

SHEETCAM

2 1/2D CAM

STABLE DESIGN

<http://www.sheetcam.com>

Copyright(c) 2005

User Manual

V5.0.0

Contents

Introduction	4	Bottom Left	12
Safety	4	Drawing centre	12
Initial setup	4	Use drawing origin	12
Setup Wizard	4	Current position	12
Linear units	5	Material bottom left	12
Angular units	5	Open Excellon options	12
Feed rate	5	File scale	12
Plasma cutting	5	Origin	12
Rotary cutting	5	Bottom Left	12
Multiple drawings	6	Drawing centre	12
Complexity	6	Use drawing origin	12
Allow multiple parts	6	Current Position	12
Plasma cutting	6	Material bottom left	13
Rotary cutting	6	Number format	13
Units setup	7	Create tools and processes	13
Linear units	7	Enabled	13
Angular units	7	Drill depth	13
Feed rate	7	Plunge rate	13
Thread pitch	7	Spindle speed	13
Post processor units	7	Open EMF options	13
Select post processor	7	Origin	13
G-code extension	8	Bottom Left	13
Use lower left coordinates	8	Drawing centre	13
Machine setup	8	Use drawing origin	13
Machine origin	8	Current Position	14
Coordinates of machine origin	8	Material bottom left	14
Working envelope size	8	Recent drawings	14
Co-ordinates of table bottom left	9	Material setup	14
Table size	9	Thickness of material	14
Max. clearance between chuck and table	9	Coordinates of bottom left corner	14
Load machine	9	Sizes	14
Save machine	9	Rapid clearance	14
Help	9	Height of bottom of material above table	14
Work flow procedures	9	Parking position	14
Creating drawings	9	Tool setup	15
Outlines must be properly closed	9	New mill/router	15
So how do I find problems?	10	New drill	17
Shapes must not self intersect	10	New V cutter	19
Explode text in DXF files	10	New automatic tap	21
Use layers	10	New rigid tap	24
Splines and bezier curves	10	New plasma cutter	26
Blocks and groups	10	New code snippet	27
Open DXF options	11	Process setup	27
File scale	11	New contour	28
Origin	11	New fill	35
Bottom Left	11	New plasma cut	38
Drawing centre	11	New drilling process	42
Use drawing origin	11	Edit G-code	46
Current Position	11	New tapping process	47
Material bottom left	12	Post process	50
Use points for drilling	12	Open and save options	51
Open HPGL options	12	Other Features	52
Origin	12	File Menu	52

Contents (cont.)

Edit Menu	52	Entering values	70
View menu	52	Finish allowance	70
Options Menu	56	Machine and material parameters	70
Help Menu	59	Machine parameters	70
Toolbars	61	Material parameters	71
View toolbar buttons	61	Mouse wheel	71
Run post processor toolbar buttons	62	Zooming and panning	71
Zoom toolbar buttons	62	Panning	71
Selection toolbar buttons	63	Rotating the screen in 3D	72
Tools toolbar buttons	67	Drag mode	72
Processes toolbar buttons	68	Tutorials	72
Lower right toolbar	69	Profile tutorial	72
Hints and Tips	69	Pocketing tutorial	76
Centre drilling	69	Plasma tutorial	78
Context sensitive help	69	Nesting Tutorial	82
Copy and duplicate differences	69	Frequently asked questions	84
Depth of cut, peck depth etc.	69	Index	86

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) 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.

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.

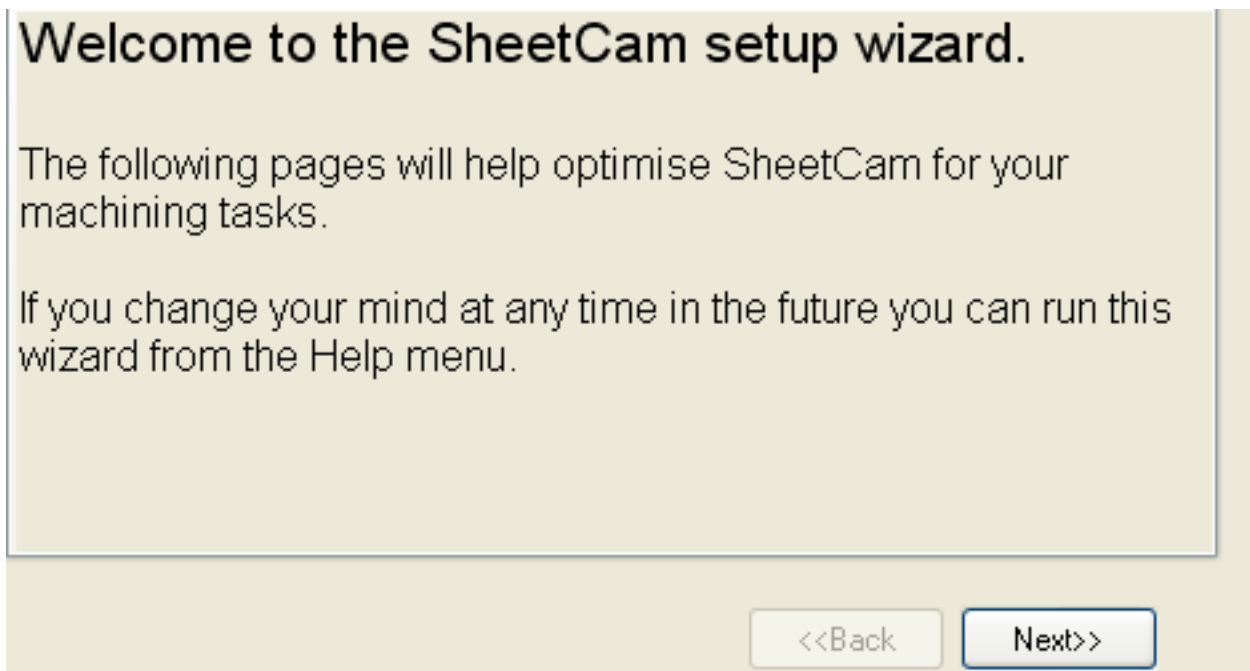
It is strongly advised that all post processor files are run through a CNC simulator of some sort to check for errors prior to running them on the machine in question. CNC simulators are available as 'freeware' at a number of sites on the internet and can be part of the control software itself. 'Stable 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.

Please select your preferred units

Linear units: inch

Angular units: degrees

Feed rate: inches/min

<<Back Next>>

Linear units

Enter the linear units from the large drop down menu. Use the scroll bar to see more options.

Angular units

Enter the angular units from the drop down menu.

Feed rate

Enter the feed rate 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 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).

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

Please choose the post processor you wish to use

Don't worry if you aren't sure. You can always change this later

Mach2

☐ Use metric units ☒ Use imperial units

Mach2 post processor

Non modal G-codes
Modal coordinates
Comments enclosed with (and)
Incremental IJ
uses G43 tool length offsets

<<Back Next>>

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

The fourth screen of the 'Setup Wizard' asks about your machine type and multiple drawings.

What type of machining do you want to do?

☐ Plasma/flame cutting
☒ Milling/routing

Do you want to be able to cut multiple drawings at the same time (manual nesting)?

☒ Yes
☐ No

<<Back Next>>

Plasma cutting

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

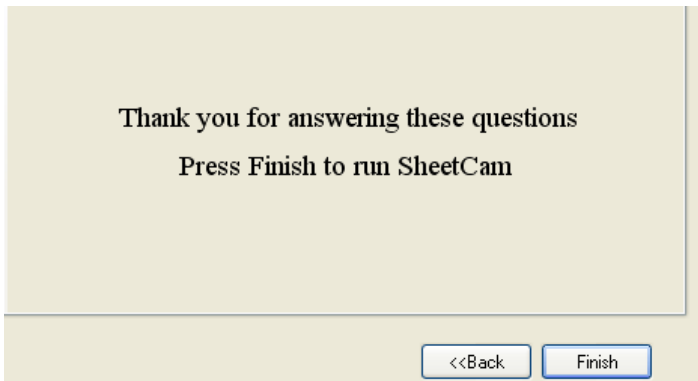
Rotary cutting

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 'Processes' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden.

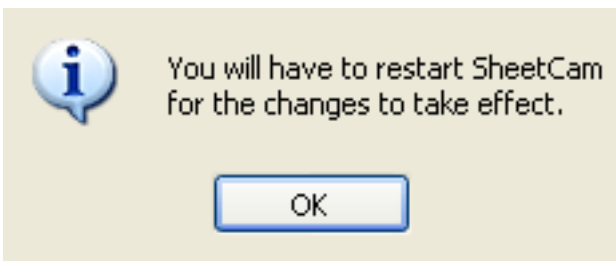
Multiple drawings

Checking this item will allow you to open multiple drawings within in the same 'job'.

The fifth screen of the 'Setup Wizard' completes the setup process. Click **<Finish>** to continue.



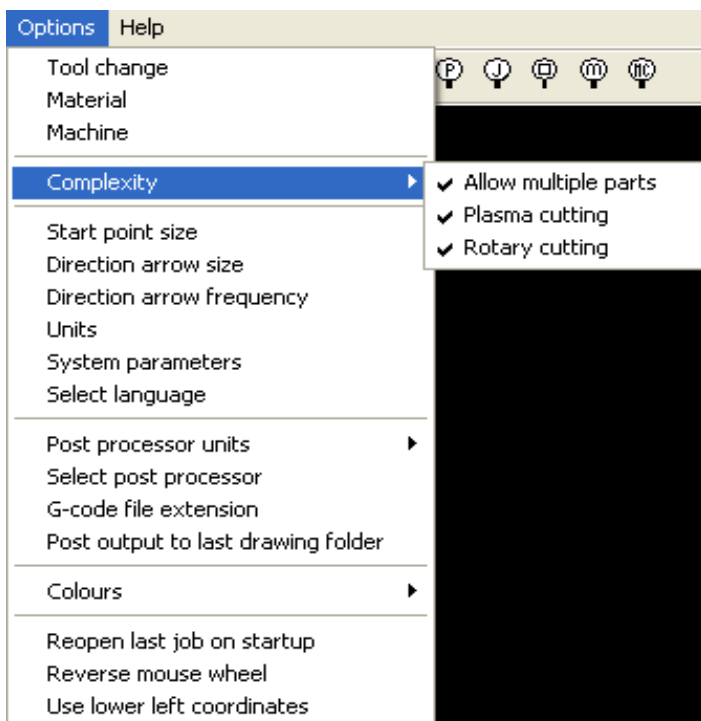
The sixth screen of the 'Setup Wizard' is a reminder to close and re-start SheetCam in order for the changes you made to take effect.



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

Complexity

'Hovering' over or clicking this menu item will open the following pop-out dialogue box.



Allow multiple parts

Checking this item will allow you to have multiple parts within in the same piece of material specified in the 'Options/material' dialogue box.

Plasma cutting

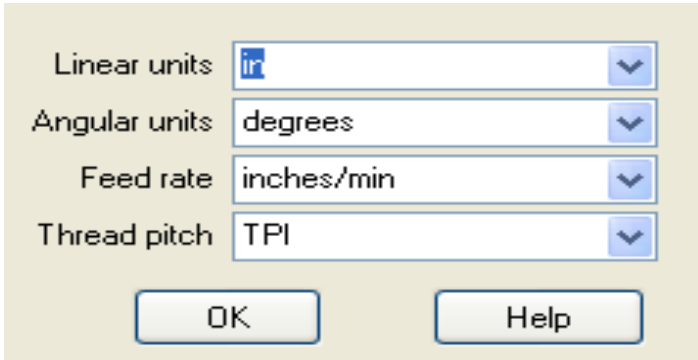
Checking this item will toggle the plasma cutting option on or off. When checked the plasma cutting 'controls' are available under the 'Tools' menu and the 'Processes' menu. The respective buttons also appear on the various 'tool bars'. When unchecked these 'controls' are hidden.

Rotary cutting

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

Units setup

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



The Units setup dialog box contains four drop-down menus and two buttons. The 'Linear units' menu is set to 'in', 'Angular units' is set to 'degrees', 'Feed rate' is set to 'inches/min', and 'Thread pitch' is set to 'TPI'. The 'OK' and 'Help' buttons are at the bottom.

Linear units

Enter the linear units from the large drop down menu. Use the scroll bar to see more options.

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.

After making any changes click **<OK>** to accept and close the box.

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).

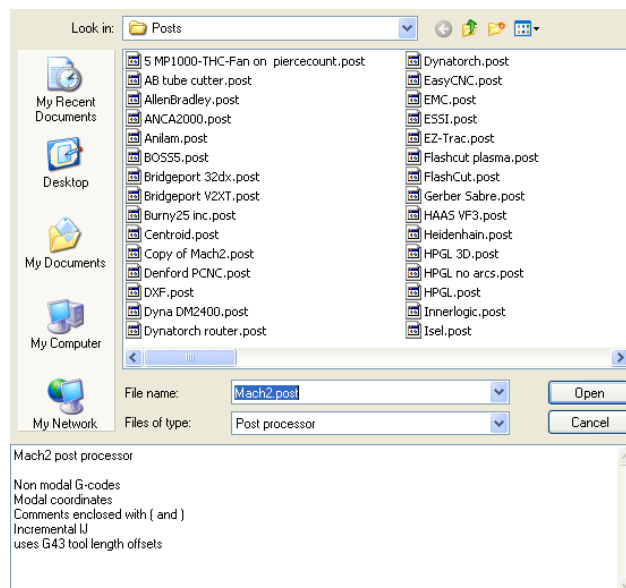
Post processor units

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

Select post processor

Calls up the standard Windows 'open file' dialogue box below.

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



G-code extension

G-code file extension

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

Use lower left coordinates

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

Machine setup

Selecting 'Machine' from the 'Options' menu will bring up the following screen, enter the relevant information as shown below.

X coordinate of machine origin	0 in
Y coordinate of machine origin	0 in
Working envelope X size	24 in
Working envelope Y size	12 in
X coordinate of table bottom left	0 in
Y coordinate of table bottom left	0 in
Table X size	36 in
Table Y size	18 in
Max clearance between chuck and table	12 in

Machine origin

Enter coordinates

Note: All units based on the selection in the 'Options/Units' menu.

The 'Setup machine parameters' dialogue box allows you to specify your machine parameters.

Machine 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.

Coordinates of machine origin

Enter the X and Y coordinates for the machine origin.

Note: This is usually a negative (-) value but could be positive (+) if required.

Working envelope size

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

Co-ordinates of table bottom left

This is the position of the bottom left hand corner of the table relative to the working envelope. Some machines have a large table but a relatively small working envelope. If you are cutting a part that is bigger than the working envelope it can be useful to see if it will fit on the table. If you don't want to display the table then enter 0 for the X and Y sizes.

Table size

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

Max. clearance between chuck and table

Enter the maximum clearance amount.

Load machine

Clicking this button will load a previously saved set of parameters.

Note: You can use SheetCam to generate G-code for several different machines (eg. router, mill and plasma cutter) by saving the parameters for each machine as separate 'machine' files and then loading the required file for the current part.

Save machine

Clicking this button will save the current values as a 'machine' file. Select a suitable location on your hard drive and type in a name for the file then click **<OK>**.

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

Once all the values are entered click **<OK>** to exit the screen.

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 DXF, HPGL or Excellon 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 purple. If your lines are very close (as defined under 'Options/Systems parameters') 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 . 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.

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).

1 contour selected
from layer SWIVEL_PLATE

OK

Note: If your drawing is all on one layer you can use SheetCam to split different processes off onto 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.

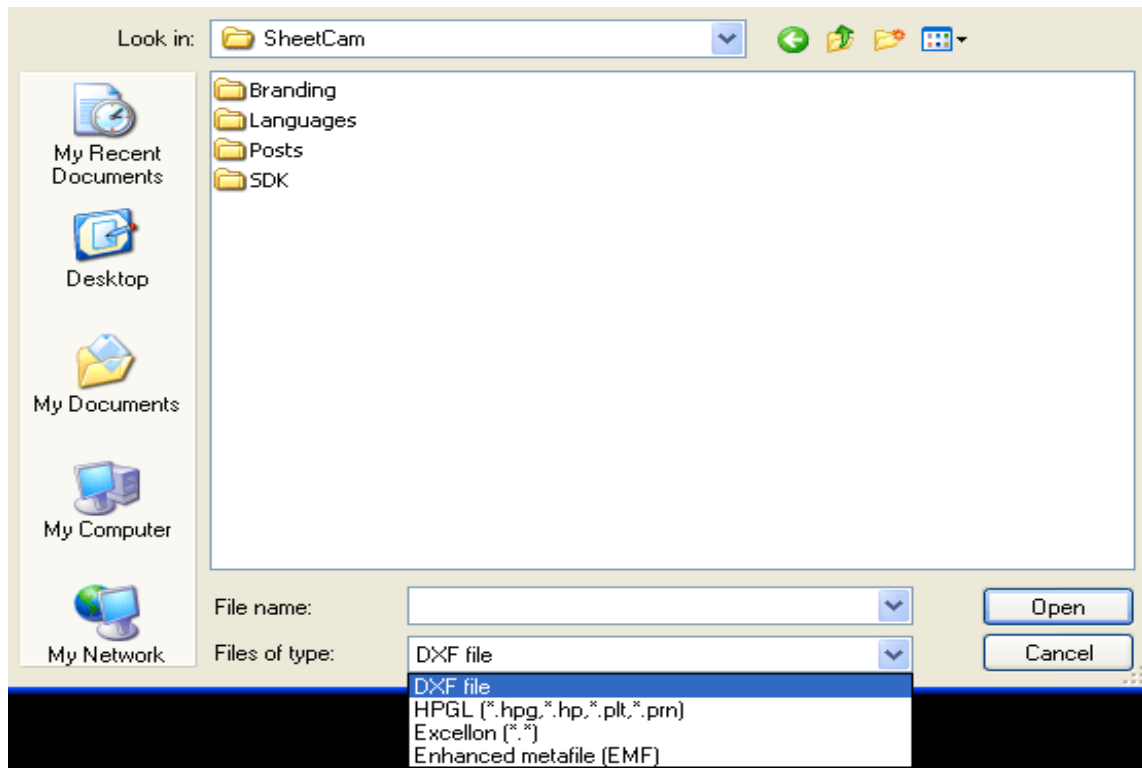
Splines and bezier curves

Try not to use splines and/or bezier curves. SheetCam cannot currently handle these entities. If you have to use splines then first of all, try to save the drawing as HPGL or AutoCad DXF V12. If that does not work then you may be able to 'explode' the splines to convert them into polylines. CorelDraw is particularly bad for this. For best results always use HPGL export instead of DXF export with CorelDraw.

Blocks and groups

Do not use blocks and/or groups. SheetCam does not currently understand blocks. Explode the blocks/groups into their component parts before saving the file.

Opening drawings

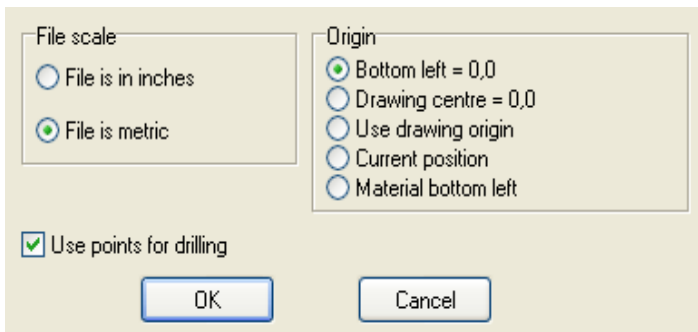


Selecting 'Open drawing' from the 'File' menu opens a pop-up window allowing you to browse for your drawing file. Acceptable formats are DXF, HPGL, Excellon or EMF files. The formats are selectable from the drop down menu. Each particular format will open its own specific settings dialogue window when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

To view the settings for each drawing type click on the file type in the graphic above.

Note: For tips on creating suitable drawings see the 'Hints and Tips' section.

Open DXF options



File scale

Select the file scale from the available options.

Origin

Select the origin location from the available options.

Bottom Left

The bottom left hand corner of the drawing is aligned to the machine 0,0 position.

Drawing centre

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

Use 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.

Current Position

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

Material bottom left

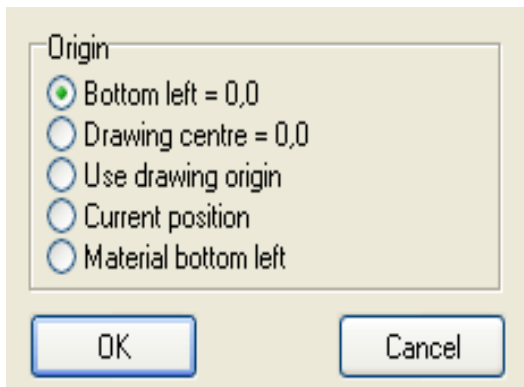
The bottom left corner of the material is aligned to the machine 0,0 position.

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.

Open HPGL options



Origin

Select the origin location from the available options.

Bottom Left

The bottom left hand corner of the drawing is aligned to the machine 0,0 position.

Drawing centre

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

Use 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.

Current position

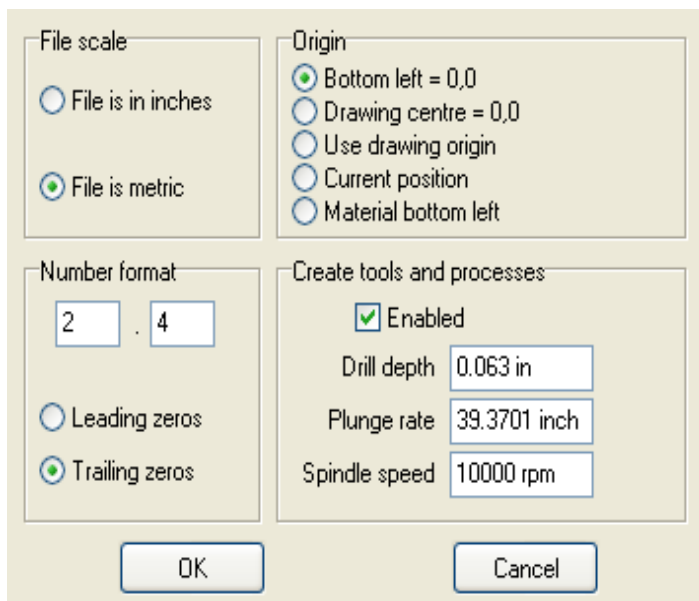
The centre of the new drawing is placed in exactly the same position as the current drawing.

Material bottom left

The bottom left corner of the material is aligned to the machine 0,0 position.

Note: For tips on creating suitable drawings see the 'Hints and Tips' section.

Open Excellon options



File scale

Select the file scale from the available options.

Origin

Select the origin location from the available options.

Bottom Left

The bottom left hand corner of the drawing is aligned to the machine 0,0 position.

Drawing centre

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

Use 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.

Current Position

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

Material bottom left

The bottom left corner of the material is aligned to the machine 0,0 position.

Number format

Excellon files use fixed point numbers. The number format specifies the number of leading and trailing digits. For instance 2.4 means two leading digits and 4 trailing digits. This is quite a common format for Inch files. Metric files often use 4.3 (4 leading, 3 trailing). To save space in the file leading or trailing zeros can be suppressed. You need to specify which system is used. If the file does not use zero suppression then it does not matter if you select 'leading zeros' or 'trailing zeros'.

Common problems:

My drawing comes out 10x too big.

If you are using leading zero suppression, increase the second digit of the number format otherwise decrease the first digit.

My drawing comes out 10x too small.

If you are using leading zero suppression, decrease the second digit of the number format otherwise increase the first digit.

My drawing comes out 25x too big or 25x too small.

You have selected metric for an inch file or vice-versa.

Note: Some files include tags for inch/metric and leading/trailing zeros. In that case they will override your selection.

Create tools and processes

Enabled

As the tools are specified in the file SheetCam can set up the tool table and processes 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.

Drill depth

Enter the required drill depth in the box.

Plunge rate

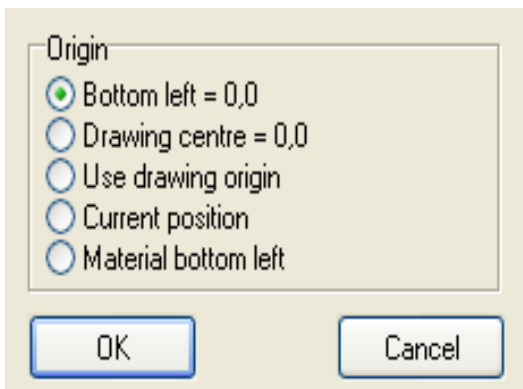
Enter the required plunge rate in the box.

Spindle speed

Enter the required spindle speed in the box.

Note: For tips on creating suitable drawings see the 'Hints and Tips' section.

Open EMF options



Origin

Select the origin location from the available options.

Bottom Left

The bottom left hand corner of the drawing is aligned to the machine 0,0 position.

Drawing centre

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

Use 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.

Current Position

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

Material bottom left

The bottom left corner of the material is aligned to the machine 0,0 position.

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.

Material setup

After opening your drawing you will need to tell SheetCam about the material you will be using.

Note: All units based on the selection in the 'Options/Units' menu.

Selecting 'Material' from the 'Options' menu opens the 'Material setup' dialogue box which allows you to specify the material parameters.

Thickness of material 0.25 in

X Coordinate of bottom left corner 1 in

Y Coordinate of bottom left corner 1 in

X size 5.75 in

Y size 9.5 in

Rapid clearance 0.25 in

Height of bottom of material above table 0

Parking position

X position 0 in ☐ Use X position

Y position 0 in ☐ Use Y position

Z position 0 in ☐ Use Z position

OK Help

Thickness of material

Enter the material thickness here.

Coordinates of bottom left corner

Enter the X and Y coordinate for the bottom left corner of the material. The 'values' are the distances from the machine zero point (set using the 'Options/machine' dialogue box) to the bottom left corner of the material, this can be a positive (+) **or** a negative (-) figure.

Sizes

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

Rapid clearance

Enter 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. Any rapid move in the Z axis will stop at this point and any further Z axis travel in a downward direction will be at the chosen feed rate.

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.

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.

Tool setup

Once you have the material information entered 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.

Tools Processes Options

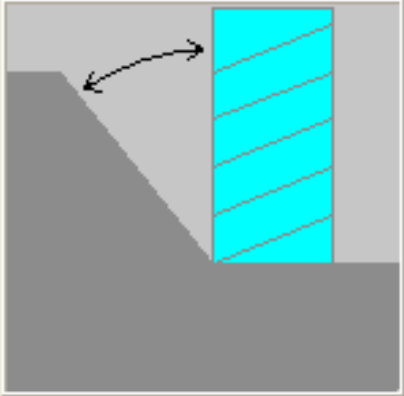
New mill/router
New plasma cutter
New code snippet

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

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

New mill/router

Tool number	1	☒ Automatically generate name
Tool name	Mill/Router, 0.25 in diameter	
Tool type	Mill/Router	
Diameter	0.25 in	
		Spindle rotation ☒ CW ☐ CCW ☐ Off
Flute length	2 in	
Tool projection	3 in	
Tool length offset	1 in	
Z increment	0.1 in	
Feed rate	3 inches/min	
Plunge rate	2 inches/min	
Ramp angle	0 degrees	
Spindle speed	1000	
OK Cancel Update processes Help		



Note: All units based on the selection in the 'Options/Units' menu.

Note: This menu item is only visible if it has been selected in the 'Options/Complexity' menu.

Tool number

Enter the tool number here.

Automatically generate name

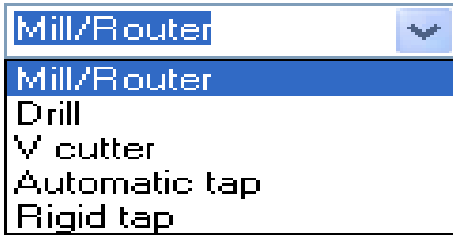
Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool name

Manually enter tool name if the box above is not checked.

Tool type

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



- Note:** A mill/router can plunge and cut sideways. It can be used by the contour, drill and pocket processes.
- Note:** A drill can plunge but cannot cut sideways. It can only be used by the drill process.
- 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.

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 projection

Enter the tool projection length here (i.e. the distance from the tip of the tool to the face of the chuck/collet).

Tool length offset

Enter the amount of tool offset here.

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.

Z increment

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.

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.

OK

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

Cancel

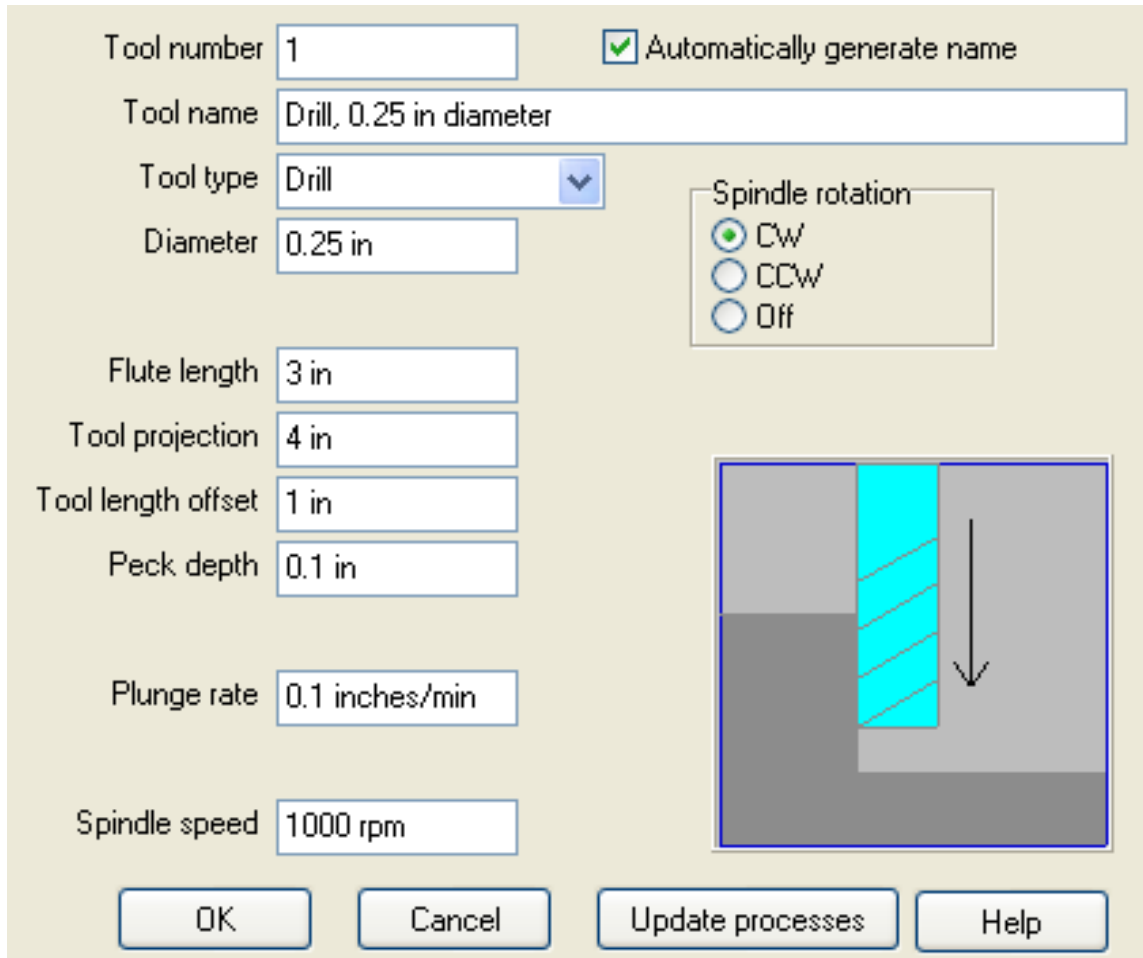
'Cancel' cancels any changes and closes the dialogue box.

Update processes

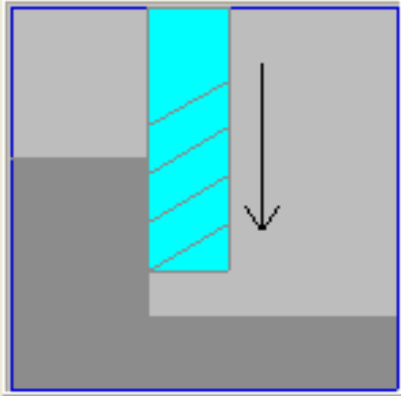
'Update processes' will copy all of the tool's parameters into **any processes** that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

Note: This dialogue box can also be accessed via the 'Add a new mill/router tool' button  located in the left-hand vertical toolbar.

New drill



The 'New drill' dialog box is a form for configuring a new drill tool. It features several input fields and a preview window. The fields are: 'Tool number' (set to 1), 'Tool name' (set to 'Drill, 0.25 in diameter'), 'Tool type' (set to 'Drill'), 'Diameter' (set to '0.25 in'), 'Flute length' (set to '3 in'), 'Tool projection' (set to '4 in'), 'Tool length offset' (set to '1 in'), 'Peck depth' (set to '0.1 in'), 'Plunge rate' (set to '0.1 inches/min'), and 'Spindle speed' (set to '1000 rpm'). There is a checkbox for 'Automatically generate name' which is checked. A 'Spindle rotation' section contains three radio buttons: 'CW' (selected), 'CCW', and 'Off'. A preview window on the right shows a 3D model of a drill bit cutting into a workpiece, with a downward arrow indicating the cutting direction. At the bottom are four buttons: 'OK', 'Cancel', 'Update processes', and 'Help'.

Tool number	1	☒ Automatically generate name
Tool name	Drill, 0.25 in diameter	
Tool type	Drill	
Diameter	0.25 in	
Flute length	3 in	
Tool projection	4 in	
Tool length offset	1 in	
Peck depth	0.1 in	
Plunge rate	0.1 inches/min	
Spindle speed	1000 rpm	
Spindle rotation		
☒ CW		
☐ CCW		
☐ Off		
		
OK Cancel Update processes Help		

Note: All units based on the selection in the 'Options/Units' menu.

Tool number

Enter the tool number here.

Automatically generate name

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

Tool name

Manually enter tool name if the box above is not checked.

Tool type

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

Mill/Router

Mill/Router

Drill

V cutter

Automatic tap

Rigid tap

- Note:** A mill/router can plunge and cut sideways. It can be used by the contour, drill and pocket processes.
- Note:** A drill can plunge but cannot cut sideways. It can only be used by the drill process.
- 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.

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 projection

Enter the tool projection length here (i.e. the distance from the tip of the tool to the face of the chuck/collet).

Tool length offset

Enter the amount of tool offset here.

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.

Peck depth

Enter the peck depth here. This function controls the depth of each peck 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.

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.

OK

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

Cancel

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

Update processes

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

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

New V cutter

The 'New V cutter' dialog box is shown with the following fields and controls:

- Tool number:** 1
- Automatically generate name:** ☒
- Tool name:** V cutter, 0.05 in diameter
- Tool type:** V cutter (dropdown menu)
- Tip diameter:** 0.05 in
- V angle:** 60
- Flute length:** 0.5 in
- Tool projection:** 2 in
- Tool length offset:** 1 in
- Z increment:** 0.2 in
- Feed rate:** 20 inches/min
- Plunge rate:** 5 inches/min
- Ramp angle:** 0 degrees
- Spindle speed:** 4000 rpm
- Spindle rotation:** ☒ CW, ☐ CCW, ☐ Off
- Diagram:** A large downward-pointing arrow representing the V-cutter.
- Buttons:** OK, Cancel, Update processes, Help

Note: All units based on the selection in the 'Options/Units' menu.

Tool number

Enter the tool number here.

Automatically generate name

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

Tool name

Manually enter tool name if the box above is not checked.

Tool type

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

Mill/Router

▼

Mill/Router

Drill

V cutter

Automatic tap

Rigid tap

- Note:** A mill/router can plunge and cut sideways. It can be used by the contour, drill and pocket processes.
- Note:** A drill can plunge but cannot cut sideways. It can only be used by the drill process.
- 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.

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 'Contour process')

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 projection

Enter the tool projection length here (i.e. the distance from the tip of the tool to the face of the chuck/collet).

Tool length offset

Enter the amount of tool offset here.

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.

Z increment

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.

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.

OK

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

Cancel

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

Update processes

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

New automatic tap

The dialog box is titled 'New automatic tap'. It contains the following fields and controls:

- Tool number:** 1
- Automatically generate name:** ☒
- Tool name:** Automatic tap, 0.375 in x 16 TPI
- Tool type:** Automatic tap (dropdown menu)
- Diameter:** 0.375 in
- Spindle rotation:** CW (selected), CCW, Off
- Flute length:** 1 in
- Tool projection:** 1.5 in
- Tool length offset:** 0 in
- Axial travel:** 0.15 in
- Reverse multiplier:** 200 %
- Pitch:** 16 TPI
- Underfeed:** 10 %
- Spindle speed:** 150 rpm

At the bottom right, there is a diagram showing a cross-section of a workpiece with a tap being inserted. The tap is highlighted in cyan, and the workpiece is grey. A vertical double-headed arrow indicates the axial travel of the tap.

At the bottom of the dialog, there are four buttons: OK, Cancel, Update processes, and Help.

Note: All units based on the selection in the 'Options/Units' menu.

Note: This menu item is only visible if it has been selected in the 'Options/Complexity' menu.

Tool number

Enter the tool number here.

Automatically generate name

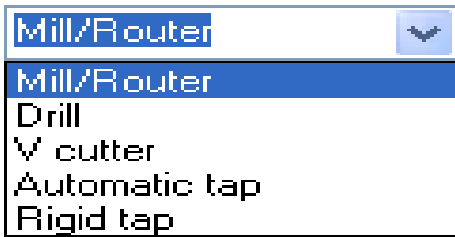
Checking this box will automatically generate the tool name based on selections made in the lower boxes.

Tool name

Manually enter tool name if the box above is not checked.

Tool type

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



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

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

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.

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 projection

Enter the tool projection length here (i.e. the distance from the tip of the tool to the face of the chuck/collet).

Tool length offset

Enter the amount of tool offset here.

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.

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.

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.

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.

OK

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

Cancel

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

Update processes

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

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

New rigid tap

Tool number	5	☒ Automatically generate name
Tool name	Rigid tap, 0.375 in x 16 TPI	
Tool type	Rigid tap	Spindle rotation ☒ CW ☐ CCW ☐ Off
Diameter	0.375 in	
Flute length	1 in	
Tool projection	1.5 in	
Tool length offset	0 in	
Reverse multiplier	150 %	
Pitch	16 TPI	
Underfeed	3 %	
Spindle speed	150 rpm	
OK Cancel Update processes Help		

Note: All units based on the selection in the 'Options/Units' menu.

Note: This menu item is only visible if it has been selected in the 'Options/Complexity' menu.

Tool number

Enter the tool number here.

Automatically generate name

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

Tool name

Manually enter tool name if the box above is not checked.

Tool type

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

Mill/Router	v
Mill/Router	
Drill	
V cutter	
Automatic tap	
Rigid tap	

- Note:** A mill/router can plunge and cut sideways. It can be used by the contour, drill and pocket processes.
- Note:** A drill can plunge but cannot cut sideways. It can only be used by the drill process.
- 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.

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 projection

Enter the tool projection length here (i.e. the distance from the tip of the tool to the face of the chuck/collet).

Tool length offset

Enter the amount of tool offset here.

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.

OK

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

Cancel

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

Update processes

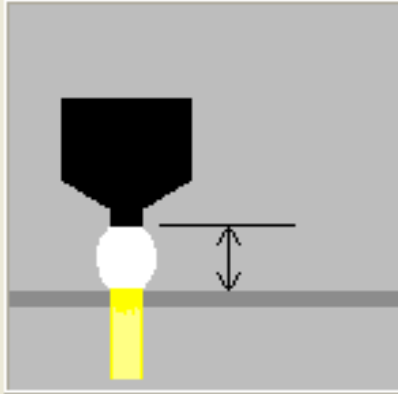
'Update processes' will copy all of the tool's parameters into any processes that use this tool. For instance this will reset the spindle speed, feed rate, ramp angle etc.

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

New plasma cutter

Tool number	1	☒ Automatically generate name
Tool name	Plasma, 0.04 in kerf	
Kerf width	0.04 in	
Feed rate	100 inches/min	
Preheat	0 s	
Pierce delay	0.2 s	
Pierce height	0.2 in	
Plunge rate	8 inches/min	
	☐ Ramp lead in	
Cut height	0.1 in	
Pause at end of cut	0 s	



Note: All units based on the selection in the 'Options/Units' menu.

Note: This menu item is only visible if it has been selected in the 'Options/Complexity' menu.

Note: If you need to use a plasma cutter to pierce holes at specific locations the 'Drilling process' can be used with a plasma tool setting for this purpose. See the 'Drilling process - Special case' section on page 42 for more information.

Tool number

Enter the tool number here.

Automatically generate name

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

Kerf width

Enter the kerf width (cut width) here.

Feed rate

Enter the required feed rate here.

Preheat

Enter the preheat time here. This is mainly for use with oxy-fuel cutting to preheat the work before turning on the oxygen to pierce and start the cut.

Pierce delay

Enter the delay time here.

Pierce height

Enter the pierce height here.

Plunge rate

Enter the plunge rate here.

Ramp lead in

Clicking this box will enable the ramp lead in option. Instead of moving straight down the cutter will ramp down to the cut start. This can help reduce tip contamination.

Cut height

Enter the cut height here.

Pause at end of cut

Enter the pause time here.

OK

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

Cancel

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

Update processes

'Update processes' will copy all of the tool's parameters into **any processes** that use this tool. For instance this will reset the kerf width, feed rate, pierce delay, etc.

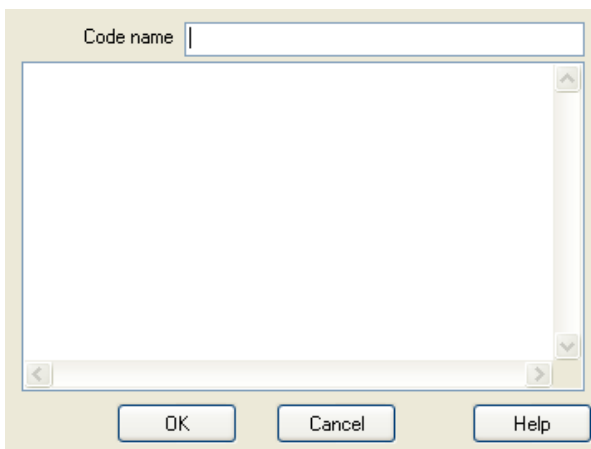
Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

Note: This dialogue box can also be accessed via the 'Add a new plasma cutter 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 code snippet



New code snippet creates a 'tool' that consists of one or more lines of G-code. This is then used by the 'Insert code' process.

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

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

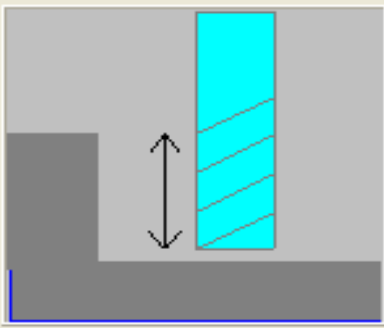
Process setup

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

New contour process
New plasma cut process
Insert code snippet
New tapping process

New fill process
New drilling process
Set post variable

New contour

Basic	Cut start	Cut path	Notes
Contour method	Outside offset		
Layer	OUTLINE		
Tool	T3: Mill/Router, 3 mm diameter		
Start depth	0		
Cut depth	0.2 in		
Z increment	0.2 in		
Finish allowance	0.1 in		
Feed rate	20 inches/min		
Plunge rate	5 inches/min		
Spindle speed	4000 rpm		
Coolant		☐ Flood ☐ Mist	
			
Cut sequence: 1 cut of 0.2 in			
OK		Cancel	
		Help	

Note: All units based on the selection in the 'Options/Units' menu.

Contour method

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

Contour method	Outside offset
Layer	Inside offset
	Outside offset
Tool	No offset

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.

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.

Layer	OUTLINE
Tool	TEXT
	OUTLINE

Tool

Select the correct tool using the down menu shown below.

Tool	T3: Mill/Router, 3 mm diameter ▼
Start depth	T3: Mill/Router, 3 mm diameter T5: Mill/Router, 0.375 in diameter

Edit

Calls up the 'Tool Edit' dialogue 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.

Z increment

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 'process' to machine the part to the 'finished' size using a finishing cutter if required and/or a different 'Z increment' 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.

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.

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

Cut start

Basic Cut start Cut path Notes

Ramp angle 0 degrees

Lead in

- ☒ None
- ☐ Arc
- ☐ Tangent
- ☐ Perpendicular

Size 0 in

Lead Out

- ☒ None
- ☐ Arc
- ☐ Tangent
- ☐ Perpendicular

Size 0 in

Use Code snippet None

OK Cancel Help

Note: All units based on the selection in the 'Options/Units' menu.

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 process will turn yellow to warn you that this is happening.

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

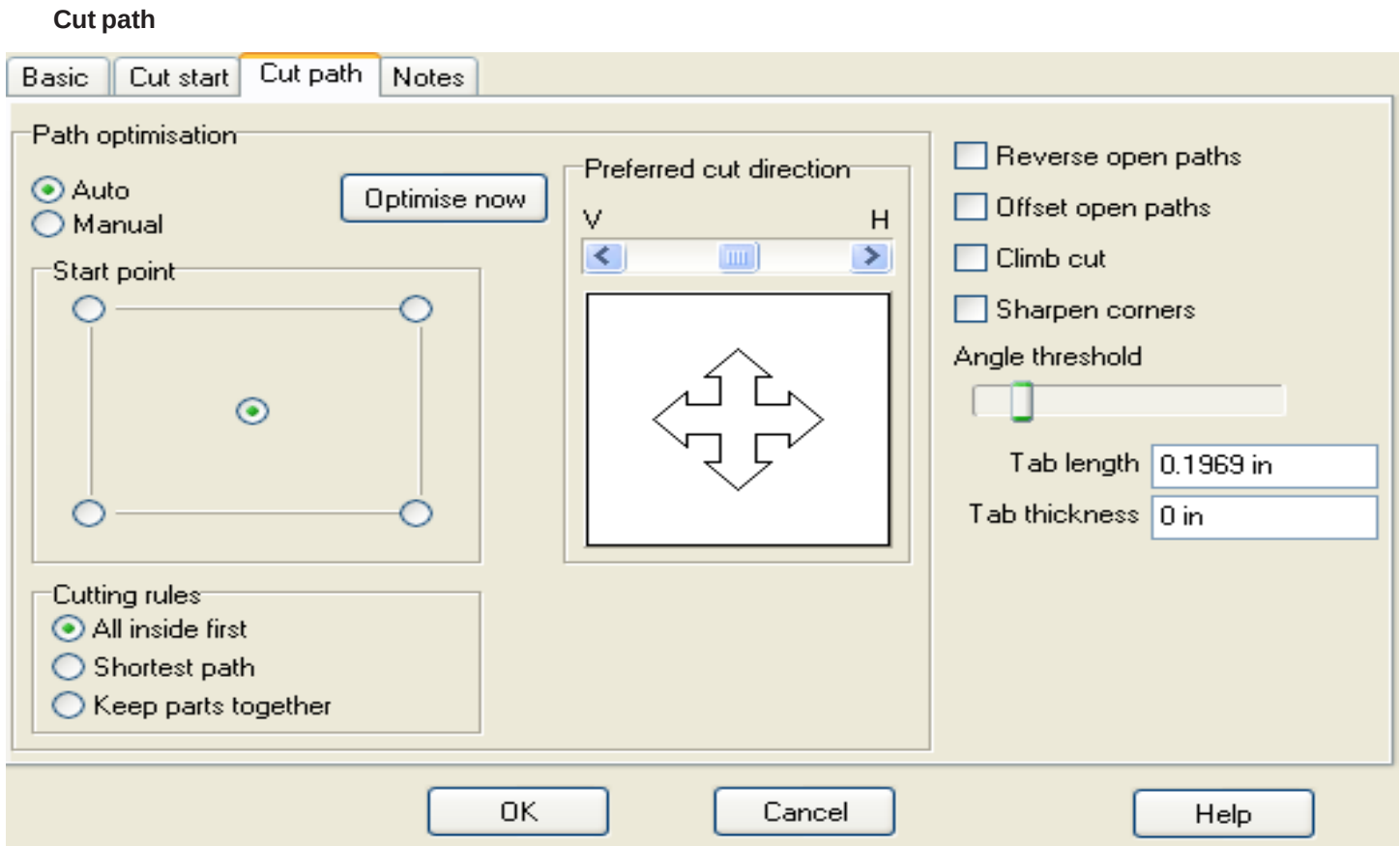
Select the type of lead out required using the 'radio' buttons and enter the size in box.

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.



Auto/manual

Select auto or manual path optimization 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 point

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.

Preferred cut direction

This adds a bias to the path optimization. For example, if you set the slider toward horizontal, 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.

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

Check this box to apply 'corner sharpening' to 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:

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 check box for 'Sharpen corners' and a slider called 'Angle threshold'. 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.

Overcut corners

Check this box to apply 'over cutting' to 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

- ☐ Reverse open paths
- ☐ Offset open paths
- ☐ Climb cut
- ☐ Sharpen corners

Angle threshold



Tab length 0.1969 in

Tab thickness 0 in

Note: This option only appears if you have selected a V cutter as your tool.

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.

Tab length/Tab thickness

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.

Tab 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.

Tab 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.

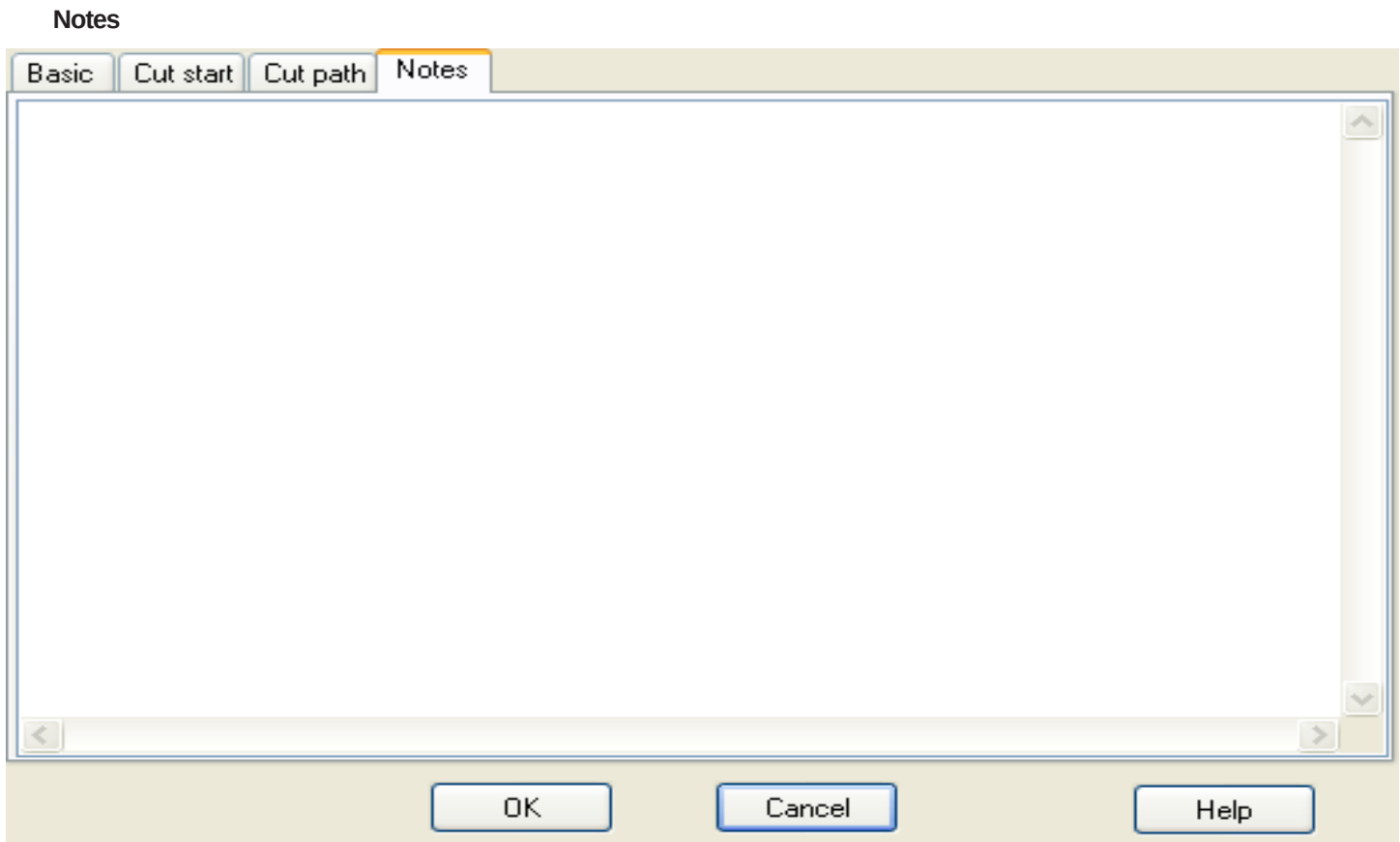
Note: By using the tab length/thickness controls you can make SheetCam clear any clamps or fixtures as it cuts provided that you know where the clamps will be. By making the 'thickness' greater than the material thickness **plus** the clamp thickness SheetCam will 'rapid' traverse for the length of the 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 process 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 process 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.

**Notes**

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

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

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

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

New fill

Basic

Notes

Pocket method Spiral pocket

Layer TEXT

Tool T3: Mill/Router, 3 mm diameter Edit...

Start depth 0 in

Cut depth 0.1 in

Z increment 0.08 in

Step over 60 %

Finish allowance 0.1 in

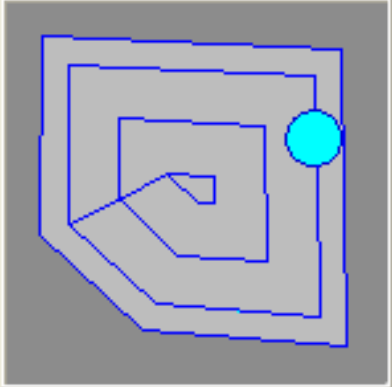
Feed rate 20 inches/min

Ramp angle 0 degrees

Plunge rate 8 inches/min

Spindle speed 10000 rpm

Coolant
☐ Flood
☐ Mist
☐ Climb cut



Cut sequence:
 1 cut of 0.0787 in
 and 1 cut of 0.0213 in

OK Cancel Help

Note: All units based on the selection in the 'Options/Units' menu.

Pocket method

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

Note: If 'Zigzag pocket' is selected a new menu item will appear as shown below.

Pocket method Zigzag pocket

Fill angle 180 degrees

Zigzag pocket

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

Layer

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

Layer OUTLINE

Tool TEXT
OUTLINE

Tool

Select the correct tool using the down menu shown below.

Tool	T3: Mill/Router, 3 mm diameter ▼
Start depth	T3: Mill/Router, 3 mm diameter T5: Mill/Router, 0.375 in diameter

Edit

Calls up the 'Tool Edit' dialogue 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.

Z increment

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.

Finish allowance

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

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 process will turn yellow to warn you that this is happening.

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).

Coolant

Select the coolant type from the available choices.

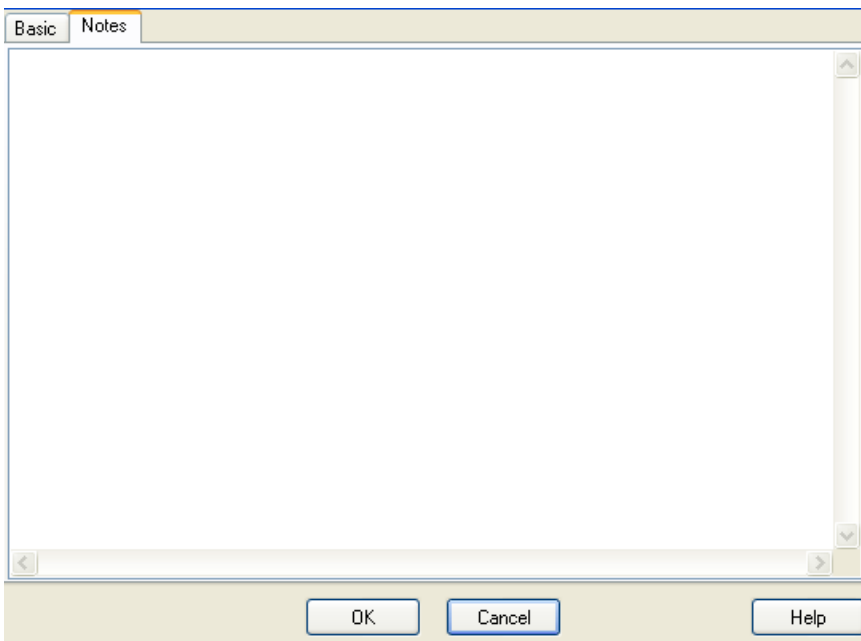
Note: These items only need to be checked if your coolant pump is controlled via software.

Climb cut

Check this box to select 'climb' cutting.

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

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

Notes**Notes**

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

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

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

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

New plasma cut

Basic Cut path Notes

Layer

Contour method

Tool

Feed rate

Loop sharp corners ☐

Loop size

Lead in

☒ None

☐ Arc

☐ Tangent

☐ Perpendicular

Size

Lead Out

☒ None

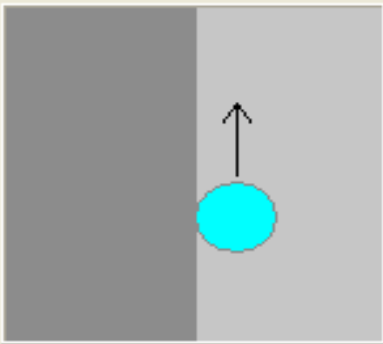
☐ Arc

☐ Tangent

☐ Perpendicular

Size

Use Code snippet



Note: All units based on the selection in the 'Options/Units' menu.

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

Layer

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

Layer

Tool

Contour method

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

Contour method

Layer

Tool

Note: 'Inside offset' will offset the cutter path to the 'inside' of the contour by half the diameter of the kerf 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 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' on the 'Cut path' tab.

Tool

Select the correct tool using the down menu shown below.

Tool	T1: Plasma, 0.04 in kerf
Feed rate	T1: Plasma, 0.04 in kerf T2: Plasma, 0.08 in kerf

Edit

Calls up the 'Plasma Edit' dialogue box.

Feed rate

Enter the required feed rate here.

Loop sharp corners

When plasma/flame/water jet 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 plasma job on a square and you will see how this works.

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.

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.

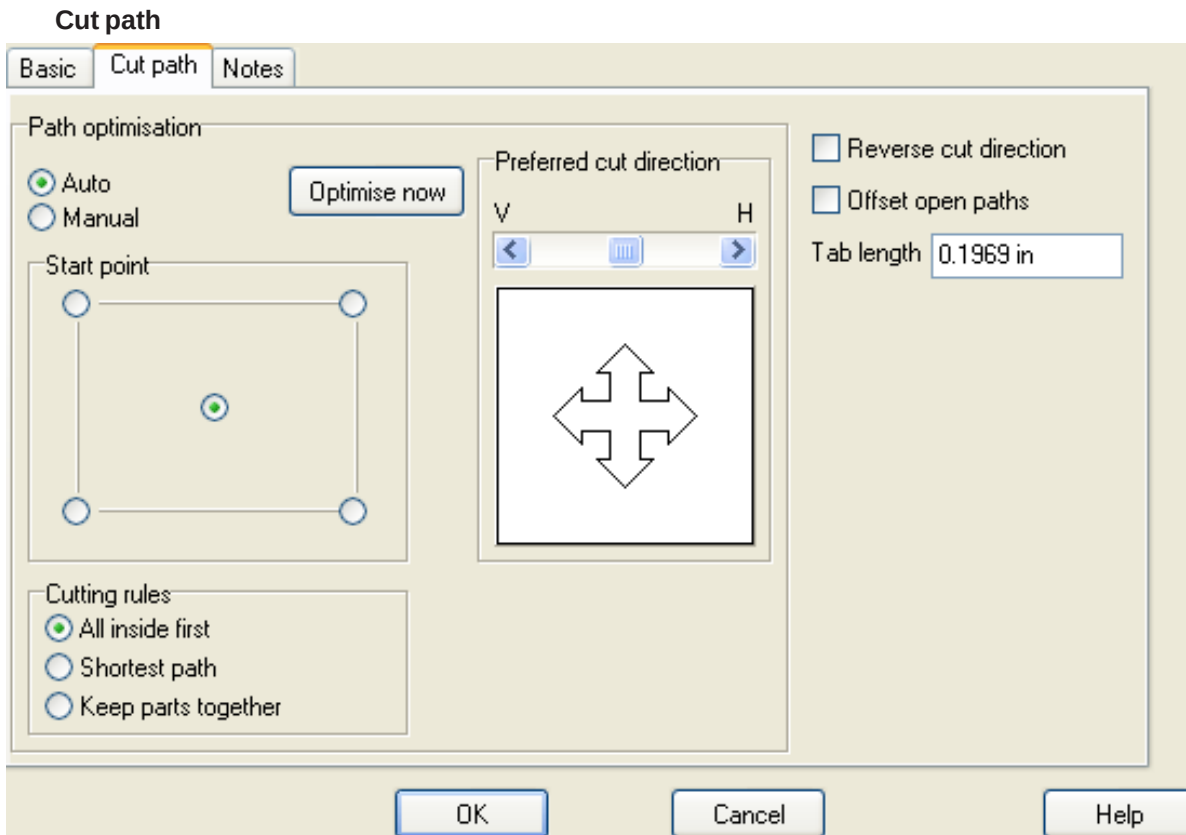
Note: Size refers to the length of the lead out.

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.

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



Auto/manual

Select auto or manual path optimization 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 point

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.

Preferred cut direction

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

Reverse cut direction

Check this box to reverse the cut direction.

Offset open paths

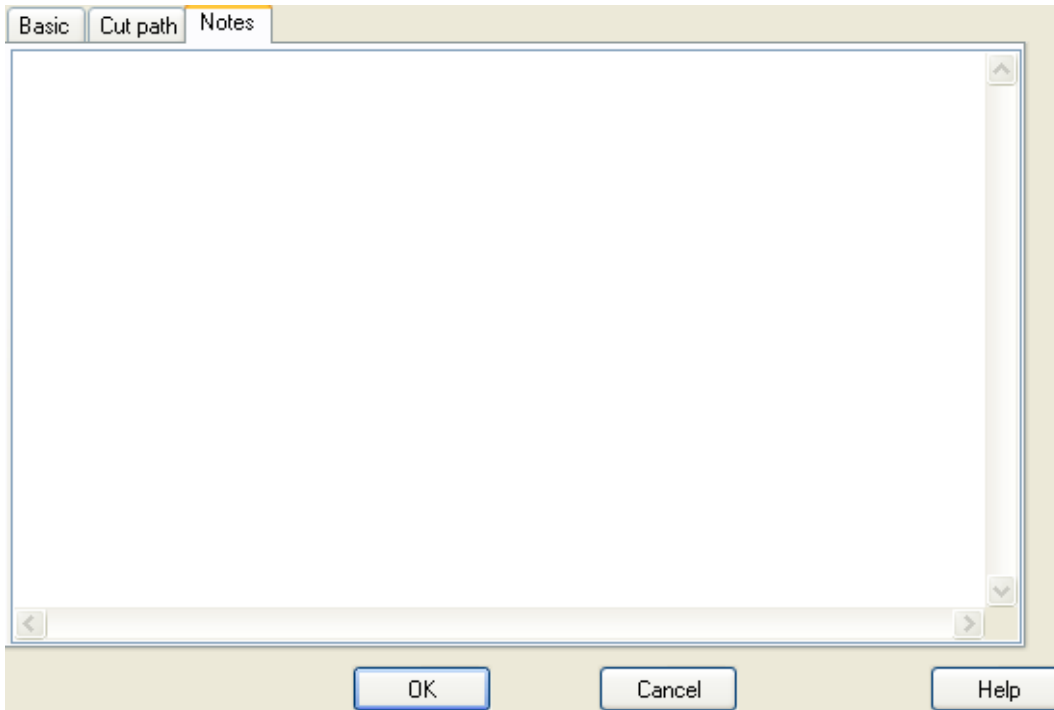
Open paths are always offset to the right of the start point. That is, if you were standing at the start point looking down the line the offset would be to your right. To reverse the side of the offset, move the start point to the other end of the line. Inside/outside offset make no difference as a line 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.

Tab 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.

Notes



Notes

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

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

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

New drilling process

Basic Cut path Notes

Layer **OUTLINE** ▼

Tool **T1: Drill, 0.25 in diameter** ▼ **Edit...**

Start depth **0 in**

Finish Depth **1 in**

Peck depth **0.2 in**

Peck retract **0.2 in**

Plunge Rate **0.1 inches/min**

Min hole size **0.2375 in**

Max hole size **0.2625 in**

Spindle speed **1000 rpm**

Coolant
☐ Flood
☐ Mist

Cut sequence:
5 cuts of 0.2 in

OK **Cancel** **Help**

Note: All units based on the selection in the 'Options/Units' menu.

Cut path tab

Click on the 'Cut path' tab to access more options.

Notes tab

Click on the 'Notes' tab to access more options.

Layer

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

Layer **OUTLINE** ▼

Tool **TEXT**
OUTLINE

Tool

Select the correct tool using the down menu shown below.

Layer **OUTLINE** ▼

Tool **T1: Drill, 0.25 in diameter** ▼

Start depth **T1: Drill, 0.25 in diameter**
T2: Drill, 0.375 in diameter

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

Edit

Calls up the 'Edit drill' dialogue 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.

Finish depth

Enter the finish 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 peck 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.

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.

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.

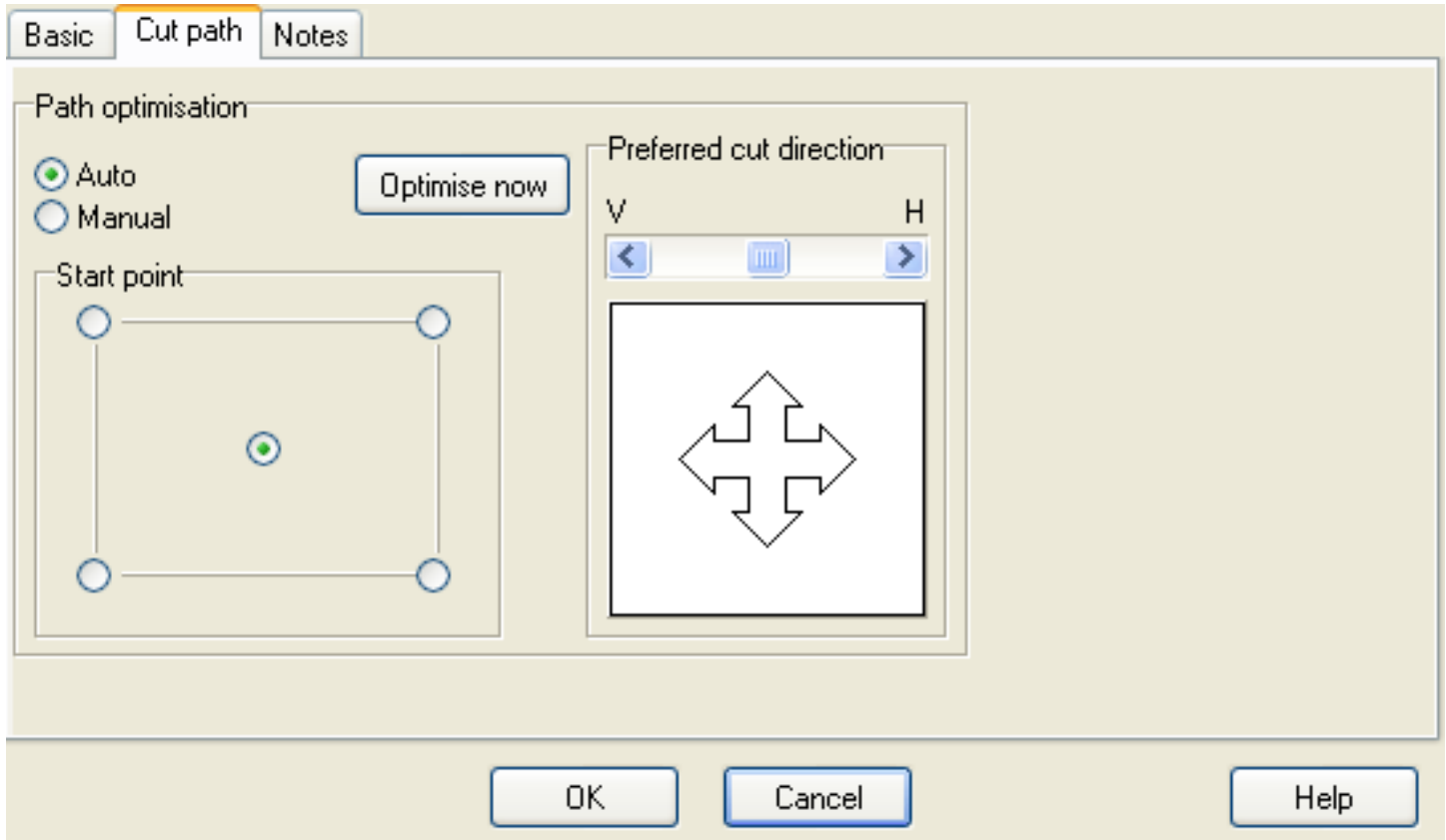
Note: These items only need to be checked if your coolant pump is controlled via software.

Special case

The drilling process can be used with a plasma cutter to cut holes at the required locations. When you select a plasma cutter as a tool in the drilling process dialogue box most of the options disappear as they are not relevant. The process 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.

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

Cut path



Auto/manual

Select auto or manual path optimization 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

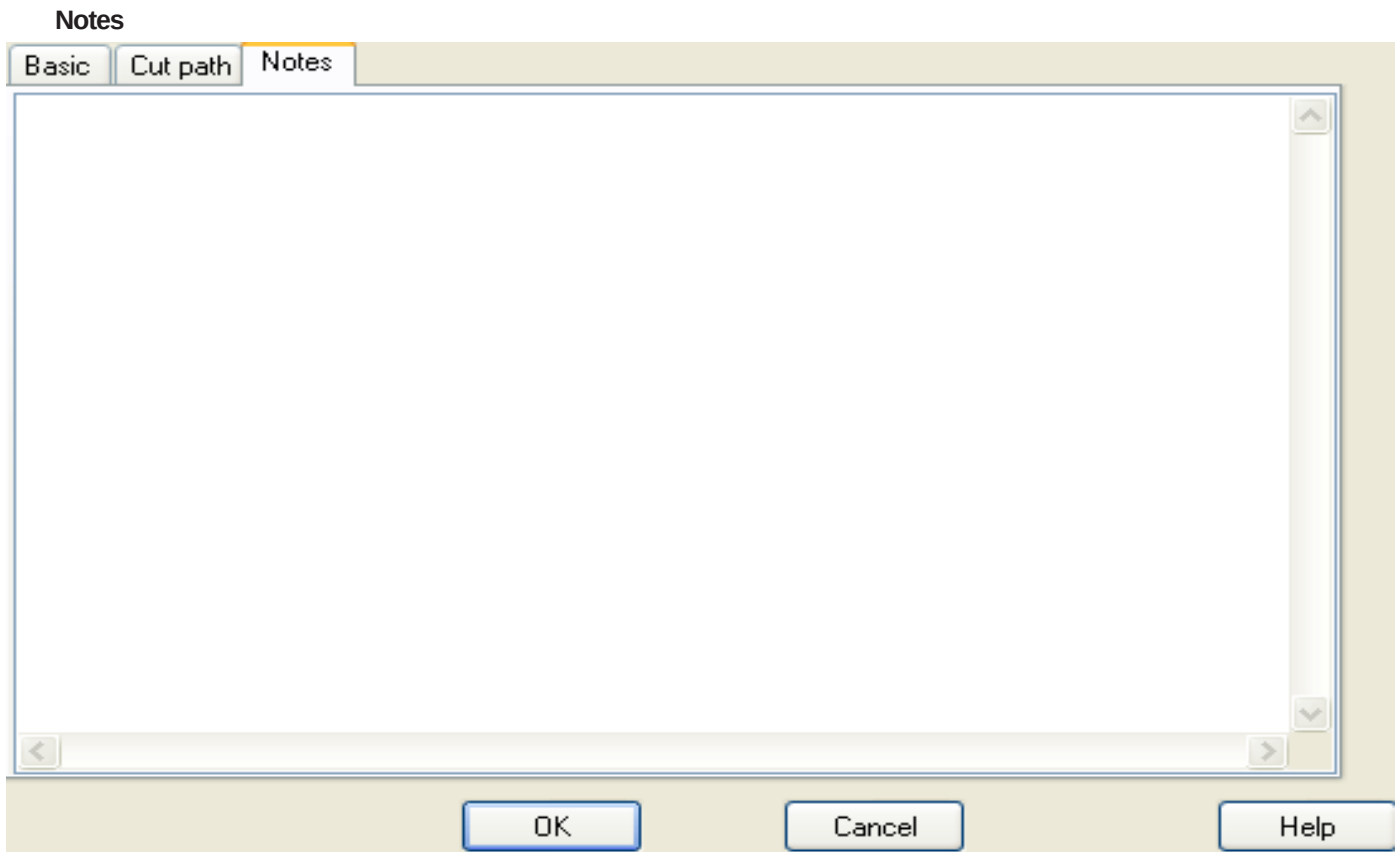
This button optimizes the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

Start point

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.

Preferred cut direction

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

**Notes**

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

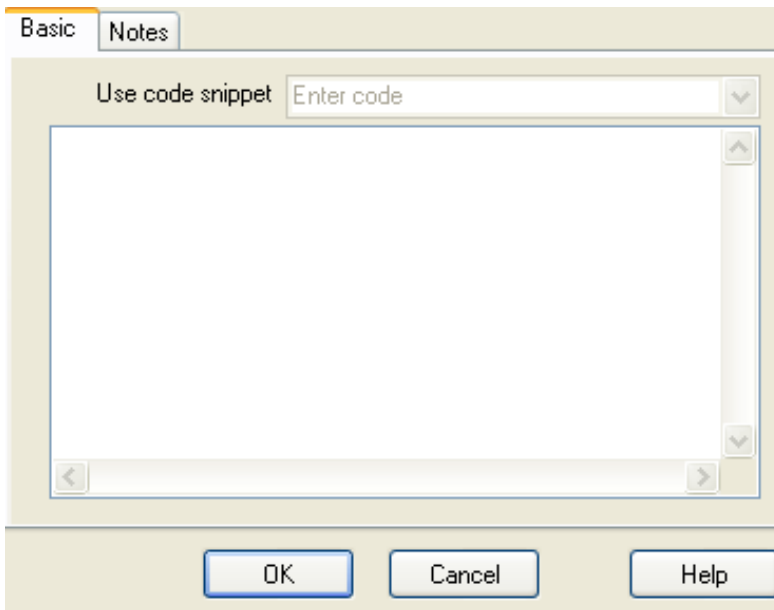
Note: If the machine understands 'comments' then these notes will also appear in the g-code as a comment.

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

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

Edit G-code

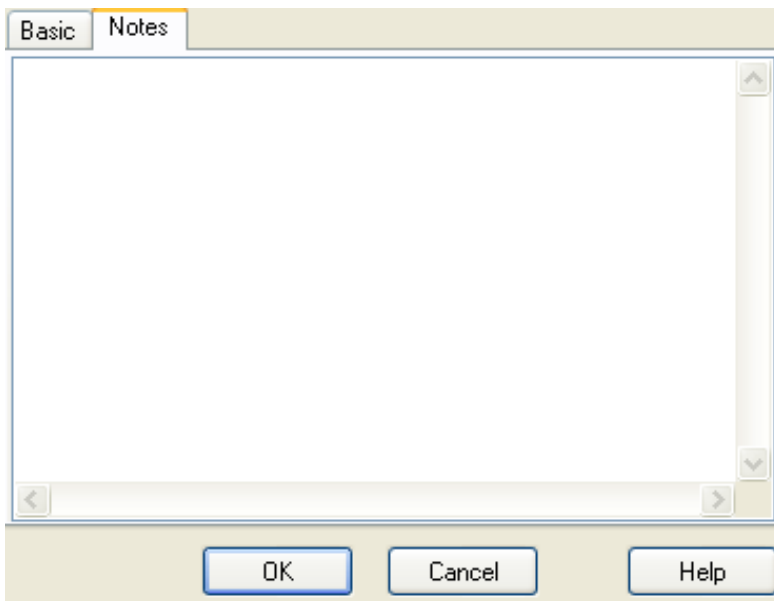


The 'Use code snippet' drop down menu will show any previously written snippets, simply select the one you want to use.

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 process.

Edit G-code Notes



Notes

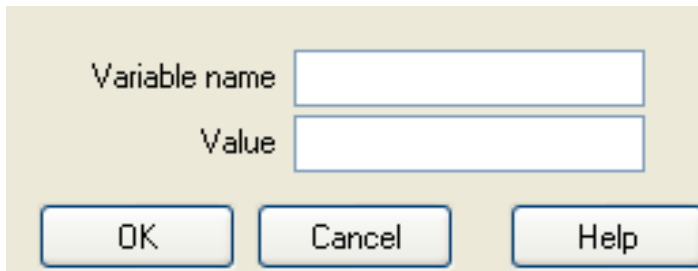
Use this area to store any notes you may wish to make regarding the process.

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

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

Set post variable



A dialog box titled 'Set post variable' with a light beige background. It contains two text input fields: 'Variable name' and 'Value'. Below the fields are three buttons: 'OK', 'Cancel', and 'Help'.

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.

Value

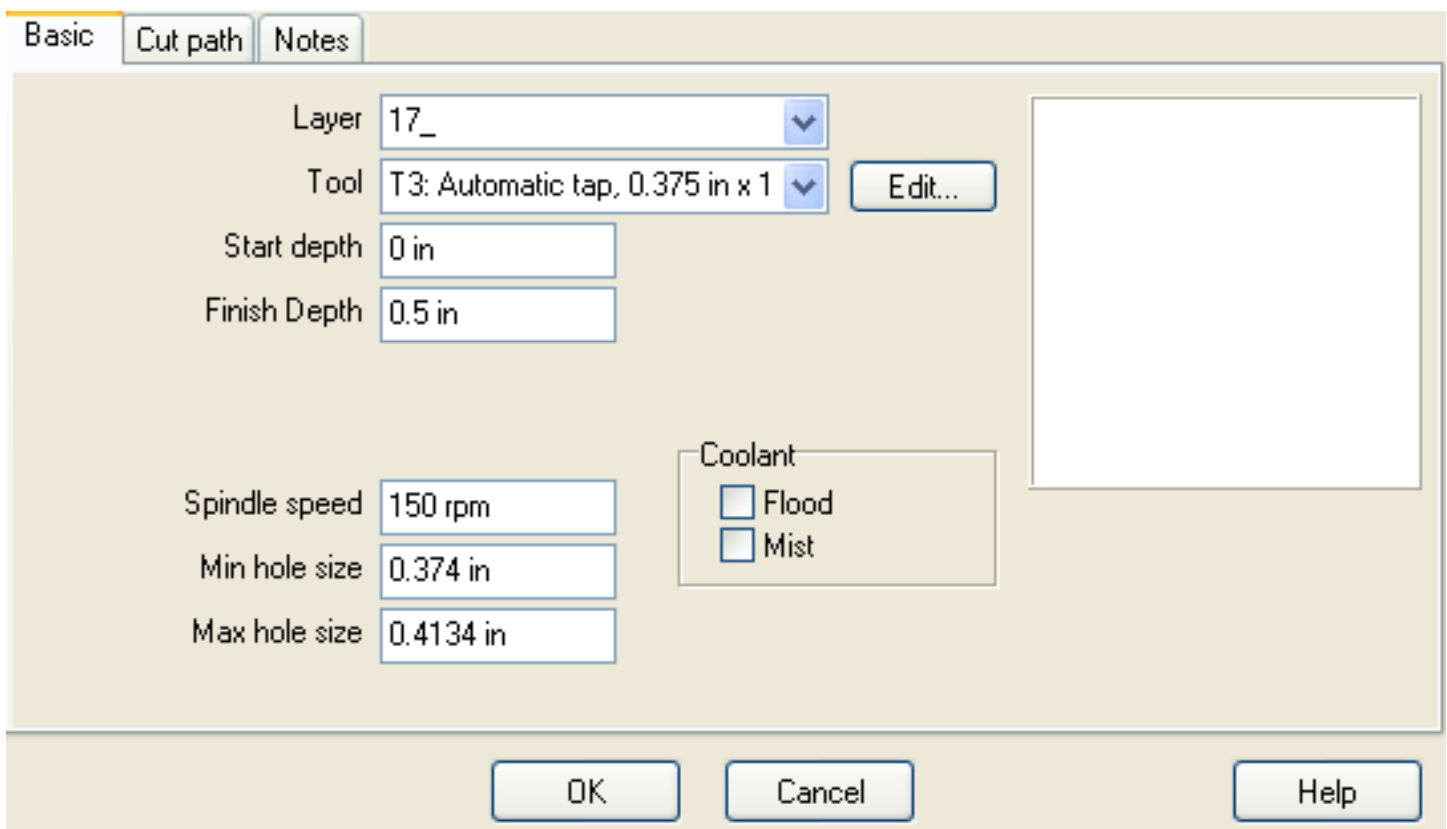
Value is the value that will be applied to that variable. This is always metric. It does not take the user's preferred units into account.

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

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

New tapping process



A dialog box titled 'New tapping process' with a light beige background. It has three tabs: 'Basic' (selected), 'Cut path', and 'Notes'. The 'Basic' tab contains several input fields: 'Layer' (dropdown menu showing '17_'), 'Tool' (dropdown menu showing 'T3: Automatic tap, 0.375 in x 1' with an 'Edit...' button), 'Start depth' (text field '0 in'), 'Finish Depth' (text field '0.5 in'), 'Spindle speed' (text field '150 rpm'), 'Min hole size' (text field '0.374 in'), and 'Max hole size' (text field '0.4134 in'). There is also a 'Coolant' section with two checkboxes: 'Flood' and 'Mist'. A large empty rectangular area is on the right side. At the bottom are three buttons: 'OK', 'Cancel', and 'Help'.

Note: All units based on the selection in the 'Options/Units' menu.

Layer

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



Two dropdown menus. The first is labeled 'Layer' and shows 'OUTLINE' selected. The second is labeled 'Tool' and shows 'TEXT' and 'OUTLINE' as options, with 'OUTLINE' highlighted.

Tool

Select the correct tool using the down menu shown below.

Tool	Automatic tap, 0.375 in x 10 TPI
Start depth	T3: Automatic tap, 0.375 in x 10 TPI T5: Rigid tap, 0.375 in x 16 TPI

Edit

Calls up the 'Edit tap' dialogue 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.

Finish depth

Enter the finish 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

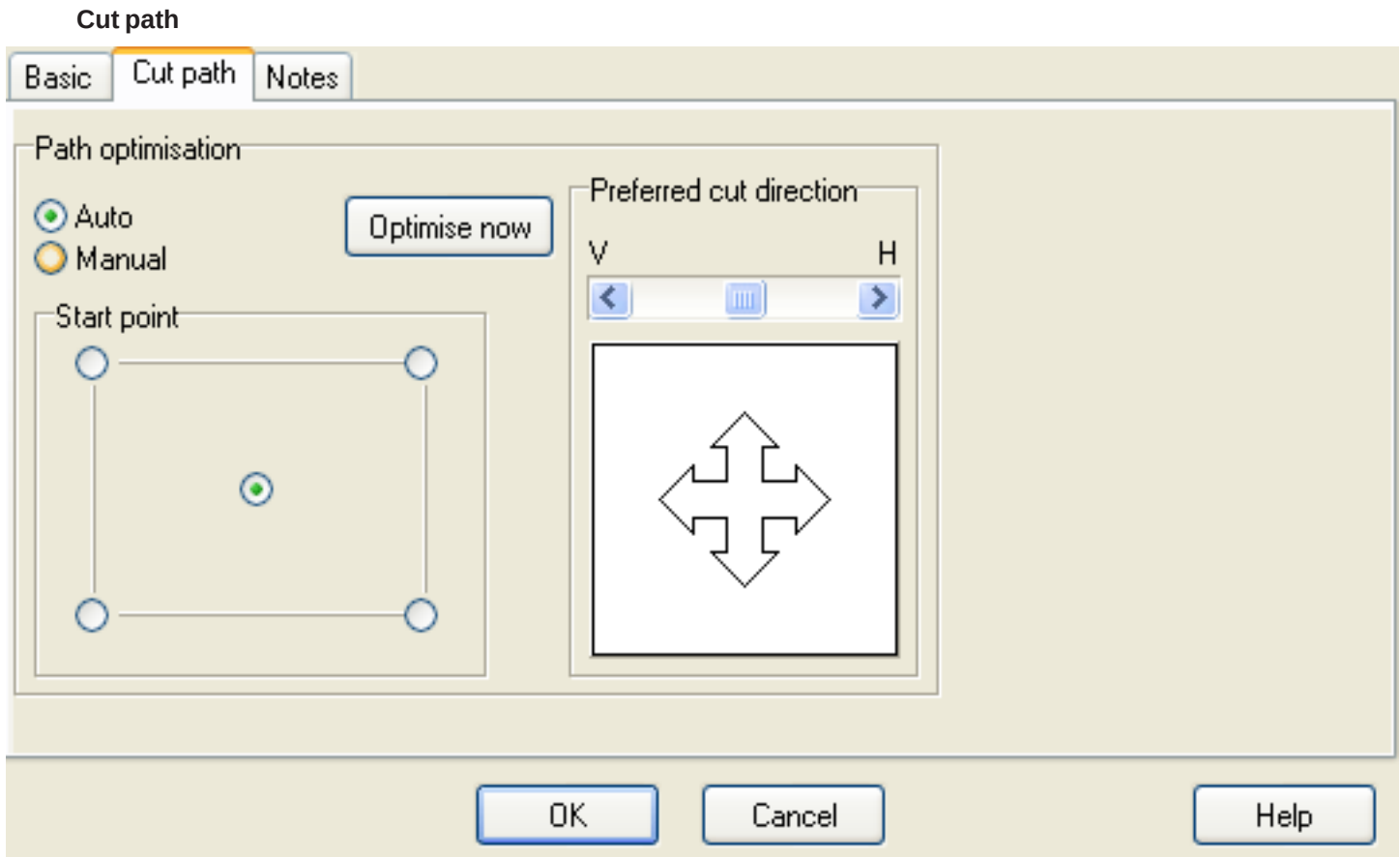
Select the coolant type from the available choices.

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

Help

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

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



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 point

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.

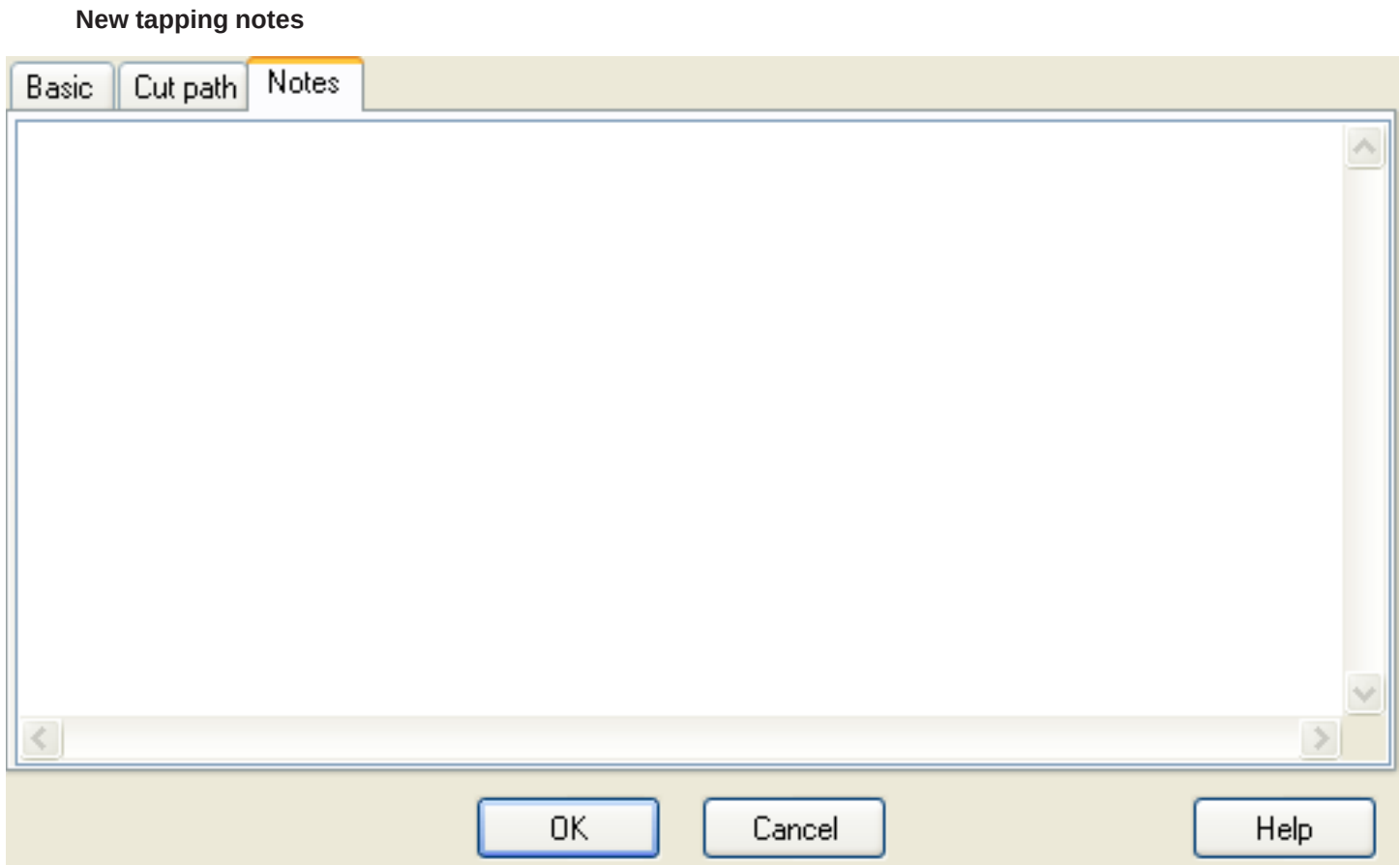
Preferred cut direction

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

Help

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

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



Notes

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

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.

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

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/Select post processor' menu. This option can also be activated by clicking on the  button on the toolbar. Enter a name for the file and a suitable location on your hard drive and then click <OK>.

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).

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

Open drawing

See Page 11 of this manual for 'Open drawing' options.

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).

Process files

Save process

Opens a pop-up window allowing you to save a process for future use as a 'process' file (.process).

Note: This option is 'greyed out' until you generate your first process.

Load process

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

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.

DNC Tool



Send data

Sends the most recently post-processed data to your machine via a RS232 interface.

Open file icon

Use this button to search for and open a G-code file. Normally the file is opened automatically when you run the post processor but you can also manually open a file to send using the DNC tool.

Configure

Opens the menu shown below.

Note: Check in your equipment manual for the correct communication port settings.

Serial port/Run external command

You can either send the file to a serial port or pass it on to an external program. Use the radio buttons to make your selection.

Command settings

Command

You can add some extra flags to the command:

%F = full file name with path

%f = file name with path but without extension

%e = file extension

%p = file path

For instance the command:

```
c:\windows\notepad.exe %F
```

would open the file in Notepad.

Run in a DOS box

If the 'Run in a dos box' checkbox is checked then the command is run in a DOS command window.

Keep open when command finishes

By checking this box the DOS command window will remain open once it has completed its task.

Wait for command to finish

By checking this box the DNC tool will not do anything else until the external program has exited.

Port settings

Comm port

Select the correct communications port for your application.

Baud rate

Select the correct baud rate for your application.

Parity

Select the correct parity for your application.

Byte size

Select the correct byte size for your application.

Stop bits

Select the correct stop bits for your application.

Flow control

Select the correct flow control for your application.

Auto send after post processing

If this option is enabled the g-code is sent immediately when the post processor is run.

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.

Colour legend

Each colour represents an item displayed in the 'view' panel.

To change the default colours double-click on either the name or the coloured box and select a new colour.

To minimize the palette click on the arrow button.

To close the palette click on the 'X' button.

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.

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 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 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.

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/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.

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/Reverse mouse wheel' option.

Options Menu

Tool change

Note: All units based on the selection in the 'Options/Units' menu.

X position ☐ Use X position
Y position ☐ Use Y position
Z position ☐ Use Z position
☐ No Z move prior to toolchange

Run code before toolchange

Run code after toolchange

OK Cancel Help

The 'Tool change' dialogue box allows you to specify a tool change location.

Position boxes

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' dialogue box) to the location chosen as the tool change position, this can be a positive (+) or a negative (-) figure.

No Z move prior to toolchange

If this is checked SheetCam will not move the Z axis prior to the first toolchange. If it is not checked then SheetCam will move to the rapid clearance height before the toolchange.

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

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

Set start point size

Note: All units based on the selection in the 'Options/Units' menu.

Start point size

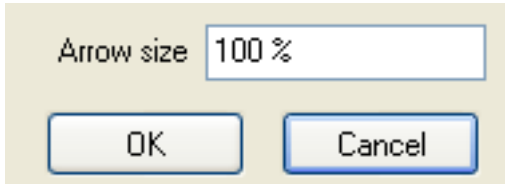
OK Cancel

This function controls the size of start point markers and 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.

Enter the required marker size and click <OK>.

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

Set direction arrow size



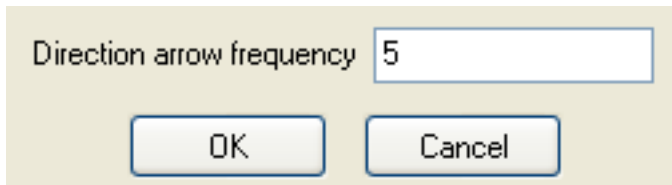
Note: All units based on the selection in the 'Options/Units' menu.

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.

Enter the required arrow size and click **<OK>**.

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.

Set arrow frequency



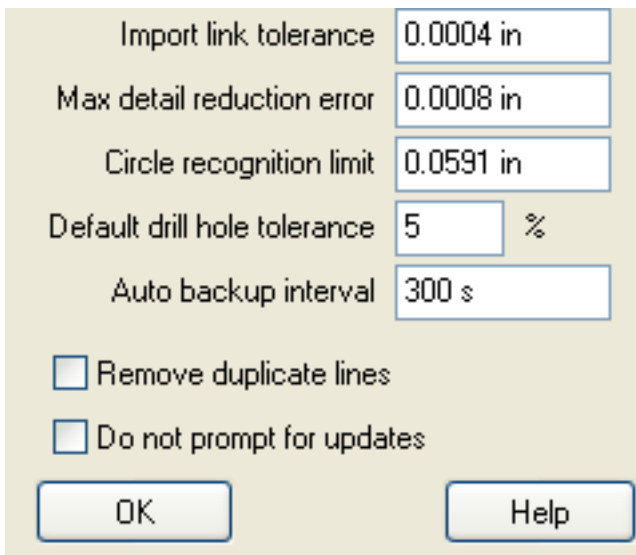
Note: This is used to control the frequency of the direction arrows shown on the cutter path display. For instance entering a figure of 5 would result in five vectors between each arrow location while entering 8 would effectively space the arrows further apart.

Enter the required arrow frequency and click **<OK>**.

System parameters

This dialogue box is used to set up the 'System parameters' 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.

Default drill hole tolerance

When you create a drill process 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.

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.

Do not prompt for updates

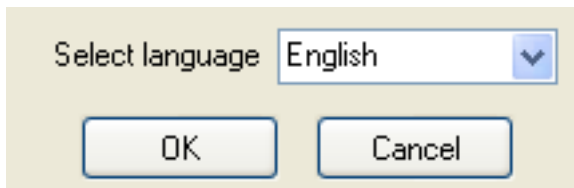
The first time SheetCam is started each day it checks for updates (if an internet connection is available). If there is a new update available it asks if you want to install it. If this box is checked you will not be prompted and updates will not be installed.

Help

Clicking on this button will open the relevant help section for that screen. This applies to all 'Help' buttons.

After making any changes click **<OK>** to accept and close the box.

Select language



This dialogue box is used to set up the 'System language' of choice.

Select from the available options in the drop down menu.

After making any changes click **<OK>** to accept and close the box.

Note: You must restart SheetCam for the changes to take effect.

Post output to last drawing folder

Checking this item will save any output from SheetCam to the last opened drawing folder on your hard drive. If this is unchecked SheetCam remembers the last folder you used.

Colours

'Hovering' over or clicking this menu item will open the following pop-out dialogue box.

Colour scheme

Clicking on either 'Colour scheme: White' or 'Colour scheme: Black' will change the background colour of the main view panel.

Other items in list

Clicking on any of the other items in the list will open up the standard Windows 'colour picker' allowing you to select a new colour for the item clicked on.

Note: The same changes can be made via the 'View/Colour legend' menu item.

Reopen last job on startup

Checking this menu item will force SheetCam to open the last job saved when it starts.

Reverse mouse wheel

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

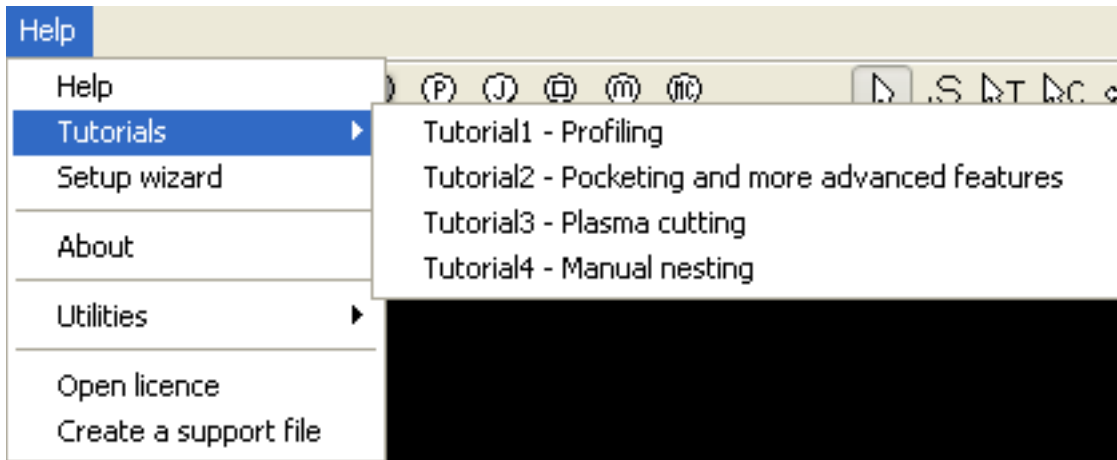
Help Menu

Help

Calls up this help file.

Tutorials

'Hovering' over or clicking this menu item will open the following pop-out dialogue box.



Select a tutorial from the available choices.

Setup wizard

Welcome to the SheetCam setup wizard.

The following pages will help optimise SheetCam for your machining tasks.

If you change your mind at any time in the future you can run this wizard from the Help menu.

<<Back

Next>>

Click on **<Next>** and follow the on-screen directions.

About

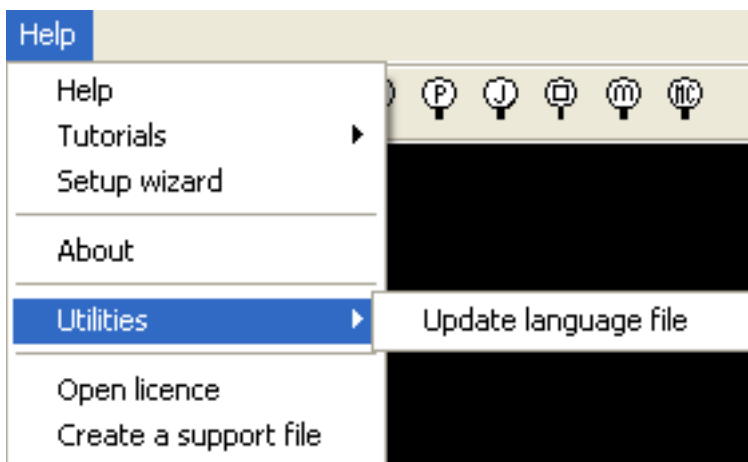


This screen gives information on the current SheetCam version and contact methods.

Clicking on any links will connect to the internet (if available).

Utilities

'Hovering' over or clicking this menu item will open the following pop-out dialogue box.



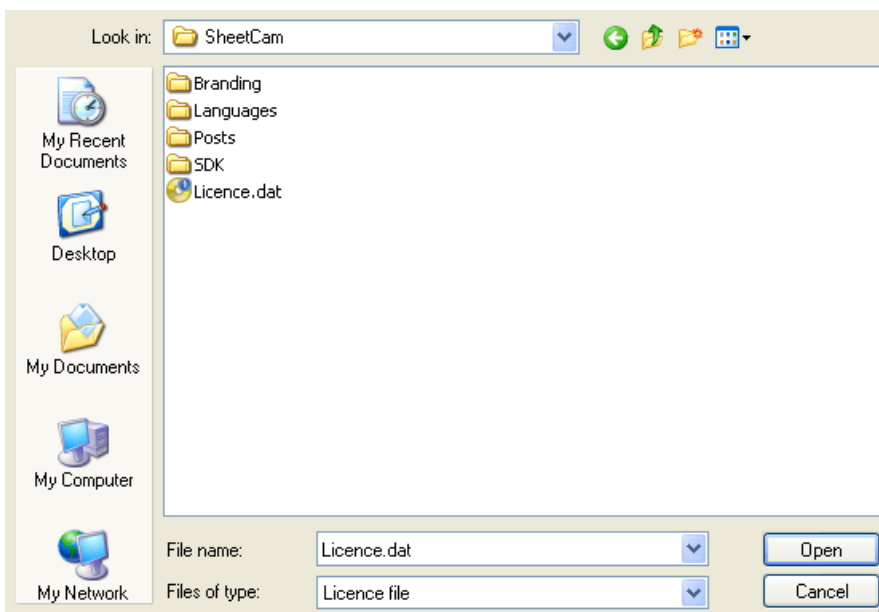
Click on **<Update language file>** to update.

Note: This only works on non-English versions.

Open licence

Locate the file **<Licence.dat>** and click **<Open>**. This will register your version of SheetCam.

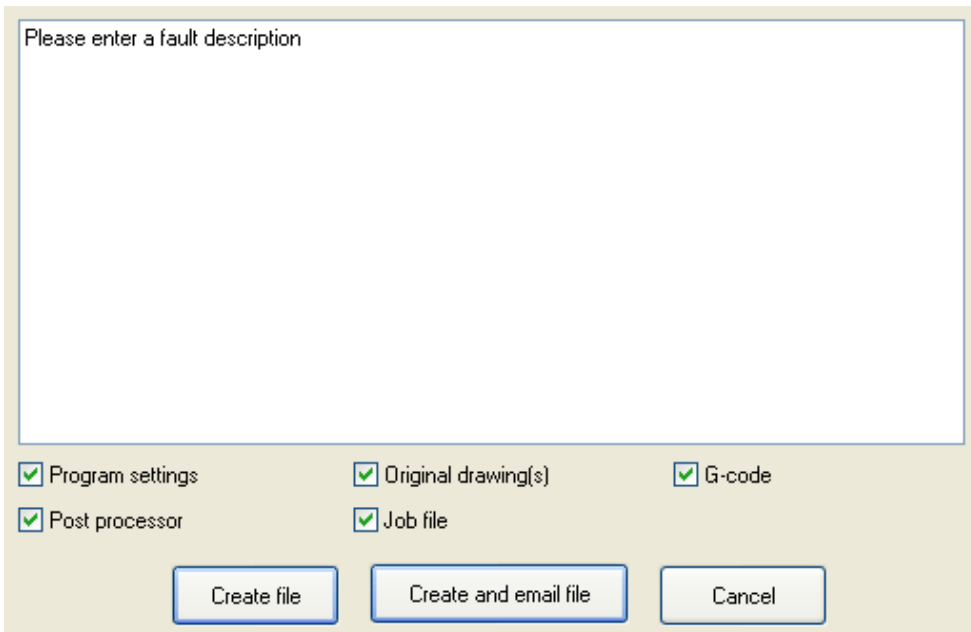
Note: This requires a restart of SheetCam to complete the registration.



Create a support file

This will create a support file that can be sent to 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 text box.

Use the check boxes to select which items are to be included with the support file.

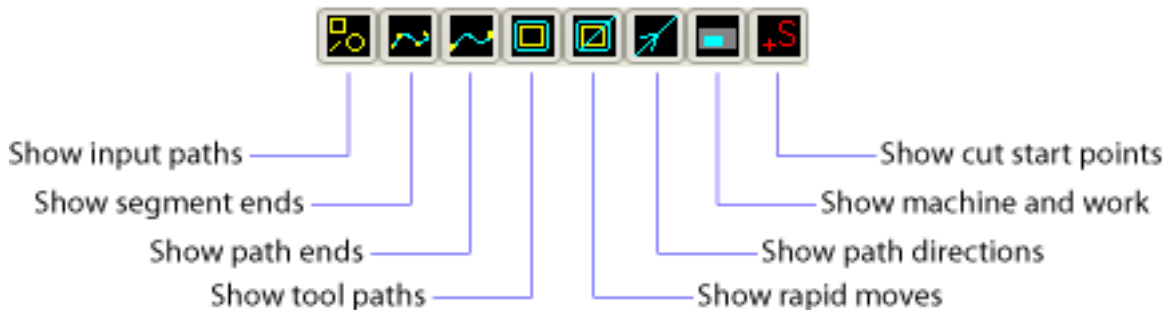
Click on **<Create file>** to save the support file to your hard drive as a zip file. Enter a suitable name/location in the appropriate boxes and click **<OK>**.

To create and then email the zipped support file (if your machine has internet access) click on **<Create and email file>**. The support file will be created as a zip file and your default email program will open with a new mail page already addressed and with the zip file attached

Toolbars

There are a number of toolbars within SheetCam and the following section details their various functions.

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.

Run post processor toolbar buttons



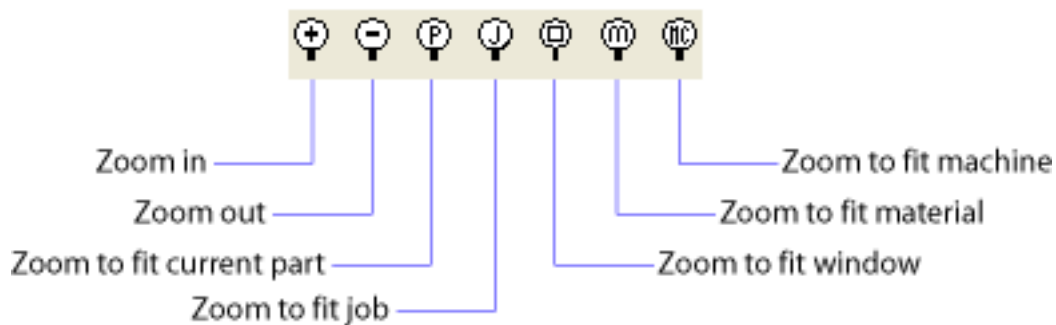
Run post processor

Runs the post processor previously chosen from the 'Options/Select post processor' menu.

Estimate cut time

Clicking on this button will estimate the time required to complete the cut.

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/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.

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/Reverse mouse wheel' option.

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.

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/Reverse mouse wheel' option.

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.

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/Reverse mouse wheel' option.

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.

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/Reverse mouse wheel' option.

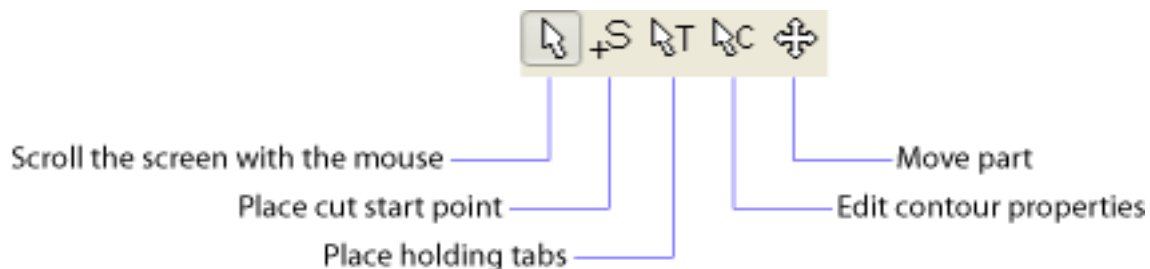
Zoom to fit machine

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.

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/Reverse mouse wheel' option.

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
Zoom in (PgUp)
Zoom out (PgDn)
Zoom to fit job
Zoom to fit material
Zoom to fit machine

Measure

To use the dimension 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 to the cursor. If you click on another point the dimension start will move to that point.

You can cancel dimensioning 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.

Place cut start point

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.

Right click options

'Right clicking' while using the 'Place cut start point' selection tool will bring up the following menu screen.

Properties
Clear all
Zoom in (PgUp)
Zoom out (PgDn)
Zoom to fit job
Zoom to fit material
Zoom to fit machine

Properties

Lead in ☒ None ☐ Arc ☐ Tangent ☐ Perpendicular Size 0.2 in	Lead Out ☒ None ☐ Arc ☐ Tangent ☐ Perpendicular Size 0 in
Use Code snippet None	
Order 2 /22	
☒ Use process settings	
OK Help	

Use code snippet

If you want to apply a previously written code snippet (see 'Tools/New code snippet') at the selected point use the drop down menu to make your selection.

Order

If you wish to change the cutting order of the parts enter the new value for the selected start point in the dialogue box.

Use process settings

If this option is selected (the default) then the start point uses the lead in/lead out settings defined in the process. If it is not selected then you can override the process settings. This is useful if the start point is in a confined space and the default settings are too big for it to fit.

Clear all

Clicking on this menu item will clear all the cut start points.

Zoom controls

Clicking on any of these items will zoom the view as required.

Place holding tab

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 clicking on the part in the required place will insert the tab. Repeat for each required tab. The size of the tab is defined in the 'Processes/New contour/Cut path' dialogue box.

Right click options

Right clicking' while the mouse pointer is over a placed tab will bring up the following menu screen.

Delete
Copy from ▶
Clear all
Zoom in (PgUp)
Zoom out (PgDn)
Zoom to fit job
Zoom to fit material
Zoom to fit machine

Delete

Clicking on this function deletes the selected tab.

Clear all

Clicking on this menu item will clear all the placed tabs.

Zoom controls

Clicking on any of these items will zoom the view as required.

Edit contour properties

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.

Right click options

'Right clicking' while using the 'Edit Contour' selection tool will bring up the following menu screen.

Properties
Move to layer ▶
Copy to layer ▶
Zoom in (PgUp)
Zoom out (PgDn)
Zoom to fit job
Zoom to fit material
Zoom to fit machine

Properties

Clicking this function opens a dialogue box that shows how many contours you have selected and which layers they are on.

Move to layer

'Hovering' over or clicking this menu item will open a pop-out dialogue box. Chose either a 'New layer' or an existing layer. This will move the selected contour(s) to the chosen layer. Choosing 'New layer' will open an input window. Enter a new name for the layer and click **<OK>**.

Copy to layer

'Hovering' over or clicking this menu item will open a pop-out dialogue box. Chose 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 an input window. Enter a new name for the layer and click **<OK>**.

Zoom controls

Clicking on any of these items will zoom the view as required.

Move part

When clicked allows you to move the part in relation to the material/machine.

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 processes. 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 processes. 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.

Right click options

'Right clicking' while using the 'Move' selection tool will bring up the following menu screen.

Copy
Duplicate
Align
Array
Delete
Disable
Set bump increment
Set grid size
Zoom in (PgUp)
Zoom out (PgDn)
Zoom to fit job
Zoom to fit material
Zoom to fit machine

Copy

Click on this function to copy the part. A part consists of a drawing with its associated processes. Copy will make a complete copy of the drawing and processes. The copy will be completely independent of the part it was copied from.

Duplicate

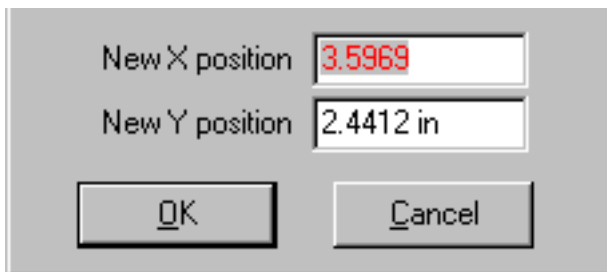
Click on this function to duplicate the part. Duplicate will perform a 'step and repeat' of the original drawing and processes. **Any changes** made to the original **will be reflected in the duplicate**.

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 dialogue box will appear.



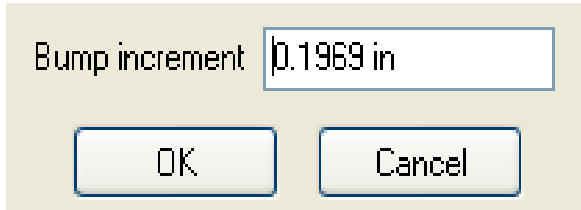
Enter the actual coordinates of this point and click <OK>. Now click on the second point on the part in the view window and enter the coordinates of that point and click <OK>. The drawing will be moved and rotated to suit 'the job'.

Delete

Clicking on this function deletes the selected item.

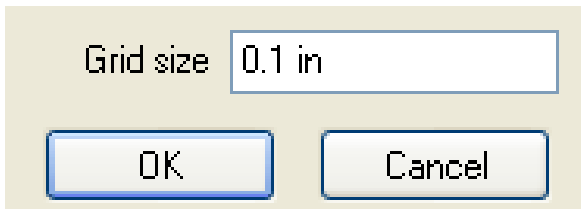
Set bump increment

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. Click on the function name to call up the dialogue box below.

A dialog box titled 'Set bump increment'. It has a label 'Bump increment' followed by a text input field containing '0.1969 in'. Below the input field are two buttons: 'OK' and 'Cancel'.

Set grid size

Click on the function name to call up the 'Set grid size' dialogue box below.

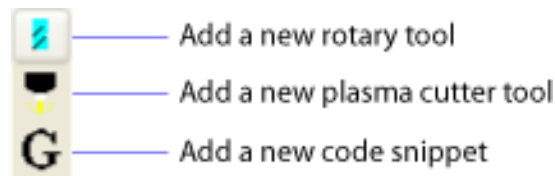
A dialog box titled 'Set grid size'. It has a label 'Grid size' followed by a text input field containing '0.1 in'. Below the input field are two buttons: 'OK' and 'Cancel'.

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. Enter the required value and click **<OK>** to accept.

Zoom controls

Clicking on any of these items will zoom the view as required.

Tools toolbar buttons



New mill button

Clicking on this button will open the 'New mill/router' set up window.

Note: This can also be accessed via the 'Tools/New mill/router' menu item.

New plasma button

Clicking on this button will open the 'New plasma cutter' set up window.

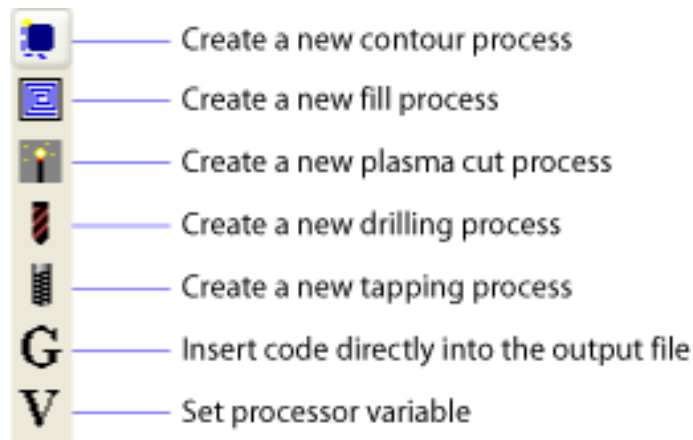
Note: This can also be accessed via the 'Tools/New plasma cutter' menu item.

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.

Processes 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 'Process/New contour' menu item.

Add new fill

Clicking on this button will open the 'New fill' set up window.

Note: This can also be accessed via the 'Process/New fill' menu item.

Add new plasma process

Clicking on this button will open the 'New plasma process' set up window.

Note: This can also be accessed via the 'Process/New plasma cut' menu item.

Add new drilling process

Clicking on this button will open the 'New drilling process' set up window.

Note: This can also be accessed via the 'Process/Drilling' menu item.

Add new tapping process

Clicking on this button will open the 'New tapping process' set up window.

Note: This can also be accessed via the 'Process/Tapping' menu item.

Edit G-code

Clicking on this button will open the 'Edit G-code' set up window.

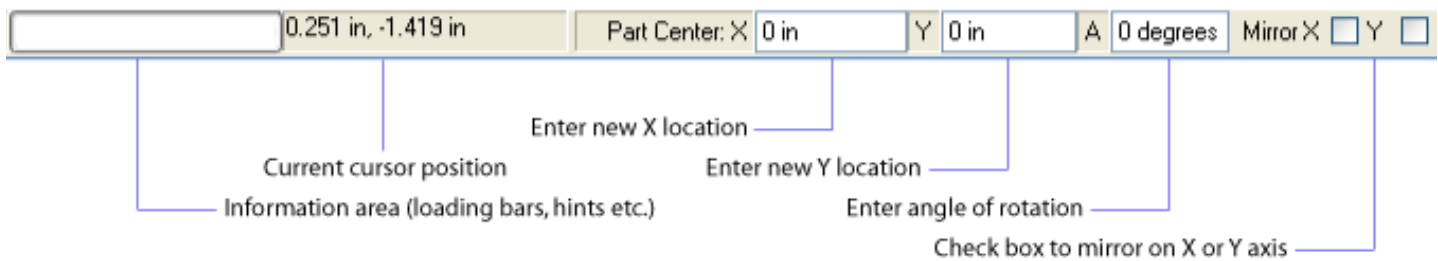
Note: This can also be accessed via the 'Process/Insert code' menu item.

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 'Process/Set post variable' menu item.

Lower right toolbar



Information area

Shows file loading bars, hints etc.

Part centre

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/Use lower left coordinates' is turned on.

Angle

To rotate the part enter the required angle.

Note: To rotate clockwise enter a **positive** (+) value, to rotate counter-clockwise 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.

Hints and Tips

Centre drilling

If you need to centre drill a series of holes prior to drilling open a new drill process 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/processes in a different way as shown below.

Copy: This will make a complete copy of the drawing and processes. 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 processes. **Any changes** made to the original **will be reflected in the duplicate**.

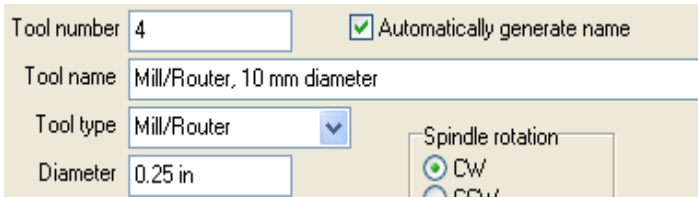
Depth of cut, peck depth etc.

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

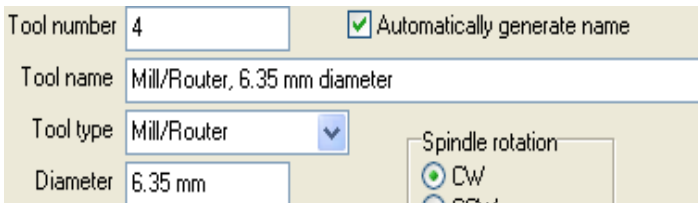
Whenever you enter a value into an edit box you can use any units you like. The units you use will be converted into your 'preferred units' as soon as you leave the edit box.

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.



The screenshot shows a tool setup dialog with the following fields: Tool number (4), Tool name (Mill/Router, 10 mm diameter), Tool type (Mill/Router), and Diameter (0.25 in). There is a checkbox for 'Automatically generate name' which is checked. A 'Spindle rotation' section is partially visible on the right.

As soon as you leave the edit box it will be converted to 6.35mm



The screenshot shows the same tool setup dialog, but the diameter field now displays '6.35 mm' after the user has left the edit box. The other fields remain the same.

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 process. 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' for more information.

Material parameters

The settings for material parameters can be accessed via the '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/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

The following tutorial pages relate to the tutorials provided within SheetCam which are accessible via the 'Help/Tutorials' menu item.

Profile tutorial

In this tutorial we will open a drawing and create the toolpaths to cut out the part. The first group of icons on the left hand side 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.



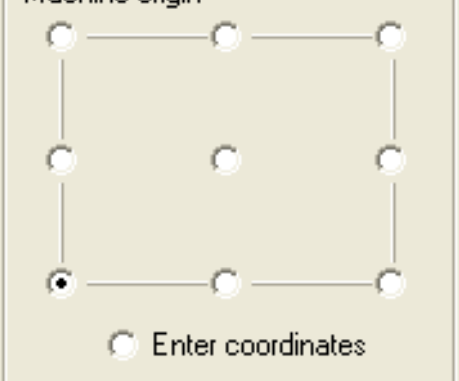
Show path ends. This highlights the start and end points of all open paths. In this case the line underneath the text is an open path because it has a start and an end point. The text consists of closed paths which have no fixed start and end points.

For the purposes of this tutorial we are working in millimetres so go to 'Options/Units' and 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

X coordinate of machine origin	0 mm
Y coordinate of machine origin	0 mm
Working envelope X size	558.8 mm
Working envelope Y size	508 mm
X coordinate of table bottom left	-25.4 mm
Y coordinate of table bottom left	-50.8 mm
Table X size	609.6 mm
Table Y size	2 feet
Max clearance between chuck and table	480 mm

Machine origin



☐ Enter coordinates

Make sure the Show machine and material button  is down. This displays the machine and material.

Now click on 'Options/Machine'. The display will zoom to fit the machine.

Let's assume our machine table is 2 feet square with a useable area of 22" by 20". The bottom left hand corner of the useable area is at coordinate 0,0.

In the 'Machine 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 useable area so enter 22" in the X size and 20" in the Y size (include the " to tell SheetCam these measurements are in inches). Note that as soon as you exit each edit box the value is converted to millimetres for you. Notice that the main drawing display changes as you change the dimensions so you get an idea what the table looks like.

The table is larger than the actual useable area so we need to enter the size and position of the table. Enter -1" and -2" for the X and Y coordinates of the table bottom left and 2 feet for the table X and Y sizes.

We won't be using the max clearance between the chuck and table so leave it as it is. Click on **<OK>**.

Thickness of material	6 mm
X Coordinate of bottom left corner	50 mm
Y Coordinate of bottom left corner	50 mm
X size	300 mm
Y size	200 mm
Rapid clearance	4 mm
Height of bottom of material above table	0.25in
OK	

Setting up the material

Now click on 'Options/Material'. The display will zoom and rotate to fit the material.

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 in the first 5 boxes. The main display will resize 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.

Now click on **<OK>**.

Defining a tool

Now we have set up the machine we need to define the tool that we will use to cut out the part.

For this job we will use a 30mm long 3mm 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

Click on the new tool icon  to bring up the tool dialogue box.

Tool number ☒ Automatically generate name

Tool name

Tool type

Diameter

Flute length

Tool projection

Tool length offset

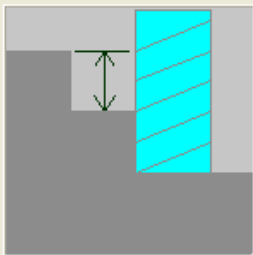
Z increment

Feed rate

Plunge rate

Ramp angle

Spindle speed



Enter 3 in the tool number and select 'Mill/Router' as the tool type. Now enter the diameter and flute length. The tool length offset is useful if you are using multiple tool holders or are using an autolock type chuck. You can enter the tool projection here so 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 (zero) here.

In 'Z increment' enter 2 as this is the maximum cut depth per pass. We will plunge the tool in at a slower rate so enter 200 in the plunge rate. Leave the ramp angle at 0 (zero) degrees. Tool projection is the amount the tool protrudes out of the chuck. In this case we will assume that it is 15mm.

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 material' button  to show the drawing. Now press the 'Create a new contour process' button  to create a new contour process.

Basic | Cut start | Cut path

Contour method

Layer

Tool

Cut depth

Z increment

Finish allowance

Feed rate

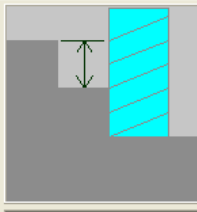
Plunge Rate

Spindle speed

Spindle direction
☒ CW
☐ CCW
☐ Off

Coolant
☐ Flood
☐ Mist

Cut sequence:
3 cuts of 2 mm
and 1 cut of 0.25 mm



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 3 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 so enter 6.25 in the 'Cut depth' box.

The box on the bottom right shows how many passes the cutter will make to cut through the material.

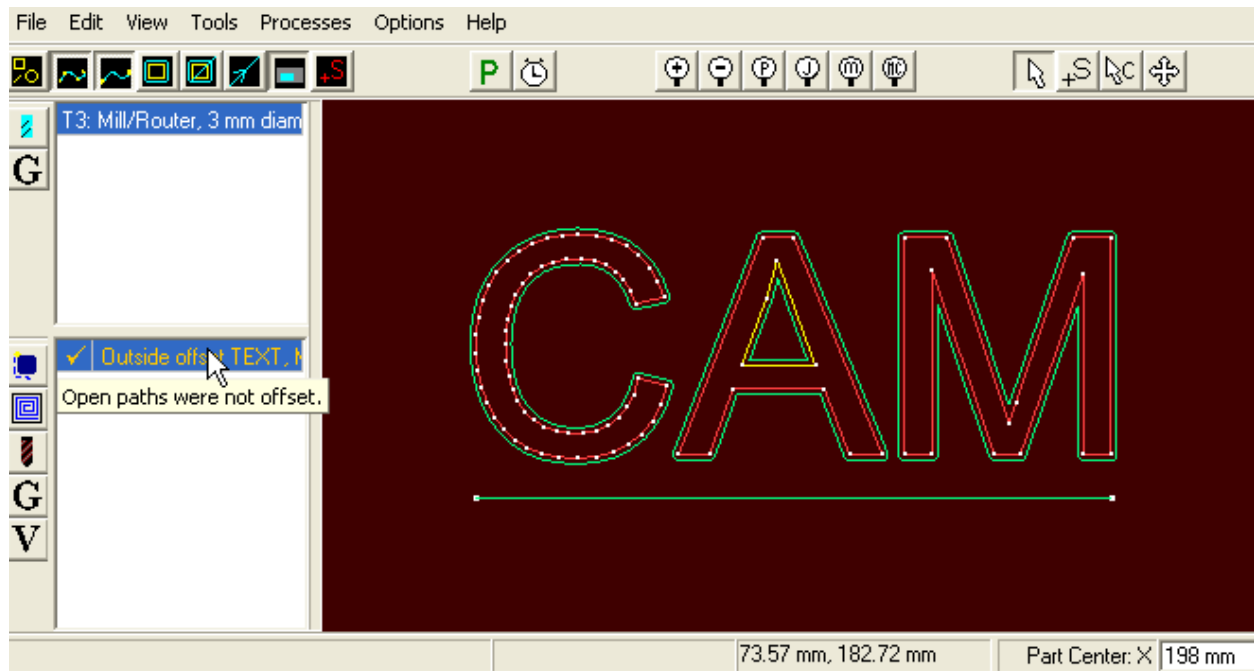
If you hover the mouse over the 'Z increment' box a box will pop telling you that the cutter will make 3 passes of 2mm and one of 0.25mm. This is not ideal for this job so change the Z increment to 1.8mm. The cutter will now make 3 passes of 1.8mm, leaving 0.85mm of material for the last cut.

Now click on **<OK>** to save the changes.

Generate the tool paths.

The toolpaths 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 process has turned yellow to warn you that there is a problem. If you hover the mouse over the process a message will pop up telling you what the problem is.



Experiment with the 'Show toolpaths' , 'Show rapid moves'  and 'Show path directions'  buttons to see what effect they have on the display.

You can drag the display around with the mouse and zoom in or out by using the mouse wheel or by holding down the right mouse button and moving the mouse up and down. You can rotate the display by holding the shift key then dragging with the mouse.

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 'Contour properties'  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.

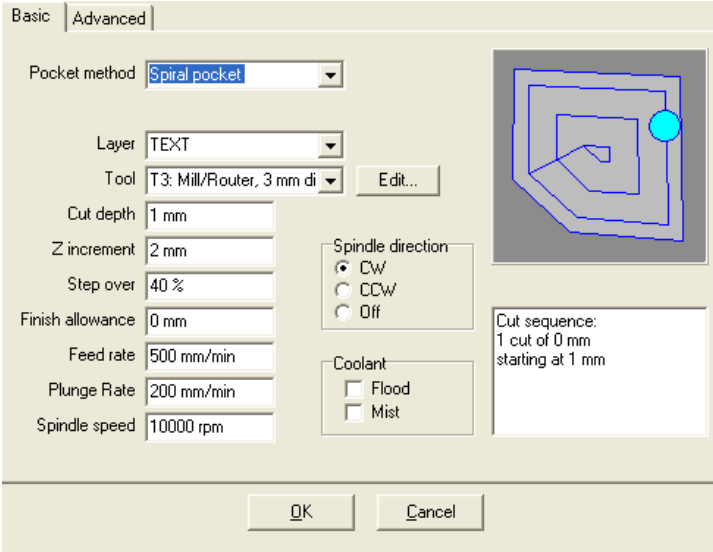
You have now successfully created your first G-code with SheetCam.

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.

First we want 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 new fill process' button  to open the pocket editor.



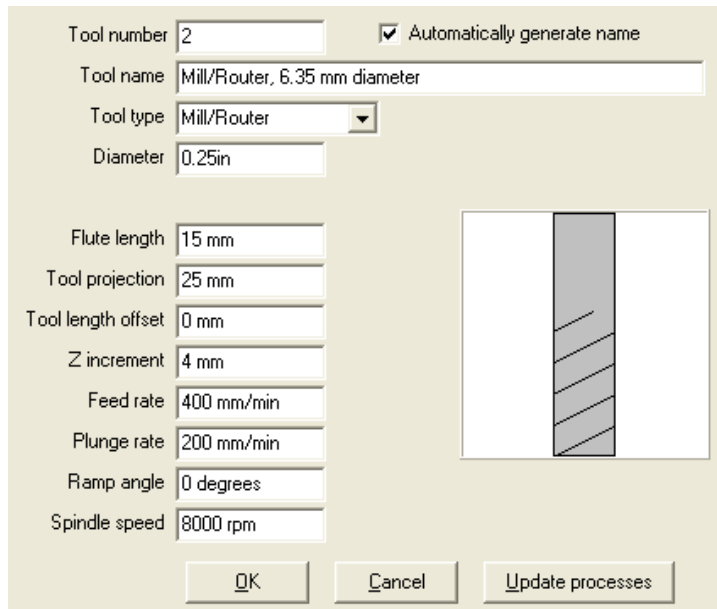
In 'Pocket method' select 'Spiral pocket'. For the layer select 'TEXT'. In the main window the text and the box around it is highlighted in white to show the parts of the drawing that are going to be used by this process.

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.

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 cutter.

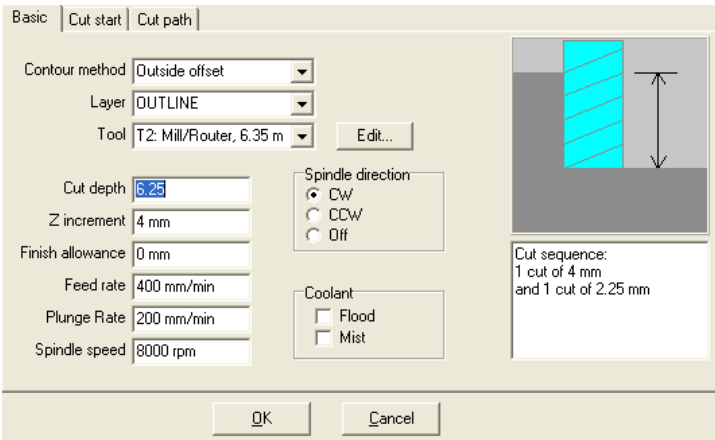
Click on the 'New tool' button  and enter the settings as shown below:



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 process' button  to open the contour editor.



The Contour editor window is shown with the 'Basic' tab selected. The 'Cut start' sub-tab is active. The 'Contour method' is set to 'Outside offset'. The 'Layer' is set to 'OUTLINE'. The 'Tool' is 'T2: Mill/Router, 6.35 mm'. The 'Cut depth' is 6.25 mm. The 'Z increment' is 4 mm. The 'Finish allowance' is 0 mm. The 'Feed rate' is 400 mm/min. The 'Plunge Rate' is 200 mm/min. The 'Spindle speed' is 8000 rpm. The 'Spindle direction' is set to 'CW'. The 'Coolant' is set to 'Flood'. The 'Cut sequence' is '1 cut of 4 mm and 1 cut of 2.25 mm'. The 'OK' and 'Cancel' buttons are at the bottom.

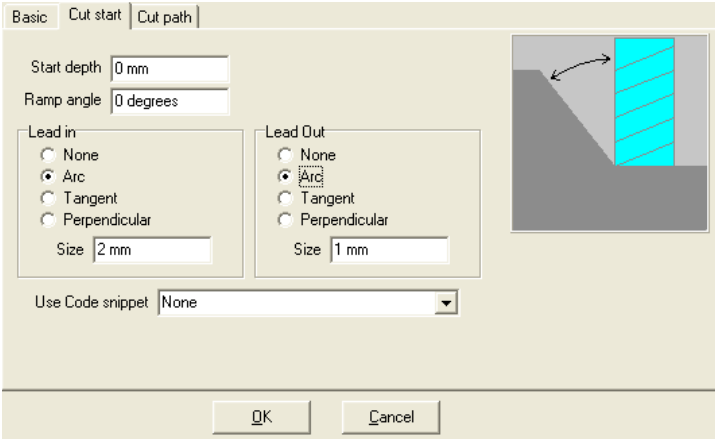
We want to cut around the outside so we use an outside offset. For the layer select 'OUTLINE' as this is the profile we want to cut. The layers 'TEXT' and 'OUTLINE' were created in the original drawing. See your CAD package documentation on how to create layers.

Select 'Tool 2' for this job and set the cut depth to 6.25mm because we want to cut all the way through our 6mm panel. If you hover the mouse over the Z increment a box will pop up and show you that the part will be cut in 1 pass of 4mm and one pass of 2.25mm.

Don't close this window yet as we will need it for the next step.

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.



The Contour editor window is shown with the 'Basic' tab selected. The 'Cut start' sub-tab is active. The 'Start depth' is 0 mm. The 'Ramp angle' is 0 degrees. The 'Lead in' is set to 'Arc' with a 'Size' of 2 mm. The 'Lead Out' is set to 'Arc' with a 'Size' of 1 mm. The 'Use Code snippet' is set to 'None'. The 'OK' and 'Cancel' buttons are at the bottom.

Click on the 'Cut start' tab and set 'lead in' to 'Arc' and enter 5mm for the size. 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.

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. Now SheetCam will start at the bottom of the pocket rather than 'cutting air' to get to depth.

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 anti-clockwise. To change this to clockwise double click on the process we have just created to open the process editor again. In the advanced tab, make sure 'Climb cut' is checked. Now click on <OK>. The outside tool path has now reversed.

Click on the 'Run post processor' button  to create your G-code.

Plasma tutorial

In this tutorial we will open a drawing and create the toolpaths to cut out the part. First make sure SheetCam is in plasma mode. Go to 'Option/Complexity' and make sure that 'Plasma cutting' is checked. You can turn off 'Rotary cutting' to simplify the display.

The first group of icons on the left hand side 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.



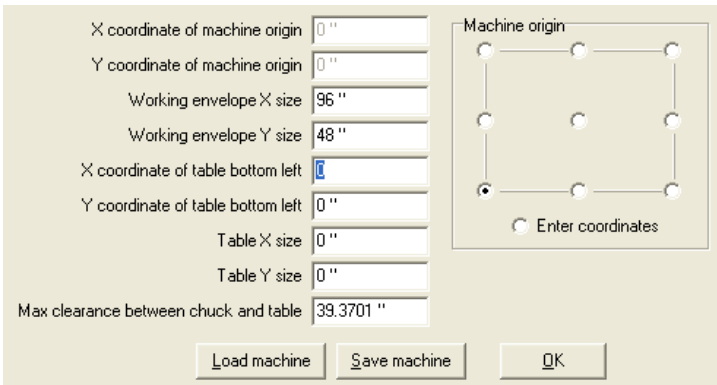
Show path ends. This highlights the start and end points of all open paths. In this case the line underneath the text is an open path because it has a start and an end point. The text consists of closed paths which have no fixed start and end points.

For the purposes of this tutorial we are working in inches so go to 'Option/Units' 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. This displays the machine and material.

Now click on 'Options/Machine'. The display will zoom to fit the machine.



Let's assume our machine table is 8' by 4'. The bottom left hand corner of the useable area is at coordinate 0,0.

In the 'Machine origin' box, click on the bottom left hand corner marker. This tells SheetCam that the origin is the bottom left hand corner.

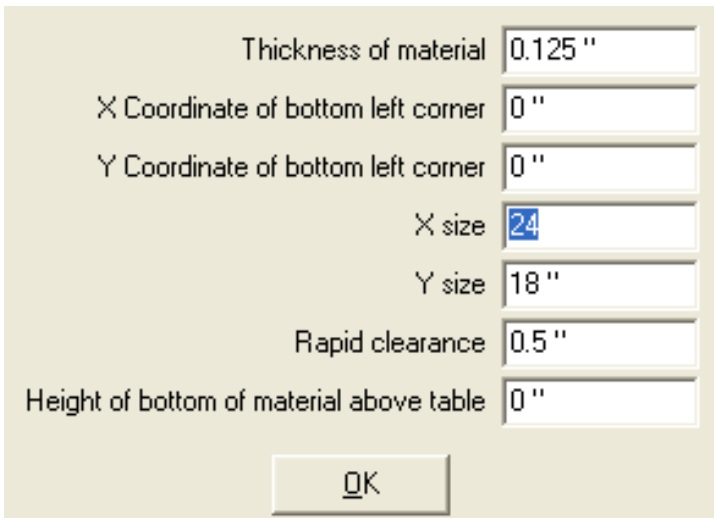
The 'Working envelope' is the useable area so enter 8 foot in the X size and 4 foot in the Y size (include feet, foot or ' to tell SheetCam these measurements are in feet). As soon as you exit each edit box the value is converted to inches for you. Notice that the main drawing display changes as you change the dimensions so you get an idea what the table looks like.

You can define the table bottom left and size 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.

We won't be using the max clearance between the chuck and table so leave it as it is. Click on <OK>.

Setting up the material

Now click on 'Options/Material'. The display will zoom and rotate to fit the material.



A dialog box for setting material parameters. It contains several input fields with labels to their left: 'Thickness of material' (0.125"), 'X Coordinate of bottom left corner' (0"), 'Y Coordinate of bottom left corner' (0"), 'X size' (24), 'Y size' (18"), 'Rapid clearance' (0.5"), and 'Height of bottom of material above table' (0"). At the bottom is an 'OK' button.

We will be cutting this sign out of a 2' x 18" off-cut of 1/8" sheet. The sheet will be placed up against the 0,0 datum bar.

Enter the dimensions and position of the material in the first 5 boxes. 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.

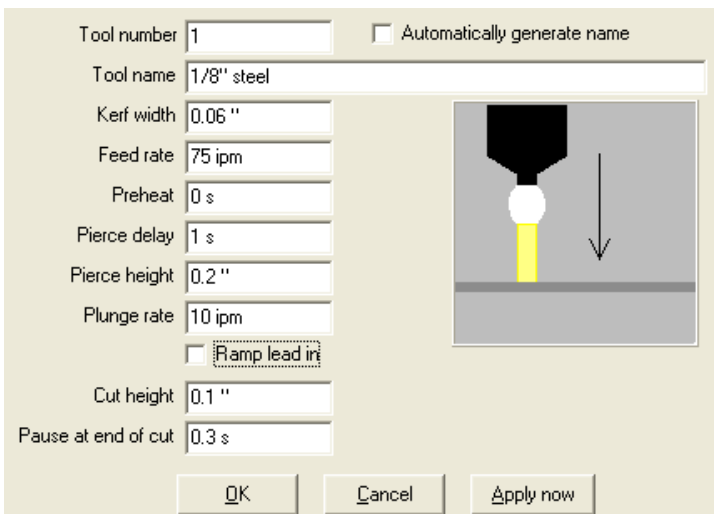
Now click on **<OK>**.

Defining a tool

Now we have set up the machine we need to define the plasma cutter settings. Click on the 'Add a new plasma cutter' icon



to bring up the plasma cutter dialogue box.



A dialog box for defining plasma cutter settings. It includes input fields for: 'Tool number' (1), 'Tool name' (1/8" steel), 'Kerf width' (0.06"), 'Feed rate' (75 ipm), 'Preheat' (0 s), 'Pierce delay' (1 s), 'Pierce height' (0.2"), 'Plunge rate' (10 ipm), 'Cut height' (0.1"), and 'Pause at end of cut' (0.3 s). There is a checkbox for 'Automatically generate name' (unchecked) and a checkbox for 'Ramp lead in' (checked). A diagram on the right shows a plasma torch cutting a piece of material. At the bottom are 'OK', 'Cancel', and 'Apply now' buttons.

Enter 1 in the tool number. 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.

Enter the kerf width (width of the cut line) and feed rate.

'Preheat' is for flame cutting. The machine will wait for this time to heat up the material before it turns on the oxygen. For plasma cutting, leave this at 0 (zero). '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 lead in' selected then this value is ignored. If 'Ramp lead in' is selected then the cutter will ramp down to cut height over the length of the lead in. This can help reduce nozzle contamination.

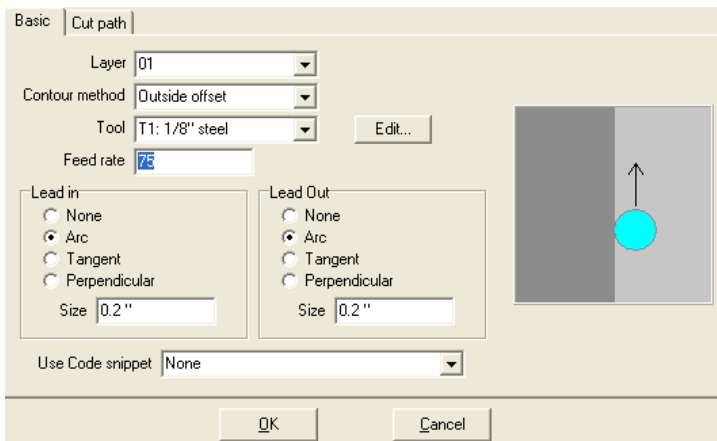
Some plasma machines have a small delay between the torch being turned off and the arc going out. If the machine starts moving before the arc has gone out you can get an ugly gouge in the work. To prevent this set the 'Pause at end of cut' to a suitable value.

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 plasma cut process' button  to create a new plasma cut process.



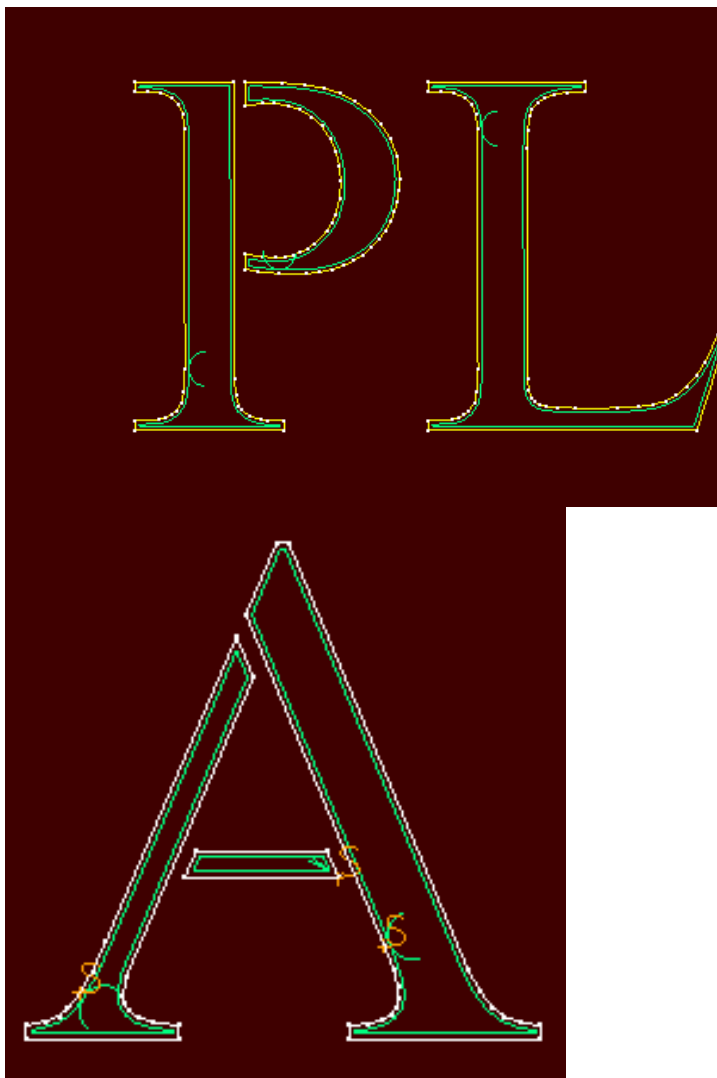
Select the '01' layer in the 'Layer' drop down list. We want to cut the part out of the sheet so select 'Outside offset' in the 'Contour method' box. Select 'Tool 1' in the 'Tool' list. Doing this sets most of the parameters from the tool definition we did earlier.

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 lead in. We can do the same when the cut ends (lead out).

Now click on **<OK>** to save the changes.

Editing start points

Notice that some of the lead ins 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 a red '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 '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.

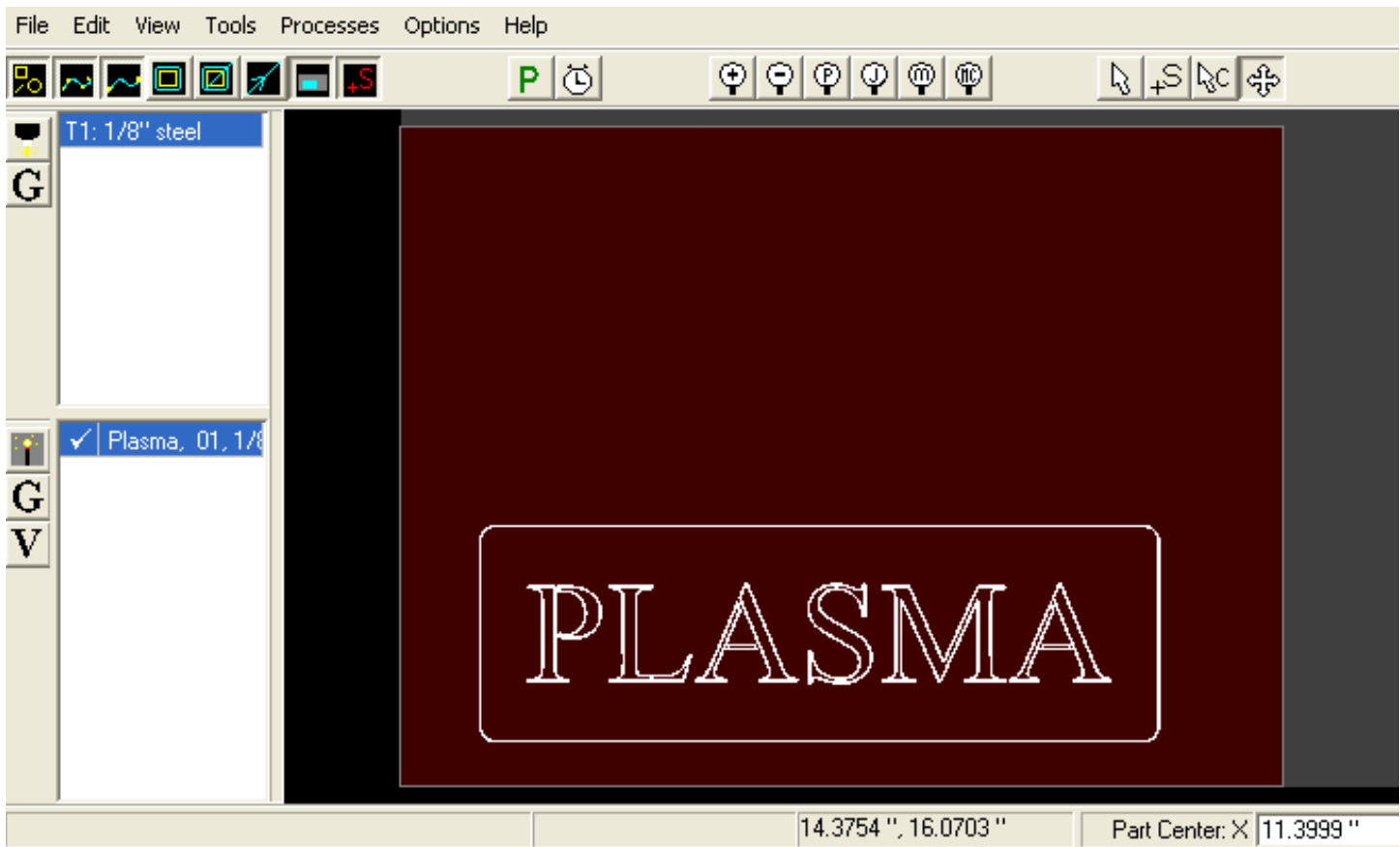
The 'A' presents some more problems. There is simply not enough room in the left hand leg for the lead in/out.

Right click on the start point and select 'Properties'. Un-check 'Use process settings' and change the lead in and out sizes to 0.15". Click on **<OK>**. You can now position the start point so there is adequate clearance.

SheetCam has realised that arc lead ins 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.

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 'Move part button' . You can now drag the part on the sheet. Alternatively you can enter values in the 'Part centre' 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.

You have now successfully created your first plasma G-code with SheetCam.

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/Complexity/Allow multiple parts' is selected.

First we need to define a bigger sheet to place the parts on. Select 'Options/Material' and increase the size to 4' x 2'.

Click on the 'Move part' 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 'Set grid size'. Enter 0.5". Now when you drag parts they will snap to the grid. You can also 'bump' them with the cursor keys. Right click and select 'Set bump increment' to set the amount each 'bump' moves the part.

To rotate a part, enter the angle into the 'Angle' box at the bottom of the screen or use the '<' and '>' keys to rotate the part in 5 degree increments.

Duplicates and copies

While in 'Move part' mode you will notice that the original part (called the parent) is orange and the duplicates are grey. 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.

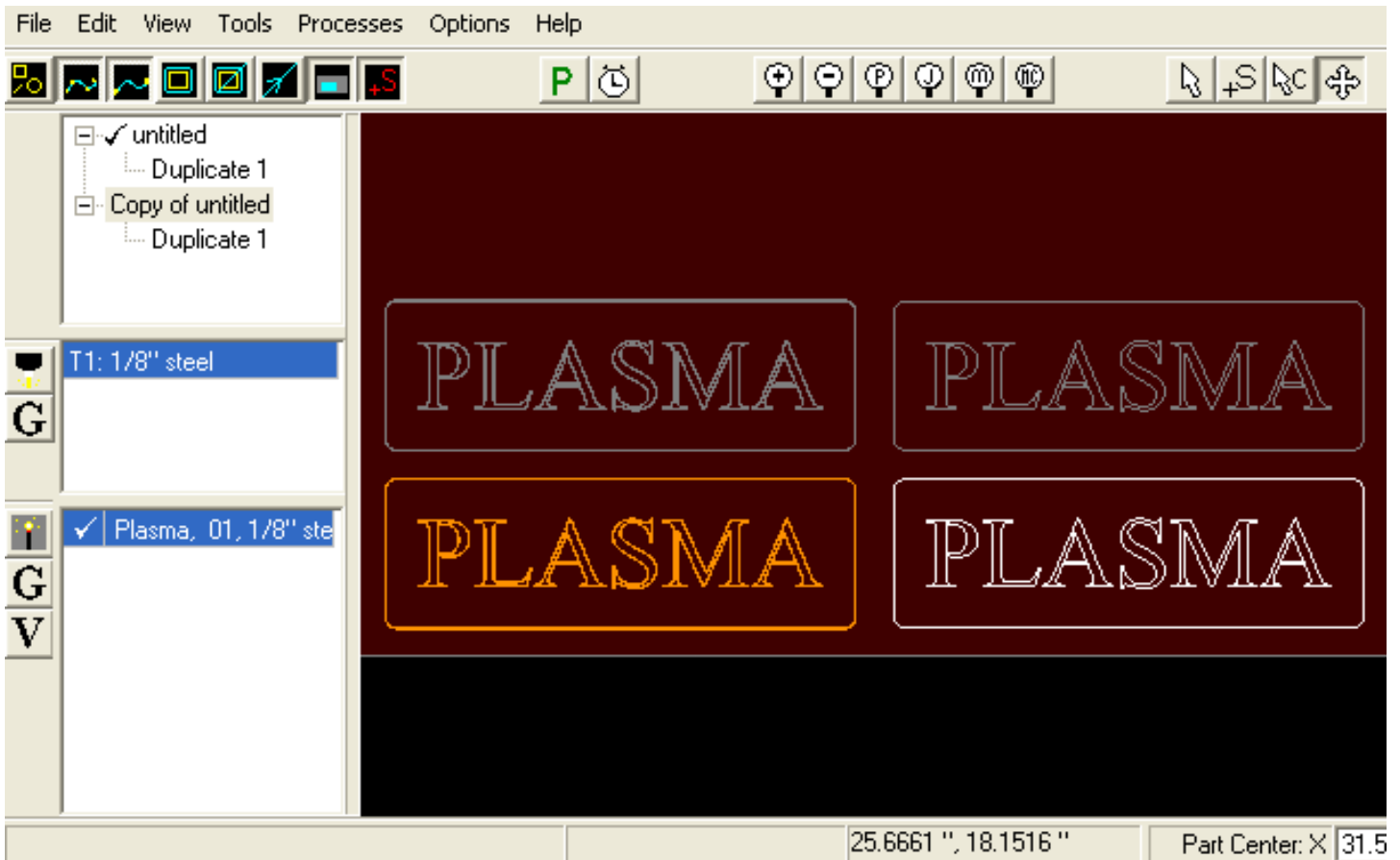
Instead of duplicating a part you can copy it. A copy is completely independent so you can change it's processes or even load a different drawing without affecting the part it was copied from. For obvious reasons you cannot copy a duplicate.

While in 'Move part' mode paths and outlines are all shown in the same colour to prevent confusion.

While in other modes the currently selected part will have it's 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.

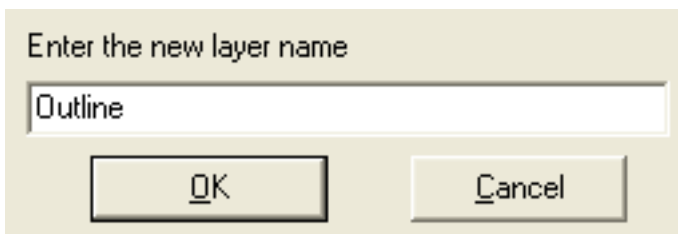
Using Copy

Now we have two signs saying 'PLASMA' we want plain backing plates that will be painted a different colour. 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 the illustration shown below.



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 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 process 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.

Frequently asked questions

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 processes. 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 process 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 process. 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: My cut paths look correct on the screen but when I try to run the G-code only part of the job gets cut.

A: SheetCam is running in demo mode. Check in 'Help/About' to see if SheetCam has found your licence.

Q: I have a simple matrix of 4 holes. There is no milling, just the drilling. I defined the drill and the drill process 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/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

A

About	60
Add new code snippet	67
Add new contour	68
Add new drilling process	68
Add new fill	68
Add new plasma process	68
Allow multiple parts	6
Angle	69
Angle threshold	33
Angular units	7
Auto/manual	31
Automatically generate name	15
Axial travel	22

B

Baud rate	53
Blocks and groups	10
Bottom Left	11
Byte size	53

C

Cancel	17
CAUTION	29
Centre drilling	69
Circle recognition limit	57
Clear all	64
Co-ordinates of table bottom left	9
Colour legend	54
Comm port	53
Configure	53
Contour method	28
Coolant	29
Coordinates of bottom left corner	14
Coordinates of machine origin	8
Copy and duplicate differences	69
Create a support file	61
Create tools and processes	13
Creating drawings	9
Current Position	11
Current position	12
Cut depth	29
Cut height	27
Cut path	31, 40, 44
Cut start	30

D

Default drill hole tolerance	57
Define a cut process	74, 80
Define another tool	76
Defining a tool	73, 79
Diameter	16
DNC Tool	52
Drag mode	72

Drawing centre	11
Drawing files	51
Drill depth	13
Duplicates and copies	82
DXF	11

E

Edit	29
Edit contour properties	65
Edit G-code	46, 68
Edit G-code Notes	46
Edit Menu	52
Editing start points	80
EMF	11
Enabled	13
Estimate cut time	62
Excellon	11
Exit	52
Explode text in DXF files	10

F

Feed rate	7
File scale	11
Finish allowance	29
Finish depth	43
Flow control	54
Flute length	16
Frequently asked questions	84

G

G-code extension	8
Generate the tool paths.	75
Generating the G-code	75, 81

H

Height of bottom of material above table	14
Help	59
Help Menu	59
Hints and Tips	69
HPGL	11

I

Import link tolerance	57
Initial setup	4
Introduction	4

J

Job files	51
-----------------	----

K

Kerf width	26
------------------	----

L

Layer	28
Layer tool	54
Lead in	30
Lead in and lead out	77
Lead out	30
Linear units	7
Load machine	9
Load process	52
Lower right toolbar	69

M

Machine and material parameters	70
Machine origin	8
Machine parameters	70
Machine setup	7, 8
Material parameters	71
Material setup	14
Max clearance	71
Max detail reduction error	57
Max hole size	43
Max. clearance between chuck and table	9
Measure	63
Min hole size	43
Mirror	69
Move part	66
Move to layer	65

N

Nesting Tutorial	82
New automatic tap	21
New code snippet	27
New contour	28
New drill	17
New drilling process	42
New fill	35
New job	51
New mill button	67
New mill/router	15
New part	51
New plasma button	67
New plasma cut	38
New plasma cutter	26
New rigid tap	24
New tapping process	47
New V cutter	19
Notes	34
Number format	13

O

OK	17
Open and save options	51
Open drawing	51
Open DXF options	11
Open EMF options	13

Open Excellon options	12
Open HPGL options	12
Open job	51
Open licence	60
Open part	51
Open toolset	51
Opening drawings	11
Optimise now	31
Options Menu	56
Order	64
Origin	11, 12
Other Features	52
Outlines must be properly closed	9
Overcut corners	32

P

Panning	71
Parity	53
Part centre	69
Part files	51
Pause at end of cut	27
Peck depth	18
Peck retract	43
Pierce delay	26
Pierce height	26
Pitch	22, 25
Place cut start point	64
Place holding tab	65
Plasma cutting	5, 6
Plasma tutorial	78
Plunge rate	13
Pocket method	35
Pocketing tutorial	76
Position boxes	56
Positioning the part on the sheet	81
Post output to last drawing folder	58
Post process	50
Post processor units	7
Preferred cut direction	32
Preheat	26
Print	52
Process files	51
Process setup	27
Processes toolbar buttons	68
Profile tutorial	72

R

Ramp angle	16
Ramp lead in	27
Rapid clearance	14, 71
Recent drawings	14
Recent jobs	51
Recent parts	51
Reopen last job on startup	58
Reverse cut direction	41
Reverse mouse wheel	58

Reverse multiplier	22, 25
Reverse open paths	32
Right click options	63
Rotary cutting	5, 6
Rotating the screen in 3D	72
Run code after toolchange	56
Run code before toolchange	56
Run post processor toolbar buttons	62

S

Safety	4
Save job	51
Save job as	51
Save machine	9
Save part	51
Save part as	51
Save process	51
Save toolset	51
Scroll the screen with the mouse	63
Select language	58
Select post processor	7
Selection toolbar buttons	63
Send data	52
Set arrow frequency	57
Set bump increment	67
Set direction arrow size	57
Set grid size	67
Set post variable	47
Set start point size	56
Setting up the machine	72, 78
Setting up the material	73, 79
Setup Wizard	4
Setup wizard	59
Shapes must not self intersect	10
Sharpen corners	32
Sharpen/Overcut corners	32
Show cut start points	62
Show input paths	54, 61
Show machine and work	54, 62
Show path directions	54, 62
Show path ends	54, 61
Show rapid moves	54, 61
Show segment ends	54, 61
Show tool paths	54, 61
Sizes	14
So how do I find problems?	10
Spindle speed	13
Splines and bezier curves	10
Start depth	29
Start point	31
Step over	36
Stop bits	53
System parameters	57

T

Tab length	33
Tab length/Tab thickness	33
Tab thickness	33
Table depth	70
Table size	9
Table width	70
tapping process	47
Thickness of material	14
Thickness of work	71
Thread pitch	7
Tip diameter	20
Tool	29
Tool change	56
Tool length offset	16
Tool name	15
Tool number	15
Tool projection	16
Tool setup	15
Tool type	16
Toolbars	61
Tools toolbar buttons	67
Toolset files	51
Tutorials	59, 72

U

Underfeed	23, 25
Update processes	17
Use code snippet	30, 64
Use drawing origin	11
Use layers	10
Use lower left coordinates	8
Use points for drilling	12
Use process settings	64
Using Copy	83
Using layers	77
Utilities	60

V

V angle	20
Value	47
Variable name	47
View menu	52

W

Work flow procedures	9
Working envelope size	8

X

X and Y coordinates of bottom left corner	71
X and Y coordinates of table origin	70
X and Y size	71

Z

Z increment	16	Zoom to fit job	54, 63
Zigzag pocket	35	Zoom to fit machine	55
Zoom in	55, 62	Zoom to fit material	55, 63
Zoom out	55, 62	Zoom toolbar buttons	62
Zoom to fit current part	54, 62	Zooming and panning	71
		Zooming in and out	71