



Must Do Steps

This is just a simple ordered necessary steps to cut something. This pdf is not intended to cover all the many aspects of cnc plasma tables, just a quick cheat sheet for new users.

- 1. Draw part in any cad program that can save file as a DXF or SVG.**
- 2. Import drawing in to sheetcam**
- 3. define operations in order you want to cut them. Making sure to use the correct "tool" for what you are cutting.**
- 4. run the post processor, and give your new file a name**
- 5. turn control box on, both ac mains power switch and front panel motor power.**
- 6. start control program**
- 7. Take out of reset**
- 8. Home z then x then y axis**
- 9. move torch via the jog keys to your desired 0,0 point on your material**
- 10. lower torch down till it is within 1" or closer to material**
- 11. zero x y and z axis**
- 12. open program generated in sheetcam**
- 13. Turn plasma cutter on, verify you have no error codes on plasma cutter (remember to turn air on!) amperage is set to the tool you used in sheetcam, and correct consumables for that amperage are installed.**
- 14. Attach work clamp to material being cut.**
- 15 Click run!**