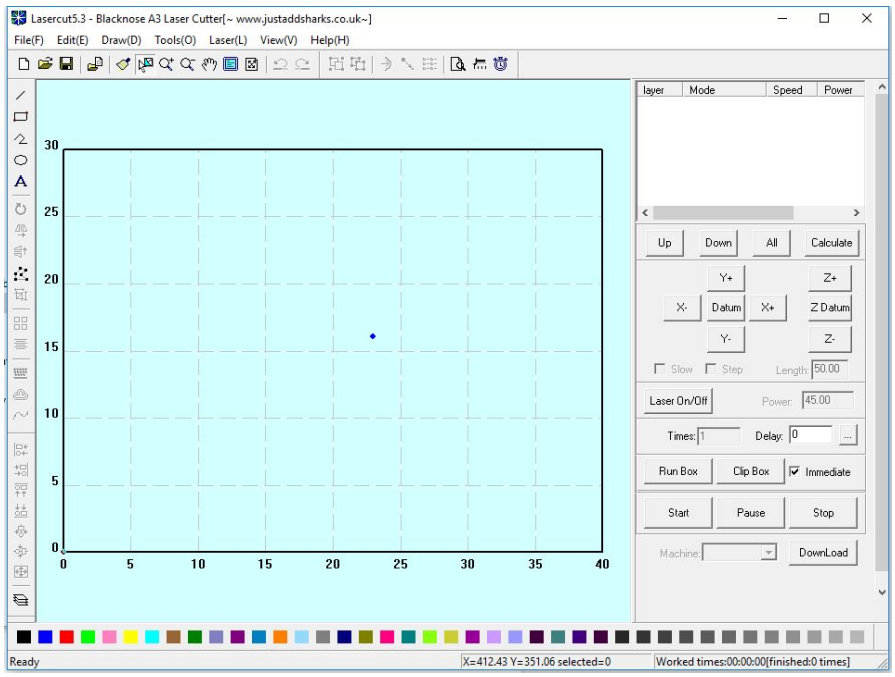
Just Add Sharks

Laser Cutters for the Evil Genius



An Introduction to Lasercut 5.3

Preparing the Artwork

Version 0.1, December 8th 2015

Precautions

WARNING!

- Never open access panels without disconnecting power
- Do not circumvent the safety cut-out switch or operate the machine with access doors open
- Never run the laser cutter unattended
- Always vent material smoke or vapour to a suitable external outlet or filter system
- Only use laser-safe materials which do not contain chlorides and formaldehydes

Introduction

LaserCut 5.3 is a dedicated laser cutter control software written by Leetro Automation Co Ltd. It is used to prepare cutting files that are sent to the Leetro MPC6515 controller hardware inside your laser cutter.

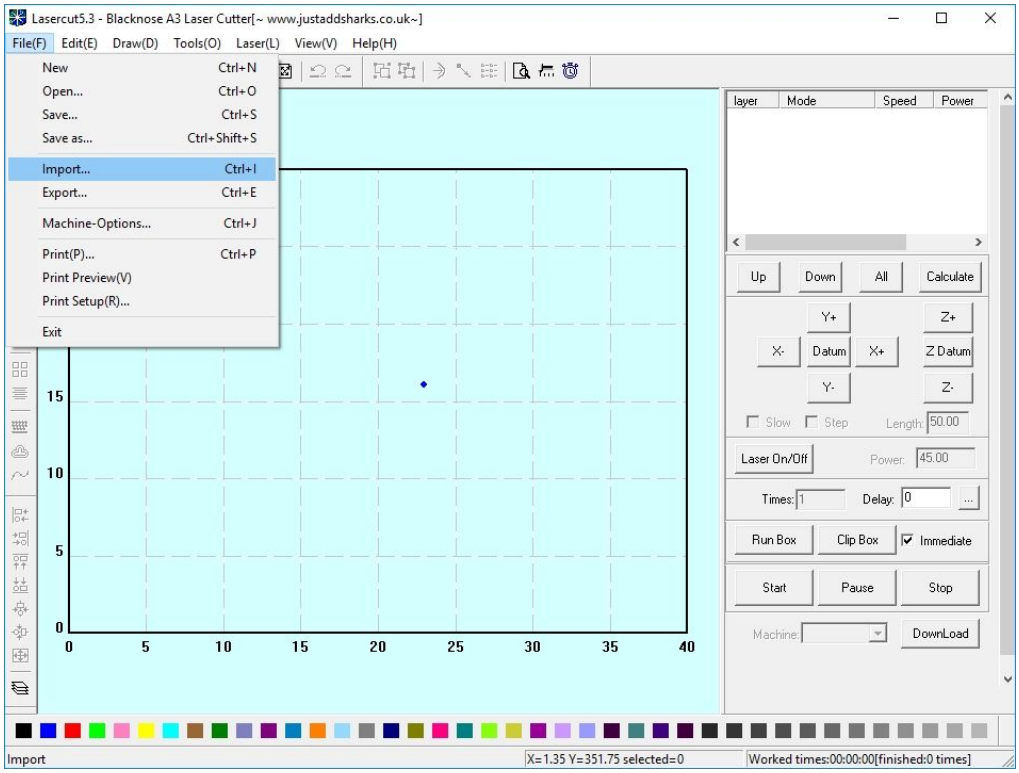
LaserCut has some basic drawing functions but it becomes most useful when dxf files are imported into the software for upload to the laser cutter. These .dxf files are created in other CAD drawing packages, such as the free, cross platform, open source vector editing package Inkscape, or the school favourite 2D Design by Techsoft. Once a .dxf file is drawn it can be imported into LaserCut.

Lasercut provides functions to simulate the cutting process that is about to occur and estimate how long the cut will take. All the information about a job can be saved and loaded by saving a project in LaserCut as an .ecp file making it easy to store settings, layouts and to repeat projects in the future.

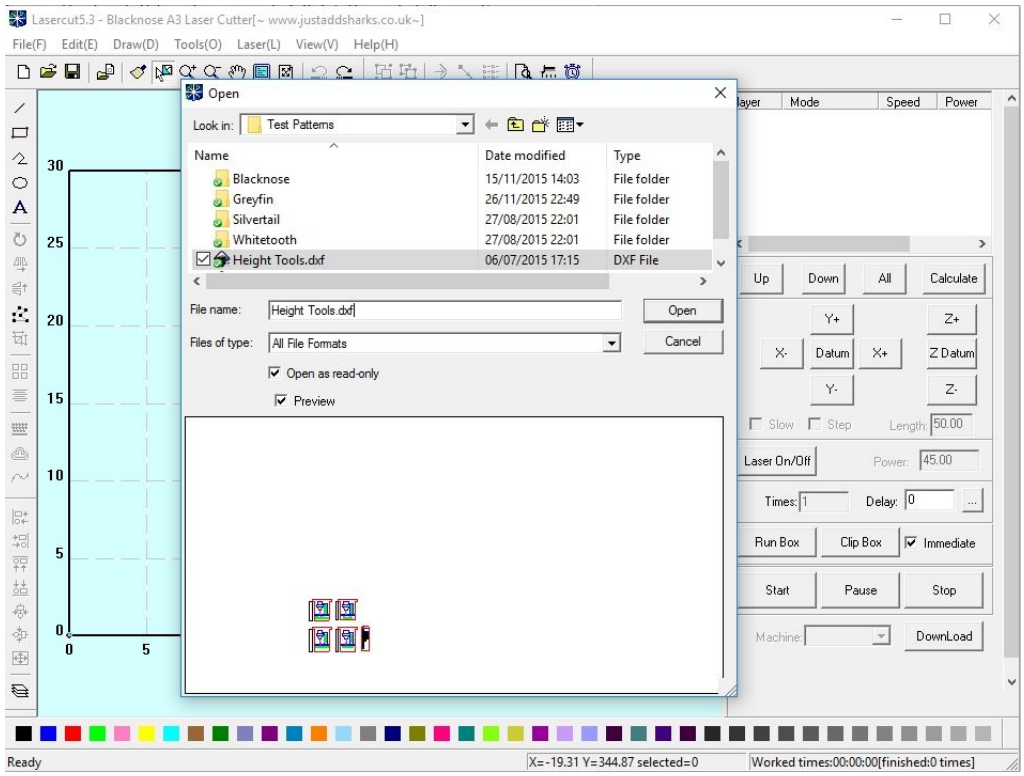
This guide will walk you through the process of importing a .dxf file into Lasercut, setting up the different cutting operations and processing the data ready to be sent to the laser.

Importing from .dxf

From the menu system in the top left hand corner, click ‘File’, then click ‘Import’ from the menu that appears.

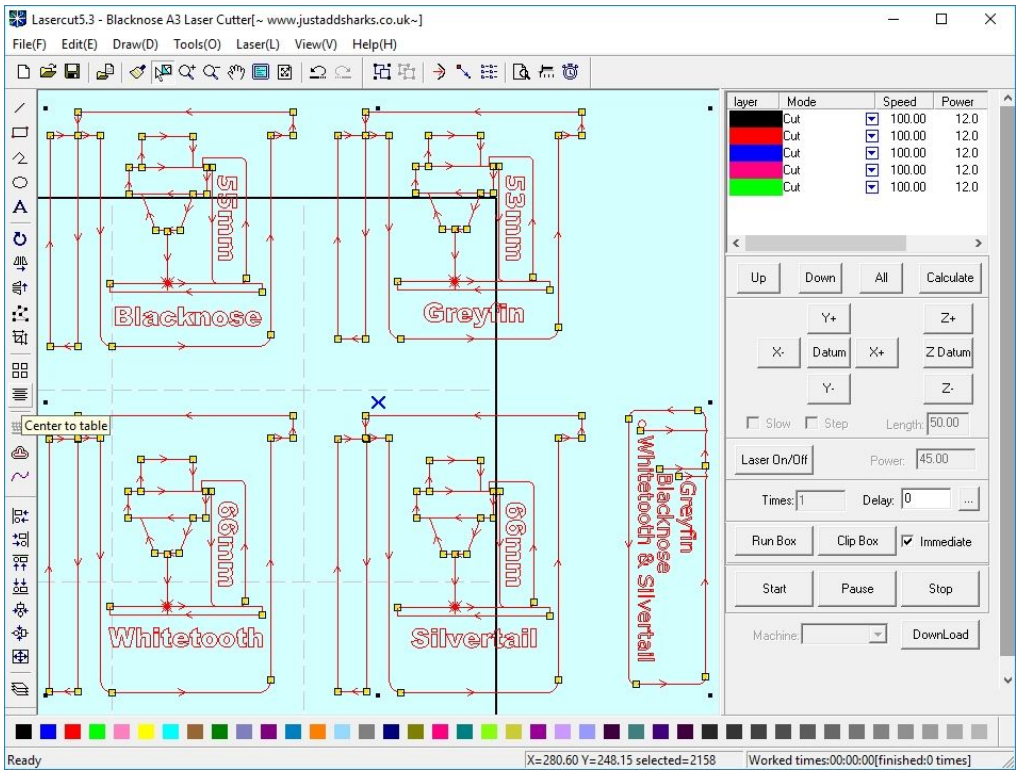


The ‘Open’ dialog window will appear which allows you to select a file to be imported into the program. Locate the ‘Test Patterns’ folder that came with the Just Add Sharks software download. Select the ‘Height Tools.dxf’ file and a small preview of the file will appear in the lower window once the file has been selected. If the preview does not appear the file may not import correctly. Click ‘Open’ once the file has been selected.

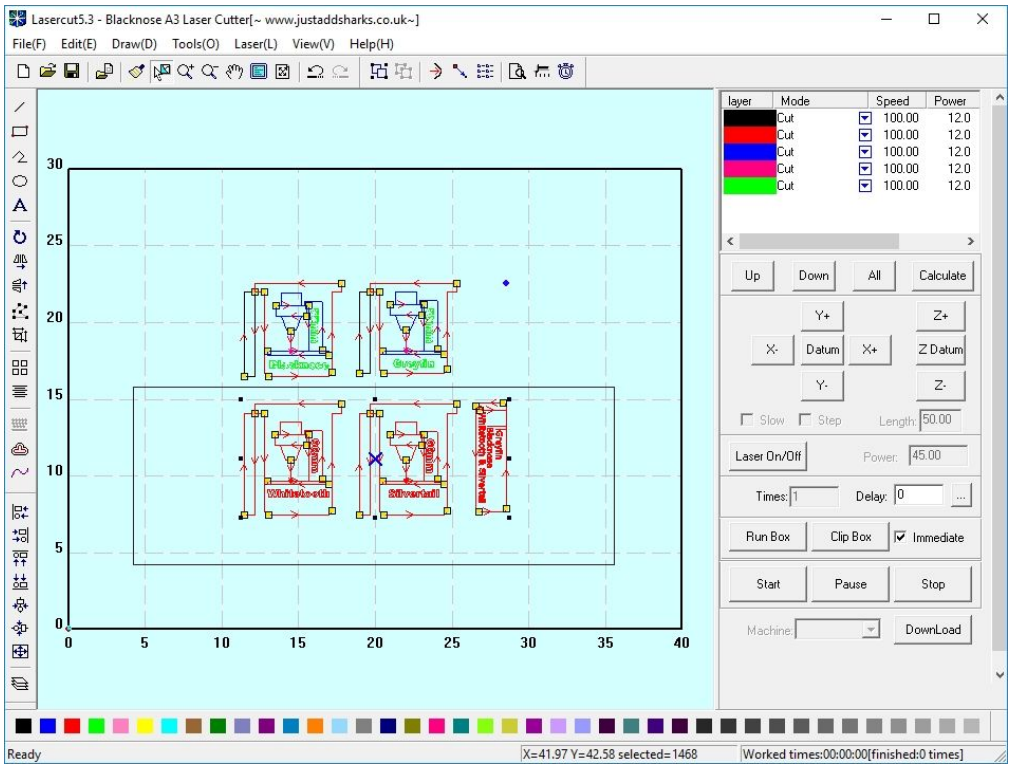


The file has to be selected by clicking on the file name, typing the name of the file into the box will not result in successful selection of the desired file.

If the file was successfully opened by Lasercut it will appear on the screen in a random location of the work bed. Often this will be outside the working area of the laser cutter.

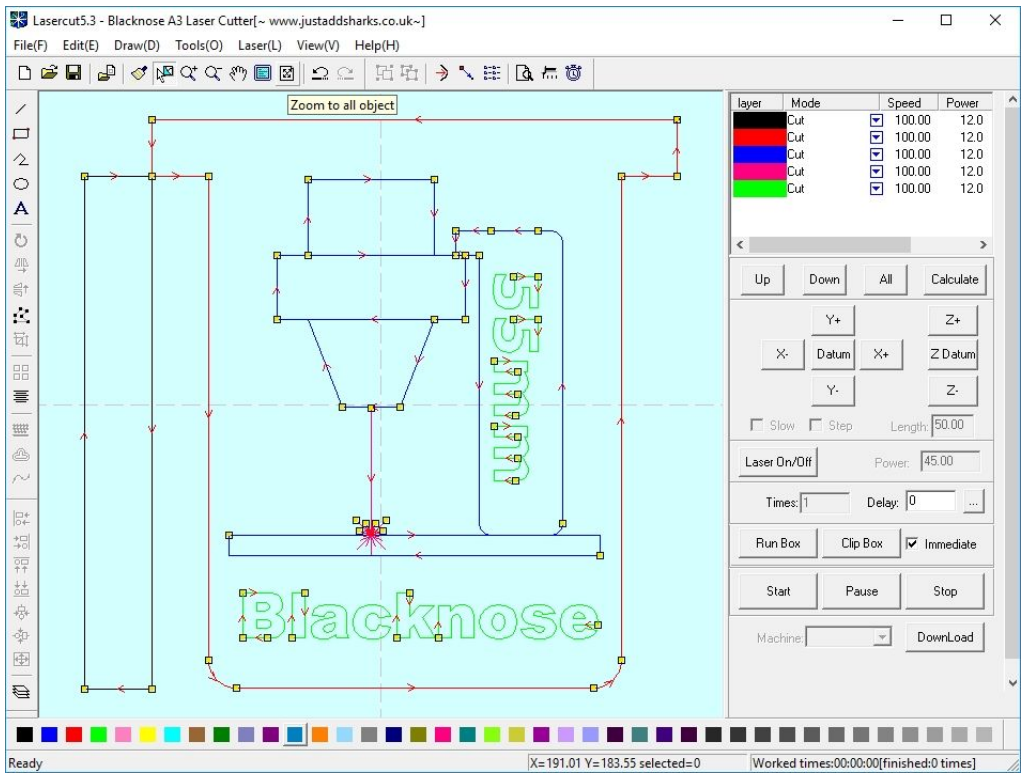


Half way down the icons on the left is a button labelled “Center to Table”. Click this button to bring the item back into the middle of the work area.



Click and drag with the mouse to select the items you don’t want to cut, in the test file you should remove the height tools for the other laser cutters.

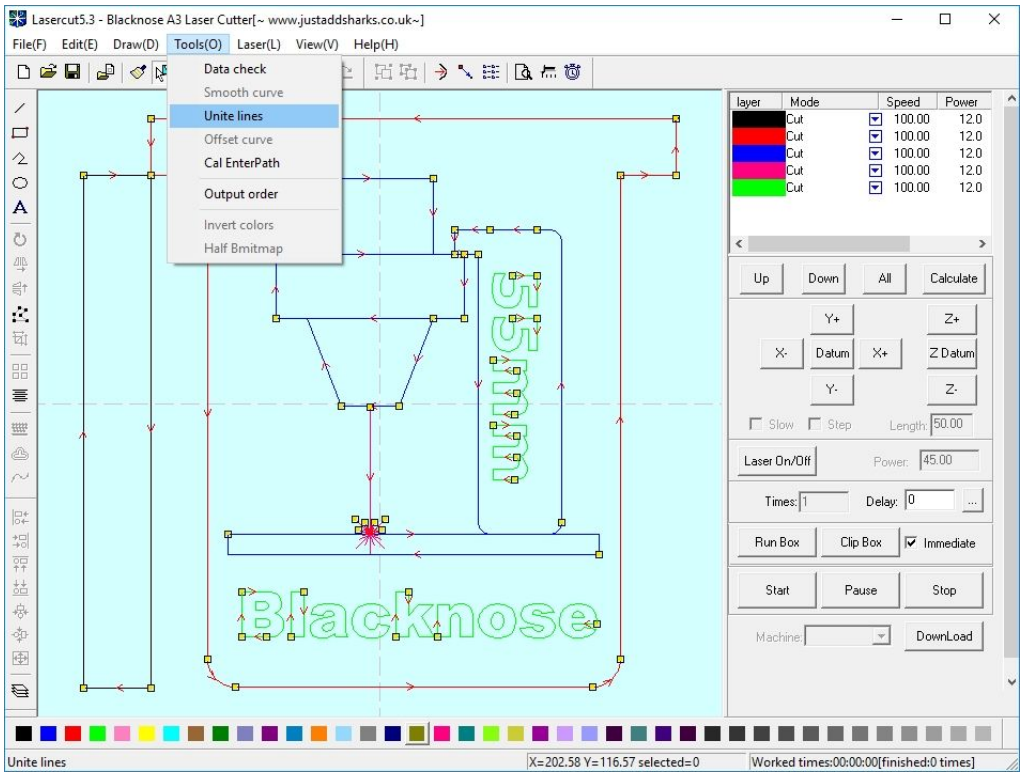
Selected items are changed to a red colour. They can be removed by pressing the ‘Del’ key.



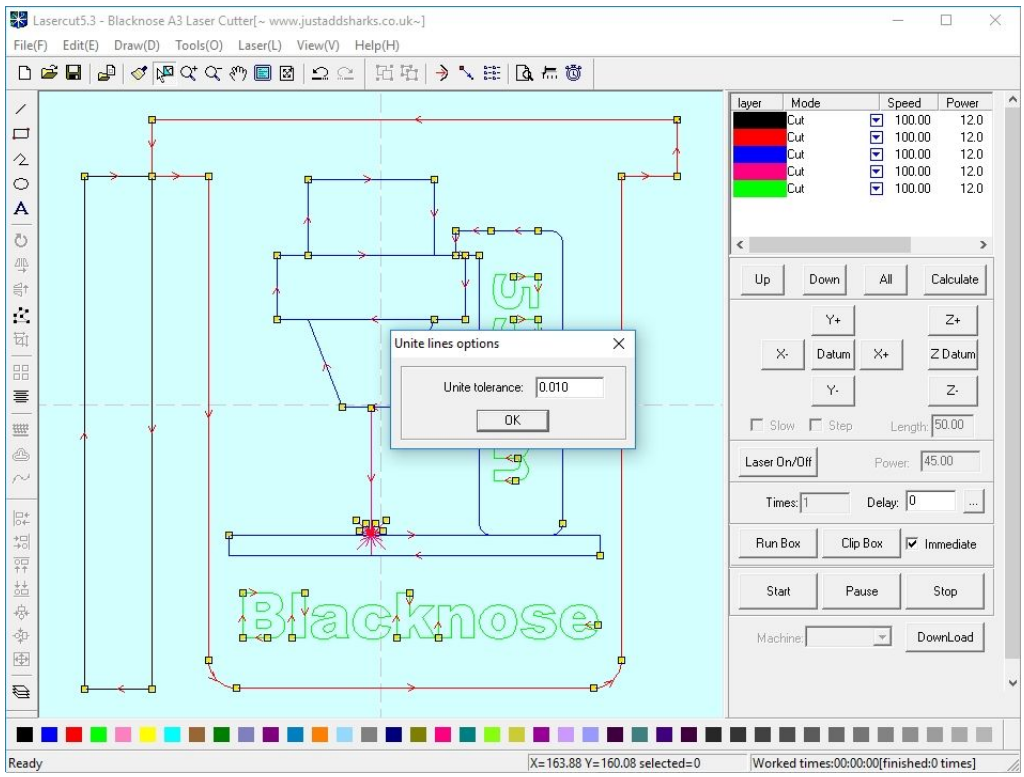
With all the other height tools deleted use the icon at the top of the screen to ‘Zoom to all Objects’. This will show you all the items currently loaded and display them as large as possible on the screen.

This is a good way to identify if there are any small pieces that have escaped deletion in the work area.

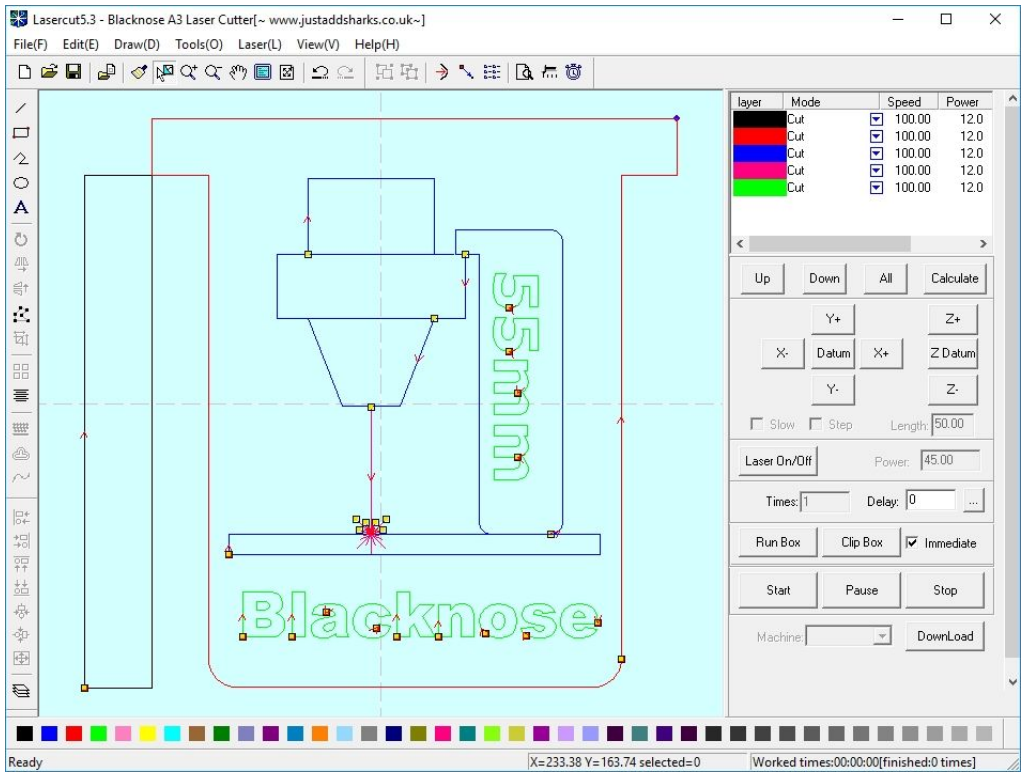
Notice how the black square on the left of the drawing has four small yellow squares on it, one in each corner. These squares indicate the start of a cut line. The Laser cutter is currently planning to cut this square out as four separate lines but it would be much more efficient for the laser to cut this as one single line. Click on ‘Tools’ and then ‘Unite lines’ to open up a dialog window that will connect these lines together.



A small dialog window will open that controls how the lines are united. The dialog window asks for a value that determines how close points on the diagram should be before they are united. Enter a value of '0.01', which means if two points in the drawing are within 0.1mm of each other they will be connected into a single point. Press 'OK' to begin the merge.



You can see that the black square on the left hand side of the drawing now has one single start point and the laser will cut the whole thing in one single motion. This function is particularly important for engraving. During the engraving operation Lasercut needs to figure out which points are inside the square so it can calculate where it needs to engrave. This is how it can tell the the letter ‘o’ should have a ring of black around the outside but not in the middle.



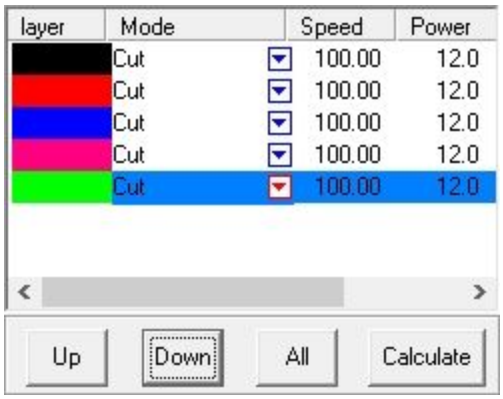
Setting up the layers

In the top right hand corner of LaserCut there is a list of all the layers being used by the current drawing. Each colour from the imported file gets set up as a new layer so you can prepare your file in advance, for the sample height tool there are five different layers.

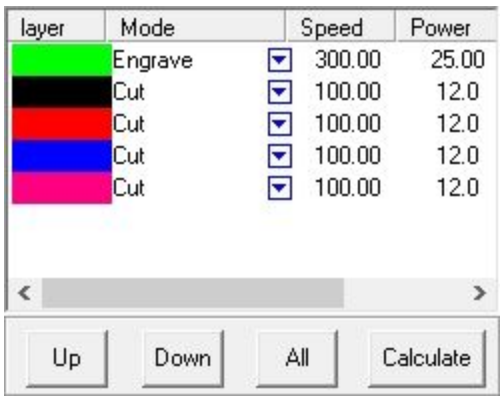
The green layer is intended to be an engrave operation. The text on the height tool will be filled in to make it easy to read.
The blue layer is intended to be a very low power cut that doesn’t go all the way through the material. This is a little diagram that shows you where to put the height tool against the laser.
The pink layer is another low power cut, it is intended to show how to order the layers and determine which parts are cut first.
The red layer is intended to be the outline cut. This will cut all the way through the material and give the actual shape of the height tool.

Black is the default import colour. If there was no colour assigned to the drawing it will be imported as a black line. Setting the black layer to 'no cut' means these items won't be cut. Black lines can be useful for guide lines and bounding boxes.
There are four different types of laser operation that could be applied to a layer.

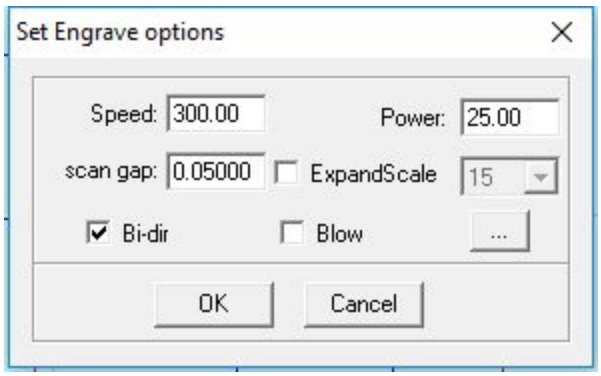
- **Cut:** This drives the laser head along the lines to trace out the item cutting shapes into the workpiece.
- **Engrave:** Drives the laser head from left to right and up a single row at a time over the target area firing a series of dots into the workpiece to leave a darkened area on the item.
- **GradeEngrave:** Primarily used to create thicker walls on rubber stamps, grade engrave puts a slope on the end of each engrave line. It takes a long time to calculate and has little improvement over a normal engrave.
- **Hole:** Puts a single laser dot in the middle of each item, or can be used to create a series of dots along the lines like the perforations of a stamp. While this can be useful it's often easier to import a dotted line in the dxf.



It is often beneficial to perform an engrave operation first particularly when the engraved area is also going to be outlined. The order of operations can be changed simply by selecting the desired layer and moving it up or down in the list. Select the Green layer and move it to the top, changed the mode to an engrave operation.

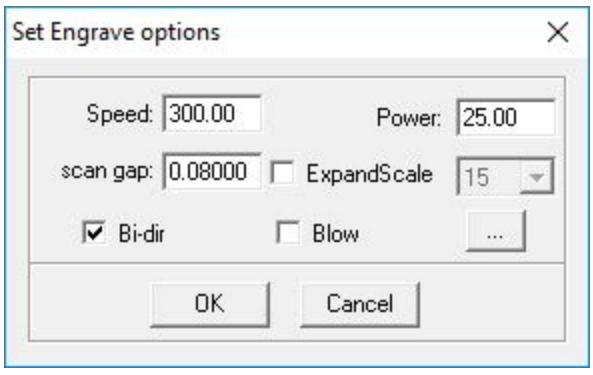


Double click the coloured section of each layer to access the speed and power settings. With the green layer set to to an engrave operation this will option the ‘Set Engrave Options’ dialog.



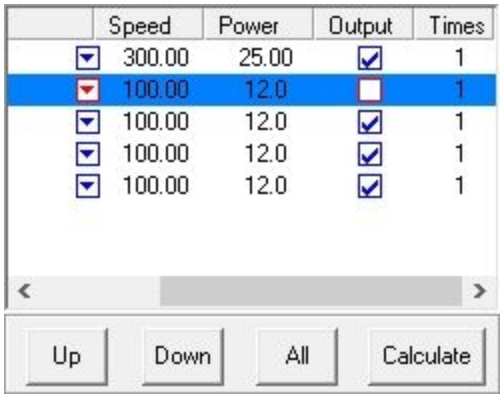
The speed is a value in mm/s that the laser head will accelerate to before engraving. The scan gap is the distance in mm, that the laser cutter will move after each row of the engraving process. It equates directly to the resolution of the image produced. The power is a value of percentage of the maximum tube power.
The darkness and depth of the engrave will vary based on these factors. It the laser head moves more slowly the laser will have more time to burn each spot in the image. If the power is increased the laser will burn each spot faster. If the scan gap is smaller the lines will be closer together and will be more burnt from previous passes. More lines will be needed to cover the same area so the engrave will take longer.

The Blacknose laser cutter is capable of travelling upto 300mm/s during an engrave operation. A scan gap of 0.08mm is a good compromise between speed and resolution. Power of 25% will create an indent about 0.5mm deep on Poplar ply.

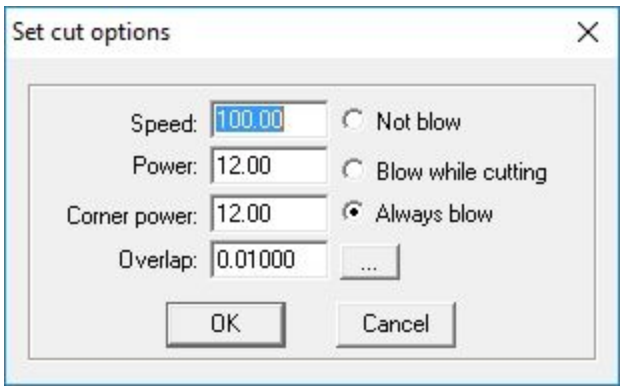


The maximum power for these machines is around 65%. The percentage controls the amount of power being sent to the laser tube and there is a peak in performance around this value. If you increase the power to 75% or 85% there is actually a decrease in cutting efficiency but you are still sending more power through the tube which will wear the tube out faster.

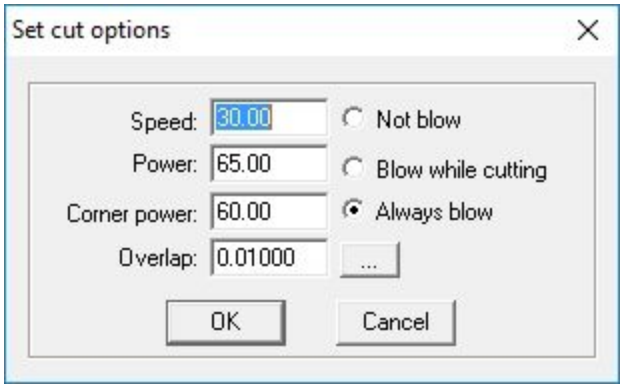
The black layer is intended to not be cut. The settings to turn the layer off are hidden from view in the layer list. It is possible to resize the control panel on the right hand side of the screen but the layer list does not change size. Scrolling to the right reveals two extra columns, output and times. The output column determines if the layer is cut or not, untick the box for the black layer. The final column changes the number of times each layer will be cut but this is a bad idea. Materials should be cut through in a single pass, after the first pass the cut fills up with debris and a second cut is not as effective. If the cut fails multiple passes can be used to complete the cut but this should be a backup plan.



Double click the red layer to access the speed and power settings for a cut operation. This was intended to be high power and cut all the way through the material.

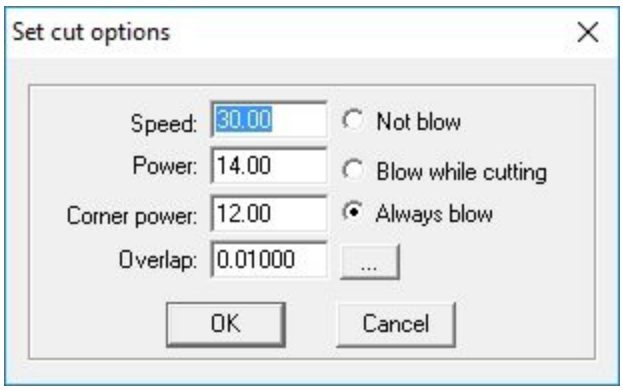


The “Speed:” is a value in mm/s, “Power:” and “Corner power:” are a percentage of the maximum tube power. The speed that the laser head moves during a cut operation is significantly lower than it’s possible speed during the engrave operation. This is because the laser head now has to move in 2 axis at once. The laser cutter has to slow down to change directions and go around corners. The corner power should be set at a lower tube power, so when cutting around a corner the machine ramps down in power (as it is moving slower), preventing the corners of the workpiece appearing to be burnt. It should typically set to 5 to 10% lower than “Power:” which is for normal straight line cutting.



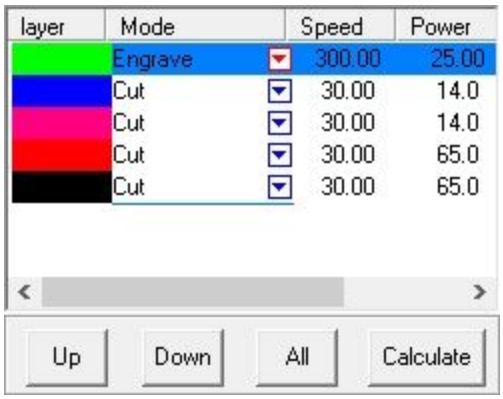
The Blacknose laser cutter is capable of travelling upto 30mm/s during a cut operation. The Power should be set to the maximum, which is around 65% as previously mentioned. The Corner Power should be 60%.

Double click the blue layer to access the speed and power settings for a cut operation. This layer is a line across the surface of the material, rather than all the way though it. The Power should be as low as possible to stop the laser cutting through the material. The laser tube has a minimum power below which is won't turn on at all and this is around 10%. To minimise burning during this operation the speed should be as high as possible.



For the Blacknose laser cutter the speed should be set to 30mm/s. The Power should be set to 14% and the corner power should be 12%. This will mark the surface of the poplar ply but not cut all the way through.

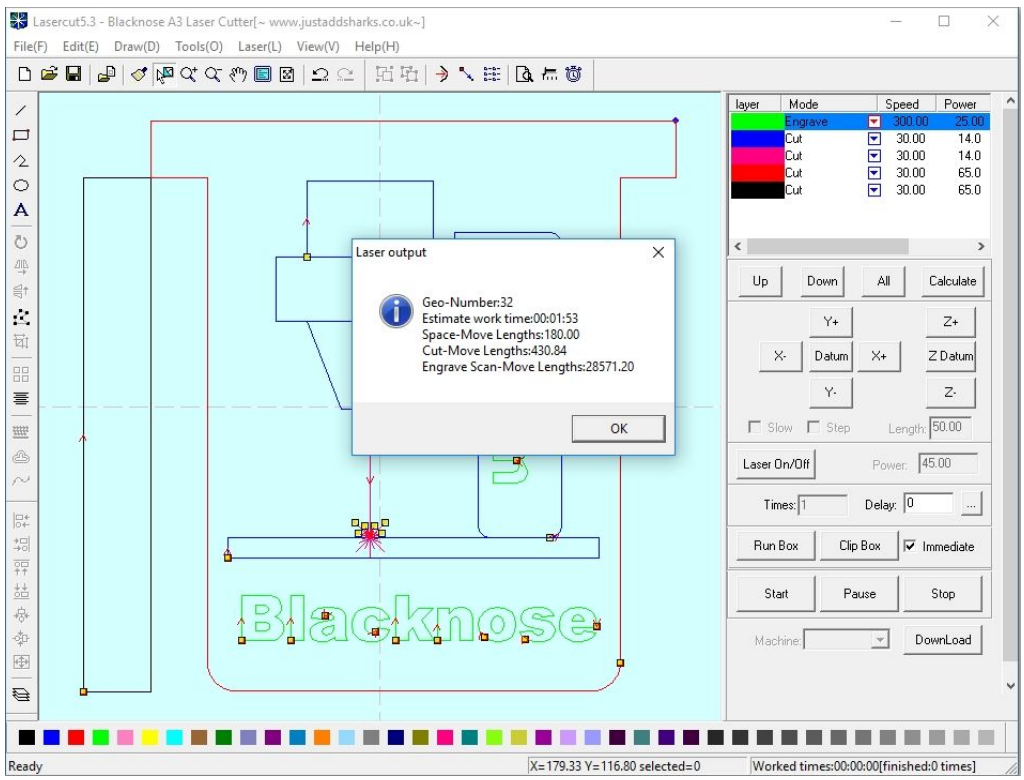
The pink layer is also a low power line cut and should be set up the same as the blue layer. It demonstrates how you can do different parts of the cut in different orders simply by moving the layer above or below the blue layer. Set up all the layers in the following order.



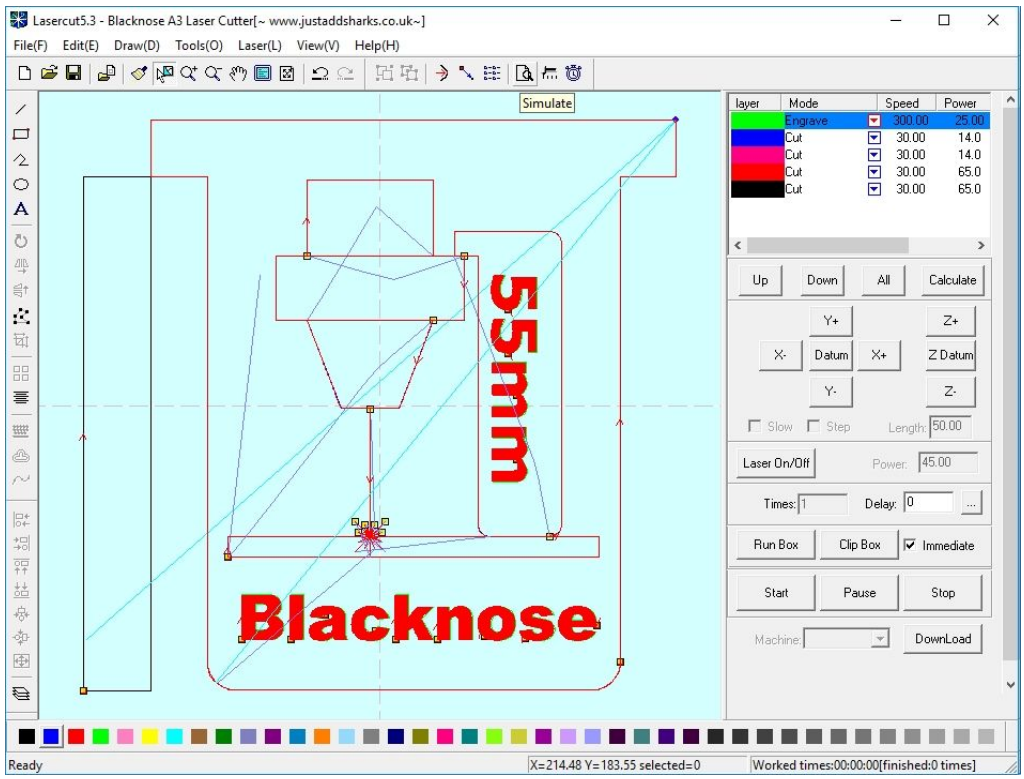
The laser will engrave the lettering first, then draw the image with low power lines and finally cut the whole thing out. It is sensible to cut the whole shape out last because once it has been cut free the shape may move around in the laser causing alignment issues between the cutting operations.

Checking the Operations

Once the layers are set up it is worth checking to make sure the laser will behave correctly. The stopwatch icon along the top of the screen will calculate all the operations the laser has to do and it will provide a time estimate for how long the processes will take. Add 50% to this time for a more accurate estimation.



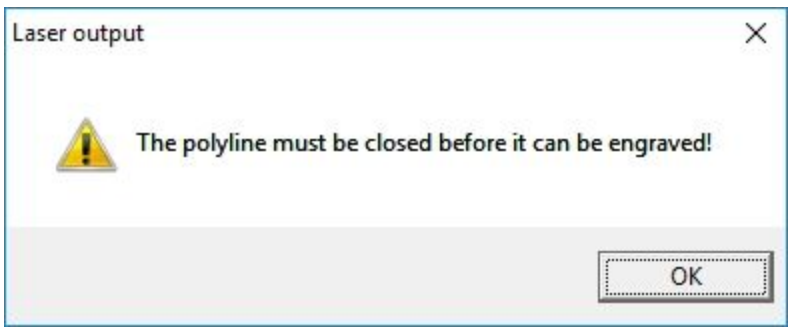
The simulate icon also on the top row will show you the route that the laser cutter intends to take around the workpiece and it will also show you which areas it intends to engrave.



Once the artwork is prepared it can be saved in an ecp file format. This data type stores the settings for each layer so the next time it is loaded it will be ready to cut.

Tips, Mistakes and Problems

Error with Engraving
If you attempt to engrave an object that has not had it's lines united you will get the following error. This means there is a gap in the object you are trying to engrave and it doesn't know where it should be engraving. In most drawings the points will be on top of each other and a value of 0.01mm will be sufficient to close the gap. You can increase the tolerance to close larger gaps but if the tolerance becomes too large you may accidentally connect the wrong points together and deform your shapes.



Scale options

Immediate Mode

Layers Turned Off

Material Rule of Thumb
If you have a new material you have never cut before, try cutting with maximum speed and maximum power. If the material fails to cut all the way through reduce the speed until it does cut. If the material cuts but it burns then reduce the power. An educated guess should be made with thin materials because they can burst into flames with this method, the power can always be increased if it didn't cut the first time.