晶研科技
JINGYAN TECHNOLOGY

JYH04手柄控制器使用说明

Instruction Manual for JYH04 Handheld Controller

适用C、M、F、V系列雕刻机 | ZH | EN

202510221.7A

使用雕刻机前，请阅读附带说明书中的安全注意事项，当心触电。

Before using the engraving machine, read the safety precautions in the supplied manual and beware of electric shock.

目录 Content

一、前言

⚠WARNING 使用设备前，请仔细阅读使用说明书，充分了解设备的相关功能，以确保正确使用雕刻机，防止意外事故的发生。若未遵守安全注意事项和使用说明可能发生触电、火灾等其他危险。请妥善保管好说明书，以便随时查阅。

1.1 安全说明

1.1.1 工作区域安全准则

◆请始终保持工作区域整洁有序、光照充足。杂乱或昏暗的环境易引发意外事故，所有工具与物料应定点存放，及时清理废料与碎屑，确保通道无障碍物。

◆严禁在易燃、易爆环境下操作设备，包括但不限于存在易燃液体、气体、粉尘（如木屑、金属粉、有机颗粒等）的场所。设备运行可能产生电火花或高温，存在燃爆风险。工作区须配备防火设施，并严禁烟火。

◆操作期间，所有无关人员——尤其是儿童及旁观者——须与设备保持安全距离。操作者应集中注意力，严禁与他人交谈、使用手机或从事任何可能分散注意力的行为，以防失控引发事故。

◆工作区应具备良好的通风条件，及时排出加工产生的烟雾与粉尘。同时应确保设备周边留有足够的操作与避险空间，避免在狭小或封闭环境中运行设备。

1.1.2 设备用电安全准则

◆雕刻机必须使用与电源插座规格完全匹配的原装插头，严禁以任何形式修改、拆除或转接设备原有插头和接线。所有电气连接须确保可靠接地，禁止使用无接地功能的插头或插座。推荐在供电回路中安装漏电保护器（GFCI/RCD），以显著降低触电风险。

◆严禁在潮湿、淋雨或易接触液体的环境中操作或存放设备。水滴或湿气进入设备内部可能引发严重触电事故。操作时，务必保持手脚及工作环境干燥，避免身体直接接触接地物体（如金属架构、水管等），以防形成回路增加电击危险。

◆禁止拉扯、缠绕电线或通过拖拽电线的方式移动设备或拔插插头。电源线应远离热源、油污、锐利边缘及设备运动部件，防止绝缘层破损或断线。定期检查电缆状态，如出现老化、裂痕、变形或过热现象应立即停用并联系专业人员进行更换。

◆非专业人员不得拆卸、改装控制箱及电机部分。任何电气维修和部件更换都必须由具备相应资质的电工完成。如发现设备漏电、异常发热、发出焦味、电弧或断路器频繁跳闸，请立即切断电源，联系售后服务，严禁带故障运行。

◆建议在设备供电回路中设置独立的过流和短路保护装置。若工作环境存在潜在液体溅射风险，应额外使用防水插座及线缆防护套。长时间不使用时，请断开设备与电源的全部连接。

1.1.3人员安全准则

◆请务必保持高度警惕，严格遵守操作规范，并在使用雕刻机前掌握基本使用常识。

◆在疲劳状态，或曾饮酒、服用药物及可能影响判断力的药品后，严禁操作雕刻机。操作过程中的任何疏忽，哪怕只是片刻，都可能导致严重的人身伤害。

◆为确保安全，请始终正确使用个人防护装备，操作期间必须佩戴护目镜，防止碎屑溅入眼睛；根据作业环境，合理选用防尘口罩、防滑安全鞋、安全帽及听力保护装置等，以有效降低伤害风险。

◆接通电源前，确认开关处于“关闭”状态，防止意外启动；及时移除主轴电机旋转部位上的扳手、刀具等物品，避免其飞出造成伤害。

◆操作时请始终保持稳定站姿和身体平衡，确保在突发情况下能迅速控制设备。

◆应穿着合适的工作服，避免宽松衣物、首饰等；长发应束起，并远离设备运动部件，防止被卷入。

◆采取有效的除尘措施，减少粉尘吸入和爆炸风险，并定期检查设备状态，确保所有安全防护装置完好可用。

◆严禁未经培训的人员操作雕刻机，日常应加强安全演练与风险教育，全面提升事故预防能力。

1.1.4 使用安全准则

◆请勿强行安装雕刻机。安装时应选用适当的电动工具，规范的工具不仅有助于提高安装效率，更能保障安装过程安全可靠。

◆注意经常维护雕刻机，检查移动部件是否错位或绑定、部件断裂以及可能影响雕刻机操作的任何其他情况。若设备已损坏，一定要将雕刻机维修好才可使用。许多事故都是由于对设备缺乏维护引起的。

◆单台设备建议由一人主导操作，他人如需协助或观察，需保持在安全距离外，避免多人同时操作引发误碰。

◆更换下来的废旧、破损刀具应放入专用的耐割容器集中处理，严禁随意放置，防止划伤。

◆任何检修、维护、清洁或调整（包括更换刀具）前，必须严格执行“上锁挂牌”程序，即切断所有能源（电、气），并挂上警示牌，防止他人误启动。

◆若电源开关无法正常启闭，请立即停止使用该设备。机器失控属于严重安全隐患，必须及时报修并由专业人员检修。在进行任何调整、更换配件或存放设备之前，务必拔掉电源插头。这一预防措施能有效避免设备意外启动。

◆闲置的雕刻机应存放于儿童无法接触的区域，严禁未阅读并理解说明书的人员操作设备。未经培训的使用者操作雕刻机极易引发危险。

◆请定期对雕刻机进行维护保养，重点关注运动部件是否错位、卡滞或存在断裂等异常情况。一旦发现损坏，必须维修完好后方可继续使用。许多事故源于设备缺乏日常维护。

◆保持刀具锋利和清洁十分重要。维护良好的锋利刀具不易断裂，切削更顺畅、更安全。

◆操作前请充分评估加工环境、材料特性及设备能力，严格遵循说明书指引。不规范的操作可能引发严重事故。此外，请在光线充足、通风良好的环境中作业，确保工作区域整洁、无杂乱物品干扰操作视线和动线。

请参考以上安全说明。在使用雕刻机、配件、刀具等工具时，请仔细考虑工作环境和需要加工的产品。若未按照本说明手册操作，可能导致危险发生。

1.2 产品简介及特点

1.2.1 产品简介

JYH04手柄系统是一款高性能三/四轴手持式脱机运动控制系统。该系统采用先进的ARM+FPGA双核处理器架构，具备强大的运算能力和高稳定性，支持2-4轴直线插补与任意2轴圆弧插补，最大脉冲输出频率达500kHz。系统具备丰富的I/O接口，包括16路输入和8路输出，支持模拟量主轴、多段速主轴及伺服主轴等多种主轴控制方式，并集成自动对刀、断点续雕、软限位保护等实用功能。其操作界面简洁直观，支持图形仿真、多坐标系切换、网络文件共享及U盘脱机加工。适用于广告雕刻、木工加工、模具制造等多种数控应用场景，是一款功能全面、易于集成、可靠性高的工业级运动控制解决方案。

1.2.2 产品特点

◆最大控制轴数：四轴, 2-4 轴直线插补、任意 2 轴圆弧插补；

◆ 用户通过U盘实现系统与外部文件交互，完全脱机工作；

◆兼容标准的G代码，支持主流的 CAD/CAM 软件，如 ArtCam、MasterCam、ProE 等；

◆多段预处理，加工轨迹的自适应速度前瞻控制，加工速度快、精度高、加工连续性好；

◆小线段连续高速度加工，在多种小线段控制算法中自动选择效率最高的算法；

◆支持超大容量的文件加工；

◆集电极开路输出，最大可输出电流500mA，可直接驱动继电器；

◆连接网络可支持文件共享及远程文件在线加工；

◆跳段执行加工功能，按照指定加工行号进行加工以及就近点加工功能，支持大文件快速定位； 暂停断点、断电断点功能及载入断点功能；

◆脉冲方向采用差分输出和双脉冲模式，最大插补脉冲输出频率500KHz；

◆A 轴支持循环编码器功能；

◆XYZA 四轴偏置(微调)功能；

◆输入端口可任意分配功能，如驱动器报警等；

◆多 I/O 点控制器方式，基本 I/O 分别为 16 输入、8 输出;

◆语言：采用国际编码，支持基本所有语系文字，客户可以自定义语言包；

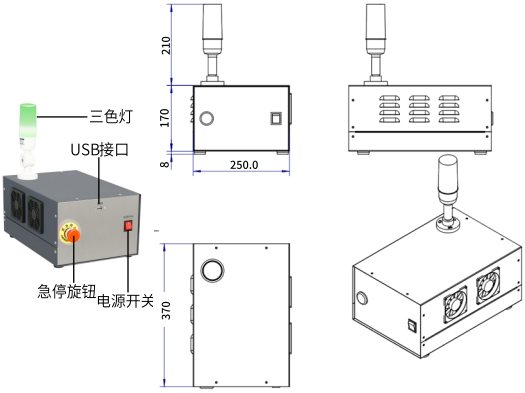
◆加减速方式：S曲线加减速；

二、控制器规格与接线

2.1 控制箱及手柄的结构

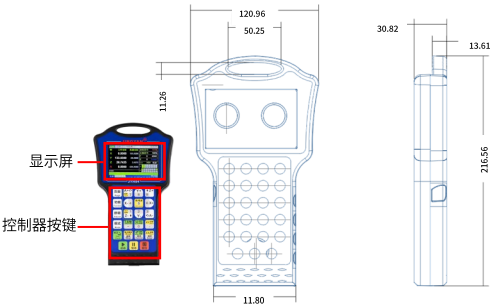
2.1.1 控制箱结构示意图

控制箱结构示意图如下：



2.1.2 手柄结构示意图

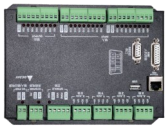
手柄的结构示意图如下：



2.2 用电器规格与接线

2.2.1 用电器规格

控制箱内部用电器规格参数如下表：

序号	用电器名称	规格	图片
1	运动控制卡	输入：AC220V 3.5A AC110V 5.5A (定制) 输出：+24V 10A 输出功率：250W 频率：50Hz / 60Hz	
2	变频器	输入电压：AC 220V 输出电压：0~220V 输出电流：7.0A 额定功率：2.2KW 相数：1PH 频率：50Hz / 60Hz	
3	控制箱	电压：AC 220V 功率：1.2KW（800W机型） 2.5KW（2.2KW机型）	
4	开关电源	输入：AC220V 3.5A AC110V 5.5A(定制) 输出：+24V 10A 输出功率：250W 频率：50Hz / 60Hz	

*务必使用符合国标交流电源插座！（建议使用带有接地保护的插座，以确保使用安全；避免与其他大功率电器共用同一电路，以防过载。）

2.2.2 航空线接线

航空插头用于实现雕刻机的电气连接，不同功能接口需对应匹配不同规格的航空插头。X、Y、Z、A四轴分别连接至对应轴的驱动电机，以控制各轴向的移动功能。限位开关与机床X、Y、Z轴的限位装置相连（系统采用XYZ轴原点开关作为限位信号源，配置为3个硬限位与3个软限位），用于实现行程保护功能。主轴马达接口连接主轴驱动线路，用于控制主轴旋转动作。水冷系统接口连接并控制主轴水冷单元的运行状态。对刀接口为扩展功能接口，可连接对刀仪，实现刀具的自动对刀操作。

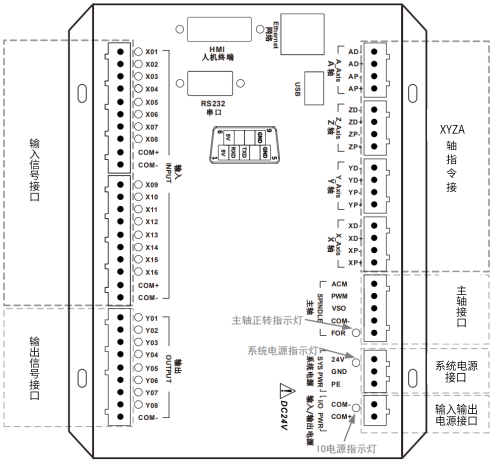
*接线备注：将雕刻机本体的航空插头线缆，依照对应标识与控制箱上的插口一一对应连接（A轴与对刀器均为选配产品，若未购买，则无需接线）。再将电源线接入控制箱的220V输入插口处。如有网络通信需求，请连接以太网端口。

航空线接线说明如下表：

端子定义 / 名称 / 引脚号	XYZA轴（7芯）	限位开关（8芯）	主轴马达（5芯）	水冷系统（4芯）	外部开关（4芯）	对刀仪（8芯）
1	24V+	X-	U	12V+	复位	24V+
2	PUL+	X+	V	GND	停止	/
3	PUL-	Y-	屏蔽网	24V+	开始	/
4	DIR+	Y+	W	GND	GND	/
5	DIR+	Z+	PE			对刀信号
6	GND	Z-				24V-
7	ALM	24V+				对刀超程
8		GND				24V-

2.2.3运动控制卡接口定义

运动控制卡接口示意图如下：



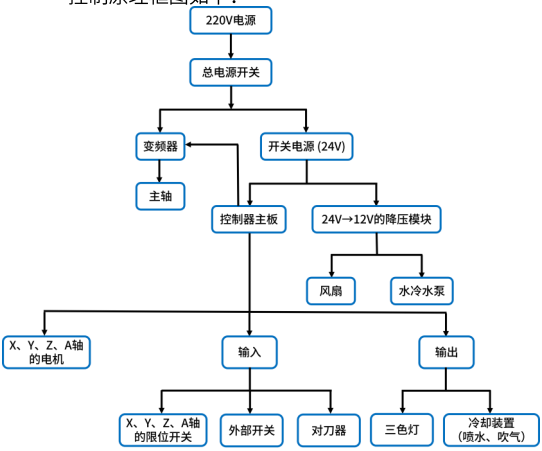
接口信号说明：

符号	种类	名称	功能	标准
PE	外壳	外壳地	接到大地上	
24V	系统电源	外部直流电源24V “+”极	1) 用户需自备的直流电源，将直流电源的正极接此端子。 2) 不可与COM+ COM-共用电源。	DC24V 3A
GND		24V 电源“−”极	直流电源 24V “−”极	
COM+	输入输出电源	外部直流电源24V “+”极	1) 用户需自备的直流电源，将直流电源的正极接此端子。 2) 不可与24V GND共用电源。	DC24V 3A
COM-		24V 电源“-”极	直流电源24V “-”极	
XP+	X轴指令	脉冲正差分信号	X 轴脉冲差分输出	线驱动输出。RS422标准插补脉冲最大。1MHZ。
XP-		脉冲负差分信号		
XD+		方向正差分信号	X 轴方向差分输出	
XD-		方向正差分信号		
YP+	Y轴指令	脉冲正差分信号	Y 轴脉冲差分输出	
YP-		脉冲负差分信号		
YD+		方向正差分信号	Y 轴方向差分输出	
YD-		方向正差分信号		
ZP+	Z轴指令	脉冲正差分信号	Z轴脉冲差分输出	
ZP-		脉冲负差分信号		
ZD+		方向正差分信号	Z轴方向差分输出	
ZD-		方向正差分信号		
4P+	4th轴指令	脉冲正差分信号	4th轴脉冲差分输出	
4P-		脉冲负差分信号		
4D+		方向正差分信号	4th轴方向差分输出	
4D-		方向正差分信号		

符号	种类	名称	功能	标准
X01 … X16	通用输入	可参数配置	通过参数配置成驱动器报警、零点、正负限位、对刀、外部急停以及功能扩展键；	支持机械、光电、接近等开关输入； 接近开关：24V类型：NPN有效电平：0V
Y01 … Y08	通用输出	可参数配置	通过参数配置成润滑，冷却等输出	集电极开路输出 内置反向二极管驱动电流500mA 驱动电压 30V
FOR	主轴开关量控制	主轴正转或启停输出	与主轴正转端子连接（若只有启停则只需接此端口即 可）	集电极开路输出 内置反向二极管驱动电流500mA 驱动电压 30V
COM-	主轴输出公共端	COM-	禁止与ACM短接	/
VSO	主轴模拟量	模拟量输出	与变频器的模拟量输入端口连接。 注意:禁止与 DCM 和 COM 短接。	0-10V模拟量
ACM		模拟量地		
PWM	PWM输出	PWM输出	与变频器的 PEM 输入端口连接。 注意:禁止 COM短接	占空比
ACM		PWM地		

三、控制原理框图

控制原理框图如下：



四、控制器功能说明

4.1 面板按键功能

面板按键的排列图如下：

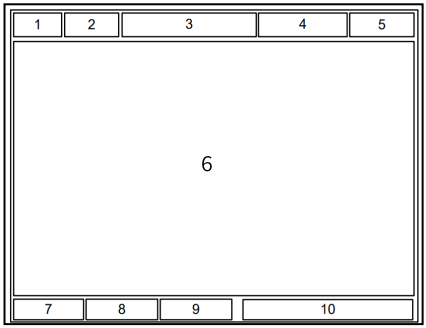
按键功能说明：所有操作都可通过按键实现，按键功能介绍如下表格。

按键图标	功能
页面 Page	主控界面、文件界面、参数界面的切换键。
功能 Function	进入功能操作菜单列表。
修调 Adjust	进给倍率、主轴倍率、进给速度（F）、主轴速度（S）、工件坐标系（G54~G59）、刀长值补正（H00）、步长等的切换键。
模式 Mode	连续移动与寸步（点动）移动模式的切换键。
A+ 倍率 % 7	A 轴正向移动; 输入数字“7”；当前焦点数据递增。
3 A-	A 轴负向移动; 输入数字“3”。
Y+ 8	Y 轴正向移动; 输入数字“8”；向上移动。
2 Y-	Y 轴负向移动; 输入数字“2”；向下移动。
% 倍率 1 Z-	Z 轴正向移动; 输入数字“9”。
Z+ 9	Z 轴负向移动; 输入数字“1”；当前焦点数据递减。
6 X+	X 轴正向移动; 输入数字“6”；向右移动; 向下翻页。

按键图标	功能
X- 4	X 轴负向移动; 输入数字“4”；向左移动；向上翻页。
高/低速 5 JOG Speed	高/低速切换; 输入数字“5”。
主轴 Spindle –	主轴启动/停止; 输入符号“-”。
回工件零 Work0 0	全轴回工件零点坐标; 输入数字“0”。
XY 清零 Zero .	X、Y 轴坐标清零; 输入符号“.”。
对刀 Tool Check	执行对刀功能。
断点继续 Break 退格 Bksp	断点继续加工; 回删。
回机床零 Home 取消 Cancel	全轴回机床零点坐标；取消。
ZA 清零 Zero 确定 Enter	Z、A轴 坐标清零；确定。
启动	启动加工；暂停后再启动。
暂停	加工操作暂停。
复位	复位功能；复位与空闲状态切换。

4.2 界面显示说明

4.2.1 主界面介绍



1.当前模式（连续/寸动） 2.当前状态（空闲/运行/复位）
3.当前加工文件 4.当前加工时间
5.登录状态 6.页面内容显示
7.主控页面 8.文件页面
9.参数页面 10.网络信息、序列号、系统日期等

4.2.2 主控页面介绍

◆路径

页面→主控页面（图 1）

◆说明

①此界面为操作主要界面，手动、自动加工、归零等都在此界面操作；
②界面提供了加工中所需监控的必要信息；

◆页面说明：

1. 坐标显示
- 此区域可同时显示工件坐标和机械坐标；
- 图示当前工件坐标系为G54；

2. 当前版本信息或加工代码
- 当前程序日期；
- 运行时的加工代码程序行；

3. 参数和故障报警信息
- 当前参数信息显示；
- 具体报警内容显示；

4. 进给倍率
- 自动加工中的进给倍率；
- 倍率范围0% - 300%；

5. 主轴倍率/寸动距离
- 自动加工中的主轴倍率；
- 倍率范围0% - 150%；
- 【连续】模式下显示为主轴倍率；
- 【寸动】模式下显示为寸动距离；

6. 进给速率（F）
- 显示机床设备的实时进给速率；
- 显示设定的进给速率(蓝色字体表示系统默认的进给速率参数, 白色字体表示采用程序中的F速度)；

7. 主轴转速（S）
- 显示机台的实时主轴转速；
- 显示设定的主轴转速（蓝色字体表示系统默认的主轴转速参数，白色字体表示采用程序中的主轴转）；

8. 当前工件坐标系
- G54、G55、G56、G57、G58、G59 工件坐标系；

9. 刀具补偿号H

10. 连续移动的高/低速显示

11. 通用输出信号Y1~Y9

12. 通用输入信号X1~X16

操作菜单列表：
- 在主控页面中，按【功能Function】 键弹出（图2）操作菜单。

图1：

图2：

4.2.3文件页面介绍

◆路径

页面→文件页面（图3）

◆说明

此界面提供多种文件管理的功能，如加选择工文件、浏览文件、一键拷贝、复制、粘贴、删除、切换磁盘（本地、U盘、网络）等。

◆页面说明：

1. 程序文件
- 【确定Enter】键选中当前文件为加工文件。

2. 当前磁盘信息
- 【功能Function】键切换磁盘（功能→切换磁盘→显示本地磁盘/U盘）。

图3：

◆操作说明：
- 按方向键【Y+】、【Y-】 上下移动光标；
- 按翻页键【X+】、【X-】 进行上下页切换；
- 按【确定Enter】键，则光标所指定的文件为当前加工文件；

操作菜单列表：
- 在文件页面中，按【功能Function】键弹出（图4）操作菜单。

4.2.4参数页面介绍

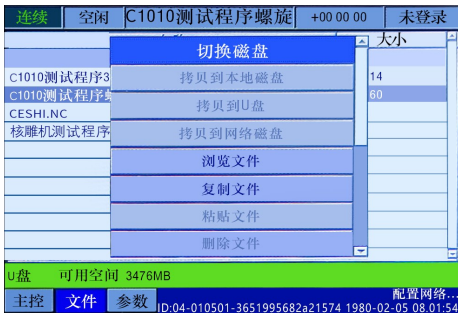
◆路径

页面→参数页面（图5）

◆说明

系统所有参数设定界面，可根据配置文件自行配置所需设置的参数。

图4：



◆页面说明：

1. 参数分类名称
2. 当前参数信息
   - 如图，“7”表示参数编号索引，“X轴电子齿轮比（分母）”表示参数名称，“1.000”表示参数值。
3. 当前选中的参数说明

◆操作说明：

- 按方向键【Y+】、【Y-】上下移动光标进行选择参数；
- 按翻页键 【X+】、【X-】进行切换参数类别；按【确定Enter】键键，则光标所指定的参数切换到录入模式，按数字键录入参数值，按【确定Enter】键键，则当前编辑的框数值更新到当前输入的参数值；若按【取消Cancel】键，则表示放弃当前参数修改。

操作菜单列表：

- 在参数页面中，按【功能Function】键弹出（图6）操作菜单。

图5：

图6：

五、设备操作流程

5.1设备开机与调试

①开机前检查机器所有接线、外接器设备是否正常，检查正常后顺时针旋起急停旋钮，按下电源按键将机器通电，等待系统界面加载完成。

②载入CNC系统后会提示“是否返回机床原点？”，点击操作系统中的【确定】按键返回机床原点。

③通过操作系统中的【X-】、【X+】、【Y+】、【Y-】、【Z+】、【Z-】、【A+】和【A-】或通过手轮来控制相应轴的移动，检测相应轴的移动功能是否正常。

④移动各轴至机器行程的最大值和最小值，检测机器的行程限位保护功能是否正常。

⑤按下操作系统中主轴开关来控制主轴的启停，检测主轴功能是否正常。（注意主轴上是否装夹刀具，如有装刀，务必确保刀具夹稳，防止刀具飞出造成伤害。）

⑥开机后检查主轴水冷风扇、水冷水泵工作是否正常启动。

5.2 定工件零点

刀具装夹至主轴上且已固定好工件后，可用手柄上的XYZ正反移动键来控制机器的正反向移动，可将机器主轴移动到编程设定的加工坐标原点。为方便定工件原点，编程时一般将加工原点设定在材料的左下角或材料中心位置。

下面以工件零点在材料左下角为例：先将主轴移动至安全高度，快速移动各轴（倍率X100或X10）至程序设置的工件原点附近（距离工件原点2-5mm），接着，缓慢移动各轴（倍率）靠近工件原点，将要接触工件原点时，点击寸动模式，且设置较小的寸动距离值（如0.01），刀具轻微碰到工件原点或转动螺帽时有微小材料切削，则可进行“清工件零”操作。至此工件零点确定完成。

分中功能也可用于定工件零点，具体操作可参考6.2.5。

本小节中的具体操作按键因操作系统不同有部分差异。

5.3 加载程序

找好工件原点后,选择需要加工的程序，将加工程序拷贝到U盘上，将需要加工的代码保存至系统本地磁盘，再加载至控制系统。

5.4 开始加工

加工程序加载好后，点击【启动】按键，机器便会开始运行程序。

六、设备常用功能

6.1 回机械零

由于加工所需的各项坐标设定都是依照机械原点为基准，所以每次控制箱开机后，都需再次确认机械原点的位置，开机后必须执行回机械零动作。

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【主控页面】

6.1.1 全部轴回机床零

◆操作流程

1）点击【回机床零Home】快捷键
2）全部轴将按照顺序移动到各轴零点开关位置
3）归零后各轴自动回退到一个固定距离（参数可设置）
4）默认回机床零顺序为：Z-X-Y-A（参数可设置）

6.1.2 单个轴回机床零

◆操作流程（以X轴为例）

1）依次点击【功能Function】按键→【回机械零】→【X轴回机械零】→【确定Enter】按键
2）系统X轴将按照归零方向参数设置的方向移动到零点开关位置
3）归零后X轴自动回退到一个固定距离（参数可设置）
4）其他轴单独回机床零操作一致

*备注：回机械零的过程中软限位无效，软限位需机械坐标系建立后生效。

6.2 清工件零

清工件零功能主要是用于设定工件坐标原点，工件零点是编程和加工的基准（通常为G54~G59坐标系的原点）。通过清零操作，将机床坐标系中的某一点设定为工件坐标系的原点，后续所有加工程序的坐标均基于此点计算。

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【主控页面】

6.2.1 工件坐标系

系统提供G54-G59六种工件坐标系，自动加工前需确定程序加工的工件坐标系。

◆操作流程

1）循环点击【修调Adjust】按键切换焦点至坐标系G54-G59
2）点击【倍率%+】和【倍率%-】 按键循环切换G54-G59工件坐标系，坐标位置显示跟随当前坐标系刷新

6.2.2 设定XYZ工件原点

在加工文件之前，用户通过手动操作调整刀具与工件的位置，以便从工件预定位置开始加工，设置工件原点前需选择正确的工件坐标系。

◆操作流程

1）将X、Y、Z轴手动移动到预定的加工起点位置
2）依次点击【功能Function】按键 →【清工件零】→【全轴清零】→按【确定Enter】键可以将当前位置的X、Y、Z轴的坐标值清零
3）也可点击【XY清零Zero】 和【ZA清零Zero】进行当前位置的X轴、Y轴和Z轴的坐标值清零

6.2.3 设定单个轴工件原点

◆操作流程（X轴为例）

1）将X轴手动移动到预定的加工起点位置
2）依次点击【功能Function】按键 →【清工件零】→【X轴清零】→输入数值“0”→按【ZA清零Zero】键可以将当前位置的X轴的坐标值清零
3）其他轴单独回机床零操作一致

6.2.4 工件原点保存与载入

该系统可保存并载入多个工件坐标原点功能。若需要记录多个工件坐标原点进行加工，要在保存该工件原点前，将XYZ轴移动至工件原点处。

◆操作流程（X轴为例）

1）依次点击【功能Function】按键→【多工件原点】→【保存工件原点】→按【确定Enter】键保存该原点

2）依次点击【功能Function】按键→【多工件原点】→【载入工件原点】→按【确定Enter】键载入该原点

以上步骤为1个工件坐标原点的设定，多个工件原点可参考此步骤依次设定，设定的工件坐标必须与加工程序的工件坐标一致。

6.2.5 分中

分中功能一般是用来寻找工件的XY的中心，也是设置工件原点的常用功能，以工件的中心作为加工的起始点，故需使用刀具触碰工件的边缘。分中需考虑刀具直径对位置确定的影响。一般使用直铣刀直接进行分中操作。若使用尖刀，需注意刀具直径随高度（变化引起的直径差可通过Z轴值控制刀具在工件两侧边缘的同一个高度进行分中）。控制器得到工件边缘的机械坐标后会自动将中心机械坐标计算求出，并将此机械坐标设入当前工件的坐标系中（G54-G59）。

本系统支持的分中方式是“两点分中”。

◆操作流程（X轴分中为例）

1）将X轴手动移动至远离工件的其中一侧，再将其进行对刀操作（此操作可参考5.4.1）。
2）将X轴工件坐标清零：依次点击【功能Function】按键→【清工件零】→【X轴清零】→输入数值0，按【确定Enter】键可以将当前位置的X轴的坐标值清零
3）将X轴手动移动到工件的另一侧，再进行对刀操作（此操作可参考5.4.1）。
4）执行分中：依次点击【功能Function】→选中【X分中】→按【确定Enter】键，则分中完成
5）Y轴分中与X轴操作一致

6.3 回工件零

回工件零功能是指将刀具或机床轴快速移动至当前运行程序的工件坐标系原点（即G54~G59等指令设定的零点）。此位置是基于机床坐标系偏移而来的、为特定加工零件单独设定的编程基准点，通常位于工件的某个角或几何中心。执行此操作的目的是建立刀具与待加工工件之间的相对位置关系，从而确保加工程序能根据正确的基准轨迹运行。

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【主控页面】

◆操作流程（X轴为例）

1）依次点击【功能Function】按键 →【回工件零】→【X轴回零】→按【确定Enter】键将回到X轴坐标原点的位置
2）其他轴回工件零与X轴回零的操作一致（⚠注意：为确保加工安全，若工件原点设置有误，Z轴寻零易产生危险，故此功能需谨慎操作，全轴寻零也需慎重！）

6.4 对刀

对刀分为手动对刀和自动对刀。这两种对刀方式的首次对刀，都要通过试切法找到刀具与工件参考面的接触点，即面板控制移动各轴至工件原点处并设定工件原点（对工件原点清零）。

6.4.1 手动对刀

当距离工件原点坐标较远时，可用快速移动的模式移动至原点附近；将要靠近原点时，可用寸动模式（设置寸动距离0.01~0.02）移动至原点，用手拨动螺帽；当工件材料略有切削或感受到有阻力时，则可判断各轴的位置，从而对各轴进行清零。

6.4.2 自动对刀

Z轴自动对刀功能是通过机台上对刀器来测量不同刀号的刀尖位置。对刀器对刀时，对刀器的触发位置到工件基准平面是固定的，使用者可以将不同刀号的刀尖位置到工件基准平面的距离输入到工件坐标系统，作为加工时刀长偏移的依据，使用此功能前请确认该设备是否配有对刀器（对刀器未包含在标准配置中，客户可根据实际需求选配。如需购买，请联系本司客服或销售）。

固定对刀仪：对刀需要事先设置对刀仪的初始XYZ位置（出货前已调试好），对刀时机床自动移动到对刀仪的固定位置进行对刀。

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【主控页面】

◆操作流程

1）执行对刀：依次点击【功能Function】按键 →【对刀（探测）】→按【确定Enter】键→【固定对刀】→按【确定Enter】键即可执行对刀动作
2）对刀操作流程：ZXY轴依次移动至机械位置，接着，Z 轴开始向下探测对刀仪，检测到对刀器开关后，回退到安全距离。

◆拓展

- 固定对刀原理：固定对刀后记录的是Z轴在对刀仪位置的机械坐标(Z轴外部补偿值)。
- Z轴清零原理：Z偏置=当前Z轴机械坐标-Z轴外部补偿值(对刀处的机械位置)-其他补偿

*备注：Z轴清零和固定对刀的先后顺序为：

- 任意一把刀先进行固定对刀
- 然后将对刀过的刀具移动到合适位置进行Z轴工件清零操作
- **不可以用没有对过刀的刀具进行清零操作**
- **固定对刀操作前需先回机械零，否则无法执行**

6.5 文件加载与管理

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【文件页面】

6.5.1 加载加工文件

系统支持三种文件存储的模式，分别为本地磁盘、U盘、网络磁盘。

◆操作流程

1）切换至文件页面：按【页面Page】键切换页面，直到显示文件页面
2）切换文件所在的磁盘：在【文件页面】，按【功能Function】键→【切换磁盘】→按【确定Enter】键→选中文件所在的磁盘（【本地磁盘】/【U盘】/【网络磁盘】）
3）【Y+】、【Y-】上下键和【X+】、【X-】翻页键将光标移动至要加工的NC文件
4）按【确定Enter】键则画面切换至【主控页面】，且该程序将被指定为当前加工程序

*备注：可通过查看界面第一行的当前加工文件名来确认是否成功加载加工程序。（若加工程序显示为红色则表示该程序已失效。）

6.5.2 拷贝文件

为确保加工过程中程序运行稳定，建议将U盘的加工文件拷贝到本地磁盘，后在本地磁盘中选择需要加工的NC文件。

◆操作流程

1）按【Y+】、【Y-】上下键选中文件
2）按【功能Function】键后选中需要拷贝的目标磁盘：【拷贝到本地磁盘】或【拷贝到U盘】或【拷贝到网络磁盘】
3）按【确定Enter】键执行文件拷贝

6.5.3 复制、粘贴文件

◆操作流程

1）按【Y+】、【Y-】上下键选中文件
2）按【功能Function】键→【复制文件】→按【确定Enter】键
3）切换到其他磁盘
4）依次点击【功能Function】→【粘贴文件】→按【确定Enter】键，则会将第二步的文件粘贴到当前位置

6.5.4 删除文件

◆操作流程

1）按【Y+】、【Y-】上下键选中文件
2）依次点击【功能Function】→【删除文件】→按【确定Enter】键，则会将当前选中文件删除

6.6 参数管理

◆操作条件：

- 系统处于【空闲】状态
- 主界面切换到【参数页面】

6.6.1 参数修改

◆操作流程

1）先点击任意参数输入密码（密码：888888，由于修改参数需要输入密码，且输入密码后会返回第一页，再次选择参数修改后不需要重新输入密码）
2）三种方式选择参数：搜索参数号、参数分类表、上下左右键手动定位
3）光标选中需要修改的参数，按【确定Enter】键
4）数字键输入参数值，按【确定Enter】键保存

6.6.2 参数备份

◆操作流程

1）按【功能Function】键→【参数备份】→按【功能Function】键→选择需要备份的目标磁盘【备份到U盘】、【备份到网络 磁盘】、【备份到系统磁盘】

2）按【确定Enter】键执行备份

6.6.3 参数恢复

◆操作流程

1）按【功能Function】键→【参数备份】→【功能Function】→选择需要恢复的目标磁盘【备份到U盘】、【备份到网络磁盘】、【备份到系统磁盘】

2）按【确定Enter】键执行恢复

6.7 系统升级

系统升级有两种方法，一种为开机自动升级，一种为开机后手动升级。

◆操作流程

- 开机自动升级

1）将升级包文件夹拷贝到U盘的根目录下。

2）插入U盘并断电重启。

3）系统将自动搜索升级文件进行升级，升级过程中界面会停留在开机界面10s左右，请耐心等待不要断电。

4）开机后查看主控界面的版本日期是否与客户提供的一样。

- 开机手动升级

1）将升级包文件夹拷贝到U盘的根目录下。

2）插入U盘。

3）依次进入【参数页面】→【高级】→【系统升级】→【从U盘升级】→按【确定】键执行系统升级。

4）升级完成后，对话框会有提示“软件升级完成，请重启系统”。

5）开机后查看主控界面的版本日期是否与客户提供的一样。

*备注：

① 升级包文件夹的名称为hsys,必须放在U盘根目录。

② 文件夹名称必须为hsys。

③ U 盘的根目录下必须是hsys升级包,且hsys文件夹下就是升级包的内容。

④ 升级成功后，将hsys文件夹从U盘中删除，否则每次断电开机都会自动升级。

七、控制箱保养

电控箱作为雕刻机核心控制单元，需定期进行清洁与检查，以确保长期稳定运行。应保持箱体干燥、无尘，每周使用干燥压缩空气清理散热孔及内部积尘，防止因散热不良导致元器件过热。同时，需检查所有电源线、电机驱动线及信号线端子是否松动，发现老化破损应及时更换。电控箱应安装在通风良好、远离强磁场和高温的环境，避免阳光直射。特别注意系统电源（24V、GND）与输入输出电源（COM+、COM-）必须分别由两个独立的开关电源供电，严禁共用，并定期检查电源滤波器工作状态，防止高频噪声影响控制系统。长期不使用时，应断电并加盖防尘罩，建议每月通电运行以驱除内部潮气，保持电路板干燥。

在使用控制箱和机器过程中，内部容易积聚粉尘，尤其在加工金属材料时，金属导电粉尘一旦进入控制箱，可能引发短路或烧毁电路。操作时，可先打开控制箱外壳，使用吹风机调至冷风档或使用冷风枪，将箱体倒置，吹除电路板及元件表面的粉尘。对于不易清理的角落，可用小刷子轻柔擦拭，确保彻底清除导电尘埃。清洁完成后，确认内部完全干燥再合盖并重新启动设备。

八、代码支持

8.1 G代码

晶研雕刻机设备常用G代码识别加工，常用G代码可参考附录一。

8.2 M代码

晶研雕刻机设备也常用M代码识别加工，常用M代码可参考附录二。

I. Introduction

⚠WARNING Before using the equipment, please read the instruction manual carefully to fully understand the relevant functions of the equipment.This ensures the correct use of the engraving machine and prevents accidents.Failure to comply with safety precautions and the instruction manual may result in electric shock, fire, or other dangers.Please keep the manual properly for future reference.

1.1 Safety Instructions

1.1.1 Work Area Safety Guidelines

◆Always keep the work area clean, orderly, and well-lit. Cluttered or dimly lit areas can easily cause accidents. All tools and materials should be stored in designated locations. Clean up waste and debris promptly to ensure a clean and unobstructed area.Never misuse cables; Do not use cables for lifting, dragging, or directly unplugging the controller.

◆Strictly prohibit operating the equipment in flammable or explosive environments, including but not limited to places with flammable liquids, gases, or dust (such as wood chips, metal powder, organic particles, etc.). Equipment operation may generate electrical sparks or high temperatures, posing a combustion risk. The work area must be equipped with fire prevention facilities, and smoking or open flames are strictly prohibited.

◆During operation, all unrelated personnel—especially children and bystanders—must maintain a safe distance from the equipment. The operator should concentrate and strictly avoid talking to others, using mobile phones, or engaging in any activity that may distract attention, to prevent loss of control and accidents.

◆The work area must have good ventilation to promptly exhaust smoke and dust generated during processing. Simultaneously, ensure there is sufficient operating and evacuation space around the equipment, and avoid operating the equipment in cramped or enclosed environments.

1.1.2 Equipment Electrical Safety Guidelines

◆The engraving machine must use the original plug that exactly matches the power socket specifications. Modifying, removing, or adapting the original plug and wiring of the equipment in any form is strictly prohibited. All electrical connections must ensure reliable grounding. Using plugs or sockets without grounding function is forbidden. It is recommended to install a Ground Fault Circuit Interrupter (GFCI/RCD) in the power supply circuit to significantly reduce the risk of electric shock.

◆Strictly prohibit operating or storing the equipment in damp, rainy, or liquid-prone environments. Water droplets or moisture entering the equipment may cause serious electric shock accidents. During operation, keep hands, feet, and the working environment dry. Avoid direct body contact with grounded objects (such as metal frames, water pipes, etc.) to prevent increased risk of electric shock.

◆Do not pull, twist cables, or move the equipment or unplug by dragging the power cord. Power cords should be kept away from heat sources, oil contamination, sharp edges, and moving parts of the equipment to prevent insulation damage or wire breakage. Regularly inspect the cable condition. If aging, cracks, deformation, or overheating are found, stop use immediately and contact professionals for replacement.

◆Non-professionals are not allowed to disassemble or modify the control box and motor parts. Any electrical repairs and component replacements must be performed by qualified electricians. If equipment leakage, abnormal heating, burning smell, arcing, or frequent circuit breaker tripping is detected, immediately cut off the power, contact after-sales service, and strictly prohibit operating with faults.

◆It is recommended to install independent overcurrent and short-circuit protection devices in the equipment's power supply circuit. If the working environment has potential liquid splash risks, use waterproof sockets and cable protection sleeves additionally. When not in use for extended periods, disconnect all connections between the equipment and the power source.

1.1.3 Personnel Safety Guidelines

◆Please maintain a high level of vigilance, strictly adhere to operating procedures, and master basic usage knowledge before using the engraving machine.

◆Strictly prohibit operating the engraving machine when fatigued, or after consuming alcohol, medication, or any substance that may impair judgment. Any negligence during operation, even for a moment, may lead to serious personal injury.

◆To ensure safety, always use Personal Protective Equipment (PPE) correctly. Safety glasses must be worn during operation to prevent debris from entering the eyes; Select and use dust masks, anti-slip safety shoes, helmets, and hearing protection appropriately based on the working environment to effectively reduce injury risks.

◆Before connecting the power, confirm the switch is in the "OFF" position to prevent accidental startup; Promptly remove wrenches, cutting tools, and other items from the rotating parts of the spindle motor to avoid ejection injuries.

◆Always maintain a stable stance and body balance during operation, ensuring quick control of the equipment in case of emergencies.

◆Wear appropriate work clothes, avoiding loose clothing and jewelry; Tie up long hair and keep it away from moving parts of the equipment to prevent entanglement.

◆Adopt effective dust removal measures to reduce dust inhalation and explosion risks, and regularly check the equipment status to ensure all safety protection devices are intact and usable.

◆Strictly prohibit untrained personnel from operating the engraving machine. Conduct daily safety drills and risk education to comprehensively improve accident prevention capabilities.

1.1.4 Usage Safety Guidelines

◆Do not force the installation of the engraving machine. During installation, use appropriate power tools. The specified tools not only improve installation efficiency but also ensure the safety and reliability of the installation process.

◆Pay attention to regular maintenance of the engraving machine. Check for misalignment or binding of moving parts, broken components, and any other conditions that may affect engraving machine operation. If the equipment is damaged, the engraving machine must be repaired before use. Many accidents are caused by lack of equipment maintenance.

◆It is recommended that a single machine be operated primarily by one person. If others need to assist or observe, they should maintain a safe distance to avoid accidental contact caused by multiple people operating simultaneously.

◆Used, damaged tools that have been replaced should be placed in dedicated, puncture-resistant containers for centralized disposal. Random placement is strictly prohibited to prevent cuts.

◆Before any inspection, maintenance, cleaning, or adjustment (including tool changes), the "Lockout-Tagout" procedure must be strictly implemented. This means disconnecting all energy sources (electrical) and attaching warning tags to prevent accidental activation by others.

◆If the power switch cannot be turned on or off normally, immediately stop using the equipment. A machine that cannot be controlled by its switch poses a serious safety hazard and must be reported for repair and inspected by professionals promptly. Before making any adjustments, replacing accessories, or storing the equipment, always unplug the power plug. This preventive measure effectively avoids accidental startup of the equipment.

◆Idle engraving machines should be placed in areas inaccessible to children. Operation by personnel who have not read and understood the manual is strictly prohibited. Operating an engraving machine by untrained users is highly likely to cause danger.

◆Perform regular maintenance on the engraving machine, focusing on whether moving parts are misaligned, stuck, or have abnormalities such as breakage. Once damage is found, it must be repaired before continued use. Many accidents stem from lack of daily equipment maintenance.

◆Keeping cutting tools sharp and clean is also very important. Well-maintained, sharp tools are less likely to break and cut more smoothly and safely.

◆Before operation, fully assess the processing environment, material characteristics, and equipment capabilities, strictly following the instructions in the manual. Non-standard operations may cause serious accidents. Additionally, work in a well-lit and ventilated environment, ensuring the work area is tidy and free of clutter that may interfere with the operator's line of sight and movement path.

Please refer to the above safety instructions. When using engraving machines, accessories, cutting tools, and other equipment, carefully consider the working environment and the products to be processed.

Failure to operate according to this manual may lead to dangers.

1.2 Product Overview and Features

1.2.1 Product Overview

The JYH04 handheld system is a high-performance three/four-axis handheld offline motion control system.This system adopts an advanced ARM+FPGA dual-core processor architecture, possessing powerful computing capability and high stability, supporting 2-4 axis linear interpolation and arbitrary 2-axis circular interpolation, with a maximum pulse output frequency of 500kHz.The system has rich I/O interfaces, including 16 inputs and 8 outputs, supporting various spindle control methods such as analog spindle, multi-speed spindle, and servo spindle, and integrates practical functions like automatic tool setting, breakpoint resume carving, and soft limit protection.Its operation interface is simple and intuitive, supporting graphic simulation, multi-coordinate system switching, network file sharing, and USB offline machining.It is suitable for various CNC application scenarios such as advertising engraving, woodworking, and mold manufacturing, making it a comprehensive, easy-to-integrate, and highly reliable industrial-grade motion controller solution.

1.2.2 Product Features

◆Maximum controlled axes: Four axes, 2-4 axis linear interpolation, arbitrary 2-axis circular interpolation;

◆Users can achieve file interaction between the system and external files via USB drive, working completely offline;

◆Compatible with standard G-code, supports mainstream CAD/CAM software such as ArtCam, MasterCam, ProE, etc.;

◆Multi-segment preprocessing, automatic speed look-ahead control for machining paths, resulting in fast machining speed, high precision, and good machining continuity;

◆High-speed continuous machining of small line segments, automatically selecting the most efficient algorithm from various small line segment control algorithms;

◆Supports ultra-large capacity file machining;

◆Open-collector output, maximum output current 500mA, can directly drive relays;

◆When connected to a network, supports file sharing and remote file online machining;

◆Skip segment machining function, machining according to specified line numbers, and nearest point machining function, supports fast positioning in large files; Pause breakpoint, power-off breakpoint function, and load breakpoint function;

◆Pulse and direction use differential output and dual-pulse mode, maximum interpolation pulse output frequency 500KHz;

◆A-axis supports cyclic encoder function;

◆XYZA four-axis offset (fine-tuning) function;

◆Input ports can be arbitrarily assigned functions, such as driver alarm, etc.;

◆Multi-I/O point controller mode, basic I/O are 16 inputs and 8 outputs respectively;

◆Language: Uses international encoding, supports basically all language scripts, customers can customize language packs;

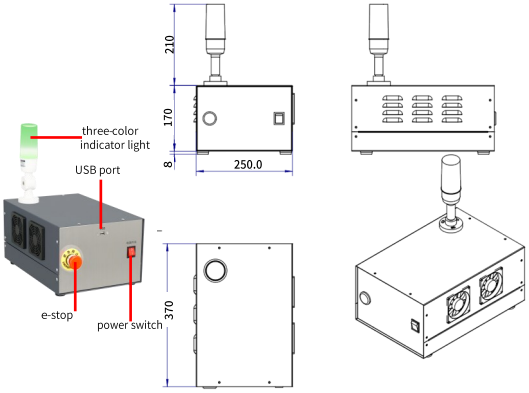
◆Acceleration/deceleration mode: S-curve acceleration/deceleration.

II. Controller Specifications and Wiring

2.1 Structure of Control Box and Handheld Unit

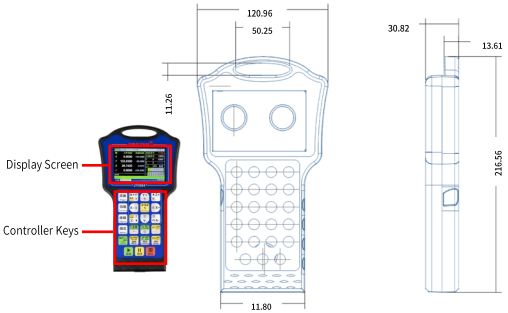
2.1.1 Control Box Structure Diagram

The control box structure diagram is as follows:



2.1.2 Handheld Unit Structure Diagram

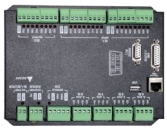
The structure diagram of the handheld unit is as follows:



2.2 Internal Electrical Components Specifications and Wiring

2.2.1 Electrical Components Specifications

The specifications of the internal electrical components in the control box are as follows:

No.	Component Name	Specifications	Picture
1	motion control card	Input: AC 110V 5.5A (Custom) AC 220V 3.5A Output: 24V 10A Output Power: 250W Frequency: 50Hz / 60Hz	
2	Frequency Converter	Input Voltage: AC 220V Output Voltage: 0~220V Output Current: 7.0A Rated Power: 2.2KW Phase: 1PH Frequency: 50Hz / 60Hz	
3	Control Box	Voltage: AC 220V Voltage: 1.2KW（800W model） 1.8KW（1.5KW model） 2.5KW（2.2KW model）	
4	Switching Power Supply	Input: AC 110V 5.5A(Custom) AC 220V 3.5A Output: +24V 10A Output Power: 250W Frequency: 50Hz / 60Hz	

*Always use AC power sockets that comply with national standards! (It is recommended to use sockets with ground protection to ensure safety; Avoid sharing the same circuit with other high-power appliances to prevent overload.)

2.2.2 Aviation Cable Wiring

Aviation connectors are used for the electrical connection of the engraving machine. Different functional interfaces need to match different specifications of aviation connectors.The X, Y, Z, A axes are connected to their respective axis drive motors to control the movement of each axis.The limit switches are connected to the limit devices on the machine's X, Y, Z axes (the system uses the XYZ axis origin switches as the limit signal source, configured as 3 hard limits and 3 soft limits) to provide travel protection.The spindle is connected to the spindle drive circuit via its interface to control the spindle rotation.The water cooling system interface connects and controls the operating status of the spindle water cooling unit.The tool setting interface is an expansion function interface that can connect a tool setter to perform automatic tool setting operations.

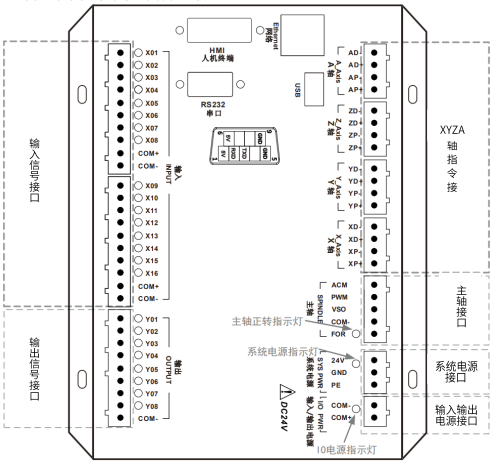
*Wiring Note: Connect the aviation connector cables from the engraving machine body to the corresponding ports on the control box one by one according to the labels. (The A-axis and tool setter are optional products. If not purchased, no connection is needed.) Then, connect the power cable to the 220V input socket on the control box. Connect the Ethernet port if network communication is required.

The aviation cable wiring description is as follows:

Name / Number	XYZA Axis (7-core)	Limit Switch (8-core)	Spindle Motor (5-core)	Water Cooling System (4-core)	External Switch (4-core)	Tool Setter (8-core)
1	24V+	X-	U	12V+	Reset	24V+
2	PUL+	X+	V	GND	Stop	/
3	PUL-	Y-	Shield	24V+	Start	/
4	DIR+	Y+	W	GND	GND	/
5	DIR+	Z+	PE			Tool Signal
6	GND	Z-				24V-
7	ALM	24V+				Tool Over-Travel
8		GND				24V-

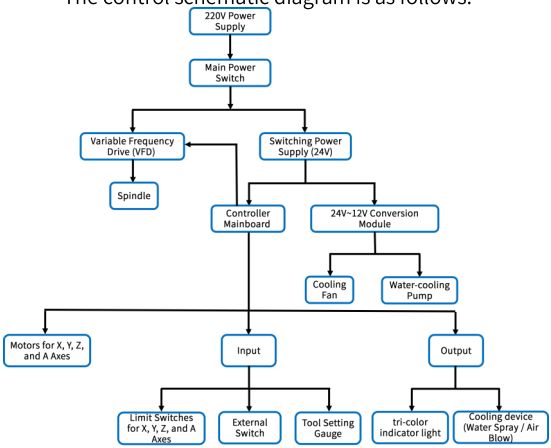
2.2.3Definition of Motion Control Card Interface

The schematic diagram of the motion control card interface is as follows.



III. Control Schematic Diagram

The control schematic diagram is as follows:



Description of interface signals:

Symbol	Type	Name	Function	Standard
PE	Enclosure	Enclosure Ground	Connected to the earth	
24V	System Power	External DC Power 24V "+" Pole	Users need to prepare their own DC power supply; connect the positive pole of the DC power supply to this terminal. Do not share the power supply with COM+ and COM-.	DC24V 3A
GND	System Power	24V Power Supply "-" Pole	"-" Pole of 24V DC power supply	DC24V 3A
COM+	I/O Power	External DC Power 24V "+" Pole	Users need to prepare their own DC power supply; connect the positive pole of the DC power supply to this terminal. Do not share the power supply with 24V and GND.	DC24V 3A
COM-		24V Power Supply "-" Pole	"-" Pole of 24V DC power supply	
XP+	X-axis Command	Positive Differential Pulse Signal	X-axis pulse differential output	Line driver output. RS422 standard. Maximum interpolation pulse: 1MHz.
XP-		Negative Differential Pulse Signal		
XD+		Positive Differential Direction Signal	X-axis direction differential output	
XD-				
YP+	Y-axis Command	Positive Differential Pulse Signal	Y-axis pulse differential output	
YP-		Negative Differential Pulse Signal		
YD+		Positive Differential Direction Signal	Y-axis direction differential output	
YD-				
ZP+	Z-axis Command	Positive Differential Pulse Signal	Z-axis pulse differential output	
ZP-		Negative Differential Pulse Signal		
ZD+		Positive Differential Direction Signal	Z-axis direction differential output	
ZD-				
4P+	4th-axis Command	Positive Differential Pulse Signal	4th axis pulse differential output	
4P-		Negative Differential Pulse Signal		
4D+		Positive Differential Direction Signal	4th axis direction differential output	
4D-				
X01...X16	General Input	Parameter-Configurable Input	Can be configured via parameters as driver alarm, zero point, positive/negative limit, tool setting, external emergency stop, and function expansion keys;	Supports mechanical, photoelectric, proximity switch inputs; Proximity switch: 24V Type: NPN Effective level: 0V

Symbol	Type	Name	Function	Standard
Y01...Y08	General Output	Parameter-Configurable Output	Can be configured via parameters as outputs for lubrication, cooling, etc.	Open-collector output with built-in reverse diode; Driving current: 500mA; Driving voltage : 30V
FOR	Spindle Digital Control	Spindle Forward Rotation or Start/Stop Output	Connect to the spindle forward rotation terminal (if only start/stop is needed, only this port needs to be connected)	Open-collector output with built-in reverse diode; Driving current: 500mA; Driving voltage: 30V
COM-	Spindle Output Common Terminal	COM-	Short-circuiting with ACM is prohibited	/
VSO	Spindle Analog Output	Analog Output	Connect to the analog input port of the frequency converter. *Note: Short-circuiting with DCM and COM is prohibited.	0-10V analog signal
ACM		Analog Ground		
PWM	PWM Output	PWM Output	Connect to the PEM input port of the frequency converter. *Note: Short-circuiting with COM is prohibited.	Duty cycle
ACM		PWM Ground		

IV. Controller Functions

4.1 Panel Key Functions

The layout diagram of the panel keys is as follows:

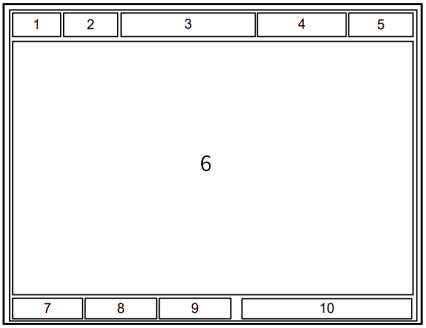
Button function description: All operations can be achieved through the keys. The key functions are introduced in the following table.

Button Icon	Function
页面 Page	Switching key for Main Control interface, File interface, and Parameter interface.
功能 Function	Enters the function operation menu list.
修调 Adjust	Toggle key for Feed Rate Override, Spindle Override, Feed Rate (F), Spindle Speed (S), Workpiece Coordinate System (G54~G59), Tool Length Compensation (H00), Step Increment, etc.
模式 Mode	Toggle key between Continuous movement and Step (Jog) movement modes.
A+ 倍率 %+ 7	A-axis positive direction movement; Input number "7"; Increments current focus data.
3 A-	A-axis negative direction movement; Input number "3".
Y+ 8	Y-axis positive direction movement; Input number "8"; Moves upward.

Button Icon	Function
2 Y-	Y-axis negative direction movement; Input number "2"; Moves downward.
Z+ 9	Z-axis positive direction movement; Input number "9".
% 倍率 1 Z-	Z-axis negative direction movement; Input number "1"; Decrements current focus data.
6 X+	X-axis positive direction movement; Input number "6"; Moves right; Page down.
X- 4	X-axis negative direction movement; Input number "4"; Moves left; Page up.
高/低速 5 JOG Speed	High/Low speed switching; Input number "5".
主轴 Spindle –	Spindle Start/Stop; Input symbol "-".
回工件零 Workp 0	Return all axes to workpiece zero point coordinates; Input number "0".
XY 清零 Zero .	X, Y axis coordinate zeroing; Input symbol "-".
对刀 Tool Check	Execute tool setting function.
断点继续 Break 退格 Bksp	Breakpoint resume machining; Backspace delete.
回机床零 Homu 取消 Cancel	Return all axes to machine zero coordinates; Cancel.
ZA 清零 Zero 确定 Enter	Z, A axis coordinate zeroing; Confirm.
启动	Start machining; Restart after pause.
暂停	Pause machining operation.
复位	Reset function; Toggle between Reset and Idle states.

4.2 Interface Display Description

4.2.1 Main Interface Overview



1.Current Mode (Continuous / Inching)
2.Current Status (Idle / Running / Reset)
3.Current Machining File
4.Current Machining Time
5.Login Status
6.Page Content Display
7.Main Control Page
8.File Page
9.Parameter Page
10.Network Information, Serial Number, System Date, etc.

4.2.2 Main Control Page Details

◆Path

Path→Main Control Page **(Fig 1)**

◆Description

①This interface is the main operation interface; manual operation, automatic machining, homing, etc., are all operated here.

②The interface provides necessary information required for monitoring during machining.

◆Page Description:

1. Coordinate Display
- This area can simultaneously display workpiece coordinates and machine coordinates.
- The diagram shows the current workpiece coordinate system is G54.

2. Current Version Information or Machining Code
- Current program date.
- Machining code line during runtime.

3. Parameter and Fault Alarm Information
- Current parameter information display.
- Specific alarm content display.

4. Feed Rate Override
- Feed rate override during automatic machining.
- Override range 0% - 300%.

5. Spindle Override / Jog Distance
- Spindle override during automatic machining.
- Override range 0% - 150%.
- Displays as Spindle Override in [Continuous] mode.
- Displays as Jog Distance in [Jog] mode.

6. Feed Rate (F)
- Displays the real-time feed rate of the machine equipment.
- Displays the set feed rate (Blue text indicates the system default feed rate parameter, white text indicates the F speed used in the program).

7. Spindle Speed (S)
- Displays the real-time spindle speed of the machine.
- Displays the set spindle speed (Blue text indicates the system default spindle speed parameter, white text indicates the spindle speed used in the program).

8. Current Workpiece Coordinate System
- Workpiece coordinate systems: G54, G55, G56, G57, G58, G59.

9. Tool Compensation Number (H)

10. High/Low Speed Display for Continuous Movement

11. General Output Signals Y1-Y9

12. General Input Signals X1-X16

Operation Menu List:
- In the Main Control page, press the [Function] key to pop up the operation menu shown in Fig 2.

4.2.3 Introduction to the File Page

◆Path

Path→File Page (Fig 3)

◆Description

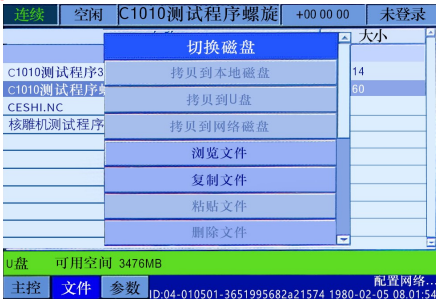
This interface provides a variety of file management functions, such as selecting machining files, browsing files, one-click copying, copying, pasting, deleting, and switching disks (local, USB flash drive, network), etc.

Fig 1:

Fig 2:

Fig 3:

Fig 4:



◆Page Description:

1. Program Files

- Press the [确定Enter] key to select the current file as the machining file.

2. Current Disk Information

- Press the [功能Function] key to switch disks (path: Function → Switch Disks → Display Local Disk/USB Flash Drive).

◆Operation Instructions:

- Press the direction keys [Y+] and [Y-] to move the cursor up and down;
- Press the page turn keys [X+] and [X-] to switch between upper and lower pages;
- Press the [确定Enter] key, and the file specified by the cursor will be set as the current machining file;

Operation Menu List:

- On the File Page, press the [功能Function] key to bring up the operation menu shown in Fig 4.

Fig 5:

Fig 6:

4.2.4 Introduction to the Parameter Page

◆Path

Path→Parameter Page (Fig 5)

◆Description

This is the setting interface for all system parameters, where the parameters to be set can be configured independently based on the configuration file.

◆Page Description:

1.Parameter Category Name

2.Current Parameter Information

- As shown in the figure, "7" represents the parameter number index, "X-axis Electronic Gear Ratio (Denominator)" represents the parameter name, and "1.000" represents the parameter value.

3.Description of the Currently Selected Parameter

◆Operation Instructions:

- Press the direction keys [Y+] and [Y-] to move the cursor up and down to select a parameter;

Press the page turn keys [X+] and [X-] to switch parameter categories;Press the [确定Enter] key: the parameter specified by the cursor will switch to input mode. Enter the parameter value using the numeric keys, then press the [确定Enter] key again to update the value in the current editing box to the newly input parameter value ; If you press the [取消Cancel] key, it means abandoning the current parameter modification.

Operation Menu List:

- On the Parameter Page, press the [功能Function] key to bring up the operation menu shown in Fig 6.

V. Equipment Operation Procedures

5.1 Equipment Startup and Debugging

① Before startup, check whether all wiring and external devices of the machine are in normal condition. After confirming they are normal, turn the emergency stop knob clockwise to release it, press the power button to energize the machine, and wait for the system interface to finish loading.

② After loading the CNC system, a prompt "Return to machine origin?" will appear. Click the【OK】 button in the operating system to return the machine to the origin.

③ Control the movement of the corresponding axes through the【X-】、【X+】、【Y+】、【Y-】、【Z+】、【Z-】、【A+】、and【A-】buttons in the operating system or via the handwheel, and check whether the movement function of each axis is normal.

④ Move each axis to the maximum and minimum positions of the machine's stroke to check whether the stroke limit protection function of the machine is normal.

⑤ Press the spindle switch in the operating system to control the start and stop of the spindle, and check whether the spindle function is normal. (Note: Check if there is a tool clamped on the spindle. If a tool is installed, ensure it is firmly clamped to prevent the tool from flying out and causing injury.)

⑥ After startup, check whether the spindle water-cooling fan and water-cooling pump start working normally.

5.2 Setting Workpiece Zero Point

After the tool is clamped on the spindle and the workpiece is fixed, the forward and reverse movement keys of XYZ on the handle can be used to control the forward and reverse movement of the machine, so that the machine spindle can be moved to the machining coordinate origin set in the program. For the convenience of setting the workpiece zero point, the machining origin is generally set at the lower left corner or the center of the material during programming.

The following takes the workpiece zero point at the lower left corner of the material as an example: first move the spindle to a safe height, quickly move each axis (with a magnification of X100 or X10) to the vicinity of the workpiece zero point set by the program (2-5mm away from the workpiece zero point). Then, slowly move each axis (with appropriate magnification) close to the workpiece zero point. When it is about to touch the workpiece zero point, click the inching mode and set a small inching distance value (such as 0.01). When the tool slightly touches the workpiece zero point or there is a small amount of material cutting when turning the nut, the "clear workpiece zero" operation can be performed. So far, the setting of the workpiece zero point is completed.

The centering function can also be used to set the workpiece zero point. For specific operations, refer to 6.2.5.

The specific operation buttons in this section may vary slightly depending on the operating system.

5.3 Loading Programs

After finding the workpiece zero point, select the program to be processed, copy the processing program to a USB flash drive, save the code to be processed to the system's local disk, and then load it into the control system.

5.4 Starting Processing

After the processing program is loaded, click the 【Start】 button, and the machine will start running the program.

VI.Common Operations

6.1 Homing (Machine Zero)

Since all coordinate settings required for machining are based on the machine origin, it is necessary to reconfirm the position of the machine origin every time the contrl box is powered on. Therefore, the homing action needs to be performed.

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [Main Control Page]

6.1.1 All Axes Return to Machine Zero

◆Operation Process

1) Click the [Home] shortcut key
2) All axes will move sequentially to their respective zero switch positions
3) After homing, each axis automatically returns a fixed distance (parameter settable)
4) The default homing sequence is: Z-X-Y-A (parameter settable)

6.1.2 Single Axis Return to Machine Zero

◆Operation Process (Taking X-axis as an example)

1) Click sequentially: [Function] key → [Return to Machine Zero] → [X-axis Return to Machine Zero] → [Enter] key
2) The system X-axis will move in the direction set by the homing direction parameter to the zero switch position
3) After homing, the X-axis automatically returns a fixed distance (parameter settable)
4) The operation for other axes to return to machine zero individually is the same

◆Note

During the homing process, the soft limits are invalid. Soft limits become effective only after the machine coordinate system is established.

6.2 Clearing Workpiece Zero

The Clear Workpiece Zero function is mainly used to set the workpiece coordinate origin. The workpiece zero is the benchmark for programming and machining (usually the origin of the G54-G59 coordinate system). Through the zeroing operation, a point in the machine coordinate system is set as the origin of the workpiece coordinate system. The coordinates of all subsequent machining programs are calculated based on this point.

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [Main Control Page]

6.2.1 Workpiece Coordinate System

The system provides six workpiece coordinate systems, G54-G59. The workpiece coordinate system for program machining needs to be determined before automatic machining.

◆Operation Process

1) Cycle click the [Adjust] key to switch the focus to the coordinate system G54-G59
2) Click the [%+] and [%-] keys to cycle through the G54-G59 workpiece coordinate systems. The coordinate

position display refreshes according to the current coordinate system.

6.2.2 Setting XYZ Workpiece Origin

Before machining a file, the user manually adjusts the position of the tool relative to the workpiece to start machining from a predetermined position of the workpiece. The correct workpiece coordinate system must be selected before setting the workpiece origin.

◆Operation Process

1) Manually move the X, Y, Z axes to the predetermined machining start position.
2) Click sequentially: [Function] key → [Clear Work Zero] → [All Axes Zero] → Press the [Enter] key to zero the coordinate values of the X, Y, Z axes at the current position.
3) You can also click [X/Y Zero] and [Z/A Zero] to zero the coordinate values of the X, Y, and Z axes at the current position.

6.2.3 Setting Single Axis Workpiece Origin

◆Operation Process (Taking X-axis as an example)

1) Manually move the X-axis to the predetermined machining start position.
2) Click sequentially: [Function] key → [Clear Work Zero] → [X-axis Zero] → Enter the value "0" → Press the [Z/A Zero] key to zero the coordinate value of the X-axis at the current position.
3) The operation for other axes to set the origin individually is the same.

6.2.4 Workpiece Origin Save and Load

This system can save and load multiple workpiece coordinate origin functions. If multiple workpiece coordinate origins need to be recorded for machining, move the XYZ axes to the workpiece origin point before saving that origin.

◆Operation Process (Taking X-axis as an example)

1) Click sequentially: [Function] key → [Multiple Work Origins] → [Save Work Origin] → Press the [Enter] key to save this origin.
2) Click sequentially: [Function] key → [Multiple Work Origins] → [Load Work Origin] → Press the [Enter] key to load this origin.

6.2.5 Centering

The centering function is generally used to find the XY center of a workpiece and is also a common function for setting the workpiece zero point. Since the center of the workpiece serves as the starting point for machining, it is necessary to use a tool to touch the edges of the workpiece.When centering, the influence of the tool diameter on position determination must be taken into account. Straight end mills are usually used directly for centering operations. If a pointed tool is used, attention should be paid to the variation of the tool diameter with height (the diameter difference caused by this variation can be controlled by adjusting the Z-axis value to ensure the tool centers the workpiece at the same height on both edges).After the controller obtains the mechanical coordinates of the workpiece edges, it will automatically calculate the mechanical coordinates of the center and set these mechanical coordinates into the coordinate system (G54-G59) of the current workpiece.

The centering method supported by this system is "Two-Point Centering".

◆Operation Process (Taking X-axis centering as an example)

1) Manually move the X-axis to one side far away from the workpiece, then perform the tool setting operation (for this operation, refer to 5.4.1).
2) Zero the X-axis workpiece coordinate: Click sequentially: [Function] key → [Clear Work Zero] → [X-axis Zero] → Enter the value 0, press the [Enter] key to zero the X-axis coordinate value at the current position.
3) Manually move the X-axis to the other side of the workpiece, then perform the tool setting operation (for this operation, refer to 5.4.1).
4) Execute centering: Click sequentially: [Function] → Select [X Centering] → Press the [Enter] key, then centering is completed.
5) Y-axis centering operation is the same as X-axis.

6.3 Returning to Workpiece Zero

The function of returning to workpiece zero refers to quickly moving the tool or machine axis to the origin of the workpiece coordinate system for the currently running program (i.e., the zero point set by commands such as G54~G59). This position is a programming reference point derived from the offset of the machine coordinate system and set independently for specific machined parts, usually located at a certain corner or geometric center of the workpiece. The purpose of performing this operation is to establish the relative positional relationship between the tool and the workpiece to be machined, thereby ensuring that the machining program can run according to the correct reference trajectory.

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [Main Control Page]

◆Operation Process (Taking X-axis as an example)

1) Click sequentially: [Function] key → [Return to Work Zero] → [X-axis Return to Zero] → Press the [Enter]

key to return to the X-axis coordinate origin position.

2) The operation for other axes to return to work zero is the same as for the X-axis. (Note: For machining safety, Z-axis zero operation is not recommended, therefore full-axis zero is also not recommended.)

6.4 Tool Setting

Tool setting is divided into manual tool setting and automatic tool setting. For the first tool setting in both methods, it is necessary to find the contact point between the tool and the workpiece reference surface through the trial cutting method, i.e., control the movement of each axis to the workpiece origin via the panel and set the workpiece origin (zero the workpiece origin).

6.4.1 Manual Tool Setting

When the distance to the workpiece origin coordinate is far, use the rapid movement mode to move near the origin. When approaching the origin, use the jog mode (set a small jog distance) to move to the origin, and manually turn the nut. When the workpiece material is slightly cut or resistance is felt, the position of each axis can be determined, and then each axis can be zeroed.

6.4.2 Automatic Tool Setting

The Z-axis automatic tool setting function measures the tool tip position of different tool numbers through the tool setter on the machine. When using the tool setter, the distance from the trigger position of the tool setter to the workpiece datum plane is fixed. The user can input the distance from the tool tip position of different tool numbers to the workpiece datum plane into the workpiece coordinate system as the basis for tool length offset during machining. Before using this function, please confirm whether the equipment is equipped with a tool setter. (The engraving machine comes standard with a tool setter; customers can choose it as an option according to actual needs. If you need to purchase, please contact our customer service or sales.)

Fixed Tool Setter: Tool setting requires pre-setting the initial XYZ position of the tool setter (debugged before shipment). During tool setting, the machine automatically moves to the fixed position of the tool setter for tool setting.

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [Main Control Page]

◆Operation Process

1) Execute tool setting: Click sequentially: [Function] key → [Tool Setting (Probe)] → Press the [Enter] key → [Fixed Tool Setting] → Press the [Enter] key to execute the tool setting action.
2) The tool setting operation procedure: Move the Z, X, and Y axes sequentially to their mechanical reference positions. Subsequently, lower the Z-axis to probe the tool setter. Upon detecting the trigger signal from the tool setter switch, retract the axis to a safe distance.

◆Extension

- Fixed Tool Setting Principle: After fixed tool setting, the recorded value is the machine coordinate of the Z-axis at the tool setter position (Z-axis external compensation value).
- Z-axis Zeroing Principle: Z Offset = Current Z-axis Machine Coordinate — Z-axis External Compensation Value (Machine position at tool setting point) — Other Compensations.

◆Note

The sequence for Z-axis zeroing and fixed tool setting

- Perform fixed tool setting with any tool first.
- Then move the tool that has been set to a suitable position and perform the Z-axis workpiece zeroing operation.
- Do not use a tool that has not been set for zeroing operation.
- Before performing fixed tool setting, homing (machine zero) must be done first, otherwise it cannot be executed.

6.5 File Loading and Management

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [File Page]

6.5.1 Loading Machining File

The system supports three file storage modes: Local Disk, USB Drive, and Network Disk.

◆Operation Process

1) Switch to the File Page: Press the [Page] key to switch pages until the File Page is displayed.
2) Switch the disk where the file is located: In the [File Page], press the [Function] key → [Switch Disk] → Press the [Enter] key → Select the disk where the file is located ([Local Disk] / [USB Drive] / [Network Disk])
3) Use the [Y+], [Y-] up/down keys and the [X+], [X-] page keys to move the cursor to the desired NC file for machining.
4) Press the [Enter] key, then the screen switches to the [Main Control Page], and this program will be designated as the current machining program.

◆ Note

You can confirm whether the machining program is successfully loaded by checking the current machining file name on the first line of the interface. (If the machining program is displayed in red, it indicates that the program has expired.)

6.5.2 Copy File

To ensure stable program operation during machining, it is recommended to copy the machining files from the USB drive to the local disk, and then select the required NC file from the local disk.

◆Operation Process

1) Press the [Y+], [Y-] keys to select the file.
2) Press the [Function] key, then select the target disk for copying: [Copy to Local Disk] or [Copy to USB Drive] or [Copy to Network Disk].
3) Press the [Enter] key to execute the file copy.

6.5.3 Copy, Paste File

◆Operation Process

1) Press the [Y+], [Y-] keys to select the file.
2) Press the [Function] key → [Copy File] → Press the [Enter] key.
3) Switch to another disk.
4) Click sequentially: [Function] → [Paste File] → Press the [Enter] key, then the file from step 2 will be pasted to the current location.

6.5.4 Delete File

◆Operation Process

1) Press the [Y+], [Y-] keys to select the file.
2) Click sequentially: [Function] → [Delete File] → Press the [Enter] key, then the currently selected file will be deleted.

6.6 Parameter Management

◆Operation Conditions:

- The system is in the [Idle] state
- The main interface is switched to the [Parameter Page]

6.6.1 Parameter Modification

◆Operation Process

1) First, click on any parameter to enter the password (Password: 888888. Since modifying parameters requires entering the password, and after entering the password it returns to the first page, selecting the parameter for modification again does not require re-entering the password).
2) Three ways to select parameters: Search parameter number, Parameter classification table, Manual positioning with arrow keys.
3) Use the cursor to select the parameter that needs modification, press the [Enter] key.
4) Enter the parameter value using the number keys, press the [Enter] key to save.

6.6.2 Parameter Backup

◆Operation Process

1) Press the [Function] key → [Parameter Backup] → Press the [Function] key → Select the target disk for backup [Backup to USB Drive], [Backup to Network Disk], [Backup to System Disk].
2) Press the [Enter] key to execute the backup.

6.6.3 Parameter Restore

◆Operation Process

1) Press the [Function] key → [Parameter Backup] → [Function] → Select the target disk for restoration [Restore from USB Drive], [Restore from Network Disk], [Restore from System Disk].
2) Press the [Enter] key to execute the restoration.

6.7 System Upgrade

There are two methods for system upgrade: one is automatic upgrade upon startup, and the other is manual upgrade after startup.

◆Operation Process

1) Automatic Upgrade upon Startup

① Copy the upgrade package folder to the root directory of the USB drive.

② Insert the USB drive and power off then restart.

③The system will automatically search for the upgrade file and perform the upgrade. During the upgrade process, the interface will stay on the startup screen for about 10 seconds. Please wait patiently and do not power off.

④ After startup, check if the version date on the main control interface matches the one provided by the customer.

2) Manual Upgrade after Startup

① Copy the upgrade package folder to the root directory of the USB drive.

② Insert the USB drive.

③ Enter sequentially: [Parameter Page] → [Advanced] → [System Upgrade] → [Upgrade from USB Drive] → Press the [Enter] key to execute the system upgrade.

④ After the upgrade is completed, a dialog box will prompt "Software upgrade completed, please restart the system".

⑤ After startup, check if the version date on the main control interface matches the one provided by the customer.

◆Note

① The upgrade package folder name must be 'hsys' and must be placed in the root directory of the USB drive.

② The folder name must be 'hsys'.

③ The root directory of the USB drive must contain the 'hsys' upgrade package, and the 'hsys' folder contains the upgrade package contents.

④ After a successful upgrade, delete the 'hsys' folder from the USB drive. Otherwise, it will automatically upgrade every time it powers on after a power-off.

VII. Control Box Maintenance

The electrical control cabinet should be installed in a well-ventilated environment away from strong magnetic fields and high temperatures, and protected from direct sunlight. Special attention must be paid to the following: The system power supply (24V, GND) and input/output power supplies (COM+, COM-) must be powered by two independent switching power supplies respectively. Sharing a single power supply is strictly prohibited. Additionally, regularly check the working status of the power filter to prevent high-frequency noise from interfering with the control system. If the cabinet will not be used for an extended period, disconnect the power supply and cover it with a dust shield. It is recommended to power it on monthly to dispel internal moisture and keep the circuit boards dry.

During the operation of the control cabinet and the machine, dust easily accumulates inside. This is especially true when processing metal materials: once conductive metal dust enters the control cabinet, it may cause short circuits or circuit burnout. When cleaning, first open the cabinet cover, use a hair dryer set to the cold air mode or a cold air gun, invert the cabinet, and blow off the dust on the surface of circuit boards and components. For hard-to-reach corners, gently wipe with a small brush to ensure thorough removal of conductive dust. After cleaning, confirm that the interior is completely dry before closing the cover and restarting the equipment.

VIII. Code Support

7.1 G-Codes

Jingyan engraving machines commonly use G-codes for machining recognition. For commonly used G-codes, please refer to Appendix I.

7.2 M-Codes

Jingyan engraving machines also commonly use M-codes for machining recognition. For commonly used M-codes, please refer to Appendix II.

The information contained in this manual is for reference only and does not constitute any operational specifications or technical guarantees. Due to differences in actual equipment models, operating environments, and specific working conditions, users must make adjustments and judgments based on the actual status of the equipment during the actual commissioning and use process. For special circumstances or technical issues not covered in the manual, do not handle them on your own; instead, contact the manufacturer or authorized service provider to obtain professional support. Product specifications and information are subject to change without prior notice, and the manufacturer also reserves the right to improve the products and documents at any time and hold the final right of interpretation. The manufacturer shall not be liable for any consequences directly or indirectly caused by operations not in accordance with this statement.

附录一：

G-Code	Instruction Name	Instruction Explanation	G-Code Example
G00	Rapid Positioning	Move to the specified position at the system's highest set speed.	G00 X10 Y20 Z30
G01	Linear Interpolation	Move to the specified position at the feed rate (F) specified by the system or the file.	G01 X50 Y60 F150
G02	Circular Interpolation CW	Clockwise circular cutting machining.	G02 X100 Y100 I20 J20
G03	Circular Interpolation CCW	Counterclockwise circular cutting machining.	G03 X100 Y100 I20 J20
G04	Dwell	Pause. When used with P, specifies the pause time in milliseconds.	G04 P200
G17	XY Plane Selection	Selects the XY plane for interpolation.	G17
G18	ZX Plane Selection	Selects the ZX plane for interpolation.	G18
G19	YZ Plane Selection	Selects the YZ plane for interpolation.	G19
G20	Inch Input	/	G20
G21	Millimeter Input	/	G21
G28	Return to Reference 1 via Intermediate Point	1.If the corresponding axis is not configured with a zero signal, the instruction for that axis is ignored. 2.If all axes have completed the HOME operation, move to the intermediate point first, then return to reference point 1. 3.If the HOME operation has not been completed, move to the intermediate point first, then perform the HOME operation.	1.G90G28X10Y10Z10A10(First move to workpiece coordinate (0,0,0,0), then return to reference point 1) 2. G91G28Z0(Z-axis returns to the Z position of reference point 1 from the current position)
G30	Return to Reference P via Intermediate Point	The P word takes values 1, 2, 3 to specify which reference point to return to. Usage is similar to the G28 instruction.	G90G30X10Y10Z10 A10P
G31	Probe.During the instructed movement, if the probe signal is detected at the specified level, the movement stops (stop mode can be configured for immediate stop or deceleration stop).	XYZA specify probe travel for each axis. L specifies the effective probe signal level. K specifies whether hard limits are disabled during probing: 0-Disabled; 1-Not disabled Q specifies the stop method after probe signal appears: 0-Deceleration stop; 1-Immediate stop. F specifies the probe speed.	G91G31Z-1000L0Q1K0F100(Z-axis probes downward 1000mm, probe signal effective level 0, hard limits disabled during probing, immediate stop upon probe signal.)
G53	Machine Coordinate System Move	The corresponding configured axis must have completed the HOME operation, otherwise the program alarms and exits.	G53X10Y10(XY axes move to machine coordinate (10,10))
G54	G54 Coordinate System	Selects the G54 workpiece coordinate system.	G54
G55	G55 Coordinate System	Selects the G55 workpiece coordinate system.	G55
G56	G56 Coordinate System	Selects the G56 workpiece coordinate system.	G56
G57	G57 Coordinate System	Selects the G57 workpiece coordinate system.	G57
G58	G58 Coordinate System	Selects the G58 workpiece coordinate system.	G58
G59	G59 Coordinate System	Selects the G58 workpiece coordinate system.	G59

G-Code	Instruction Name	Instruction Explanation	G-Code Example
G73	High-Speed Peck Deep Hole Drilling Cycle	Deep hole drilling cycle, used with F. X_Y hole position data; Z_ distance from R point to hole bottom; R_ distance from initial position to R point; Q_ cutting depth per feed; P_ dwell time; K_ repeat count.	G73X_Y_Z_R_Q_F_K_
G73	High-Speed Peck Deep Hole Drilling Cycle	Deep hole drilling cycle, used with F. X_Y hole position data; Z_ distance from R point to hole bottom; R_ distance from initial position to R point; Q_ cutting depth per feed; P_ dwell time; K_ repeat count.	G73X_Y_Z_R_Q_F_K_
G74	Left-Hand Tapping Cycle	The spindle must be configured as a servo spindle. G74 will reverse at the set speed to the hole bottom (Z) then forward back to the R point. G84 will forward at the set speed to the hole bottom (Z) then reverse back to the R point. X_Y hole position data; Z_ distance from R point to hole bottom; R_ safety plane; P_ dwell time.	G98/G99 G74 X_ Y_ Z_ R_ P_ F_
G84	Right-Hand Tapping Cycle	The spindle must be configured as a servo spindle. G74 will reverse at the set speed to the hole bottom (Z), then forward back to the R point. G84 will forward at the set speed to the hole bottom (Z), then reverse back to the R point. X_Y hole position data; Z_ distance from R point to hole bottom; R_ safety plane; P_ dwell time.	G98/G99 G84 X_ Y_ Z_ R_ P_ F_
G81	Drilling Cycle	Drilling cycle, used with F. X_Y hole position data; Z_ distance from R point to hole bottom; R_ distance from initial position to R point; F_ cutting feed rate; K_ repeat count.	G81 X_Y_Z_R_F_K_
G82	Boring Cycle	Boring cycle, used with F. X_Y hole position data; Z_ distance from R point to hole bottom; R_ distance from initial position to R point; F_ cutting feed rate; K_ repeat count; P_ dwell time at hole bottom.	G82X_Y_Z_R_P_F_K_
G83	Peck Deep Hole Drilling Cycle	G83 retracts rapidly to the hole bottom between feeds. X_Y hole position data; Z_ distance from R point to hole bottom; R_ distance from initial position to R point; Q_ cutting depth per feed; F_ cutting feed rate; K_ repeat count.	G83X_Y_Z_R_Q_F_K_
G90	Absolute Programming	In G90 input programs, all machining operation points start from the same initial point.	G90
G91	Incremental Programming	In G91 input programs, all machining operation points are relative to the previous machining point, representing a relative coordinate increment.	G91

附录二：

M-Code	Instruction Name	Instruction Explanation
M00	Program Stop	Pause
M03	Spindle Forward (CW)	Spindle forward output control signal set active.
M04	Spindle Reverse (CCW)	Spindle reverse output control signal set active.
M05	Spindle Stop	Spindle forward output control signal set inactive.
M08	Coolant On	Coolant output control signal set active.
M09	Coolant Off	Coolant output control signal set inactive.
M10	Lubrication On	Lubrication output control signal set active.
M11	Lubrication Off	Lubrication output control signal set inactive.
M29	Rigid Tapping Mode	Enter rigid tapping mode, called before G74/G84 instructions.
M30	Program End and Reset	Ends all instructions.
M105	X-axis Homing Command	If these commands appear on the same line, it indicates the corresponding axes perform homing simultaneously.
M106	Y-axis Homing Command	
M107	Z-axis Homing Command	
M108	A-axis Homing Command	

SERVICE

E-Mail:support@jingyan.tech

Website:http://www.jingyan-tech.com/


注意安全!


当心触电!


注意强磁场!


佩戴护目镜!


阅读说明书!


请勿将用电器丢入垃圾桶!