



**Setup
and
Operation Manual**

v3.1.17

Introduction

Welcome to SheetCam, an affordable but powerful 2 1/2 D CAM program.

SheetCam has been designed to fill a niche in the CAM marketplace by providing an easy to use application for machining 'sheet' goods (metal plates, plastic sheets, thin woods etc.) It will generate the required code for inside and outside contours, pockets and drilling cycles and will work with milling machines, routers, engravers and plasma cutters.

SheetCam accepts data in the form of DXF files (CAD drawings), HPGL files (line art), SVG files, G-code drawings and Excellon files (circuit boards) and has several, configurable, post processors to meet the needs of the many control packages available. Custom post processors can also be written to cope with non-standard applications.

SheetCam will allow the nesting of parts and has features for copying, duplicating, rotating and mirroring parts to reduce wastage to the minimum. SheetCam can also allow for parts that are not aligned perfectly along the machine axes by aligning the drawing to the actual part.

SheetCam will show cutter paths, rapid moves, layers etc. and the part can be rotated in three dimensions in the view panel to check for errors prior to machining.

Document navigation

This PDF file has been created to make navigation to relevant information easier for the user. Please use the 'Bookmarks' on the left to jump to specific chapters and sub-sections. Clicking on a line item in the 'Table of Contents' will take you to that specific page. Clicking on a line item in the 'Index' will take you to that specific topic. **Blue text** is a 'hot link' which will take you to the relevant section of the document.

Work flow

New users are sometimes confused as to the 'correct' way to get SheetCam to manipulate their drawings/files. The information below is intended to be a guide to the optimal steps in the process from drawing to compiled G-code. There are normally three or four steps in any CAD/CAM process:

1. Create your part/design in either a CAD package or an 'art' application like Corel Draw.
2. Open/import the drawing as a DXF, HPGL or Excellon file into the CAM program and set up your tooling sizes and cutting operations. Once these are set run the 'post processor' to generate the required G-code.
3. (Optional but very worthwhile) Run the generated G-code in a CNC simulator (lots of free versions available on the net) to check for any errors. (e.g. CNCSimulator
<http://www.cnccomputer.com>)
4. Once the code has been verified open the G-code file in your CNC control package and run the program to create the part.

Creating drawings

In order to create usable drawings in either a CAD or art application certain fundamental principles **must** be adhered to otherwise SheetCam will not be able to manipulate the resulting DXF, HPGL or Excellon file. An outline to creating acceptable drawings can be found in the '[Hints & Tips/Creating your drawing](#)' section of this document.

Context sensitive help

Clicking on an open palette or settings window within the SheetCam application and pressing the <**F1**> key will open the specific help file for that item.

Table of Contents

<u>Safety</u>	1	<u>Basic tab</u>	42
<u>Initial setup</u>	2	<u>Cut start tab</u>	45
<u>Setup wizard</u>	2	<u>Cut path tab</u>	45
<u>Preferred units</u>	5	<u>Notes Tab</u>	50
<u>Machine type</u>	6	<u>New pocket</u>	51
<u>Setting up the machine</u>	6	<u>Basic tab</u>	51
<u>Machine type tab</u>	7	<u>Notes tab</u>	54
<u>Post processor tab</u>	7	<u>Tapping</u>	54
<u>Working envelope tab</u>	8	<u>Basic tab</u>	54
<u>Table display tab</u>	8	<u>Cut path tab</u>	56
<u>Save/load tab</u>	9	<u>Notes tab</u>	57
<u>Setting up the material</u>	10	<u>Thread milling</u>	58
<u>Material tab</u>	10	<u>Basic tab</u>	58
<u>Work flow procedures</u>	11	<u>Cut path tab</u>	60
<u>Creating drawings</u>	11	<u>Notes tab</u>	60
<u>Outlines must be properly closed</u>	11	<u>Plasma cut</u>	61
<u>So how do I find problems?</u>	12	<u>Basic tab</u>	61
<u>Shapes must not self intersect</u>	12	<u>Cut path tab</u>	63
<u>Explode text in DXF files</u>	12	<u>Notes tab</u>	66
<u>Use layers</u>	12	<u>Drilling</u>	67
<u>Opening drawings</u>	13	<u>Basic tab</u>	67
<u>Open SVG options</u>	13	<u>Cut path tab</u>	69
<u>Open HPGl options</u>	14	<u>Notes tab</u>	70
<u>Open G-code options</u>	14	<u>Insert code</u>	70
<u>Open Excellon options</u>	14	<u>Notes tab</u>	71
<u>Open DXF options</u>	15	<u>Set post variable</u>	71
<u>Recent drawings</u>	15	<u>Basic tab</u>	71
<u>Tool setup</u>	16	<u>Notes tab</u>	71
<u>Rotary tools</u>	16	<u>Move to</u>	72
<u>New mill</u>	16	<u>Basic tab</u>	72
<u>New mill 'Notes' tab</u>	18	<u>Notes tab</u>	73
<u>New drill</u>	19	<u>Post process</u>	73
<u>New drill 'Notes' tab</u>	20	<u>Open and save options</u>	73
<u>New V cutter</u>	21	<u>Job files</u>	73
<u>New V cutter 'Notes' tab</u>	23	<u>Template files</u>	74
<u>New tapping head</u>	23	<u>Part files</u>	74
<u>New tapping head 'Notes' tab</u>	25	<u>Drawing files</u>	74
<u>New rigid tap</u>	25	<u>Toolset files</u>	74
<u>New rigid taps 'Notes' tab</u>	27	<u>Other features</u>	76
<u>New thread mill</u>	28	<u>File Menu</u>	76
<u>New thread mill 'Notes' tab</u>	29	<u>Edit Menu</u>	76
<u>Jet cutting tools</u>	30	<u>View menu</u>	76
<u>New plasma tool</u>	30	<u>Zoom menu</u>	77
<u>New plasma tool 'Notes' tab</u>	31	<u>Options menu</u>	79
<u>New flame tool</u>	32	<u>Job Options</u>	79
<u>New flame tool 'Notes' tab</u>	33	<u>Machine options</u>	85
<u>New waterjet tool</u>	34	<u>Application options</u>	89
<u>New waterjet tool 'Notes' tab</u>	35	<u>Plugins menu</u>	97
<u>New laser tool</u>	36	<u>Help menu</u>	98
<u>New laser 'Notes' tab</u>	37	<u>Toolbars</u>	102
<u>New plate marker tool</u>	38	<u>View toolbar buttons</u>	102
<u>New plate marker tool 'Notes' tab</u>	39	<u>Run post processor toolbar buttons</u>	103
<u>New code snippet</u>	40	<u>Zoom toolbar buttons</u>	103
<u>New code snippet 'Notes' tab</u>	40	<u>Selection toolbar buttons</u>	105
<u>Tool table</u>	41	<u>Calculator button</u>	118
<u>Operation setup</u>	42	<u>Tools toolbar buttons</u>	118
<u>Contour</u>	42	<u>Operations toolbar buttons</u>	119

<u>Lower right toolbar</u>	120	<u>Depth of cut, peck depth etc</u>	129
<u>Plugins</u>	121	<u>Entering values</u>	130
Autoload Plugin	121	Finish allowance	130
Batch Update Plugin	122	Machine and material parameters	131
Calculator Plugin	122	Mouse wheel	132
DXF plugin	123	Zooming and panning	132
Excelon plugin	123	<u>Tutorials</u>	132
G-code plugin	124	Machine setup	133
HPGL plugin	124	Material setup	133
Mach plugin	125	Profile tutorial	133
Simulation Plugin	126	Pocketing tutorial	139
SVG plugin	127	Jet Cutting tutorial	141
Text plugin	127	Flame cutting options	147
<u>Hints and tips</u>	128	Waterjet cutting options	147
Centre drilling	128	Laser cutting options	148
Context sensitive help	128	Plate marker options	148
Copy and duplicate differences	128	<u>Nesting tutorial</u>	149
<u>Creating your drawing</u>	128	<u>Frequently asked questions (FAQ)</u>	151

Safety

Care **must** be taken when using computer controlled equipment to ensure the safety of the operator, bystanders and the equipment itself. While SheetCam does not directly control the machine (except in special circumstances) the post processor code generated by SheetCam is used by other software that in turn controls the machine operation.

It is strongly advised that **all** post processor files are run through a CNC simulator of some sort to check for errors **prior** to running them on the machine in question. CNC simulators are available as 'freeware' at a number of sites on the internet and can be part of the control software itself. 'Stable Design' cannot be held responsible for **any damage** caused by incorrect programming by the user.

Initial setup

Before you can generate G-code SheetCam needs to know certain facts about your specific machine and unit preferences in order to calculate the required tool paths. When you install SheetCam for the first time the 'Setup Wizard' will start and ask a number of questions. If you need to change any of the parameters or if you entered something incorrectly the various parameters can be found under the 'Options' menu on the main toolbar. Let's look at the 'Setup Wizard' first and then look at what can be changed under the 'Options' menu.

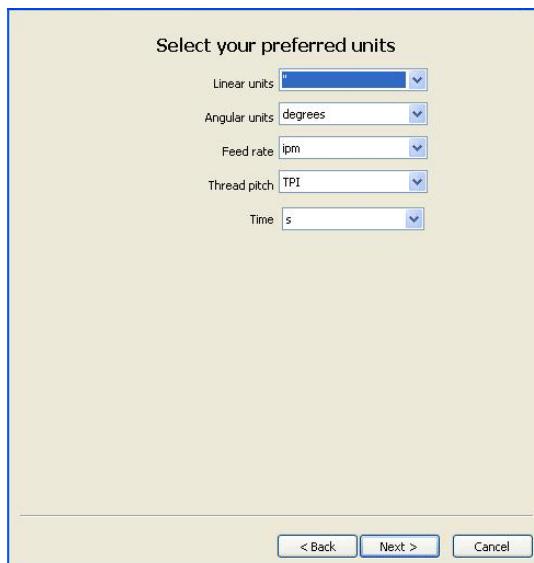
Setup wizard



The first screen you will see is the 'Welcome' screen.

Click on <Next> to proceed.

The second screen of the 'Setup Wizard' deals with the preferred units.



Linear units

Enter the linear units from the large drop down menu.

Angular units

Enter the angular units from the drop down menu.

Feed rate

Enter the feed rate units from the drop down menu.

Tread pitch

Enter the thread pitch units from the drop down menu.

Time

Enter the time units from the drop down menu.

After making your choices click <Next> to move onto the next option.

Note: You can use any units when entering values into any edit boxes (with the exception that linear edit boxes must have linear units and angular edit boxes must have angular units). If you do not specify any units then the preferred units (as set above) are used. If you specify any of the units in the list they will be converted to your preferred units as soon as you leave the edit box.

Example 1:

If your preferred units are inches and you type <10mm> into a cut depth box the value will change to 0.3937 inches as soon as you leave the box.

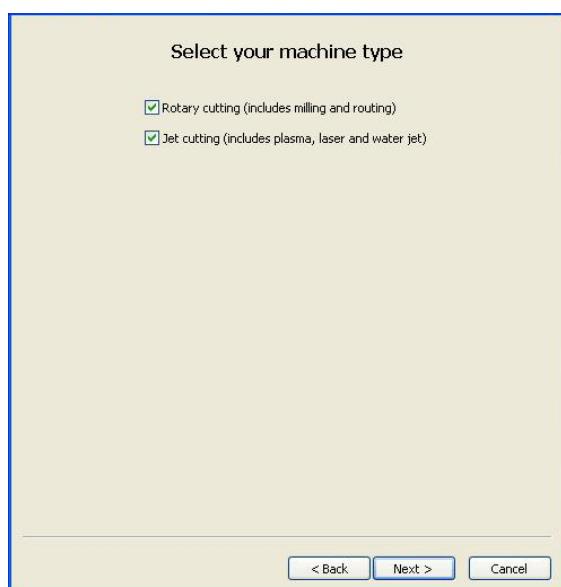
Example 2:

If your preferred units are mm and you type <1"> into a cut depth box the value will change to 25.4mm as soon as you leave the box.

Example 3:

If your preferred units are mm and you type <1 inch> into a cut depth box the value will change to 25.4mm as soon as you leave the box.

The third screen of the 'Setup Wizard' deals with your 'machine type'.



Note: At least **one** option **must** be selected at all times.

Rotary cutting

Checking this item will toggle the rotary cutting option on or off. When checked the rotary cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden. Rotary cutting includes milling, routing, drilling, tapping etc.

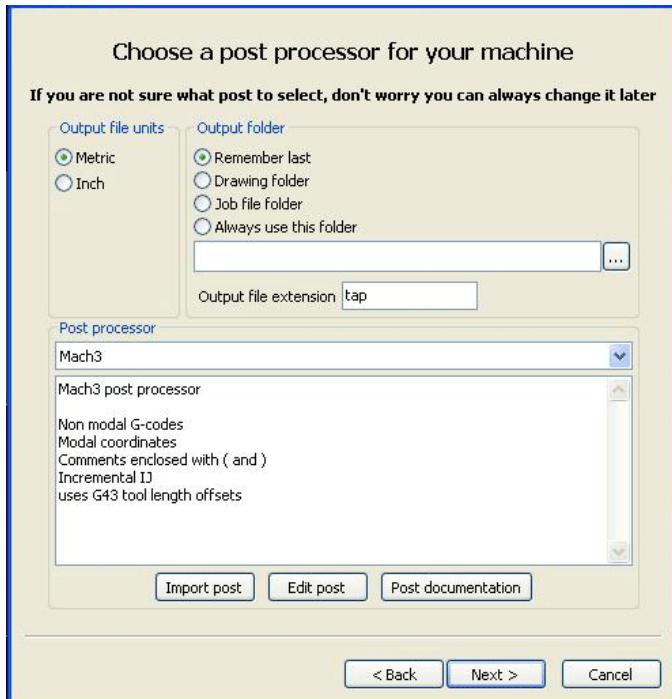
Jet cutting

Checking this item will toggle the jet cutting option on or off. When checked the jet cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective

buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden. Jet cutting includes plasma, laser, flame, water jet etc.

Note: Check both boxes if you want rotary controls **and** jet cutting controls available under the 'Tools' menu and the 'Operation' menu.

The fourth screen of the 'Setup Wizard' deals with 'Post processor' selection.



Select the correct processor for your machine controller. A brief description of the processor file will appear in the lower window (if available).



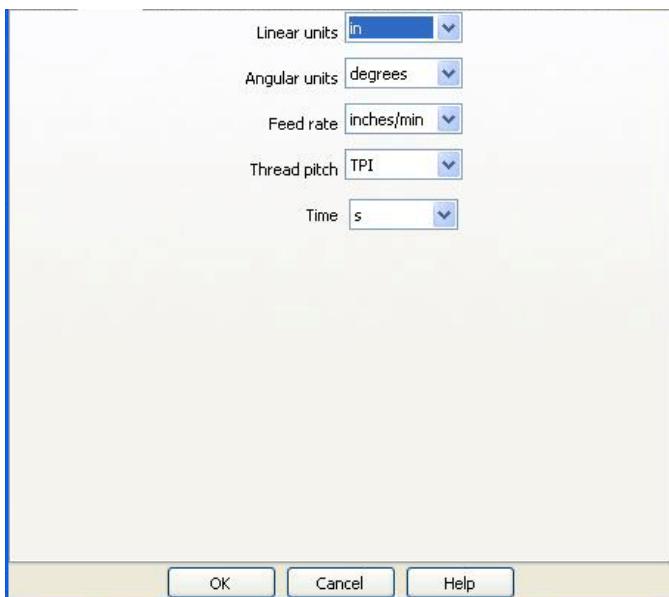
The fifth screen of the 'Setup Wizard' completes the setup process.

Click <Finish> to continue.

Now let's look at where you can enter the same information in the 'Options' menu.

Preferred units

Select '**Options/Application options**' from the main tool bar and ensure the '**Units**' tab is 'active' as shown below:



This dialog box is used to set up the 'base' or 'preferred' units used by many of SheetCam's functions.

Linear units

Enter the linear units from the large drop down menu.

Angular units

Enter the angular units from the drop down menu.

Feed rate

Enter the feed rate units from the drop down menu.

Thread pitch

Enter the thread pitch units from the drop down menu.

Time

Enter the time units from the drop down menu.

Note: You can use **any units** when entering values into **any** edit boxes (with the exception that linear edit boxes must have linear units and angular edit boxes must have angular units). If you do not specify any units then the preferred units (as set above) are used. If you specify any of the units in the list they will be converted to your preferred units as soon as you leave the edit box.

Example 1:

If your preferred units are inches and you type <10mm> into a cut depth box the value will change to 0.3937 inches as soon as you leave the box.

Example 2:

If your preferred units are mm and you type <1"> into a cut depth box the value will change to 25.4mm as soon as you leave the box.

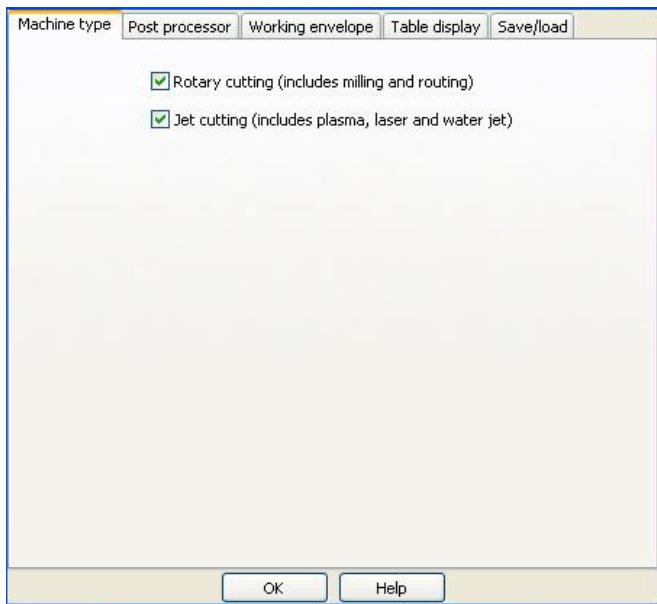
Example 3:

If your preferred units are mm and you type <1 inch> into a cut depth box the value will change to 25.4mm as soon as you leave the box.

Machine type

Select '**Options/Machine**' from the main tool bar and ensure the '**Machine type**' tab is 'active' as shown below:

Note: At least **one** option **must** be selected at all times.



Rotary cutting

Checking this item will toggle the rotary cutting option on or off. When checked the rotary cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden. Rotary cutting includes milling, routing, drilling, tapping etc.

Jet cutting

Checking this item will toggle the jet cutting option on or off. When checked the jet cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective

buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden. Jet cutting includes plasma, laser, flame, water jet etc.

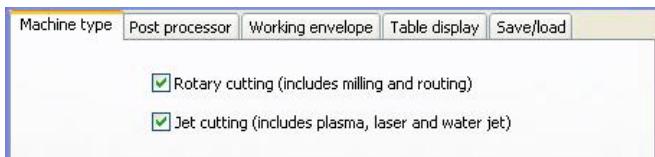
Note: Check both boxes if you want rotary controls **and** jet cutting controls available under the 'Tools' menu and the 'Operation' menu.

Setting up the machine

Make sure the '**Show machine and work**' button  is active (the button is active when it has a grey lined box around it). This displays the machine and the work material in the main window.

Now click on the '**Options/Machine**' menu item. A pop-up window will appear with several tabs that allow control of the various functions.

Machine type tab



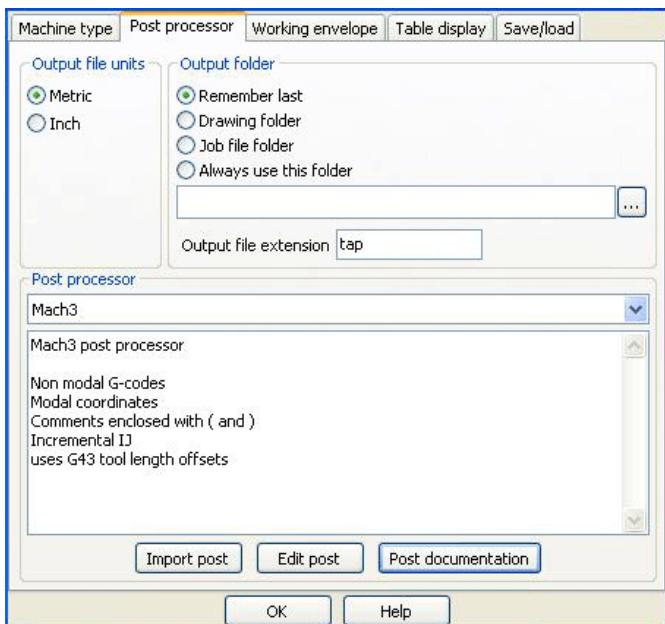
Select the check box most applicable to your operation.

If you have a milling machine or router select '**Rotary cutting**'.

If you have a jet cutter of some sort (i.e. plasma, laser, water etc.) then select '**Jet cutting**'.

If you want SheetCam to output G-code for both types of machine then select **both** rotary and jet cutting.

Post processor tab



In the '**Output file units**' section select the correct unit radio button depending on your machine's requirements.

Note: The units selected in the '**Output file units**' section only affect the generated code. They do not alter the units used in SheetCam.

In the '**Output folder**' section select your preferred location using the radio buttons.

In the '**Post processor**' section select the correct processor for your machine using the drop down menu.

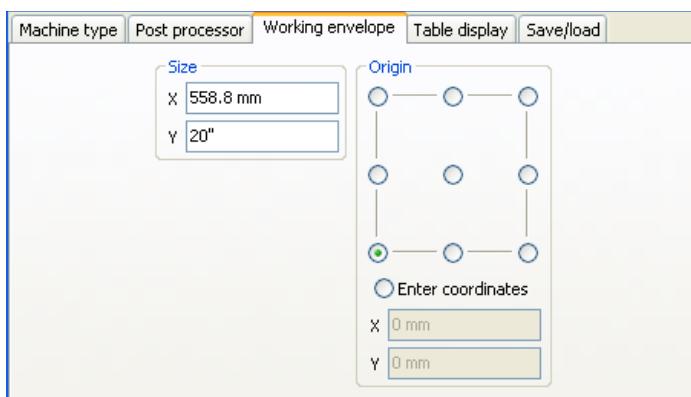
Note: While G-code is supposedly a standard, most machines speak different 'dialects' of G-code. To get around this problem SheetCam uses 'post processors'. These are little programs that run inside SheetCam and create code customized for particular machines.

The '**Import post**' button allows you to import additional post processor files if required.

The '**Edit post**' button allows you to edit the currently selected post processor.

The '**Post documentation**' button opens a pop-up window that provides information on how to write post processor files for those who need to make custom files.

Working envelope tab



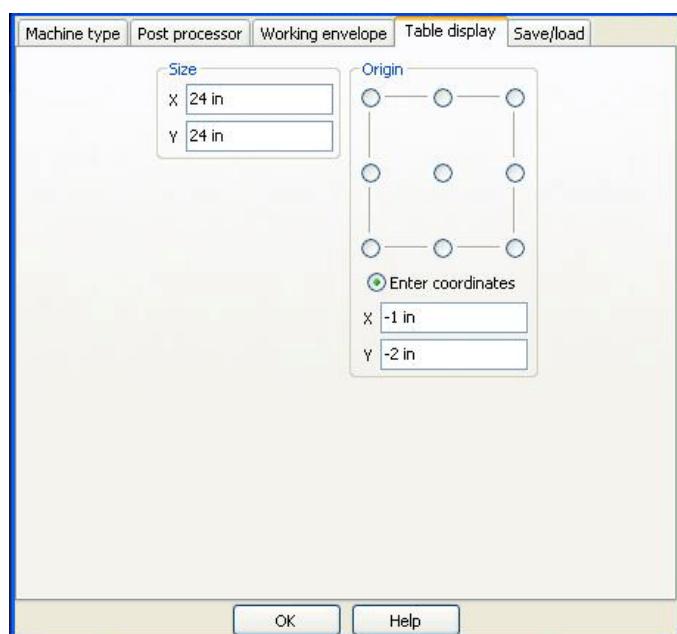
tells SheetCam that the origin is the bottom left hand corner.

The '**Working Envelope**' is the usable area so enter <22"> in the 'X' size and <20"> in the 'Y' size (include the " to tell SheetCam these measurements are in inches) as shown.

Note: This item is purely for reference and it does not affect the G-code in any way. If you do not wish to use this feature set the sizes to 0, 0.

Table display tab

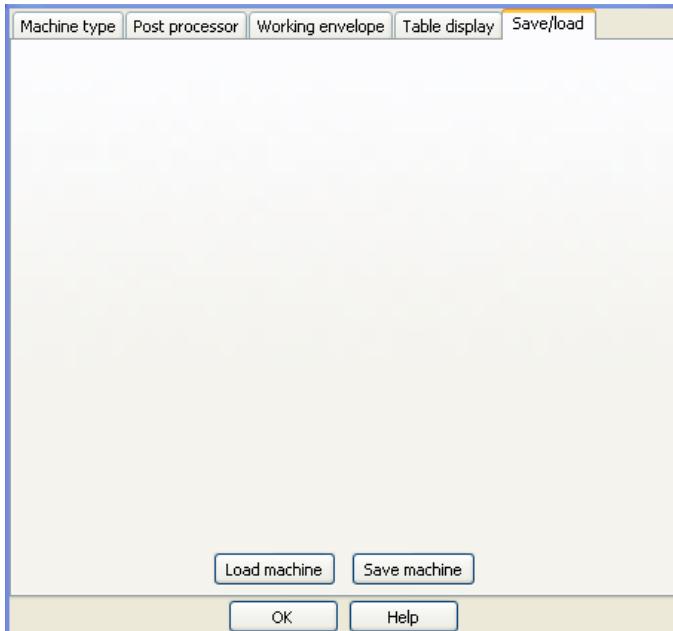
The table is larger than the actual usable area so we need to enter the size and position of the table. Let's assume the '**Working Envelope**' origin (0,0) is 1" x 2" away from the corner of the table (**you should obviously substitute the correct dimensions/values for your machine**). Click on the 'Enter coordinates' radio button in the 'Origin' section and enter <-1"> and <-2"> for the 'X' and 'Y' coordinates of the table bottom left corner and enter <2 feet> for the 'X' and 'Y' dimensions in the 'Size' section as shown.



Note: As soon as you exit each edit box the value is converted to millimetres for you. Notice too that the main drawing display changes as you change the dimensions so that you get an idea what the table looks like.

Note: This item is purely for reference and it does not affect the G-code in any way. If you do not wish to use this feature set the sizes to 0, 0.

Save/load tab



You can save your machine setup for future use or if you use SheetCam to output G-code to a number of different machines you can save each setup for later recall. Click on '**Save machine**' to call up a save dialog box or '**Load machine**' to load a previously saved file.

When you have made all the changes outlined above click on the <OK> button at the bottom of the dialog box to save your changes.

Setting up the material

Click on the '**Options/Job options**' menu item. A pop-up window will appear with several tabs that allow control of the various functions. The display will zoom and rotate to fit the last specified material.

Material tab

For the purposes of this example our material is a 300mm x 200mm sheet of 6mm plastic that is clamped to the table with it's bottom left hand corner at X=50mm,Y=50mm. The sheet is clamped on to a piece of 0.25in board to allow the cutter to cut right through without damaging the table. **You should obviously substitute the correct dimensions/values for your material.**

Enter the dimensions and position of the material as shown below. The main display will re-size to show the material as you enter the values. To be safe we will run the cutter 4mm above the panel when moving between cuts so enter '4' in the '**Rapid clearance**' box. The sacrificial board is 0.25in thick so enter '**0.25in**' for the height of the work above the table.

Plunge safety clearance is used when doing a rapid move to cut height. When SheetCam moves to the top of the cut it stops slightly short and moves the remaining distance at feed rate. This allows for inaccuracies in your setup and any chips that may be in the way. The last thing you want is to slam the tool into the work at full speed. By default the clearance is 0.5mm which seems to work pretty well in most cases. Older versions of SheetCam have always had this hard coded to 0.5mm. Enter your required distance in the box.

For convenience you can save your material setup for later use if you use a lot of the same type/size material. Click on the '**Save material**' to call up a save dialog box or '**Load material**' to load a previously saved file.

Now click on <OK> to save your changes.

Work flow procedures

New users are sometimes confused as to the 'correct' way to get SheetCam to manipulate their drawings/files. The information below is intended to be a guide to the optimal steps in the process from drawing to compiled G-code. There are normally three or four steps in any CAD/CAM process:

1. Create your part/design in either a CAD package or an 'art' application like Corel Draw.
2. Open/import the drawing as a DXF, HPGL or Excellon file into the CAM program and set up your material sizes, tooling sizes and cutting processes. Once these are set run the 'post processor' to generate the required G-code.
3. (Optional but very worthwhile) Run the generated G-code in a CNC simulator (lots of free versions available on the net) to check for any errors.
4. Once the code has been verified open the G-code file in your CNC control package and run the program to create the part.

Creating drawings

In order to create usable drawings in either a CAD or art application certain fundamental principles must be adhered to otherwise SheetCam will not be able to manipulate the resulting file.

Outlines must be properly closed

A closed shape is any shape where the lines completely enclose an area. A square is a simple example of a closed shape. It has an inside and an outside so SheetCam needs to work out which side you want to cut. A simple line is an open shape as there is no inside or outside therefore SheetCam treats it differently.

So what is meant by 'properly closed'?

Let's take a simple example. You draw a rectangle using four lines. However two of the lines don't quite meet. On the screen it looks fine unless you zoom in very close. When you load the drawing into SheetCam it will recognize that two ends don't meet and it will assume that the shape is open so you can't cut it out. All open lines will be shown on the screen in grey (if you haven't changed the default colours). If your lines are very close (as defined under 'Options/Application options/Drawing import tab/Import link tolerance') SheetCam will automatically move the ends so that they touch.

Note: This should not be used to compensate for poor drawings!

So how do I find problems?

Turn off the show segment ends button  and turn on the show path ends button 

Note: 'Snaps' are your best weapon against problems like this. Use 'grid' snaps and 'end' snaps whenever possible. They are the best way to create accurate drawings. If you are not familiar with using snaps then look in the documentation for your CAD/drawing program. Virtually all CAD/drawing programs have some form of snap capability.

Shapes must not self intersect

A good example of a self intersecting shape is the figure 8. In the middle of the 8 the lines cross over each other. Imagine a path in the shape of an 8 and you are walking along it. At one point the outside is on your left and another point the outside is on your right. There is no rational way to cut this shape. SheetCam tries but the results will, almost undoubtedly, not be what you are after.

Explode text in DXF files

DXF files cannot handle 'True Type' fonts very well as there is no 'standardized' format to follow. To guarantee that the cut path is exactly the same as your drawing you need to convert any text into lines. This is often called 'exploding'. If you are not familiar with 'exploding' text look in the documentation for your CAD/drawing program to find out more. If your CAD/drawing package supports HPGL export then try exporting your drawing as HPGL. Most CAD/drawing packages will automatically convert text to lines when exporting HPGL.

Note: Some drawing programs have an option to 'convert text to lines' rather than having an 'explode' function but the end result is the same, text blocks are converted to lines that SheetCam will recognize.

Use layers

If you want to perform more than one operation on your part then you will need to separate your drawing into layers. Each time you run a process in SheetCam the process is applied to everything on that layer. You can create your drawing using 'layers' in CAD programs and in 'most' drawing programs (check your CAD/drawing program documentation for more information).

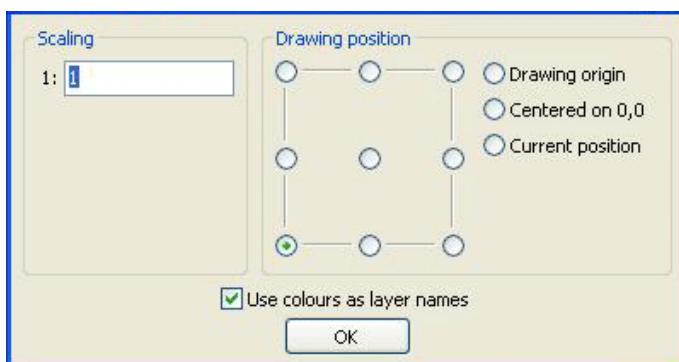
Note: If your drawing is all on one layer you can use SheetCam to split different processes into their own layers. Use the 'Edit contour' window to move parts to a different layer by first selecting the 'Edit contour properties' button. Then click on a drawing section to select it and right-click to call up the 'Edit contour' window. Select either a new layer (and provide a unique name) or an existing layer to move the selection to.

Opening drawings

Selecting '**Import drawing**' from the '**File**' menu opens a pop-up window allowing you to browse for your drawing file. Acceptable formats are SVG, HPGL, G-code, Excellon or DXF files. The formats are selectable from the drop down menu. Each particular format will open its own specific settings dialog window when you select the file and click on <**Open**>.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

Open SVG options



Scaling

Set the scale ratio in the box.

Drawing position

Click one of the position radio buttons or select the origin location from the following options:

Drawing origin

The origin specified in the drawing is used as the machine 0,0 position. The position of the origin in the drawing varies between different cad packages. Some use the page bottom left hand corner, some use the page centre and some use the origin (sometimes called the datum) as it is placed in the drawing.

Centered on 0,0

The centre of the drawing is aligned to the machine 0,0 position.

Current Position

The centre of the new drawing is placed in exactly the same position as the current drawing.

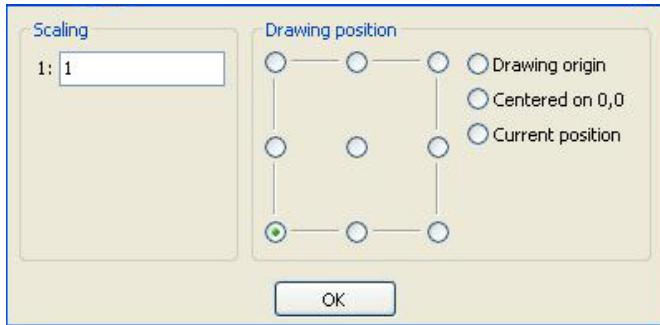
Use colours as layer names

Check this box to use colours in the drawing as layer names.

Note: The 'Scaling' and 'Drawing position' items above are applicable to all drawing import dialog boxes.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

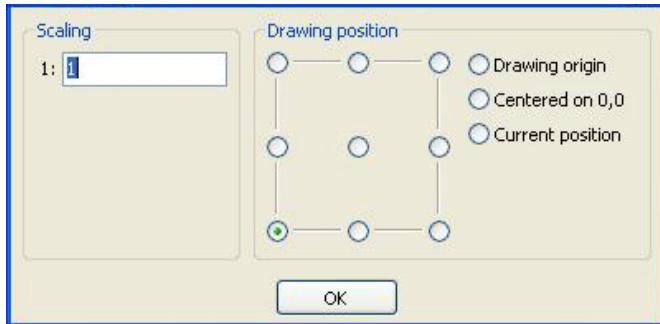
Open HPGL options



For 'Scaling' and 'Drawing position' information see 'Open SVG options' above.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

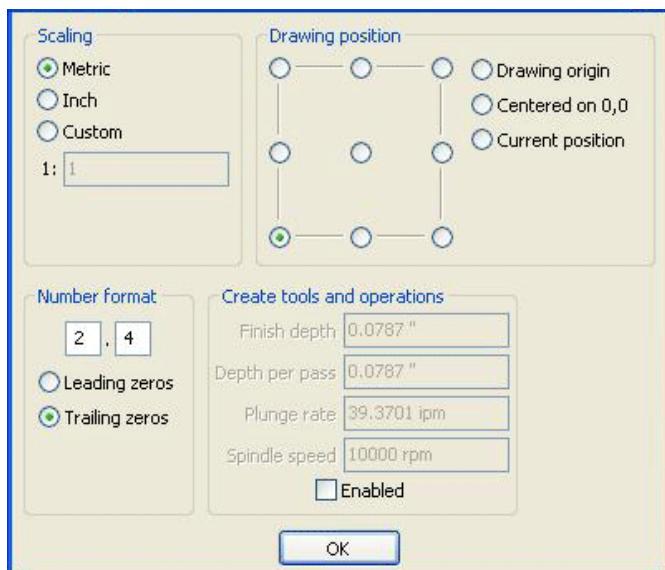
Open G-code options



For 'Scaling' and 'Drawing position' information see 'Open SVG options' above.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

Open Excellon options



For 'Scaling' and 'Drawing position' information see 'Open SVG options' above.

Number format

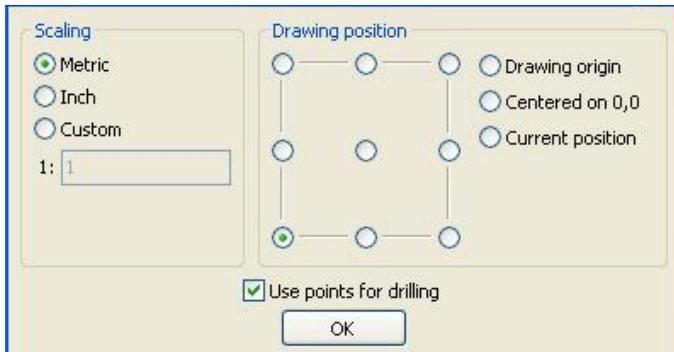
Use the boxes to set the number format and the radio buttons to select either leading zero or trailing zero suppression.

Create tools and operations

If this section is 'Enabled' then SheetCam can set up the tool table and operations to run the job. With this option turned on you can simply load a file and immediately post process, without having to do anything else.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

Open DXF options



For 'Scaling' and 'Drawing position' information see '[Open SVG options](#)' above.

Use points for drilling

Check this option if you marked the hole locations with 'points' in your original drawing.

Note: For tips on creating suitable drawings see the '[Hints and Tips](#)' section.

Recent drawings

Opens a pop-up window showing the last five drawings opened.

Tool setup

Once you have the machine and material information entered and a drawing selected you can define the tools you intend to use. Tools are selected by clicking on the 'Tools' menu item, select the correct type of tool from the drop down menu.

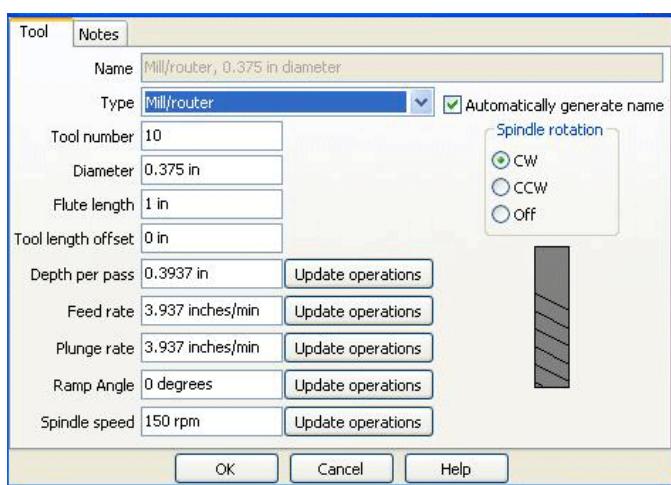


Note: The tools shown in the drop down menu are based on the choices made in the 'Options/Machine' menu.

Note: Tools can also be selected by clicking on the required tool button on the left of the main screen.

Rotary tools

New mill



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct tool type from the drop down menu shown below.



Note: This page shows setting options for a new mill/router (rotary cutter). To see the other tool setup pages refer to the relevant sections below.

operations.

Note: A mill/router can plunge and cut sideways. It can be used by the contour, drill and pocket

Note: A drill can plunge but cannot cut sideways. It can only be used by the drill operation.

Note: A V cutter is offset depending on the angle and depth. The deeper you cut, the more the cut path is offset.

Note: An automatic tapping head automatically reverses when you retract the spindle.

Note: A rigid tapping head does not reverse when you retract the spindle, the spindle is reversed instead.

Note: A thread mill is a single or multi-point thread cutting tool.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Depth per pass

Enter the depth here. This function controls the depth of cut per pass.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Feed rate

Enter the required feed rate here.

Plunge rate

Enter the plunge rate here.

Ramp angle

'Ramping' allows the cutter to enter the work while travelling in a forward direction. This reduces the load on the Z axis and also allows you to plunge with a 'non-centre cutting' tool (i.e. some 4-flute end mills). The cutter will travel forwards as it plunges, cutting a ramp into the work. This ramp is then machined away. If you are ramping into an open contour or between tabs then the cutter ramps down then backs up to the start of the cut, reverses up to the start point then reverses again and carries on to the end. If the contour is open then the cutter simply carries on at the end of the cut until the ramp is machined away. For small contours the cutter will spiral down to depth. The ramp angle specifies the angle at which the cutter will plunge into the work . 0 degrees or 90 degrees are straight down (no ramping), 5

degrees is very shallow and 85 degrees is very steep.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software.
However, the post processor will issue a warning if the speed is set to zero (0).

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operations** that use this tool.
For instance this will reset the spindle speed, feed rate, ramp angle etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

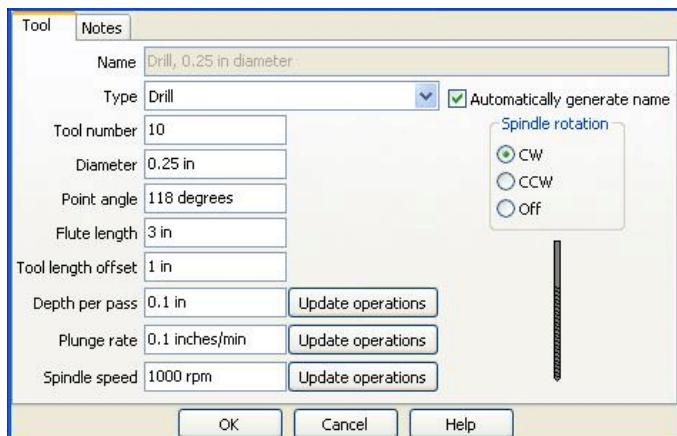
Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

New mill 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New drill



below is not checked.

Type

Select the correct tool type from the drop down menu shown below.



Note: This page shows setting options for a new drill (rotary cutter). To see the other tool setup pages refer to their relevant sections.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Point angle

Enter the point angle here.

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box

Depth per pass

Enter the depth here. This function controls the depth of cut per pass.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Plunge rate

Enter the plunge rate here.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software. However, the post processor will issue a warning if the speed is set to zero (0).

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the spindle speed, feed rate, etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

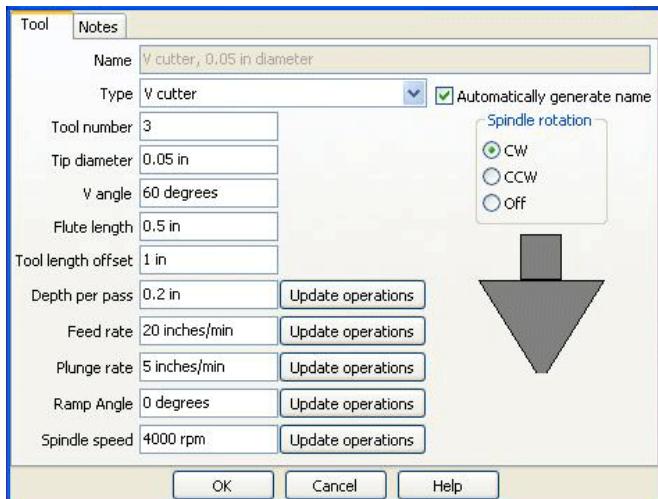
Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

New drill 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New V cutter



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

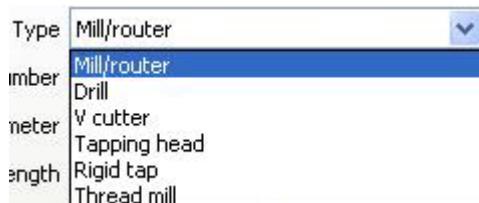
Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct tool type from the drop down menu shown below.



Note: This page shows setting options for a new V cutter (rotary cutter). To see the other tool setup pages refer to their relevant sections.

Note: See '[New mill](#)' for a full description of the various tool types.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Tip diameter

Enter the tip diameter of the tool here.

V angle

Enter the angle of the tool tip here.

Note: This value is used in the corner sharpening function (see '[Operations\Contour\Cut path tab](#)')

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Depth per pass

Enter the depth here. This function controls the depth of cut per pass.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Feed rate

Enter the required feed rate here.

Plunge rate

Enter the plunge rate here.

Ramp angle

'Ramping' allows the cutter to enter the work while travelling in a forward direction. This reduces the load on the Z axis and also allows you to plunge with a 'non-centre cutting' tool (i.e. some 4-flute end mills). The cutter will travel forwards as it plunges, cutting a ramp into the work. This ramp is then machined away. If you are ramping into an open contour or between tabs then the cutter ramps down then backs up to the start of the cut, reverses up to the start point then reverses again and carries on to the end. If the contour is open then the cutter simply carries on at the end of the cut until the ramp is machined away. For small contours the cutter will spiral down to depth. The ramp angle specifies the angle at which the cutter will plunge into the work . 0 degrees or 90 degrees are straight down (no ramping), 5 degrees is very shallow and 85 degrees is very steep.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software. However, the post processor will issue a warning if the speed is set to zero (0).

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool

diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

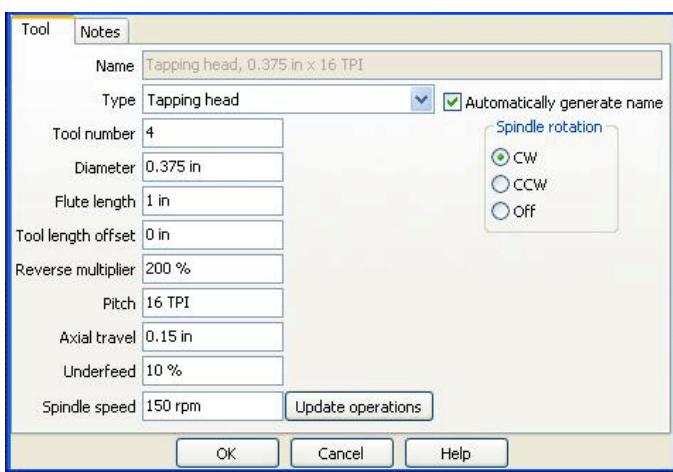
Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

New V cutter 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New tapping head



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

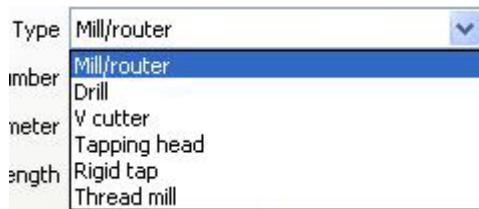
Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct tool type from the drop down menu shown below.



Note: This page shows setting options for a new tapping head (rotary cutter). To see the other tool setup pages refer to their relevant sections.

various tool types.

Note: See '[New mill](#)' for a full description of the

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Reverse multiplier

Normally tapping heads spin faster in reverse than forwards. If your reverse speed is 1.5x as fast as the plunge speed then use a value of 150%, if it is twice as fast use 200%.

Pitch

Enter the thread pitch if you are using a metric machine or the threads per inch (TPI) if you are using an imperial machine.

Axial travel

An automatic tapping head (e.g. TapMatic) automatically reverses when you retract the spindle. There is a small amount of backlash when you start to retract as the reverse clutch engages. This value is that backlash.

Underfeed

If you feed faster than the tap cuts you will either break the tap or strip the thread. The worst that can happen if you underfeed is that the clutch in the head disengages for a short time. It is customary to feed slightly slower than the theoretical speed to make sure you don't overfeed. Normally 5 - 10% is enough. If the tapping head 'chatters' while tapping, reduce the amount of underfeed.

Spindle speed

Enter the spindle speed here.

Note: Spindle speed **must** be entered. It is used with the tap pitch to work out the feed rate.

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

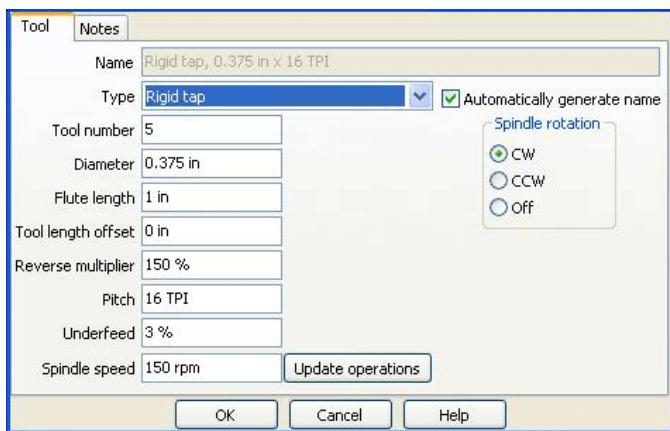
New tapping head 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New rigid tap

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct tool type from the drop down menu shown below.

Type	Mill/router	<input checked="" type="button"/>
Number	Mill/router	
Material	Drill	
Material	V cutter	
Material	Tapping head	
Material	Rigid tap	
Material	Thread mill	

Note: This page shows setting options for a new rigid tap (rotary cutter). To see the other tool setup pages refer to their relevant sections.

Note: See '[New mill](#)' for a full description of the various tool types.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Reverse multiplier

To speed up the tapping cycle you can use a higher spindle speed when retracting. Make sure you don't exceed the machine's maximum rated spindle speed (spindle speed x reverse multiplier).

Pitch

Enter the thread pitch if you are using a metric machine or the threads per inch (TPI) if you are using an imperial machine.

Underfeed

Rigid tap holders normally have some axial travel to allow for slight variations in spindle speed. It is sometimes useful to feed slightly slower than the ideal to prevent overfeeding and breaking the tap. Normally you would use 0 - 3%.

Spindle speed

Enter the spindle speed here.

Note: Spindle speed **must** be entered. It is used with the tap pitch to work out the feed rate.

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

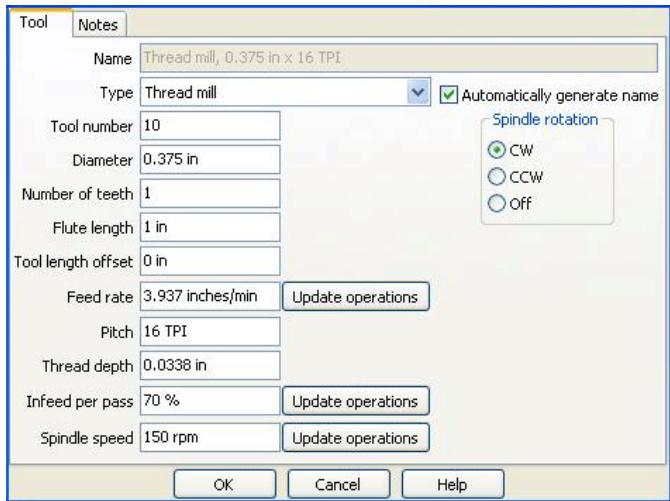
Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

New rigid taps 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New thread mill



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Note: This menu item is only visible if 'Rotary cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct tool type from the drop down menu shown below.



Note: This page shows setting options for a new thread mill (rotary cutter). To see the other tool setup pages refer to their relevant sections.

Note: See '[New mill](#)' for a full description of the various tool types.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Number of teeth

Enter the number of teeth on the tool here.

Flute length

Enter the tool flute length here (i.e. the length from the tip of the tool to the end of the cutting flute).

Tool length offset

Enter the amount of tool offset here. This should only be used if the machine controller does not have its own tool length compensation.

Note: Tool length offset is the length of the tool from a particular reference point. This may be the spindle nose or any other convenient reference point. It is used to compensate for different length tools. The post processor may use this value in one of two ways: If the controller supports tool length offsets then they are used. If not the offset is added to the Z position.

Feed rate

Enter the required feed rate here.

Pitch

Enter the thread pitch if you are using a metric machine or the threads per inch (TPI) if you are using an imperial machine.

Infeed per pass

Infeed per pass controls the depth of cut. 100% will cut the thread in one pass. 70% would cut 70% in one pass and 30% in the last pass. 25% would cut four equal passes.

Spindle speed

Enter the spindle speed here.

Note: Spindle speed **must** be entered. It is used with the tap pitch to work out the feed rate.

Spindle rotation

Select the spindle direction from the available choices.

Note: These items only need to be checked if your spindle motor is controlled via software.

Note: The spindle direction also affects the cut direction. For instance if climb cutting, outside contours are cut clockwise if the spindle is clockwise or anticlockwise if the spindle is anticlockwise.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create new rotary tool' button  located in the left-hand vertical toolbar.

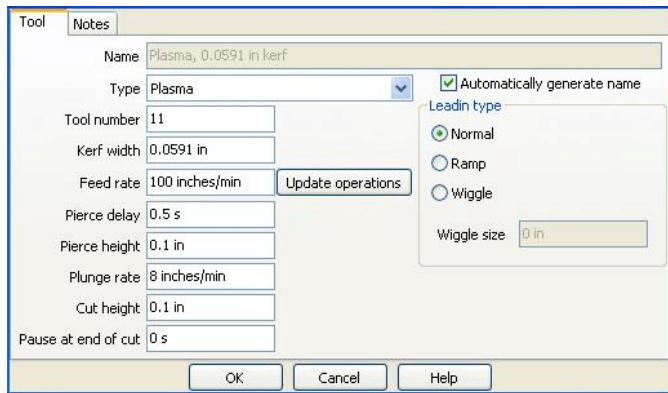
New thread mill 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Jet cutting tools

New plasma tool



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Note: This menu item is only visible if 'Jet cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Note: If you need to use a plasma cutter to pierce holes at specific

locations the 'Drilling operation' can be used with a plasma tool setting for this purpose.
See the '[Drilling operation](#)' page for more information.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct jet cutting type from the drop down menu shown below.



Note: This page shows setting options for a new plasma cutter. To see the other tool setup pages refer to the relevant sections below.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Kerf width

Enter the kerf width (cut width) here.

Feed rate

Enter the required feed rate here.

Pierce delay

Enter the delay time here.

Pierce height

Enter the pierce height here.

Plunge rate

Enter the plunge rate here.

Cut height

Enter the cut height here.

Pause at end of cut

Enter the pause time here.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Leadin type

Select the leadin type using the radio buttons.

Enter the wiggle size in the box.

Note: Wiggle piercing moves the cutting head back and forth to help prevent damage to the head.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new jet cutting tool' button  located in the left-hand vertical toolbar.

Note: Pierce height, plunge rate, ramp leadin and cut height are only available for machines where the Z axis is controlled via G-code. Some machines perform these operations automatically.

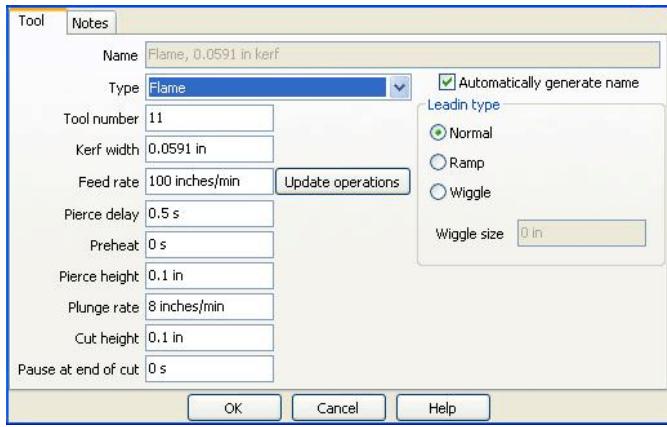
New plasma tool 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New flame tool

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Note: This menu item is only visible if 'Jet cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Note: If you need to use a flame cutter to pierce holes at specific locations the 'Drilling operation' can be used with a flame tool setting for this purpose. See the '[Drilling operation](#)' page for more information.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct jet cutting type from the drop down menu shown below.



Note: This page shows setting options for a new flame cutter. To see the other tool setup pages refer to the relevant sections below.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Kerf width

Enter the kerf width (cut width) here.

Feed rate

Enter the required feed rate here.

Pierce delay

Enter the delay time here.

Preheat

Enter the preheat time here.

Pierce height

Enter the pierce height here.

Plunge rate

Enter the plunge rate here.

Cut height

Enter the cut height here.

Pause at end of cut

Enter the pause time here.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Leadin type

Select the leadin type using the radio buttons.

Enter the wiggle size in the box.

Note: Wiggle piercing moves the cutting head back and forth to help prevent damage to the head.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new jet cutting tool' button  located in the left-hand vertical toolbar.

Note: Pierce height, plunge rate, ramp leadin and cut height are only available for machines where the Z axis is controlled via G-code. Some machines perform these operations automatically.

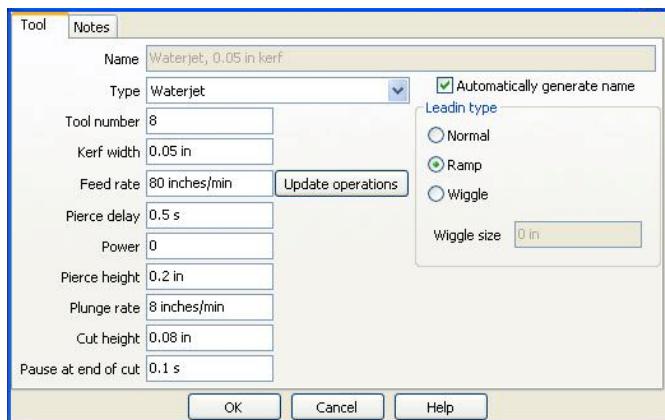
New flame tool 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New waterjet tool

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Note: This menu item is only visible if 'Jet cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Note: If you need to use a waterjet cutter to pierce holes at specific locations the 'Drilling operation' can be used with a waterjet tool setting for this purpose. See the '[Drilling operation](#)' page for more information.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct jet cutting type from the drop down menu shown below.



Note: This page shows setting options for a new waterjet cutter. To see the other tool setup pages refer to the relevant sections below.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Kerf width

Enter the kerf width (cut width) here.

Feed rate

Enter the required feed rate here.

Pierce delay

Enter the delay time here.

Power

Set your power level in this box if your system has G-code controlled power settings.

Pierce height

Enter the pierce height here.

Plunge rate

Enter the plunge rate here.

Cut height

Enter the cut height here.

Pause at end of cut

Enter the pause time here.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Leadin type

Select the leadin type using the radio buttons.

Enter the wiggle size in the box.

Note: Wiggle piercing moves the cutting head back and forth to help prevent damage to the head.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new jet cutting tool' button  located in the left-hand vertical toolbar.

Note: Pierce height, plunge rate, ramp leadin and cut height are only available for machines where the Z axis is controlled via G-code. Some machines perform these operations automatically.

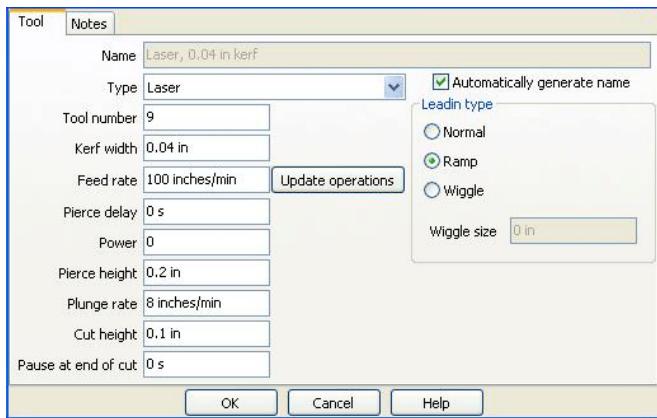
New waterjet tool 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New laser tool

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Note: This menu item is only visible if 'Jet cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Note: If you need to use a laser to pierce holes at specific locations the 'Drilling operation' can be used with a laser setting for this purpose. See the '[Drilling operation](#)' page for more information.

Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct jet cutting type from the drop down menu shown below.



Note: This page shows setting options for a new laser. To see the other tool setup pages refer to the relevant sections below.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Kerf width

Enter the kerf width (cut width) here.

Feed rate

Enter the required feed rate here.

Pierce delay

Enter the delay time here.

Power

Set your power level in this box if your system has G-code controlled power settings.

Pierce height

Enter the pierce height here.

Plunge rate

Enter the plunge rate here.

Cut height

Enter the cut height here.

Pause at end of cut

Enter the pause time here.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Leadin type

Select the leadin type using the radio buttons.

Enter the wiggle size in the box.

Note: Wiggle piercing moves the cutting head back and forth to help prevent damage to the head.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new jet cutting tool' button  located in the left-hand vertical toolbar.

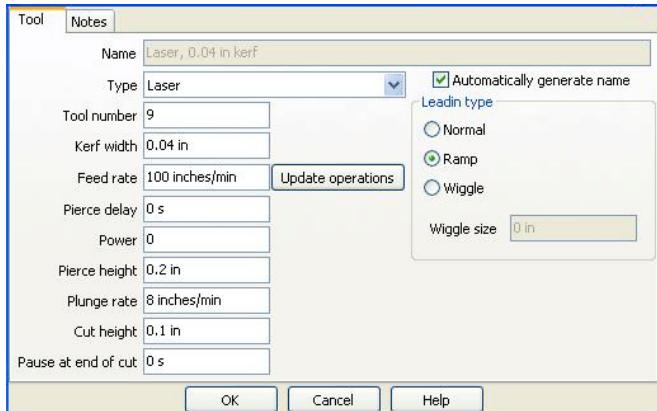
Note: Pierce height, plunge rate, ramp leadin and cut height are only available for machines where the Z axis is controlled via G-code. Some machines perform these operations automatically.

New laser 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New plate marker tool



Name

Manually enter the tool name if the 'Automatically generate name' box below is not checked.

Type

Select the correct jet cutting type from the drop down menu shown below.



Note: This page shows setting options for a new plate marker tool. To see the other tool setup pages refer to the relevant sections below.

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool number

Enter the tool number here.

Diameter

Enter the diameter of the tool here.

Feed rate

Enter the required feed rate here.

Cut height

Enter the cut height here.

Start delay

Enter the delay time here.

Pause at end of cut

Enter the pause time here.

Update operations

'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Note: This menu item is only visible if 'Jet cutting' has been selected in the '[Options\Machine\Machine type](#)' menu.

Leadin type

Select the leadin type using the radio buttons.

Enter the wiggle size in the box.

Note: Wiggle piercing moves the cutting head back and forth to help prevent damage to the head.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new jet cutting tool' button  located in the left-hand vertical toolbar.

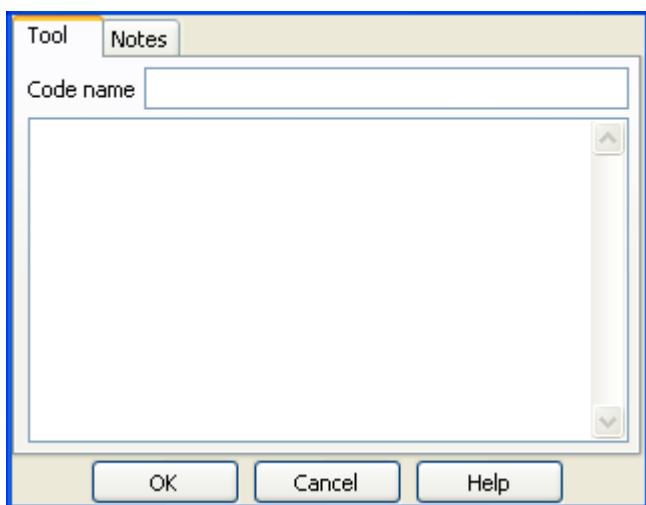
Note: Pierce height, plunge rate, ramp leadin and cut height are only available for machines where the Z axis is controlled via G-code. Some machines perform these operations automatically.

New plate marker tool 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New code snippet



Selecting 'New code snippet' opens a 'tool' that allows for the creation of one or more lines of G-code. This is then used by the '[Insert code](#)' operation.

Enter a name for the code in the top window and the body of the code in the lower window. Click <OK> to accept.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create new code snippet' button  located in the left-hand vertical toolbar.

New code snippet 'Notes' tab

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Tool table

Mill/router	V cutter	Drill	Thread mill	Rigid tap	Tapping head	Plasma	Flame	Waterjet	Laser	Plate marker
Name	Tool number	Diameter	Flute length	Tool length offset	Depth per pass	Feed rate	Plunge rate	Ramp Angle	Spindle speed	
Mill/router, 0.25 in diameter	1	0.25 in	2 in	1 in	0.1 in	3 inches/min	2 inches/min	0 degrees	1000 rpm	
Drill, 0.25 in diameter	2									
V cutter, 0.05 in diameter	3									
Tapping head, 0.375 in x 16 TPI	4									
Rigid tap, 0.375 in x 16 TPI	5									
Plasma, 0.05 in kerf	6									
Flame, 0.15 in kerf	7									
Waterjet, 0.05 in kerf	8									
Laser, 0.04 in kerf	9									

Automatically generate name

[OK](#) [Cancel](#) [Help](#)

Selecting 'Tool table' opens a window that allows you to edit the tools that have been created.

Use the tabs at the top of the table to navigate to a tool type and click once in a highlighted (white) box to select it and click again to edit the value (double-clicking also works).

Automatically generate name

Checking this box will automatically generate the tool name based on selections made in the boxes.

OK

'OK' keeps the changes and applies them to **any new operations** that use that tool. The tool diameter however is immediately applied to **all operations** that use the tool.

Cancel

'Cancel' cancels any changes and closes the tool table window.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Tool table' button  located in the left-hand vertical toolbar.

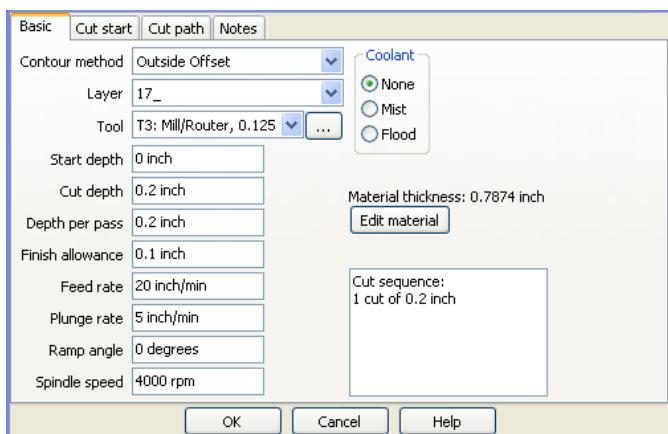
Operation setup

You will now need to select the 'operation' you require from the following options:

Contour	Drilling
Pocket	Insert code
Tapping	Set post variable
Plasma cut	Move to

Contour

Basic tab



Note: All units based on the selection in the ['Options\Application options\Units'](#) menu.

Contour method

Select the contour method required using the drop down menu.



Note: 'Inside offset' will offset the cutter path to the 'inside' of the contour by half the diameter of the tool chosen below.

Note: 'Outside offset' will offset the cutter path to the 'outside' of the contour by half the diameter of the tool chosen below. It should be noted that the cut side will be reversed if you cut an inside contour. For instance, if you were to cut a washer then 'Outside offset' will cut the inside of the hole and then the outside of the washer (in that order).

Note: 'No offset' will set the centre of the tool on the contour line.

Note: The above is for closed contours, for open contours see '['Offset open paths'](#)' on the 'Cut path' tab.

Layer



Select the layer you wish to apply the contour to using the drop down menu.

Tool



Select the correct tool using the down menu.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the '[Tool Edit](#)' dialog box.

Start depth

This function specifies the depth at which to start cutting.

Note: This function could be used if you want to cut a second pocket into the floor of a previously machined pocket that is 1" deep. Set the start '**depth**' to 1" (a **positive** (+) dimension) and SheetCam will start machining at this point. This saves a lot of 'air cutting' time.

CAUTION: Improper use of this function could lead to tool/part crashes! **Only** change the value from zero (0) (the default value) if you are actually cutting a 'pocket within a pocket'.

Cut depth

Enter the cut depth here. This function controls the total depth of cut.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Depth per pass

Enter the Z increment depth here. This function controls the depth of cut per pass.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Finish allowance

Enter the finish allowance here. This will force SheetCam to leave the part either oversized or undersized depending on the 'Contour method' chosen above. You will then need to specify a new 'Operation' to machine the part to the 'finished' size using a finishing cutter if required and/or a different 'Depth per pass' depth.

Note: Leaving the allowance at zero (0) will machine the contour to full size with no need for a finishing pass.

Feed rate

Enter the required feed rate here.

Plunge rate

Enter the plunge rate here.

Ramp Angle

'Ramping' allows the cutter to enter the work while travelling in a forward direction. This reduces the load on the Z axis and also allows you to plunge with a 'non-centre cutting' tool (i.e. some 4-flute end mills). The cutter will travel forwards as it plunges, cutting a ramp into the work. This ramp is then machined away. If you are ramping into an open contour or between tabs then the cutter ramps down then backs up to the start of the cut, reverses up to the start point then reverses again and carries on to the end. If the contour is closed then the cutter simply carries on at the end of the cut until the ramp is machined away. For small contours the cutter will spiral down to depth. The ramp angle specifies the angle at which the cutter will plunge into the work . 0 degrees or 90 degrees are straight down (no ramping), 5 degrees is very shallow and 85 degrees is very steep.

Note: If a cut path is very short it will not be possible to ramp into the cut. In this case the cutter will be plunged straight down. The cut operation will turn yellow to warn you that this is happening.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software. However, the post processor will issue a warning if the speed is set to zero (0).

Coolant

Select the coolant type from the available choices.

Note: These items only need to be checked if your coolant pump is controlled via software.

Edit material

Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

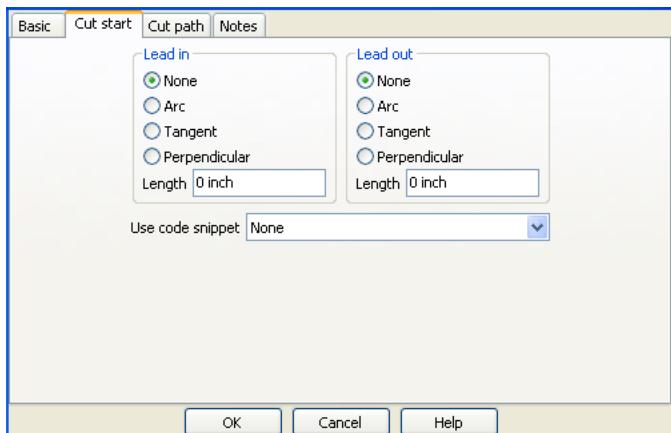
Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new contour operation' button located in the left-hand vertical toolbar.



Cut start tab



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Lead in

Select the type of lead in required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead in.

Lead out

Select the type of lead out required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead out.

Use code snippet

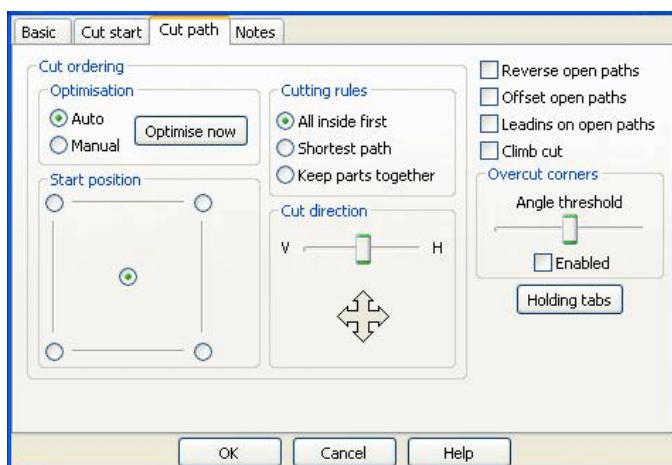
Select the required code snippet from the drop down menu.

Note: The code snippet needs to be previously defined using the '[Tools/New code snippet](#)' function. The snippet is inserted into the code at the start of the cut just before the cutter plunges to depth.

Help

'Help' opens the help file at the relevant location.

Cut path tab



Cut ordering

Optimisation

Auto/manual

Select auto or manual path optimisation using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while

adhering to the rules shown under 'Cutting rules' below. If 'Manual' is selected then you can manually set the cut sequence (by editing the start points).

Optimise now

This button optimises the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start position

This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

Cutting rules

All inside first

Inside contours are cut first then outside. This is the way SheetCam always used to work. Useful for plasma or for milling/routing when you are cutting all the way through.

Shortest path

This option uses the shortest possible route between contours. This is the one you would use for most milling/routing jobs where you aren't cutting right through.

Keep parts together

Like all inside first, it cuts inside then outside. If your drawing contains more than one part then each part is cut out completely before moving on to the next. This is useful for plasma, where heat distortion can cause problems if you cut all the insides of all parts then cut all the outsides. By the time you get to the last part heat distortion of the sheet can result in the inside not lining up with the outside.

Cut direction

This adds a bias to the path optimisation. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.

Reverse open paths

Check this box to reverse the cut direction of open paths.

Note: If this item is selected and you are cutting to depth in a number of passes then the cutter will reverse direction on each pass. If this is turned off the cutter will lift and return to the start of the path for each pass. It can give a better finish but obviously takes longer.

Offset open paths

Open paths are always offset to the right of the start point. That is, if you were standing at the start point looking down the line the offset would be to your right. To reverse the side of the offset, move the start point to the other end of the line. Inside/outside offset make no difference as a line doesn't have an inside or outside.

Note: If you have climb cut selected the cut will start at the opposite end of the path to the start point. I know this is counter intuitive but it is the only way to maintain the 'offset to the right' rule.

Leadins on open paths

Check this box to apply leadins to open paths.

Climb cut

Check this box to select 'climb' cutting.

Note: This will reverse the cut direction on closed paths.

Sharpen/Overcut corners

Note: 'Sharpen corners' will only appear if you have selected a V cutter while 'Overcut corners' will only appear if you have selected a mill/router.

Sharpen corners

Note: If you are using a V cutter on inside corners the radius of the cut is bigger at the top than at the bottom due to the taper of the cutter. Turning on 'sharpen corners' ramps the cutter up on inside corners so that the radius of the cut is the same at the top as it is at the bottom. This is a benefit when engraving text or graphics. This option is not available for mill/router cutters. The steps for using corner sharpening are shown below:

Angle threshold

First define your tool. It must be defined as a V cutter and you must enter the V angle. Pocket the area to clear the majority of the material using a regular milling cutter. Now do an 'inside contour' using the V tool. With the V tool selected go to the 'Cut path' tab. You should see a slider called 'Angle threshold' and a check box for 'Enabled'. If you are engraving text or something that is made up of lots of short line segments you may find that SheetCam is sharpening some very shallow corners which wastes time and does not make much difference to the final result. Increasing the angle threshold will remove these.

Note: When sharpening corners you may not want to sharpen shallow angles. This option controls how sharp an angle has to be to trigger corner sharpening. Consider the letter 'C' where the outline consists of a large number of short line segments. You don't normally want to sharpen the join between every line.

Enabled

Check this box to apply 'corner sharpening' to corners.

Overcut corners

Note: If this is selected the cutter will cut into inside corners to allow room for a sharp cornered part to fit. For instance if you are cutting a rebate for a square part to fit into this option will save manually squaring out the corners. This option is not available for V cutters.

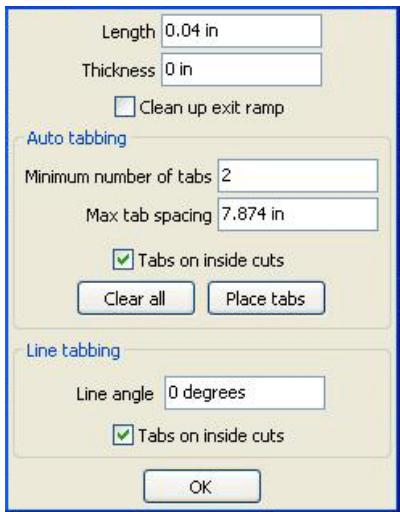
Angle threshold

Angle threshold adjusts how sharp the corner needs to be before the overcutting is applied.

Enabled

Check this box to apply 'Overcut corners' to corners.

Holding Tabs



Clicking on this button will open the dialog box shown below. The tab length and tab thickness controls are used to specify the respective sizes of holding tabs (i.e. the material left behind to hold parts in place). To place tabs use the 'Holding tab tool'  on the tool bar.

Note: This button will be 'greyed' out if no tabs are set.

Length

This control is used to specify the length of the holding tab. Tabs can be any size to suit the application but generally longer tabs have a reduced thickness while shorter tabs are usually left at the material thickness.

Thickness

This control is used to specify the thickness of the holding tab. Setting the tab thickness to less than the material thickness will result in thin tabs that can be easily cut/broken when machining is complete. It is worth mentioning that tab thickness is based on the material thickness, not the cut depth. For instance, if you have 10mm thick material and specify 2mm thick tabs you will need to cut at least 8mm deep to see the tabs at all.

Note: By using the tab length/thickness controls you can make SheetCam clear any clamps or fixtures as it cuts provided that you know where the clamps will be. By making the 'thickness' greater than the material thickness **plus** the clamp thickness SheetCam will 'rapid' traverse for the length of the tab after the cutter has withdrawn to the correct height provided that the combined thickness is greater than the rapid clearance height (as set under the '[Options/Material](#)' menu item). The steps for using 'tabs' are shown below:

Set up your contour operation as normal and then go to the 'Cut path' tab and set the tab length and tab thickness. Tab length is the length of the tab at the narrowest point. Thickness is the **amount of material left** in the tab.

Once your operation is set up click on the 'Tabs' button . The cursor will change to an arrow with a 'T' next to it if it is near a valid tab position. Now click on the outline where you want to place the tab. You can place as many tabs as you like. To delete a tab point at it (it will change to white), right click and select delete. To check if the tabs have been placed correctly click on the 'Scroll screen' button  then rotate the display by holding the shift key and dragging with the mouse. You will see the cut paths lift up where you have placed the tabs.

If you set the tab thickness to greater than the cut depth + rapid clearance then the cutter will rapid when it moved from one end of the tab to another. This is handy if you have a holding clamp in the way. You can set the tab thickness so that the cutter will lift over the clamp even if this is higher than the rapid clearance.

If you are **not** using tabs it is recommend that you set the tab thickness to 0 (zero). When using a tab thickness greater than 0 (zero) SheetCam uses a slightly different cutting strategy which can be a little less efficient. This is particularly noticeable if you use ramping

in conjunction with the tabs.

Clean up exit ramp

When ramp cutting with tabs, enabling this will clean up the trailing ramp on each tab. This takes longer to cut but leaves square edged tabs. With this option disabled the ramp is left so the entry to the tab is square but the exit is tapered.

Auto tabbing

Auto tabbing automatically places tabs on every outline.

Min number of tabs

The number of tabs needed depends very much on the material you are working with. Generally the minimum number of tabs should be set to 2. This way, even small parts will have two tabs.

Max tab spacing

The max tab spacing is the maximum distance between tabs. This effectively sets the number of tabs on larger outlines. Even on big parts you probably don't need more than 5 so the max tab spacing can be quite large.

Tabs on inside cuts

Check this box to create tabs on the inside of the cut line.

Clear all

Clicking this button clears all tabs.

Place tabs

Clicking this button places the tabs.

Line tabbing

Line tabbing is used while manually placing tabs. See the '[Edit tabs](#)' section for more information.

Line angle

Sets the angle of the tab line.

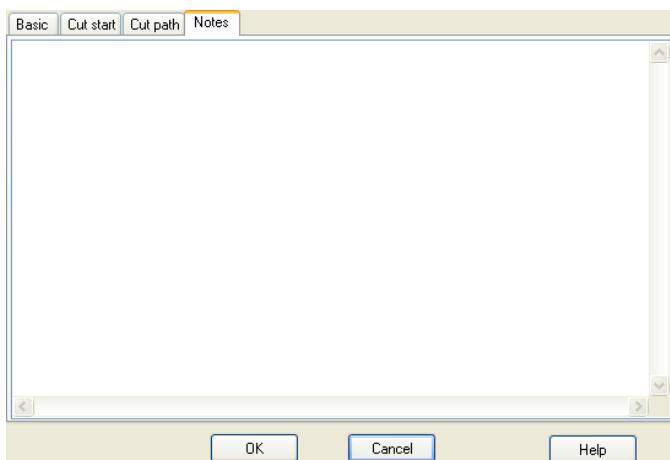
Tabs on inside cuts

Check this box to create tabs on the inside of the cut line.

Help

'Help' opens the help file at the relevant location.

Notes Tab

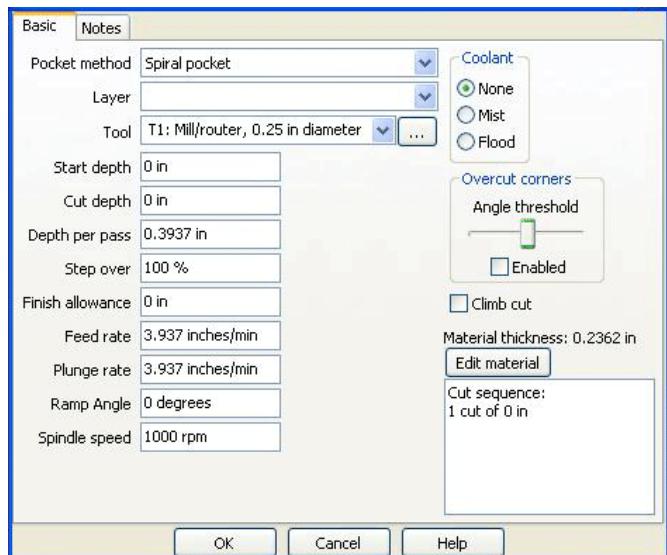


Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

New pocket

Basic tab



Note: All units based on the selection in the ['Options\Application options\Units'](#) menu.

Pocket method

Select either 'Spiral pocket' or 'Zigzag pocket' from the drop down menu.

Layer

Select the layer you wish to apply the pocket to using the drop down menu shown below.



Tool

Select the correct tool using the down menu.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the ['Tool Edit'](#) dialog box.

Start depth

This function specifies the depth at which to start cutting.

Note: This function could be used if you want to cut a second pocket into the floor of a previously machined pocket that is 1" deep. Set the start '**depth**' to 1" (a **positive** (+) dimension) and SheetCam will start machining at this point. This saves a lot of 'air cutting' time.



CAUTION: Improper use of this function could lead to tool/part crashes!

Only change the value from zero (0) (the default value) if you are actually cutting a 'pocket within a pocket'.

Cut depth

Enter the cut depth here. This function controls the total depth of cut.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Depth per pass

Enter the Z increment depth here. This function controls the depth of cut per pass.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Step over

Set the 'Step over' amount as a percentage of the cutter diameter.

Fill angle

Note: This item is only displayed when 'Zigzag pocket' is selected in 'Pocket method' above.

Changing the 'Fill angle' changes the direction of cut for the zigzag pocket.

Finish allowance

Enter the finish allowance here. This will force SheetCam to leave the part undersized. You will then need to specify a new 'Operation' to machine the part to the 'finished' size using a finishing cutter if required and/or a different 'Depth per pass' depth.

Note: Leaving the allowance at zero (0) will machine the contour to full size with no need for a finishing pass.

Feed rate

Enter the required feed rate here.

Plunge rate

Enter the plunge rate here.

Ramp angle

'Ramping' allows the cutter to enter the work while travelling in a forward direction. This reduces the load on the Z axis and also allows you to plunge with a 'non-centre cutting' tool (i.e. some 4-flute end mills). The cutter will travel forwards as it plunges, cutting a ramp into the work. This ramp is then machined away. If you are ramping into an open contour or between tabs then the cutter ramps down then backs up to the start of the cut, reverses up to the start point then reverses again and carries on to the end. If the contour is open then the cutter simply carries on at the end of the cut until the ramp is machined away. For small contours the cutter will spiral down to depth. The ramp angle specifies the angle at which the cutter will plunge into the work . 0 degrees or 90 degrees are straight down (no ramping), 5 degrees is very shallow and 85 degrees is very steep.

Note: If a cut path is very short it will not be possible to ramp into the cut. In this case the cutter will be plunged straight down. The cut operation will turn yellow to warn you that this is happening.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software. However, the post processor will issue a warning if the speed is set to zero (0).

Coolant

Select the coolant type from the available choices.

Note: These items only need to be checked if your coolant pump is controlled via software.

Sharpen/Overcut corners

Note: 'Sharpen corners' will only appear if you have selected a V cutter while 'Overcut corners' will only appear if you have selected a mill/router.

Sharpen corners

Note: If you are using a V cutter on inside corners the radius of the cut is bigger at the top than at the bottom due to the taper of the cutter. Turning on 'sharpen corners' ramps the cutter up on inside corners so that the radius of the cut is the same at the top as it is at the bottom. This is a benefit when engraving text or graphics. This option is not available for mill/router cutters.

The steps for using corner sharpening are shown below:

Angle threshold

First define your tool. It must be defined as a V cutter and you must enter the V angle. Pocket the area to clear the majority of the material using a regular milling cutter. Now do an 'inside contour' using the V tool. With the V tool selected go to the 'Cut path' tab. You should see a slider called 'Angle threshold' and a check box for 'Enabled'. If you are engraving text or something that is made up of lots of short line segments you may find that SheetCam is sharpening some very shallow corners which wastes time and does not make much difference to the final result. Increasing the angle threshold will remove these.

Note: When sharpening corners you may not want to sharpen shallow angles. This option controls how sharp an angle has to be to trigger corner sharpening. Consider the letter 'C' where the outline consists of a large number of short line segments. You don't normally want to sharpen the join between every line.

Enabled

Check this box to apply 'corner sharpening' to corners.

Overcut corners

Note: If this is selected the cutter will cut into inside corners to allow room for a sharp cornered part to fit. For instance if you are cutting a rebate for a square part to fit into this option will save manually squaring out the corners. This option is not available for V cutters.

Angle threshold

Angle threshold adjusts how sharp the corner needs to be before the overcutting is applied.

Enabled

Check this box to apply 'Overcut corners' to corners.

Climb cut

Check this box to select 'climb' cutting.

Note: This will reverse the cut direction on closed paths.

Edit material

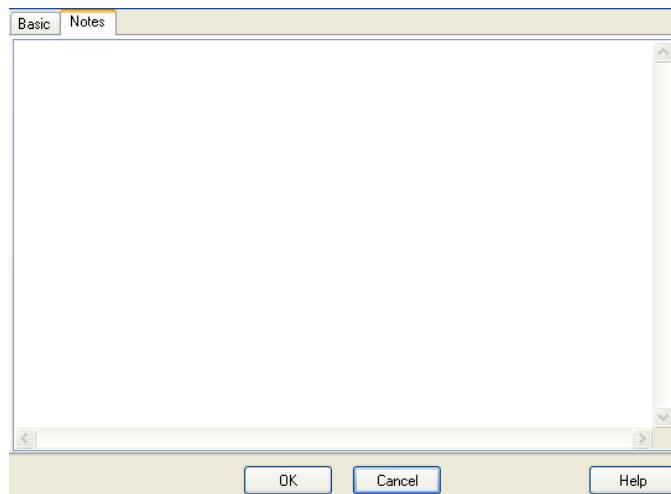
Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new fill operation' button  located in the left-hand vertical toolbar.

Notes tab



Notes

Use this area to store any notes you may wish to make regarding the operation.

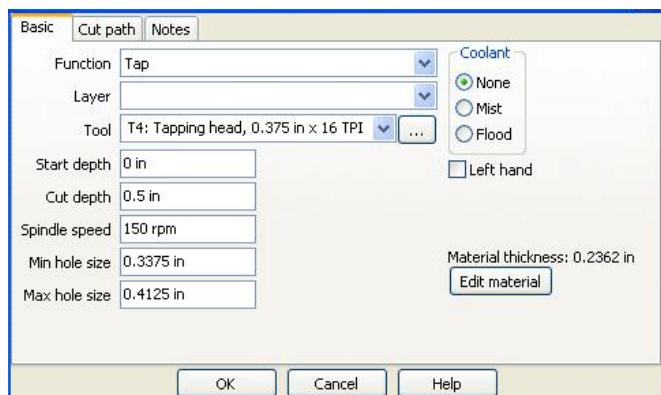
Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Help

'Help' opens the help file at the relevant location.

Tapping

Basic tab



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Function

Select the correct threading type from the drop down menu shown below.

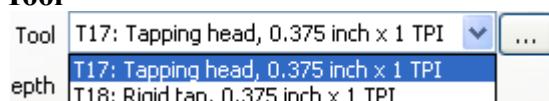
Note: This page shows setting options for a new tap. To see the 'Thread mill' setup pages refer to the relevant section below.

Layer



Select the layer you wish to apply the tapping operation to using the drop down menu shown.

Tool



Select the correct tool using the down menu shown.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the '[Tool Edit](#)' dialog box.

Start depth

This function specifies the depth at which to start tapping.

Note: This function could be used if you want to tap a hole into the floor of a previously machined pocket that is 1" deep. Set the start '**depth**' to 1" (a **positive** (+) dimension) and SheetCam will start machining at this point. This saves a lot of 'air cutting' time.

Cut depth

Enter the cut depth here. This function controls the total depth of the tapped hole.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of the hole. Some users expect this to be a minus (-) figure.

Spindle speed

Enter the spindle speed here.

Note: Spindle speed **must** be entered. It is used with the tap pitch to work out the feed rate.

Min hole size

Set the minimum hole size here.

Max hole size

Set the maximum hole size here.

Note: Only holes that fall in the range covered by the max. and min. settings will be machined.

Note: If holes have been specified as points in the drawing then SheetCam has no way of knowing the desired hole size. Holes specified as points are always tapped.

Coolant

Select the coolant type from the available choices.

Note: These items only need to be checked if your coolant pump is controlled via software.

Left hand

Check this box if you are cutting a left hand thread.

Edit material

Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

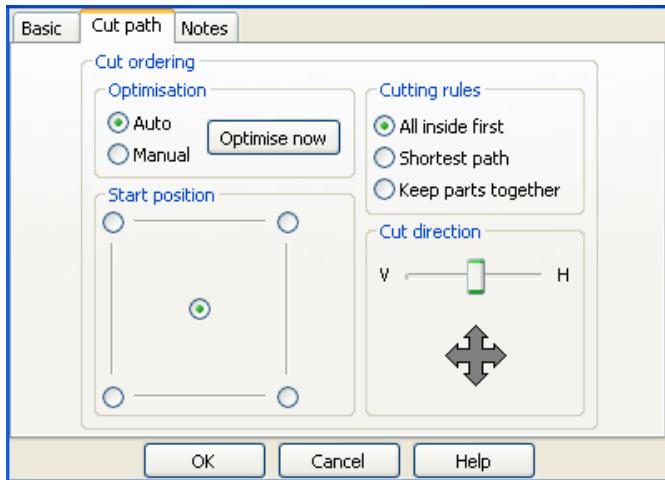
Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new tapping operation' button  located in the left-hand vertical toolbar.

Cut path tab

Cut ordering



Optimisation

Auto/manual

Select auto or manual path optimisation using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while adhering to the following rules:

- 1) Open contours are cut first.
- 2) Inside contours are cut next (i.e. the hole in a washer).
- 3) Outside contours are cut last.

If 'Manual' is selected then you can manually set the cut sequence (by editing the start points).

Optimise now

This button optimises the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start position

This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

Cutting rules

All inside first

Inside contours are cut first then outside. This is the way SheetCam always used to work. Useful for plasma or for milling/routing when you are cutting all the way through.

Shortest path

This option uses the shortest possible route between contours. This is the one you would use for most milling/routing jobs where you aren't cutting right through.

Keep parts together

Like all inside first, it cuts inside then outside. If your drawing contains more than one part then each part is cut out completely before moving on to the next. This is useful for plasma, where heat distortion can cause problems if you cut all the insides of all parts then cut all the outsides. By the time you get to the last part heat distortion of the sheet can result in the inside not lining up with the outside.

Cut direction

This adds a bias to the path optimisation. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.

Help

'Help' opens the help file at the relevant location.

Notes tab**Notes**

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

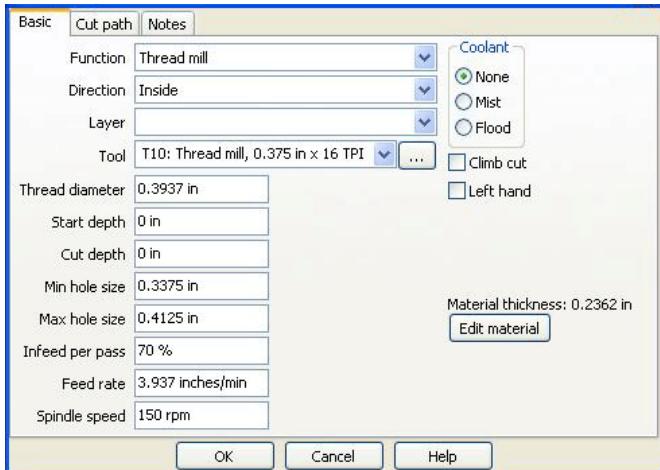
Help

'Help' opens the help file at the relevant location.

Thread milling

Basic tab

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Function

Select the correct threading type from the drop down menu shown below.



Note: This page shows setting options for a new Thread mill. To see the 'Tap' setup pages refer to the relevant section above.

Direction

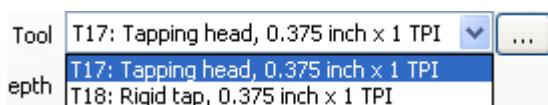
Select 'Inside' or 'Outside' from the drop down menu.

Layer



Select the layer you wish to apply the tapping operation to using the drop down menu shown.

Tool



Select the correct tool using the down menu shown.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the '[Tool Edit](#)' dialog box.

Thread diameter

Enter the thread diameter here.

Start depth

This function specifies the depth at which to start tapping.

Note: This function could be used if you want to tap a hole into the floor of a previously machined pocket that is 1" deep. Set the start '**depth**' to 1" (a **positive** (+) dimension) and SheetCam will start machining at this point. This saves a lot of 'air cutting' time.

Cut depth

Enter the cut depth here. This function controls the total depth of the tapped hole.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of the hole. Some users expect this to be a minus (-) figure.

Min hole size

Set the minimum hole size here.

Max hole size

Set the maximum hole size here.

Note: Only holes that fall in the range covered by the max. and min. settings will be machined.

Note: If holes have been specified as points in the drawing then SheetCam has no way of knowing the desired hole size. Holes specified as points are always tapped.

Infeed per pass

Infeed per pass controls the depth of cut. 100% will cut the thread in one pass. 70% would cut 70% in one pass and 30% in the last pass. 25% would cut four equal passes.

Feed rate

Enter the required feed rate here.

Spindle speed

Enter the spindle speed here.

Coolant

Select the coolant type from the available choices.

Note: These items only need to be checked if your coolant pump is controlled via software.

Climb cut

Check this box to select 'climb' cutting.

Note: This will reverse the cut direction on closed paths.

Left hand

Check this box if you are cutting a left hand thread.

Edit material

Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

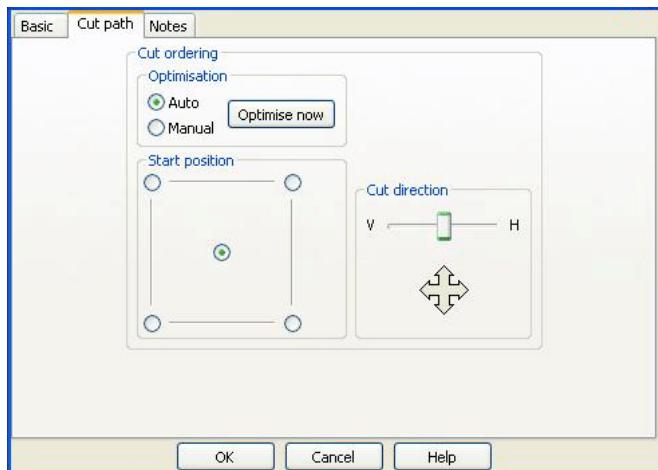
Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new tapping operation' button  located in the left-hand vertical toolbar.

Cut path tab

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



Cut ordering

Optimisation

Auto/manual

Select auto or manual path optimisation using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while

adhering to the rules shown under 'Cutting rules' below. If 'Manual' is selected then you can manually set the cut sequence (by editing the start points).

Optimise now

This button optimises the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start position

This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

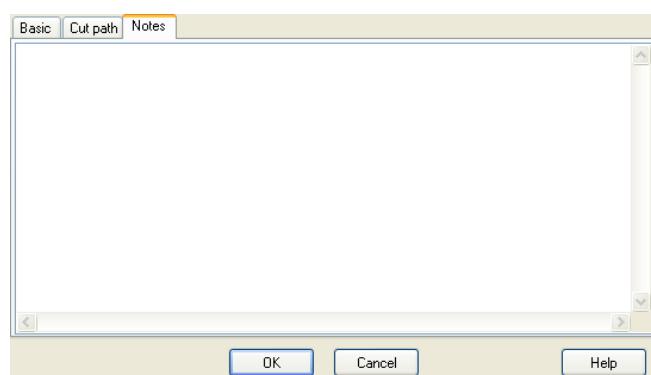
Cut direction

This adds a bias to the path optimisation. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.

Help

'Help' opens the help file at the relevant location.

Notes tab



Notes

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

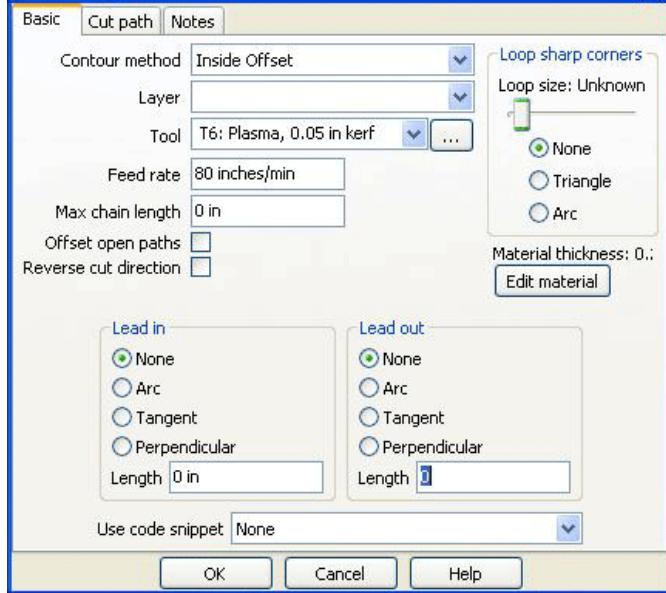
Help

'Help' opens the help file at the relevant location.

Plasma cut

Basic tab

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



the kerf of the tool chosen below.

Note: If you need to use a plasma cutter to pierce holes at specific locations the 'Drilling operation' can be used with a plasma tool setting for this purpose. See the '[Drilling operation](#)' page for more information.

Contour method

Select the contour method required using the drop down menu shown below.

Note: 'Inside offset' will offset the cutter path to the 'inside' of the contour by half the diameter of

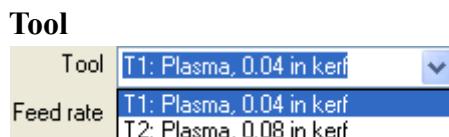
Note: 'Outside offset' will offset the cutter path to the 'outside' of the contour by half the diameter of the kerf of the tool chosen below.

Note: 'No offset' will set the centre of the tool on the contour line.

Note: The above is for closed contours, for open contours see 'Offset open paths' below.



Select the layer you wish to apply the contour to using the drop down menu shown.



Select the correct tool using the down menu shown.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the '[Tool Edit](#)' dialog box.

Feed rate

Enter the required feed rate here.

Max chain length

Specify the maximum length allowed for chain cuts in this box. SheetCam will then use this value when it calculates the chains between parts. Piercing wears consumables so if possible SheetCam will try to cut from the end of one outline to the start of the next. The maximum chain length defines the maximum distance to cut. If the distance is further than this value the tool will lift and move to the next cut start as normal.

Offset open paths

Open paths are always offset to the right of the start point. That is, if you were standing at the start point looking down the line the offset would be to your right. To reverse the side of the offset, move the start point to the other end of the line. Inside/outside offset make no difference as a line doesn't have an inside or outside.

Note: If you have climb cut selected the cut will start at the opposite end of the path to the start point. I know this is counter intuitive but it is the only way to maintain the 'offset to the right' rule.

Reverse cut direction

Check this box to reverse the cut direction.

Loop sharp corners

When plasma/flame/water jet/laser cutting the exit point of the jet lags behind the entry point. This causes rounding of sharp corners. To get around this problem the corner is cut in a loop instead. Set up a simple jet cutting job on a square and you will see how this works. Triangle loops take up the least space but are relatively slow to cut due to the sharp corners. Arc loops are smoother to cut but take more room. If an arc loop would overlap an existing cut, SheetCam tries a triangle. If the triangle still won't fit, the corner is not looped.

Note: Loops will be left out if they would overlap an existing cut path.

Loop size

The slider adjusts the size of the loop used above. This is an arbitrary function and can only be established using trial and error methods as it is dependant upon flame/jet size on your machine. Set the slider, make a test cut and adjust as required.

Use the radio buttons to select the type of loop.

Edit material

Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

Lead in

Select the type of lead in required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead in.

Lead out

Select the type of lead out required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead out.

Use code snippet

Select the required code snippet from the drop down menu.

Note: The code snippet needs to be previously defined using the '[Tools/New code snippet](#)' function. The snippet is inserted into the code at the start of the cut just before the cutter plunges to depth.

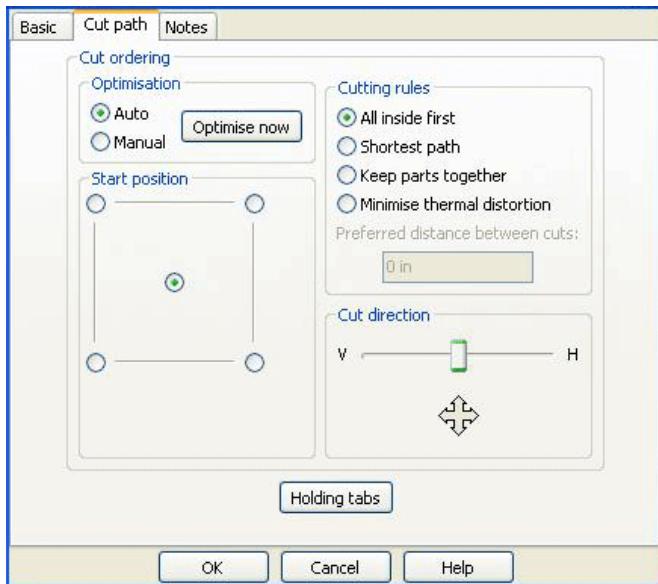
Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new plasma cut operation' button  located in the left-hand vertical toolbar.

Cut path tab

Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.



sequence (by editing the start points).

Optimise now

This button optimises the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start position

This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

Cut ordering

Optimisation

Auto/manual

Select auto or manual path optimisation using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while adhering to the rules shown under 'Cutting rules' below. If 'Manual' is selected then you can manually set the cut

Cutting rules

All inside first

Inside contours are cut first then outside. This is the way SheetCam always used to work. Useful for plasma or for milling/routing when you are cutting all the way through.

Shortest path

This option uses the shortest possible route between contours. This is the one you would use for most milling/routing jobs where you aren't cutting right through.

Keep parts together

Like all inside first, it cuts inside then outside. If your drawing contains more than one part then each part is cut out completely before moving on to the next. This is useful for plasma, where heat distortion can cause problems if you cut all the insides of all parts then cut all the outsides. By the time you get to the last part heat distortion of the sheet can result in the inside not lining up with the outside.

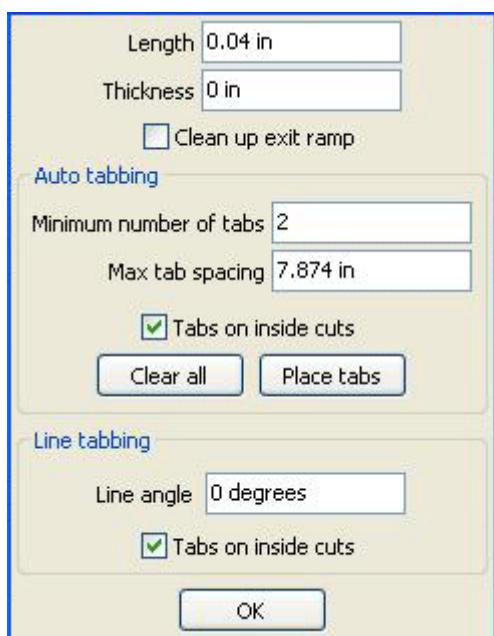
Minimise thermal distortion

This option tries to avoid cutting outlines that are close to other recently cut outlines. This helps to spread the heat over the sheet and minimize distortion. Increasing 'Preferred distance between cuts' controls the minimum distance between adjacent cuts. Increasing this value spreads the heat more evenly but increases overall cut time because of the extra rapid moves between cuts.

Cut direction

This adds a bias to the path optimisation. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.

Holding Tabs



Clicking on this button will open the dialog box shown below. The tab length and tab thickness controls are used to specify the respective sizes of holding tabs (i.e. the material left behind to hold parts in place). To place tabs use the 'Holding tab tool' 

Note: This button will be 'greyed' out if no tabs are set.

Length

This control is used to specify the length of the holding tab. Tabs can be any size to suit the application but generally longer tabs have a reduced thickness while shorter tabs are usually left at the material thickness.

Thickness

This control is used to specify the thickness of the holding tab. Setting the tab thickness to less than the material thickness will result in thin tabs that can be easily cut/broken when machining is complete. It is worth mentioning that tab thickness is based on the material thickness, not the cut depth. For instance, if you have 10mm thick material and specify 2mm thick tabs you will need to cut at least 8mm deep to see the tabs at all.

Note: By using the tab length/thickness controls you can make SheetCam clear any clamps or fixtures as it cuts provided that you know where the clamps will be. By making the 'thickness' greater than the material thickness **plus** the clamp thickness SheetCam will 'rapid' traverse for the length of the tab after the cutter has withdrawn to the correct height provided that the combined thickness is greater than the rapid clearance height (as set under the '[Options/Material](#)' menu item).

The steps for using 'tabs' are shown below:

Set up your contour operation as normal and then go to the 'Cut path' tab and set the tab length and tab thickness. Tab length is the length of the tab at the narrowest point. Thickness is the **amount of material left** in the tab.

Once your operation is set up click on the 'Tabs' button . The cursor will change to an arrow with a 'T' next to it if it is near a valid tab position. Now click on the outline where you want to place the tab. You can place as many tabs as you like. To delete a tab point at it (it will change to white), right click and select delete. To check if the tabs have been placed correctly click on the 'Scroll screen' button  then rotate the display by holding the shift key and dragging with the mouse. You will see the cut paths lift up where you have placed the tabs.

If you set the tab thickness to greater than the cut depth + rapid clearance then the cutter will rapid when it moved from one end of the tab to another. This is handy if you have a holding clamp in the way. You can set the tab thickness so that the cutter will lift over the clamp even if this is higher than the rapid clearance.

If you are **not** using tabs it is recommend that you set the tab thickness to 0 (zero). When using a tab thickness greater than 0 (zero) SheetCam uses a slightly different cutting strategy which can be a little less efficient. This is particularly noticeable if you use ramping in conjunction with the tabs.

Clean up exit ramp

When ramp cutting with tabs, enabling this will clean up the trailing ramp on each tab. This takes longer to cut but leaves square edged tabs. With this option disabled the ramp is left so the entry to the tab is square but the exit is tapered.

Auto tabbing

Auto tabbing automatically places tabs on every outline.

Min number of tabs

The number of tabs needed depends very much on the material you are working with. Generally the minimum number of tabs should be set to 2. This way, even small parts will have two tabs.

Max tab spacing

The max tab spacing is the maximum distance between tabs. This effectively sets the number of tabs on larger outlines. Even on big parts you probably don't need more than 5 so the max tab spacing can be quite large.

Tabs on inside cuts

Check this box to create tabs on the inside of the cut line. These are useful to stop the waste piece you have cut out from moving and possibly getting caught by the cutter.

Clear all

Clicking this button clears all tabs.

Place tabs

Clicking this button places the tabs.

Line tabbing

Line tabbing is used while manually placing tabs. See the '[Edit tabs](#)' section for more information.

Line angle

Sets the angle of the tab line.

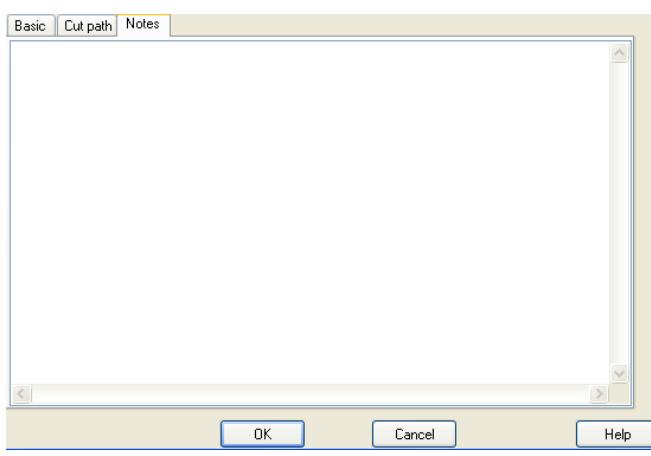
Tabs on inside cuts

Check this box to create tabs on the inside of the cut line.

Help

'Help' opens the help file at the relevant location.

Notes tab



Notes

Use this area to store any notes you may wish to make regarding the operation.

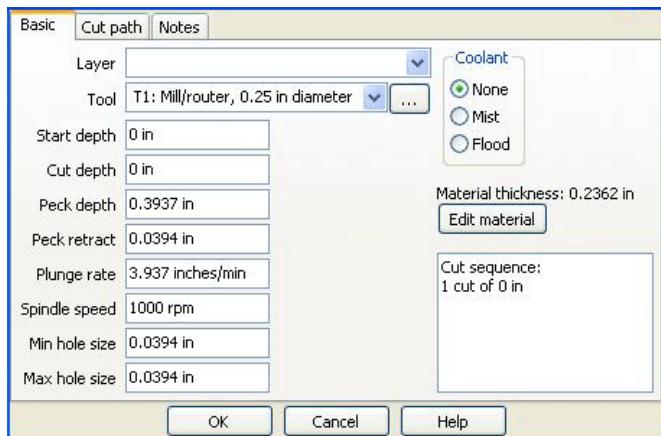
Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Help

'Help' opens the help file at the relevant location.

Drilling

Basic tab



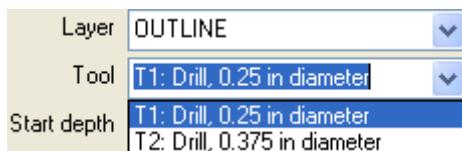
Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Layer

Select the layer you wish to apply the contour to using the drop down menu shown below.



Tool



Select the correct tool using the down menu shown.

Note: A special case exists for using the drilling operation with a jet cutter. See the 'Special case' section below for more information.

Edit

Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the '[Tool Edit](#)' dialog box.

Start depth

This function specifies the depth at which to start drilling.

Note: This function could be used if you want to drill a hole into the floor of a previously machined pocket that is 1" deep. Set the start '**depth**' to 1" (a **positive** (+) dimension) and SheetCam will start machining at this point. This saves a lot of 'air cutting' time.

Cut depth

Enter the cut depth here. This function controls the total depth of the drilled hole.

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of the hole. Some users expect this to be a minus (-) figure.

Peck depth

Enter the 'peck' depth here. This function controls the depth of each pass of the drilled hole (i.e. how deep the drill will travel before backing out to relieve/break chips).

Note: This is a **positive** (+) figure as you are specifying the '**depth**' of the peck. Some users expect this to be a minus (-) figure.

Peck retract

Enter the peck retract distance here. This function controls the distance the drill 'backs out' of the hole to relieve/break chips.

Note: This is a **positive** (+) figure as you are specifying the '**retract distance**' of the peck.

Plunge rate

Enter the plunge rate here.

Spindle speed

Enter the spindle speed here.

Note: Speed only needs to be specified if your spindle motor is controlled via software.
However, the post processor will issue a warning if the speed is set to zero (0).

Min hole size

Set the minimum hole size here.

Max hole size

Set the maximum hole size here.

Note: Only holes that fall in the range covered by the max. and min. settings will be machined.

Note: If holes have been specified as points in the drawing then SheetCam has no way of knowing the desired hole size. Holes specified as points are always drilled.

Tip: If you need to centre drill a series of holes prior to drilling set the minimum hole size to zero (0) and the maximum hole size to the size of the largest hole. All the holes on that layer will then be centre drilled.

Coolant

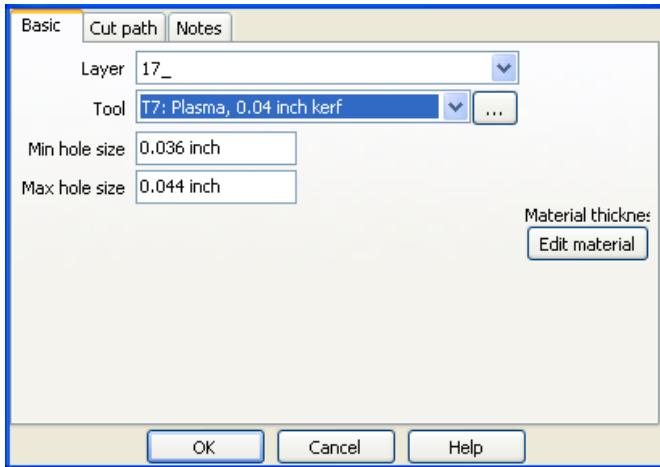
Select the coolant type from the available choices.

Note: These items only need to be checked if your coolant pump is controlled via software.

Edit material

Clicking on the 'Edit material' button calls up the '[Material](#)' dialog box.

Special case



The drilling operation can be used with a jet cutter to cut holes at the required locations. When you select a jet cutter as a tool in the drilling operation dialog box most of the options disappear as they are not relevant (see screenshot). The operation performs a function similar to centre drilling but pierces a hole with the cutter instead. The hole can then be drilled to size, on a different machine or by hand, at a later time. Some plasma cutters have

a centre punch built into the head. In that case this function can be used to fire the centre punch instead.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new drilling operation' button  located in the left-hand vertical toolbar.

Cut path tab

Cut ordering

Optimisation

Auto/manual

Select auto or manual path optimisation using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while adhering to the rules shown under 'Cutting rules' below. If 'Manual' is selected then you can manually set the cut sequence (by editing the start points).

Optimise now

This button optimises the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start position

This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

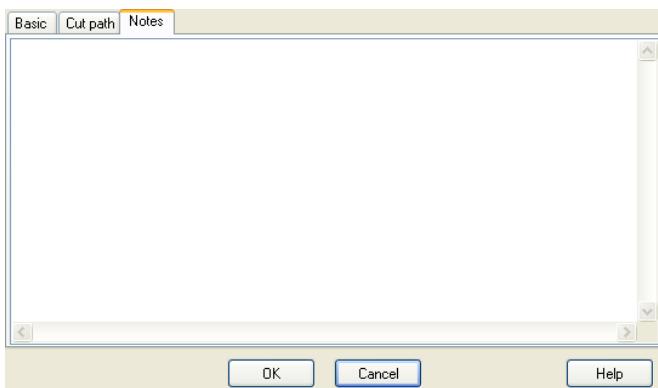
Cut direction

This adds a bias to the path optimisation. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.

Help

'Help' opens the help file at the relevant location.

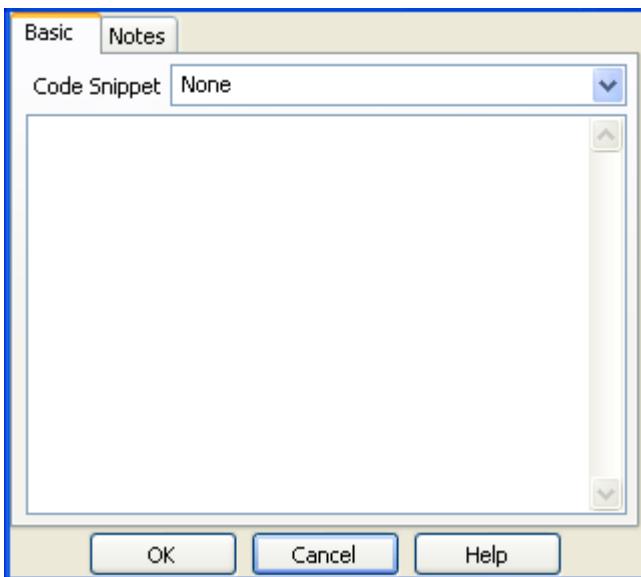
Notes tab



Help

'Help' opens the help file at the relevant location.

Insert code



Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Insert code' button  located in the left-hand vertical toolbar.

Notes

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

This function allows for the insertion of code snippets previously written in the ['New code snippet'](#) tool in the 'Tools' menu.

The 'Use code snippet' drop down menu will show any previously written snippets, simply select the one you want to use.

If the drop down menu says 'Enter code' then code can be entered directly into the lower window.

Click <OK> to apply it as a operation.

Notes tab

Notes

Use this area to store any notes you may wish to make regarding the operation.

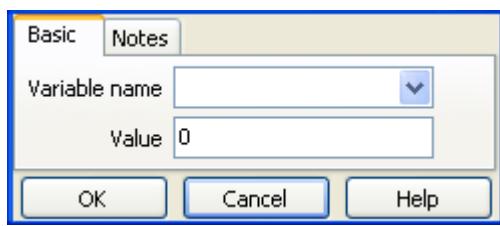
Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Help

'Help' opens the help file at the relevant location.

Set post variable

Basic tab



This function is used to set post processor variables. Sometimes a particular post may implement features that are not otherwise available in SheetCam. 'Set post variable' is a way of passing parameters to the post processor to control these functions. The variable names and functions are defined in the post.

Variable name

Variable name is defined in the post processor. Use the drop down menu to select an item (if available).

Value

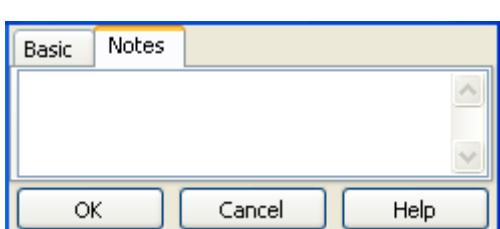
Value is the value that will be applied to that variable.

Help

'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Set post processor variable' button  located in the left-hand vertical toolbar.

Notes tab



Notes

Use this area to store any notes you may wish to make regarding the operation.

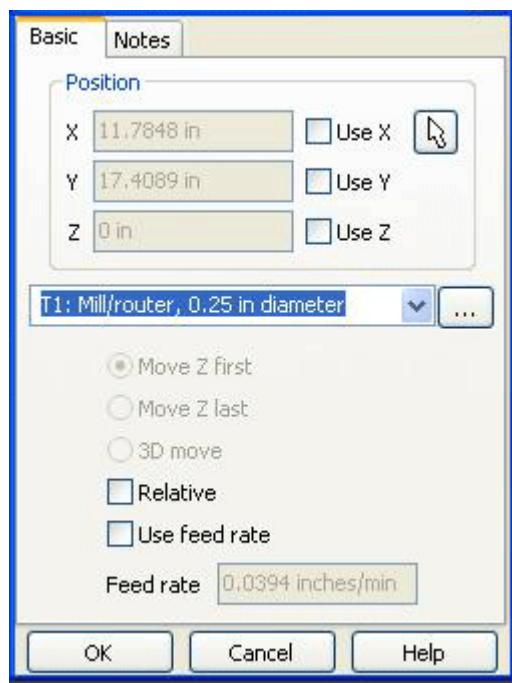
Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Help

'Help' opens the help file at the relevant location.

Move to

Basic tab



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Position

Click on the 'Use...' button to enable that axis and enter the location in the box.

Note: The move only takes place in the selected axis (or axes) so selecting only the Y axis would not move the X or Z axes.

Mouse select button

Clicking on the 'Mouse select' button temporarily closes the dialog box and allows you to click anywhere on the table or workpiece in the main viewing window. The click position is then recorded and displayed when the dialog box reappears.

Tool type

Select the tool type from the drop down menu.

Move

The move radio buttons only become active if the Z axis is selected above.

Move Z first - Moves the Z axis before the X and/or Y axes.

Move Z last - Moves the Z axis after the X and/or Y axes.

3D move - Moves all selected axes at the same time.

Relative

Clicking this box forces the move(s) to be made relative to the current location rather than to the machine zero location.

Note: You **cannot** use a relative move as the first operation. SheetCam has no way of knowing where the machine is when you start the code so it cannot move relative to an unknown position.

Use feed rate

If 'Use feed rate' is disabled the move will be a G0 rapid move. If it is enabled the move will be a G1 move at the feed rate specified in the box below.

Feed rate

Enter the required feed rate in the box.

Note: This dialog box can also be accessed via the 'Move to' button  located in the left-hand vertical toolbar.

Notes tab



Notes

Use this area to store any notes you may wish to make regarding the operation.

Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Post process

Once you have all your tools and processes set up it is time to 'post' your process. Select 'Run post processor' from the 'File' menu. This runs the 'post processor' chosen from the '[Options\Machine\Post processor](#)' menu. This option can also be activated by clicking on the  button on the toolbar.

Open and save options

Once you have 'posted' your G-code you can save various parameters for later use. The 'File' menu has the following options to open and save files.

Job files

New job

Opens a new job.

Open job

Opens a pop-up window allowing you to browse for an existing job.

Recent jobs

Opens a pop-up window showing the last five jobs opened.

Save job

Opens a pop-up window allowing you to save your current job as a 'job' file (.job).

Save job as

Opens a pop-up window allowing you to save your current job under a different name or location as a 'job' file (.job).

Template files

Open job template

Opens a pop-up window allowing you to browse for an existing job template.

Recent job templates

Opens a pop-up window showing the last five job templates opened.

Save job as template

Opens a pop-up window allowing you to save your current job under a different name or location as a 'job template' file (.jobt).

Part files

New part

Opens a pop-up window allowing you to browse for your part file. Acceptable formats are DXF, HPGL or Excellon files.

Open part

Opens a pop-up window allowing you to browse for a previously saved 'part' file (.part).

Recent parts

Opens a pop-up window showing the last five parts opened.

Save part

Opens a pop-up window allowing you to save your current part as a 'part' file (.part).

Save part as

Opens a pop-up window allowing you to save your current part under a different name or location as a 'part' file (.part).

Drawing files

Import drawing

See [Page 11](#) of this manual for 'Import drawing' options.

Recent drawings

Opens a pop-up window showing the last five drawings opened.

Toolset files

Open toolset

Opens a pop-up window allowing you to browse for a previously saved 'tools' file (.tools).

Save toolset

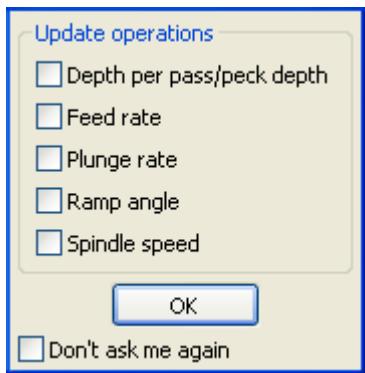
Opens a pop-up window allowing you to save your current tools as a 'tools' file (.tools).

Save default toolset

Makes the current toolset the default. This is the toolset that will be loaded when TNG first starts.

Note: A 'toolset' contains a list of all the tools you have defined.

Load default toolset



when loading a default toolset.

Update operations

If any of the boxes are ticked their parameters are applied to any **existing operations** that use that tool. Future operations are not affected. For instance if Tool 1 has a spindle speed of 1000 rpm and you select 'Spindle speed' then all operations that use Tool 1 will have their spindle speed set to 1000 rpm. Click <OK> after making any selections to save the changes.

Don't ask me again

If you check this box the dialog box above will not appear

Other features

Congratulations, you have just ‘worked through’ the procedures for setting up your first SheetCam job. There are a number of other features and functions within SheetCam and the following pages will look at these features in detail.

File Menu

Print

Prints the current item(s) in the 'view' panel. You must have a printer connected and 'online' to print.

Note: If the 'view' panel has a black background all the colours will be inverted to save ink.

Exit

Exits SheetCam. If you have any unsaved items you will be prompted to save first.

Edit Menu

Undo

Undo the last action.

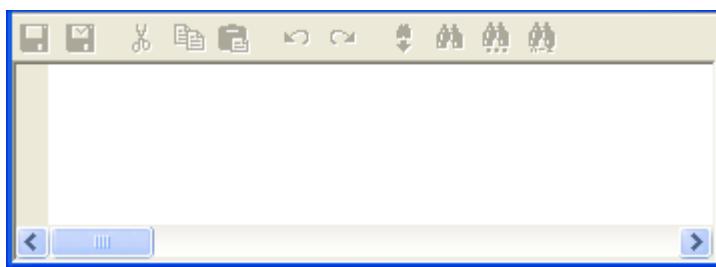
View menu

The view menu contains a number of viewing options.

Layer tool

To turn a layer on or off double-click on the check mark.
To rename a layer double-click on the name.
To minimize the palette click on the arrow button.
To close the palette click on the 'X' button.

Code editor



Selecting this menu item displays an editing window that allows you to edit/modify the generated G-code. The default location is at the bottom of the main window but the editing window can be 'dragged' to any location the user prefers. Drag it back to the bottom of the screen to restore it to its original position. Clicking on the 'X' in the top right corner will close the window.

Show input paths

Toggles the input paths on or off. This option can also be activated by clicking on the button in the toolbar.

Show segment ends

Toggles the segment ends on or off. This option can also be activated by clicking on the button in the toolbar.

Show path ends

Toggles the path ends on or off. This option can also be activated by clicking on the  button in the toolbar.

Note: This can be useful for locating 'non-joined' lines which may cause erratic behaviour.

Show tool paths

Toggles the tool paths on or off. This option can also be activated by clicking on the  button in the toolbar.

Show rapid moves

Toggles the rapid move paths on or off. This option can also be activated by clicking on the  button in the toolbar.

Show path directions

Toggles the path directions on or off. This option can also be activated by clicking on the  button in the toolbar.

Show machine and work

Toggles the machine and work on or off. This option can also be activated by clicking on the  button in the toolbar.

Show start points

Toggles the start points on or off. This option can also be activated by clicking on the  button in the toolbar.

Show true width

Toggles the true width of cut on or off. This option can also be activated by clicking on the  button in the toolbar.

Zoom menu

Zoom in

Zooms the 'view' in. This option can also be activated by clicking on the  button in the toolbar or by pressing <Page Up> on the keyboard.

Note: If you have a 'scroll mouse' the 'mouse wheel' can also be used to zoom in and out. If the zoom directions are opposite to the way you like to work you can change the direction of zoom using the '[Options\Application options\Reverse mouse wheel](#)' option.

Zoom out

Zooms the 'view' out. This option can also be activated by clicking on the  button in the toolbar or by pressing <Page Down> on the keyboard.

Zoom to fit current part

Zooms the 'view' panel to fit the current part. This option can also be activated by clicking on the  button in the toolbar.

Zoom to fit job

Zooms the 'view' panel to fit the job. This option can also be activated by clicking on the  button in the toolbar.

Zoom to fit window

Clicking on this button will zoom the view to fit the window 'dragged' on the main screen.

This option can also be activated by clicking on the  button in the toolbar.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit material

Zooms the 'view' panel to fit the material. This option can also be activated by clicking on the  button in the toolbar.

Zoom to fit machine

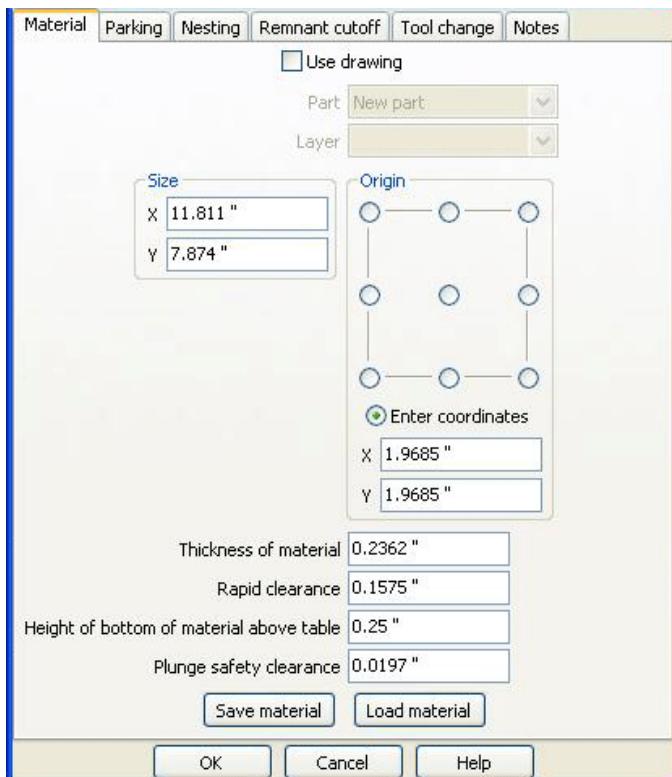
Zooms the 'view' panel to fit the machine. This option can also be activated by clicking on the  button in the toolbar.

Options menu

Job Options

Material tab

Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.



The 'Material' dialog box allows you to specify material parameters.

Use drawing

Checking this box allows you to use a drawing to specify the material shape rather than having a simple rectangle. Checking the box also activates the two drop down menus below:

Note: Very complex shapes will slow down the graphic display in the main window. It is recommended that you have a reasonably powerful graphics card on your machine if you plan to use this feature.

Part

Use this drop down menu box to select the part in the drawing.

Layer

Use this drop down menu box to select the layer in the drawing.

Size

Enter the size of your material in the X and Y directions.

Origin

Clicking on one of the 'radio buttons' will quickly establish the material origin relative to the machine origin.

Clicking on the 'Enter coordinates' button will allow you to enter coordinates in the 'coordinates' windows.

Note: 'Origin' is usually taken as the bottom left corner of the machine but can be anywhere.

Thickness of material

Enter the material thickness here.

Rapid clearance

Enter the the amount of 'clearance' you wish to apply during rapid moves.

Note: This is a **positive** (+) figure as you are specifying the '**clearance**' amount and will be the height above Z0 (which is **always** the top face of the material) to the bottom of the tool. Z axis travel below the rapid clearance plane can be at G0 rate. It only switches to G1 at the plunge safety clearance.

Height of bottom of material above table

Enter the distance the bottom of the material sits above the table.

Note: This is a **positive** (+) figure as you are specifying the '**height**'. This is used where the part requires a 'sacrificial board' under it in order to machine it (i.e. a 'through' pocket or cut-out in a sheet of material). SheetCam **will** let you cut into the board but it **will not** let you cut into the table.

Plunge safety clearance

Plunge safety clearance is used when doing a rapid move to cut height. When SheetCam moves to the top of the cut it stops slightly short and moves the remaining distance at feed rate. This allows for inaccuracies in your setup and any chips that may be in the way. The last thing you want is to slam the tool into the work at full speed. By default the clearance is 0.5mm which seems to work pretty well in most cases. Older versions of SheetCam have always had this hard coded to 0.5mm. Enter your required distance in the box.

Save material

Clicking this button will allow you to save the material settings to a suitable location.

Load material

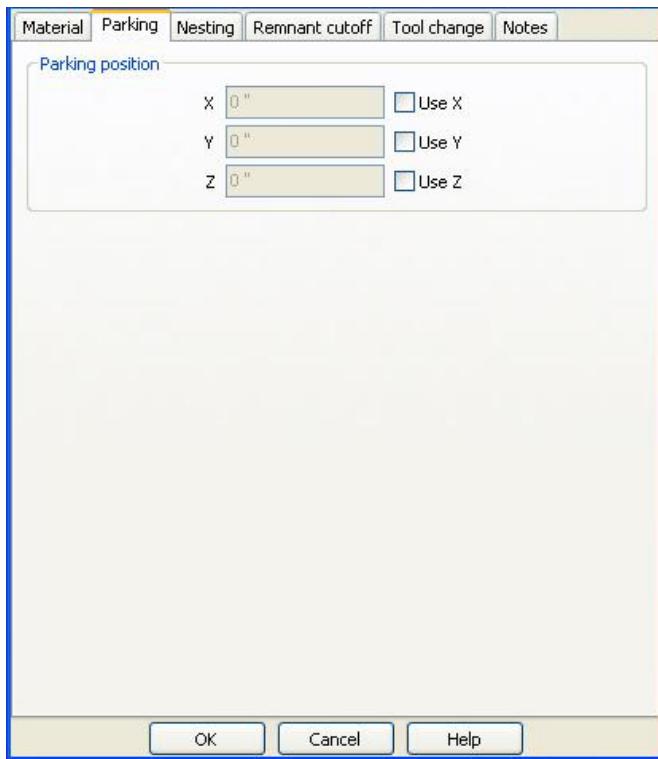
Clicking this button will allow you to load a previously saved material.

Help

'Help' opens the help file at the relevant location.

Parking Tab

Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.



The 'Parking' dialog box allows you to specify parking position parameters.

Parking position

If any of the 'Use' position boxes are checked then the machine will go to the position entered into the preceding box at the end of the program. For instance, if you just want the tool to lift well clear of the work at the end of the program then just enter the required position and place a check mark in the Z position check box. If you want to move to a specific X,Y coordinate (i.e. to clear a work load/unload area) then use those as well.

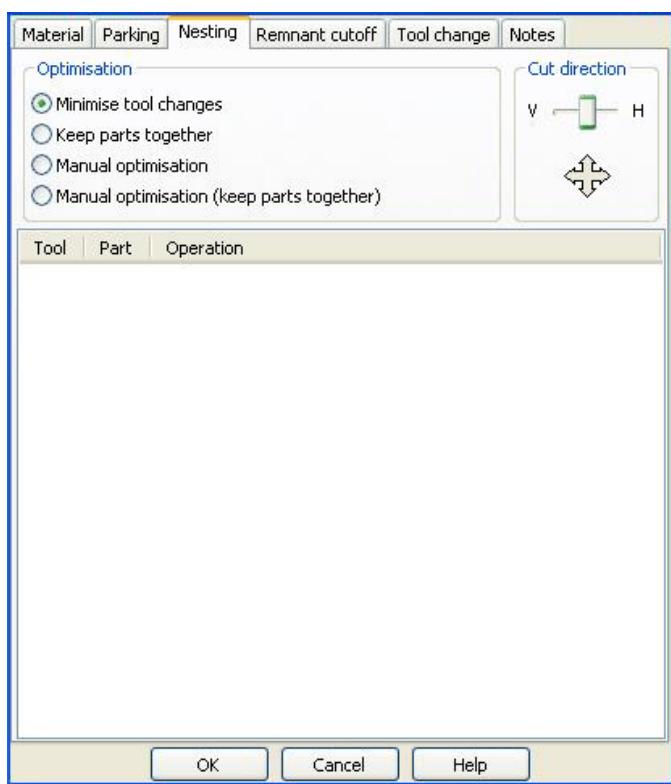
Note: If a Z position is specified and it is above the rapid height then the Z axis will move first and then the X,Y axes simultaneously. If Z is below the rapid clearance height (unlikely but it is allowed for) then the Z axis moves last.

Help

'Help' opens the help file at the relevant location.

Nesting Tab

Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.



Optimisation

Minimise tool changes

For multi-part jobs, SheetCam will try to order the operations in all parts to minimise tool changes. All of the operations in any specific part will always be cut in order but operations in different parts may be cut together. This can save time while milling/routing but is not a good idea for plasma and laser as material distortion could cause accuracy problems.

Keep parts together

Each part is cut before the next one (i.e. the opposite of the above).

Manual optimisation

This feature allows you to drag operations up and down the list. You **cannot** change the order of operation in any one part.

For example let's assume you have two parts:

part 1 op 1
part 1 op 2
part 2 op 1
part 2 op 2

You can drag part 1 op 2 anywhere in the list as long as it appears after part 1 op 1. Once you have moved it you can move part 1 op 1 as long as it appears before part 1 op 2. The same applies for part 2. Basically it is a manual version of 'Minimise tool changes'

Manual optimisation (keep parts together)

This feature is similar to the one above but you can only move whole parts up and down the list. In this mode you **cannot** move individual operations. This is basically a manual version of 'Keep parts together'.

Cut Direction

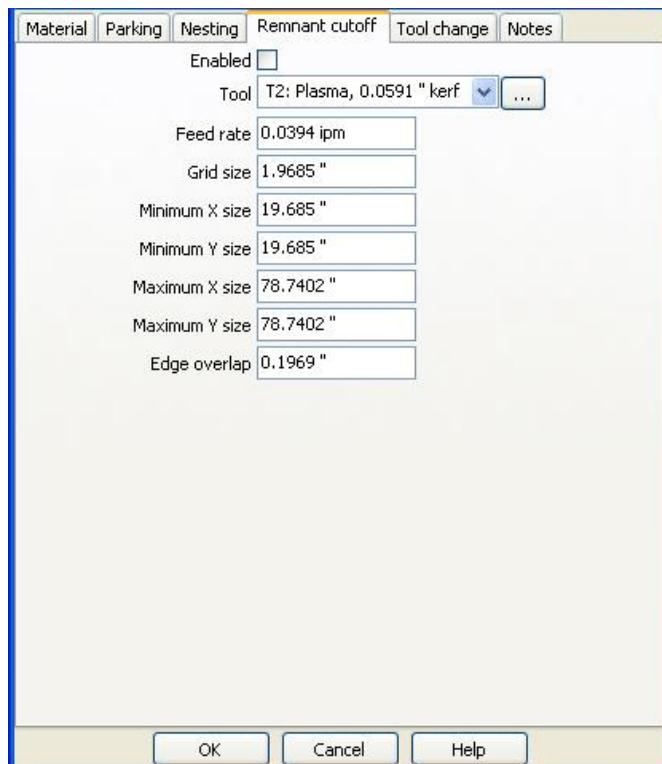
When optimising the part cut order, this sets the preferred cut direction. For instance, if it is fully towards 'H', the optimisation will prefer to move to the next part on the left/right rather than up/down.

Help

'Help' opens the help file at the relevant location.

Remnant cutoff tab

Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.



The 'Remnant cutoff' dialog box allows you to specify parameters for removing scrap material from the sheet.

Note: This function is only available for jet cutters and the tab only appears if jet cutting is enabled in the '[Options/Machine](#)' dialog box.

Enabled

Checking this box turns this feature on/off.

Tool

Select the correct tool using the drop down menu.

Feed rate

Enter the required feed rate here.

Grid size

Remnant cutoff calculates the remnant area by breaking the whole sheet into small squares. The grid size specifies the size of these squares. Decreasing the grid size increases accuracy but slows down processing. Generally 50mm or 2" is about right. Minimum and maximum sizes will be rounded to multiples of this value.

Minimum X,Y size

These boxes specify the minimum useful remnant sizes. SheetCam will do its best not to generate remnants smaller than this.

Maximum X,Y size

Large pieces of heavy material can be a problem to handle so these boxes specify the largest remnant size. All remnants will be no greater than this.

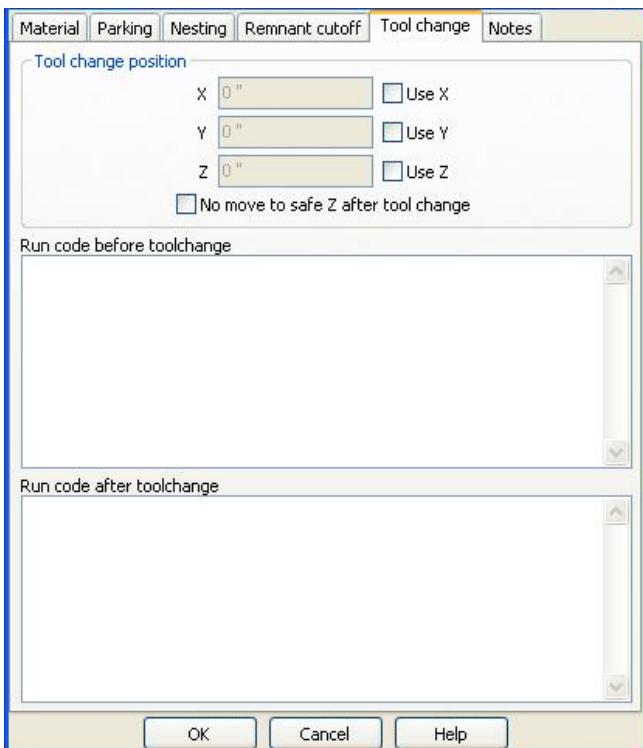
Edge overlap

It is usually not advisable to stop the torch exactly on the edge of the sheet. If the sheet is slightly misaligned then the remnants may be stuck together with small uncut tabs. This value makes the torch overshoot the edge by the given amount. When cutting remnants SheetCam will always cut towards the edge of the sheet. It will never start on an edge and cut inwards.

Help

'Help' opens the help file at the relevant location.

Tool change tab



Note: All units based on the selection in the '[Options\Application options\Units](#)' menu.

Tool change position

Click in the small box to the right and enter the tool change position in the window on the left. The 'position' is the distance from the machine zero point (set using the '[Options\Machine\Working envelope](#)' dialog box) to the location chosen as the tool change position, this can be a positive (+) **or** a negative (-) figure.

No move to safe Z after tool change

Checking this box prevents the tool from moving to the safe Z position after a tool change.

Run code before toolchange

If you need to run a block of code prior to a tool change it can be entered here. This is run before the X, Y, Z position moves. For instance if you are using tool length offsets on the machine you can cancel the offset and move the Z axis to a safe height. Examples are:

- G0 Z??? - where ??? is a position to move the Z axis to in order to clear clamps etc. prior to the move to the toolchange position.
- M0 - Pause the program execution to allow you time to change tools.

Run code after toolchange

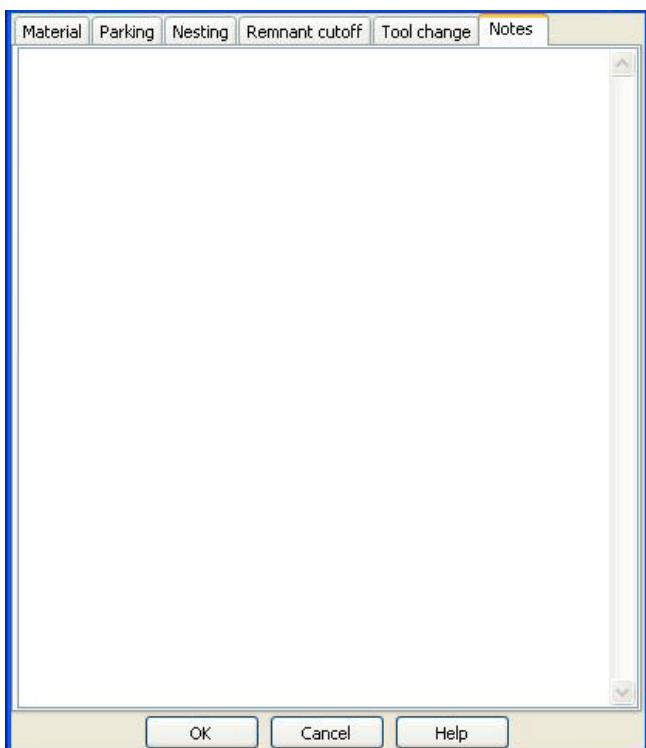
If you need to run a block of code after a tool change it can be entered here. Examples are:

- X/Y/Z ??? - where??? is a suitable location to start the next operation.
- Code to reference/zero the tool.

Help

'Help' opens the help file at the relevant location.

Notes tab



Notes

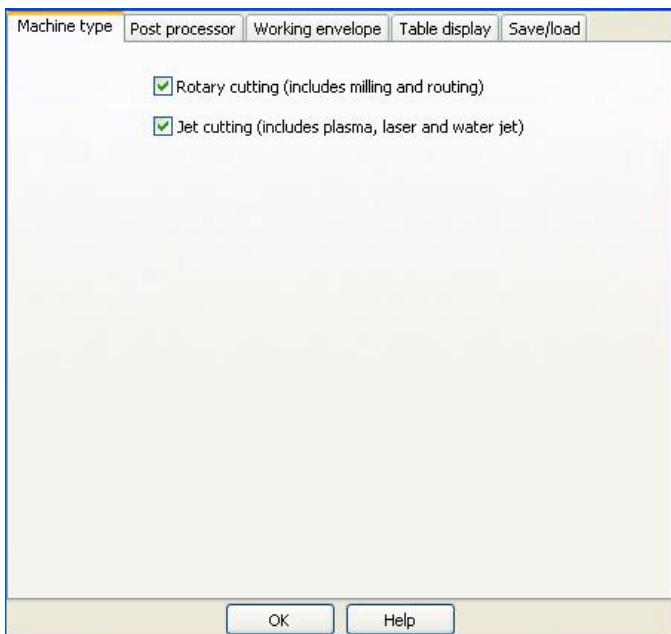
Use this area to store any notes you may wish to make regarding the Job options settings.

Help

'Help' opens the help file at the relevant location.

Machine options

Machine type tab



Note: At least **one** option **must** be selected at all times.

Rotary cutting

Checking this item will toggle the rotary cutting option on or off. When checked the rotary cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden.

Jet cutting

Checking this item will toggle

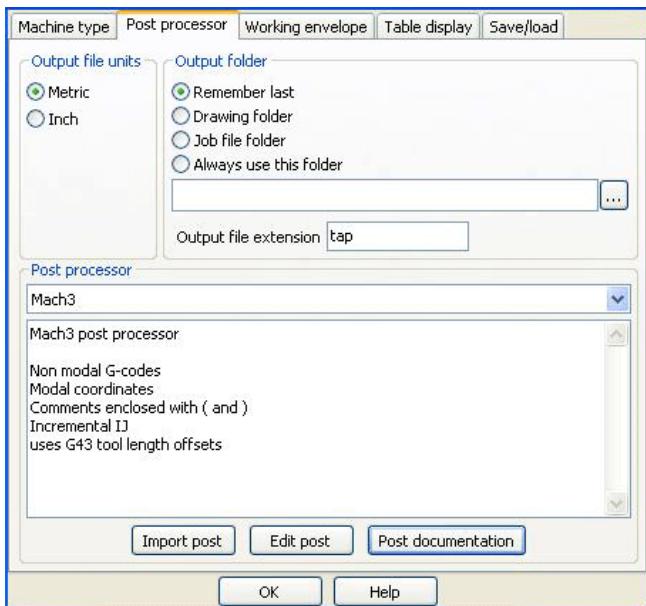
the jet cutting option on or off. When checked the jet cutting 'controls' are available under the 'Tools' menu and the 'Operation' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden.

Note: Check both boxes if you want rotary controls **and** jet cutting controls available under the 'Tools' menu and the 'Operation' menu.

Help

'Help' opens the help file at the relevant location.

Post processor tab



option enter the required extension for your CNC controller (txt, cnc, tap, nc, etc.).

Post processor

Select the correct processor for your machine controller using the drop down menu. A brief description of the processor file will appear in the lower window (if available).

Import post

This button allows you to import a new processor file.

Edit post

This button will open an editing tool which will allow you to edit the selected post processor.

Post documentation

This button opens up a window explaining the 'Post' language and functions.

Help

'Help' opens the help file at the relevant location.

Output file units

Selecting either 'Metric' or 'Inch' will determine the units used by the post processor.

Output Folder

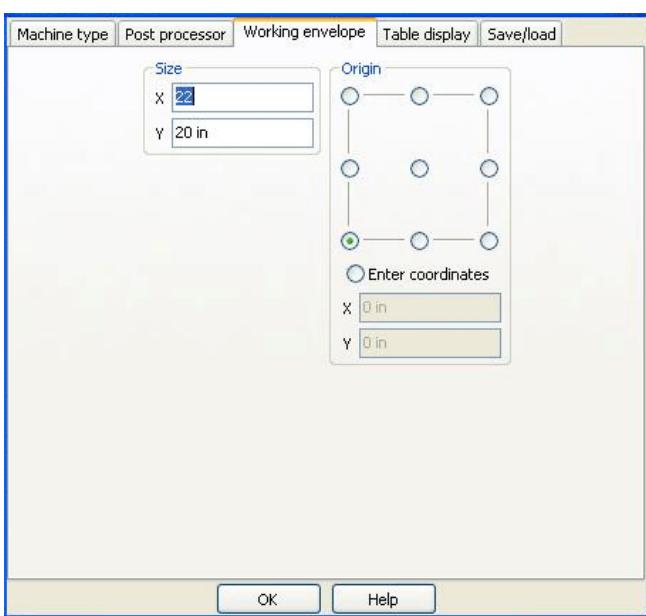
Use the 'radio buttons' to select the folder to be used when saving the post process file.

For the 'Always use this folder' option use the navigation button on the right (the one displaying three dots) to select the relevant location.

For the 'Output file extension'

option enter the required extension for your CNC controller (txt, cnc, tap, nc, etc.).

Working envelope tab



This tab controls the size of the machine working envelope.

Size

This is the area of the machine that the cutter can physically cut.

Origin

Clicking on one of the 'radio buttons' will quickly establish the working origin relative to the dimensions entered in the 'Size' windows.

Clicking on the 'Enter coordinates' button will allow you to enter coordinates in the 'coordinates' windows.

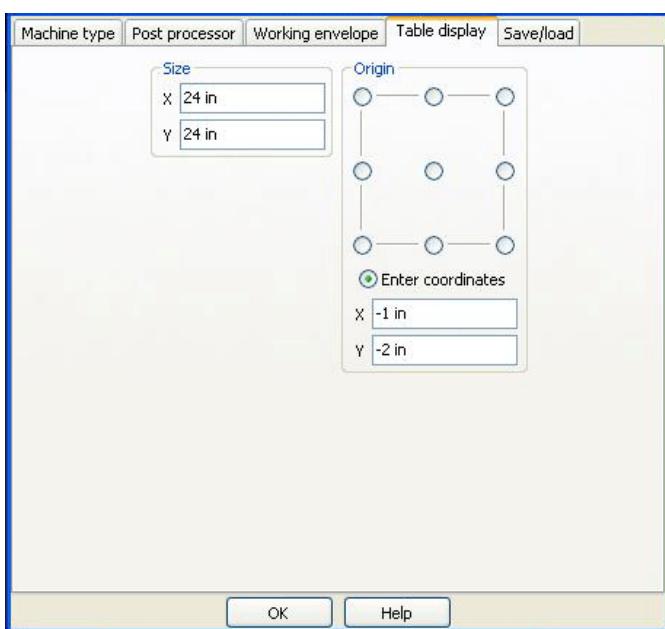
Note: 'Machine origin' is usually taken as the bottom left corner of the machine but can be anywhere.

Note: This item is purely for reference and it does not affect the G-code in any way. If you do not wish to use this feature set the sizes to 0, 0.

Help

'Help' opens the help file at the relevant location.

Table display tab



Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.

This tab controls the size of the machine table displayed by SheetCam.

Size

Enter the size of the table in the X and Y directions.

Origin

Clicking on one of the 'radio buttons' will quickly establish the machine origin relative to the dimensions entered in the 'Table size' windows.

Clicking on the 'Enter coordinates' button will allow you to enter coordinates in the 'coordinates' windows.

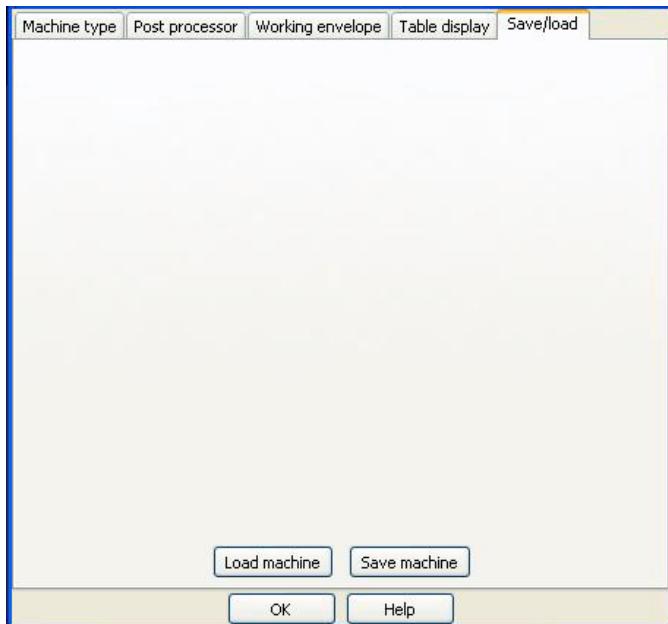
Note: 'Machine origin' is usually taken as the bottom left corner of the machine but can be anywhere.

Note: This item is purely for reference and it does not affect the G-code in any way. If you do not wish to use this feature set the sizes to 0, 0.

Help

'Help' opens the help file at the relevant location.

Save/load tab



Load machine

This button will load the settings of a previously saved machine.

Save machine

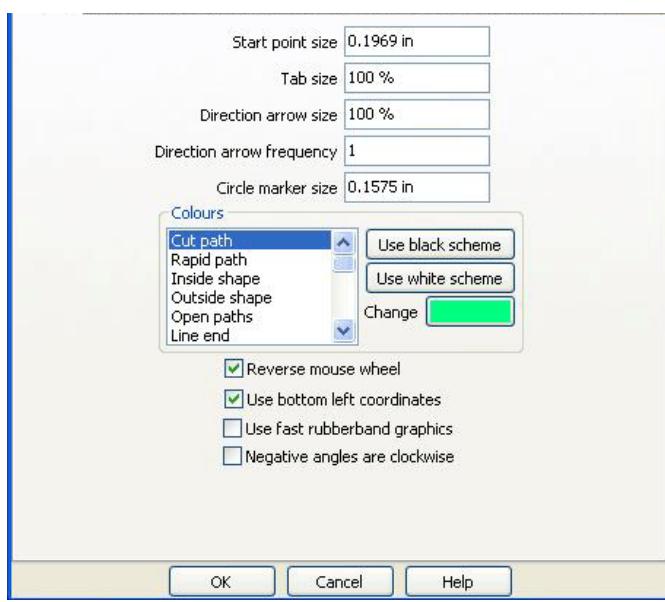
This button will save the current settings to a suitable location.

Help

'Help' opens the help file at the relevant location.

Application options

Display tab



Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.

Start point size

This function controls the size of start point markers. The size depends on the size of your drawing. If they are too large reduce this value, if they are too small increase this value.

Tab size

This function controls the size of the tabs. Enter a value as a percentage of the current size.

Direction arrow size

The size depends on the size of your drawing. If they are too large reduce this value, if they are too small increase this value.

Note: This function is used to control the size of the direction arrows shown on the cutter path display and is a percentage of the tool diameter.

Direction arrow frequency

This is used to control the frequency of the direction arrows shown on the cutter path display. For instance entering a figure of 5 would result in five vectors between each arrow location while entering 8 would effectively space the arrows further apart.

Circle marker size

This function controls the size of hole centre markers. The size depends on the size of your drawing. If they are too large reduce this value, if they are too small increase this value.

Note: Hole centre markers will only update when you next open a drawing.

Colours

Click on an item you would like to change in the scrolling menu and then click in the 'Change' box to pick a new colour.

Colour scheme

Clicking on either of the 'scheme' buttons will change the background colour of the main view panel.

Reverse mouse wheel

Checking this item will switch the way the mouse wheel controls the zoom in the main view panel.

Use bottom left coordinates

The part position boxes at the bottom of the screen normally show the coordinates of the centre of the part. Checking this item changes them to show the coordinates of the lower left corner.

Use fast rubberband graphics

Some temporary graphics can be quite slow on slow graphics cards, for instance when moving text or when dragging a box around several items to select them. Enabling this option can speed up the display but the graphics may flicker. It may also actually slow down some fast graphics cards.

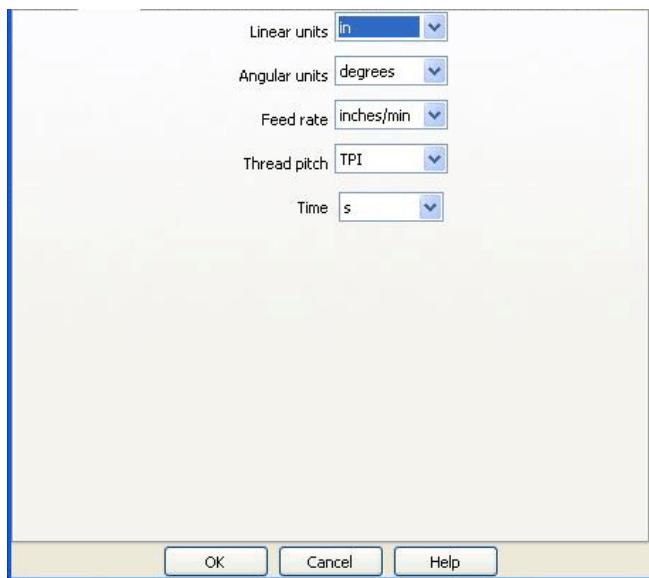
Negative angles are clockwise

SheetCam shows the part angle with positive angles clockwise and negative angles counter-clockwise by default. This option reverses the angles.

Help

'Help' opens the help file at the relevant location.

Units tab



This dialog box is used to set up the 'base' or 'preferred' units used by many of SheetCam's functions.

Linear units

Enter the linear units from the large drop down menu.

Angular units

Enter the angular units from the drop down menu.

Feed rate

Enter the feed rate units from the drop down menu.

Thread pitch

Enter the thread pitch units from the drop down menu.

Time

Enter the time units from the drop down menu.

Note: You can use **any units** when entering values into **any** edit boxes (with the exception that linear edit boxes must have linear units and angular edit boxes must have angular units). If you do not specify any units then the preferred units (as set above) are used. If you specify any of the units in the list they will be converted to your preferred units as soon as you leave the edit box.

Example 1: If you type 10mm (with the unit designator **mm**) into a cut depth box, it will change to 0.3937 inches as soon as you leave the box (assuming your preferred units are inches).

Example 2: If you type 1" (with the unit designator **"**) into a cut depth box, it will change to 25.4mm as soon as you leave the box (assuming your preferred units are mm).

Example 3: If you type 1 inch (with the unit designator **inch**) into a cut depth box, it will change to 25.4mm as soon as you leave the box (assuming your preferred units are mm).

Help

'Help' opens the help file at the relevant location.

Annoying dialogs tab



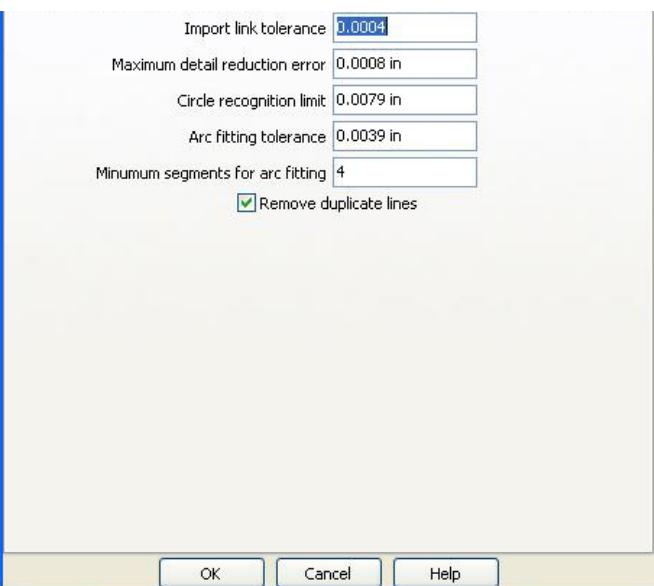
Dialog selection

Check the relevant boxes to prevent SheetCam from displaying them again. Uncheck them to have the dialog boxes reappear.

Help

'Help' opens the help file at the relevant location.

Drawing import tab



Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.

This dialog box is used to set up the 'Drawing import' settings used by many of SheetCam's functions.

Note: These values should never need changing under 'normal' conditions.

Import link tolerance

Due to rounding errors or

drawing inaccuracies sometimes lines and arcs in an outline do not join up exactly. If they are this distance apart or closer the ends are moved so that they join up exactly. This should not be used to compensate for poor drawings.

Max detail reduction error

Some drawings consist of a large number of very small line segments (especially polyline curves converted to DXF). This can slow down processing and produce very large g-code files. It can also slow down machining. SheetCam tries to reduce the number of lines when it loads the drawing. Removing lines introduces small errors and this value is the maximum allowable error. For Plasma cutting you can afford to have quite a large error (say .020"). The setup wizard adjusts this value to suit.

Circle recognition limit

When SheetCam loads a drawing it looks for circles to identify holes for drilling.

Arc fitting tolerance

If a series of line segments approximate an arc, SheetCam will convert them to an arc. This tolerance is the maximum error introduced by the conversion. Increasing the tolerance will create more arcs, resulting in smoother cutting and smaller code.

Minimum segments for arc fitting

This is the minimum number of line segments that can be considered to be part of an arc.

Remove duplicate lines

Sometimes drawings can contain multiple overlapping copies of the same lines. These lines normally confuse SheetCam. If this option is enabled SheetCam will delete all but one copy of the lines. This is time consuming so if you have very complex drawings that take a long time to load, turning this option off will speed up loading considerably. It is recommended to keep this on unless your drawings do take a long time to load and you are really sure there are no overlapping lines.

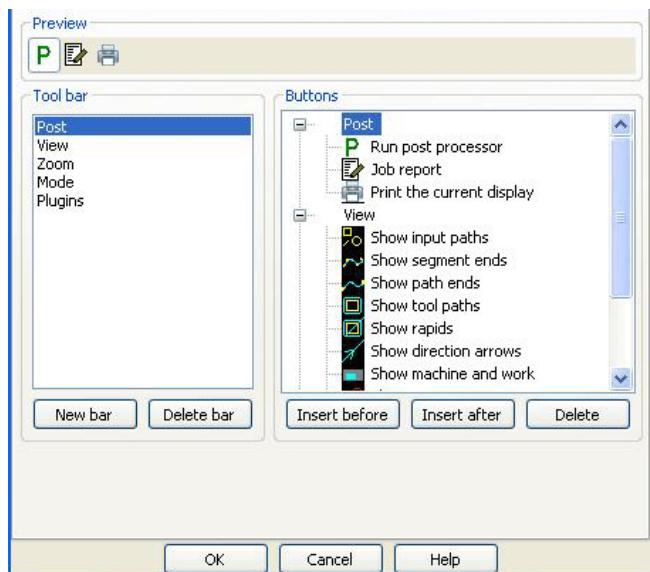
Help

'Help' opens the help file at the relevant location.

Toolbars tab

SheetCam includes a toolbar editor that allows you to make custom toolbars. Right click on any tool bar to open the editor dialog box shown below.

Preview



The preview area shows the current tool icons plus any additions you make.

Tool bar

Select the toolbar you wish to edit from the available list.

New Bar

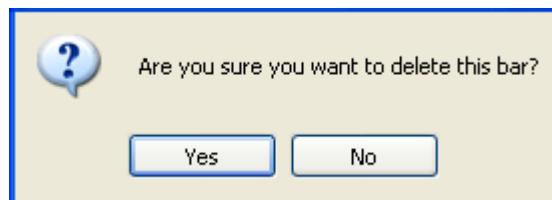
Clicking the 'New bar' button opens up the dialog box shown below:



Enter the name for the new bar and click <OK>. The new bar will then appear in the 'Tool bar' pane.

Delete bar

To delete a toolbar first click on the name in the 'Tool bar' pane and then click the 'Delete bar' button. The following dialog box will then appear:



Click <Yes> to delete the toolbar.

Buttons

This pane shows the currently assigned tool icons for the toolbars. Click on the '+' button to expand the tree or the '-' button to contract a tree.

Insert before

This button inserts a new toolbar icon before the selected one. Click on a toolbar icon in the 'Preview' pane then scroll down to find the toolbar icon you wish to add, select it and then click <Insert before>.

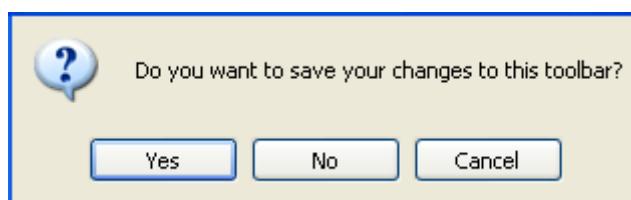
Insert after

This button inserts the new toolbar icon after the selected one. Click on a toolbar icon in the 'Preview' pane then scroll down to find the toolbar icon you wish to add, select it and then click <Insert after>.

Delete

This button deletes a toolbar icon from a toolbar. Click on a toolbar icon in the 'Preview' pane then click <Delete>.

Note: Whenever you switch between toolbars in the 'Tool bar' pane you will get the following dialog box:

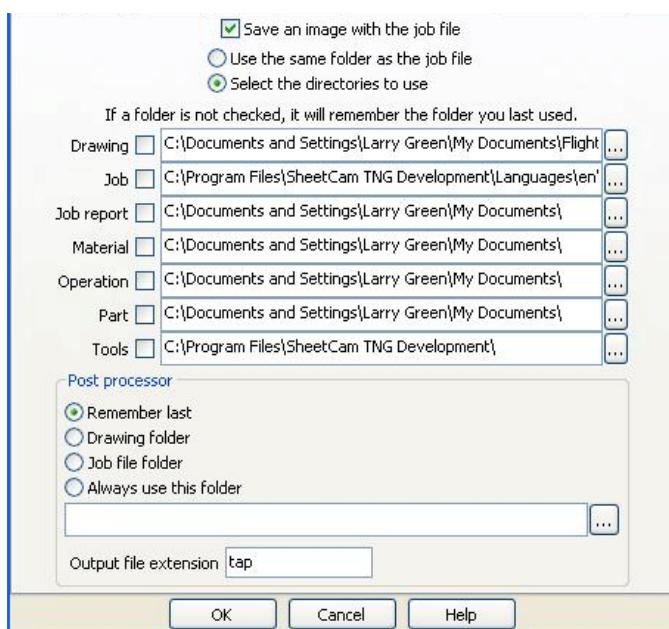


Select <Yes>, <No> or <Cancel> as required.

Help

'Help' opens the help file at the relevant location.

Default directories tab



Save an image with the job file
Check this box to save an image of the job (in PNG format) along with the job file.

Use the same folder as the job file

Click this 'radio button' to use the same folder as the job file.

Select the directories to use

Click this 'radio button' to select the directories to use and then select the directories using the menu items below.

Directory locations

These dialog boxes are used to display the default directory locations. To change the default location click on the check box and then click on the button to the right (the one displaying three dots) and select a new location.

Note: In practice each of the boxes shown above will contain information relevant to your particular computer/installation.

If a folder is not checked SheetCam will use the last used folder.

Post processor

Use the 'radio buttons' to select the folder to be used when saving the post process file.

For the 'Always use this folder' option use the navigation button on the right (the one displaying three dots) to select the relevant location.

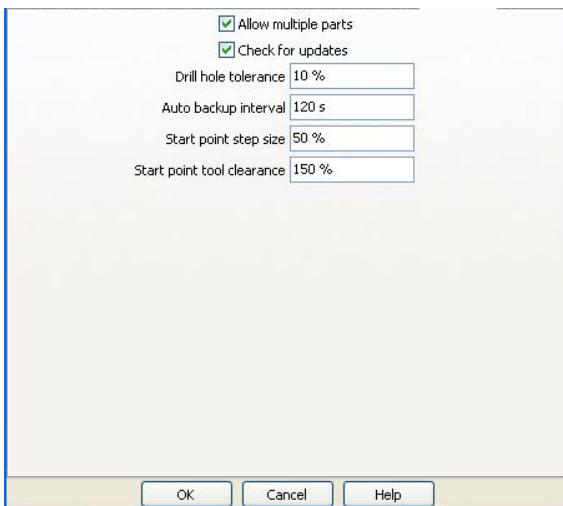
For the 'Output file extension' option enter the required extension for your CNC controller (txt, cnc, tap, nc, etc.).

Help

'Help' opens the help file at the relevant location.

Advanced tab

Note: All units based on the selection in the '[Options/Application options/Units](#)' menu.



This dialog box is used to set the 'Advanced' settings used by many of SheetCam's functions.

Note: These values should never need changing under 'normal' conditions.

Allow multiple parts

Checking this item will allow you to have multiple parts within in the same piece of material specified in the '[Options/Job options/Material](#)' dialog box.

Check for updates

With this enabled, the first time SheetCam is started each day, it will check the SheetCam website for updates.

Note: This requires that the computer is connected to the Internet to complete this function.

Drill hole tolerance

When you create a drilling operation SheetCam automatically sets the minimum hole diameter and maximum hole diameter to be just under and just over the actual drill size. This value controls how much tolerance is allowed. If you increase this number the minimum hole diameter will decrease and the maximum hole diameter will increase.

Auto backup interval

Enter the required time (in seconds) for the automatic backup function.

Note: The automatic backup is saved to a temporary file in the SheetCam program folder. If SheetCam is closed normally it deletes the temporary file. If it crashes, the next time you start SheetCam it will detect the file and load it.

Start point step size

If a start point's leadin or leadout will not fit where it has been placed, SheetCam automatically tries moving it to somewhere where it will fit. Decreasing this value makes the calculations more accurate but slows down path generation. In general you should not need to alter this value.

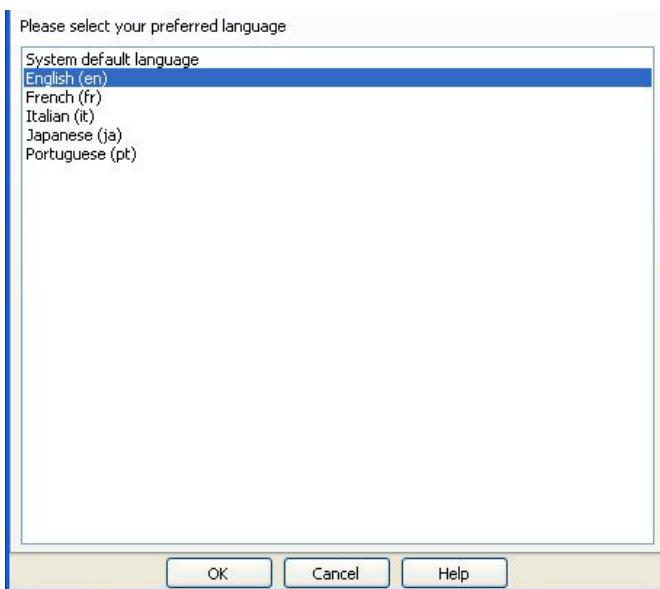
Start point tool clearance

When working out if a start point will fit, SheetCam allows for the tool plunging at the start of the leadin. For plasma, laser and waterjet cutting the pierce hole is bigger then the kerf width. This value is the percentage size increase. For instance if the pierce hole is twice the size, set this value to 200%.

Help

'Help' opens the help file at the relevant location.

Language tab

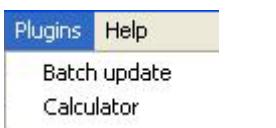


This dialog box is used to set the 'Language' used by SheetCam. Click on the language of choice and then click <OK>.

Help

'Help' opens the help file at the relevant location.

Plugins menu



Batch update

Selecting this option opens a dialog window as shown below:

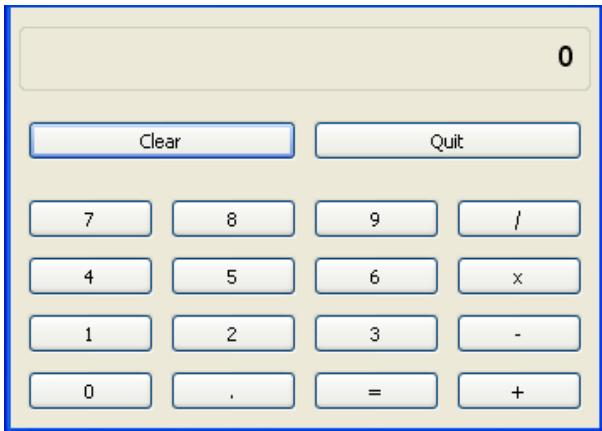


Use the directory tree to navigate to your required folder. SheetCam will load each job in turn then save it with an image (in PNG format).

Note: If your directory has a lot of files you may want to go and have a cup of coffee while you wait.

Note: This process puts a heavy stress on the SheetCam internal code so there is a possibility it may crash and some of the tool paths may not be generated on complex jobs. Check your job file afterwards if there is any doubt.

Calculator



Toggles a simple calculator on or off. This option can also be activated by clicking on the  button in the toolbar.

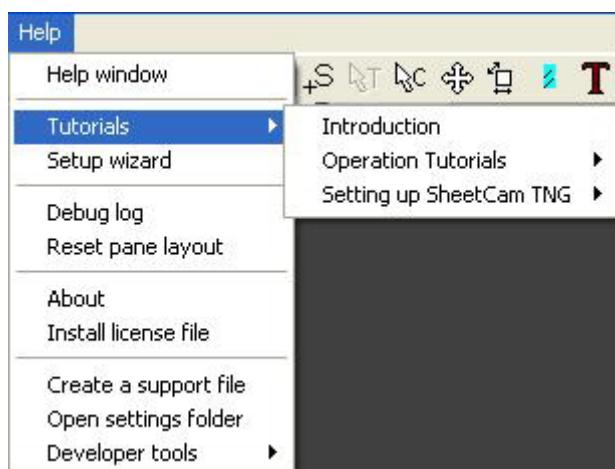
Help menu

Help

Calls up the help file.

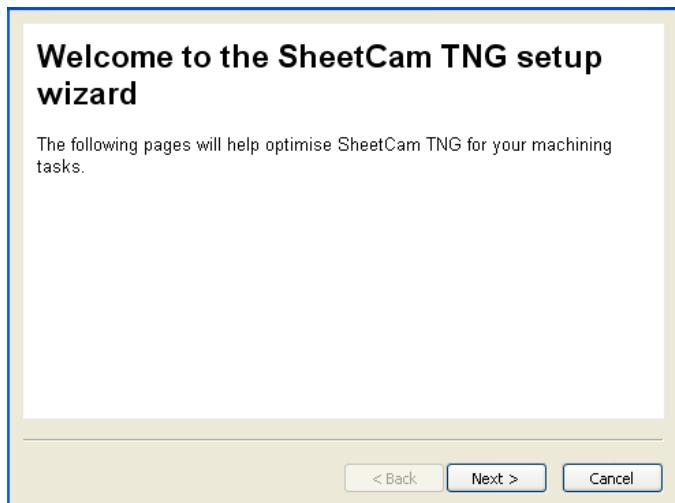
Tutorials

'Hovering' over or clicking this menu item will open the following pop-out dialog box.



Select a tutorial from the available choices.

Setup wizard



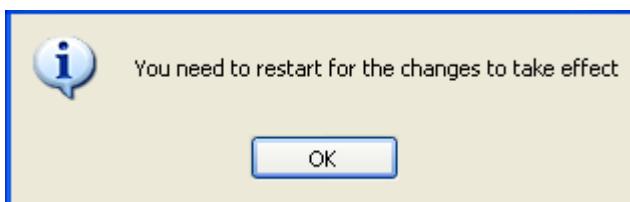
Click on <Next> and follow the on-screen directions.

Debug log

Calls up a 'Debug log' window at the bottom of the main window.

To close the 'Debug log' window either click on the 'X' in the upper right corner of the 'Debug log' window or select 'Help/Debug log' again to cancel the action.

Reset pane layout



This menu item resets the pane layout to the default settings. This can be useful if the panes get misplaced on the screen.

Note: Changes only take place after a restart of SheetCam.

About



This screen gives information on the current SheetCam version.

The 'Credits' section lists information on the tools used to build SheetCam.

Show licence

Clicking this button will open a window displaying the software licence agreement.

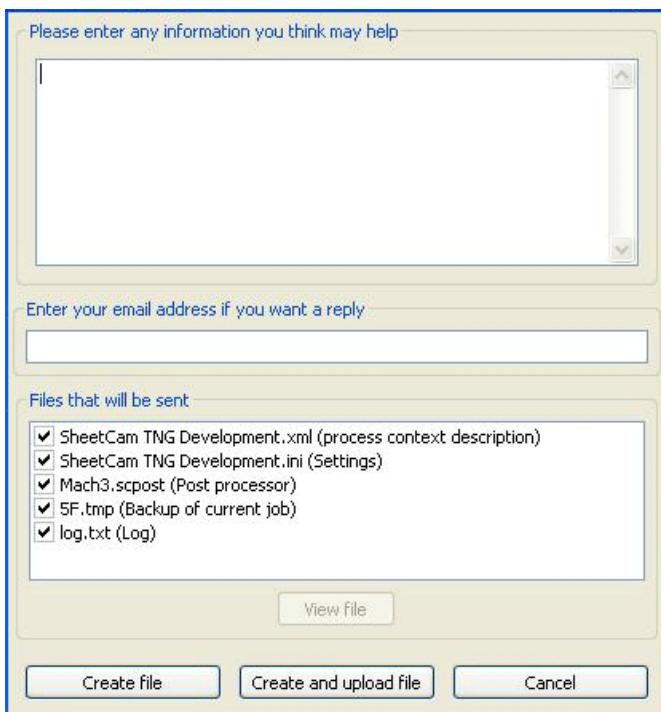
Install licence file

Locate the file 'license.camlic' and click <Open>. This will register your version of SheetCam.

Note: This requires a restart of SheetCam to complete the registration.

Create a support file

This will create a support file that can be sent to the creators of SheetCam for analysis.



Note: If possible you should have the problem job open when creating a support file.

Enter a brief description of the fault encountered in the upper text box.

Enter a valid email address in the middle text box. **You cannot receive a reply if you do not supply an address.**

Use the check boxes to select which items are to be included with the support file.

Click on <Create file> to save the support file to your hard drive as a zip file. Enter a suitable name/location in the appropriate

boxes and click <OK>.

Once saved you can access that file and attach it to an email for support.

Click on <Create and upload file> to automatically send the package. You **must** have an active internet connection at the time in order for this feature to work.

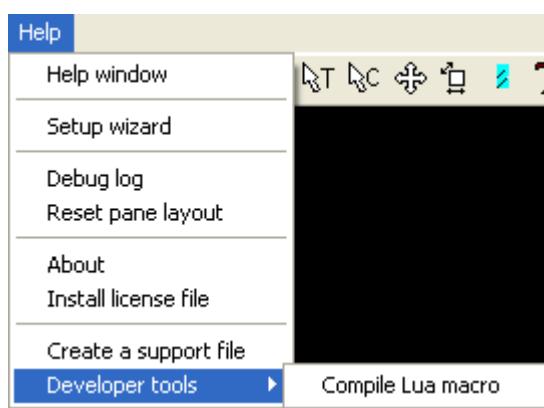
Click on <Cancel> to close the window.

Open settings folder

Opens a pop-up window showing the SheetCam settings folder.

Developer tools

Hovering your mouse over the 'Developer tools' menu item will open the following pop-out menu list:



Compile Lua macro

Selecting this item opens a pop-up window. Select the file you require and click <Open>.

Used mainly by developers. Plugins can be written in the Lua macro programming language. Normally these are plain text files. If you want to hide your code you can compile the plugin so no-one can read the code.

Toolbars

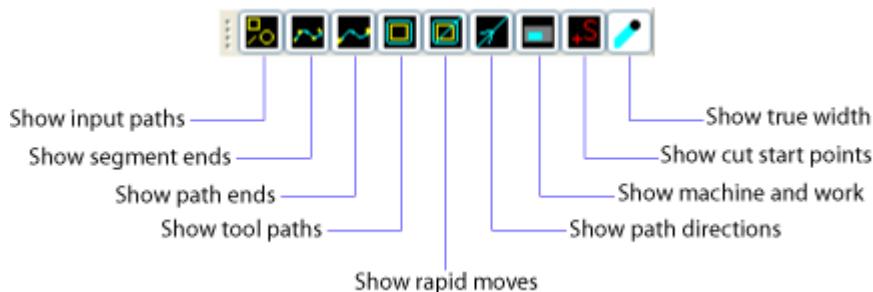
There are a number of toolbars within SheetCam and the following section details their various functions.

Note: All toolbars can be 'dragged' from their default locations by clicking and dragging on either the three dots to the left of the toolbar or by clicking and dragging the 'header bar' in the case of the 'Parts', 'Tools' and 'Operations' toolbars.

Click and drag to replace them at the default location.

Note: SheetCam includes a Toolbar Editor that allows you to make custom toolbars. It can be found under the 'Toolbars' tab of the '[Options\Application options](#)' menu item.

View toolbar buttons



Show input paths

Toggles the input paths on or off.

Show segment ends

Toggles the segment end points on or off.

Show path ends

Toggles the path end points on or off.

Note: This can be useful for locating 'non-joined' lines which may cause erratic behaviour.

Show tool paths

Toggles the tool paths on or off.

Show rapid moves

Toggles the rapid moves on or off.

Show path directions

Toggles the path directions on or off.

Show machine and work

Toggles the machine and work display on or off.

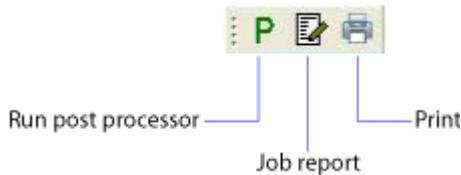
Show cut start points

Toggles the cut start points on or off.

Show true width

Toggles the true width of the cutter/jet on or off.

Run post processor toolbar buttons



Run post processor

Runs the post processor previously chosen from the '[Options\Machine\ Post processor](#)' menu.

Job report

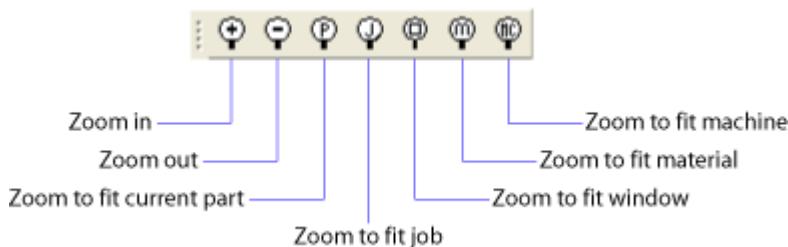
Clicking on this button will display the job report dialog box.

Print button

Clicking on this button will open the 'Print' set up window.

Note: This can also be accessed via the 'File\Print' menu item.

Zoom toolbar buttons



Zoom in

Clicking on this button will zoom the view in.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Note: If you have a 'scroll mouse' the 'mouse wheel' can also be used to zoom in and out. If the zoom directions are opposite to the way you like to work you can change the direction of zoom using the '[Options\Application options\Display\Reverse mouse wheel](#)' option.

Zoom out

Clicking on this button will zoom the view out.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit current part

Clicking on this button will zoom the view to fit the current part in the viewing window.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit job

Clicking on this button will zoom the view to fit the job in the viewing window.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit window

Clicking on this button will zoom the view to fit the window 'dragged' on the main screen.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit material

Clicking on this button will zoom the view to fit the material in the viewing window.

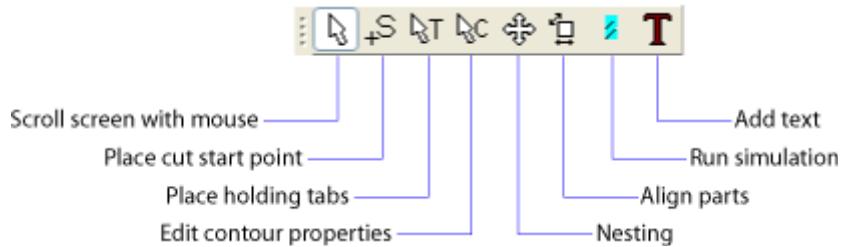
Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Zoom to fit machine

Clicking on this button will zoom the view to fit the machine in the viewing window.

Note: Zooming can also be achieved by using the <Page Up> and <Page Down> keyboard buttons.

Selection toolbar buttons



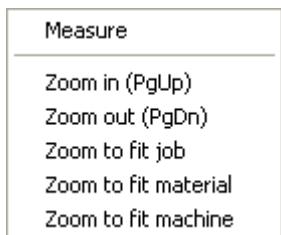
Scroll the screen with the mouse

When clicked allows you to scroll the view panel by 'dragging' with the mouse.

Note: Hold <Shift> and drag with the mouse to rotate the drawing in three dimensions. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Right click options

'Right clicking' while using the 'Scroll screen' selection tool will bring up the following menu screen.



Measure

To use the measure tool click on the point you want to measure from.

Note: If you click near a vertex (a join between two lines/arcs) or the centre of a circle the dimension will snap to that point.

When you move the mouse the coordinate display at the bottom of the screen shows the X,Y length and the number next to the cursor is the distance from the start point to the cursor. If you click on another point the measure start point will move to that location. You can cancel the measure function by switching to another mode, pressing <Escape> or by right clicking and selecting 'End measuring'.

Zoom controls

Clicking on any of these items will zoom the view as required.

Edit cut start points

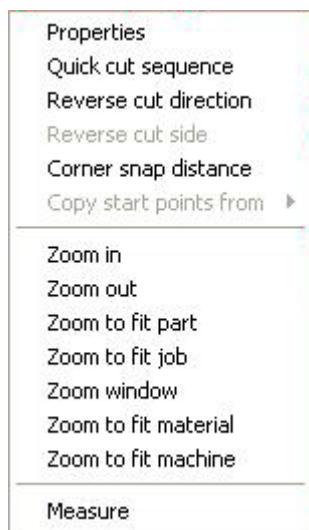
When clicked allows you to insert the 'Cut start point' on the part at a location of your choosing. The mouse cursor changes to a pointer with an 'S' if you are near a valid start point.

Note: Hold down the <Shift> key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.

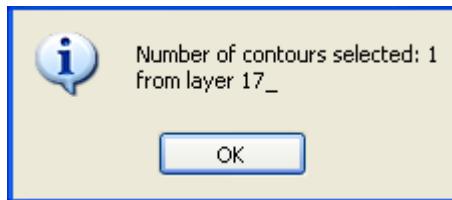
Right click options

'Right clicking' while using the 'Place cut start point' selection tool will bring up the following menu screen.



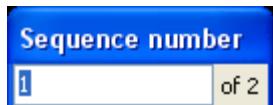
Properties

Clicking this function opens a dialog box that shows how many contours you have selected and which layers they are on.



Click <OK> to close the box.

Quick cut sequence



This is a quick way of manually setting the cut order. The sequence number box shows the current start point number and the total number of start points. If you click on a start point it's number will be changed to the sequence number and the sequence number will be incremented by 1. If the sequence number reaches the total number of start points it will wrap round to 0. The 'Escape' key cancels the command.

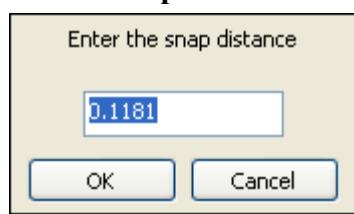
Reverse cut direction

This reverses the cut direction for the current start point only allowing you to switch between climb/conventional cutting for individual start points.

Reverse cut side

Reverse cut side is only used when using 'No offset' and appears as 'greyed out' at all other times. As there is no real inside or outside when using 'No offset' leadins and leadouts can end up on the wrong side of the cut. 'Reverse cut side' will place the leadin/leadout on the other side.

Corner snap distance



If you try placing a start point within the specified distance of a corner (an intersection between a line/arc and another line/arc) then the start point will snap to that corner.

The 'Escape' key cancels the command.

Copy start points from

'Hovering' over or clicking this menu item will open a pop-out dialog box allowing you to copy start points from another operation using the same layer. If no other operations use this layer then the 'Copy start points from' feature will be 'greyed' out.

You need at least two operations that use the same layer for the option to become available. Say you have two outside offset operations, one to do a roughing pass and one to do a finishing pass. You can copy the start points from one pass to the other so that all of the cut starts are identical.

Zoom controls

Clicking on any of these items will zoom the view as required.

Measure

To use the measure tool click on the point you want to measure from.

Note: If you click near a vertex (a join between two lines/arcs) or the centre of a circle the dimension will snap to that point.

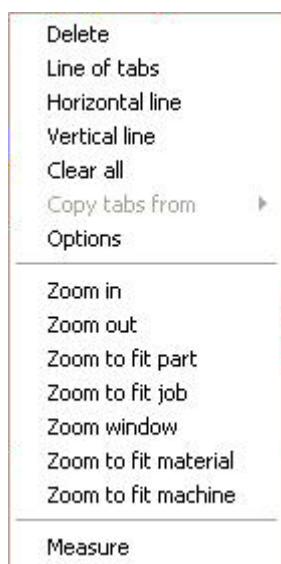
When you move the mouse the coordinate display at the bottom of the screen shows the X,Y length and the number next to the cursor is the distance from the start point to the cursor. If you click on another point the measure start point will move to that location. You can cancel the measure function by switching to another mode, pressing <Escape> or by right clicking and selecting 'End measuring'.

Edit tabs

When clicked allows you to insert a 'holding tab' on the part at a location of your choosing. The mouse cursor changes to a pointer with a 'T' and a white line will appear at the tab location. Move the mouse around to move the tab location. Clicking on the part in the required place will insert the tab.

Note: Select the required operation in the 'Operations' list before placing tabs.

Repeat for each required tab. The size of the tab is defined in the '[Operation/Contour/Cut path](#)' dialog box.



Right click options

'Right clicking' while the mouse pointer is over a placed tab will bring up the following menu screen.

Delete

Clicking on this function deletes the selected tab.

Note: You must have at least one tab before this menu item appears in the list.

Line of tabs

Clicking on this function adds a line of tabs at the angle set in the 'Options' dialog box.

Horizontal line

Clicking on this function adds a horizontal line of tabs.

Vertical line

Clicking on this function adds a vertical line of tabs.

Clear all

Clicking on this menu item will clear all the placed tabs.

Copy tabs from

'Hovering' over or clicking this menu item will open a pop-out dialog box allowing you to copy tabs from another operation using the same layer. If no other operations use this layer then the 'Copy tabs from' feature will be 'greyed' out.

You need at least two operations that use the same layer for the option to become available. Say you have two outside offset operations, one to do a roughing pass and one to do a finishing pass. You can copy the tabs from one pass to the other so that all of the tabs are identical.

Options

Clicking on this function opens the 'Options' dialog box. See '[Jet cutting\Cut path tab\Holding tabs](#)' for more information.

Zoom controls

Clicking on any of these items will zoom the view as required.

Measure

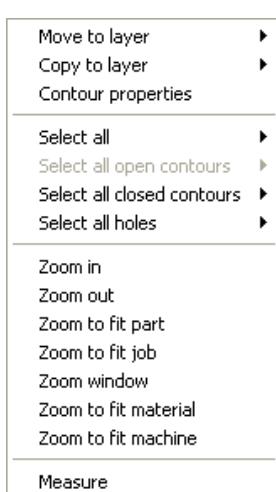
To use the measure tool click on the point you want to measure from.

Edit contours

Use this function to move or copy contours to different layers. Click on a contour to select it. Hold the <Control> key to select multiple contours.

Note: Hold down the <Shift> key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the  button in the toolbar.



Right click options

'Right clicking' while using the 'Edit Contours' selection tool will bring up the following menu screen.

Move to layer

'Hovering' over or clicking this menu item will open the following pop-out dialog box.



Choose either a 'New layer' or an existing layer. This will move the selected contour(s) to the chosen layer.

Choosing 'New layer' will open the following window.



Enter a new name for the layer and click <OK>.

Copy to layer

'Hovering' over or clicking this menu item will open the following pop-out dialog box.



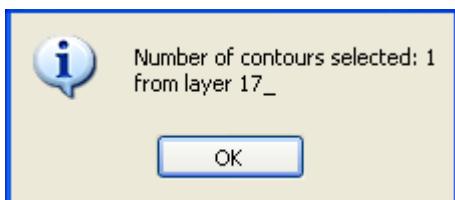
Choose either a 'New layer' or an existing layer. This will copy the selected contour(s) to the chosen layer (but leave the original contour(s) intact on the original layer).

Choosing 'New layer' will open the following window.



Enter a new name for the layer and click <OK>.

Contour Properties



Clicking this function opens a dialog box that shows how many contours you have selected and which layers they are on.

Click <OK> to close the box.

Select options

'Hovering' over or clicking on any items in this menu area will open the following pop-out dialog box.



Select all

This will select all contours from the selected layer(s).

Select all open contours

This will select all open contours from the selected layer(s).

Note: If there are no open contours this option will be 'greyed' out.

Select all closed contours

This will select all closed contours from the selected layer(s).

Note: If there are no closed contours this option will be 'greyed' out.

Select all holes

This will select all open holes from the selected layer(s).

Note: If there are no holes contours this option will be 'greyed' out.

Zoom controls

Clicking on any of these items will zoom the view as required.

Measure

To use the measure tool click on the point you want to measure from.

Nesting

This feature allows you to arrange multiple parts on the same workpiece.

Click on the part to select it then drag it to the new position. You can also copy and duplicate the part with this function.

TIP: 'Copy' and 'Duplicate' have different functions as follows:

Copy: This will make a complete copy of the drawing and operations. The copy will be **completely independent** of the part it was copied from.

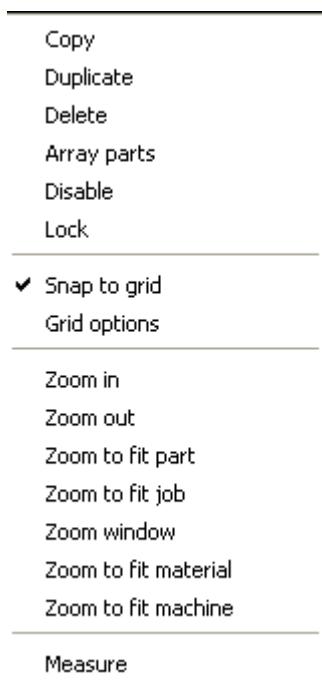
Duplicate: This will perform a 'step and repeat' of the original drawing and operations. **Any changes** made to the original **will be reflected in the duplicate**.

Note: Hold down the <Shift> key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the  button in the toolbar.

Right click options

'Right clicking' while using the 'Nesting' tool will bring up the following menu screen.



Copy

Click on this function to copy the part. A part consists of a drawing with its associated operations. Copy will make a complete copy of the drawing and operations. The copy will be **completely independent** of the part it was copied from.

Duplicate

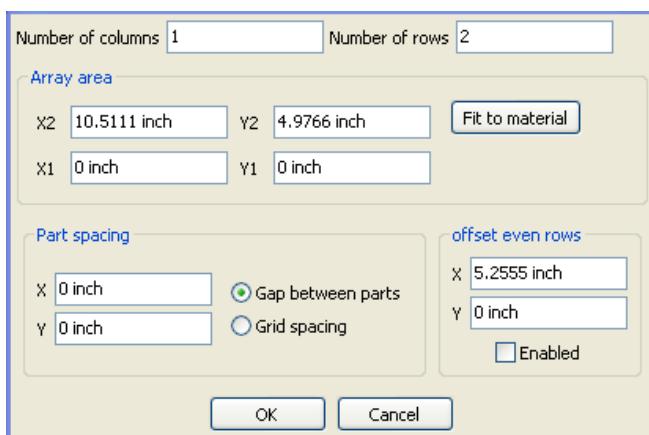
Click on this function to duplicate the part. Duplicate will perform a 'step and repeat' of the original drawing and operations. **Any changes made to the original will be reflected in the duplicate.**

Delete

Clicking on this function deletes the selected item.

Array parts

Clicking on 'Array' will open the following dialog box.



This creates an array of duplicates (step and repeat). Select the part(s) you want to array using 'Ctrl-click' or drag a box to select multiple parts then 'right-click' and select 'Array'.

Number of columns/rows

Enter the number of columns and rows required. A matrix of

parts will be generated with the number of columns and rows specified.

Array area

Use the boxes in this section to specify the size of the array area. X1 and Y1 are the bottom left corner and X2 and Y2 are the upper right corner. The boxes can be used in two ways:

1. If you enter a value in the columns/rows boxes above or change the part spacing or offset rows then the boxes show the area the array will cover.
2. If you directly enter a value the number of columns/rows will be updated to fit as many parts as possible into the specified area.

Fit to material

The 'Fit to material' button sets the area to the size of the material and then updates the number of columns/rows to suit.

Part spacing

If '**Gap between parts**' is selected then X and Y represent the space between the duplicates.

Note: To calculate the spacing the parts are assumed to be rectangular, with the rectangle being the smallest that will completely enclose the part. For some oddly shaped parts you may well end up using a negative X or Y to minimize wasted material.

If '**Grid**' is selected then X and Y represent the horizontal and vertical increment. For instance if X is 2" then the duplicates will be placed every 2" in the X axis.

Offset even rows

If you are laying out irregularly shaped or circular parts then simply placing them on a regular grid would be inefficient. Instead, you can offset alternate rows to pack them in more tightly. If an array does not come out the way you want it to you can always undo (Edit->Undo or Ctrl-Z) the operation.

Enabled

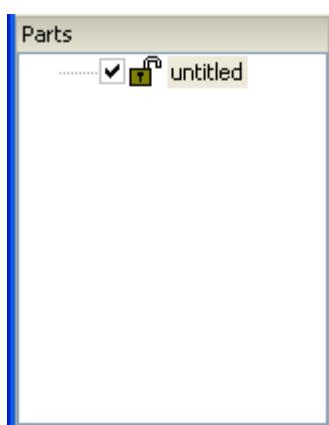
Clicking the 'Enabled' check box enables the 'Offset even rows' function. Dimensions can be left in the boxes and used as required by simply toggling the 'Enabled' check box on or off.

Disable/Enable

A disabled part will not be cut. Using the right-click menu is the same as using the check box next to the part name.

Lock/Unlock

When in nesting mode the part tree shows a padlock symbol next to the part name (see below).

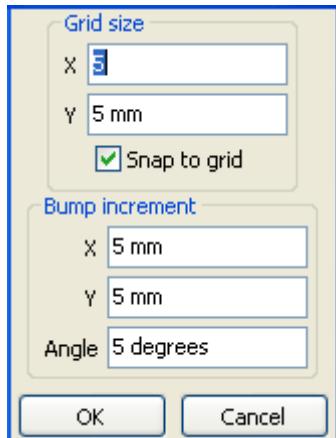


This symbol indicates if the part is locked or not. Locked parts cannot be moved while in nesting mode. Lock/Unlock appears in the right-click menu in the part tree and the main graphics windows in nesting mode.

Snap to grid

Clicking on this item toggles the 'Snap to grid' function on or off.

Grid options



Grid size

If you set the grid size to be greater than zero (0) when you move the part it will always 'snap' to the nearest multiple of the grid size.

Snap to grid - check the box to enable 'snapping'.

Bump increment

When moving a part you can 'bump' it a fixed amount using the arrow keys. 'Bump increment' sets the amount each 'bump' moves the part.

Zoom controls

Clicking on any of these items will zoom the view as required.

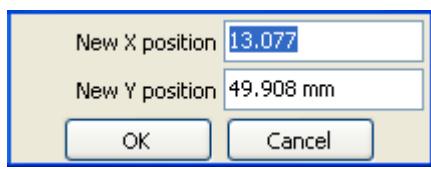
Measure

To use the measure tool click on the point you want to measure from.

Align

Click on this function to align the drawing in SheetCam to the work on your machine. For example, imagine you have a component or piece of material (here after called 'the job') that you have clamped to the table. Instead of carefully positioning 'the job' to line up with the drawing/machine axes you can clamp 'the job' roughly in place then line the drawing up to match it. This is also very useful when drilling circuit boards.

Pick two points on 'the job' that are spaced as far apart as possible and measure their coordinates by jogging the machine and using an edge finder or lining up a pointed cutter by eye. Click on <Align> and then click on the first point on the part in the view window and the following dialog box will appear.



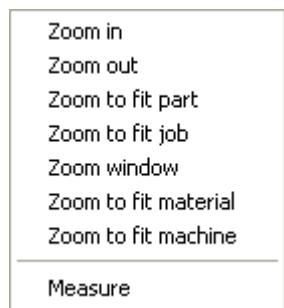
Enter the actual coordinates of this point and click <OK>. Now click on the second point on the part in the view window and enter the coordinates of that point and click <OK>. The drawing will be moved and rotated to suit 'the job'.

Note: Hold down the <Shift> key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the  button in the toolbar.

Right click options

'Right clicking' while using the 'Align' selection tool will bring up the following menu screen.



Zoom controls

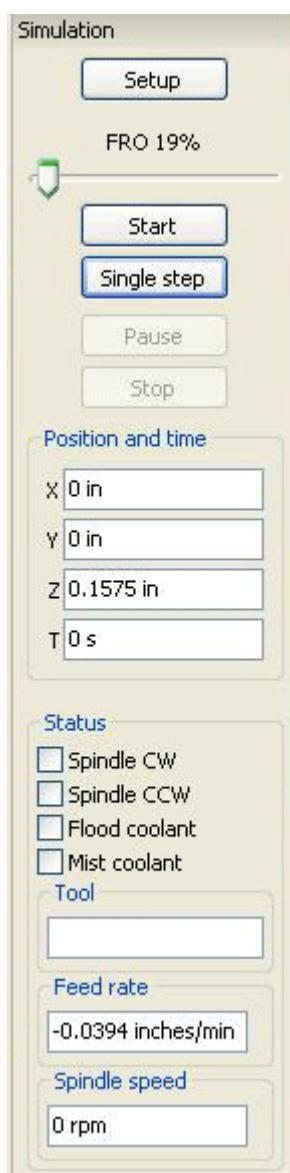
Clicking on any of these items will zoom the view as required.

Measure

To use the measure tool click on the point you want to measure from.

Run simulation

Clicking on this item will run the 'Simulation plugin'.

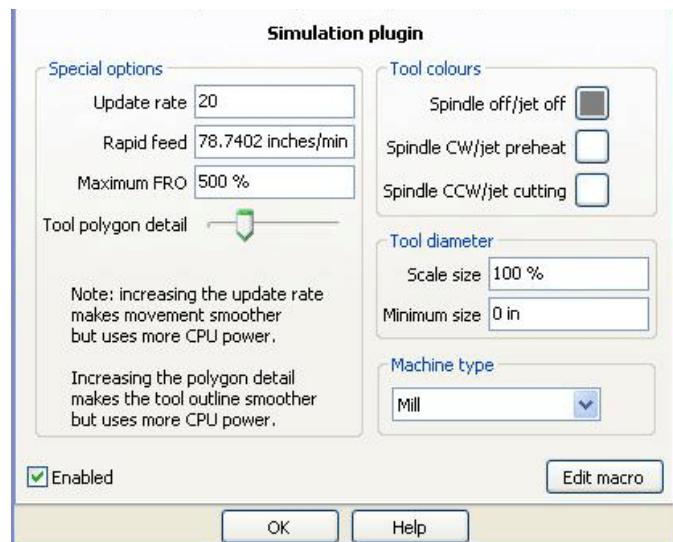


Note: Hold down the <Shift> key to scroll the screen in 3D with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.

Setup

Clicking on the 'Setup' button will open the following dialog box:



Special options

Update rate

Enter the required update rate in the box.

Note: Increasing the update rate makes movement smoother but uses more CPU power.

Rapid feed

Enter the required rapid feed rate in the box.

Maximum FRO

Enter the maximum feed rate override (FRO) amount in the box as a percentage.

Tool polygon detail

Use the slider to control the tool polygon detail.

Note: Increasing the polygon detail makes the tool outline smoother but uses more CPU power.

Tool colours

Click on a box next to an item to open the colour picker window then select a new colour.

Tool diameter**Scale size**

Enter the scale size as a percentage.

Minimum size

Enter the minimum size as a dimension.

Machine type

Select the machine type using the drop down menu.

Enabled

This check box toggles the plugin on or off.

Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

FRO Slider

Move the slider to control the amount of feed rate override (FRO). The further to the right you move the slider the faster the simulation will run.

Start

Clicking the 'Start' button will start the simulation.

Single step

Clicking the 'Single step' button will step through the simulation one move at a time.

Pause

Clicking the 'Pause' button will pause the simulation.

Stop

Clicking the 'Stop' button will stop the simulation.

Position and time

The four boxes show the current location of the tool in the X, Y and Z planes and the time taken to get there.

Status

A check mark in a box indicates that specific item is in use at that time.

Tool

This box displays the current tool in use.

Feed rate

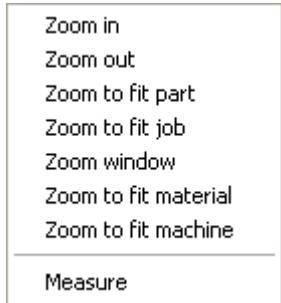
This box displays the current feed rate.

Spindle speed

This box displays the current spindle speed.

Right click options

'Right clicking' anywhere in the viewport window while running the simulator will bring up the following menu screen.

**Zoom controls**

Clicking on any of these items will zoom the view as required.

Measure

To use the measure tool click on the point you want to measure from.

Text



This places text directly in your part. It uses AutoCad .shx fonts.

Note: Hold down the <Shift> key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.

Layer

The layer the text will be placed on.

Font

Select the required font from the drop down menu or use the button (the one with three dots) to search for a font folder.

Size/Position

Height - The height of a capital letter. This does not include descenders on lower case fonts.

Horizontal spacing - Horizontal distance between characters. 100% is the width of one character. 50% is half the width.

Vertical spacing - Vertical distance between lines. 100% is the height of one character. 50% is half the height.

Rotation - The amount to rotate the text in degrees.

Alignment

Select the alignment style using the buttons.

Text Window

Enter the required text in this area.

Note: The text can be 'dragged' with the mouse to position it on the drawing.

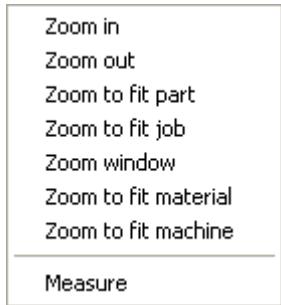
Add to current part

Click this button to generate the text and add it to the part.

Note: You can undo this but you cannot delete or edit already placed text.

Right click options

'Right clicking' while using the 'Text' selection tool will bring up the following menu screen.



Zoom controls

Clicking on any of these items will zoom the view as required.

Measure

To use the measure tool click on the point you want to measure from.

Calculator button

Toggles a simple calculator on or off.

Tools toolbar buttons



- Add a new rotary tool
- Add a new jet cutting tool
- Add a new code snippet
- Access tool table

New rotary tool button

Clicking on this button will open the 'New rotary tool' set up window.

Note: Use this button for **ALL** rotary tools (mills, drills, taps etc.)

Note: This can also be accessed via the '[Tools\New rotary tool](#)' menu item.

New jet cutting button

Clicking on this button will open the 'New jet cutting tool' set up window.

Note: Use this button for **ALL** jet cutting tools (plasma, flame, waterjet, laser etc.)

Note: This can also be accessed via the '[Tools\New jet cutting tool](#)' menu item.

Add new code snippet

Clicking on this button will open the 'New code snippet' set up window.

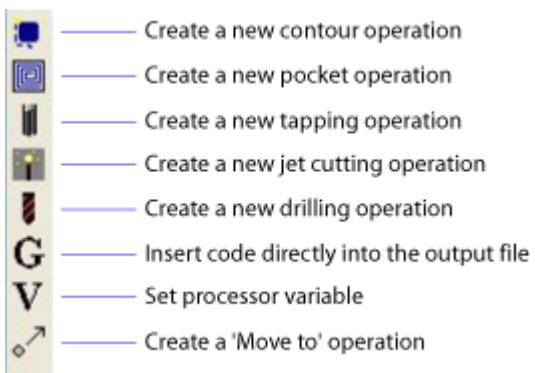
Note: This can also be accessed via the '[Tools\New code snippet](#)' menu item.

Tool table button

Clicking on this button will open the 'Tool table' window.

Note: This can also be accessed via the '[Tools\Tool table](#)' menu item.

Operations toolbar buttons



Add new contour

Clicking on this button will open the 'New contour' set up window.

Note: This can also be accessed via the '[Operation\Contour](#)' menu item.

Add new pocket

Clicking on this button will open the 'New pocket' set up window.

Note: This can also be accessed via the '[Operation\Pocket](#)' menu item.

Create new tapping operation

Clicking on this button will open the 'Tapping operation' set up window.

Note: This can also be accessed via the '[Operation\Tapping](#)' menu item.

Create new jet cutting operation

Clicking on this button will open the 'Plasma cut' set up window.

Note: This can also be accessed via the '[Operation\Plasma cut](#)' menu item.

Create new drilling operation

Clicking on this button will open the 'Drilling operation' set up window.

Note: This can also be accessed via the '[Operation\Drilling](#)' menu item.

Insert G-code

Clicking on this button will open the 'Insert G-code' set up window.

Note: This can also be accessed via the '[Operation\Insert code](#)' menu item.

Set post processor variable

Clicking on this button will open the 'Set post variable' set up window.

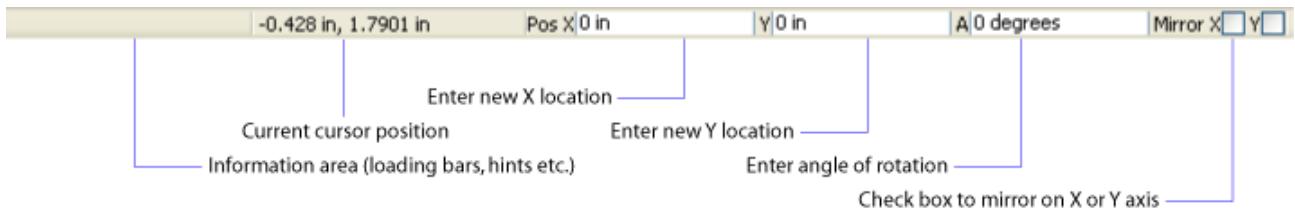
Note: This can also be accessed via the '[Operation\Set post variable](#)' menu item.

Move to operation

Clicking on this button will open the 'Move to' operation set up window.

Note: This can also be accessed via the '[Operation\Move to](#)' menu item.

Lower right toolbar



Information area

Shows file loading bars, hints etc.

Cursor position

Displays the current cursor position.

Enter new X/Y location

Enter the location of the part centre in the X and Y directions to move the part.

Note: Part centre becomes lower left corner if 'Options\Application options\Display\Use bottom left coordinates' is turned on.

Angle

To rotate the part enter the required angle.

Note: To rotate clockwise enter a **positive (+)** value, to rotate counterclockwise enter a **negative (-)** value.

Mirror

Checking the <X> box will mirror the part in the X plane.

Checking the <Y> box will mirror the part in the Y plane

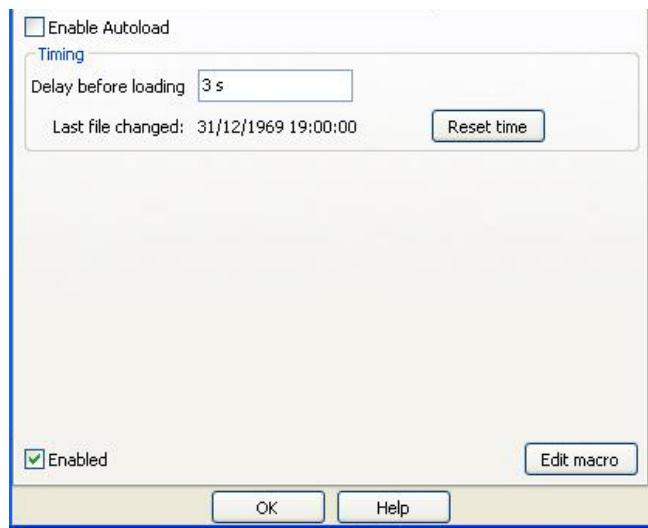
Checking both boxes will mirror the part in **both** planes.

Plugins

SheetCam features several 'plugins' as standard and more can be written by outside developers. The standard ones are as follows and can be accessed via the '[Options\Plugin options](#)' menu item:

Autoload Plugin

This dialog box controls the Autoload plugin.



Enable Autoload

Check/uncheck this box to enable/disable the Autoload plugin.

Timing

Delay before loading

Autoload is triggered by a change in the file. If it takes your CAD package a while to save the file it is possible that autoload may try loading it before your CAD package has finished. To work around this there is a delay before the file is loaded.

Last file changed

This is the time the last file changed. Files with modification dates before this time will be ignored.

Reset time

If you change your clock you may need to click on 'Reset time' to reset this value.

Enabled

This check box toggles the plugin on or off.

Edit macro

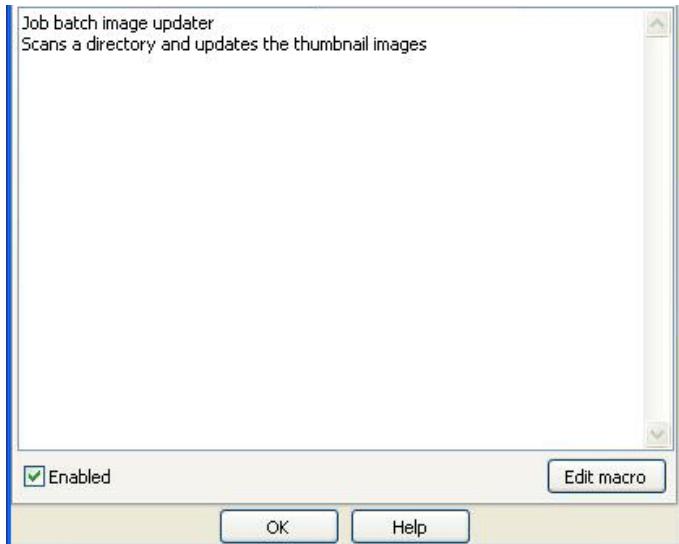
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Batch Update Plugin

This tab controls the Batch update plugin.



Enabled

This check box toggles the plugin on or off.

Edit macro

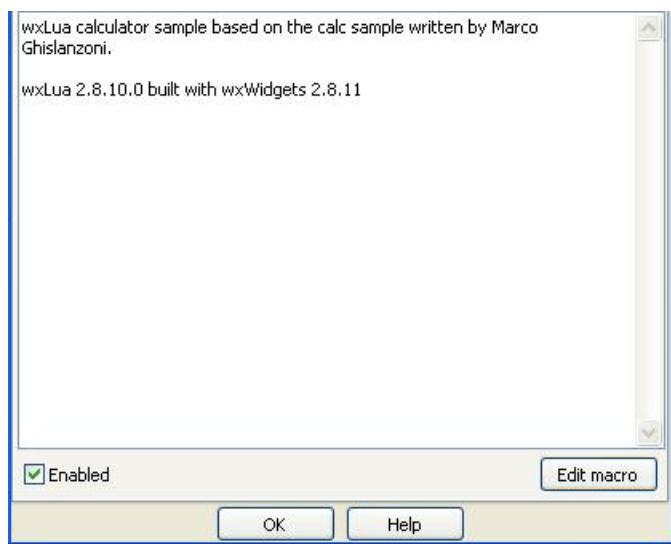
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Calculator Plugin

This tab controls the calculator plugin and adds the calculator toolbar button.



Enabled

This check box toggles the plugin on or off.

Edit macro

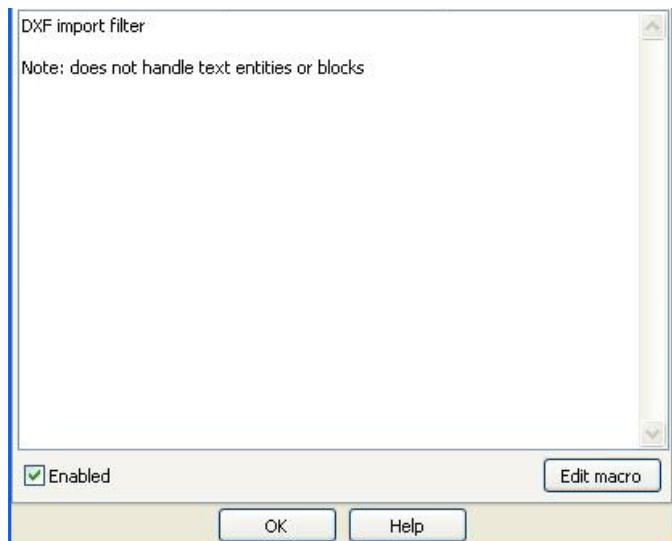
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

DXF plugin

This plugin controls the DXF drawing import features.



Enabled

This check box toggles the plugin on or off.

Edit macro

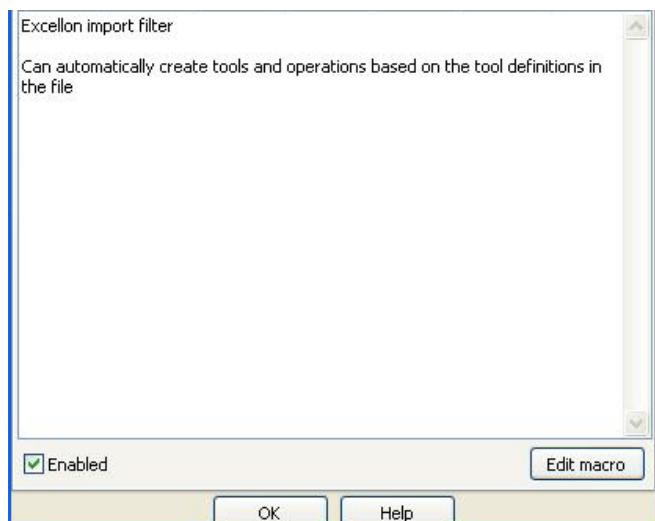
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Excellon plugin

This plugin controls the Excellon drawing import features.



Enabled

This check box toggles the plugin on or off.

Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

G-code plugin

This plugin controls the G-code drawing import features.



Enabled

This check box toggles the plugin on or off.

Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

HPGL plugin

This plugin controls the HPGL drawing import features.



Enabled

This check box toggles the plugin on or off.

Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

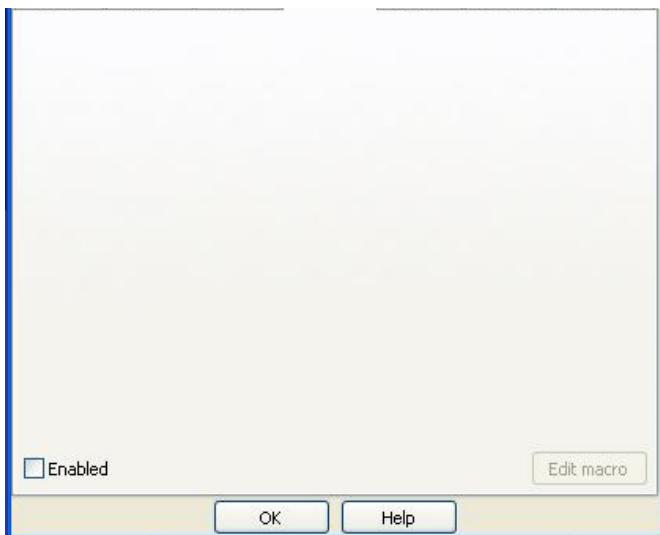
Help

'Help' opens the help file at the relevant location.

Mach plugin

This tab controls the Mach plugin.

If Mach is installed and it is a reasonably recent version you will see the message 'A compatible version of Mach was detected'. You will also see a toolbar button with a big M in it and the menu entry 'Plugins\Enable Mach'. When the button or menu entry is selected and Mach is running, it will be notified when you run the post processor. If Mach is not currently running a program it will close its current file and load the new one.



Enabled

This check box toggles the plugin on or off.

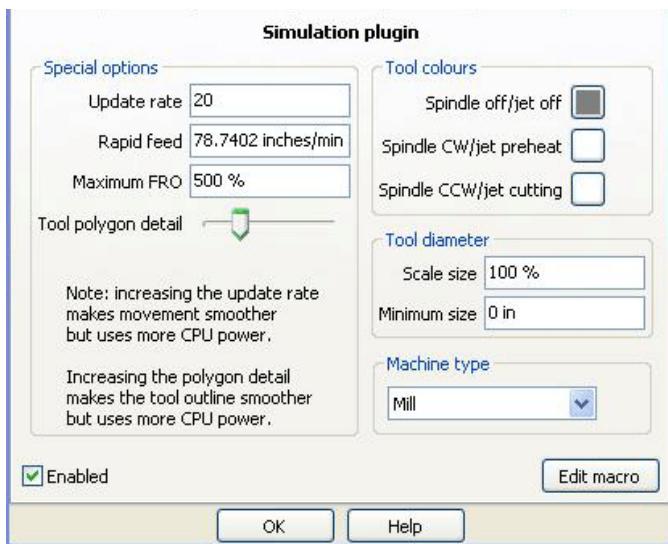
Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Simulation Plugin



Special options

Update rate

Enter the required update rate in the box.

Note: Increasing the update rate makes movement smoother but uses more CPU power.

Rapid feed

Enter the required rapid feed rate in the box.

Maximum FRO

Enter the maximum feed rate override (FRO) amount in the box as a percentage.

Tool polygon detail

Use the slider to control the tool polygon detail.

Note: Increasing the polygon detail makes the tool outline smoother but uses more CPU power.

Tool colours

Click on a box next to an item to open the colour picker window then select a new colour.

Tool diameter

Scale size

Enter the scale size as a percentage.

Minimum size

Enter the minimum size as a dimension.

Machine type

Select the machine type using the drop down menu.

Enabled

This check box toggles the plugin on or off.

Edit macro

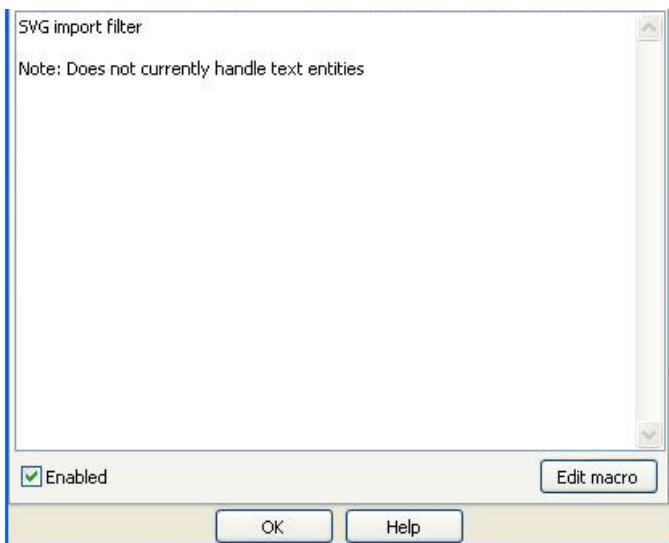
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

SVG plugin

This plugin controls the SVG drawing import features.



Enabled

This check box toggles the plugin on or off.

Edit macro

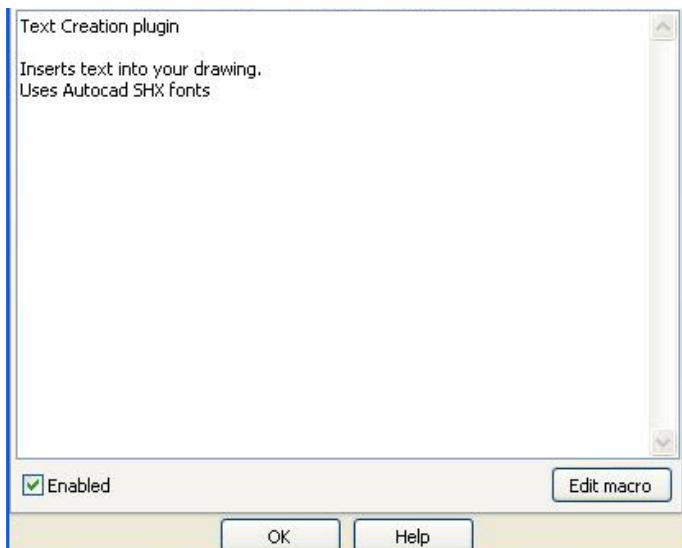
Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Text plugin

This tab controls the text plugin.



Enabled

This check box toggles the plugin on or off.

Edit macro

Clicking this button will open an editing window to allow editing of the plugin.

Help

'Help' opens the help file at the relevant location.

Hints and tips

Centre drilling

If you need to centre drill a series of holes prior to drilling open a new drill operation then set the minimum hole size to zero (0) and the maximum hole size to the size of the largest hole. All the holes on that layer will then be centre drilled.

Context sensitive help

Clicking on an open palette or settings window and pressing the <F1> key will open the specific help file for that item.

Copy and duplicate differences

The 'copy' and 'duplicate' commands have very specific meanings in SheetCam. Each acts upon the drawing/operations in a different way as shown below.

Copy: This will make a complete copy of the drawing and operations. The copy will be **completely independent** of the part it was copied from.

Duplicate: This will perform a 'step and repeat' of the original drawing and operations. **Any changes** made to the original **will be reflected in the duplicate**.

Creating your drawing

To allow SheetCam to understand your drawing you need to follow a few simple rules.

Outlines must be properly closed

A closed shape is any shape where the lines completely enclose an area. A square is a simple example of a closed shape. It has an inside and an outside so SheetCam needs to work out which side you want to cut. A simple line is an open shape as there is no inside or outside therefore SheetCam treats it differently.

So what is meant by 'properly closed'? Let's take a simple example. You draw a rectangle using four lines. However two of the lines don't quite meet. On the screen it looks fine unless you zoom in very close. When you load the drawing into SheetCam it will recognize that two ends don't meet and it will assume that the shape is open so you can't cut it out. All open lines will be shown on the screen in purple. If your lines are very close (as defined under '[Options\Application options\Drawing import](#)') SheetCam will automatically move the ends so that they touch. It is **not** good practise to rely on this feature, **always** try to draw properly closed shapes.

So how do I find problems?

Turn off the show segment ends button  and turn on the show path ends button 

128

drawing and fix the problem.

Note: 'Snaps' are your best weapon against problems like this. Use 'grid' snaps and 'end' snaps whenever possible. They are the best way to create accurate drawings. If you are not familiar with using snaps then look in the documentation for your CAD/drawing program. Virtually all CAD/drawing programs have some form of snap capability.

Shapes must not self intersect

A good example of a self intersecting shape is the figure 8. In the middle of the 8 the lines cross over each other. Imagine a path in the shape of an 8 and you are walking along it. At one point the outside is on your left and another point the outside is on your right. There is no rational way to cut this shape. SheetCam tries but the results will almost undoubtedly not be what you are after.

Explode text in DXF files

DXF files cannot handle 'True Type' fonts very well as there is no 'standardized' format to follow. To guarantee that the cut path is exactly the same as your drawing you need to convert any text into lines. This is often called 'exploding'. If you are not familiar with 'exploding' text look in the documentation for your CAD/drawing program to find out more. If your CAD/drawing package supports HPGL export then try exporting your drawing as HPGL. Most CAD/drawing packages will automatically convert text to lines when exporting HPGL.

Note: Some drawing programs have an option to 'convert text to lines' rather than having an 'explode' function but the end result is the same, text blocks are converted to lines that SheetCam will recognize.

Use layers

If you want to perform more than one operation on your part then you will need to separate your drawing into layers. Each time you run an operation in SheetCam the operation is applied to everything on that layer. You can create your drawing using 'layers' in CAD programs and in 'most' drawing programs (check your CAD/drawing program documentation for more information).

Note: If your drawing **is** all on one layer you can use SheetCam to split different operations off onto their own layers. Use the '[Edit contour](#)' window to move parts to a different layer by first selecting the 'Edit contour' button . Then click on a drawing section to select it and right-click to call up the 'Edit contour right click options' window. Select either a new layer (and provide a unique name) or an existing layer to move the selection to.

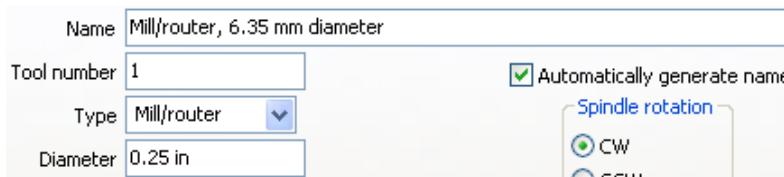
Depth of cut, peck depth etc.

Any '**depths**' are **always** entered as a **positive** (+) figure as you are specifying the '**depth**' of cut. Some users expect this to be a minus (-) figure.

Entering values

Whenever you enter a value into an edit box you can use any units you like. The units you use will be converted into your '[Preferred units](#)' as soon as you leave the edit box.

For example if your default units are millimetres but you are using a 0.25 inch cutter then just type in **0.25 in**. The units are not case sensitive so 0.25IN 0.25 inch or 0.25 Inch would all work. If you type in a value without any units SheetCam assumes you are using your preferred units.



As soon as you leave the edit box it will be converted to 6.35mm



The available linear units are:

Metric: um, mm, cm, m

Inch: mil, thou, in, inch, ", feet, foot, '

The available angular units are:

degrees, deg, radians, rad

The available feed rate units are

Metric: mm/min, m/min, mm/sec, m/sec

Inch: IPM, inch/min, IPS, inch/sec

Finish allowance

The finish allowance option forces SheetCam to leave the part under or over sized. A positive finish allowance will leave some material behind for a finishing pass using a new operation. This allows you to use a different cutter and/or Z increment for the finishing pass. A negative finish allowance will remove more material. This is useful if you have a close tolerance hole and you need to make it slightly bigger. Note that the biggest negative finish allowance you can use is slightly less than the cutter radius.

Note: Leaving the allowance at zero (0) will machine the contour/pocket to full size with no finishing pass.

Machine and material parameters

SheetCam can show the machine table and material outlines for reference purposes. It also uses the material thickness and height above the table to work out if the cutter is likely to crash into the table.

Machine parameters

The settings for machine parameters can be accessed via the '[Options/Machine](#)' menu item.

X and Y coordinates of table origin

These are the absolute coordinates of the bottom left hand corner of the table.

Table width

This is the width of the table (X axis).

Table depth

This is the depth of the table (Y axis).

Max clearance between the chuck and the table

This is the maximum available clearance on the Z axis (i.e. the clearance between the chuck and table with the Z axis raised to the maximum height). It is used to work out if it is possible to fit the work on the machine.

Note: As with any input boxes you can enter values in any measurement system. See '[Entering values](#)' on the previous page for more information.

Material parameters

The settings for material parameters can be accessed via the '[Options\Job options\Material](#)' menu item.

Thickness of work

This is the total thickness of the material being machined.

X and Y coordinates of bottom left corner

These are the absolute coordinates of the bottom left hand corner.

X and Y size

The overall size of the work (X and Y axes).

Rapid clearance

The height above the work to run the tool when moving between cuts.

Height of bottom of material above table

Enter the distance the bottom of the material sits above the table.

Note: This is a **positive** (+) figure as you are specifying the '**height**'. This is used where the part requires a 'sacrificial board' under it in order to machine it (i.e. a 'through' pocket or cut-out in a sheet of material). SheetCam **will** let you cut into the board but it **will not** let you cut into the table.

Note: As with any input boxes you can enter values in any measurement system. See '[Entering values](#)' for more information.

Mouse wheel

If you have a 'scroll mouse' the 'mouse wheel' can also be used to zoom in and out. If the zoom directions are opposite to the way you like to work you can change the direction of zoom using the [Options\Application options\Reverse mouse wheel](#) option.

Zooming and panning

Zooming in and out

There are three methods of zooming in and out:

- The mouse wheel zooms the screen at any time.
- Use the zoom buttons  on the toolbar.
- Use the <Page Up> and <Page Down> keys on the keyboard.

Panning

There are two methods of panning the screen:

- Use the keyboard arrow keys to pan the screen at any time.
- If you are in 'drag' mode (i.e. the  toolbar button is selected) holding down the left mouse button and moving the mouse will drag the drawing.

Rotating the screen in 3D

You have to be in 'drag' mode (i.e. The  toolbar button is selected) to rotate the screen. Hold down the <Shift> key and move the mouse to rotate the drawing.

Drag mode

If the pointer button  is selected then you will automatically be in drag mode as soon as you press a mouse button. Holding down the <Shift> key will enable drag mode while any other selection tool is active. The mouse pointer will change into a hand to show you are in drag mode. On releasing the <Shift> key the previous selection tool will be reactivated.

Tutorials

A number of tutorials covering different aspects of the software can be found under the 'Help\Tutorials' menu item. These are reproduced below so that users can print them out if required.

Machine setup

The machine setup is covered on pages 6-9 of this manual.

Material setup

The material setup is covered on page 10 of this manual.

Profile tutorial

In this tutorial we will open a drawing and create the toolpaths to cut out the part.

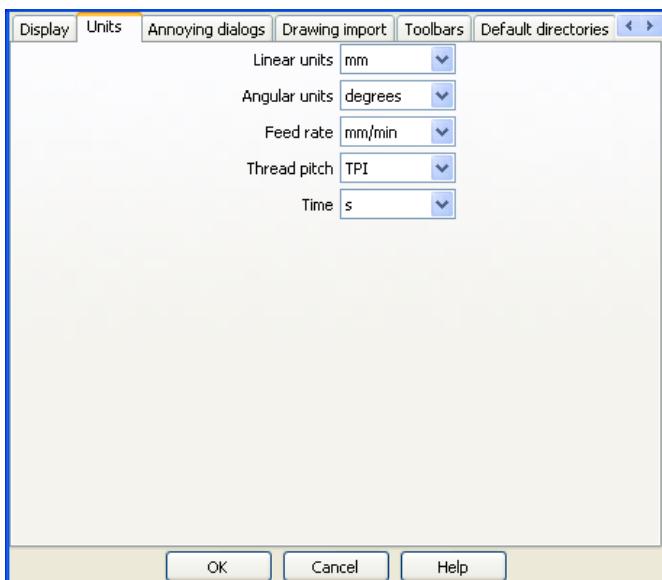
The second group of icons on the left hand side of the upper toolbar control how the drawing is displayed. Experiment with the first three to see what they do.

 Show the input paths. This shows or hides your drawing

 Show segment ends. This highlights the start and end points of each line or arc segment in the drawing. The curves in this drawing consist of lots of small line segments so they show up as almost completely white.

 Show path ends. This highlights the start and end points of all open paths. In this case the line underneath the text is an open path because it has a start and an end point. The text consists of closed paths which have no fixed start and end points.

For the purposes of this tutorial we are working in millimetres so go to the '**Options\Application options**' menu item and select the '**Units**' tab once the pop-up window opens (see image below).



Make sure '**Linear units**' are set to mm and '**Feed rate**' is set to '**mm/min**' then click on '**OK**'. If you prefer inches then you can change the units when you have finished the tutorial.

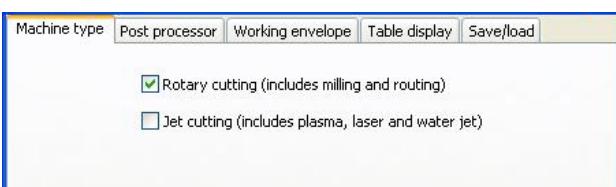
Setting up the machine

Make sure the '**Show machine and work**' button is active (the button is active when it has a grey lined box around it). This displays the machine and the work material.

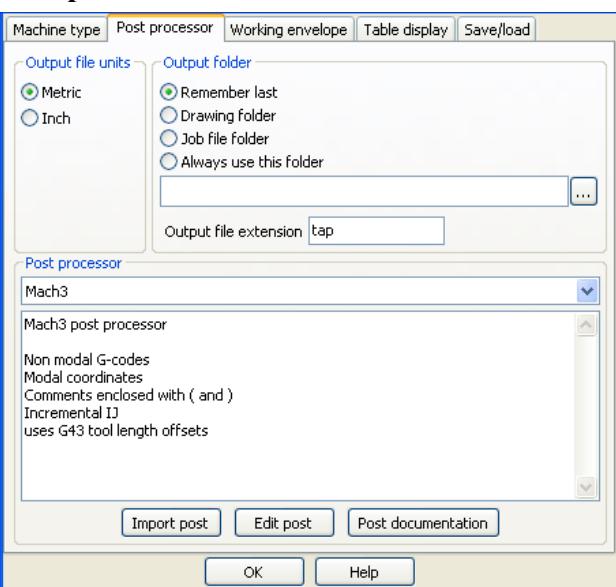
Now click on the '**Options\Machine**' menu item. A pop-up window will appear with several tabs that allow control of the various functions.

Machine type tab

For this tutorial select the '**Rotary cutting**' check box only.



Post processor tab



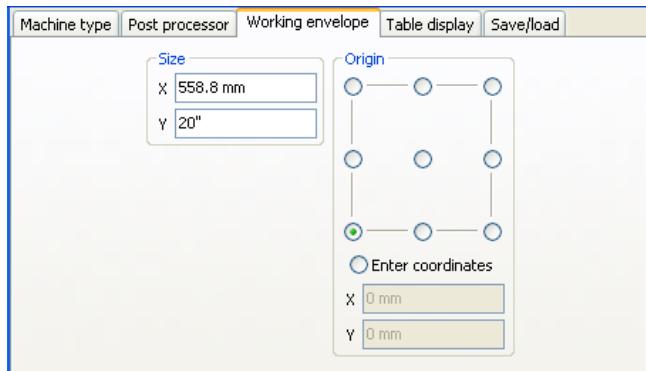
Select the '**Metric**' radio button in the '**Output file units**' section and '**Mach 3**' in the '**Post processor**' drop down menu as shown.

Note: The units selected in the '**Output file units**' section only affect the generated code. They do not alter the units used in SheetCam.

Note: While G-code is supposedly a standard, most machines speak different 'dialects' of G-code. To get around this problem SheetCam uses 'post processors'. These are little programs that run inside SheetCam and create code customized for particular machines. You should select the correct 'post processor' for your machine using the drop down menu when making your own files but for this tutorial select 'Mach 3'.

Working envelope tab

Let's assume our machine table is 2 feet square with a usable area of 22" by 20". The bottom left hand corner of the usable area will be at coordinate 0,0.



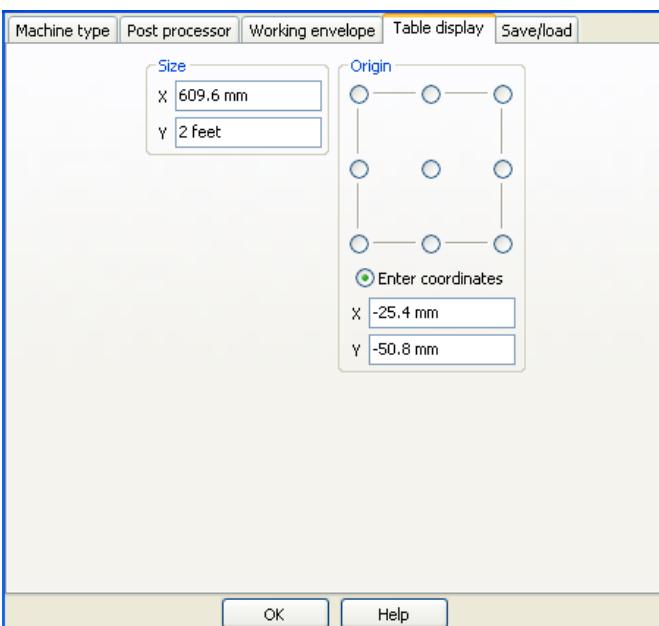
In the '**Origin**' box, click on the bottom left hand corner marker. This tells SheetCam that the origin is the bottom left hand corner.

The '**Working Envelope**' is the usable area so enter <22"> in the '**X**' size and <20"> in the '**Y**' size (include the " to tell SheetCam these measurements are in inches) as shown.

Note: As soon as you exit each edit box the value is converted to millimetres for you. Notice too that the main drawing display changes as you change the dimensions so that you get an idea what the table looks like.

Table display tab

The table is larger than the actual usable area so we need to enter the size and position of the table. Let's assume the 'Working Envelope' origin (0,0) is 1" x 2" away from the corner of the table. Click on the '**Enter coordinates**' radio button in the '**Origin**' section and enter <-1"> and <-2"> for the '**X**' and '**Y**' coordinates of the table bottom left corner and enter <2 feet> for the '**X**' and '**Y**' dimensions in the '**Size**' section as shown.



Note: As soon as you exit each edit box the value is converted to millimetres for you. Notice too that the main drawing display changes as you change the dimensions so that you get an idea what the table looks like.

When you have made all the changes outlined above click on the **<OK>** button at the bottom of the dialog box to save your changes.

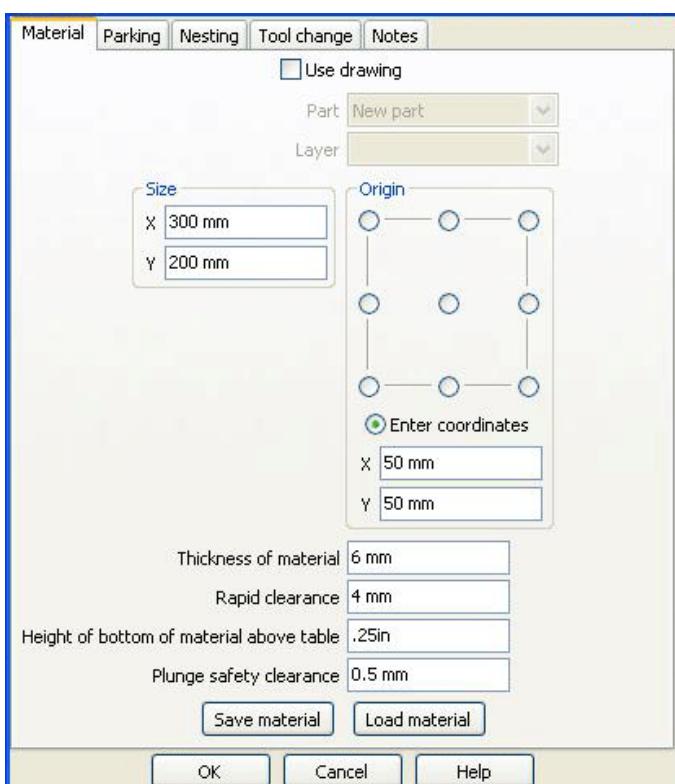
Setting up the material

Now click on the '**Options\Job options**' menu item. A pop-up window will appear with several tabs that allow control of the various functions. The display will zoom and rotate to fit the last specified material.

Material tab

We will be cutting this sign out of a 300mm x 200mm sheet of 6mm plastic that is clamped to the table with it's bottom left hand corner at X=50mm, Y=50mm. As we intend to cut the parts out completely the sheet is clamped on to a piece of 0.25in board to allow the cutter to cut right through without damaging the table.

Enter the dimensions and position of the material as shown below.



most cases.

Older versions of SheetCam have always had this hard coded to 0.5mm. Enter your required distance in the box.

Now click on <OK> to save your changes.

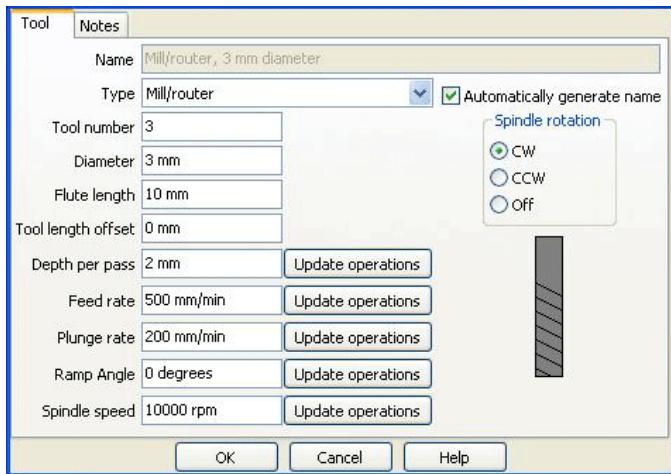
Defining a tool

Now we have set up the machine we need to define the tool that we will cut the part out with. For this job we will use a 30mm long x 3mm diameter spiral router with 10mm flute length. The spindle speed will be 10,000 RPM and the feed rate will be 500mm/min. The maximum safe cut depth per pass in this material is 2mm. On the machine it is tool number 3.

The main display will re-size to show the material as you enter the values. To be safe we will run the cutter 4mm above the panel when moving between cuts so enter '4' in the '**Rapid clearance**' box. The sacrificial board is 0.25in thick so enter '**0.25in**' for the height of the work above the table.

Plunge safety clearance is used when doing a rapid move to cut height. When SheetCam moves to the top of the cut it stops slightly short and moves the remaining distance at feed rate. This allows for inaccuracies in your setup and any chips that may be in the way. The last thing you want is to slam the tool into the work at full speed. By default the clearance is 0.5mm which seems to work pretty well in

Click on the '**Create a new rotary tool**' icon located on the left side tool bar to bring up the tool dialog box shown below:



located on the left side tool bar to bring up

Select '**Mill/Router**' as the tool type and enter <3> in the tool number box. Now enter the '**Diameter**' and '**Flute length**' information. The '**Tool length offset**' box is useful if you are using multiple tool holders or are using an autolock type chuck. You can enter the tool projection here so that SheetCam can automatically compensate for different length tools. This will either be done by offsetting the Z position or specifying the tool

length if your machine understands tool length offsets.

As we are only using one tool just enter <0> here.

In the '**Depth per pass**' box enter <2> as this is the maximum cut depth per pass and enter <500> in the '**Feed rate**' box. We will plunge the tool in at a slower rate so enter <200> in the '**Plunge rate**' box. Leave the '**Ramp angle**' at 0 degrees. Enter <10000> in the '**Spindle speed**' box.

Double check all your settings are the same as shown above and then click on <OK> to save your changes.

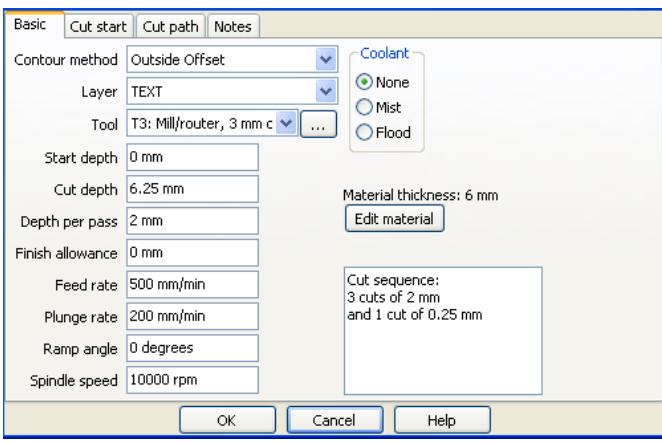
Define a cut operation

Now we have defined the tool we need to use it to cut the letters out.

Press the '**Zoom to fit material**' button  on the upper tool bar to show the drawing and material. Now press the '**Create a new contour operation**' button  to create a new contour operation.

We want to cut the letters out of the sheet so select '**Outside offset**' in the '**Contour method**' box. Select the '**TEXT**' layer in the '**Layer**' drop down list. Select tool '**T3**' in the '**Tool**' list. Doing this sets most of the parameters from the tool definition we did earlier.

We want to cut right through the material and slightly into the sacrificial board below so enter <6.25> in the '**Cut depth**' box.

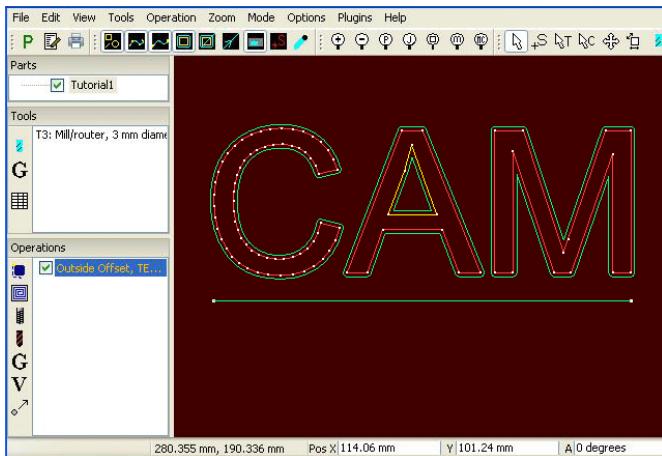


The box at the bottom right shows how many passes the cutter will make to cut through the material. It is currently telling you that the cutter will make 3 passes of 2mm and one of 0.25mm. This is not ideal for this job so change the '**Depth per pass**' value to <1.8>. The cutter will now make 3 passes of 1.8mm, leaving 0.6mm of material left for the last cut (plus the 0.25mm into the sacrificial board of course).

Now click on <OK> to save the changes.

Generate the tool paths

The tool paths will now be created. While the paths are being calculated an animated icon  will be displayed. As this is a very simple job the icon will disappear very quickly.



SheetCam has worked out that the inside of the 'A' has to be cut on the opposite side. It will also be cut before the outside. The cutter has followed the line exactly because SheetCam does not know what to do with it. Note that the '**Operation**' has turned yellow to warn you that there is a problem. If you hover the mouse over the operation a message will pop up advising you what the problem is.

Experiment with the '**Show tool paths**' , '**Show rapids**'  and '**Show direction arrows**'  buttons to see what effect they have on the display.

You can 'drag' the display around with the mouse and zoom in/out by using the mouse wheel. You can rotate the display by holding the shift key then 'dragging' in the display area with the mouse (double-click in the display area to return to the top down view).

As we are only interested in cutting out the letters there is no point in cutting the line. To stop SheetCam from cutting it we need to move it to another layer. Click on the '**Edit contours**' button  then click on the line. It will be highlighted in white. While you are in this mode you can scroll the display around by holding the shift key and dragging with the mouse or using the arrow keys. Now right-click and select '**Move to layer\New layer**' and type anything in for the new layer name.

We will deal with layers in more detail in another tutorial.

Generating the G-code

Now we have defined what is to happen we can generate the G-code ready for the machine to run.

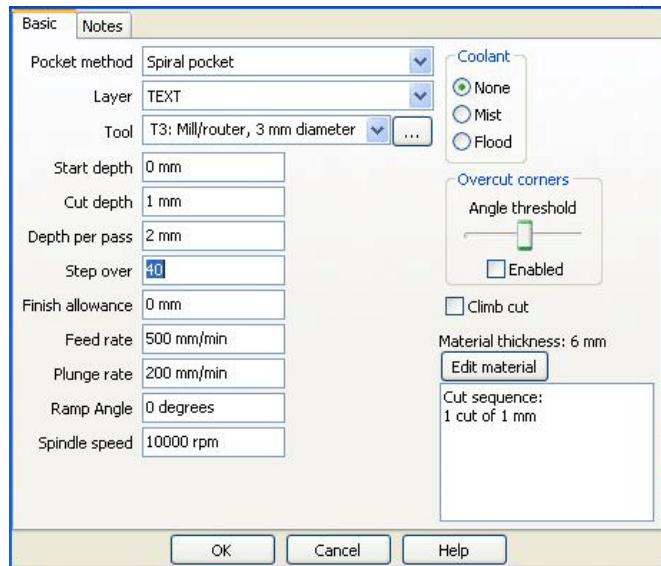
Click on the '**Run post processor**' button  to generate the G-code. If this is the first time you have post processed a drawing then you may be asked to select a post processor to run. Try to choose one that best fits your machine.

You will then be prompted for the file name for the G-code file you are about to create.

You have now successfully created your first G-code with SheetCam.

Pocketing tutorial

In the previous tutorial we cut some letters out of a sheet of plastic. In this tutorial we will make a sign with raised letters and a raised border.



The first step is to engrave around the letters to form a recess 1mm deep.

SheetCam has already loaded the same tool settings as the last tutorial so we can get straight on to cutting plastic.

First click on the '**Create a pocket**' button  to open the pocket editor. In '**Pocket method**' select '**Spiral pocket**'. For the '**Layer**' select '**TEXT**'. In the main window the text and the box around it is highlighted in white to show the parts of the drawing that are going to be used by this operation.

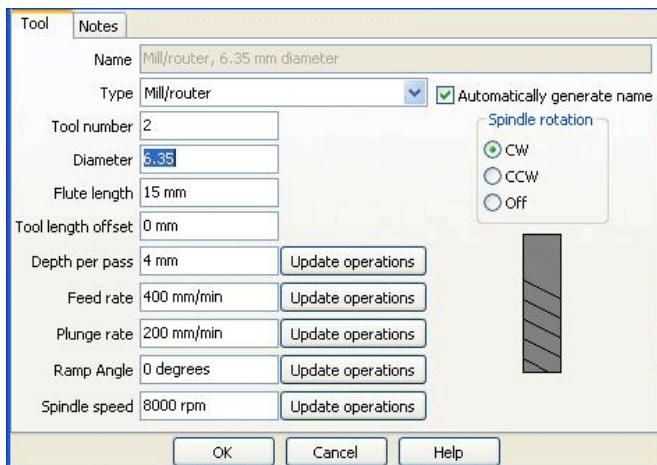
Click on the drop down arrow for the tool and select the only tool in the list. Doing this sets most of the parameters from the tool definition we did in the previous tutorial. Set the '**Cut depth**' to 1mm and the '**Step over**' to 40%. The step over is the percentage of the tool width that is cut at each pass.

Double-check that your settings match the screen shot above and then click <OK>.

Define another tool

Now we need to cut the sign out. Just because we can we will use a different cutter. This time we will use a 0.25in diameter cutter.

Click on the 'Create a new rotary tool' button  and enter the settings as shown below:

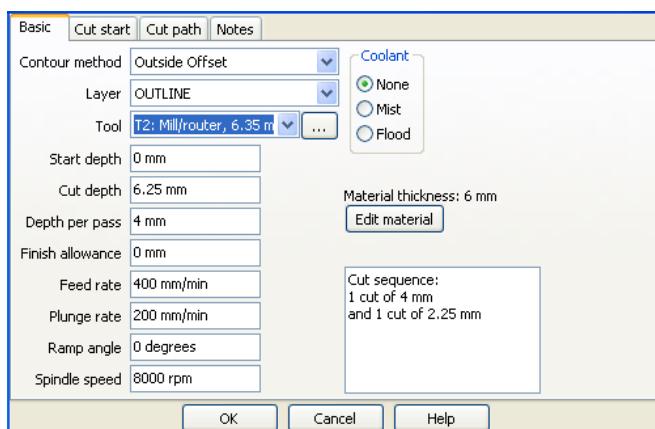


Remember that as soon as you leave the diameter box the diameter will be converted to millimetres. Now click on <OK> to save the settings.

Using layers

Click on the 'Create a new contour operation' button  to open the contour editor. We want to cut around the outside so select 'Outside Offset' from the drop down menu.

For the layer select 'OUTLINE' as this is the profile we want to cut. The layers 'TEXT' and 'OUTLINE' were created in the original drawing. See your CAD package documentation on how to create layers. If your CAD package does not support layers, you can use the layers feature in SheetCam (refer to the 'Help' section) but in many cases it is easier to set up the correct layers in the CAD package before exporting the drawing file.



Select 'Tool 2' (T2) for this job and set the cut depth to '6.25mm' because we want to cut all the way through our 6mm panel. The box in the lower right corner shows you that the part will be cut in 1 pass of 4mm and one pass of 2.25mm.

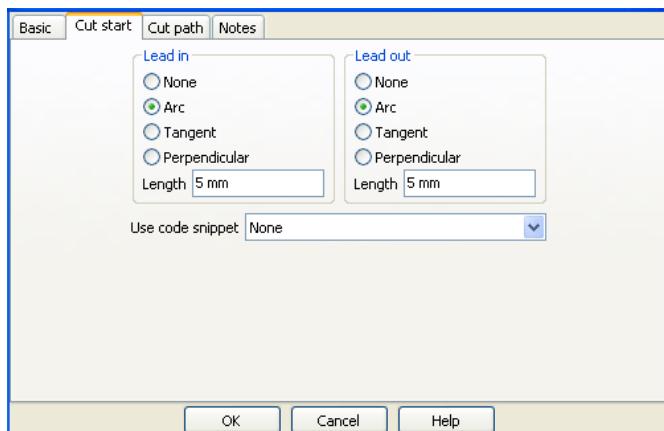
'Start depth' can save time when machining an area that has already been machined in a previous operation. For instance if you are cutting a hole in a pocket. You can set the start depth to the

depth of the pocket then SheetCam will start machining at the bottom of the pocket rather than 'cutting air' to get to depth. For this tutorial leave it set to zero.

Do not close this dialog box yet as we will need it on the next page.

Lead in and lead out

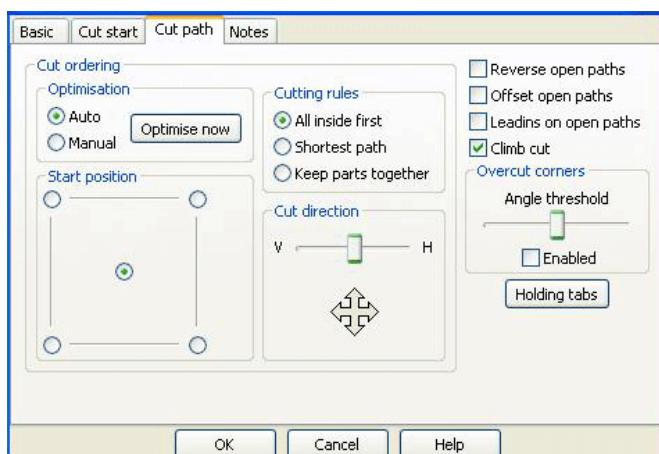
When the cutter plunges into the work it may burn and leave a mark. To avoid this we can tell SheetCam to plunge the tool in close to the cut then move in and start cutting the profile. This is called lead in. Plasma cutters tend to make a messy hole when they start and finish so lead in and lead out can be very handy if you are plasma cutting.



Click on the 'Cut start' tab and set 'Lead in' to 'Arc' and enter 5mm for the 'Length'. The cutter will now plunge in 5mm from the start point then enter the profile smoothly without leaving any marks. You can do the same for the 'Lead out'.

When your settings look like the screen shot above click on <OK>.

This process has been added to the process list on the left hand side.



If you turn on the show path directions button you can see what directions the cuts will be in.

In this case the outline is cut anticlockwise. To change this to clockwise double click on the process we have just created to open the process editor again. In the 'Cut path' tab, make sure 'Climb cut' is checked as shown.

Now click on <OK>. The outside tool path has now reversed.

Click on the 'Run post processor' button to create your G-code.

Jet Cutting tutorial

In this tutorial we will open a drawing and create the toolpaths to cut out the part.

First make sure SheetCam is in jet cutting mode. Go to '**Options\Machine**' and make sure the '**Jet cutting**' check box is checked. You can turn off 'Rotary cutting' to simplify the display.

The second group of icons on the left hand side of the upper toolbar control how the drawing is displayed. Experiment with the first three to see what they do.



Show the input paths. This shows or hides your drawing.



Show segment ends. This highlights the start and end points of each line or arc segment in the drawing. The curves in this drawing consist of lots of small line segments so they show up as almost completely white.



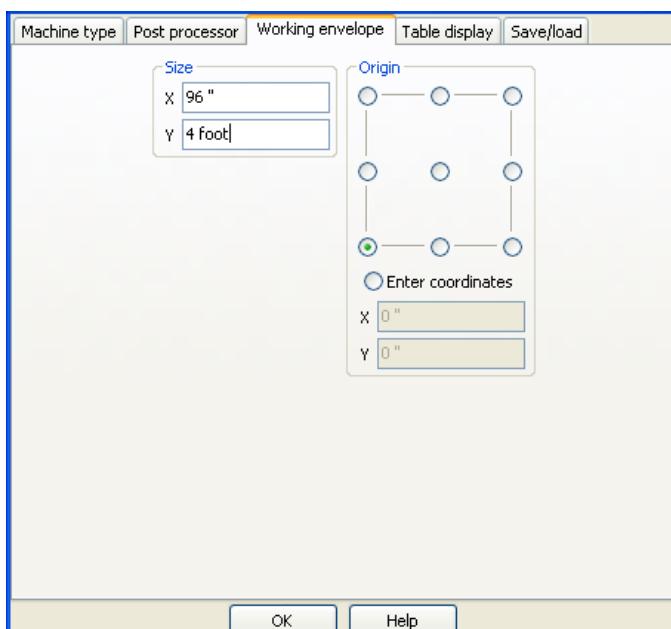
Show path ends. This highlights the start and end points of all open paths. In this case the line underneath the text is an open path because it has a start and an end point. The text consists of closed paths which have no fixed start and end points.

For the purposes of this tutorial we are working in inches so go to '**Options\Application options**' then click on the '**Units**' tab and make sure linear units are set to ' " ' and feed rate is set to '**ipm**' then click on <OK>. If you prefer millimetres then you can change the units when you have finished the tutorial.

Setting up the machine

Make sure the '**Show machine and material**' button is down (i.e. outlined by a grey box). This displays the machine and material.

Now click on '**Options\Machine**' and select the '**Working envelope**' tab.

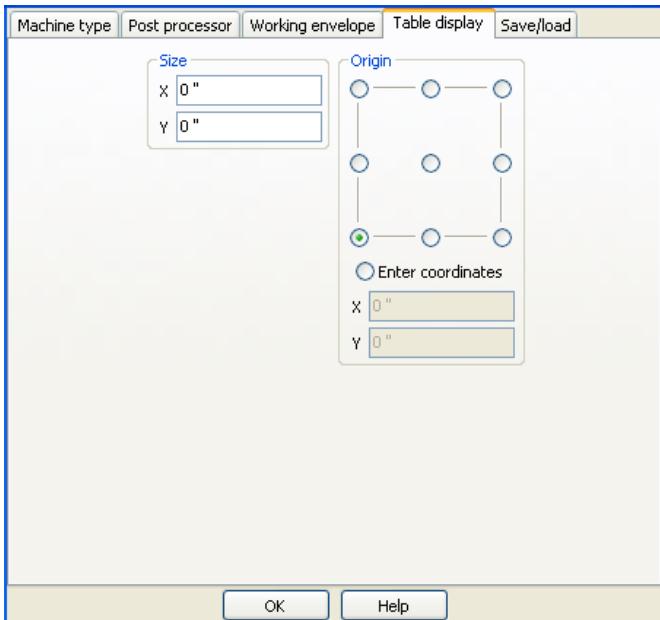


Let's assume our machine table is 8' by 4' and the bottom left hand corner of the useable area is at coordinate 0,0.

In the '**Origin**' box, click on the bottom left hand corner marker. This tells SheetCam that the origin is the bottom left hand corner.

The '**Working Envelope**' is the usable area so enter '**8 foot**' in the '**X**' size and '**4 foot**' in the '**Y**' size (include **feet**, **foot** or " to tell SheetCam these measurements are in feet) as shown.

As soon as you exit each edit box the value is converted to inches for you. Notice that the main drawing display changes as you change the dimensions so you get an idea what the table looks like.



Click on the '**Table display**' tab. You can define the table bottom left and size in this section if you want to show the actual size of the machine as well as the working area. This can be useful if you have a small machine and can fit on larger sheets than the cutting envelope.

For simplicity in this example we will leave these values at zero.

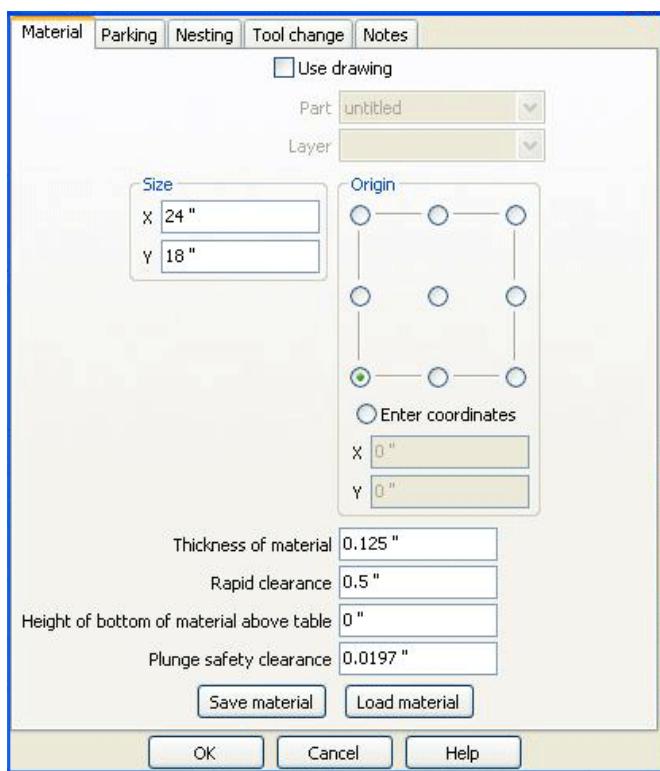
Click on <OK> to save your changes.

Setting up the material

Now click on '**Options\Job options**'. The display will zoom and rotate to fit the material.

We will be cutting this sign out of a 2' x 18" offcut of 1/8" sheet. The sheet will be placed up against the 0,0 datum bar.

Enter the dimensions and position of the material as shown below. The main display will resize to show the material as you enter the values. To be safe we will run the cutter 0.5" above the panel when moving between cuts so enter '**0.5**' in the '**Rapid clearance**' box. Leave the last value as zero because this is not used for plasma cutting.



Plunge safety clearance is used when doing a rapid move to cut height. When SheetCam moves to the top of the cut it stops slightly short and moves the remaining distance at feed rate. This allows for inaccuracies in your setup and any chips that may be in the way. The last thing you want is to slam the tool into the work at full speed. By default the clearance is 0.5mm (0.0197") which seems to work pretty well in most cases. Older versions of SheetCam have always had this hard coded to 0.5mm. Enter your required distance in the box.

Now click on <OK>.

Defining a tool

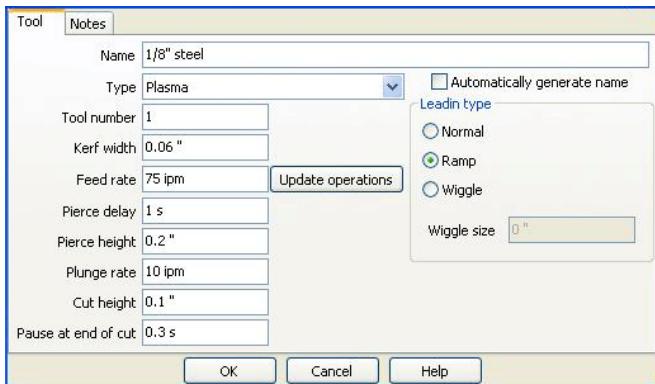
Now we have set up the machine we need to define the plasma cutter settings.

Click on the 'Create a new jet cutting tool' icon  to bring up the plasma cutter dialog box. For the 'Type' select 'Plasma' in the drop down menu.

Enter '1' in the 'Tool number' box. Tools are always displayed in numerical order so the tool number can be used to group different tool settings. Turn off 'Automatically generate name' and enter a name for this tool. You can have a library of tool definitions for different thicknesses of material, machine settings etc.

Note: If you select 'Flame', 'Waterjet', 'Laser' or 'Plate marker' then you will see slightly different options/boxes as each type uses different features. To see the specific features for each type click on the relevant link: [Flame](#), [Waterjet](#), [Laser](#), [Plate marker](#).

Enter the 'Kerf width' (width of the cut line) and 'Feed rate' (plus all other values) as shown.



'Pierce delay' is the time the torch takes to pierce from the moment the cutter is turned on. Set the 'Pierce height' as high as possible. If the pierce height is too high then the arc won't transfer to the material. If it is too low the nozzle could be damaged by droplets of steel blown back as the arc pierces.

'Plunge rate' is the speed the torch plunges down to cut height. If you have 'Ramp' selected in the 'Leadin type' area then this value is ignored. If a ramp lead in is selected then the cutter will ramp down to cut height over the length of the leadin. This can help to reduce nozzle contamination.

Some plasma machines have a small delay between the torch being turned off and the arc going out. If the machine starts moving before the arc has gone out you can get an ugly gouge in the work. To prevent this set the 'Pause at end of cut' to a suitable value.

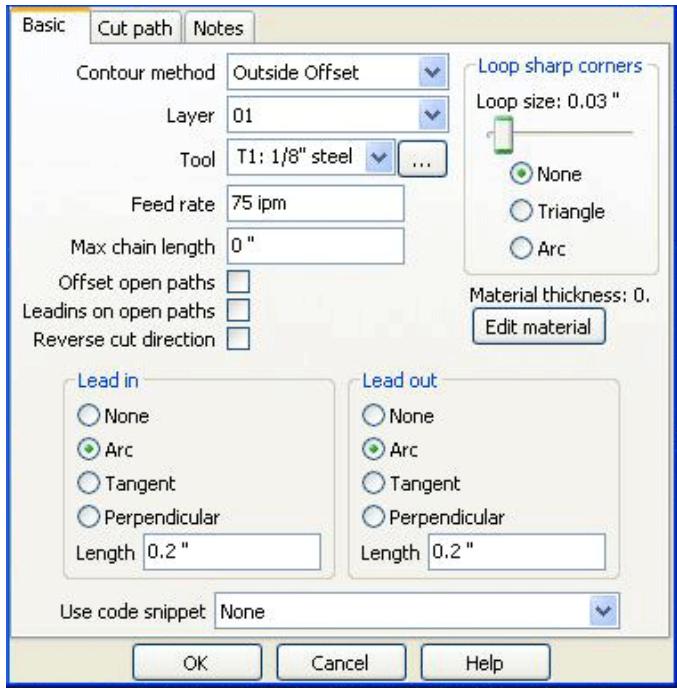
'Update operations' will copy all of the tool's parameters into **any operation** that use this tool. For instance this will reset the feed rate.

Click on <OK> to save the tool.

Define a cut process

Now we have defined the tool we need to use it to cut the letters out.

Press the 'Zoom to fit part' button  to show the drawing. Now press the 'Create a new jet cutting operation' button  to create a new plasma cut operation.



We want to cut the part out of the sheet so select '**Outside offset**' in the '**Contour method**' box. Select the '**01**' layer in the '**Layer**' drop down list. Select '**Tool 1**' (T1) in the '**Tool**' list. Doing this sets most of the parameters from the tool definition we did earlier.

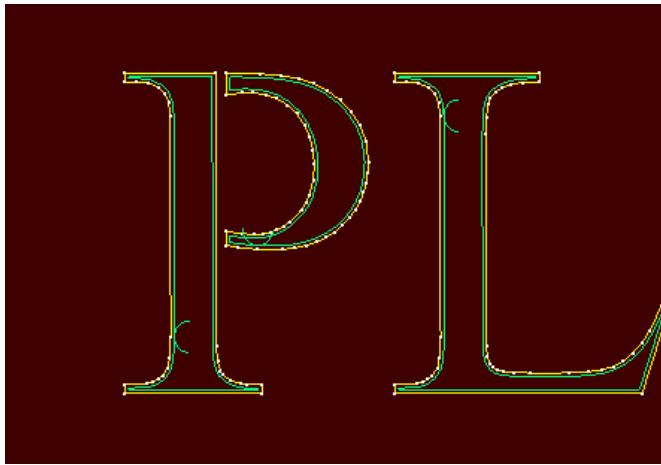
When the plasma cutter pierces the work it leaves an ugly mark. We can tell SheetCam to pierce away from the cut edge then move in and cut the outline. This is called a leadin. We can do the same when the cut ends (leadout).

Ensure that your dialog box looks like the one shown.

Now click on <OK> to save the changes.

Editing start points

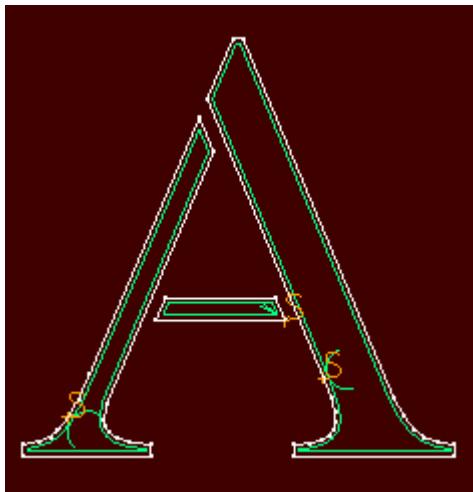
Notice that some of the leadins don't fit properly. This drawing was deliberately drawn to emphasize the problem.



Click on the '**Place cut start point**' button . Each start point is marked with an orange 'S'. Start points can be placed on any junction between lines/arcs (a segment end). If you have the '**Show segment ends**' button down you will see these points marked with white dots. The cursor changes to an arrow with an 'S' when it is near a segment end.

Click on a segment end in the larger part of the loop part of the letter 'P' to move the start point. As the mouse button is being used to place start points you can't use it to scroll the screen. Hold down the shift key and drag with the mouse to scroll.

The letter 'A' presents some more problems. There is simply not enough room in the left hand leg for the leadin/out.

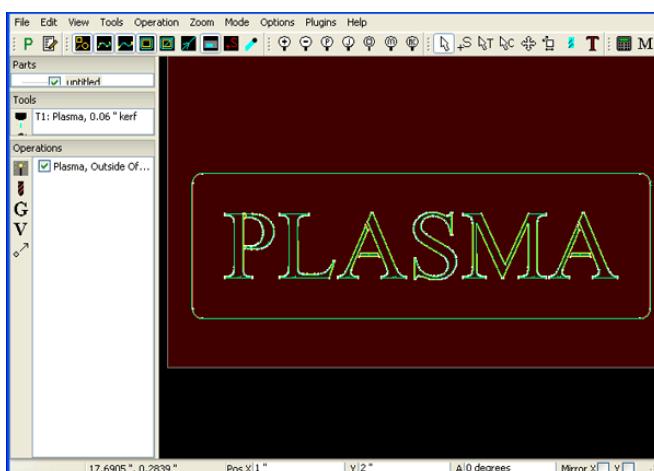


Right click on the start point and select '**Properties**'. Uncheck '**Use operation leadin/leadout**' and change the lead in and out sizes to '**0.15"**' in both cases. Click on <**OK**>. You can now position the start point so there is adequate clearance.

SheetCam has realised that arc leadins won't work in the horizontal part of the 'A' and has shortened them and changed them to lines.

Even so they won't fit properly so you need to shorten them a bit and could even try changing them to 'Perpendicular' from 'Arc'. You will know when you get it right as the yellow warning highlight on the 'Operation' toolbar will disappear when all the issues have been resolved.

Positioning the part on the sheet



At the moment the part is stuck in the bottom left hand corner of the material. We need to move it closer to the middle of the sheet. Click on the '**Nesting**' button  which also allows you to move parts . You can now drag the part on the sheet. Alternatively you can enter values in the '**Pos X**' and '**Y**' boxes on the bottom of the screen.

Generating the G-code

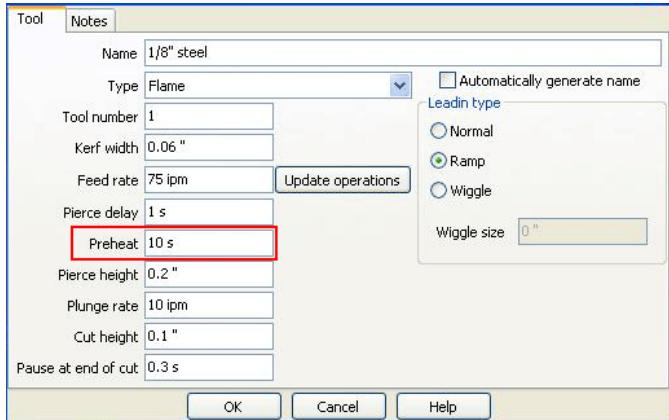
Now we have defined what is to happen we can generate the G-code ready for the machine to run.

Click on the '**Run post processor**' button  to generate the G-code. If this is the first time you have post processed a drawing then you may be asked to select a post processor to run. Try to choose one that best fits your machine.

You will then be prompted for the file name for the G-code file you are about to create.

Flame cutting options

When you select '**Flame**' in the 'Type' drop down menu of the '**Tool**' dialog box you get different options which are applicable to flame cutters only. These are described below:

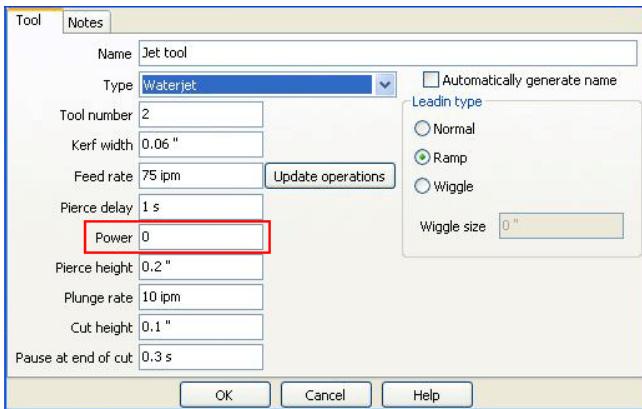


Preheat

The machine will wait for this amount of time to heat up the material before it turns on the oxygen.

Waterjet cutting options

When you select '**Waterjet**' in the 'Type' drop down menu of the '**Tool**' dialog box you get different options which are applicable to waterjet cutters only. These are described below:

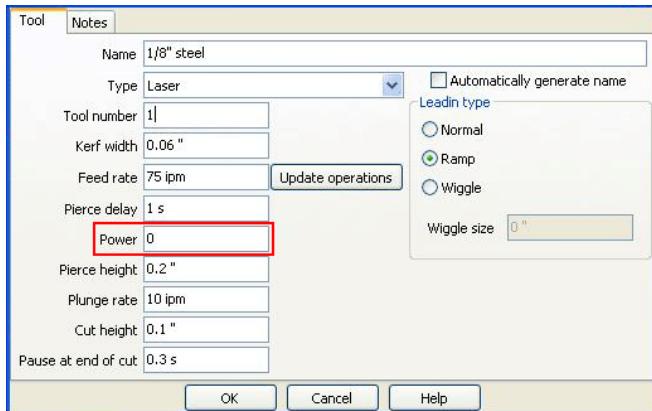


Power

Some waterjet cutters have the ability to adjust the power of the jet via software. Use this box to set the power setting if your machine uses this feature. If your machine does not use this feature then set this value to zero ('0').

Laser cutting options

When you select '**Laser**' in the 'Type' drop down menu of the 'Tool' dialog box you get different options which are applicable to laser cutters only. These are described below:

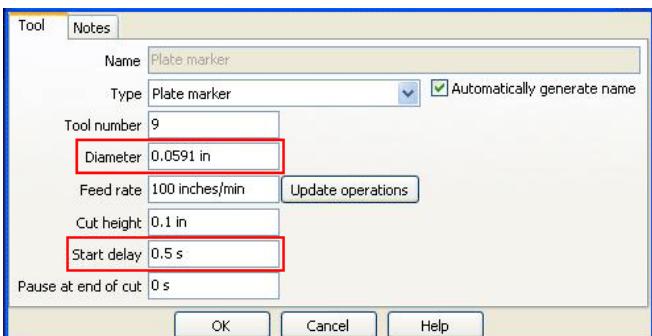


Power

Some laser cutters have the ability to adjust the power of the beam via software. Use this box to set the power setting if your machine uses this feature. If your machine does not use this feature then set this value to zero ('0').

Plate marker options

When you select '**Plate marker**' in the 'Type' drop down menu of the 'Tool' dialog box you get different options which are applicable to plate markers only. These are described below:



Diameter

Enter the diameter of the tool here.

Start delay

Enter the delay time here.

Nesting tutorial

In the previous tutorial we plasma cut a single part. In this tutorial will cut a number of parts. You need to have nesting enabled to do this. Make sure that '**Options\Application options\Advanced tab\Allow multiple parts**' is selected.

First we need to define a bigger sheet to place the parts on. Select '**Options\Job options\Material tab**' and increase the size to 4' x 2'.

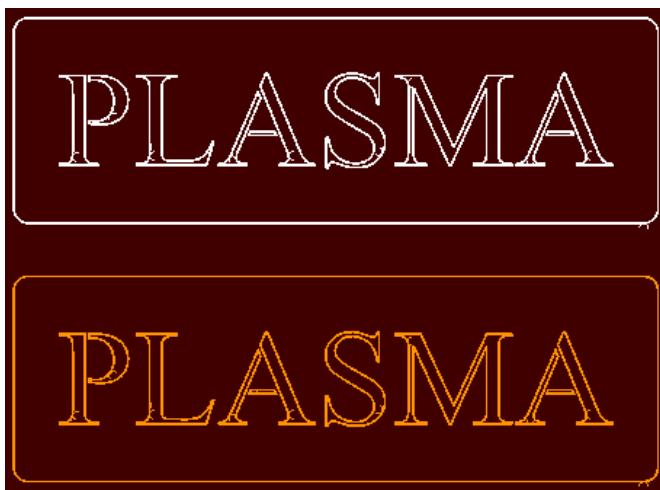
Click on the '**Nesting**' button  then click on the part select it. Right click and select '**Duplicate**'. A duplicate will be created and attached to the mouse cursor. Drag it above the original part and drop it by clicking with the mouse. The duplicate will appear in the part list in the top left hand corner. If you need to you can press the escape key to cancel the operation.

If you are placing a number of parts it helps to have a grid to line them up. Right click and select '**Grid options**'. Enter '**0.5"**' in the '**X**' and '**Y**' boxes in the '**Grid size**' section. Now when you drag parts they will snap to the grid. You can also 'bump' them with the cursor keys.

In the '**Bump increment**' section set the amount each 'bump' moves the part. To rotate a part, enter the 'bump angle' into the '**Angle**' box and set the keys you would like to use rotate the part clockwise or counter-clockwise in the relevant boxes.

Duplicates and copies

Switch to the 'normal' mode by clicking on the '**View toolpaths**' button  . While in 'View toolpaths' mode you will notice that the original part (called the parent) is orange and the duplicates are grey as shown below.



Duplicates simply offset, scale and rotate the code generated from the original part. Any changes to the parent part are reflected in the duplicates. You cannot change any process settings for duplicates as they are tied to their parent.

Instead of duplicating a part you can copy it by selecting '**Copy**' from the right-click menu. A copy is completely independent so you can change its processes or even load a

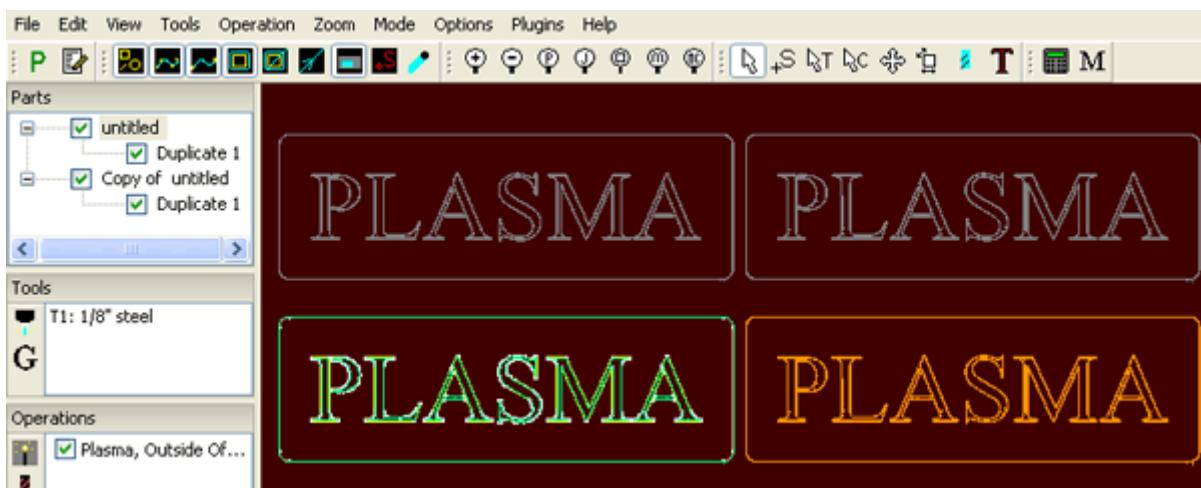
different drawing without affecting the part it was copied from. For obvious reasons you cannot copy a duplicate.

While in 'Nesting' mode paths and outlines are all shown in the same colour to prevent confusion. When in other modes the currently selected part will have its paths and outlines colour coded. All other parts and duplicates will be shown in either grey or orange. To select a part click on it in the part list.

Using Copy

Now we have two signs saying 'PLASMA' and we want plain backing plates that will be painted a different colour.

Make sure you are in '**Nesting**' mode by clicking on the  button then click on the original part to select it then right click and select '**Copy**'. Drop the copy to the right-hand side of the original part. Now right click and select '**Duplicate**'. Drop the duplicate above the new copy. You should end up with something like this:



We now have four identical signs. Two of them need the text removed so they can be used as backing plates.

Click on the '**Contour properties**' button  then click on the copy in the part list (on top left hand side). Click on the outline of the copied part. Right click and select '**Move to layer\New layer**'. Type 'Outline' and click on <OK>. The outline has now been moved to a new layer called 'Outline'.



There will now be no paths generated for the outline but the text will be cut out. This is the opposite of what we want so double-click on the current operation in the '**Operations**' toolbar and change the '**Layer**' to '**Outline**'. The outline will now be cut out and the text won't. If you look closely at the duplicate of this part there are no paths shown for the text in the duplicate.

If you want the practice you can now click on the '**Run post processor**' button  to generate the G-code.

Frequently asked questions (FAQ)

Q: I have a circular pocket with a small keyway tangent to the outside contour. Is there a way to machine the circular pocket then do a tool change to a smaller tool and re-machine the keyway.

A: The easiest way is to use two contour operations. First cut the shape with your larger cutter. As the cutter won't fit in the keyway it will just leave a small bump where the keyway should be. You can now create another contour operation using the smaller cutter. This is slightly wasteful because the small cutter will travel round the whole outline even though you only need to cut the keyway.

A better alternative is to draw a rectangle on another layer that covers the keyway and extends outwards by at least 1/2 the cutter diameter. You can then use an inside profile on this rectangle to just cut the keyway.

Is the key going to have rounded corners to compensate for the cutter radius? If not then turn on 'overcut corners' in the 'cut path' tab of the contour operation. This will cut into the corners of the keyway to allow the square key to fit.

Q: When you are using 'No offset' is there a way to change the start point from the outside of the cut to the inside.

A: With 'No offset' SheetCam has no way of working out what side of the cut you want to keep. A way to work around this is to use an inside or outside offset and use a negative finish allowance to return the cut path to nearly on the line.

Example: If you are using a 0.25in cutter, use a finish allowance of -0.121" (SheetCam will not allow you to use a bigger finish allowance). It will mean that your part will come out 0.004" over/under size.

Q: My outline is shown in grey and I cannot offset it.

A: Somewhere on the outline the lines don't quite join up. Turn off the show segment ends button  and turn on the show path ends button . The start and end points of each line will be shown. A closed shape has **no start** and **no end** so there will be no markers if the shape is fully closed. If you see the end markers on a 'supposedly' closed shape it means the lines are **not joined** at that location. You can now go back to your drawing and fix the problem then resave and open it again in SheetCam.

Note: 'Snaps' are your best weapon against problems like this. Use 'grid' snaps and 'end' snaps whenever possible. They are the best way to create accurate drawings. If you are not familiar with using snaps then look in the documentation for your CAD/drawing program. Virtually all CAD/drawing programs have some form of snap capability.

Q: I have a simple matrix of 4 holes. There is no milling, just the drilling. I defined the drill and the drill operation on the appropriate layer. The graphics look OK except there are no rapid moves shown. No way for the tool to get to the holes. When I run the post processor the code has no moves in it and no drilling.

A: If there are no rapid moves shown then the holes are not being drilled. The most likely reason is that the min. hole size and max. hole size are not set correctly. Only holes between those two sizes will be drilled. Try increasing the max. hole size and decreasing the min. hole size.

Q: I have drawn a shape with sharp edges but when SheetCam shows the cut path the corners are rounded. What is happening?

A: The display shows the centre line of the cutter. Imagine your cutter is a piece of solid bar. As it rolls round the corner the centre line of the bar follows an arc centred on the point of the corner. The cutting edge follows the sharp corner precisely. This is the most efficient way to cut a corner.

Q: I created a new tool and then when I loaded a previously saved job the tool disappeared.

A: 'Job' files contain their own tool sets. If you load a 'job' SheetCam will use that toolset. The way around the problem is to save your toolset before loading the job. You can then reload it at a later date.

Q: I would like to see an accurate description of 'polyline' and 'spline'. I know these are basic words but an understanding of them is taken for granted in most documents and they are not self explanatory.

A: A polyline is a line that can contain multiple vertices and segments; the segments being either straight lines or arcs. It can also be a single segment straight line or arc. In AutoCAD for example you can convert an arc into a polyline. They appear to be exactly the same, which they are, but the information is stored differently.

A spline is a curve, not actually made up of multiple arcs, but defined by a mathematical equation.

Q: I am profiling a part mounted on top of a spoil board. I want it to cut slightly into the spoil so I get a nice sharp edge on my part but SheetCam shows an error and changes the cut depth.

A: You can define a spoil board thickness using the 'Height of bottom of material above table' setting found under the 'Options\Job options\Material' menu item. You will still get a warning if you cut right through the spoil board.

Q: Why would the lead in options be disabled? They used to work but are now greyed out.

A: You have 'Overcut corners' or 'Sharpen corners' on the 'Cut Path' tab checked. You cannot use lead in when overcutting or sharpening corners.

Index

About.....	100	Hints and tips.....	128
Add new code snippet.....	118	HPGL plugin.....	124
Add new contour.....	119	Information area.....	120
Add new pocket.....	119	Initial setup.....	2
Advanced tab.....	96	Insert code.....	70
Angle.....	120	Insert G-code.....	119
Annoying dialogs tab.....	92	Install licence file.....	100
Application options.....	89	Introduction.....	2
Autoload Plugin.....	121	Jet cutting tools.....	30
Batch update.....	97	Jet Cutting tutorial.....	141
Batch Update Plugin.....	122	Job files.....	73
Calculator button.....	118	Job Options.....	79
Calculator Plugin.....	122	Job report.....	103
Centre drilling.....	128	Language tab.....	97
Code editor.....	76	Layer tool.....	76
Context sensitive help.....	iii, 128	Load default toolset.....	75
Contour.....	42	Lower right toolbar.....	120
Copy and duplicate differences.....	128	Mach plugin.....	125
Create a support file.....	101	Machine and material parameters.....	131
Create new drilling operation.....	119	Machine options.....	85
Create new jet cutting operation.....	119	Machine parameters.....	131
Create new tapping operation.....	119	Machine setup.....	133
Creating drawings.....	iii, 11	Machine type.....	6
Creating your drawing.....	128	Machine type tab.....	7, 85
Cursor position.....	120	Material parameters.....	131
Default directories tab.....	95	Material setup.....	133
Depth of cut.....	129	Material tab.....	10, 79
Developer tools.....	101	Mirror.....	120
Display tab.....	89	Mouse wheel.....	132
Document navigation.....	2	Move to.....	72
Drag mode.....	132	Move to operation.....	119
Drawing files.....	74	Nesting.....	110
Drawing import tab.....	92	Nesting Tab.....	82
Drilling.....	67	Nesting tutorial.....	149
DXF plugin.....	123	New code snippet.....	40
Edit contours.....	108	New drill.....	19
Edit cut start points.....	105	New flame tool.....	32
Edit Menu.....	76	New jet cutting button.....	118
Edit tabs.....	107	New laser tool.....	36
Enter new X/Y location.....	120	New mill.....	16
Entering values.....	130	New plasma tool.....	30
Excellon plugin.....	123	New plate marker tool.....	38
Exit.....	76	New pocket.....	51
Explode text in DXF files.....	12, 129	New rigid tap.....	25
FAQ.....	151	New rotary tool button.....	118
File Menu.....	76	New tapping head.....	23
Finish allowance.....	130	New thread mill.....	28
Frequently asked questions.....	151	New V cutter.....	21
G-code plugin.....	124	New waterjet tool.....	34

Open and save options.....	73	Show path ends.....	77, 102
Open DXF options.....	15	Show rapid moves.....	77, 102
Open Excellon options.....	14	Show segment ends.....	76, 102
Open G-code options.....	14	Show start points.....	77
Open HPGL options.....	14	Show tool paths.....	77, 102
Open settings folder.....	101	Show true width.....	77, 103
Open SVG options.....	13	Simulation Plugin.....	126
Opening drawings.....	13	Special case.....	69
Operation setup.....	42	SVG plugin.....	127
Operations toolbar buttons.....	119	Table display tab.....	8, 87
Options menu.....	79	Tapping.....	54
Outlines must be properly closed.....	11	Template files.....	74
Panning.....	132	Text.....	117
Parking Tab.....	81	Text plugin.....	127
Part files.....	74	Thread milling.....	58
Plasma cut.....	61	Tool change tab.....	84
Plugins.....	121	Tool setup.....	16
Plugins menu.....	97	Tool table.....	41
Pocketing tutorial.....	139	Tool table button.....	118
Post process.....	73	Toolbars.....	102
Post processor tab.....	7, 86	Toolbars tab.....	93
Preferred units.....	5	Tools toolbar buttons.....	118
Print.....	76	Toolset files.....	74
Print button.....	103	Tutorials.....	132
Profile tutorial.....	133	Undo.....	76
Rotary tools.....	16	Units tab.....	91
Rotating the screen in 3D.....	132	Use layers.....	129
Run post processor.....	103	View menu.....	76
Run post processor toolbar buttons.....	103	View toolbar buttons.....	102
Run simulation.....	114	Work flow.....	2
Safety.....	1	Work flow procedures.....	11
Save/load tab.....	9, 88	Working envelope tab.....	8, 87
Scroll the screen with the mouse.....	105	Zoom in.....	77, 103
Selection toolbar buttons.....	105	Zoom menu.....	77
Set post processor variable.....	119	Zoom out.....	77, 104
Set post variable.....	71	Zoom to fit current part.....	77, 104
Setting up the machine.....	6	Zoom to fit job.....	78, 104
Setting up the material.....	10	Zoom to fit machine.....	78, 104
Setup wizard.....	2, 99	Zoom to fit material.....	78, 104
Shapes must not self intersect.....	12, 129	Zoom to fit window.....	78, 104
Show cut start points.....	102	Zoom toolbar buttons.....	103
Show input paths.....	76, 102	Zooming and panning.....	132
Show machine and work.....	77, 102	Zooming in and out.....	132
Show path directions.....	77, 102		