Processor = LinuxCNC – Output File Units* - Z zero**
Working Envelope – Select Origin*** and enter Table Size (Max 46in X 46 in)
Table Display – Normally this would be set to match Working Envelope)

*If you change Output File Units, then you need to restart SheetCAM for it to take effect. (Your part will be the same size, whether output in inches or metric.)
** Always leave Z zero to 'Top of Work'
*** Usually it will be Center or Lower Left. ALL Origins must match – Machine/Working envelope & Table Display; Job/Material; and when importing a drawing

Options/Job Options - Material Tab –
Enter material description (Optional)
Choose Origin to match Machine Options
Set size of workpiece (in X and Y)
Set Material Thickness **Accurately** (use calipers to measure)
Rapid Clearance = .5 inches (this is how high the bit rises before performing a rapid move – to miss clamps, etc.)
Height of Material above Table – Must be greater than zero - .5 inches is a good number
Plunge Safety Clearance = .25 inches

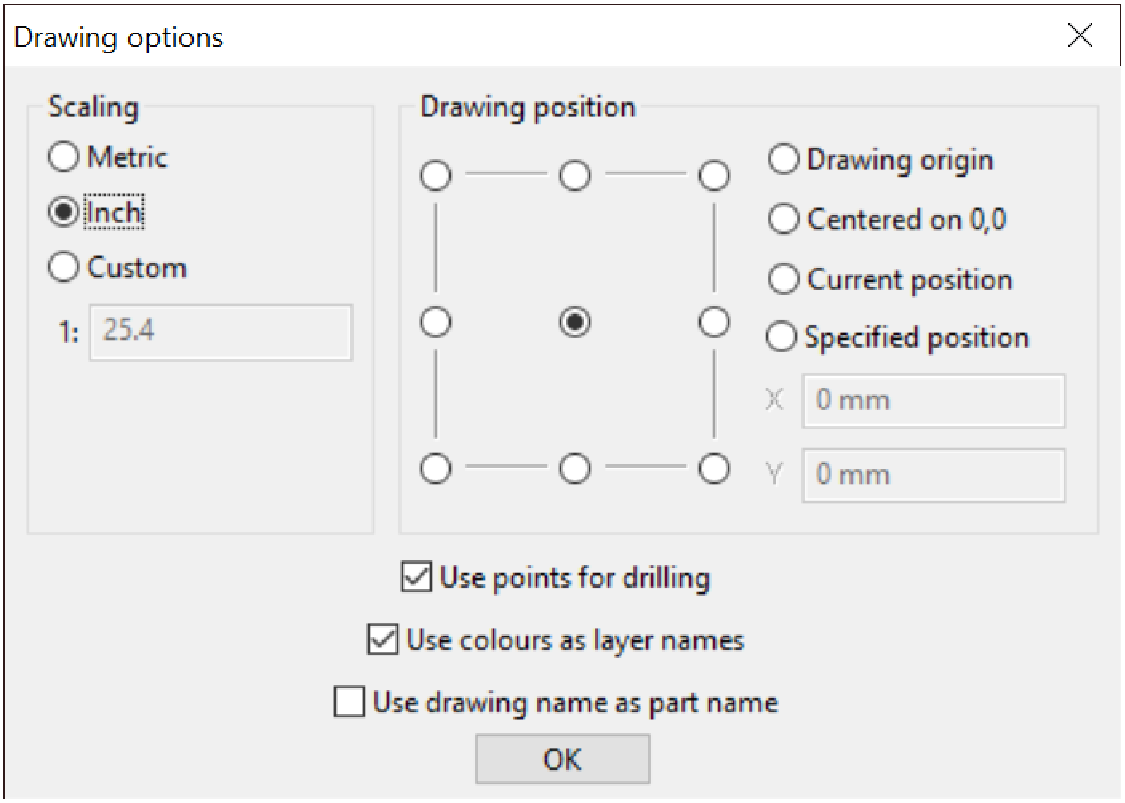
View Drop Down – Click to make sure Layer Tool is visible on the left side of the screen (You should see 4 'panes' on the left side of the screen)

There may or may not be entries in the 4 panes on the left of the screen
You do not need to delete these, just 'uncheck' the box to the left of the ones you will not be using (Layers & Operations) But deleting them could avoid some confusion..

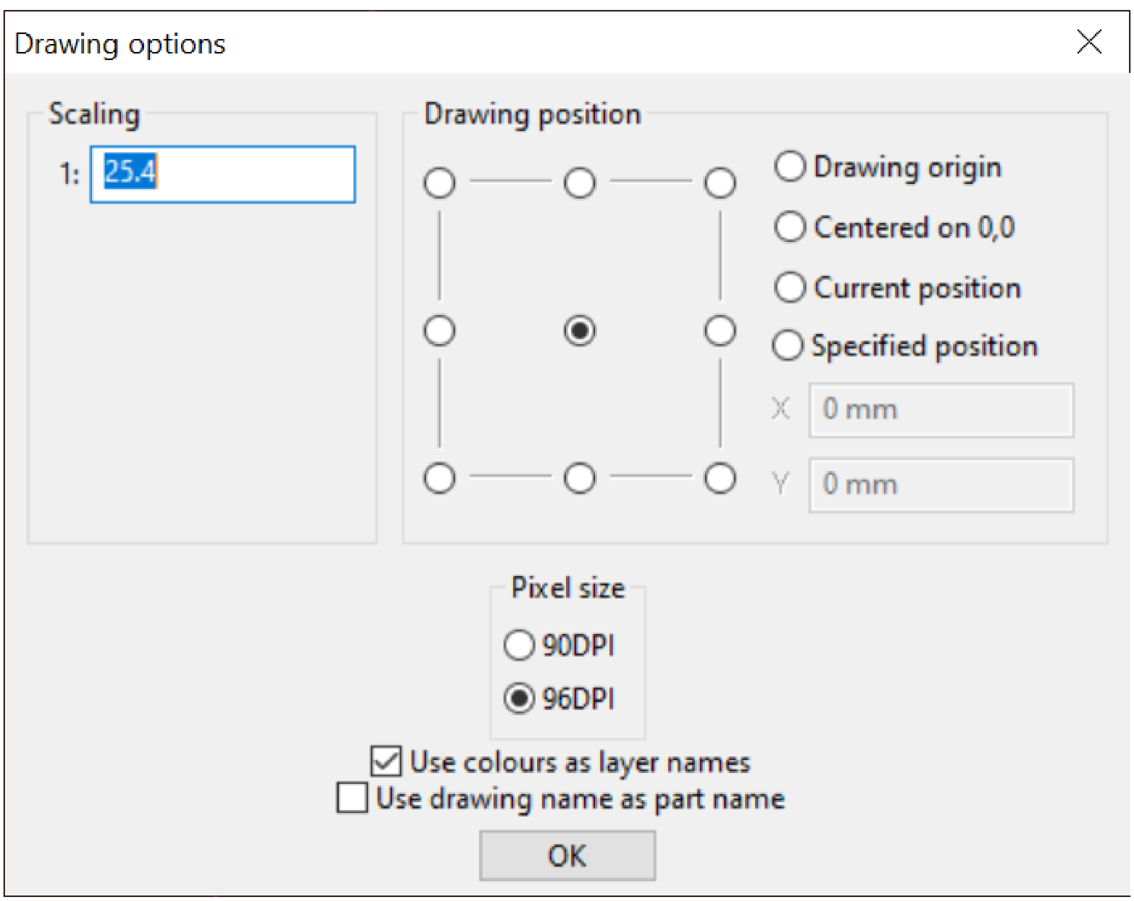
File/Import Drawing – You can import a .svg file or a .dxf file

Note: Be careful with drawing names. You are asked if you want to use the drawing name as the job name when you import a drawing. Look at the name that SheetCAM gives to the current job. If it is the same as a previous job that you want to save, then re-name it. Always use names that you can easily recognize.

.dxf Import:



.svg Import:



Important!: When importing a drawing, the origin of the workspace in the Machine and Job options must match each other ***and*** the Import window. Otherwise, the Gcode created will be 'offset' from where you expect it to be.

A note on scaling: SheetCAM assumes all drawings are in **mm**. It applies a 1:25.4 Scale to a .dxf file if you select **inches** in the Import Drawing window. (You are telling SheetCAM that you intended the drawing to be in **inches**.) For an .svg file you need to set the Scale to 1:25.4 yourself if you intended the drawing to be in **inches**.

.svg and .dxf drawings are 'unitless'. They have geometry described by numbers, but these numbers do not inherently have a 'size'. i.e. '23' does not mean '23 inches' or '23 mm', just '23 units'. You must tell SheetCAM what the '23' means – 'inches' or 'mm'. It then will scale the drawing to match the units of your current program.

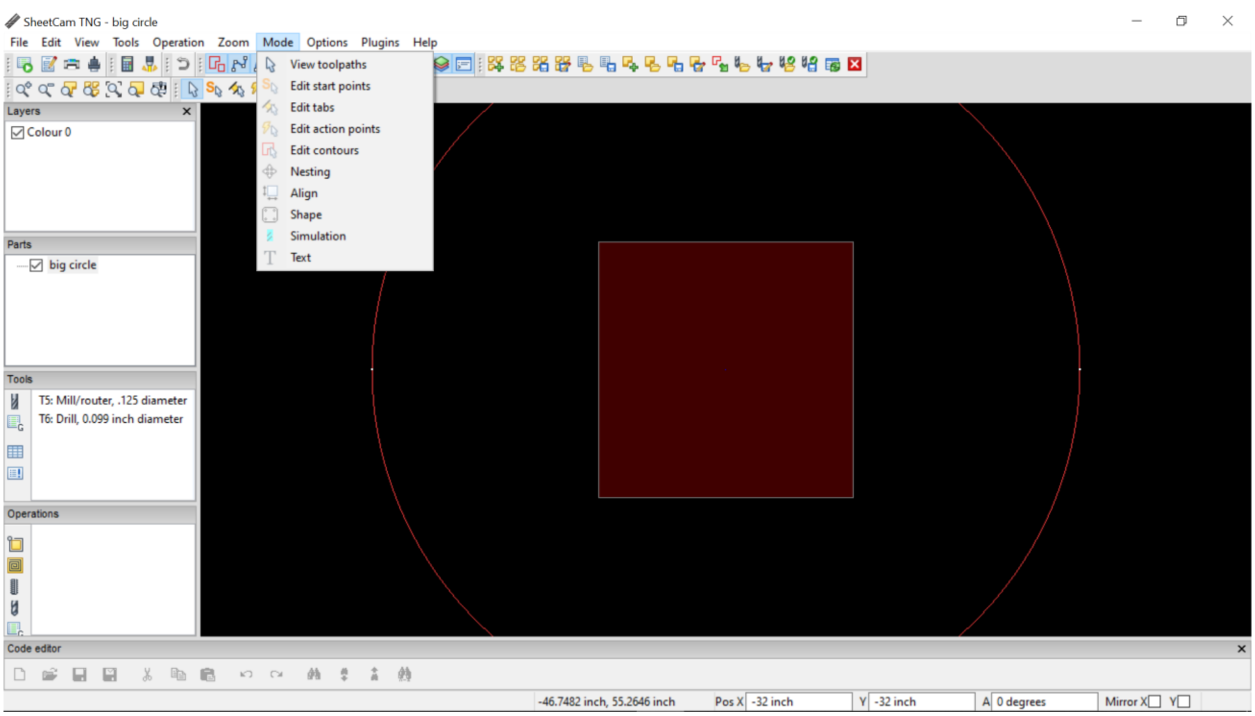
.svg file – Scaling - needs to be 1:25.4 if the drawing units were intended as **inches,** or 1:1 if the drawing units were intended to be **mm**.

Pixel Size – I leave it at 96 dpi
Use Colors as Layer Names – leave checked
Use File Name as Job Name – Usually OK to leave this checked

.dxf file – Scaling – Select 'Inch' if the drawing were intended as **inches**, or 'Metric' if the units were intended as **mm**.
Use Points for Drilling – Leave checked
Use Colors as Layer Names – leave checked
Use File Name as Job Name – Usually OK to leave this checked

In the end, you just need the drawing to 'look the right size' on the Working Table. If you use the 'Measure' feature (Right click and select MEASURE from the menu. Then click and drag to see measurements)

MODES



Modes determine what actions can be taken. I will cover the View Toolpaths, Edit Tabs, Edit Contours and Simulation Modes.

View Toolpaths – Toolpaths are shown. Rapid moves are shown in blue, 'feed speed' moves in green.

Edit Tabs – Here you place Tabs on a cut path (it must be highlighted in the Operations Window) by left clicking, and delete by hovering with the mouse and pressing the Delete key (or Right clicking and selecting Delete).

Edit Contours – In this Mode contours (either open or closed) can be moved and/or copied to new layers. Various choices for selecting contours will appear in the drop down menu.

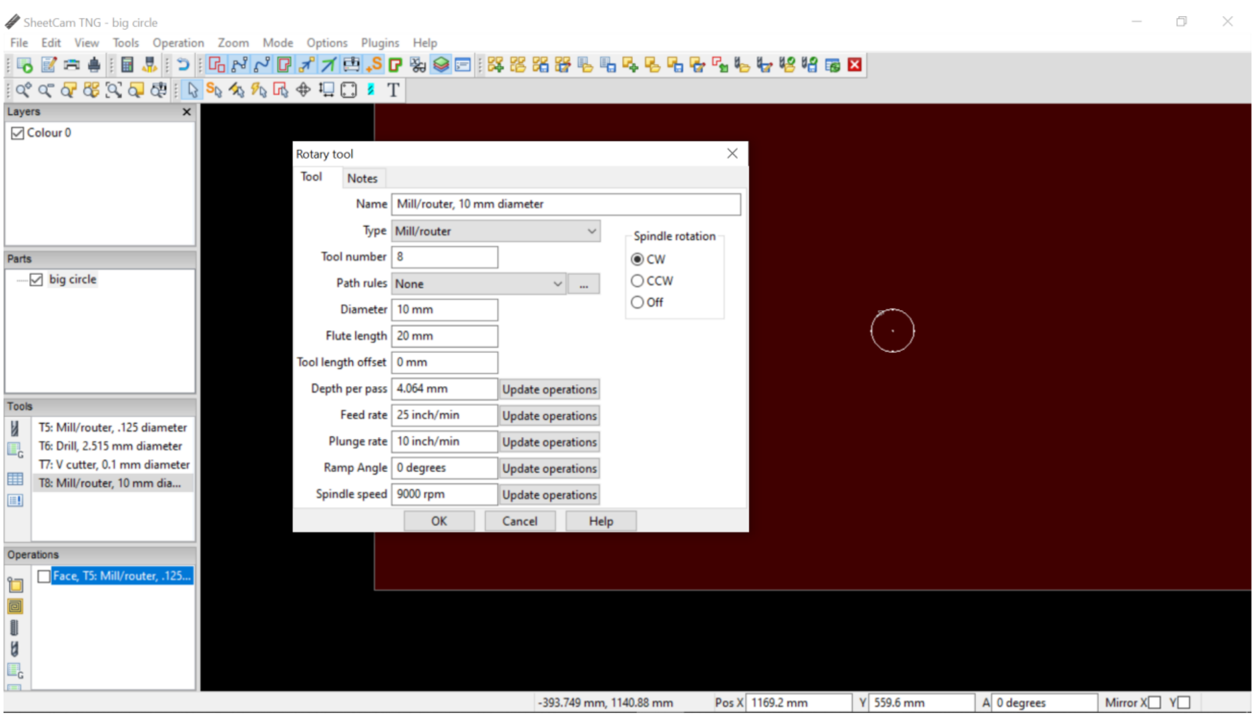
Simulation – In this Mode a simulation can be run, which shows a simulated tool following the toolpaths. A 3D view of the simulation can be seen by using Shift-Left Button (hold down the Shift Key and the left mouse button, the view will 'tilt' to show a 3D view of the Toolpaths. Double Click to return to a TOP VIEW.)

<u>Information Panes</u>

Layers Pane – This pane shows the layers that exist in the program. De-selecting a layer makes its contents invisible and prevents them from generating any G code. (If this pane is not visible, then go to the View drop down menu and click on Layer Tool)

Parts Pane – We do not use this pane, it is more for production programming.

Tools Pane – Tools are created here by clicking the milling cutter icon on the left of the pane. The router uses three tool types: Vbit, drill bit and square mill cutter.



Note: It is best to delete unwanted tools before creating new ones. The reason is to keep 'Tool Numbers' below about 8. LinuxCNC likes the lower numbers better.

Settings: Type – Select tool type

Tool Number – Give the lowest available number (You can type in an unused number)
Path Rules – skip
Diameter – 'Usually' enter the actual diameter of the bit – use '.125in' if you are in mm and using inch tool sizes. Use '3mm' if you are in inches and using metric tooling. SheetCAM will do the conversion.
Flute Length – This is the length of the cutting edge. (Flutes are needed to clear the chips as the cutter cuts them. If they are not cleared, they will get rubbed into dust and heat up the work and the tooling. Never cut deeper than the flutes on a cutter.)

Tool Length Offset – This is for use with a tool changer. We don't use it.

Depth per Pass – A conservative setting is the diameter of the tool, i.e. .25 for a ¼" dia. Tool. Do not go more than 2x the tool diameter.

Feed Rate – Using the formula below, try for a chip thickness of .001 to .004. 30 inches per minute is a 'safe' feed rate. You could go 40 or 50, but cut quality might suffer.
Plunge Rate – 15 inches/minute is 'safe'

Ramp Angle – This is **always** zero, for our purposes.

Spindle Speed – This is one of the factors in the formula below. It is also discussed in the article 'The Zen of CNC'. 8,000 to 12,000 is almost always 'safe'. I use 10,000 a lot.

Spindle Rotation is always CW.

FEEDS and SPEEDS (Otherwise known as 'Chip Thickness')

Discussion: Chips remove heat from the workpiece and tooling. Dust does not. The thicker the chip the better – up to a point. Taking a thicker chip will give a rougher cut, so there is a practical limit to chip thickness. It is about 0.004". This number represents 250 cuts per inch of travel. See the following formula:

Feed Speed/#Flutes x Spindle Speed = Chip thickness

Feed Speed = in/min
divided by
Cuts(flutes) = chips/revolution X rev/min
Equals
in/chip

40/2x10,000 = 0.002 (2 flute cutter at 40 ipm and 10,000 rpm)

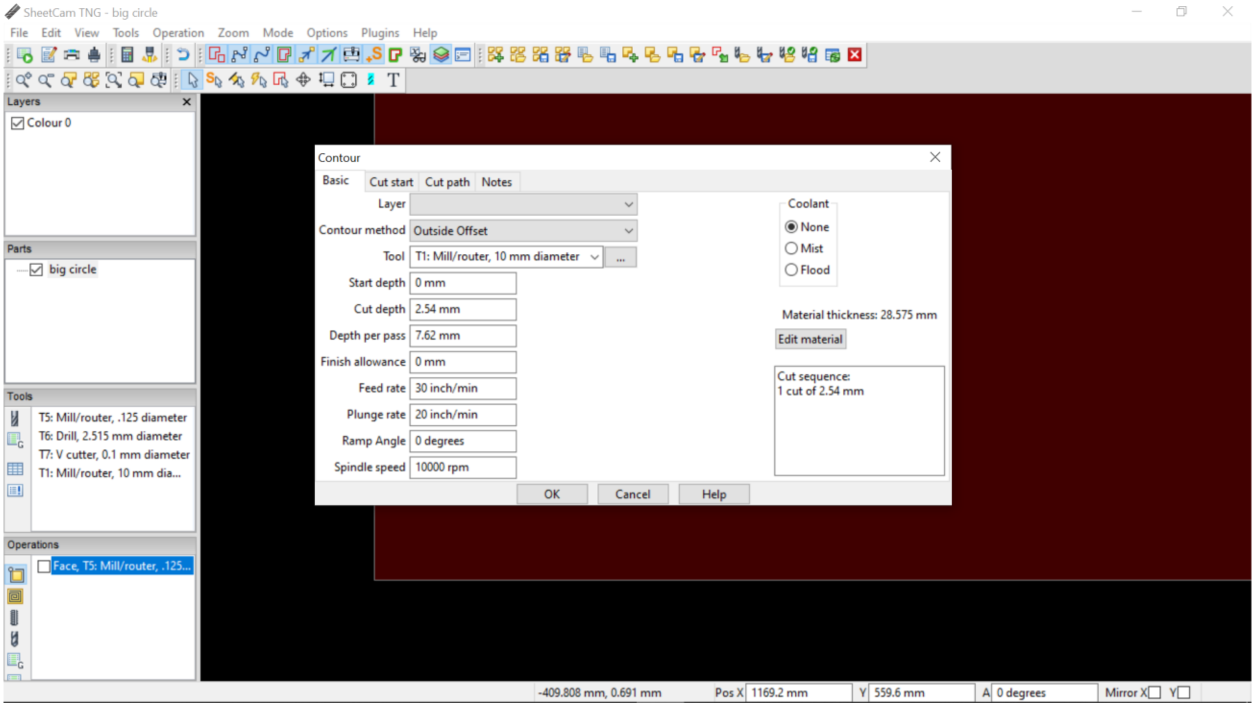
30/1X10,000 =0.0033 (single flute cutter at 30 ipm and 10,000 rpm)

Caution: Each time a tool is used in a new operation, even though all of the tool parameters were set when the tool was created, they **all** must be checked. If a parameter is overridden in a particular operation, it might affect that parameter the next time that tool is used.

OPERATIONS

Three types of operations are used with the CNC router: Contour, Pocket and Drill. To see the effect of a contour or pocket operation, toggle the Show True Width button on the top Icon Ribbon or select 'Show True Width' from the View drop down menu.

CONTOUR OPERATION



Layer – Choose the layer that has the contours you want to cut. A contour is used to create a tool path, according to the Method chosen (See Contour Method).

Contour Method – Choose the Offset for the cutter. Outside moves the center of the cutter outside of the closed contour by the radius of the cutter. Inside Offset moves it the radius to the inside of the closed contour. No Offset leaves the center of the cutter on the contour.

Note: A contour must be closed in order to apply an offset. If you think the contour should be closed, but SheetCAM thinks it is not, you can try re-importing your drawing. Before re-importing, open Application Options/Drawing Import and increase the Import Link Tolerance to about .004. This will 'bridge' larger gaps in the contour to make it a closed contour. If that doesn't work, then you need to go back to your drawing program, fix the drawing and save it. When you re-import it, you will need to recreate the layers (tools and operations will remain from your first attempt).

Another problem is 'duplicate lines'. Usually SheetCAM will remove the duplicates and notify you, but sometimes there will still be duplicates. They cause the most problems when they share one or two end points.

Tool – select the tool you want from the drop down list

Start Depth – This is normally zero

Cut Depth – Total depth of the cut you want to make (if making a through cut, try to limit it to .020" or .5mm deeper than your material thickness. This will prevent unnecessary 'chewing up' of the spoil board)

Depth per Pass – A 'safe' number is the diameter of the tool

Finish Allowance – This will leave the amount you enter for a final 'finishing' pass (of the Depth)

Feed Rate – 30 to 50 inches per minute

Plunge Rate – 15 in/min is 'safe'

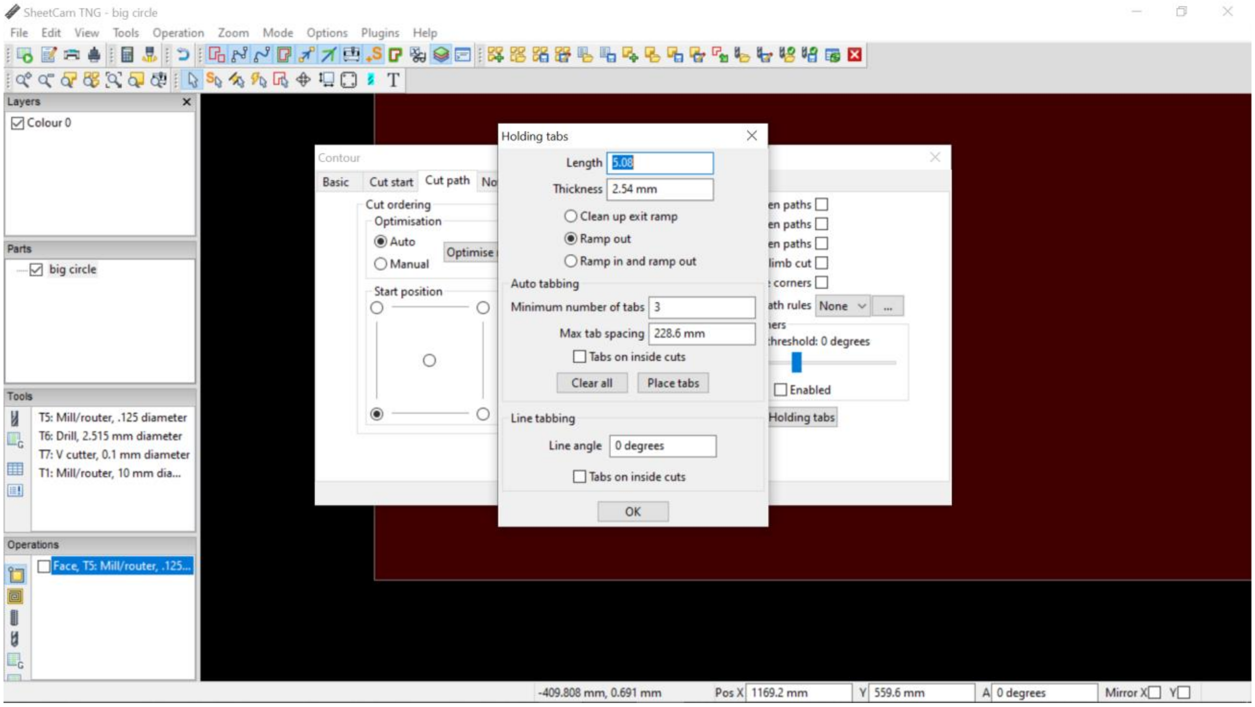
Ramp Angle – Always leave this as zero

Spindle Speed – 8K to 12K

Note1: Pictures appear to the right of each parameter as you enter it to show what that parameter affects.

Note2: Depth per Pass, Feed Rate, Plunge Rate and Spindle Speed can be adjusted by clicking the UPDATE button within the tool definition, but ALWAYS CHECK - EACH OPERATION - to make sure the values are what you expect.

HOLDING TABS

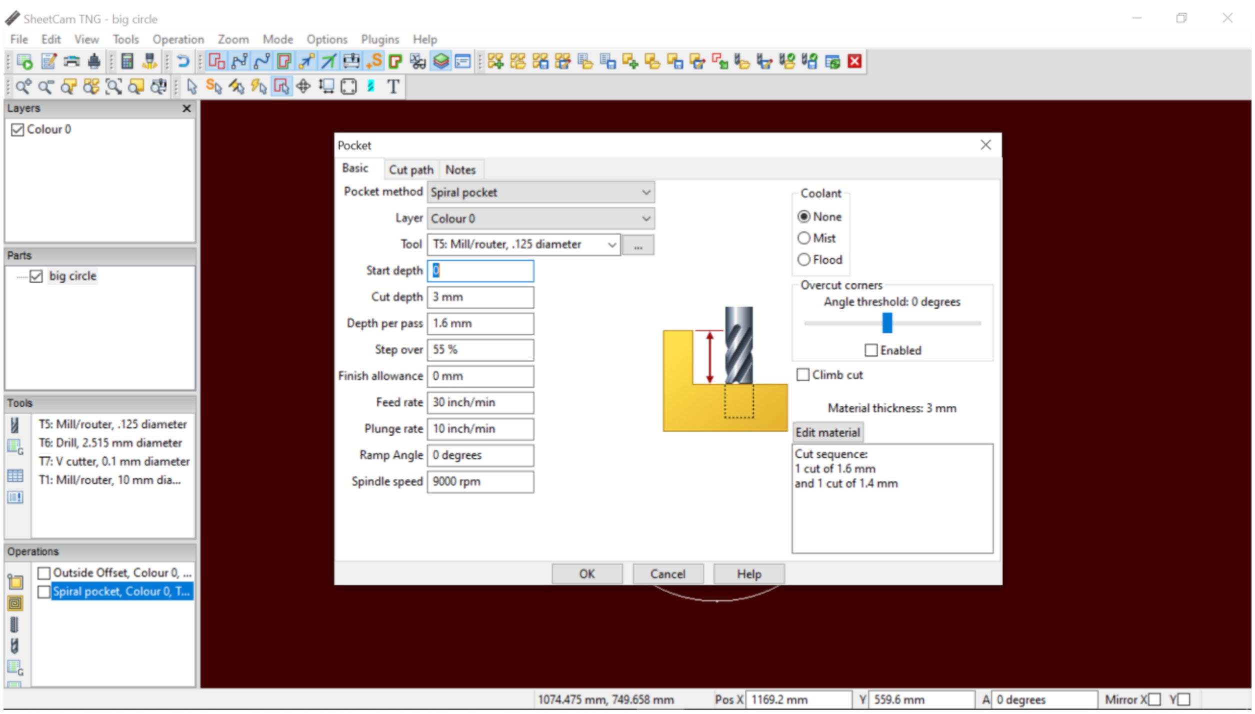


Holding Tabs keep your parts attached to the parent material when cutting all the way through, so the part does not move after it is cut through. They can be placed two ways – 'Automatically', by clicking the Cut Path tab in a contour operation or by 'manually' placing them in the Edit Tabs Mode. Note: Tabs will not be placed if your contour cut is not deeper than the material thickness (adjust material thickness in Options/Job Options).

'Automatic' – In the Holding Tabs dialog box, set the desired Length(+/- .2) and Thickness(+/- .1); leave Ramp Out checked; Minimum Number of Tabs and Min Spacing are OK (since you will be moving the tabs after they are 'automatically' placed); THEN click on Place Tabs and click OK to exit the dialog and OK to exit the contour operation.

'Manual' – Using the Mode menu drop down, click on Edit Tabs; click on the desired operation; hover to see the blue 'blob' indicating a Tab and left click to place it. Hover and press the Delete key or right click and select Delete to remove a Tab

POCKET OPERATION

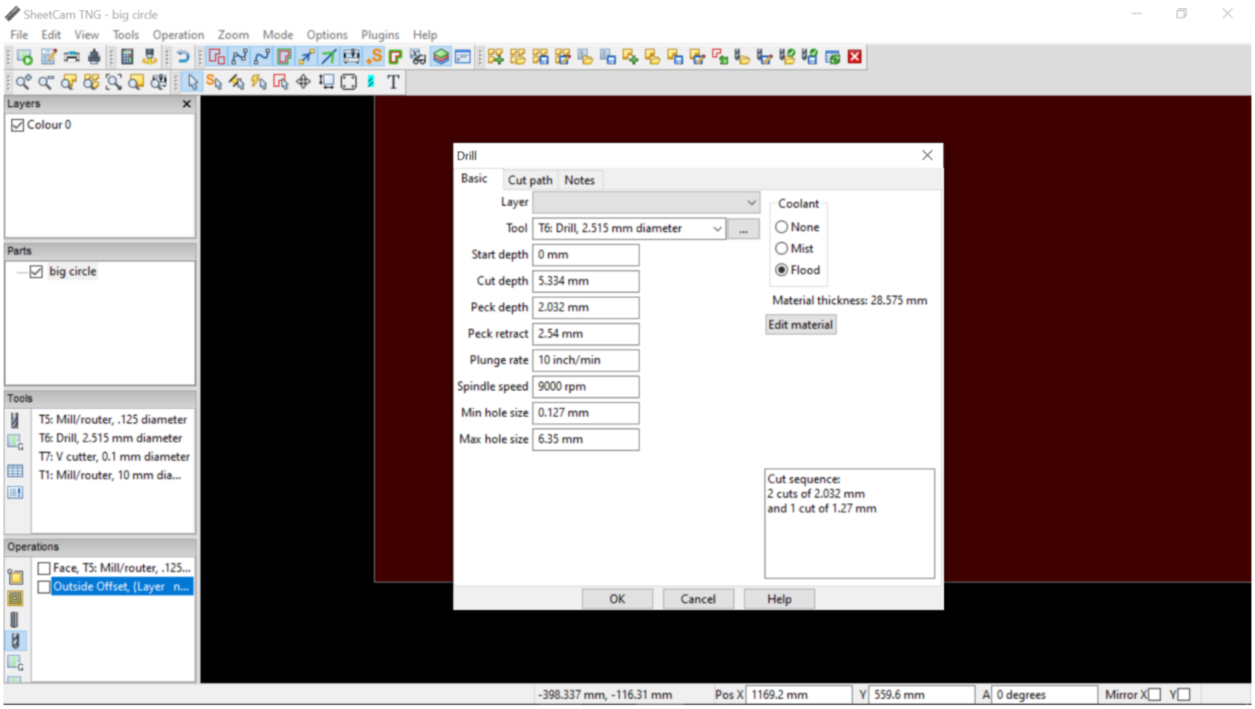


The Pocket Operation creates an overlapping tool path that clears the inside of a closed contour, usually starting at the center. If the overlap (Step Over %) is set too high, then ‘islands’ of uncut material will be created (you will see red spots in the green if ‘Show True Width’ is active). Some patterns need a Step Over as low as 50(%). Try starting at 80 and working down until no ‘islands’ remain. (Use ‘Show True Width’ to reveal uncut ‘islands’). Note: if you use a step over greater than 50% then you will get ‘strips’ of wood peeled off by the router bit. This should not affect the final finish of you work. If it does then set step over below 50%.

Note: I almost always use ‘Spiral pocket’. ‘Zigzag’ might do better with a regular, rectangular pocket shape.

‘Step Over % ’ is the only new parameter. It is explained above.

DRILL OPERATION



The drilling operation is accomplished in a series of partial depth ‘pecks’ with a ‘retract’ following each ‘peck’. The ‘retract’ is meant to help clear the chips from the hole. You can draw a circle the size of the hole you want and place these where desired (this is done in your drawing program). SheetCAM will find the circles and designate them for drilling IF it decides that they fit between the Minimum and Maximum Hole Size in the Drill Operation dialog box. You sometimes need to play with the Min./Max. size parameters to get Sheet CAM to recognize the holes for drilling.

Use caution when drilling with a milling cutter!! Set Plunge Rate to 5 and ‘Peck Retract to 3X the ‘Peck Depth’. Otherwise, you could start a fire. (Chips get trapped in the hole and ground into powder. This retains heat which can ignite the powder. MDF is especially prone to fires.)

Tool – select the tool you want from the drop down list (it will normally be a mill/cutter)
Start Depth – This is normally zero
Cut Depth – Total depth of the hole
Peck Depth– A ‘safe’ number is the diameter of the tool
Peck Retract – set to (at least) 3X the peck depth
Plunge Rate – 5 in/min is ‘safe’
Spindle Speed – **3,000**
Min Hole Size – Set to just smaller than the cutter you will use
Max Hole Size – Set just larger than cutter.

FILE/ Save Job As

Before running the post processor, you may want to save your job. This will allow you to open it at a later time and make changes. Select the location, give it a recognizable name and click 'Save'.

FILE/ Run Post Processor

Make sure all operations that you intend to use are checked, then Select File/Run Post Processor. Select the destination you want and give the file a name you will recognize. The file type is always '.tap'.

Click 'Save'. You will get a message window showing all of the operations that were performed and if they were or were not successful.

Open a Saved Job

When you open a Saved Job, you need to check all of the choices in
- Options/Machine:
  - Make sure the Machine Type is correct
  - Make sure the PostProcessor is correct
  - Check the Working Envelope and Table display for Origin and Dimensions
- And Options/Job
  - Check Origin and ALL Dimensions

Layers, Tools and Operations should all be restored

Check that TABS were restored to Operations that had them.

Final Thoughts –

Programming is an iterative process. It may take several trips from the router and back to SheetCAM to get the desired result, so have extra material on hand, or run the program on a scrap piece first. For a complex or new program, you can 'run in air' first – just set the 'Z' axis to an inch or so above the material and run the program. This will show if the location on the work piece or some feed speed or depth of cut is way off.

It is always the one setting that you forget to check that messes up your program. 'Experience' is your best teacher, but unfortunately, we are always learning.

SheetCAM has way more features than we have covered here. These are the basics, which will cover 90% of what you will do on the router. Many of SheetCAM's features are intended for 'production' router work.