

Sheetcam Setup

During initial installation select Jet cut tool and metric units

After program has installed

From Tool bar click on options

Job options

Used for setting up sheet sizes etc varies on individual jobs

Machine

Machine type

Check jet cutting is ticked rotary cutting is unticked

Post processor

Select Hypertherm edge from drop down list

Units metric

Output folder (normally remember last)

Out put file extension tap

Working envelope (optional)

Set working envelope of machine and origin

Table display (optional)

As above

NB if setting the origin in any option ensure it is always the same corner or odd results will occur

Creating a tool

You need to create a tool for each kerf (cut width) value for your plasma

Depending on plasma type I would suggest from 1 to 5mm in 0.5mm steps

On left hand side

Tools box

Top icon (create new jet cutting tool)

In menu select type ie plasma

Select kerf width ie 1mm

There is no need to change any of the other settings as our controller sets these itself

Click ok

Repeat until you have all your required kerf settings

If using a Marker use tool 9 to tell the controller to mark rather than cut.

Set T9 to Marker set kerf width to 0

Bringing in a part

Create your part in autocad ,draftsight etc

Only draw what you want to cut ie no dimensions , no borders no construction lines etc
all parts of the shape should be on the same layer (I would suggest layer 0) all lines
should be solid continuous

On sheetcam press File

New part (about half way down menu)

In the drop down box ensure dxf files is selected in the bottom right corner

Then navigate to where your part is stored, select and press open

In the drawing options

Select metric

1:1

Drawing position set same as previous origins

I usually set use drawing name as part name

Untick use points for drilling

Untick use colours as layer names

Press ok

Your part should appear on screen if the outside is red and holes are yellow all is well
If any part of the shape is white or insides are red it means there is something wrong with
the dxf drawing – sheet cam will try it's best to remove any double lines join up any gaps
but can only do so much. However it will show you where to look for errors by putting
white dots where there is an issue

Once you have a good shape / nest

On the left side operations

Top icon (magic wand /light house ????) click

In jet cutting

Basic

Contour method outside offset

Layer select layer part was drawn on ie 0

Tool select the relevant tool from the ones created earlier

Offset open paths untick

Leadins on open paths untick

Reverse cut direction tick

Lead in

I usually use perpendicular

Length depends on material thickness (rule of thumb is lead in should equal thickness – however this is a personal preference)

Lead out

Usual set to omm

Cut path

Cut ordering –auto

Start position , you've guessed it same as other origins

Cutting rules

Keep parts together Tick

Click ok

If you are using marking, in your CAD programme, draw the parts to be marked on a different layer to the profile. Process the marker layer first by selecting the marker layer in the jet cutting menu accessed from the top icon on the Operations box, set the tool to T9.

The part should have a green out line (kerf) and show the start points

From top tool bar press the green P

Select where you would like to save the tap file ie memory stick

Take memory stick to machine and chop out the shape

When exiting Sheetcam ensure you click ok when asked to save default tool set or all your good work will be for nothing

Now the info is saved you can process a shape in a few clicks