Collision Detector

ver 1.3.5 David benson
2nd September 2015

Requirements

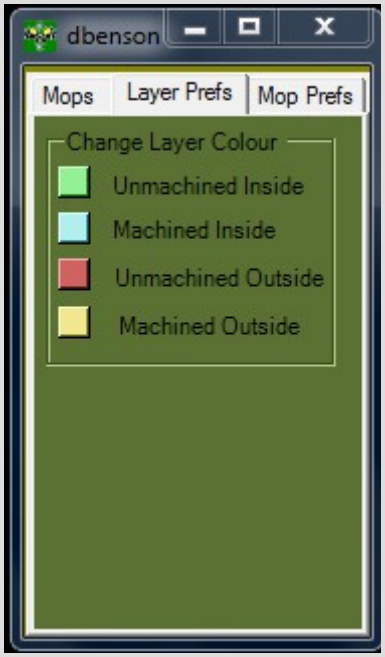
CamBam plus [0.9] Rel 8P (beta-2)
Copyright © 2013 HexRay Ltd
CamBam.CAD : 0.9.5035.22791
CamBam : 0.9.5035.22792
Build Date : 14/10/2013 1:39:42 PM
Framework Version : 2.0.50727.5485

The collision detector is a plugin for CamBam and is designed to speed up the generation of a cad file to be used on cnc machines.
It does this in several ways, Firstly it gives a visual representation of the areas in a file that will be cut with the currently selected cutter and importantly those that cannot be cut.
There are three .cb files included in the zip file which you can load up and follow along with the tutorial.
The .dll file should be placed in your CamBam plugin's directory.
To do this in a easy to understand manner:
Four new layers are inserted into your CamBam file.
They are in order:

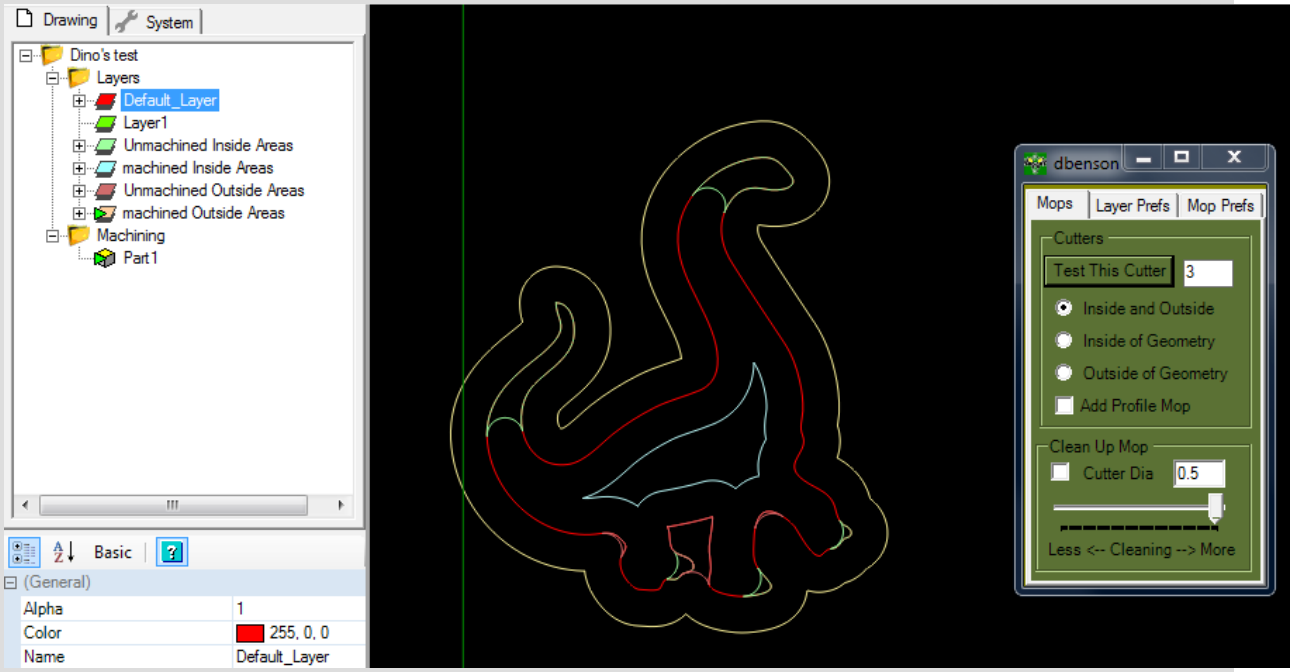
1. Unmachined Inside Areas
2. Machined Inside Areas
3. Unmachined Outside Areas
4. Machined Outside Areas

Layer Unmachined Inside Areas:
Displays the areas on the Inside of your currently selected Shape that can not be cut with the presently selected cutter.
Layer Machined Inside Areas:
Displays the areas on the Inside of your currently selected Shape that can be cut with the presently selected cutter.
Layer Unmachined Outside Areas:
Displays the areas on the Outside of your currently selected Shape that can not be cut with the presently selected cutter.
Layer Machined Outside Areas
Displays the areas on the Outside of your currently selected Shape that can not be cut with the presently selected cutter.

There is a Tab on the Collision Detector Form (Layer Prefs) where you can set the Layer colours to your own Preference.
See below:

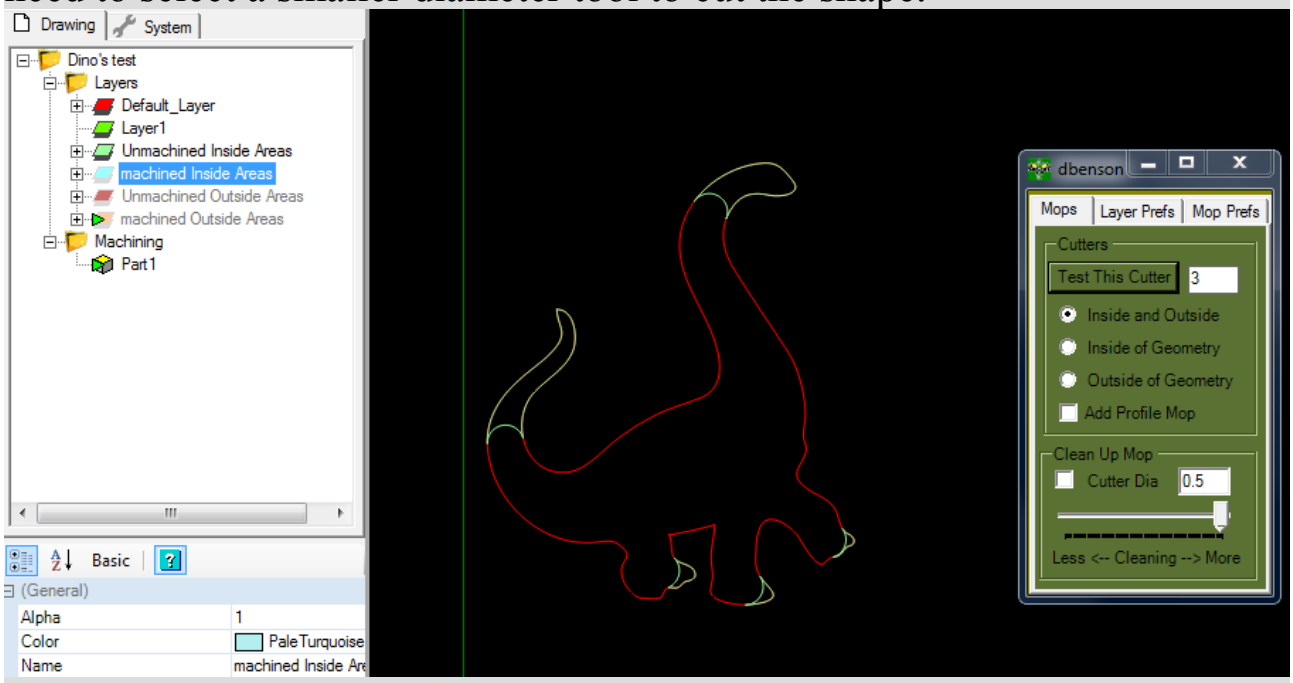


Just click on the coloured square beside each layer name and a colour dialogue will pop up allowing you set the colour.
Load up the Dino's .CB file to follow along with the tutorial.
Ok so lets have look at what the layers look like on the screen with some sample data and of course you can display and hide the layers by simply clicking on any Layer in the tree view and pressing the space bar.



So what does all this mean and how can it help you, Well in the figure below all of the layers have been turned off except the Unmachined inside layer

What this shows you, are the areas on the inside of the shape that cannot be cut because your selected tool diameter is too large, and you would need to select a smaller diameter tool to cut the shape.



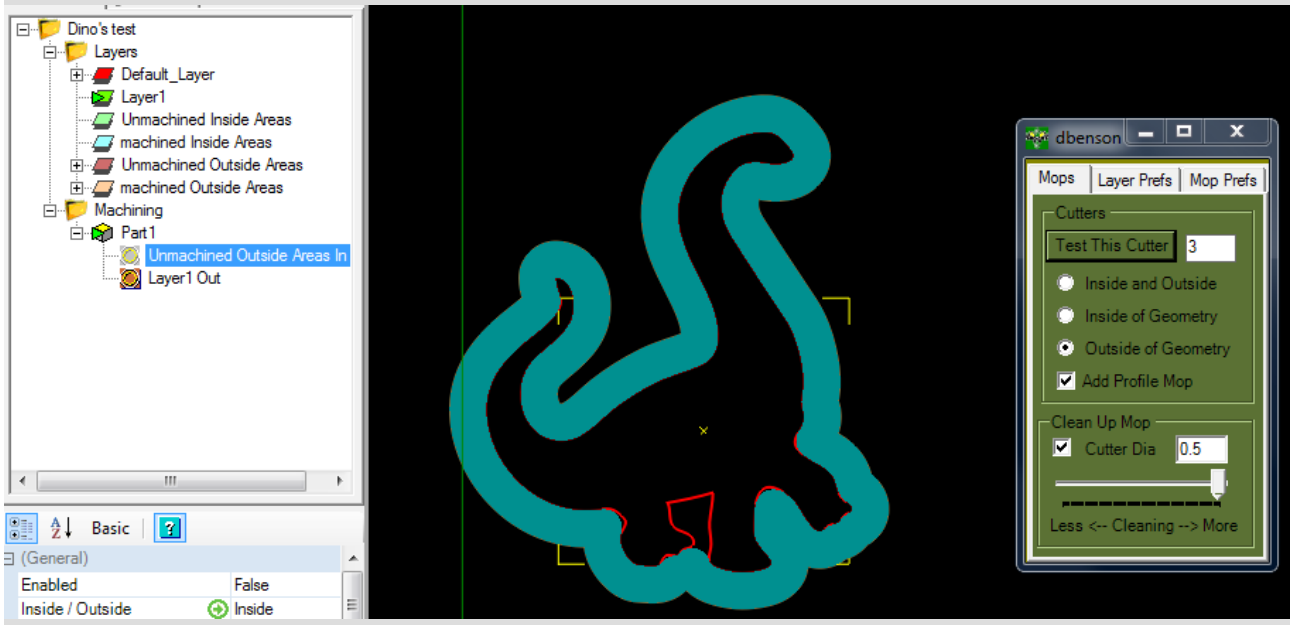
In the figure below all of the layers have been turned off except the Unmachined outside layer

What this shows you, are the areas on the outside of the shape that cannot be cut because your selected tool diameter is too large.and you would need to select a smaller diameter tool to cut the shape.

So I here you say what is the difference between this, and selecting the shape and applying a profile mop and selecting “show cut widths” from the menu.
Well if you have complex geometry or a lot of it, It's easy to miss small areas that can't be cut especially if there in the sub millimetre range.
And this method will show any uncut areas no matter how small and put them on a separate layer so there easy to see.

Cutting the shape out

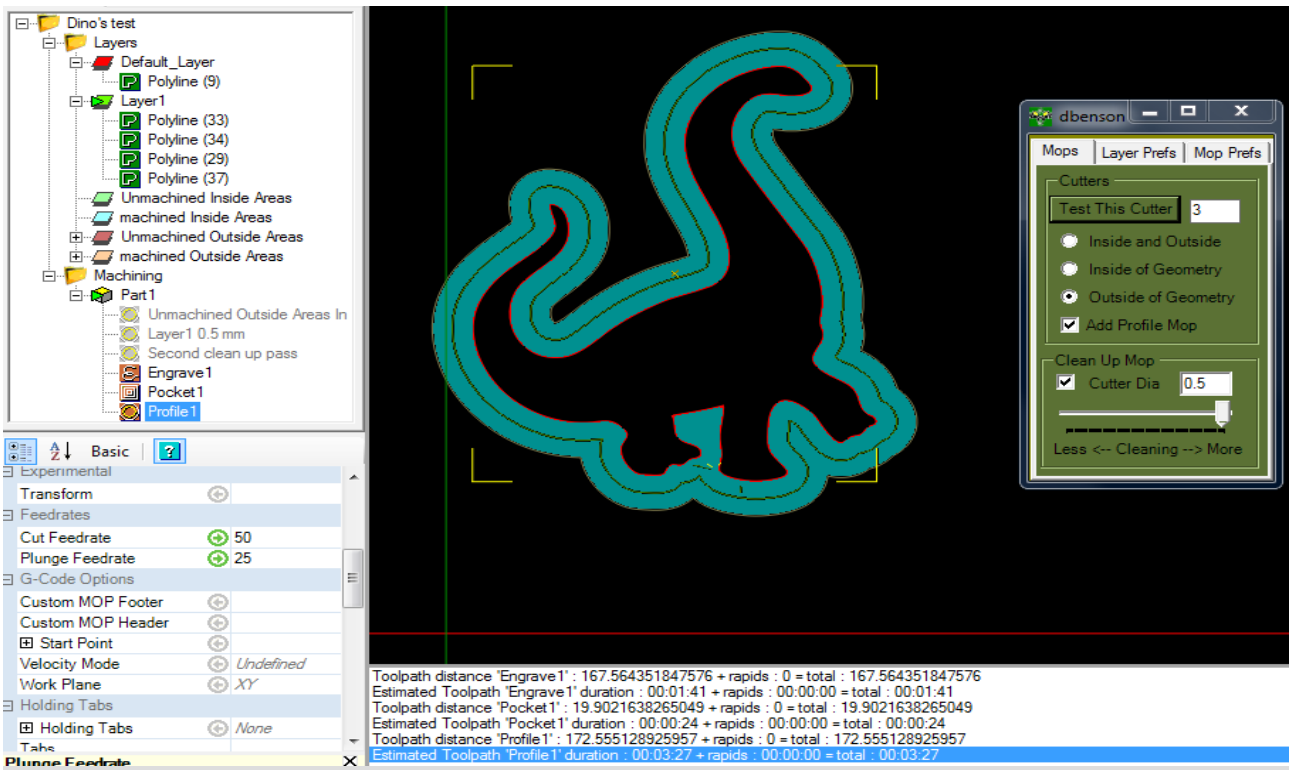
The method I used to cut the shape before the collision detector was to select a tool with a diameter small enough to cut all of the detail out but often this was a small tool and I had use use low feed rates to suit the cutter and this always took a long time, so to improve the cutting time I would select a larger cutter (with which I could use higher feed rates) and then use a mop with a smaller cutter and go around the shape again But this time scrolling through the code to the pieces of uncut material and selecting (on Mach3) the “run from here” option.
This was quicker but you had to be at the machine and was error prone.
So how can the collision detector help!
Well if you select the Add Mop option and the Add Clean up mop option two new mops will be added to your file.
Now if you go to the menu and select generate tool paths and hide the Unmachined outside layers Layer you will see a screen like the figure below



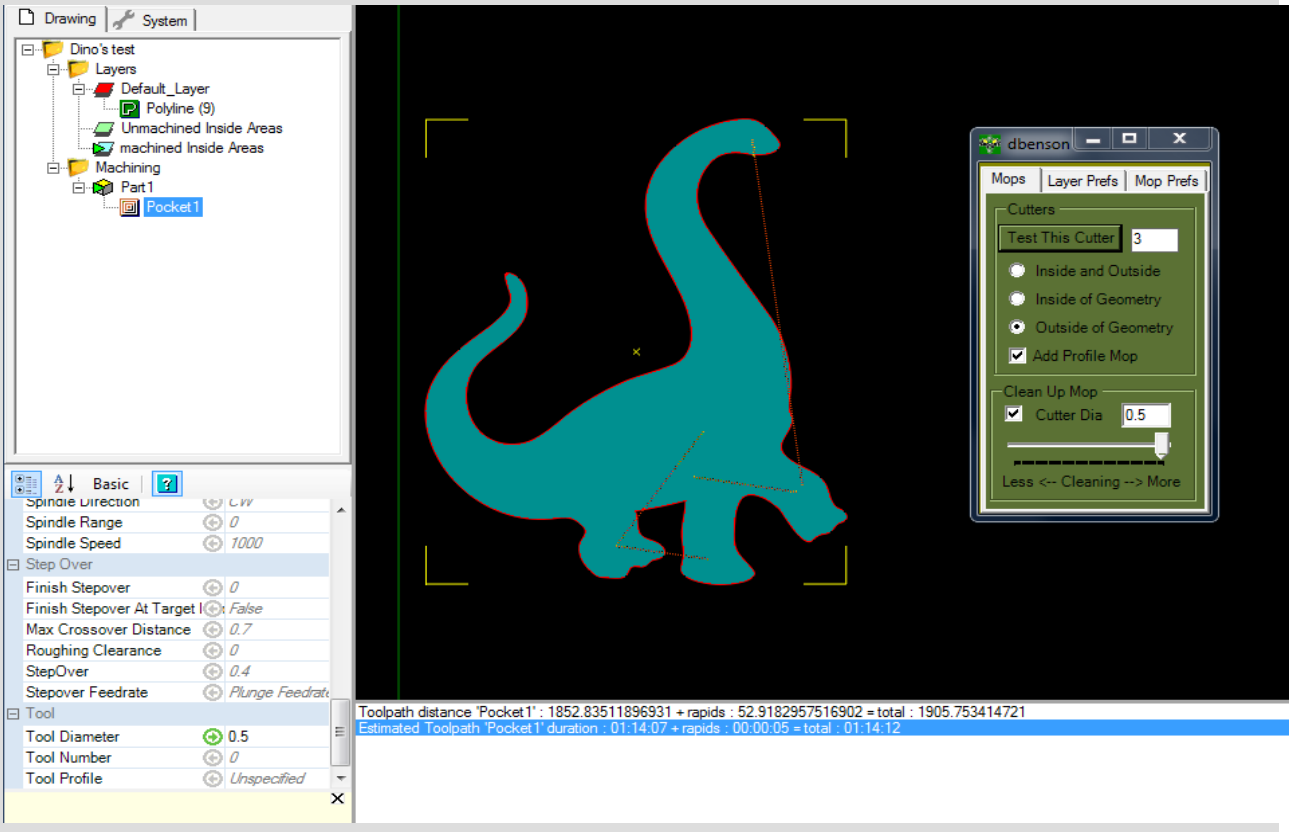
where you can see the larger diameter cutting path and the areas that can't be cut with it.

Now if you show the Unmachined Outside Layers and hide Layer 1 out you can see that tool paths have been generated for the uncut parts of your shape.

So now your cutting time will be dramatically shortened compared to selecting a mop for the shape and travelling all the way around the perimeter.
As you can see in the figure below there is still one area between dino's legs that is left uncut you have three choices either select another larger tool size in the Clean up mop at the loss of some cleaning ability.
As I wanted to cut the shape as accurately as possible I chose another method where I turned all the layers off but the Unmachined outside layer selected the tool path and the copied and pasted another Clean-up mop and associated only with that one tool path. The other method is to select the tool path join it up with a poly-line and use a pocket mop.
You may choose the method that suits the shape you are working with.
Below are the two method I tried.

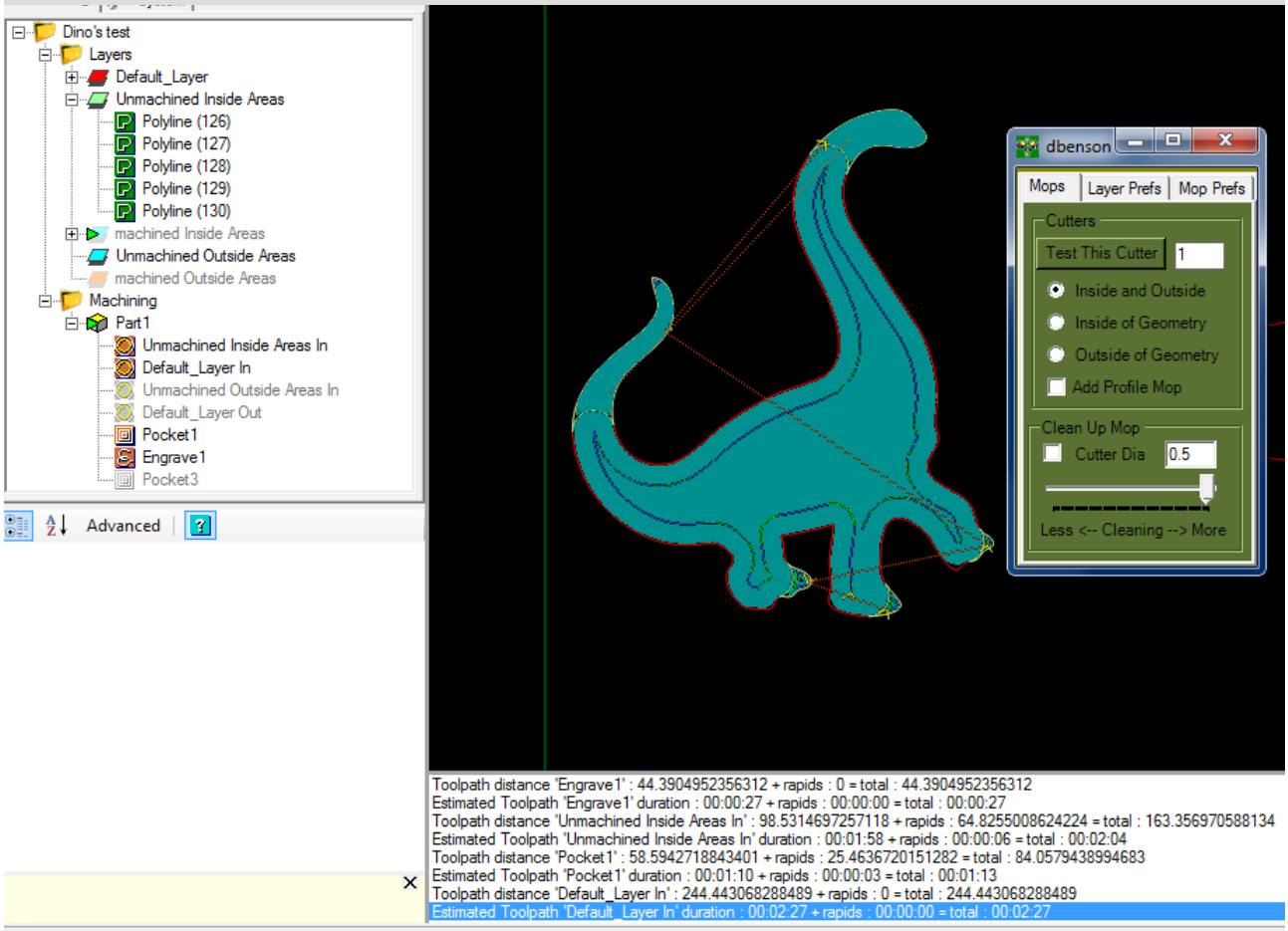


So how much time did we save by using the Collision Detector
In this case 50 % and this is largely dependant on the length of the clean up pass.
So what about pocketing the shape out for an inlay, are there any gains to be had here.



So about 1 hour and 14 minutes to do the pocket if you have selected a tool that will cut out all the detail and just produce a pocket.

Of course I never do this and try to optimise the cutting time whenever I can by using a larger tool and drawing regions in the selected shape and some times doing an offset and engraving that as well, but this can get tricky and time consuming depending on the size and shape of the object.

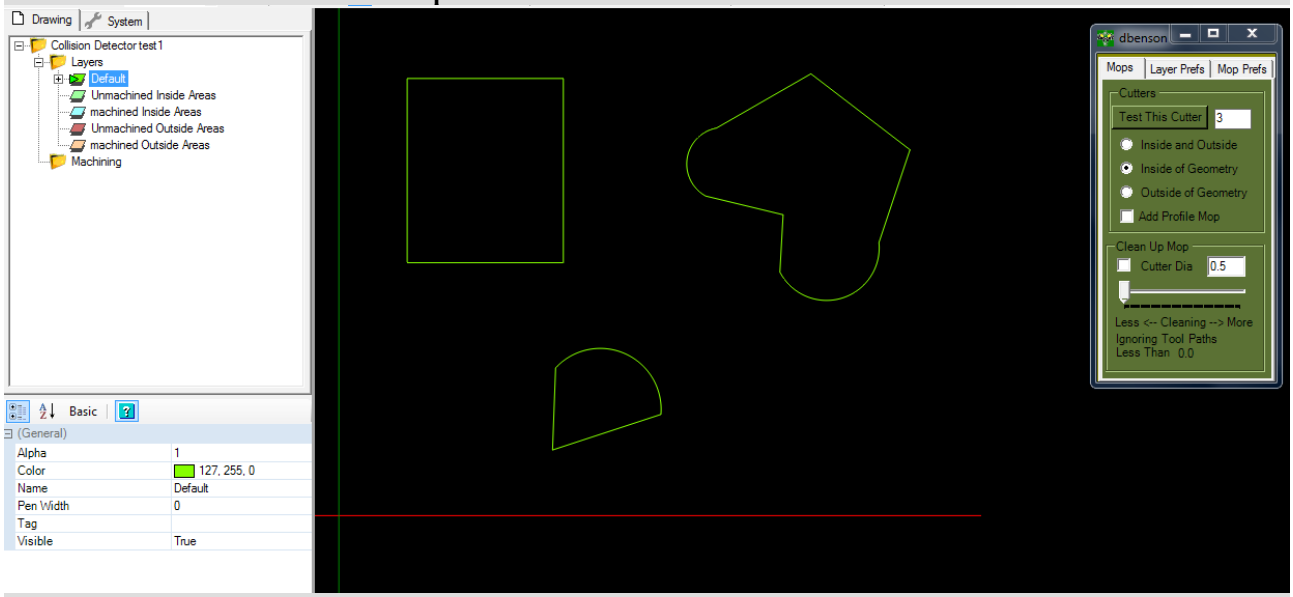


And so after a bit of playing around the cutting time is reduced to about 6 minutes of course you can get good results by hand editing the file producing your own tool paths but this takes a bit of fiddling whereas the plugin does this legwork for you

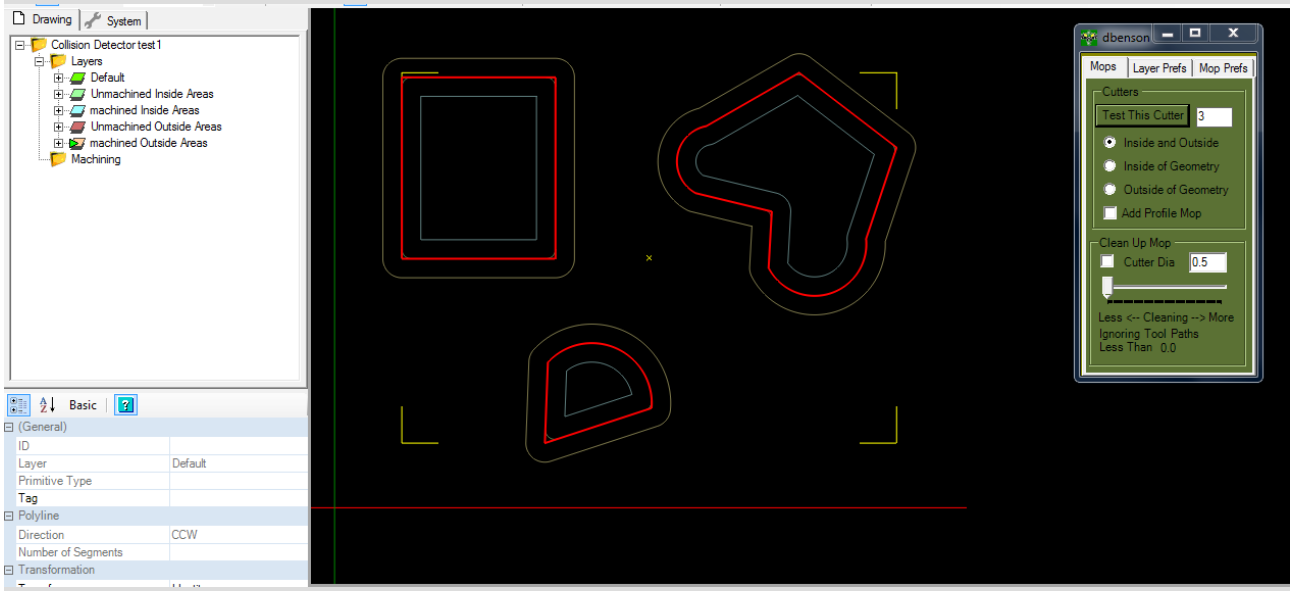
Load up the Collision Detector test1.cb

OK so lets move on to doing multiple shapes, you can follow along with the Collision Detectortest1.cb file.

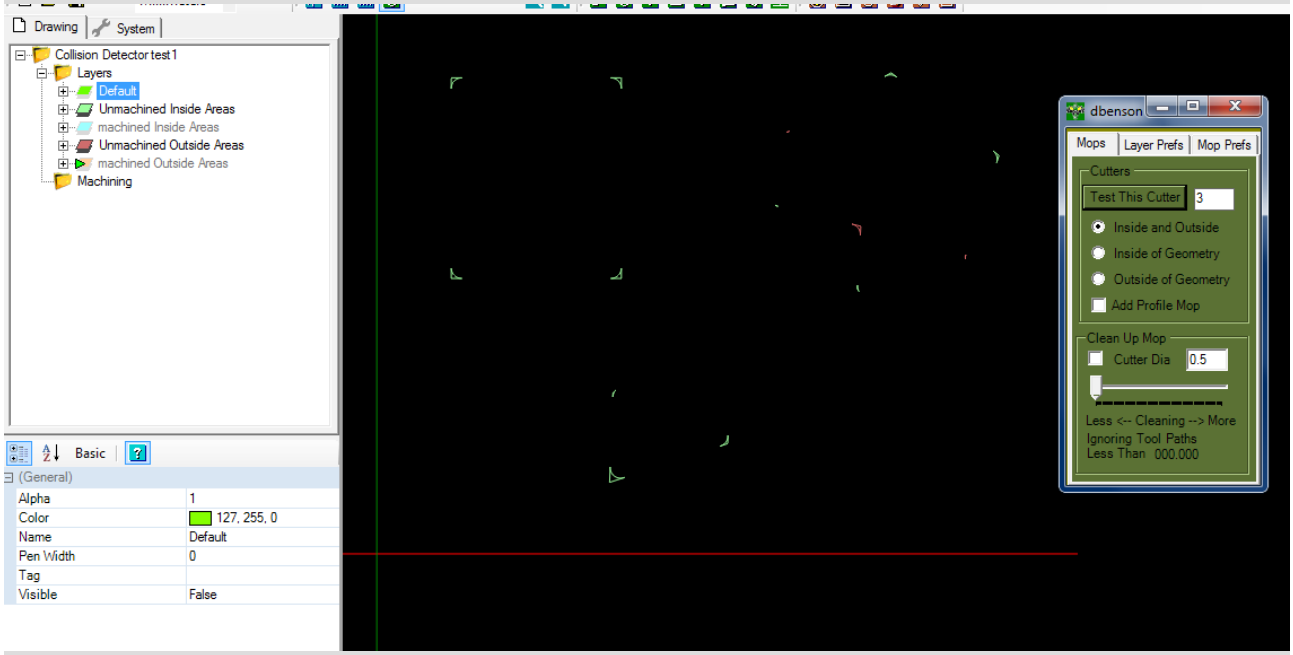
When you open up the file there are 3 shapes. Open up the plugin and move it to a convenient spot on the screen.



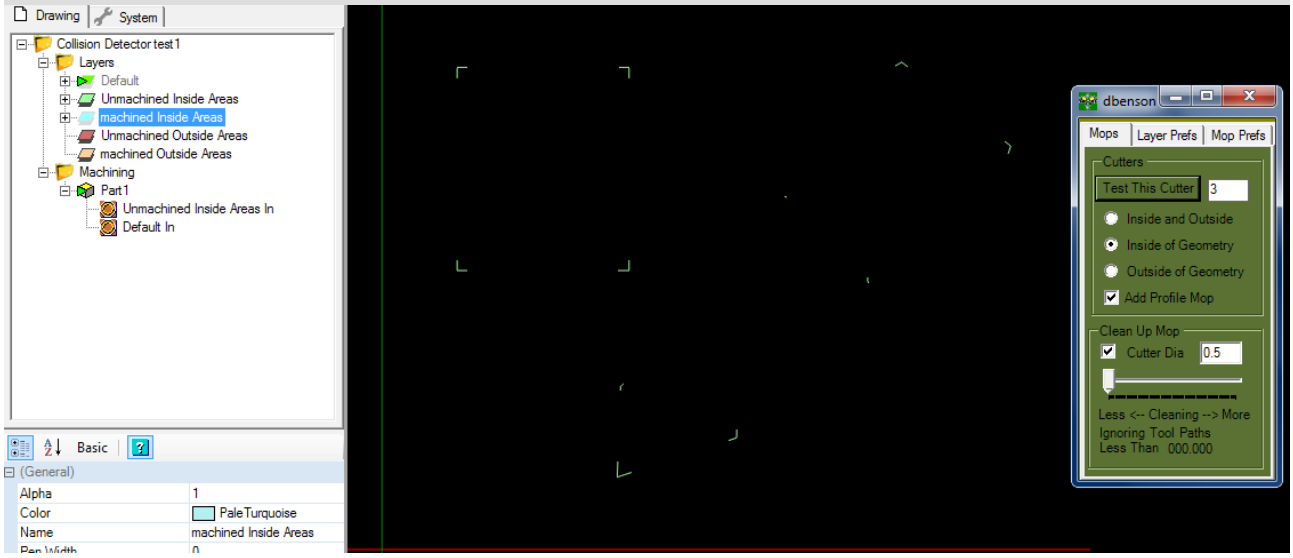
Now click the test this endmill button and You will be presented with this screen.



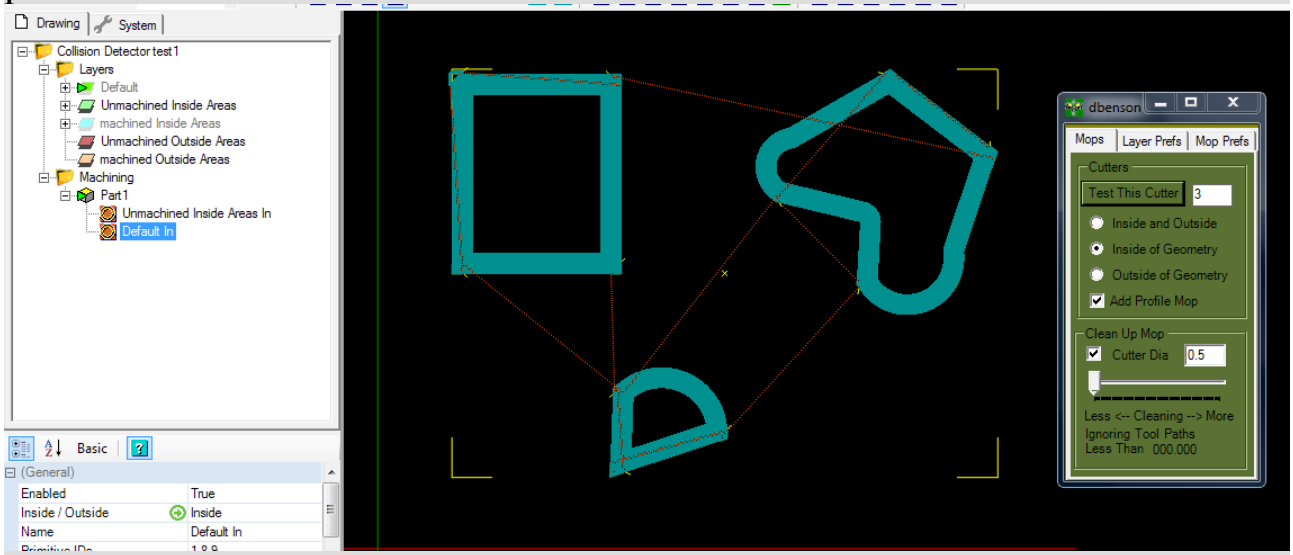
If you deselect the machined inside and outside Layers you will be able to see the areas of the shapes that can not be cut with the selected 3mm tool and the screen will look like the figure below.

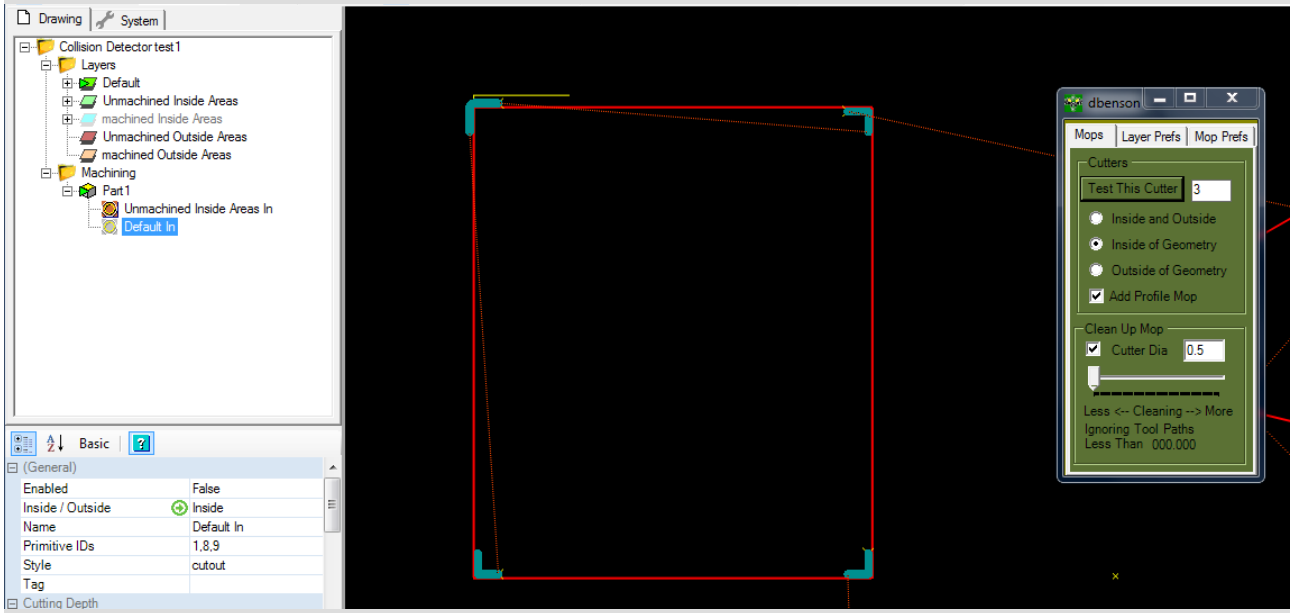


Ok so now we have identified that there are quite a few areas that will not be cut let us see what we can do about this.
Firstly delete all the Layers except the default Layer.
Now select the “Inside of Geometry” radio button.
Now check the “Add Profile Mop “ and “Add Clean Up Mop “Check boxes
You can leave the cleaning slider at zero for now.
Now click the “Test this Endmill “ button and you will be presented with Two Mops “Default in” and “Unmachined Inside Areas”



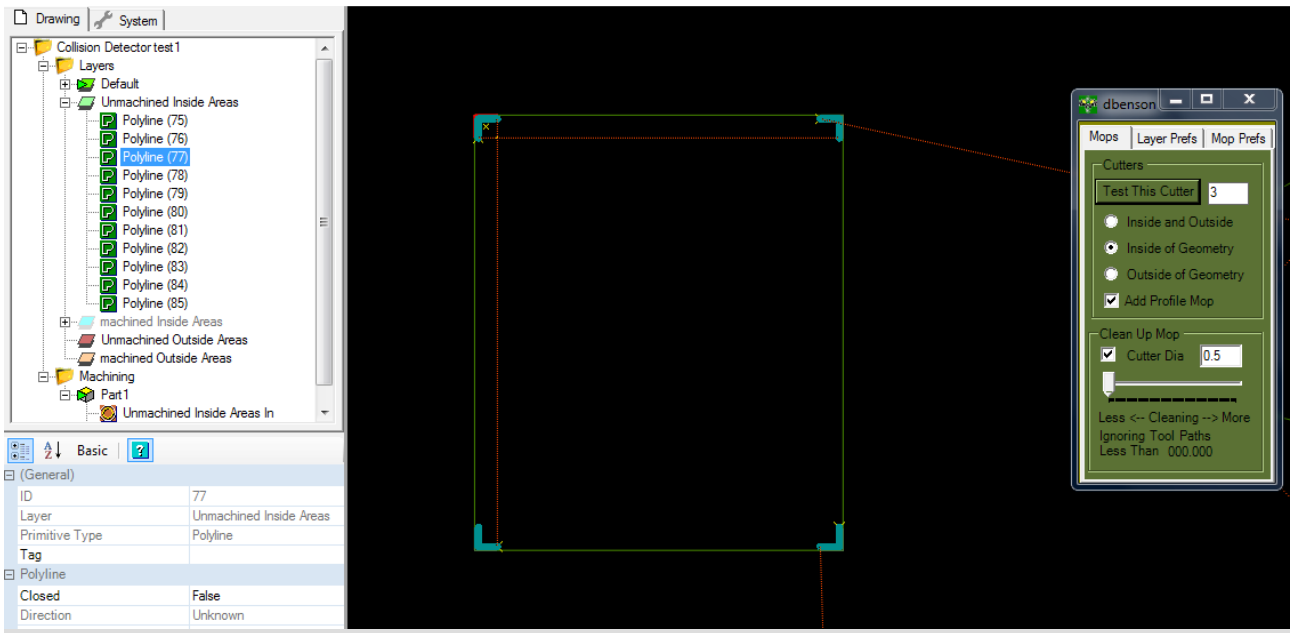
Now From the machining menu select Generate tool paths and you will be presented with a screen that looks like this.

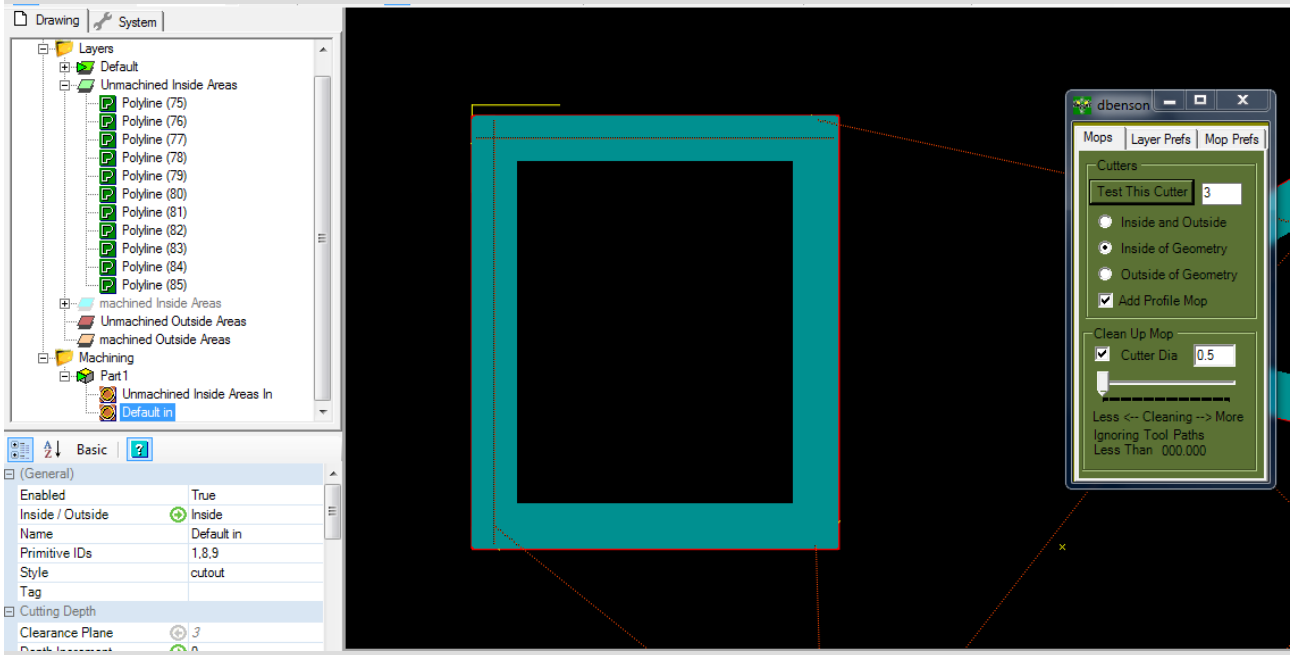




Now you should turn off the Default Mop and inspect that all of the tool paths are indeed on the inside of the shapes and they all seem alright except one at the top left hand corner of the square.

To fix this and place the tool path on the inside of the shape simply select the tool path, and from the Edit menu do a poly-line reverse and now generate the tool paths again and the tool paths are all correctly placed.

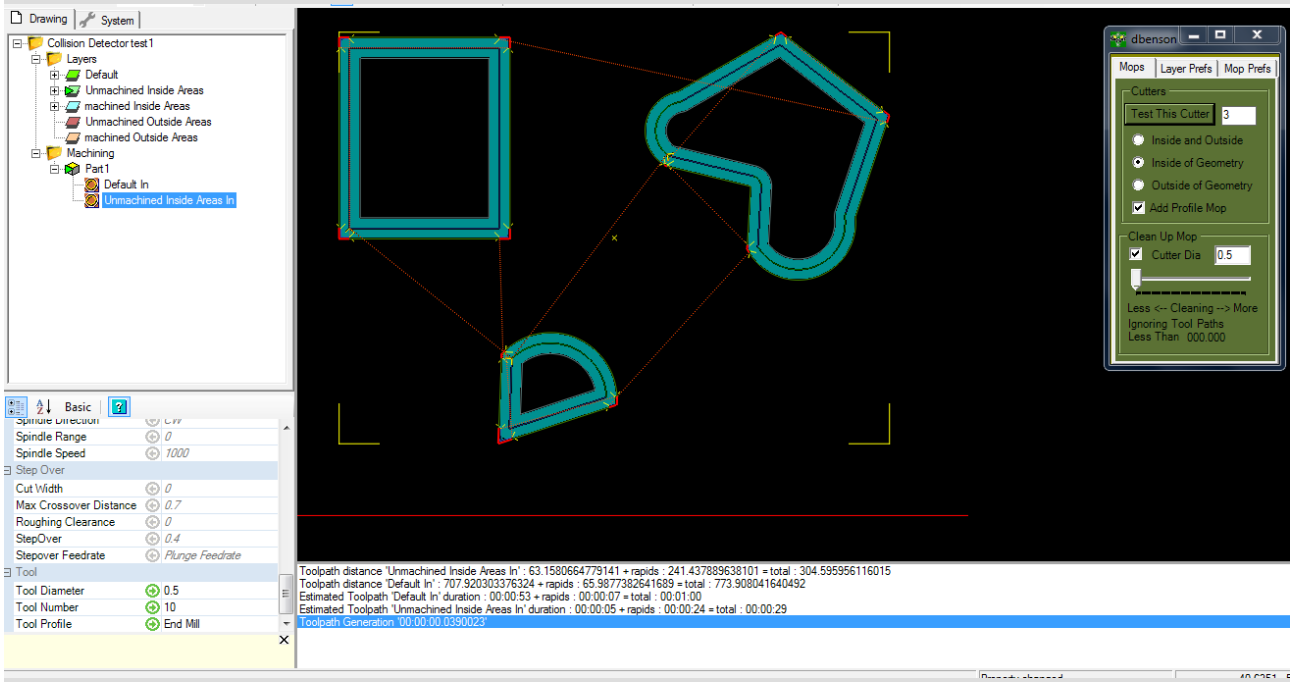




Once you have done this click on the Default Mop and press the space bar and you will be able to see the combined tool paths and just how much of the shape you can cut.

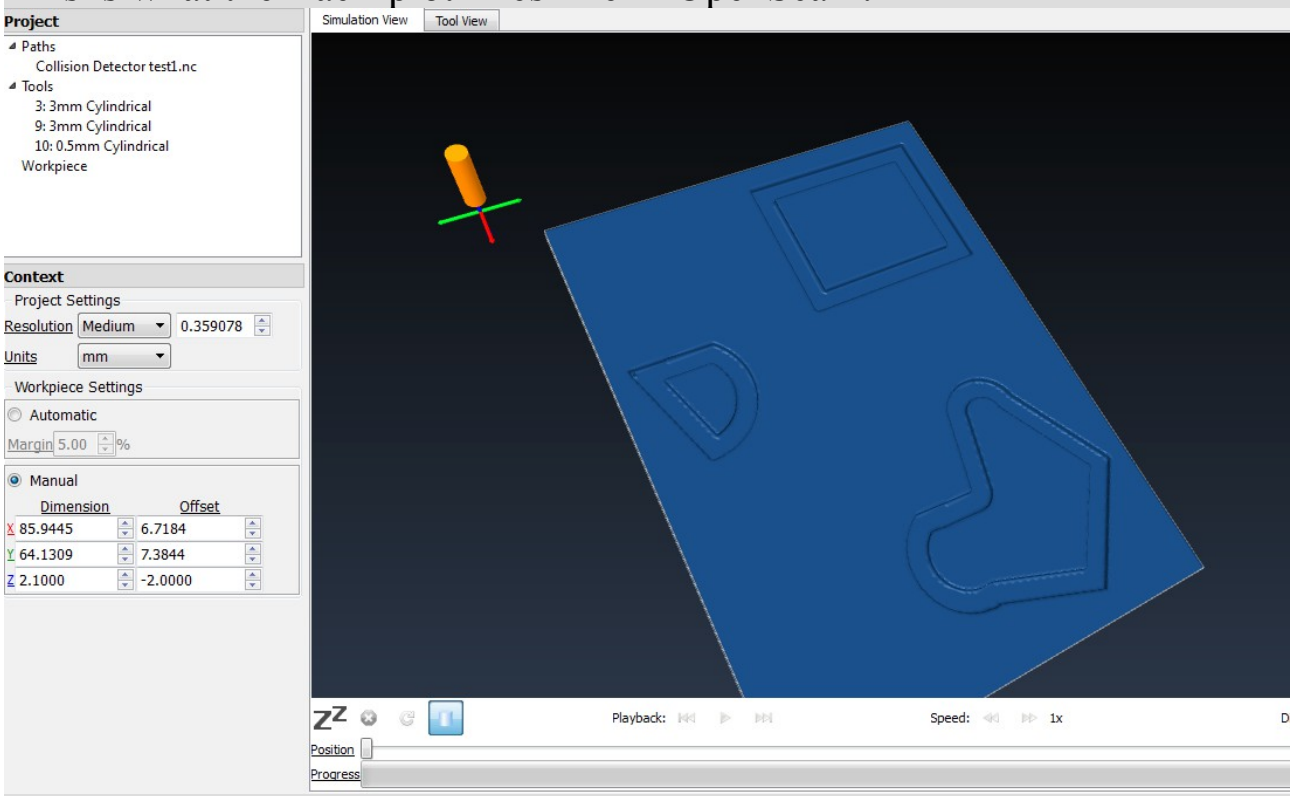
Now zoom out and you can see that all of the shapes have a tool path and your good go and all you have to do is insert the appropriate speeds and feeds,and other details into each Mop for your machine.

Your screen should look like this after generating the G code.



Now it's always prudent to simulate your work if you have a Back-plotter Cut Viewer (Commercial software) or OpenScam (Open Source) will do the Job.

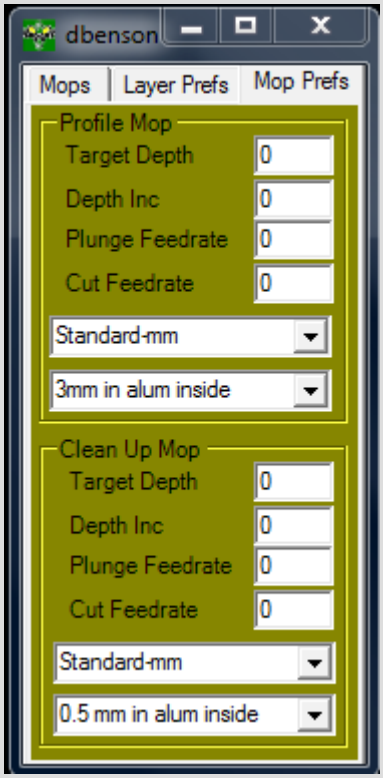
This is what the Back-plot likes like in OpenScam:

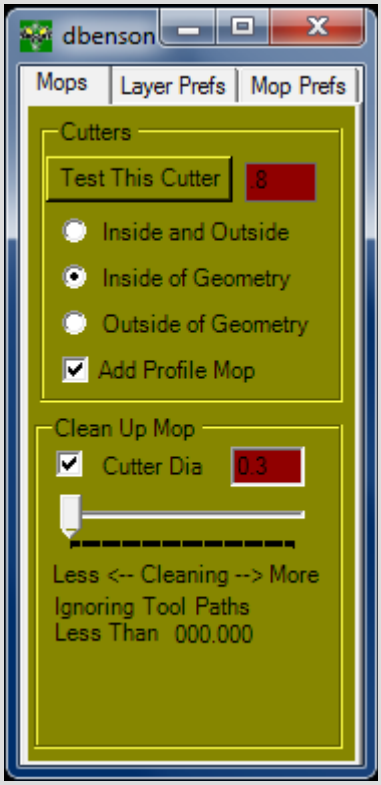


The last tab the Mop Prefs just lets you enter some predetermined values that will be entered into the mop when they are made for you.
One thing to note is that you don't have to put a “-” sign in front of the target depth.

As of version 1.3.5

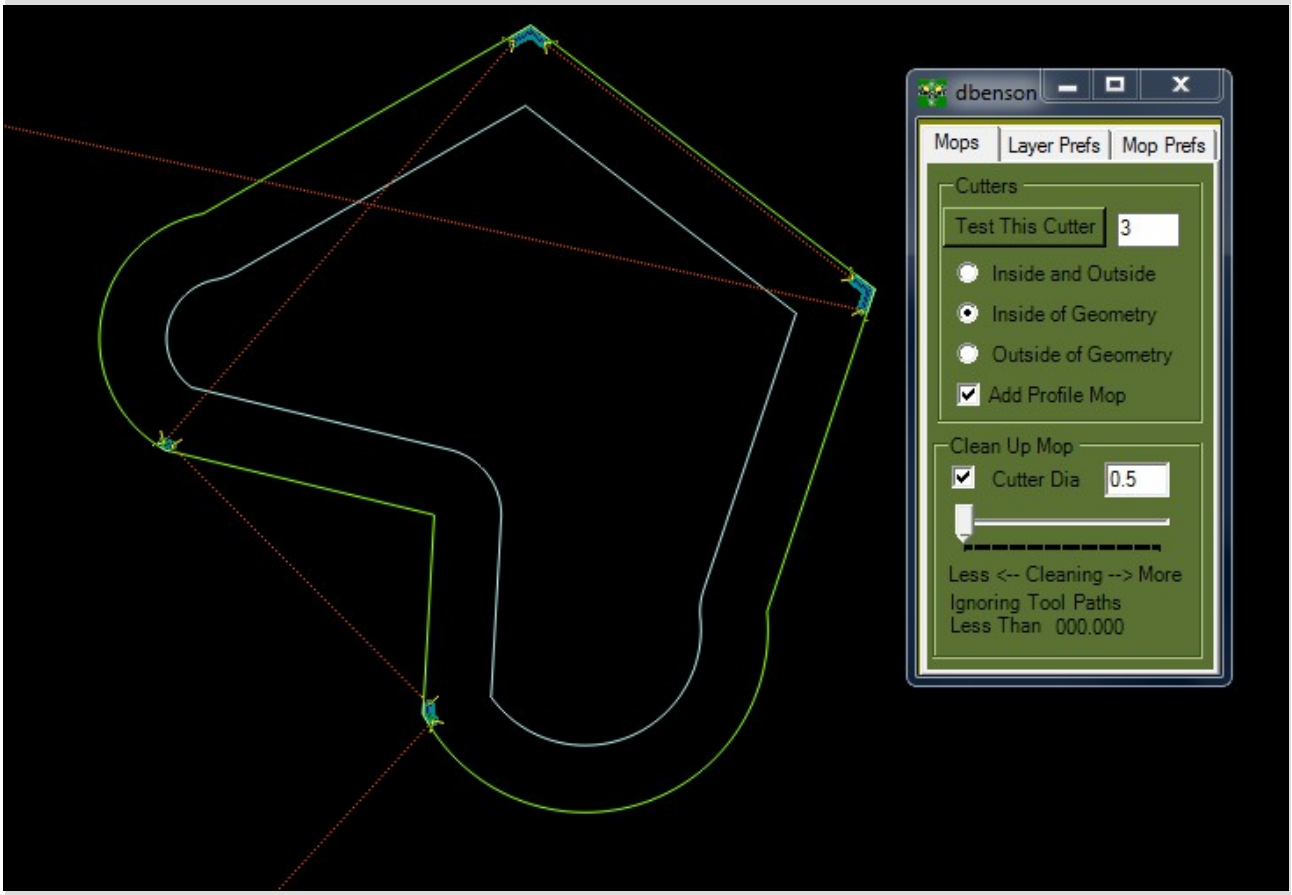
You can now set one of CamBam's predefined styles or your own custom style to the mops to be produced . Which means that the mops will be fully specified with no adjustments necessary after they are generated.
If you select this option the Tool Diameter textboxes will reflect this by changing colour to red to indicate that the Tool Diameters from the Style or Custom Style are being used rather than the Tool Diameters in these text boxes.



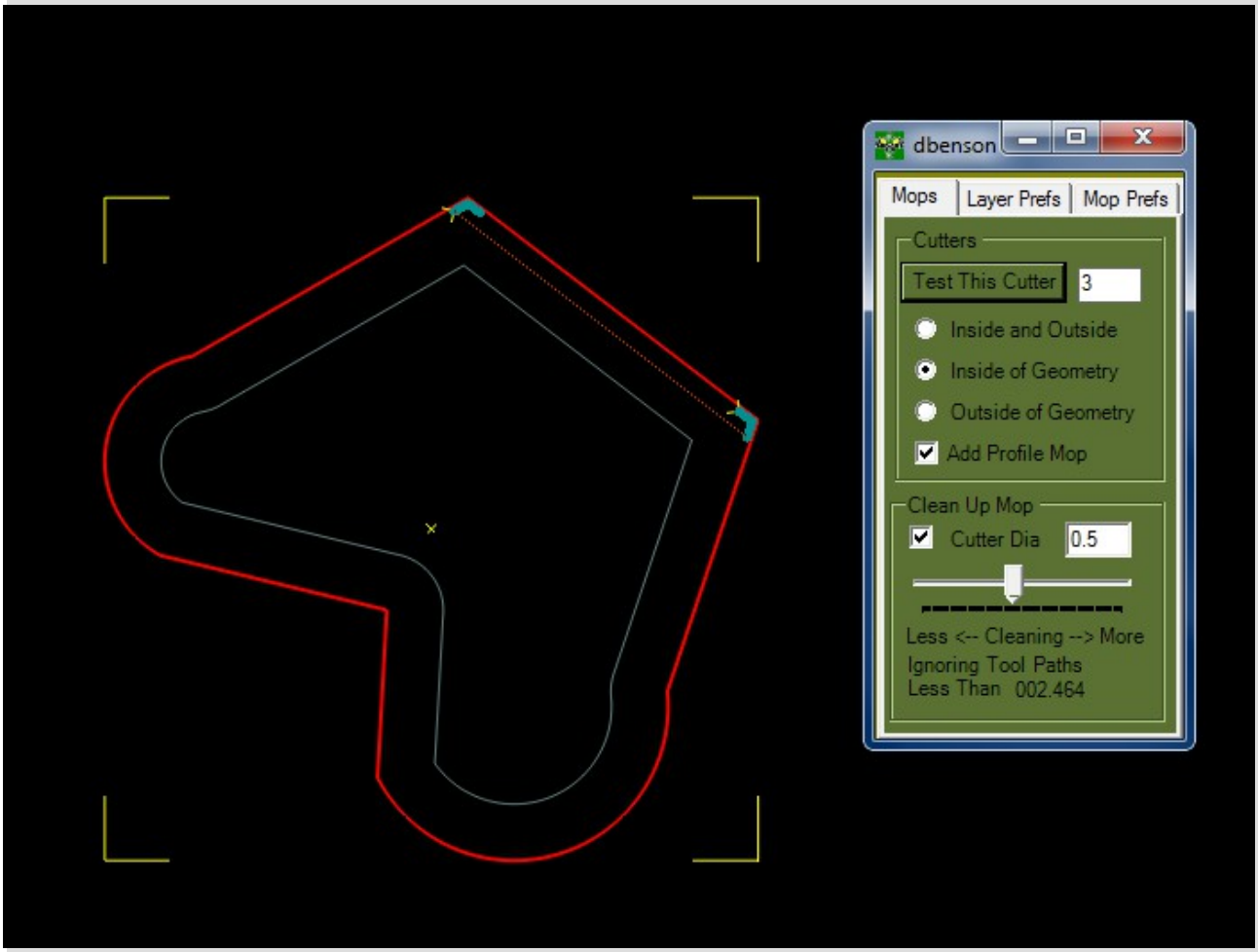


One last thing to mention is the tool path cleaning slider on the bottom of the form.
It's default value is zero which means that all of the tool paths will be generated for the shape, even if they are very small and you may make a value judgement that you don't want to cut them, so you progressively eliminate them by sliding the slider to the right.
See the before and after figures below and note the sliders position.

Before with slider at the zero position

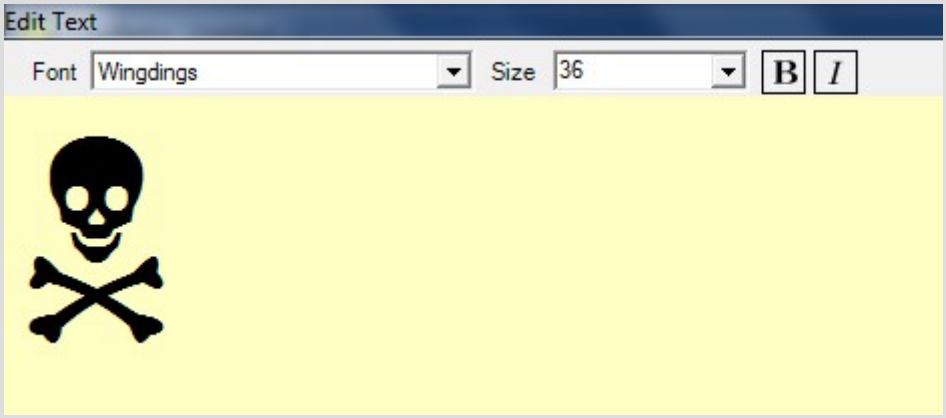


After with slider at about the half way position.

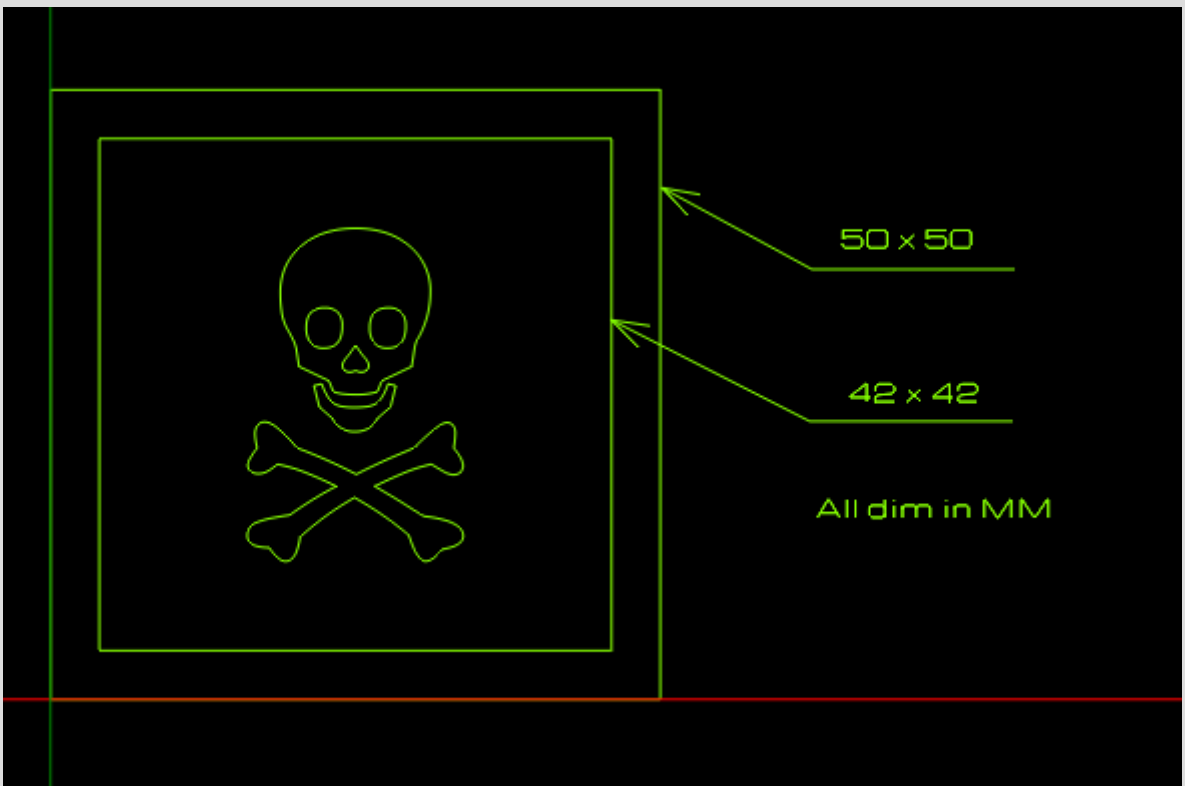


Text

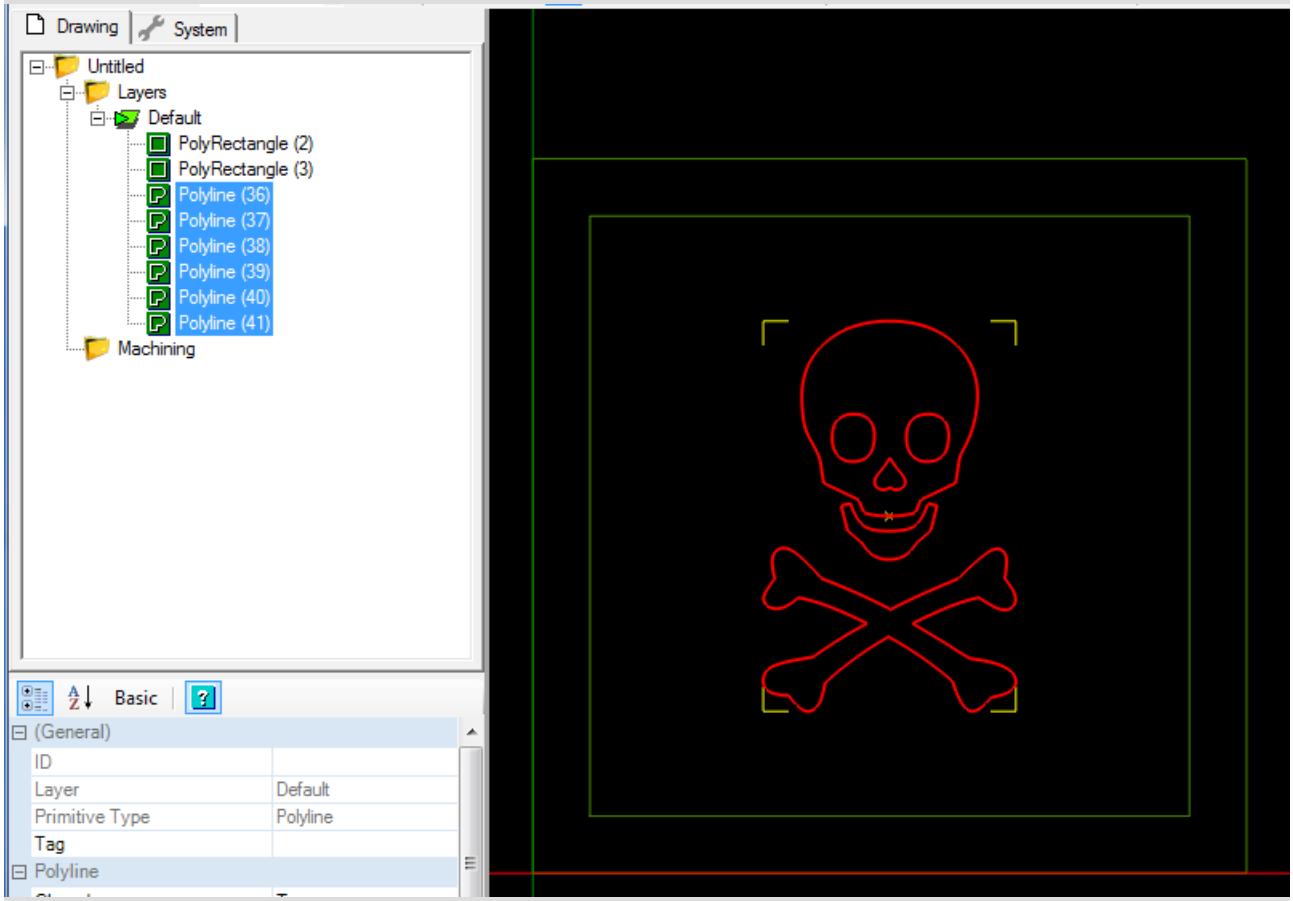
Text inserted into your document from the CamBam menu is a collection of poly-lines that are grouped together into a region, and need to be converted into poly-lines to work with the plugin.
However after you have created Mops with the collision Detector and renamed the mops and associated Layers, You can if you need to, convert the poly-lines back into regions.
Ok so lets do some text, Insert some text into your file using the Cambam menu tool bar icon.
Select the Wingdings font and use the capital letter “N” with a size of 36.



Then create two squares one 50 x 50 mm and the other 42 x 42 mm and place the text into the centre of the boxes.

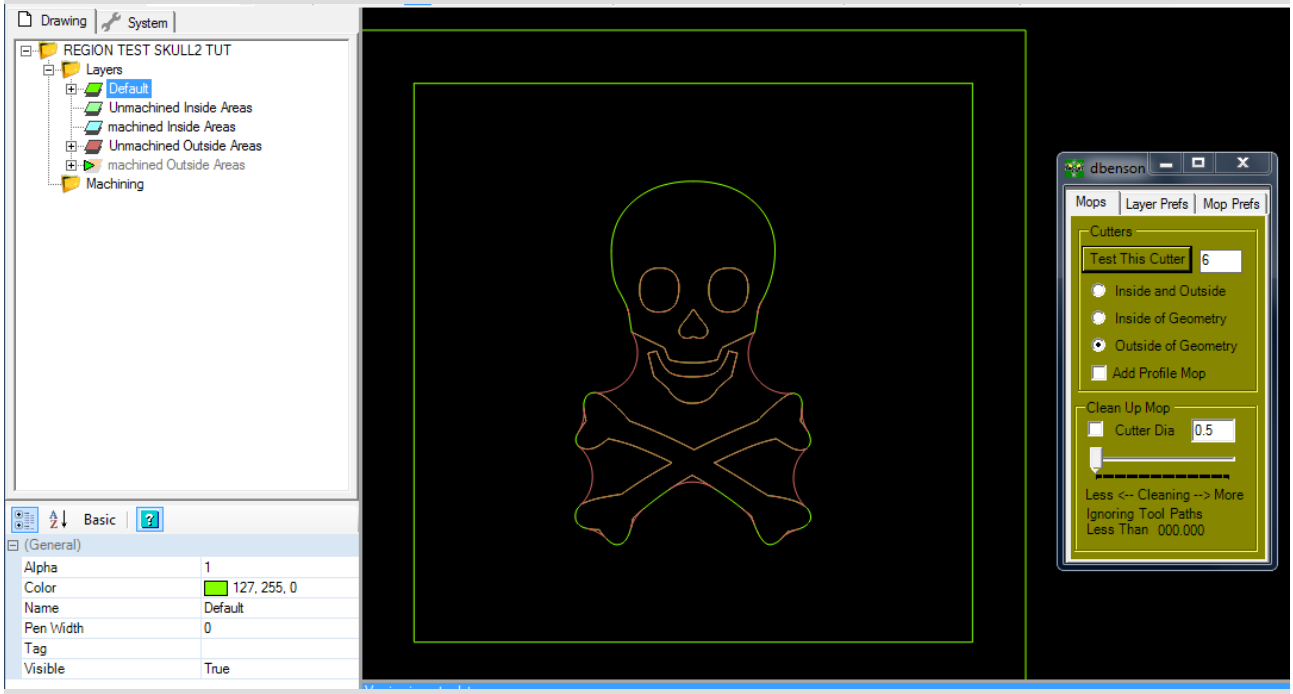


Then select your text (the Jolly Roger shape) and covert it to poly-lines

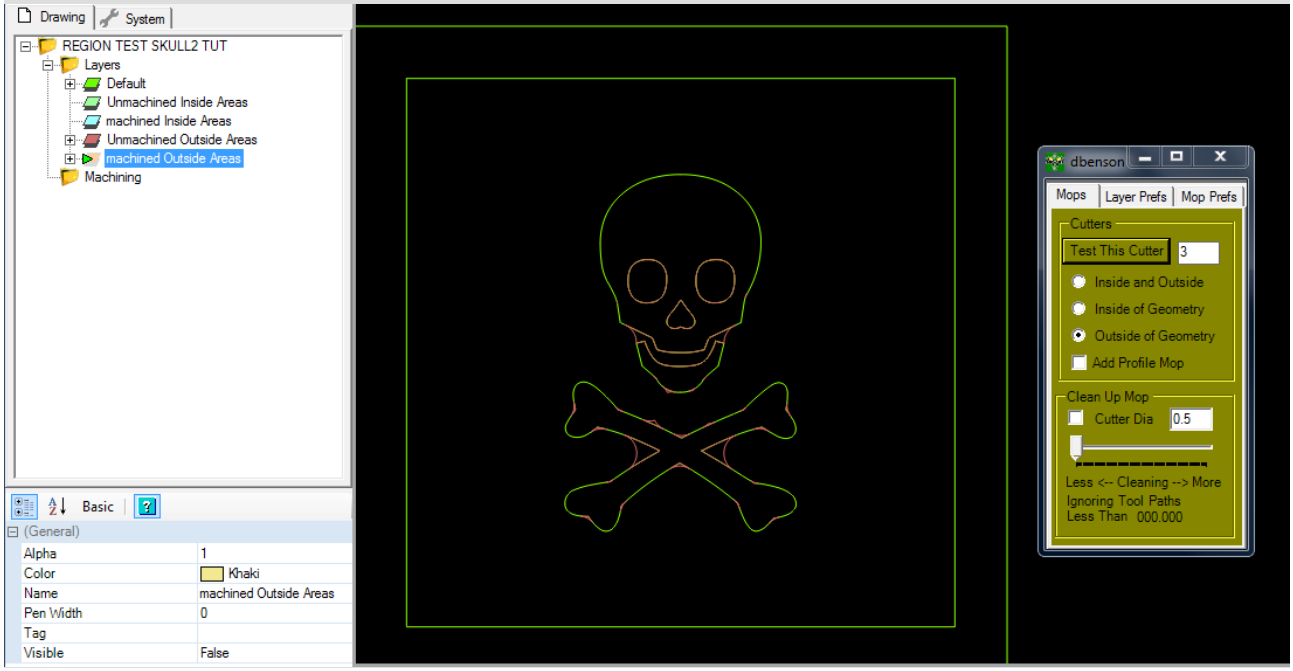


At this stage I moved the bones down 0.5 mm down in the y axis just for aesthetics.
The tools I had on hand to cut this shape out were a 6mm four flute endmill,3mm two flute slot drill,1.5875 mm two flute endmill and a 0.5 mm reground Vee Cutter.

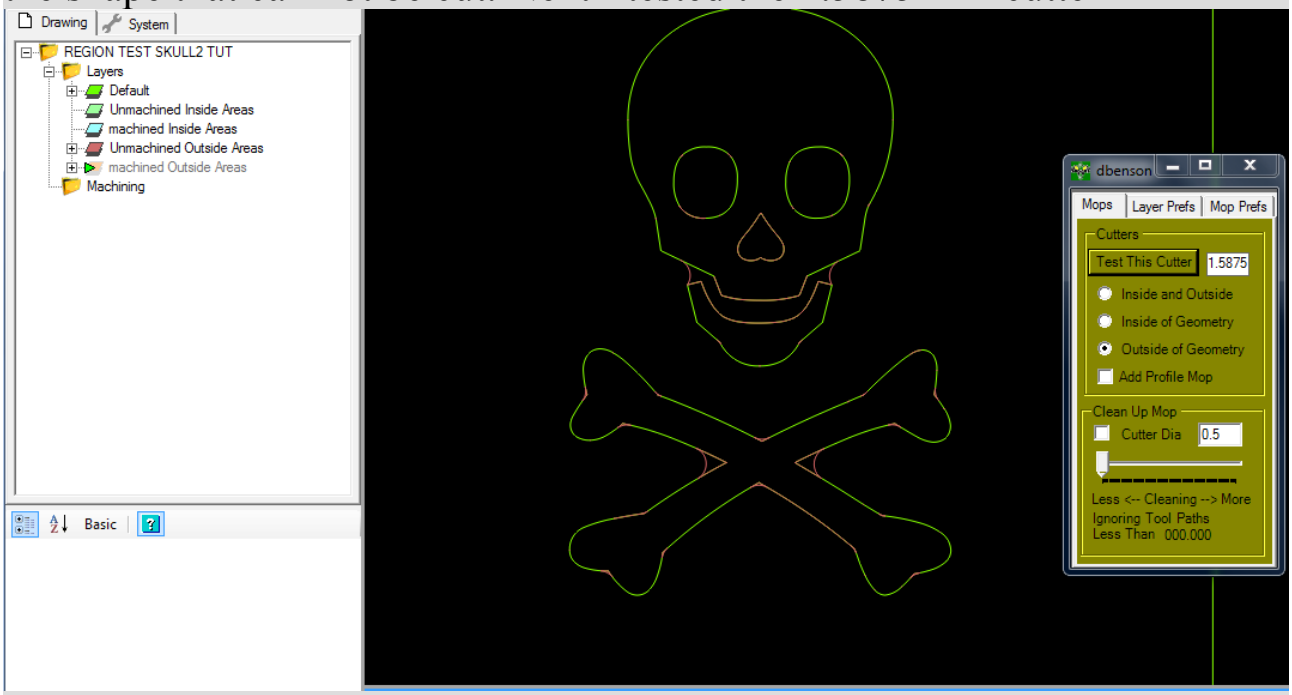
Next I loaded up the Collision Detector Plugin, Typed 6 into Test this endmill text entry box and clicked the Test this endmill button so that I could see which portions of the shape could be cut and those that could not.



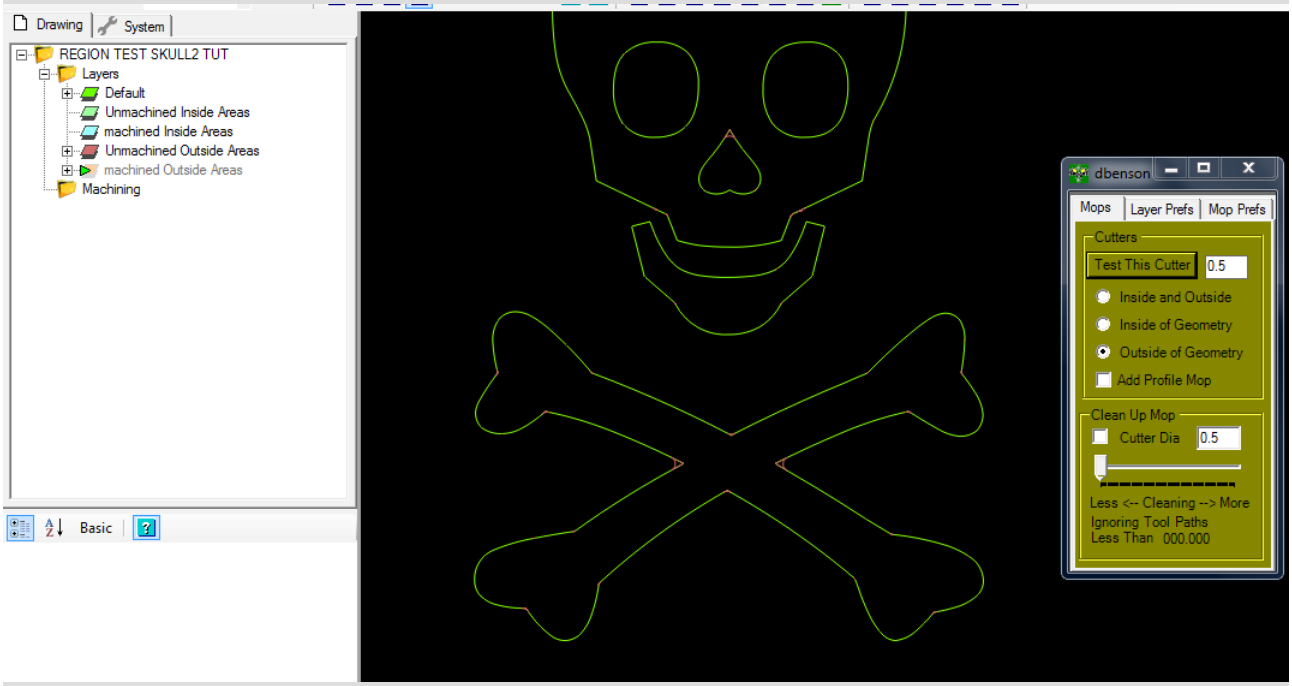
You can toggle on and off the Default layer to see just how much of the shape will be cut. As you can see there quite a few areas that will not be cut with the 6 mm cutter, So next I tested a 3 mm cutter to see how more of the shape could be cut.



As you can see a lot more of the shape can be cut but there are still parts of the shape that can not be cut. Next I tested the 1.5875 mm cutter



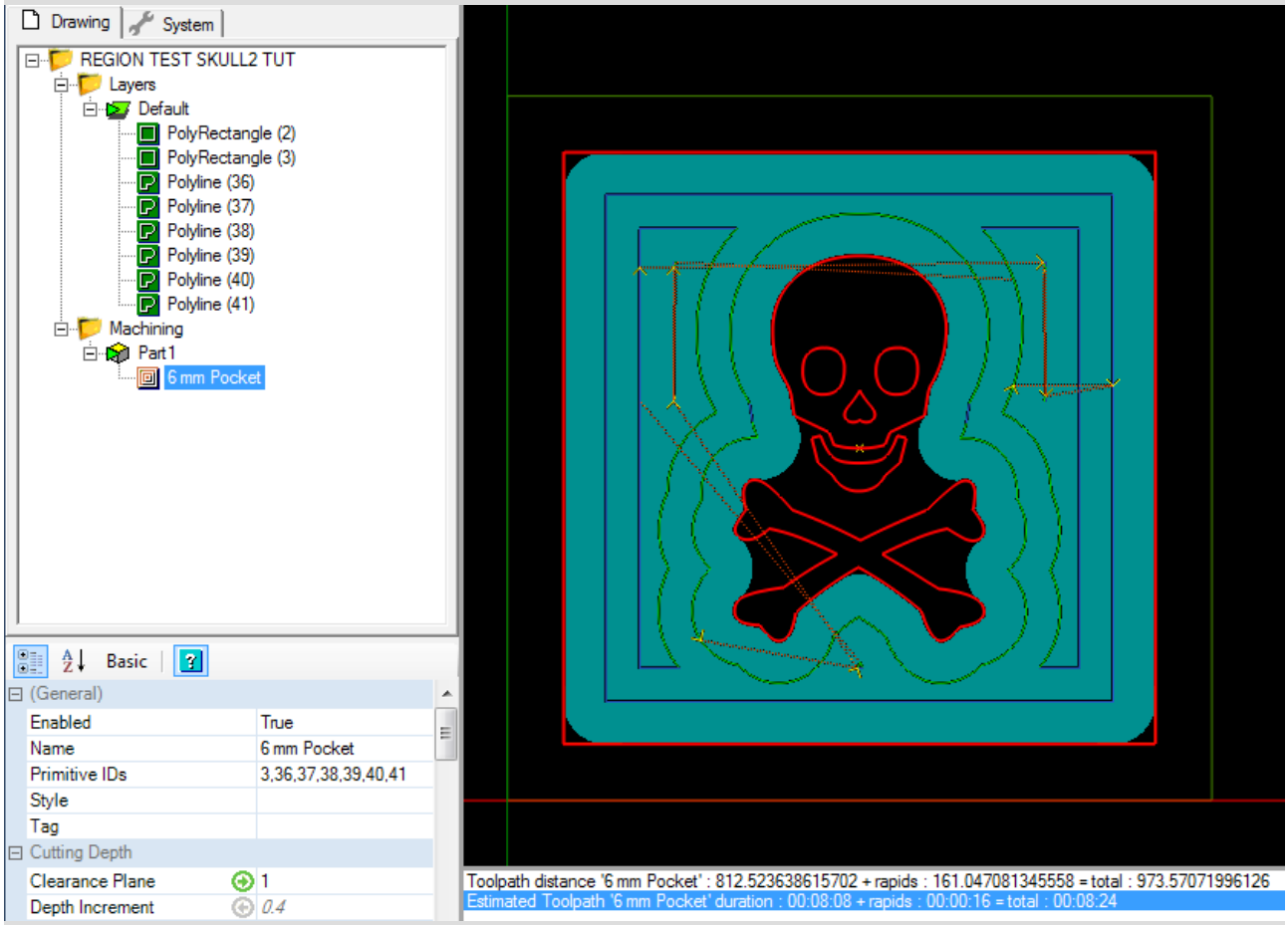
and finally I tested the 0.5 mm cutter.



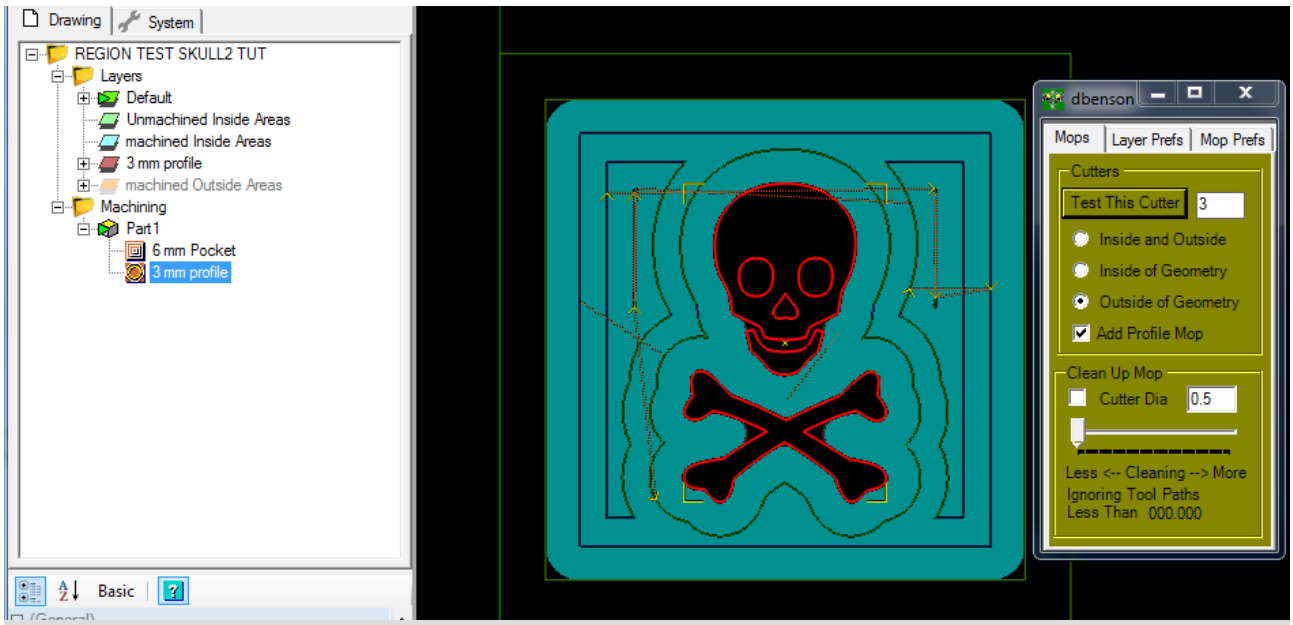
Which I was satisfied with, And now on to applying mops to the shape for cutting.

Note. What you may see on the screen only sometimes (depending on cutter size and the geometry of the shape are small specks and other debris, these will be automatically cleaned by the plugin when it produces the mops according to the value set by the cleaning slider.

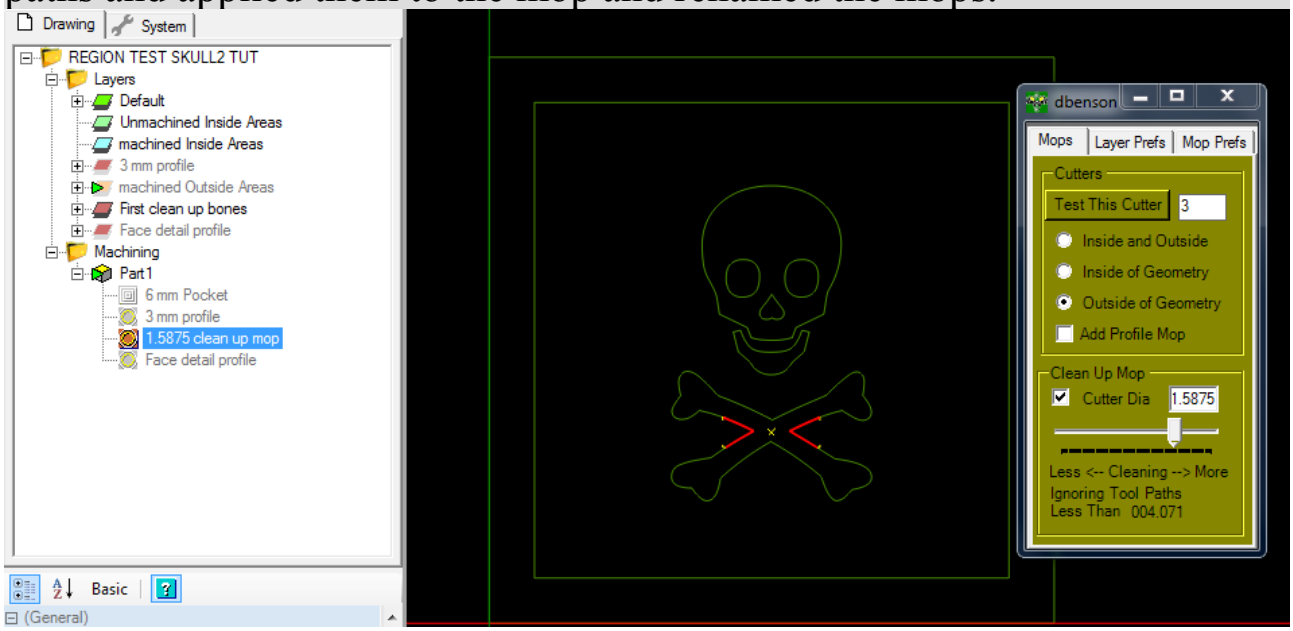
The first mop to make was a pocket mop between the interior square and the the outline of the shape. Turn show cut widths on in the view menu.
Select your objects and press the pocket mop in the toolbar.
And insert your values for your machine into the mop and press (cntrl T).



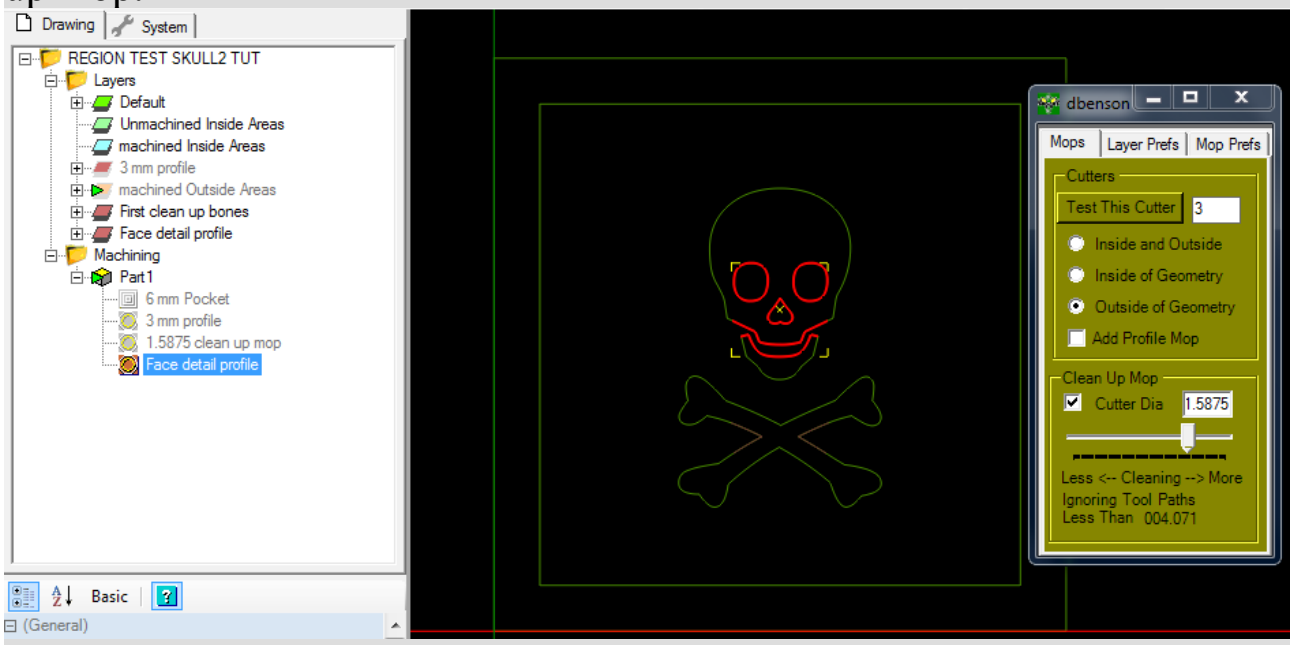
Now on to use the collision detector to make some mops.
Enter 3 into the test this endmill text box, check the Add profile mop check box and click the test this endmill button and press (cntrl T).
It's also important to re-name your new mop and the Layer it was created from. I just copy and paste the mop name into the layer name as you can see below. This is very important as if you don't rename the layer it will be overwritten and you will lose the tool paths associated with it in the mop.



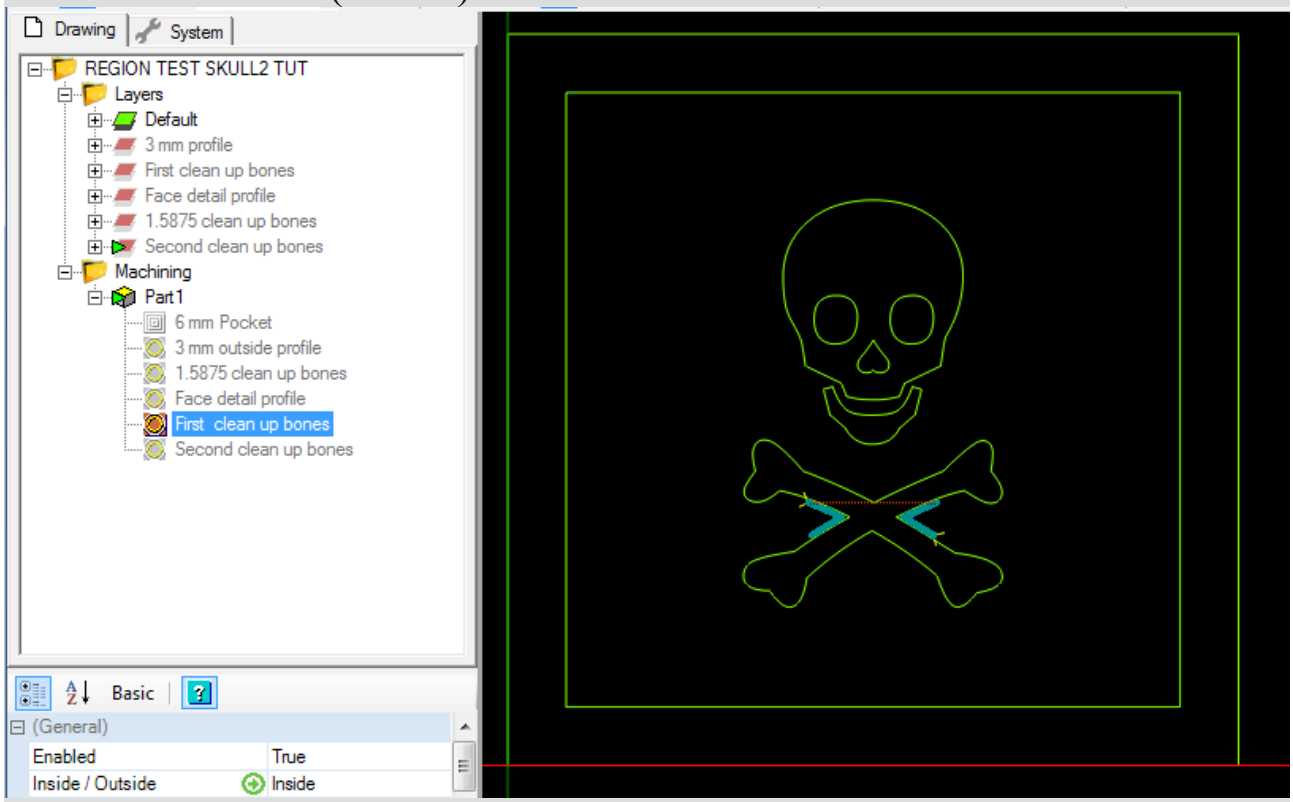
Now the shape is almost completely cut out, but we still need another mop to clean up the uncut areas.
So now uncheck the add profile mop check box, and check the Clean up mop check box and enter a value of 1.5875 and move the cleaning slider to ignore tool paths less than 4 mm and press the test this endmill checkbox. And insert your values for your machine into the mop like you would normally do for a regular mop or if you have set up some values in the mop preferences tab in the collision detector, those will entered automatically for you. Remember to do a (cntrl T)
Turn off the 6 mm profile pocket and the 3mm profile mops
and you can see the 1.5875 mm clean-up mop tool paths
Because I was interested in cleaning up the bones I just selected those tool paths and applied them to the mop and renamed the mops.



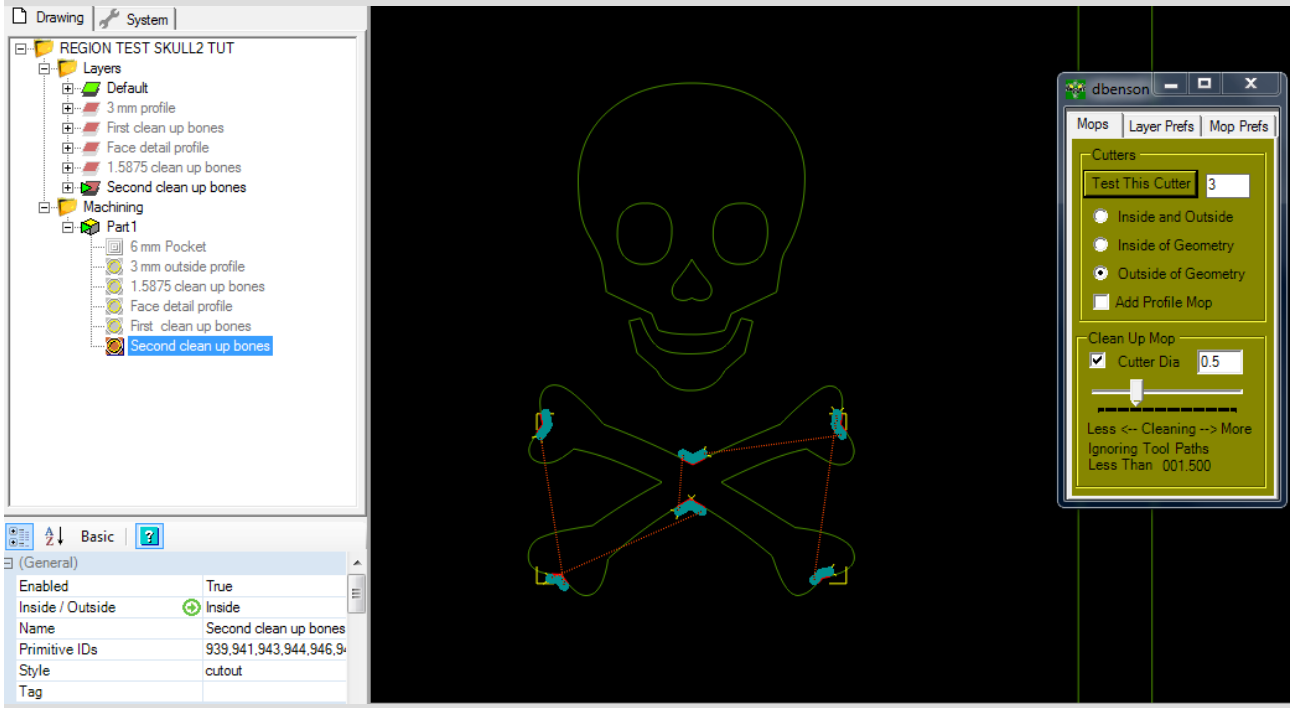
Now type 0.5mm into the Clean up mop text box and select the skull poly-lines and click the test this cutter button and generate the clean up mop.



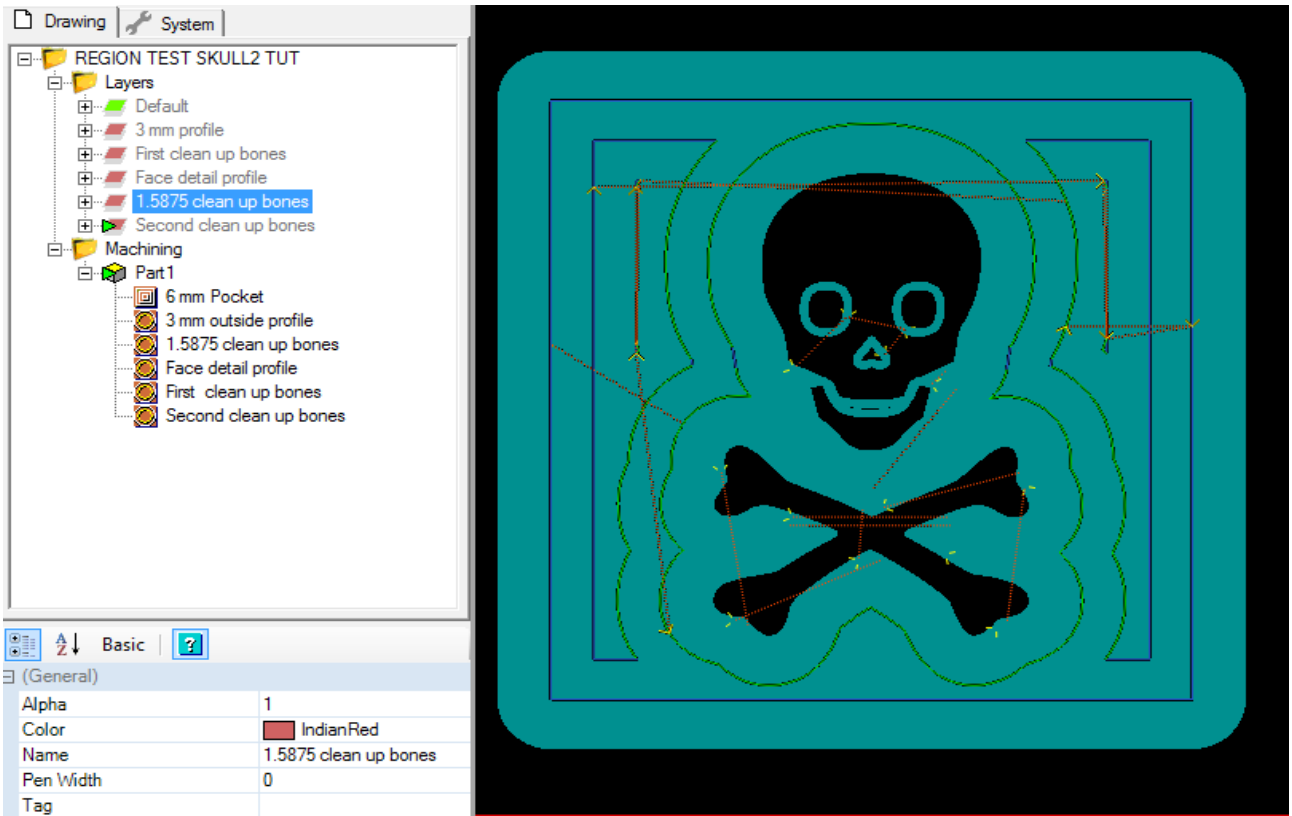
Now enter a 0.5 mm into the clean up mop text box and select the bones poly-line and generate the mop by clicking the test this endmill button, rename it to “first clean up bones” and rename the regenerated layer also. Remember to do a (cntrl T).



Finally select the bones poly-line once again and move the clean slider to ignore tool paths less than about 1.5 mm and generate the mop and rename it to “second clean up mop” and then right click on the mop and select the drawing objects as illustrated. Remember if any of the cuts appear on the wrong side of the line, select them and do a poly-line reverse and then a (cntrl T).



Now if you activate all the mops and generate the tool paths either by the menu or (cntrl + T) you'll see a screen like the one below.



And an image of the file that has been cut out on the cnc mill.

The picture on the Left has the clean up mops enabled and shows greater detail than the one on the right and the one on the left cuts much faster.