## 2.5 Axis CNC Routing Workflow

In this example two types of machining will be utilized – **Pocketing** and **Engraving**.

Pocketing machining refers to the removal of material within a closed 2D shape to a specified depth. In this operation the bit is held to the edge of the shape or '*region*'.

Engraving machining refers to the removal of material along a line or '*path*' to a specified depth. In this operation the bit is centered on the path.

Geometry Prep in Rhino

Draw 2D 'regions' that depict the perimeter of level planes at set elevations. (These are indicated in yellow in the image)
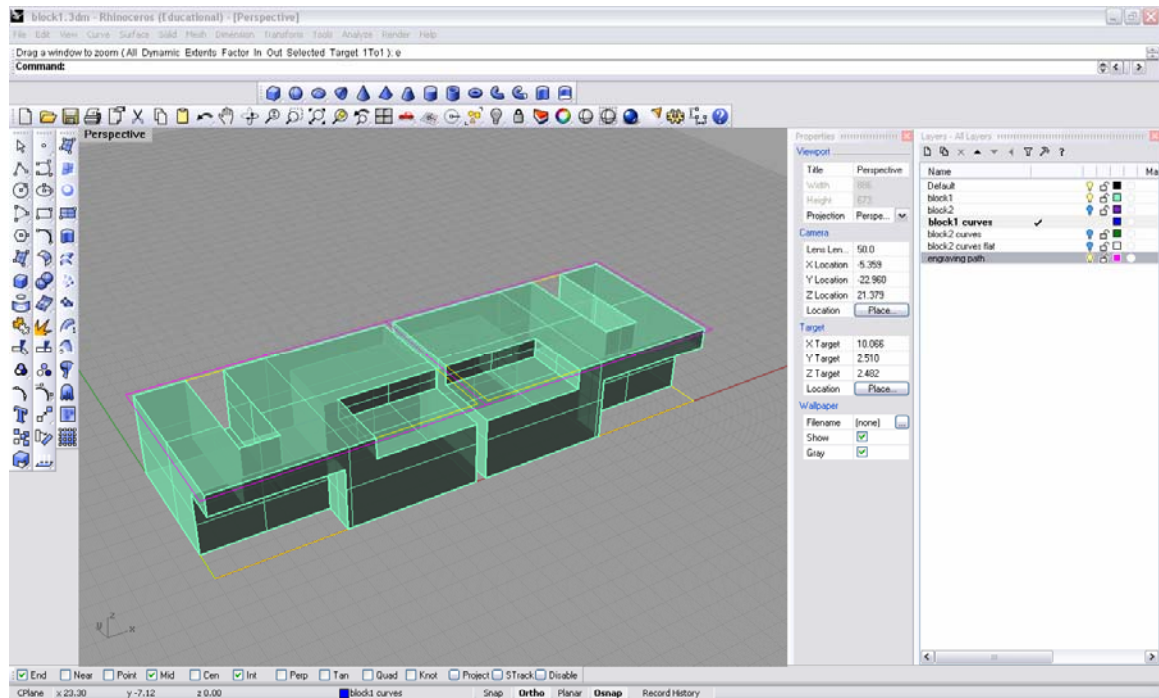

Draw 2D 'engraving' lines that will become the path the bit will follow. (These are indicated in magenta in the image)

Position the object to be cut correctly relative to the x,y,z origin. The CNC router bed is 4'x8'. The orientation is 8' in the y-axis and 4' in the x-axis. The front left corner of the machine is 0,0. The surface of the bed is Z 0. Objects to be cut must always be oriented to fit within this area and positioned with a corner at the origin and all geometry in the positive x,y and z as shown above.

Once file is prepared with appropriate geometry and layers, save in rhino format.

Open Visual Mill and open your rhino file. All layers and colors established in rhino will be transferred. Use this to your advantage. While to 3d geometry was transferred for visual purposes, all machine operations will be generated using only the 2D geometry.
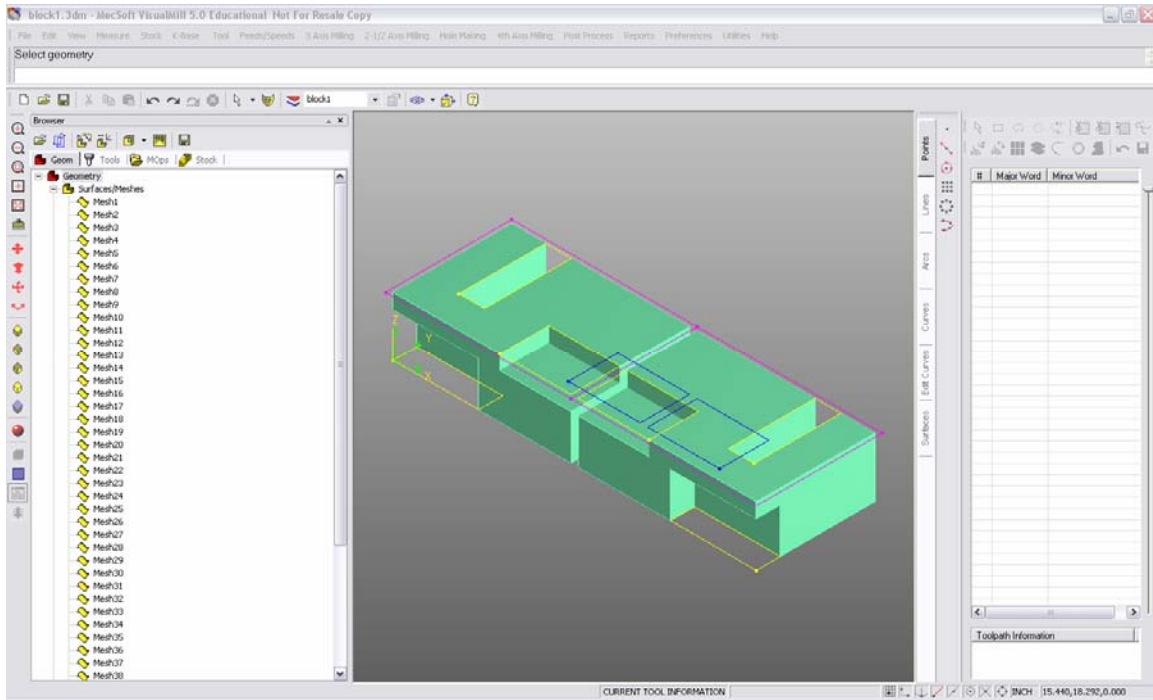

Steps to generating toolpaths:

1.   open file
2.   define stock dimensions (stock refers to the material you will cut your object from, in this case foam) For this project we will use box stock.
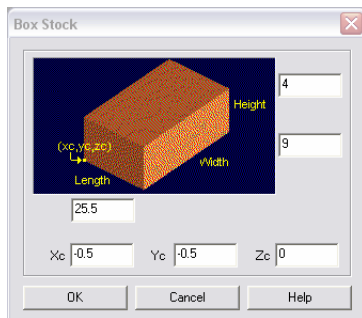

3.   pick the '*tool*' (bit) that will be used to cut the object. Only one tool will be used for this project. It is **Tool 2** within the tool library. The settings for the tool are preset and must not be changed. We only need to choose the bit. This is done by double clicking on the bit within the library list.
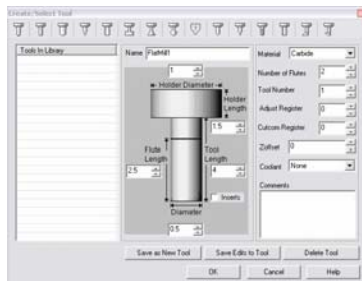
4. select the geometry that will be used to generate the toolpath(s). Geometry can be picked line by line or selected with a rectangle as done within rhino. It is useful to turn off the layers not needed to facilitate the selection process.

5. pick the appropriate toolpath (either **pocketing** or **engraving**, these are both found within the 2.5 axis menu)
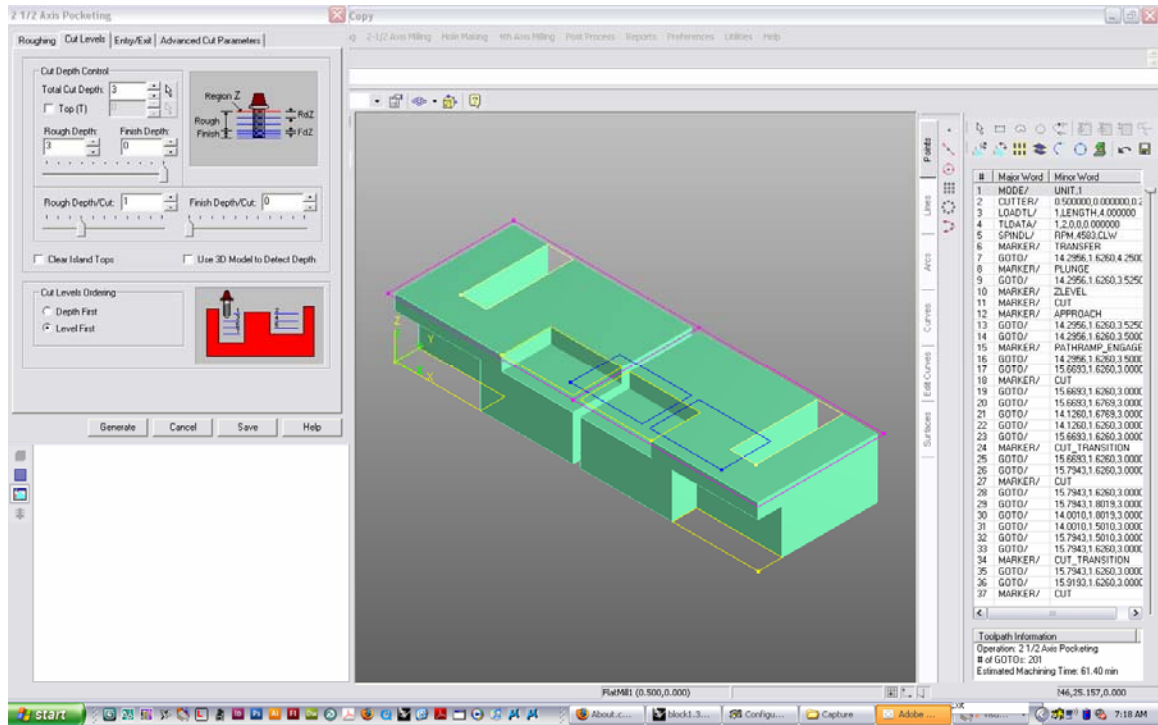

6. enter the appropriate variables for machining. [CUT LEVELS TAB] These are **Total Cut Depth** (overall depth of our machining) and **Rough Depth/Cut** (the depth that will be cut with each pass) This must not be above 1". In the example below we are cutting to a total depth of 3" in 1" increments, hence the machine will make three '*passes*'. [ROUGHING TAB] **Tolerance** and **Stock** must be set to 0.001 this will ensure the machine will cut precisely to our geometry. The only other variable to set will be **Step Distance** which specifies how much the machine will 'step-over' from '*pass*' to pass. This should be set to **50% of the diameter of the bit**.
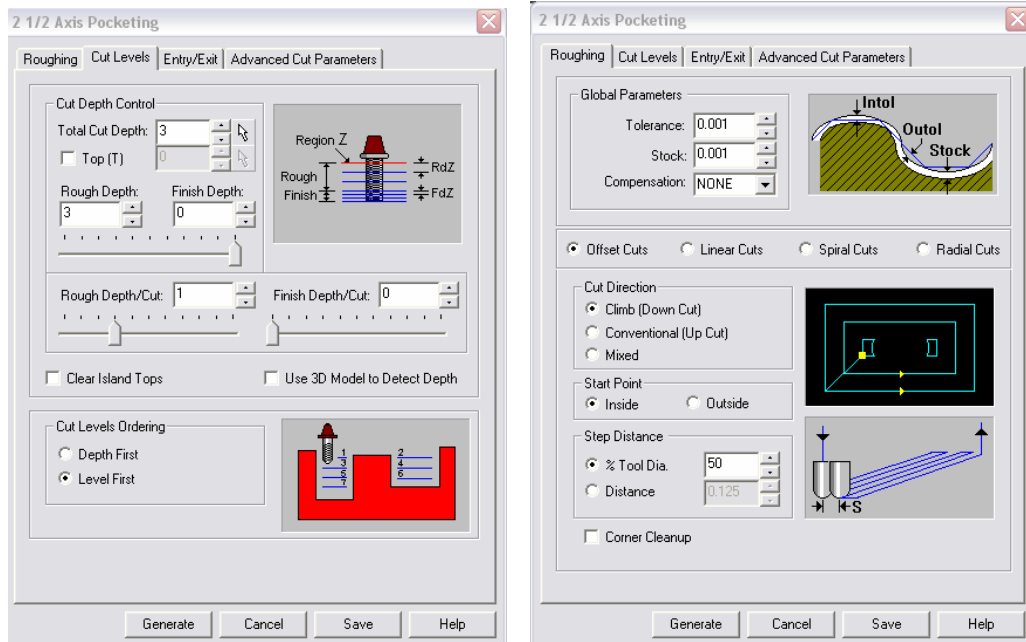
7.　　　once these variables are set correctly the toolpath can be '*generated*'. In the image below, the cyan refers to the step over pocketing cuts, the yellow refers to the plunging of the bit into the material and the red refers to the rapid transfer of the bit (vertically and horizontally). We can see the 3 vertical steps of 1" increments.
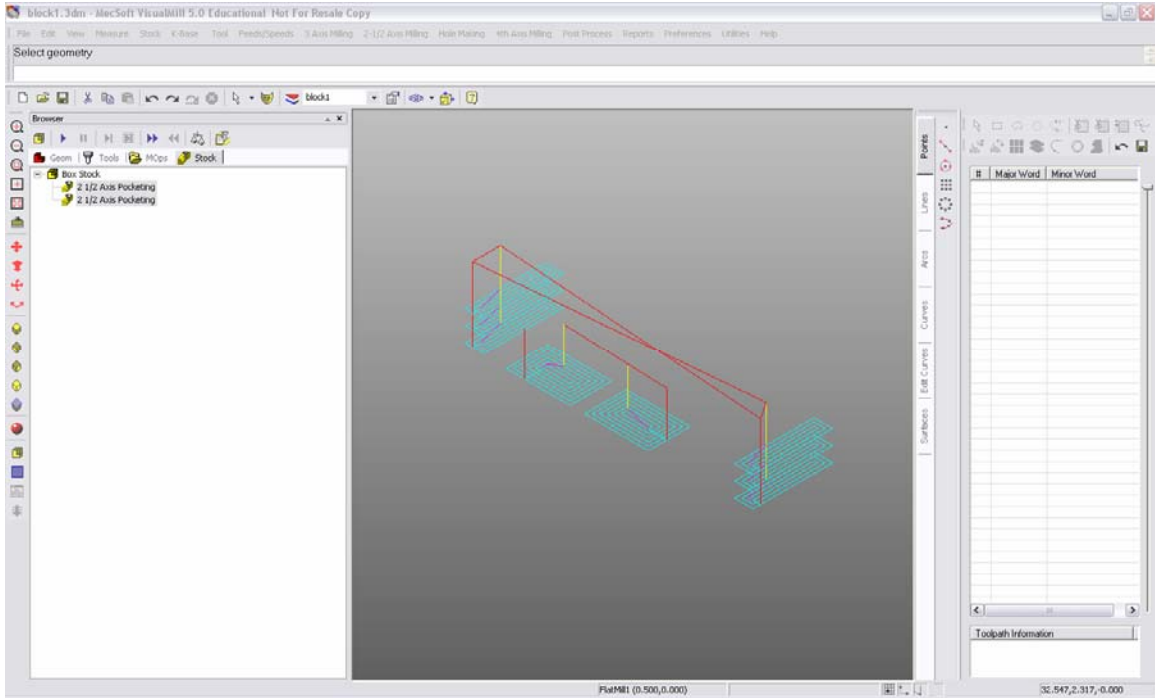

8.　　　preview the milling within the **Stock tab.** This will allow us to see a preview of how our object will be cut.
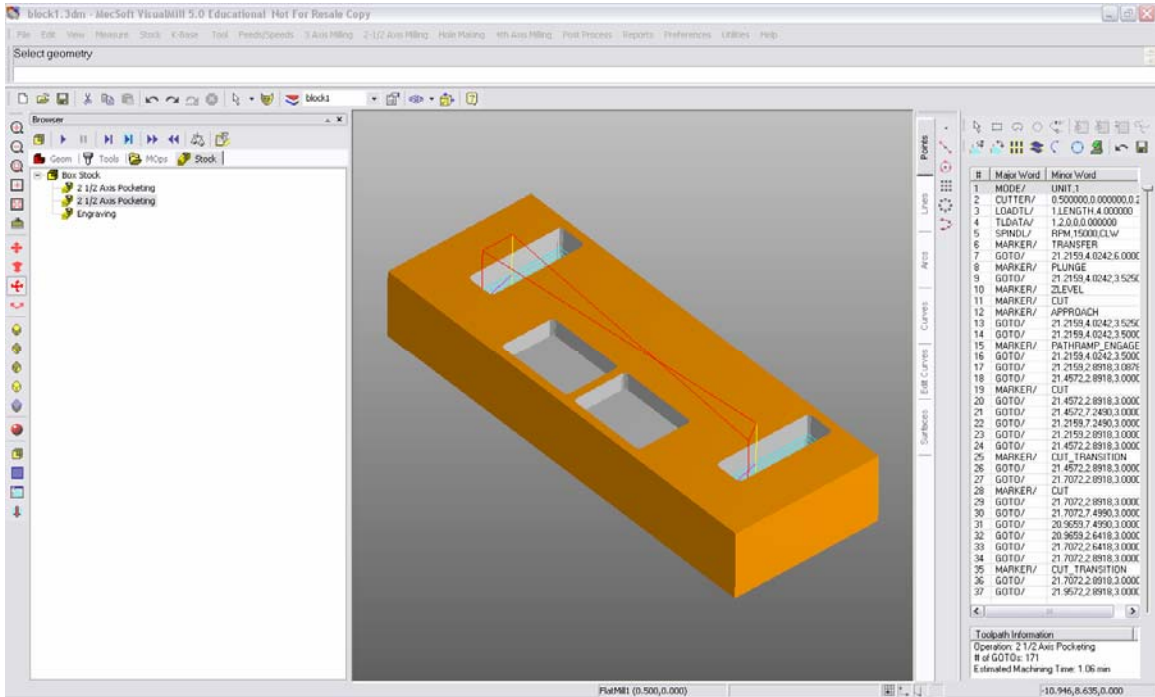
9.   The next step is to define and generate the engraving toolpaths to cut out our two blocks. Similar to the previous steps we will need to select our geometry and pick the engraving option within 2.5 axis milling. Here we will specify the **Total Depth Cut** (4" because our geometry is 4" above Z 0) and number of passes within the **Rough Depth/Cut** option (there will be two 2" deep passes for this engraving toolpath – this number must never be greater than 2")
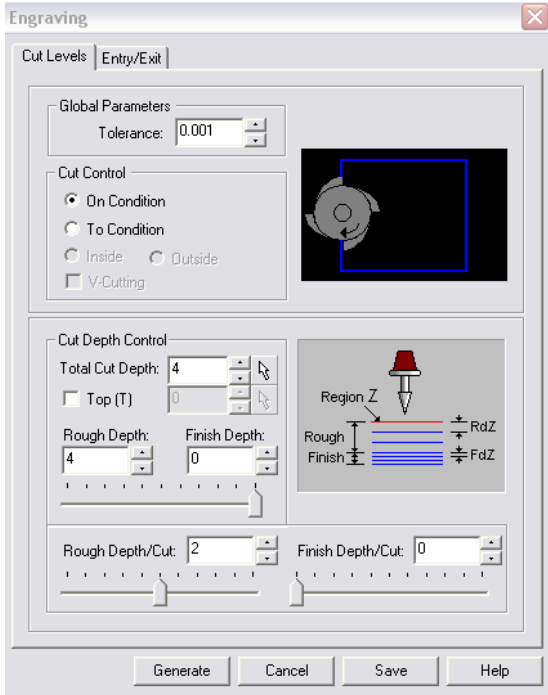

10.   Generate toolpath and preview within stock tab
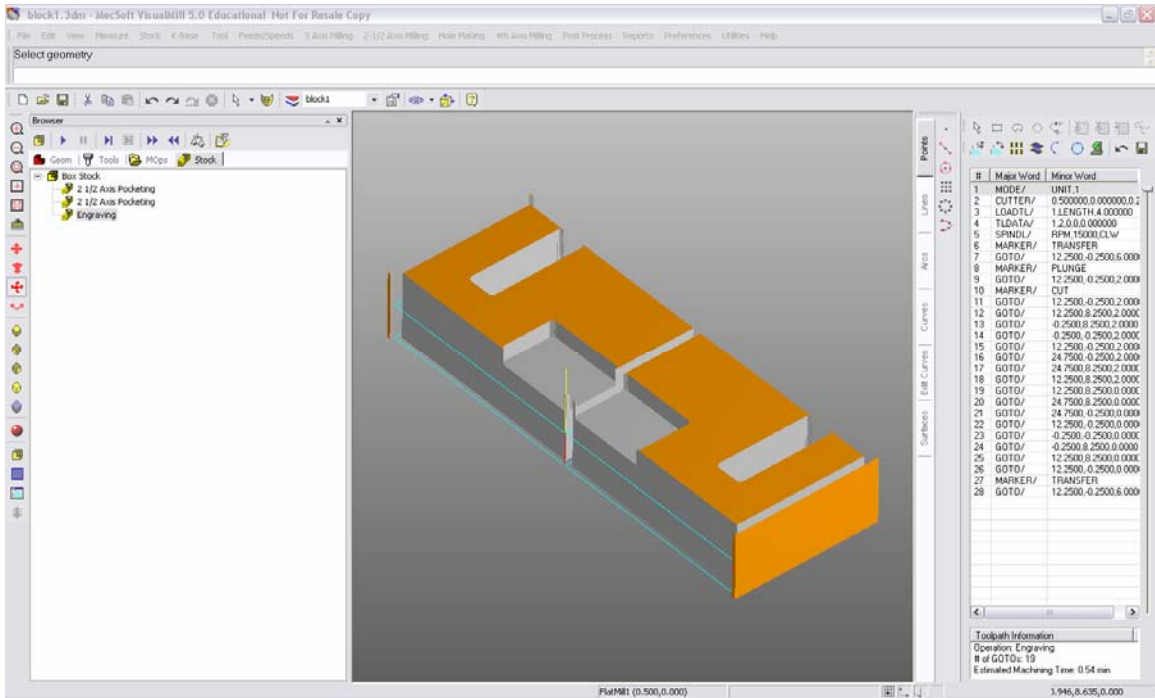
11. Set **Speeds and Feeds** (speed is the RPM of the spindle and cutting bit, Feed is the rate in inches per minute that the machine will cut) Speed must be set to 15000RPM and Feed rates must be set as shown below. This must be done for all toolpaths.
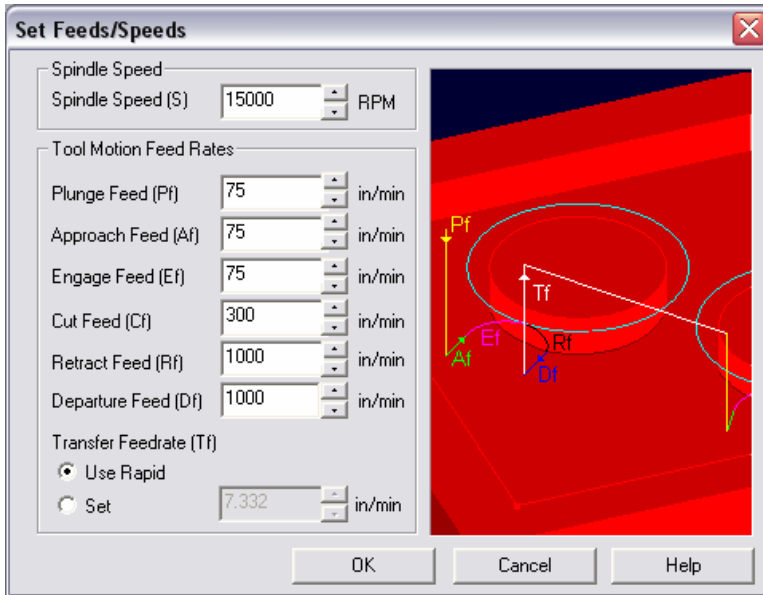
12. Once the Feeds and Speeds are set all toolpaths must me regenerated.
13. **Post Process** toolpaths to generate G-Code for machine. Use the CNT MOTION 950 post processor as shown below. Name your file and place a .NC suffix. Example Block1.NC
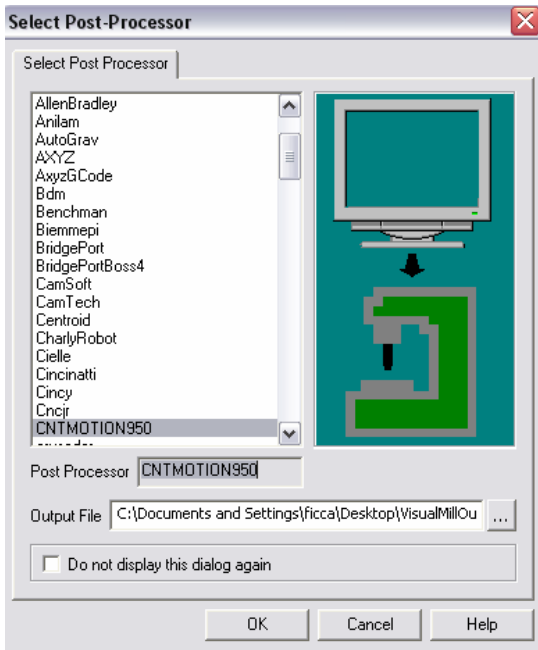
14. Milling steps will be covered in a separate PDF.