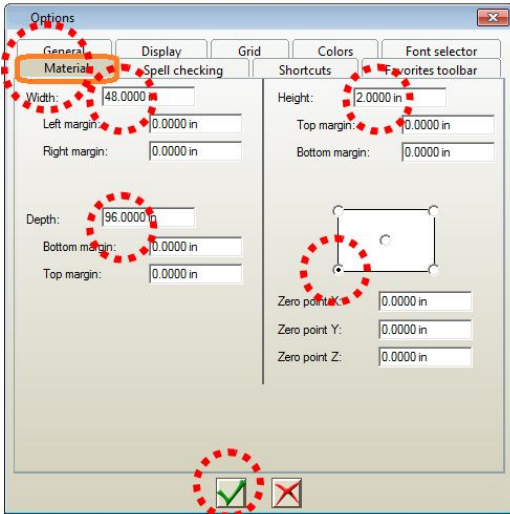




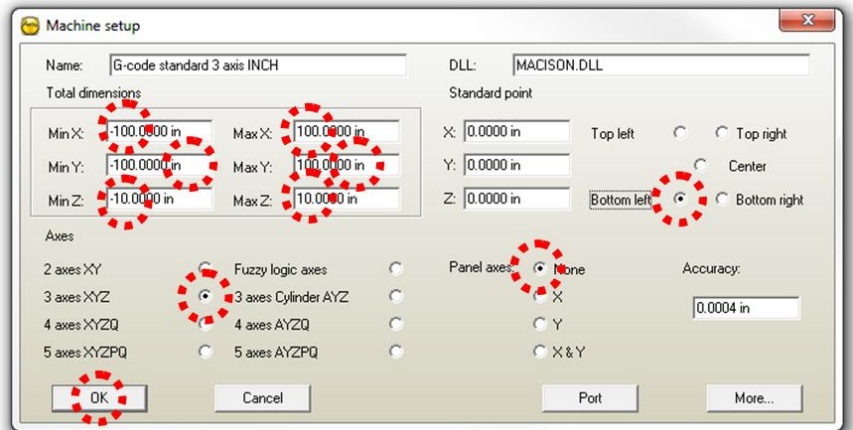
Unleash your Router's 3D Capabilities

How to adjust machining settings to avoid overflows?

It happens at the time of the post-processing, error messages regarding overflows are displayed. Those messages are often due to one or more inaccurate settings between either the machine size defined, the size of the Working Area/Artwork size set, and/or the toolpath area built. To help going through the adjustments of all these elements, this tutorial explains the proper order of the installation steps to remove those blocking messages.



In tutorial #8 **How to set machine limits in ArtClip3D**, the working area definition has been explained, this virtual black rectangle defining the size of your machine bed is important to set limits. Once the Width, Depth and Height are set accordingly to your machine size in the F10 options, those dimensions will constantly come back at the opening of ArtClip3D. This will always be the physical machine size you see at the opening, everything built inside this black rectangle will be within your machine; everything outside this black rectangle will not machine and will result of an overflow message.

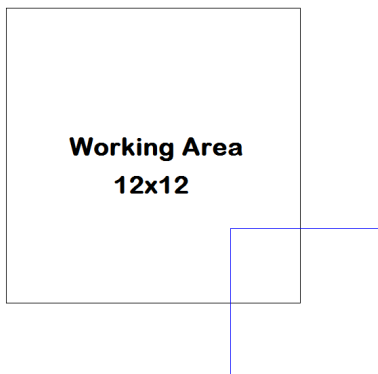


In Tutorial #11 **How to install a machine in ArtClip3D**, the installation of a post-processor has been demonstrated. That little application taking care of the translation of the toolpaths into the language of your machine also contains important dimensions interacting with the post-processing.

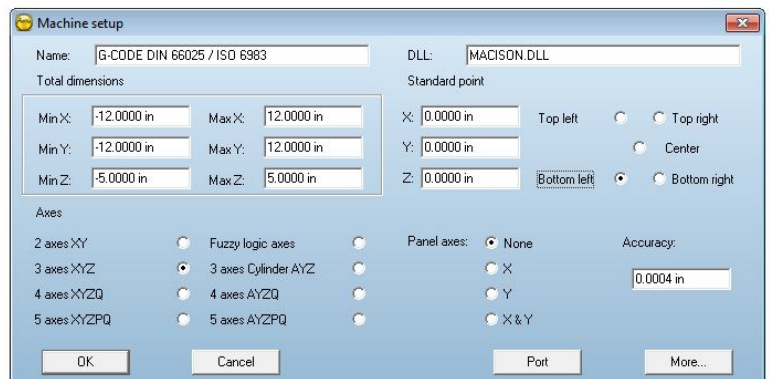
Let' see how all of this information explained in those 2 previous tutorials now interacts.

Below is an example of a 12 x 12 Working Area, which could correspond to a machine bed size or a material size depending on how this area is used. Some CNC machine are as small as 3 in by 3 in and others can be unlimitedly bigger, while the material size can be of any size; once the machine size is set it is easy to know the machine limits thanks to the Working area defining either the machine or the material size.

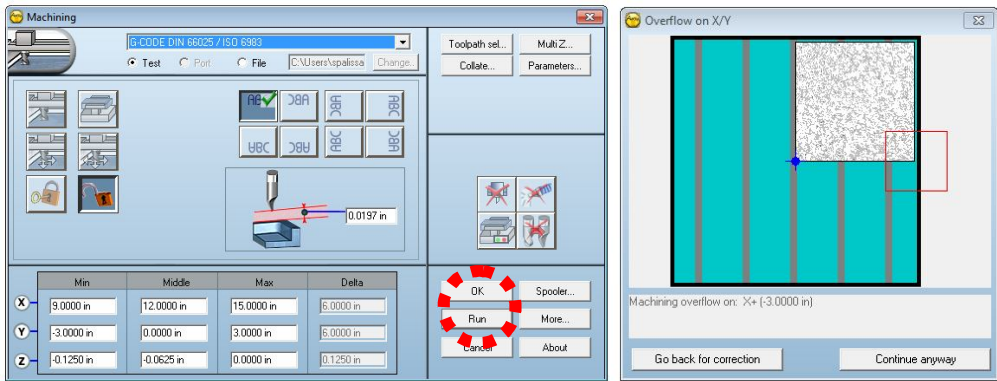
A square vector of 6x6 has been drawn outside the working area. We read earlier that anything inside in the working area will machine straight from the machining window, and that anything outside will bring overflow messages.



Square Vector 6x6



If the current square which is outside the working area is used to generate a toolpath, by extension this toolpath will be as well outside the Working area. As a result we will obtain an overflow message as this one shown below on the right, soon as the OK or RUN button is clicked:

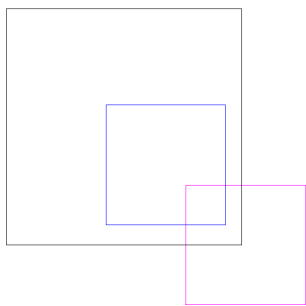


In this Overflow window there are 4 important elements:

- o The **biggest square in blue azure** corresponds to the size of the machine defined in the postprocessor installation as explained in Tutorial #11, with the total area dimension of 24 x 24 as shown in the Machine Set up window further above.
- o The **marble square** corresponds to the Working area of 12 x 12 as we defined it in this tutorial and explained in the Tutorial #8 with different machine dimensions.
- o The **red square** corresponds to the toolpaths area shown as a bounding box. This box is not contained inside the Marble Area (Working area), neither within the Blue Azure area (Machine definition). When this is the case then an overflow most likely will happen.
- o The message itself, stating where the overflow occurred; here in X at **3 mm** from the working area.

To solve this overflow it is needed to have the Red area within at least the Machine definition so the toolpath can be post-processed.

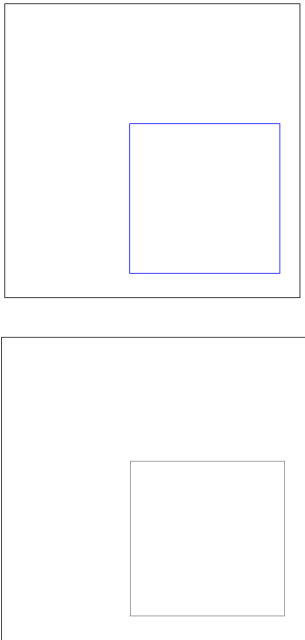
To reach this point several sequences are possible. Here is the easiest:



The toolpath should be generated within the Working area, so this supposes to go back inside the 2D editor and modify the position of the square. Having the 2D or even a 3D object area within the Working area will insure you will be able to send it to the machine.

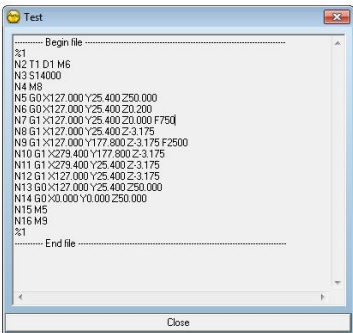
Once done keep in mind that only was changed the vector position. The toolpath position needs to be recomputed as well to follow the new position of the vector (Right).

Once the toolpath is recomputed to the proper position, you can then try to send it to the machining and postprocessing.



The overflow message will not come back if you either click the OK or RUN button as the machine, the toolpath and the working area are properly aligned to post-process.

The G-Code will output right away as a program file towards the location you defined on your hard drive or as a quick preview with the TEST position as shown on left.



Frequently Asked Questions

- **I am having an overflow on Z, why?**

Like any overflow your toolpath has reached a position in XY or here in Z that is not contained in the machine definition of ArtClip3D (Post-processor) or within the working area defined. Extend the size of the Z directions (Z travel) and the overflow will be removed the same way as for X or Y overflows.

- **I do not have any overflow, but my 3D toolpath went way lower what I was expecting. Is there any reason?**

There is only one reason that would explain a toolpath depth going deeper than expected. The 3D toolpath is actually going to that depth. By checking the Z Mini while post-processing you can check how deep the toolpath is set for.

To set an overflow here as a security make sure that the Z mini in the Postprocessor is set to the maximum thickness or depth you want to go. Setting up a Z mini will secure anything you have underneath the material cut and will protect as well the tool to reach depth it is not supposed.