



Unleash your Router's 3D Capabilities


How to manage multi-tools programming in ArtClip3D?

It might become handy to set tool numbers activating an Automatic Tool Changer from ArtClip3D in order to reduce the tool change cycles and optimizing the overall machining time . This supposes your CNC machine has an Automatic Tool Changer. If this is not the case this tutorial is still interesting since it shows how to organize your toolpaths manually and also explain the output logics towards your machine.

- **Setting up a tool database with Tool numbers assignments**

Go directly to the CAM module.



Click the tool database icon  and set if not in the list already a **Conical 20 Deg Tip: 0.015in** tool and an **1/8 End Mill** tool with the respective number 1 and 5. Right click the existing tools or create new ones.

Expand the list by clicking on the (+).

Right click the tool and Edit it.

In the **Tool Editor**, the **Technology** tab shows the **Tool index** field. The number entered in this field will be postprocessed to the machine as **M6 T5 D5***. Make sure - if the machine controller deals with Tool correction - that the **Corrector** in the Turret tab shows the same number as the tool index.

The screenshot shows three overlapping windows from the ArtClip3D software. The background window is the 'Tools database' showing a tree view of tool categories like 'INCH Tools', 'Aluminum', 'High Density Ure', 'Plastic', 'Steel', 'Wood', and 'Used tools'. The middle window is the 'Tool editor' in the 'Technology' tab, showing fields for 'Reference', 'Supplier', 'name', and 'Adjustment' (Manual/Stand). The 'Tool index' field is highlighted with a red dashed circle and contains the number '5'. The foreground window is the 'Tool editor' in the 'Turret' tab, showing a turret diagram with 10 tool slots and a table below it. The table has columns for 'Tool index', 'Correct', 'Max length', and 'Used length'. The 'Correct' column for the first row is highlighted with a red dashed circle and contains the number '5'. At the bottom of the Turret tab, there is a checkbox labeled 'Index = Corrector' which is checked.

	Tool index	Correct	Max length	Used length
1	5	5	3937007874015.7480 in	0.0000 in
2	1	1	3937007874015.7480 in	0.0000 in
3	1	1	3937007874015.7480 in	0.0000 in
4	1	1	3937007874015.7480 in	0.0000 in
5	1	1	3937007874015.7480 in	0.0000 in
6	1	1	3937007874015.7480 in	0.0000 in
7	1	1	3937007874015.7480 in	0.0000 in
8	1	1	3937007874015.7480 in	0.0000 in
9	1	1	3937007874015.7480 in	0.0000 in
10	1	1	3937007874015.7480 in	0.0000 in

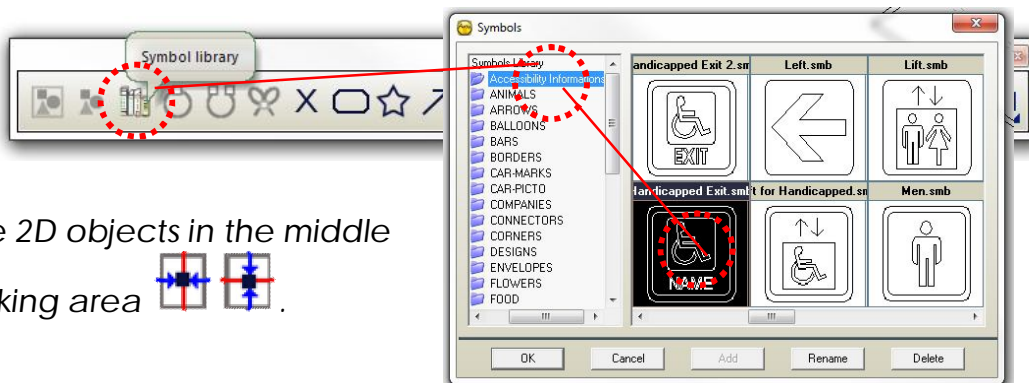
(*M6 T5 D5 code corresponds to the standard G-Code for Tool Calls)

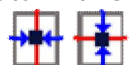
J.Note: When all the tools located in your carrousel (tool changer) have the right tool number assigned, then you can start using the tool changer knowing that your tools will be called accordingly to the tool index set in the ArtClip3D database. Let's practice that!

- **Setting up multi toolpaths for a Sign.**

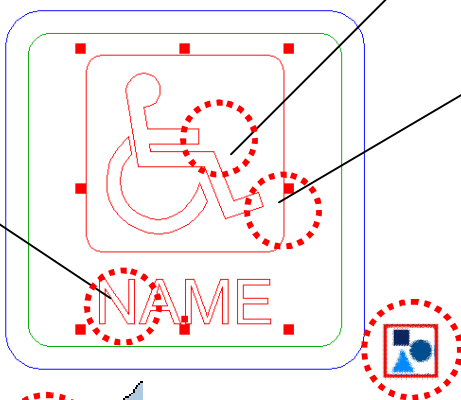


From the 2D module, open the favorites tool  and import from the **“Symbol Library”** icon the Handicapped Exit.smb file from the **“Accessibility Informations”** directory.



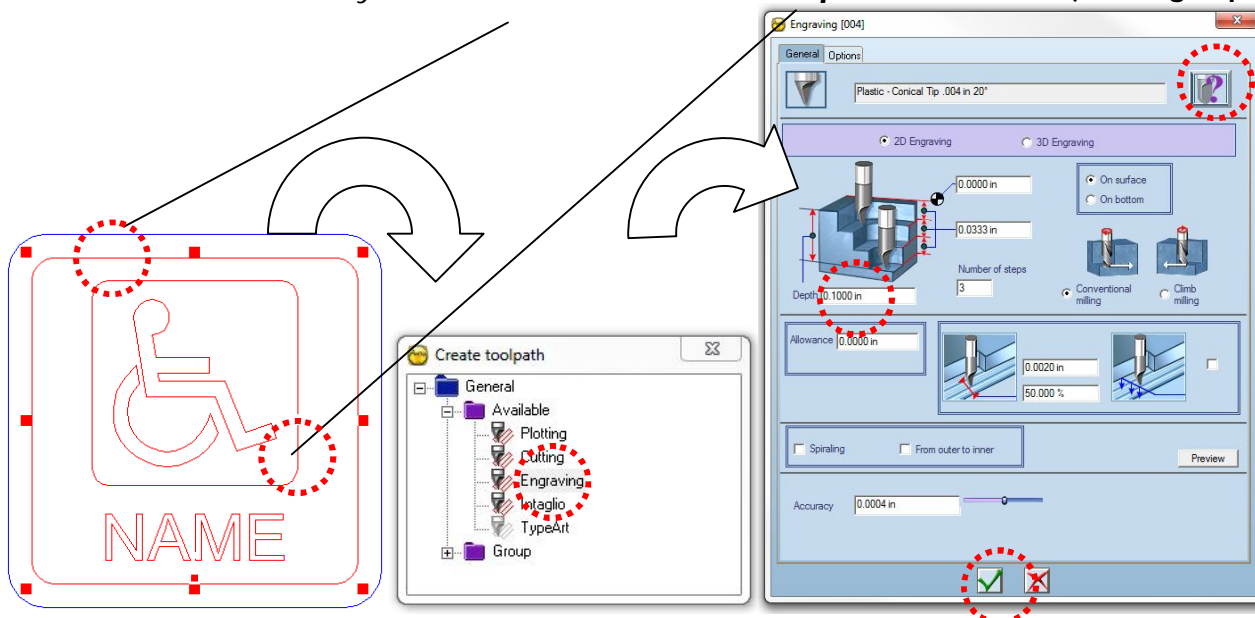
Center the 2D objects in the middle of the working area .


Ungroup the 2D Model imported and regroup it lighter with the **chair + its bounding box + the text NAME** as show below in red:



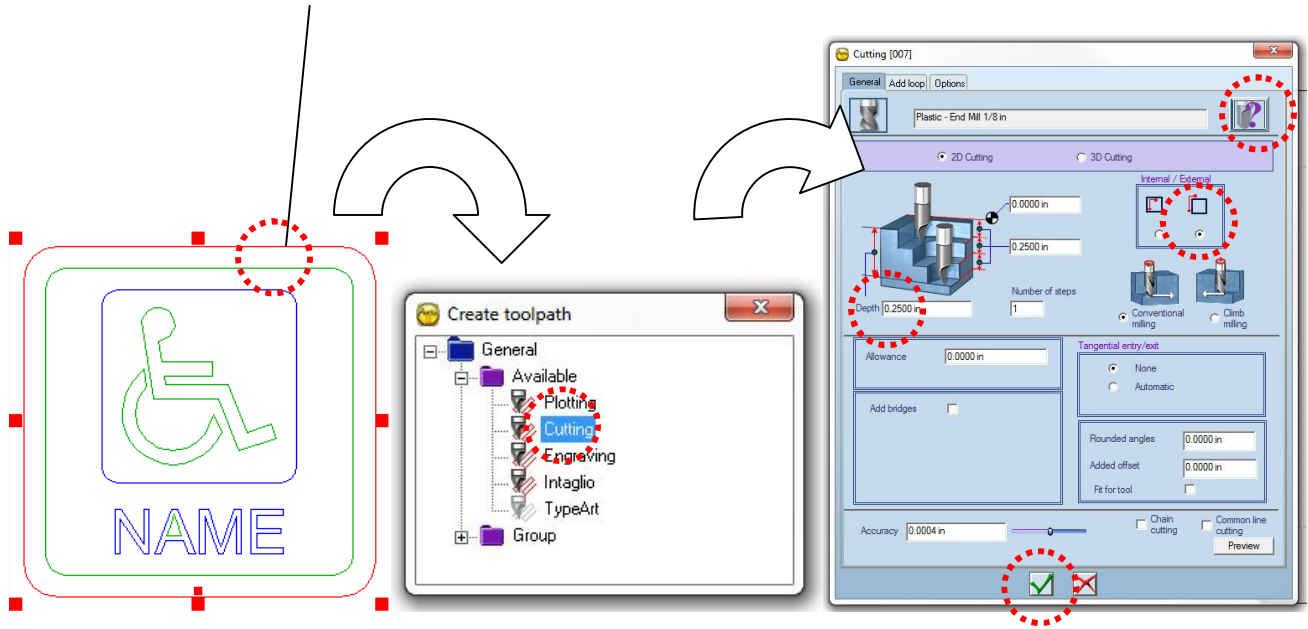
Go then to the **Cam** module. 

Select with the **CTRL** key the **2nd outline** and the **Group** made earlier. (Do not group them)



In the **Create toolpath** windows double click the **Engraving** toolpath and set it as shown above. If the windows is not shown, click on the  **“Create toolpath”** icon.


Select then the **outer limit** of the 2D model and double click the **Cutting** toolpath.



Once the toolpaths are generated you check them in the NC Simulation.

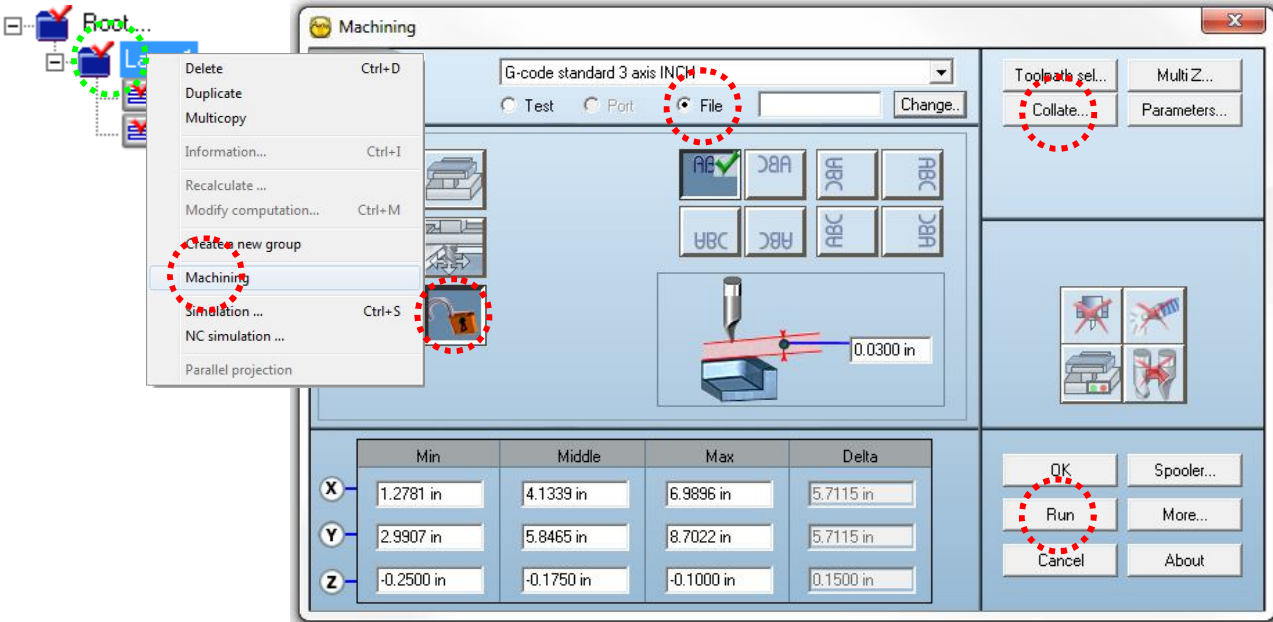
Right click the **Layer 1** and select the **NC Simulation** command as shown below.



Run the simulation by clicking the Play button . This allows you to make sure that the paths created will give you the expected result on the machine. Once simulated you can zoom or rotate the result. When done, close the window.



Right click then the **Layer 1** and select the **Machining** command.



The **Collate** button is the button setting up the way to output a program towards CNC machines with Automatic Tool Changer capability or not. In the 2 G-Code outputs shown below we can notice the 2 tool numbers which are different. This output may not work with a machine without tool changer. It is important in that case to make sure the tools are all showing the number 1.

```

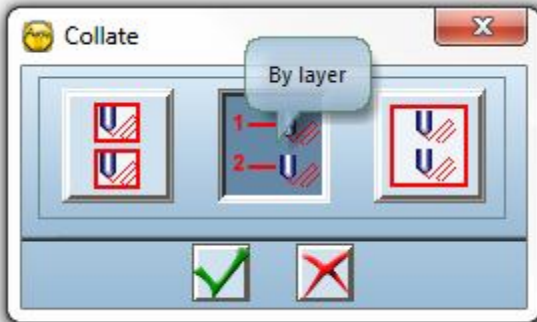
G0 G90
T2 M6
S15000 M3
G0 Z0.8 F79.
G0 X0. Y0.
X1.894 Y8.702
Z0.05
G1 Z-0.25 F20.
X6.373 F59.
G2 X6.99 Y8.086 I0. J-0.616
G1 Y3.607
G2 X6.373 Y2.991 I-0.617 J0.
G1 X1.894
G2 X1.278 Y3.607 I0. J0.616
G1 Y8.086
G2 X1.894 Y8.702 I0.616 J0.
G0 Z0.4
X0. Y0.
M2
    
```

```

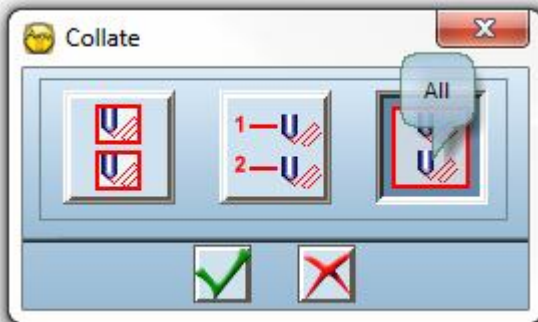
G0 G90
T1 M6
S16500 M3
G1 Z0.8 F24.
G1 X0. Y0.
G0 X3.738 Y4.122
Z0.05
G1 Z-0.1 F39.
X3.737 F79.
X3.738 Y4.123
Y4.122
X3.76 Y4.107
X3.716
X3.738 Y4.166
X3.76 Y4.107
X3.782 Y4.092
X3.694
X3.738 Y4.208
X3.782 Y4.092
G0 Z0.03
X3.477 Y7.404
    
```

The Collate mode set on the **None** Mode will send **one** toolpath **by one** toolpath. This mode has to be used for Machine without Automatic Tool Changer. For machines having the Automatic Tool Changer, then the next modes work with added options. Click the **RUN** button to output your program.

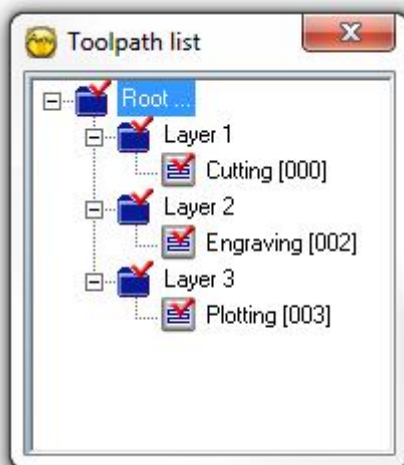
Further information:



The **"By layer"** Mode will automate the output to 1 program file only per Layer used. You can have as many toolpaths generated within the layer, only the toolpaths present on this layer will be output to the machine controller. If you have an automatic tool changer then the program output will manage the tool change accordingly to the tool index entered and will execute a Stop when the layer is finished to machine.



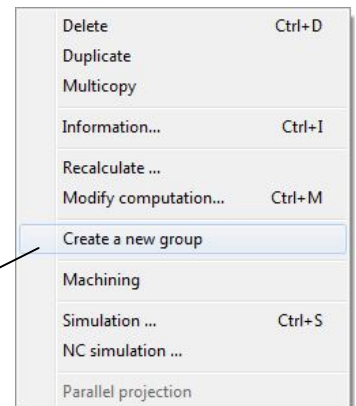
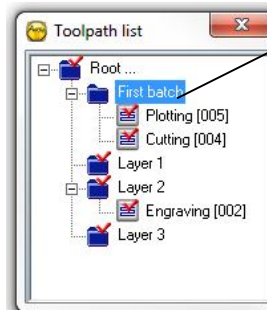
The **"All"** Mode will gather as many as Layers and toolpaths under the root selected and will output only 1 program file to the machine. This is the mode to use if an Automatic Tool Changer is available on the CNC machine. The program will have all the layers with all the toolpaths and will not stop between layers.



If the **Root** is right clicked then the complete tree will be sent to the machining.

If the **Layer1** is right clicked, only the toolpath present under this path will be sent to the machine.

You can add a special group in which you want to add only toolpath you want to machine separately of others. Right click for that in the tree and select the **"Create a new group"** command.



J.Tip!: It might become handy to limit the display of the toolpath when those ones are constantly refreshed by the computer. To stop any refreshing of the display hold the **ESC** key on your keyboard. If you want to stop re-displaying the paths you can double click on the File icon to uncheck the mark, and un-display the toolpath on screen, speeding up then the action on screen.