








1. Turn on control box power switch

SHEETCAM

2. Open SheetCAM
3. In Parts panel: right-click, New part, choose DXF file, Open, OK
4. Select nesting tool , choose and right-click part, Array parts: create multiples, OK
5. Select toolpath tool 
6. In Operations panel: select jet cutting operation tool 
7. Choose appropriate cut parameters, OK
8. Using start point tool , make changes to your start points if needed
9. Click post processor  at top left to output G-Code

MACH3

10. Open Mach3
11. Click 'ESTOP', when solid green click 'Home X/Y/Z'
12. Click 'LOAD G-CODE', find '.tap' file made in SheetCam
13. Ensure either 'Auto Volt Set' is on or 'Set Volts In' is set correctly
14. Make sure consumables are correct for cut & material
15. Make sure plasma is turned on and set to correct amperage
16. Drive torch over material to start point, press blue 'X' button on controller or 'Zero All' on screen
17. Click 'Run' or press green 'A' button on controller to start cut

