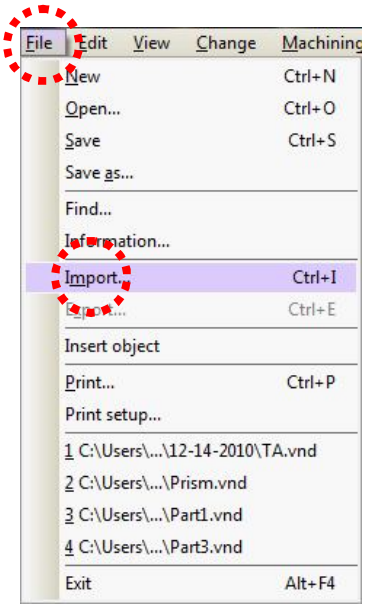
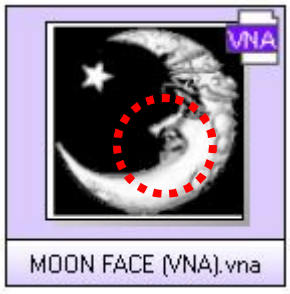
Unleash your Router's 3D Capabilities

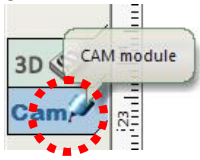
How to machine a 3D Scene?

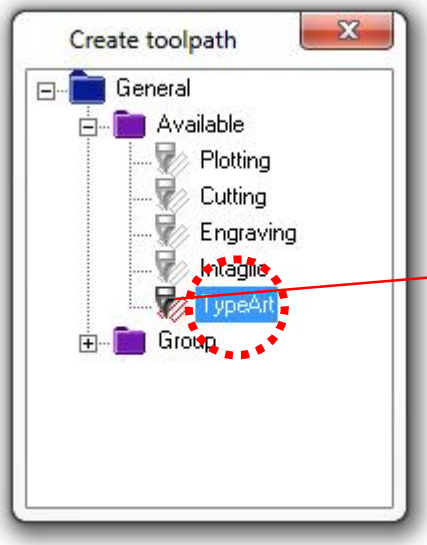
- *Import any 3D model from the "**3D Models**" folder.*





- *On the left side of the screen click the "**Cam**" tab to be directed in the Machining Module.*



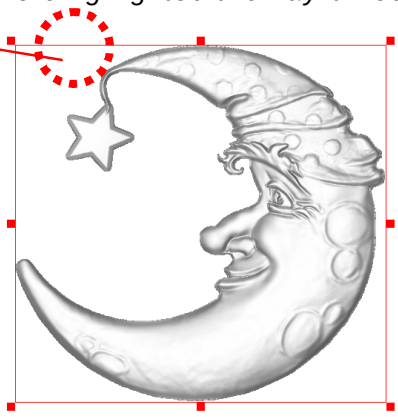


- *Open the **Create toolpath** window if not yet displayed.*

  *The **Create toolpath** icon*  *opens/closes this window.*

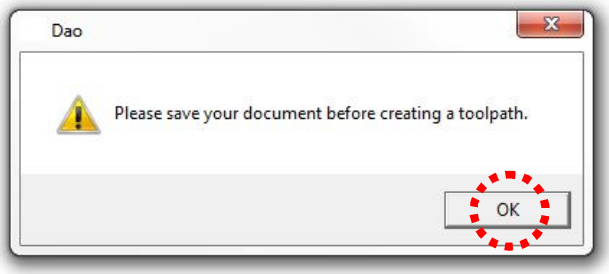
- *Select the 3D model so the "**TypeArt"** toolpath highlights.*

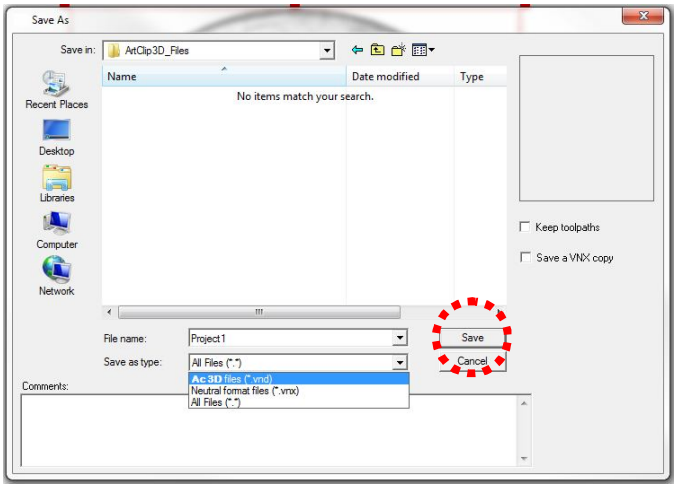
When a toolpath name is highlighted this way it means it is available.



- *Double click the **TypeArt** toolpath.*

*A **Dao** message prompts you to save the file.*

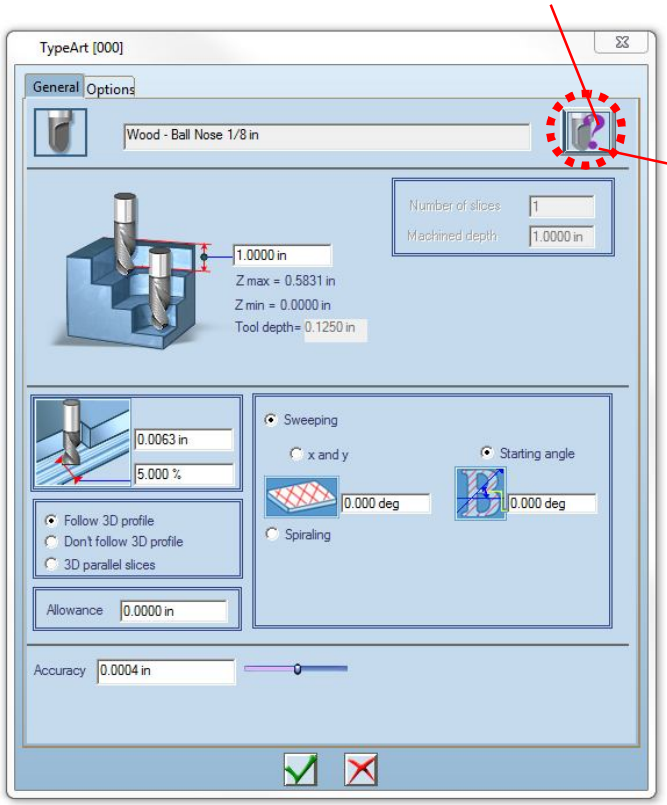


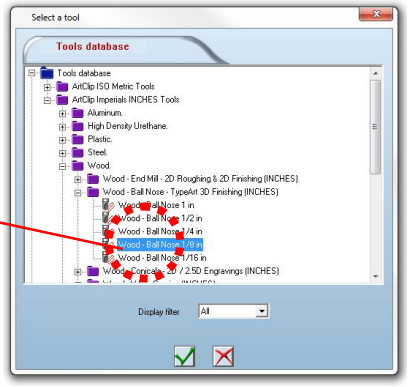


- *Click **OK**, the "**Save As**" window follows.*
- *Save the VND file under any desired folder name by clicking on **Save**. Project1 will be the name used here as the file name.*

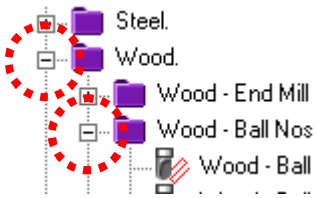
*Soon as the file saves a new window titled "**TypeArt**" comes up.*
This is the master control setting up all the machining parameters for 3D models cut.

- *First thing to set is the tool choice.*



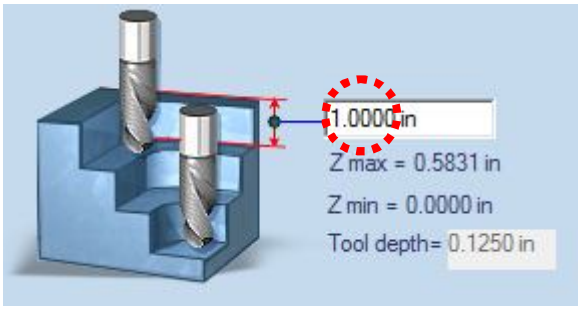


*The tool with the question mark button gives direct access to the **tool database**. Chose the Unit between **Metric** and **Inches** by expanding the base using the **+** sign on the left of the groups.*



- *Double click the tool desired and set then the cut parameters as explained below.*

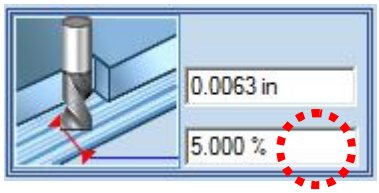
*The **step** field allows the definition of the quantity of materials that will be removed in one step. This sets the tool position for as many steps needed to cut the complete height of the 3D model.*



***Zmax** shows the Maximum Height of the 3D model from the Z zero of the machine.*

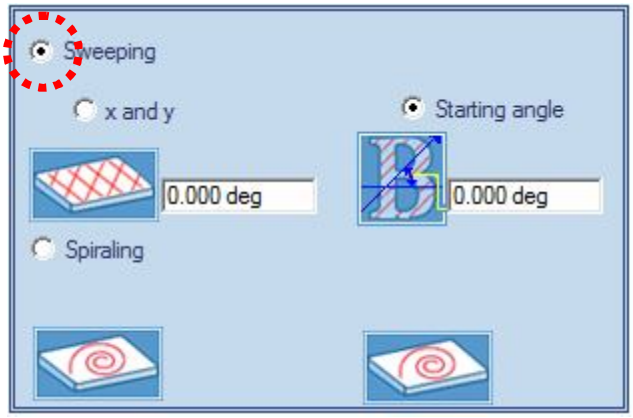
***Z min** shows the position of the bottom of the 3D model from the Z zero of the machine.*

***Tool depth** reminds the value entered in the tool database for the **Max Depth of Cut** (see **How to define a tool?**).*



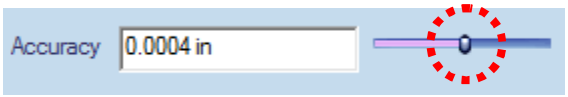
*The **Stepover** (or **Overlap**) can be set at a percentage of the tool diameter or at a sweeping gap. The smaller the Stepover is the smoother the Surface finish will be. On the other hand increasing the number of steps will result in a longer machining code and also a longer machining time. It is important then to adjust this setting according to the surface finish to reach and the overall time desired to be spent to achieve it.*

*3 toolpath generation modes can be set. The **Follow 3D profile** will insure the whole surface of the 3D model will be covered by the tool, while managing a multi steps computation and insuring a tool efficiency to cut the design in the most optimized number of steps.*



***Sweeping** will cover the 3D model from top to bottom; changing the **Starting angle** will change the direction of the sweeping accordingly.*

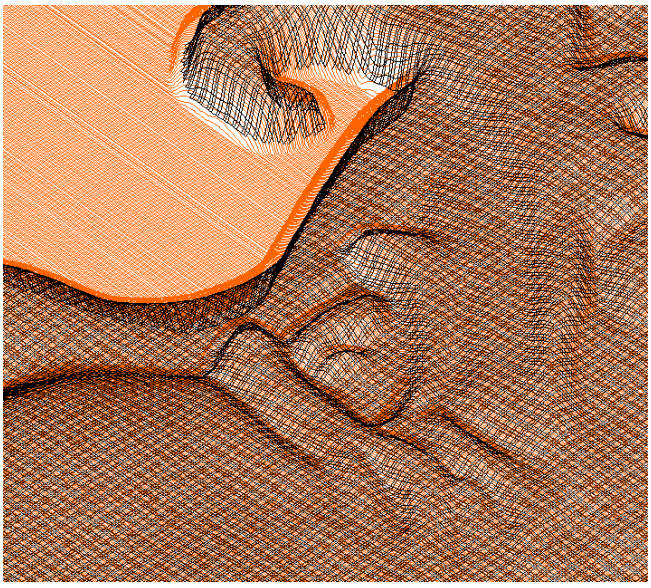
*The **Allowance** can offer the capability of leaving an **overthickness** all over the 3D model allowing managing a roughing, semi finishing, and finishing passes if needed. Be careful when using this feature with multiple tools and no tool changer on the machine, the Z needs to be set for each individual tool, see the Tip section later on explaining the process further.*

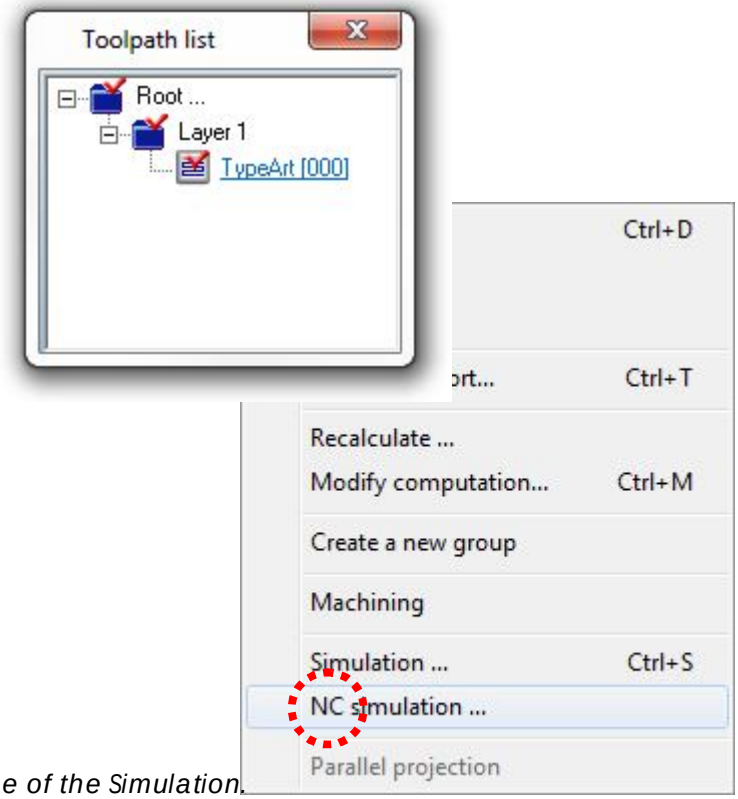


*The Accuracy is not to be changed except if it is needed to play on the number of points the toolpath generates. It is kept most of the time at **0.0004in** insuring the best balance between accuracy and machining time.*

- *Click the tick sign to validate the computation of the toolpath*  *.*

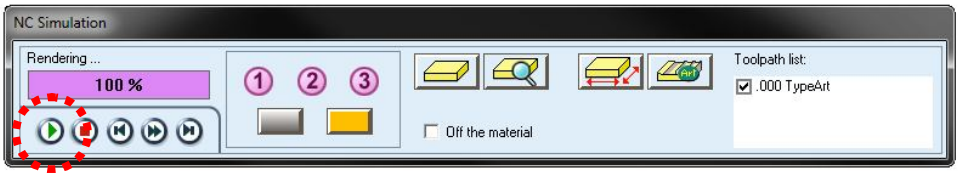
The toolpath is being computed live. Once generated it will appear in the toolpath list window that can be open and close from the following icon:

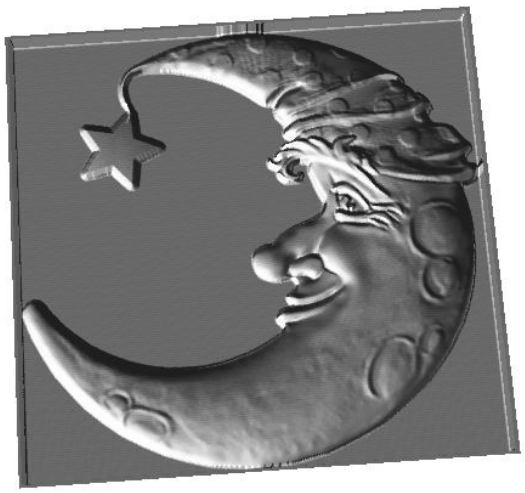


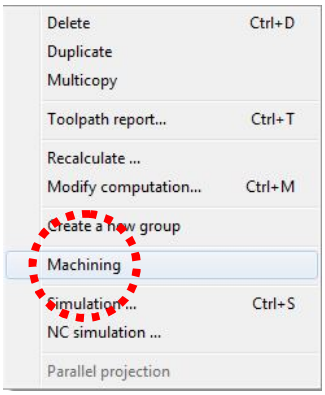


The toolpath computed can be then checked in one of the Simulation.

- *Right click the Toolpath and from the sub menu select the **NC simulation...** command.*
- *Click the Play button the toolpath is shown in a block of material pointing any imperfection that may appear on the machine allowing correcting prior machining.*

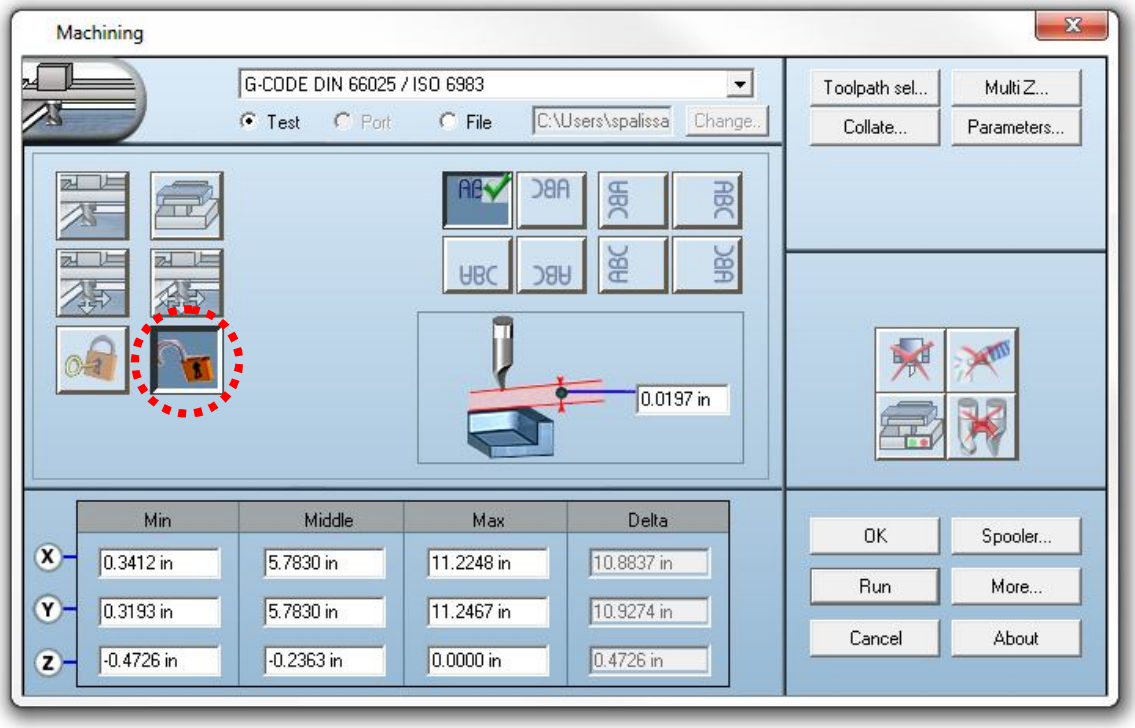


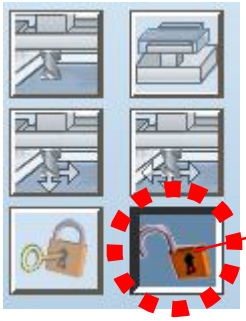




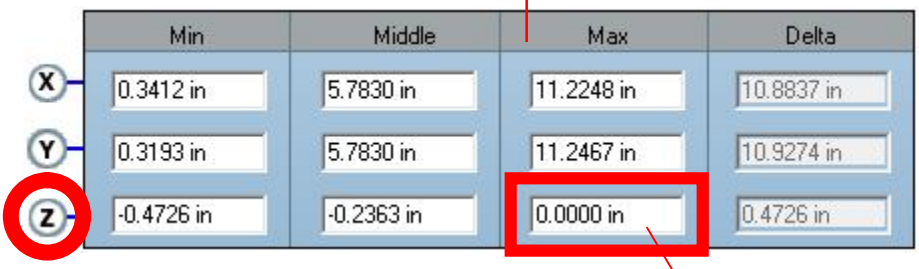
- *Close the Simulation window from the Cross close command, and right click one more time on the toolpath to access its sub menu.*
- *Select the "**Machining**" command.*

A new window pops up which prepares the toolpath processing to the machine.





- *Make sure the lock is on its opened position.*
  This allows the access to the lower Table defining the position of the toolpath within the machine area.



- *In the **MAX** column enter **0** for the **Z**. This will move the toolpath within the material (sinking it).*

The other fields of the table set the toolpath's position within the machine area.

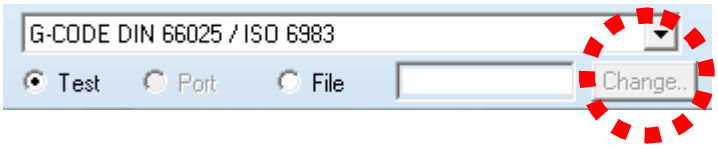
J.Tip!

All the 3D models imported from VectorClip3D are set with the background at zero.
*This means all the 3D models will be imported all at the same unique Z position. While generating the toolpath, the Z motions are being created **above the Z=0**. To machine completely in the material the Z motions must be below zero level (top of the material). By lowering the MAX Z position you will insure the path is within the material range definition.*

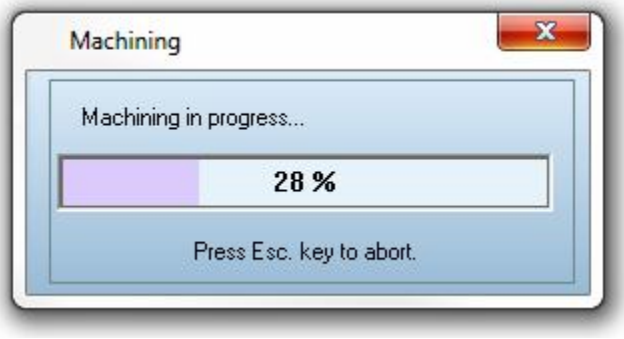
It happens also that the Top Surface of the Material is not completely flat. If this is the case rather than surfacing flat with an additional toolpath prior running the 3D path, it is possible to go a little bit deeper in the material from its top. To do this, just enter in the Z max a negative value such as -0.020 in or more as it would be needed so the toolpath start below 0 at a comfortable depth.

Also, very important to know is the need of lowering the toolpaths when you do a multi tool machining using the Allowance option. For each individual toolpath you will have to enter the Z negative position where it starts from the Z zero top of material. This can be easily determined by subtracting the allowance from the Top of the material. Z start of machining = Z top of material – Z top of the 3D model + Allowance entered.

*To output the machine file it is needed to define a defined path. By clicking on the **Change** button a path can be set allowing retrieving the machining file at a desired location on the hard drive, or through the network.*

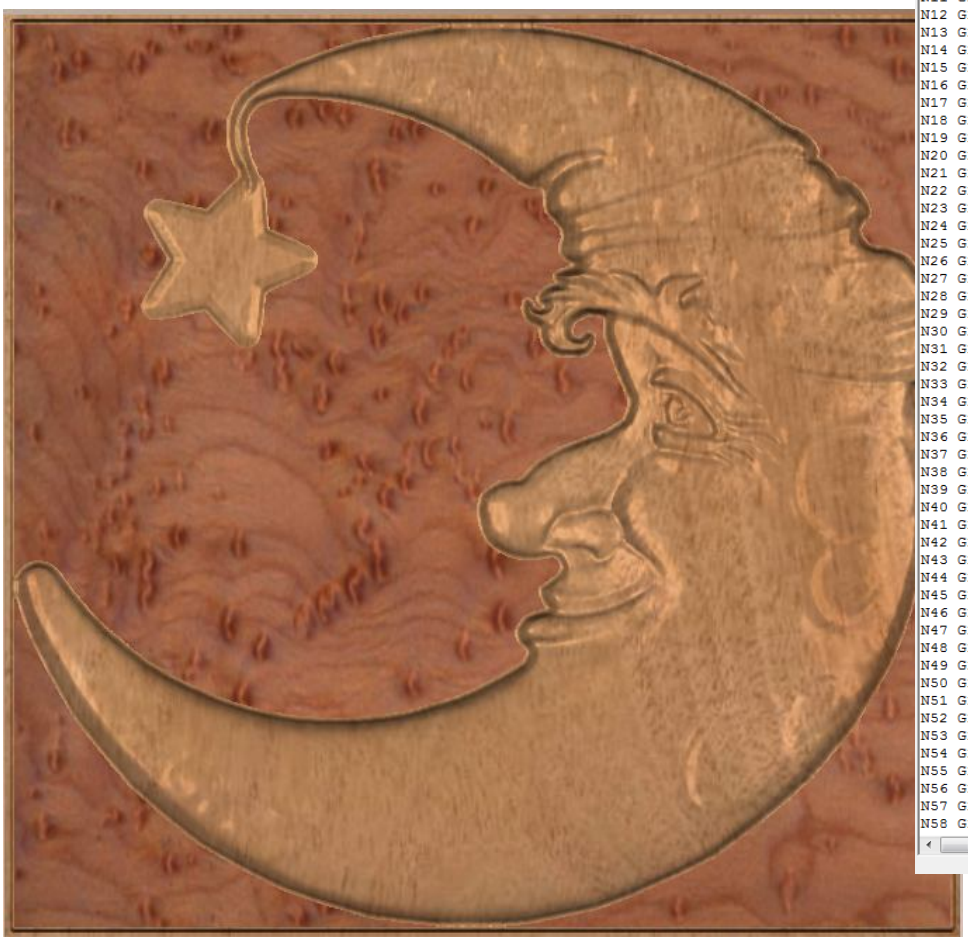


- *When all of the above is set as explained you ready to click on the **Run** button.*



The toolpath is then "postprocessed" in the language of the machine at the location as defined within the Machining table window.

```
%1
N2 T1 M6
N3 ( S19750)
N4 M8
N5 G0 X357.602 Y362.734 Z50.000
N6 G0 X357.602 Y362.734 Z0.200
N7 G1 X357.602 Y362.734 Z-14.809
N8 G1 X485.445 Y362.734 Z-14.809 F2000
N9 G1 X485.545 Y362.734 Z-14.795
N10 G1 X485.745 Y362.734 Z-14.756
N11 G1 X485.845 Y362.734 Z-14.683
N12 G1 X486.045 Y362.734 Z-14.519
N13 G1 X486.245 Y362.734 Z-14.369
N14 G1 X486.345 Y362.734 Z-14.299
N15 G1 X486.546 Y362.734 Z-14.209
N16 G1 X486.646 Y362.734 Z-14.171
N17 G1 X486.846 Y362.734 Z-14.129
N18 G1 X486.946 Y362.734 Z-14.109
N19 G1 X487.146 Y362.734 Z-14.075
N20 G1 X487.246 Y362.734 Z-14.060
N21 G1 X487.546 Y362.734 Z-14.024
N22 G1 X487.746 Y362.734 Z-14.002
N23 G1 X488.046 Y362.734 Z-13.972
N24 G1 X488.346 Y362.734 Z-13.938
N25 G1 X488.646 Y362.734 Z-13.908
N26 G1 X488.846 Y362.734 Z-13.885
N27 G1 X488.946 Y362.734 Z-13.867
N28 G1 X489.146 Y362.734 Z-13.819
N29 G1 X489.246 Y362.734 Z-13.771
N30 G1 X489.446 Y362.734 Z-13.653
N31 G1 X489.547 Y362.734 Z-13.580
N32 G1 X489.747 Y362.734 Z-13.427
N33 G1 X489.947 Y362.734 Z-13.316
N34 G1 X490.047 Y362.734 Z-13.263
N35 G1 X490.247 Y362.734 Z-13.191
N36 G1 X490.347 Y362.734 Z-13.160
N37 G1 X490.547 Y362.734 Z-13.117
N38 G1 X490.647 Y362.734 Z-13.098
N39 G1 X490.847 Y362.734 Z-13.065
N40 G1 X491.047 Y362.734 Z-13.036
N41 G1 X491.247 Y362.734 Z-13.009
N42 G1 X491.547 Y362.734 Z-12.972
N43 G1 X491.847 Y362.734 Z-12.938
N44 G1 X492.147 Y362.734 Z-12.904
N45 G1 X492.548 Y362.734 Z-12.859
N46 G1 X492.648 Y362.734 Z-12.846
N47 G1 X492.848 Y362.734 Z-12.817
N48 G1 X492.948 Y362.734 Z-12.795
N49 G1 X493.148 Y362.734 Z-12.745
N50 G1 X493.348 Y362.734 Z-12.684
N51 G1 X493.448 Y362.734 Z-12.653
N52 G1 X493.748 Y362.734 Z-12.586
N53 G1 X493.948 Y362.734 Z-12.543
N54 G1 X494.048 Y362.734 Z-12.524
N55 G1 X494.248 Y362.734 Z-12.492
N56 G1 X494.348 Y362.734 Z-12.478
N57 G1 X494.548 Y362.734 Z-12.454
N58 G1 X494.648 Y362.734 Z-12.444
```

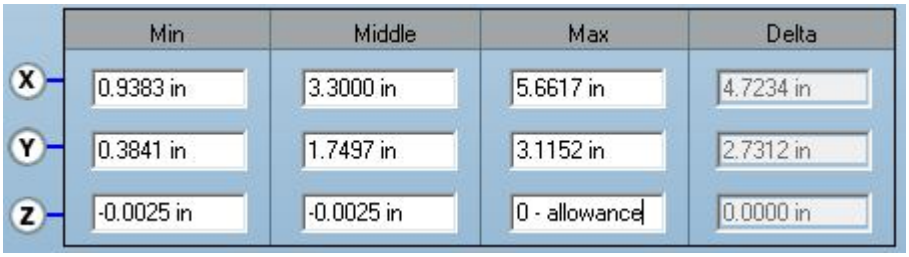


*(This above rendering is obtained from the **Advanced NC Simulation**. Hold **SHIFT** key while starting the **NC Simulation**)*

Frequently Asked Questions

I created a 3D path with a defined allowance, and it seems my CNC is cutting above the piece when I run the 2nd toolpath. Why is that?

When you create allowances on 3D path, it is similar as creating an offset in 3 directions (XYZ). This 3D offset leaving material all around the design needs a secondary toolpath to be generated; usually a smaller tool can remove the extra material still in place. However you no longer machine from the same Z top origin, your design has sunk of the 3D size in Z + the allowance set. This allowance needs to be entered in the Z max as a negative value so the CNC starts the machining at the proper height, the one left by the previous tool.



How do you set several material colors in the Advanced NC Simulation?

The Advanced NC Simulation has a second tab named Preferences where you can set a Main material and a 2nd color for all the cut made at the same depth.