I. Safety warning:

1. Improper use of the CNC engraving machine can cause serious health risks and risk of damage to the machine.
2. If you are using the CNC engraving machine, you are responsible for the safety of all people and equipment affected by using the machine.
3. Only persons with proper training and up-to-date knowledge of the safe operation procedures should use the CNC machine.
4. If you are in doubt about the safe and proper operation, you MUST ask for assistance before operating the machine.
5. Failure to operate the CNC machine in a safe and responsible way might result in serious consequences such as being held liable for damages and being banned from using the machine in the future.

II. Experimental Safety Precautions:

1. Replace water inside the tank of cooling system before each use.
2. Turn on the computer, software and then open the spindle controller to avoid damaging machine.
3. Simulate cutting tool path without installing cutting tool. Make sure there is no problems before installing cutting tool.
4. Operator has to stay with the machine during machining.
5. Don’t connect the computer to internet to avoid virus.
6. Remove the water pump from the tank if the CNC engraving machine is not in use.

III. Purpose of the experiment:

Use CNC engraving machine properly.

IV. Experiment principle:

Computer controlled engraving process.
V. Experimental equipments:

1. Monitor: (x 1)

2. Computer: (x1)
3. Spindle controller: (x1)

4. Pumping motor (Put in the water tang during machining):
5. Water tank: (x1)

6. High-speed spindle x 1 and platform x 1:
7. Vise × 4 and wrench x2

VI. Experimental setup:
1. Connect spindle controller to computer.
2. Connect water pump to controller.
3. Plugin the power of computer, monitor and spindle controller.
4. Make sure water level in water tank is higher than the pump.

VII. Experimental procedure:
1. Sign the user record book.
2. Turn on computer.
3. Open simulation software CIMCO Edit V5. Figure 1 appears. File->load the NC program, press the path simulation icon.

Figure 1
4. Once path simulation starts, Figure 2 appears. Left hand side of the window displays NC code, the right hand side shows machining path. Control buttons are at the bottom right. Press the playback, NC code runs, and check the machining path.

![Figure 2](image1)

5. Activate Mach3Mill software by double click Mach3Mill icon, and Figure 3 appears. Be sure to start Mach3Mill before turning on the spindle controller to avoid sudden movement of machining tool. (See appendix 1 for more info of use of Mach3Mill).

![Figure 3](image2)
6. Turn on the controller (at the rear of the controller case) as shown below, check if water in the water tank is circulating (Do not use if the water is not circulating to avoid over heating of spindle may and damaging spindle).

![Figure 4](image)

Figure 4

7. Press Reset, use the left and right keys on keyboard to control the X-axis motion, the up and down keys to control the Y axis motion, Page Up and Page Down keys to control the Z-axis motion.


![Figure 5](image)

Figure 5
9. G-code shown in the program window and the tool path shown in Figure 6.

![Figure 6](image)

10. Before installing tool, Z-axis must have a safety height and put a soft pad below the tool holder to protect tool if tool drops accidentally. If cutting depth is 3 mm, Tool bottom must be at least 5 mm above the platform bottom (cutting depth + 2 mm = height of tool bottom above the platform to avoid tool hitting the platform). The X and Y axis must be located at the center of the platform to avoid tool hitting the limit switch.

- Press Reset to set position 0 for X, Y and Z axis, respectively.
- Press 開始 (Start) to start path simulation.
- Press 暫停 (Suspend) to stop simulation temporarily and press 開始 to restart simulation from where the NC code is suspended.
- Press 停止 (Stop) to stop NC code.

11. Open up acrylic lid and use vises to fix the workpiece on the platform as shown.

![Figure 7](image)
Install the milling tool on tool holder of the spindle.
12. Use the keyboard up, down, left and right to move spindle to the machining origin of the workpiece, then use Page Up and Page Down keys to move the Z-axis down to contact the machining surface.

13. Press Reset zero X, zero Y and zero Z, to set the machining origin.

14. Close acrylic cover. Then move tool up without contact with the workpiece.

Press RUN on the controller (as shown in Figure 8), the spindle rotates (Use the black knob to adjust frequency of controller. The relationship between spindle speed and frequency of controller can be found in Appendix II).

![Spindle Speed Display](image)

**Figure 8**

15. Once spindle starts rotating, Press Run to start machining. The operator must stay with the machine. Do not leave the machine untended. Always try lower spindle speed first before using high spindle speed. Contact experienced users for suggestions on spindle speed.

16. Once machining complete, press STOP on the spindle controller to stop the spindle.

17. Using the keyboard on, down, left, right, Page Up and Page Down keys to move the spindle to its original state.

18. Turn off the spindle controller and computer.

19. Remove the workpiece and the tool. Place vise and wrench in the original drawer.

20. First use brush to clean machine and platform and then use vacuum to clean up machine and platform. Make sure to vacuum the interior of the platform to remove cutting leftovers (chips).

21. Close the acrylic cover of machine.

22. Remove the water pump from the tank if the CNC engraving machine is not in use.

23. Sign off the user record.
Appendix I.

Window of Mach3

1. 系统视窗：Switch between various monitor windows (Just use the window after running the program)
2. 程式视窗：Loaded NC program shown here. You can check the program line by line.
3. 座標軸視窗：View relative coordinates and machine coordinates
4. 圖像路徑視窗：After loading NC code, machining path is shown here.
5. 操作視窗：Operate machining.
   5.1 開始：Start running loaded program.
   5.2 暫停：Suspend at the line of the program, click start to run NC program again.
   5.3 停止：Stop machine, Click 開始 run NC program from the beginning
5.4 編輯程式碼：Edit NC program and save.

5.5 關閉程式碼：Close NC program.

5.6 載入程式碼：Load new NC program.

5.7 重頭開始：Start NC program from the beginning.

5.8 單行執行：Run NC program one line.

5.9 刪除此行：Delete selected code.

6. 進給率視窗：Stop feed rate (Adjust feed rate from the slide bar).

7. 重置：Reset system, click this button to restart system and connect to controller.
## Appendix II

<table>
<thead>
<tr>
<th>Frequency</th>
<th>rpm</th>
<th>Frequency</th>
<th>rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>600</td>
<td>210</td>
<td>12600</td>
</tr>
<tr>
<td>20</td>
<td>1200</td>
<td>220</td>
<td>13200</td>
</tr>
<tr>
<td>30</td>
<td>1800</td>
<td>230</td>
<td>13800</td>
</tr>
<tr>
<td>40</td>
<td>2400</td>
<td>240</td>
<td>14400</td>
</tr>
<tr>
<td>50</td>
<td>3000</td>
<td>250</td>
<td>15000</td>
</tr>
<tr>
<td>60</td>
<td>3600</td>
<td>260</td>
<td>15600</td>
</tr>
<tr>
<td>70</td>
<td>4200</td>
<td>270</td>
<td>16200</td>
</tr>
<tr>
<td>80</td>
<td>4800</td>
<td>280</td>
<td>16800</td>
</tr>
<tr>
<td>90</td>
<td>5400</td>
<td>290</td>
<td>17400</td>
</tr>
<tr>
<td>100</td>
<td>6000</td>
<td>300</td>
<td>18000</td>
</tr>
<tr>
<td>110</td>
<td>6600</td>
<td>310</td>
<td>18600</td>
</tr>
<tr>
<td>120</td>
<td>7200</td>
<td>320</td>
<td>19200</td>
</tr>
<tr>
<td>130</td>
<td>7800</td>
<td>330</td>
<td>19800</td>
</tr>
<tr>
<td>140</td>
<td>8400</td>
<td>340</td>
<td>20400</td>
</tr>
<tr>
<td>150</td>
<td>9000</td>
<td>350</td>
<td>21000</td>
</tr>
<tr>
<td>160</td>
<td>9600</td>
<td>360</td>
<td>21600</td>
</tr>
<tr>
<td>170</td>
<td>10200</td>
<td>370</td>
<td>22200</td>
</tr>
<tr>
<td>180</td>
<td>10800</td>
<td>380</td>
<td>22800</td>
</tr>
<tr>
<td>190</td>
<td>11400</td>
<td>390</td>
<td>23400</td>
</tr>
<tr>
<td>200</td>
<td>12000</td>
<td>400</td>
<td>24000</td>
</tr>
</tbody>
</table>