



Plasma CNC Power-On Checklist

Checklist Frequency: when powering machine from an dormant condition

	Procedure	Remarks
1	CNC Plug-and-Play Controller: Set Red power lever to “on” position	CNC Controller Powered
2	CNC PC: Connected to CNC Controller (Ethernet)	
3	CNC PC: Mach software loaded with correct machine profile	
4	Machine: Verify all E-Stop switches are unlocked	Twist handle to unlock a locked E-stop switch
5	CNC PC: Click on red “Reset” button to clear emergency stop condition	If Emergency Stop condition persists, verify e-stop switches and limit switches are not active
6	CNC PC: Check TMC communications lights are green	TMC communications lights are found on Mach3 Program Run screen.
6	CNC PC: Verify machine moves using the jog keys (arrow keys)	Be aware of current machine position before jogging
7	Plasma Torch Power Supply: Verify compressed air line is connected and pressurized	See your plasma torch manual for air pressure and flow rate requirements
8	Plasma Torch Handle: If equipped with a torch lock, Verify torch lock set to “ready to fire” position.	Applies to powermax 45 XP machine torches. Note: Torch will need to be test fired once after switching from locked to ready-to-fire to prime torch.
9	Plasma Torch Power Supply: Set power switch to “on”.	Machine is now ready!



Plasma CAM Checklist

Checklist Frequency: When defining every new job (program)

	Procedure	Remarks
1	SheetCAM: Create new or Open your plasma “.job” file	
2	SheetCAM: Import vectors for this job as “new part”	Verify your new part is selected
3	SheetCAM: Create or edit existing tool to verify that Kerf width, Feed rate, Pierce delay, Pierce height, and Cut Height match settings found in your “Hypertherm torch operator manual” for your material type and thickness	Refer to Cut Charts and Marking Charts in torch manual
4	SheetCAM: Create new “Plasma cut” Operation	Operation > Plasma cut
5	SheetCAM: Choose Contour Method	Outside offset is typical for jobs with mix of closed/open profiles
6	SheetCAM: Select Layer that contains the vectors you want to cut	
7	SheetCAM: Enter feed rate based on your material type and thickness	Refer to Cut Charts and Marking Charts in torch manual
8	SheetCAM: Configure lead-in and lead-out settings	Edit individual start points which do not conform to global lead in lead out requirements
9	SheetCAM: Press OK and verify cutting preview shows a clockwise motion around your outer part profiles	enable Reverse Cut Direction if cutting preview needs to be reversed
10	SheetCAM: Run the Post Processor and create your gcode file	Ensure you use CNC Router Parts Ohmic



Normal Plasma Operations Checklist

Checklist Frequency: When running a job (program)

	Procedure	Remarks
1	Machine: Load Material onto water table and attach grounding clip	Ensure grounding clip is directly attached to your material and has a solid connection with bare/exposed metal
2	CNC PC: Program Target Voltage (Mach3 program run screen) based on material selection and target cut height	Refer to Cut Charts and Marking Charts in torch manual
3	Plasma Power Supply: Verify cutting amperage setting	Refer to Cut Charts and Marking Charts in torch manual
4	CNC PC: Verify THC (Torch Height Control) is enabled	THC button on program run screen should be green
5	CNC PC: Load program (g-code) using "Load File"	
6	Torch Mount: Verify torch mount is fully engaged with magnetic alignment features in base plated	
7	CNC PC: Zero X, Y on the left/front corner of where you want your job to start	
8	CNC PC: Zero Z with torch approximately .25" above work surface	
9	CNC PC: Press "Cycle Start" to begin cutting program	Note; wear appropriate eye protection. See your torch manual for proper shade level