

---

## 目录

第一章 概述 .....	4
1.1 使用安全注意事项 .....	4
1.2 加工流程 .....	4
第二章 安装与安全事项 .....	4
2.1 系统配置 .....	5
2.2 安全提醒与系统安装 .....	5
2.2.2 雕刻机安装环境 .....	5
2.2.3 保存及搬运时的注意事项 .....	6
2.2.4 一般注意事项 .....	6
2.2.5 使用安全注意事项 .....	7
2.2.6 禁止事项 .....	8
2.2.7 废弃时的注意事项 .....	8
2.3 电缆要求 .....	9
2.4 常见 EMC 问题解决建议 .....	9
第三章 系统接线说明 .....	11
3.1 端子定义说明 .....	12
3.2 接线示例 .....	16
第四章 系统操作界面 .....	17
4.1 系统界面 .....	17
4.2 工具栏 .....	18
4.3 状态栏 .....	19
4.4 加工轨迹窗口 .....	21
第五章 导入加工文件 .....	22
5.1 在程序管理导入 .....	22
5.2 在系统内手动编写 .....	22
5.3 复制批量粘贴到共享文件夹 .....	22
第六章 回机械原点 .....	24
第七章 设置工件原点 .....	26
7.1 手动清原点 .....	26
7.2 固定对刀 .....	26
7.3 浮动对刀 .....	26

---

7.4 保存和选取工件原点 .....	27
第八章 手动加工 .....	29
8.1 手轮脉冲方式 .....	29
8.2 连续点动方式 .....	30
8.3 步进方式 .....	30
第九章 加工操作 .....	31
9.1 装载文件 .....	31
9.2 设置工件原点 .....	31
9.3 自动加工 .....	31
9.3.1 开始 .....	31
9.3.2 暂停 .....	33
9.3.3 停止 .....	34
9.3.4 微调 .....	34
9.3.5 断点继续 .....	34
9.3.6 高级开始 .....	34
9.3.7 阵列加工 .....	35
9.4 手轮引导加工 .....	36
第十章 检查加工文件 .....	37
第十一章 铣底、铣边框操作 .....	39
第十二章 程序管理 .....	40
12.1 新建 .....	40
12.2 编辑 .....	40
12.3 删除 .....	41
12.4 装载 .....	41
第十三章 参数管理 .....	42
13.1 设置参数 .....	42
13.2 恢复厂商参数 .....	43
13.3 备份参数 .....	43
13.4 恢复参数 .....	43
13.5 修改密码 .....	44
13.6 客户端设置 .....	45
13.7 自定义快捷输出、自定义动作按钮、自定义M代码 .....	45

---

13.8 刀具补偿设置.....	46
13.9 刀具偏置.....	47
13.10 参数修改方法.....	47
13.11 参数总览.....	47
13.11.2 轴参数.....	53
13.11.3 主轴参数.....	54
13.11.4 原点参数.....	54
13.11.5 补偿参数.....	55
13.11.6 手轮参数.....	55
13.11.7 刀库参数.....	56
13.11.8 SL 总线参数.....	57
13.11.9 双工位参数.....	58
13.11.10 上下料参数.....	60
13.11.11 贴标.....	62
第十四章 辅助功能.....	63
14.1 当前版本.....	63
14.2 重启控制器软件.....	63
14.3 程序负载监测.....	64
14.4 打开/关闭软件开机自启动.....	64
14.5 软键盘.....	64
14.6 定时暂停.....	64
14.7 导出安装包.....	65
14.8 升级主站.....	65
第十五章 I/O 状态.....	66
15.1 输出 I/O 测试.....	67
15.2 输入/输出 I/O 配置.....	67
15.3 端口极性修改.....	69
15.4 SL 总线配置.....	69
更新日志.....	71

---

## 第一章 概述

欢迎您使用本公司生产的 L68 控制系统。本说明书详细介绍了本控制系统的各个功能的详细操作，并配以大量实例和图表加以说明。在使用雕刻机或切割机之前请您仔细阅读本操作说明书，以确保正确使用加工，防止意外事故发生。并请妥善保存此说明书，以便随时查阅。

本系统采用工控主机+L68 控制器方式，需要配置 PC 机。操作简单、易学易懂，安装方便，占用体积小，适用于石材雕刻、铝板切割。

### 1.1 使用安全注意事项

- 严禁在强干扰、强磁场环境中使用本产品；
- 不可带电插拔操作盒电源；
- 注意防水、防尘、防火；
- 防止金属等导电物质进入壳内；
- 严禁非授权的拆卸，内部无用户可修复部件；
- 插拔其他连线时用力要适度；
- 长时间不使用，请注意断电，并妥善保存；
- 检修、调整机器时，必须关闭电源；
- 操作及维修人员必须经过培训。

### 1.2 加工流程

系统加工流程为：

1. 回零前调试
2. 回机械原点
3. 设工作原点
4. 装载加工程序
5. 选择加工方式
6. 执行加工

具体操作，请查阅以下功能介绍，进行相关操作。

## 第二章 安装与安全事项

---


## 2.1 系统配置

- ◆ 3C-I011A 主控、工控机、电源盒、显示器、扩展板
- ◆ 工业网线

## 2.2 安全提醒与系统安装

在使用本控制系统前，请您仔细阅读本手册后再进行相关的操作。

仔细阅读本操作说明书，以及用户安全须知，使用者应该做相应保护，采取必要的安全防护措施，方可进行操作。初次进行操作的操作者，应在了解相应功能的正确使用方法后，方可进行相应的操作，对于不熟悉的功能或参数，严禁随意操作或更改系统参数。

本手册有关安全内容使用标识：，有关作业安全的内容十分重要，请务必遵守。没有按照要求操作会造成危险情况，可能导致轻伤或中度伤害，以及设备损坏的情况。

**注意：对于重型机械，容易引发人身安全事故的设备，不可使用本控制器。**

### 2.2.1 系统安装事项

- 1) 配线作业必须由专业电工进行。
- 2) 确认电源断开后才能开始作业。
- 3) 请安装于金属等阻燃物上并远离可燃物。
- 4) 使用时必须安全接地。
- 5) 外部电源发生异常，控制系统会发生故障，为使整个系统安全工作，请务必在控制系统的外部设置安全电路。
- 6) 安装、配线、运行、维护前，必须熟悉本手册内容；使用时也必须熟知相关机械、电子常识及一切有关安全的注意事项。
- 7) 安装控制器的电箱，应具备通风良好、防油、防尘的条件。若电控箱为密闭式则易使控制器温度过高，影响正常工作，须安装抽风扇，电箱内适宜温度为 40℃ 以下，不要使用在结露及冰冻的地方。
- 8) 控制器安装应尽量避免与接触器、变压器等交流配件布置过近，避免不必要的突波干扰。

### 2.2.2 雕刻机安装环境

- 1) 地面坚固；

- 
- 2) 避免阳光直射;
  - 3) 为保养检修留一定空间;
  - 4) 空间温度: 5—40℃;
  - 5) 相对湿度: 30—95%;
  - 6) 设备安装要水平;
  - 7) 通风良好。

### 2.2.3 保存及搬运时的注意事项

**\*注意: 请勿保存、放置在下述环境中, 否则可能会导致火灾、触电或机器损坏。**



- 1) 阳光直射的场所, 环境温度超过保管放置温度条件的场所, 相对湿度超过保管放置湿度的场所, 温差大、结露的场所。
- 2) 接近腐蚀性气体、可燃性气体的场所, 尘土、灰尘、盐分及金属粉尘较多的场所, 有水、油及药品滴落的场所, 振动或冲击可传递到主体的场所。
- 3) 请勿握住线缆进行搬运, 否则会导致机器损坏或故障。


### 2.2.4 一般注意事项

- 1) 请勿过多的将本产品叠加放置在一起, 否则可能会导致损坏或使用时发生故障。
- 2) 本产品为一般性工业制品, 不是以伤害任何生命健康为使用目的的产品。
- 3) 若应用于可能因本产品故障引发重大事故或损坏的装置时, 请配备安全装置。
- 4) 若应用于硫磺或硫化性气体浓度较高的环境下, 请注意可能因硫化反应, 使得芯片电阻断线或出现点接触不良等情况。
- 5) 若输入超过本产品电源额定范围的电压, 可能因内部部件的损坏出现冒烟、起火等现象, 请充分注意输入电压。
- 6) 请注意本产品无法保证超过产品规格范围的使用。
- 7) 本说明书如有与系统功能不符、不详尽处, 以系统软件功能为准。
- 8) 控制系统功能改变或完善(升级), 恕不另行通知。如若用户有其他需求,

请与本公司联系。

### 2.2.5 使用安全注意事项

	<ul style="list-style-type: none"><li>◆ 严禁在强干扰、强磁场环境中使用本产品。</li><li>◆ 注意防水、防尘、防火。</li><li>◆ 插拔 U 盘和其他连线时用力要适度。</li><li>◆ 长时间不使用，请注意断电，并妥善保管。</li><li>◆ 防止金属等导电物质进入壳内。</li><li>◆ 编码器必须使用屏蔽线，且屏蔽层必须保证单端可靠接地！</li><li>◆ 请勿在伺服驱动器周围安装变压器等产生电磁波或干扰的设备，否则会导致伺服驱动器误动作，如需安装此类设备，应在其与伺服驱动器之间设置屏蔽板。</li></ul>
注意	<ul style="list-style-type: none"><li>◆ 请遵照当地标准，进行支路、短路回路的保护。如果支路、短路回路的保护措施不当，可能会导致伺服驱动器损坏。</li><li>◆ 请勿与焊机或需要大电流的动力机器等共用接地线，否则会导致伺服驱动器或机器的动作不良。</li><li>◆ 如果机器明显损坏或者有部件丢失，请勿连接或进行操作。接线、检查等请由专业人员进行。</li><li>◆ 严禁非授权的拆卸，内部无用户可修复部件；</li><li>◆ 旋转的电机会向伺服驱动器馈送电能，这样即使在电机停止并切断电源时也会造成伺服驱动器带电。在伺服驱动器上开展维护保养工作之前，请确保电机伺服驱动器安全断开连接。</li></ul>
 危险	<ul style="list-style-type: none"><li>◆ 请勿在电源通电的状态下进行接线作业，否则会有触电的危险。进行检查前，请切断所有设备的电源。即使切断电源，内部电容器中还有残余电压。切断电源后，请至少等待 10 分钟。</li><li>◆ 雕刻刀十分锋利，运行时禁止用手触摸，以防伤害。也不要用手帕、丝巾接触，以防卷入造成伤害或损坏设备；</li></ul> <p style="text-align: center;">上电后</p>

	<ul style="list-style-type: none"> <li>◆ 上电后不要打开控制盒盖板，否则有触电的危险！</li> <li>◆ 不可带电插拔操作盒电缆。</li> <li>◆ 请勿在通电状态下拆下伺服驱动器的盖板或触摸印刷电路板，否则会有触电的危险。</li> </ul>
<b>运行中</b>	
	<ul style="list-style-type: none"> <li>◆ 非专业技术人员请勿在运行中检测信号，否则可能引起人身伤害或设备损坏！</li> <li>◆ 请勿触摸散热风扇及放电电阻以试探温度，否则可能引起灼伤！</li> </ul>
<b>维护保养时</b>	
	<ul style="list-style-type: none"> <li>◆ 操作及维修人员必须经过培训</li> <li>◆ 检修、调整机器时，必须关闭电源。</li> <li>◆ 没有经过专业培训的人员请勿对伺服驱动器实施维修及保养，否则造成人身伤害或设备损坏！</li> <li>◆ 请勿带电对设备进行维修及保养，否则有触电危险！</li> <li>◆ 所有可插拔插件必须在断电情况下插拔！</li> <li>◆ 更换伺服驱动器后必须进行参数的设置和检查。</li> <li>◆ 请勿上电运行已经损坏的机器，否则会扩大机器的损坏。</li> </ul>
 警告	<ul style="list-style-type: none"> <li>◆ 有些系统在通电时机械可能会出现自行动作，请小心，否则有导致死亡或重伤的可能。</li> <li>◆ 非电气施工专业人员请勿进行安装、维护、检查或部件更换，否则会有触电的危险。</li> <li>◆ 严禁非授权的拆卸，内部无用户可修复部件。</li> </ul>

**\*注意：处理不当可能会引起危险，包括人身伤害或设备事故等。**

### 2.2.6 禁止事项

除本公司工作人员外，请勿进行拆卸修理工作。

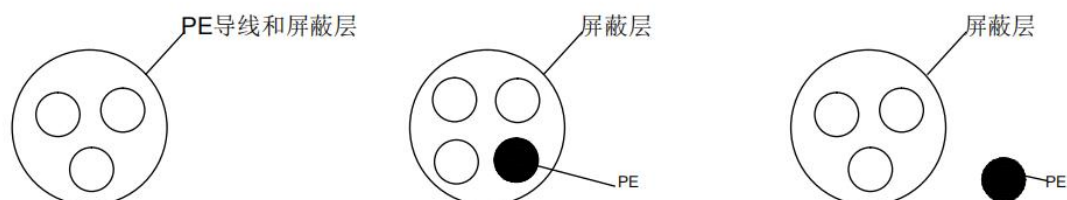
### 2.2.7 废弃时的注意事项

产品正常使用之后需作为废品处理时，有关电子信息产品的回收、再利用事宜，请遵守有关部门的法律规定。

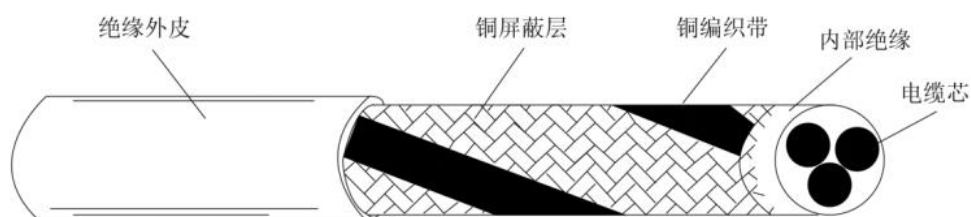


## 2.3 电缆要求

为了满足 EMC 的要求，编码器线必须采用带有屏蔽层的屏蔽电缆，动力线建议采用有屏蔽层的屏蔽电缆，屏蔽电缆有三根相导体的屏蔽电缆和四根相导体的屏蔽电缆，其中一根为 PE 线，如下图所示：



为了有效抑制射频干扰发射和传导，屏蔽线的屏蔽层由同轴的铜编织带组成。为了增加屏蔽效能和导电性能，屏蔽层的编织密度应大于 90%。如下图所示：



安装注意事项：

(1) 所有屏蔽电缆推荐使用屏蔽对称电缆，对于输入电缆也可以采用四芯电缆；

(2) 电机电缆及其 PE 屏蔽导线（绞合屏蔽）应尽量短，以降低电磁辐射以及电缆外部的杂散电流和容性电流；

(3) 建议所有控制电缆都需要采用屏蔽电缆；

(4) 驱动器的输出动力线建议使用屏蔽电缆，或使用钢管屏蔽动力线，且屏蔽层要可靠接地，于受干扰设备的引线建议使用双绞屏蔽控制线，并将屏蔽层可靠接地。

## 2.4 常见 EMC 问题解决建议

驱动器产品属于强干扰设备，在使用过程中因为布线、接地等存在问题时，仍然可能出现干扰现象，当出现与其他设备相互干扰的现象时，还可以采用以下的办法进行整改。

干扰类型	
漏电保护断路器 开关跳闸	<p>降低载频；</p> <p>减少驱动线长度；</p> <p>输入驱动线上加绕磁环（不绕 PE 线）；</p> <p>上电瞬间跳闸的，需断开输入端较大对地电容；（断开外置或内置滤波器的接地端，输入端口对地 Y 电容的接地端）</p> <p>运行或使能跳闸的，需在输入端加装漏电流抑制措施（漏电流滤波器、安规电容 + 绕磁环、绕磁环）</p>
通讯干扰	<p>电机外壳连接到驱动器 PE 端；</p> <p>驱动器 PE 端连接电网 PE ；</p> <p>在输入电源线上加绕磁环；</p> <p>通讯线源和负载端加匹配电阻；</p> <p>通讯线差分线对外加通讯公共地线；</p> <p>通讯线用屏蔽线，屏蔽层接通讯公共地线；</p> <p>通讯布线需要用双绞方式；</p>

表 2-1 常见 EMC 干扰问题与处理方法

### 第三章 系统接线说明

图 3-1 总线接线图

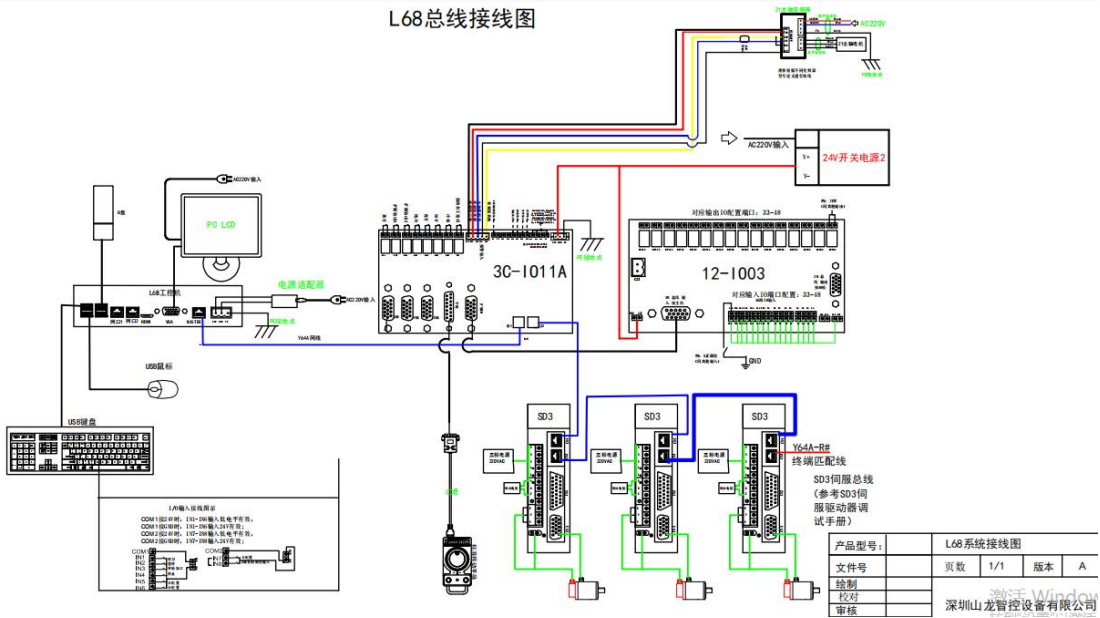
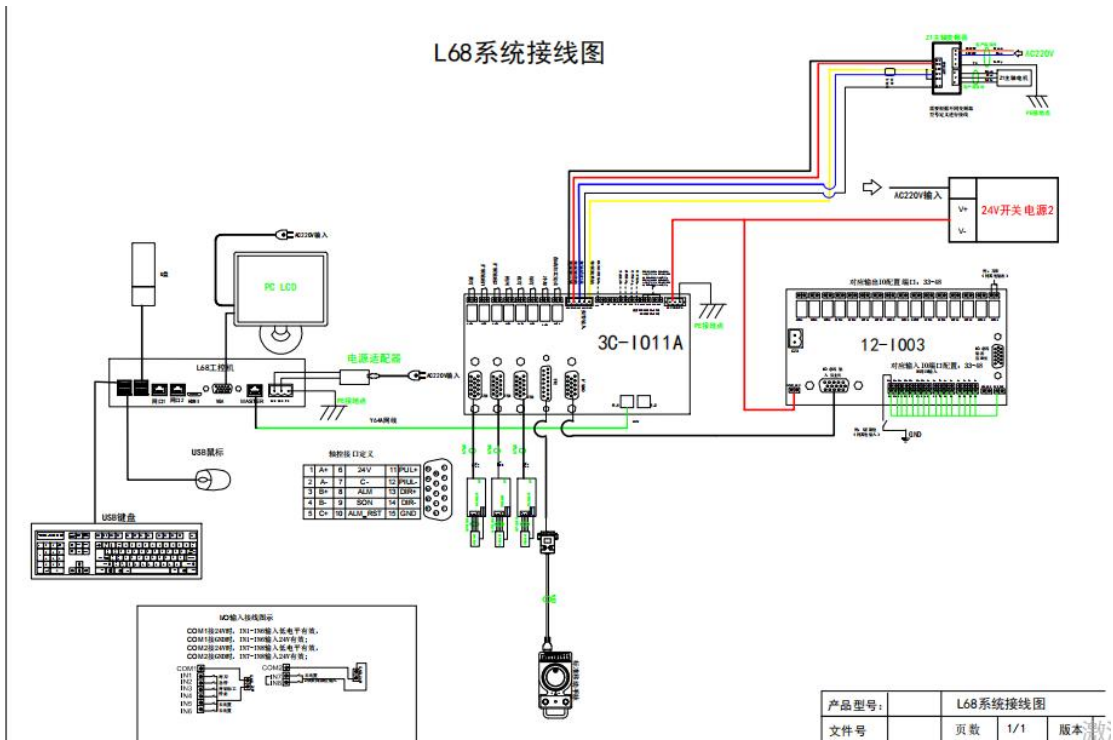


图 3-2 脉冲接线图

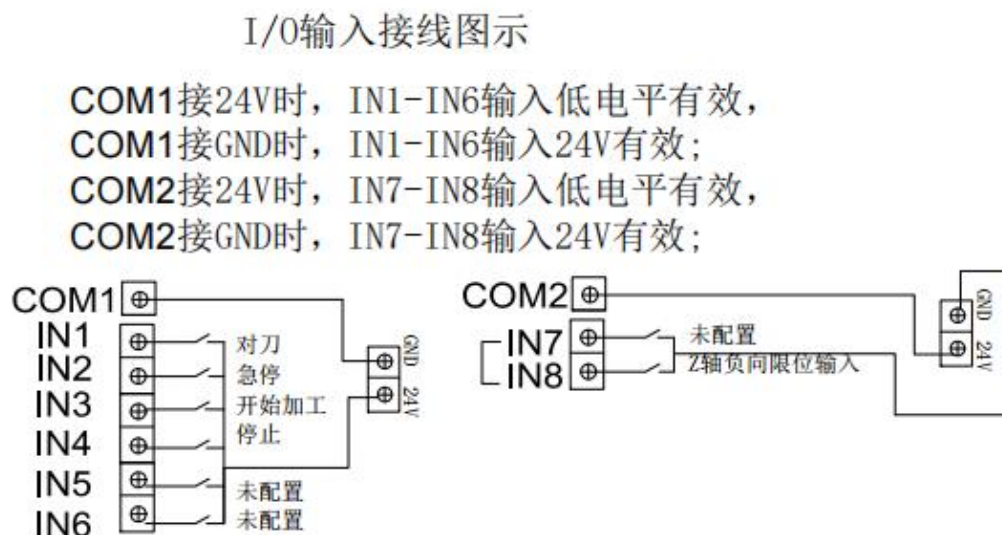


### 3.1 端子定义说明

主控输入 I0 1~8、输出 I0 1~8、扩展板输入输出各有 16 路 对应输入 I0 33~48、输出 I0 33~48 可以自由设置端口功能，配置了就可以使用。

注：输入 I0 有两种接法，如图所示

图 3-3 输入 I0 接线图



分类	端口	定义	说明
电源输入	24V	24V 电源输入端	直流 24V 输入，提供系统工作用电。
	GND	24V GND	
	PE	地线端	
I0 输入	24V	24V 电源输入端	24V 电源输入，给光电开关供电。
	GND	公共端	电源地及开关量公共端。
	COM1	公共端	参考图 3-3
	COM2	公共端	参考图 3-3
	GX1	对刀	开关量输入，可接常开、常闭。
	GX2	急停	开关量输入，可接常开、常闭。
	GX3	开始加工	开关量输入，可接常开、常闭。
GX4	暂停加工	开关量输入，可接常开、常闭。	

	GX5	未配置	开关量输入，可接常开、常闭。
	GX6	未配置	开关量输入，可接常开、常闭。
	GX7	未配置	开关量输入，可接常开、常闭。
	GX8	Z 轴负向限位	开关量输入，可接常开、常闭。
轴控	X 轴	X 轴轴控接口	与外部驱动器连接，具体参考图 3-4
	Y 轴	Y 轴轴控接口	与外部驱动器连接，具体参考图 3-4
	Z 轴	Z 轴轴控接口	与外部驱动器连接，具体参考图 3-4
手轮输入	手轮	电子手轮接口	手轮引脚参考图 3-5
IO 输出	YM1	自动加工完成	晶体管输出
	YM2	冷却	晶体管输出
	YM3	绿灯	晶体管输出
	YM4	红灯	晶体管输出
	YM5	抱闸	晶体管输出
	YM6	拓展输出 2	晶体管输出
	YM7	扩展输出 1	晶体管输出
	YM8	黄灯	晶体管输出
主轴	GND	公共端	电源地及开关量公共端。
	FWD	主轴使能输出端	主轴启动，一般接变频器正转信号。
	REV	主轴使能反转	主轴启动,一般接变频器反转信号
	ALM	主轴报警	主轴报警
	AVI	主轴调速	主轴调速输出
	网口	RJ1	网口
RJ2		网口	SLBUS 协议接口

图 3-4 系统轴控接口

端口 ( IN )	定义	说明	端口 ( IN )	定义	说明
01	A+	A 相反馈+	09	SON	伺服 ON
02	A-	A 相反馈-	10	ALM_RST	报警清除
03	B+	B 相反馈+	11	PUL+	脉冲输出+
04	B-	B 相反馈-	12	PUL-	脉冲输出-
05	C+	C 相反馈+	13	DIR+	方向输出+
06	24V	24V 输出	14	DIR-	方向输出-
07	C-	C 相反馈-	15	GND	公共端
08	ALM	报警输入			

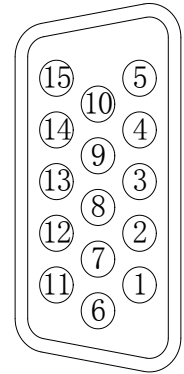
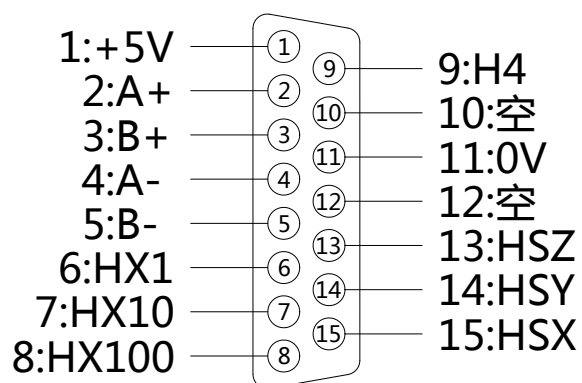


图 3-5 手轮接口定义



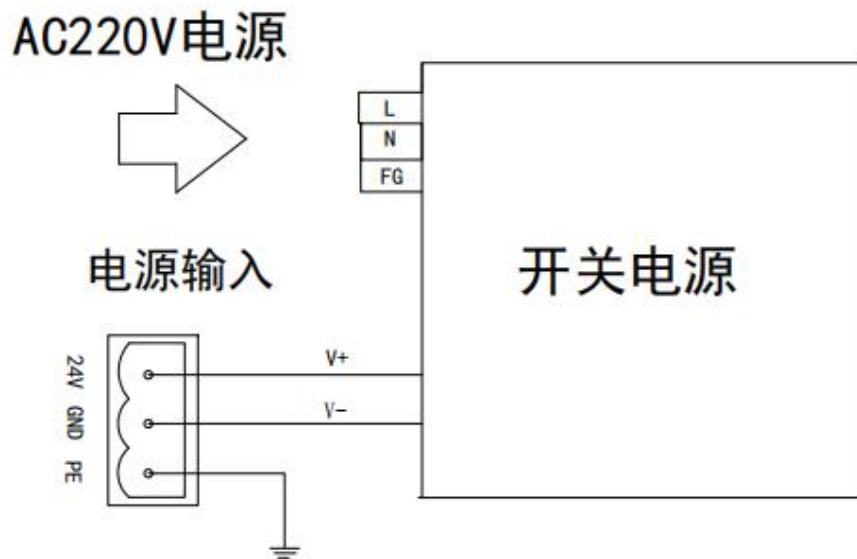
端口 ( IN )	定义	说明	端口 ( IN )	定义	说明
1	+5V	为手轮供电	9	H4	选择 4 轴
2	A+	编码器 A 信号	10	空	空
3	B+	编码器 B 信号	11	0V	数字地
4	A-	编码器 A 信号地	12	空	空
5	B-	编码器 B 信号地	13	HSZ	选择 Z 轴
6	HX1	选择 X1 倍率	14	HSY	选择 Y 轴
7	HX10	选择 X10 倍率	15	HSX	选择 X 轴
8	HX100	选择 X100 倍率			

表 3-1 系统端子定义说明

### 3.2 接线示例

L68 系统采用直流供电，系统额定功率为 24V/2.2A，如图 3-2 所示，请按外部继电器和的电磁阀等其他外接配件的实际使用，配备足够功率的开关电源。建议使用防水电源。

图 3-6 电源输入接线图





## 第四章 系统操作界面

### 4.1 系统界面

系统界面由标题栏、菜单栏、状态栏、工具栏、加工轨迹窗口和功能窗口组成。如下图图 4-1 所示：

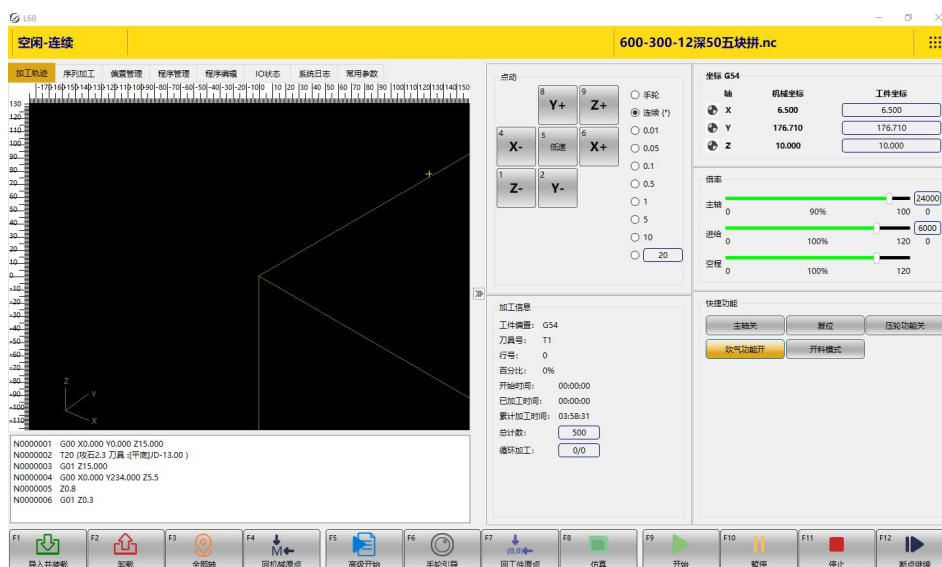


图 4-1 系统界面

**标题栏：**显示软件名称和已装载文件名

**菜单栏：**包含多个下拉子菜单，分别代表六类主要操作，“自动”、“手动”、“参数管理”、“进给速度”、“辅助”、“控制界面”。可通过右上角九宫格进入【菜单栏】选择功能对应菜单项执行某个动作或实现某个功能。



图 4-2 菜单栏

**工具栏：**左侧为快捷操作按钮通过按钮执行对应操作。右侧为信息提示框，显示报警或提示信息

**状态栏：**包括四个信息显示窗口：“坐标信息”、“倍率”、“加工信息”、“快捷功能”，主要用于显示加工过程中的一些状态信息，通过点击显示按钮，更改对应的坐标或速度参数。

**加工轨迹窗口：**用来显示仿真或加工的三维轨迹图像，可通过放大、缩小、移动、查看加工细节、切换视角（加工轨迹窗口内单击鼠标右键）。

**多功能窗口：**包括：“对刀”、“程序管理”、“程序编辑”、“IO 状态”、“系统日志”、“常用参数”、“偏置管理”、“换刀”、“轴控调试”、“诊断数据”、“打孔”、“测板长”，每一个窗口分别代表一项分类功能。没有的窗口可以在菜单栏-参数管理-客户端设置里配置

## 4.2 工具栏

最下面是工具栏，工具栏由一些操作按钮组成，分别对应一些菜单命令或选项功能，鼠标可以直接点击这些按钮实现对应的指定功能。


工具栏右侧为信息提示框，显示报警或提示信息，方便于人机交互。




图 4-3 工具栏

工具栏按钮功能：

F1  : 导入文件并装载到系统

F2  : 卸载已装载文件

F7  : 回工件原点

F3  : 设置当前点（所有轴）为工件原点

F4  : 回机械原点

F5  : 高级开始，选定行号加工

F6  : 手轮引导

- F8  : 仿真
- F9  : 开始
- F10  : 暂停
- F12  : 断点继续
- F11  : 停止

### 4.3 状态栏

状态栏：包括五个信息显示窗口，如下图所示：



图 4-4 状态栏

“坐标”为坐标信息

显示当前机械坐标与工件坐标。以随时将当前点位置设置为工件原点，只需鼠标点击工件坐标，便弹出对话框，点击“确定”就可以把对应轴的坐标位置设为工件原点。

“倍率”为进给速度区

可以设定进给速度，调整进给倍率、显示倍率和进给速度实际值。另外还显示手动高低速，G00 固定速度。

在自动加工过程中，用户可以通过鼠标拉动滑杆调节进给倍率来进行加工速度的调整。或是通过【进给速度】的子菜单选择调节进给倍速。

鼠标拉动滑杆，可以在 0~120%范围内调节当前运动速度倍率，进给倍率以百分数的形式显示出来，实际最高速度=设定速度值\*速度倍率。

系统空闲时，点击设定值的速度框，会弹出修改速度对话框，输入数值，点击确定，即可完成修改默认进给速度和空行速度值。加工速度的大小设置不能超过参数中设置的单轴最大速度，否则系统报错。



图 4-5 调整进给速度

可以设定主轴速度、调整主轴倍率、显示倍率和主轴速度实际值。还可以在快捷功能里启动/停止主轴的旋转。通过鼠标拉动滑杆和修改主轴速度参数调整主轴速度。



图 4-6 调整主轴转速

系统空闲时，点击设定值的速度框，回会弹出修改速度对话框，输入数值，点击确定，即可完成对加工速度的修改。加工速度的大小设置不能超过参数中设置的主轴最大速度，否则系统报错。

“加工信息”为加工信息区

显示当前工件坐标系、当前行号、开始加工时间、已加工时间、完成百分比，以及当前所用刀具的刀具号，分别 T1、T2... 显示。

“点动”为轴控窗口

初始默认为连续，此时黑色圆点在连续圆框中；如果连上手轮则默认为手轮，此时黑色圆点在手轮圆框中；下方 0.01—10 为步进距离，如黑点位于 1 的圆框内（即为选中状态）单次点动某一轴即可控制机床行进 1mm，最下方蓝色框为自定义步进距离。

“快捷功能”为轴控窗口

可以手动控制主轴启停，压轮开关，吹气开关，开料模式，可以在【参数管理】界面的【自定义快捷输出】添加的自定义快捷输出，也会在该窗口显示。

#### 4.4 加工轨迹窗口

在机床执行加工程序或仿真的时候，加工轨迹窗口可以实时跟踪刀具加工轨迹。通过

跟踪加工轨迹的三维实时显示功能，用户可以直观的检测刀具所走路径，以确认加工正确。

在三维跟踪模式，鼠标右键点击加工轨迹窗口，可以切换不同视图，有正视图、仰视图、俯视图、后视图、左视图、右视图、西南等轴视图、东南等轴视图、东北等轴视图、西北等轴视图，方便用户从不同角度，以合适的缩放比例查看图形。如图 4-7 所示。

用户可以通过鼠标滚轮缩小/放大、按住鼠标左键拖动当前显示的加工轨迹。当进行第二次加工或手动加工时，若需要清除先前的加工轨迹，可点击鼠标右键点击【清除】以免混淆。

除加工轨迹窗口外，还有程序管理、程序编辑、I/O 状态等窗口，可通过屏幕上方按键“程序管理”、“程序编辑”、“系统日志”进行切换。

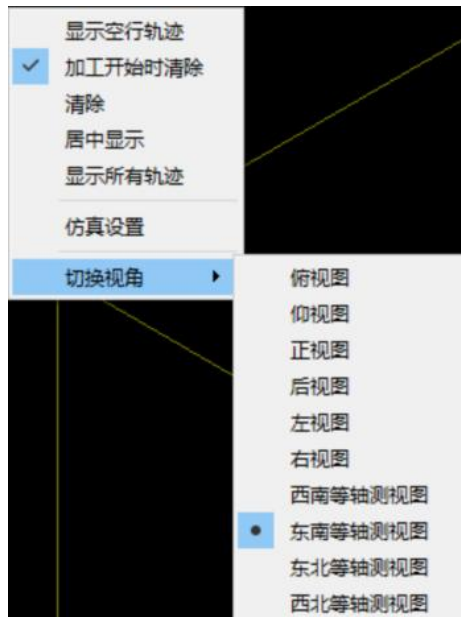



图 4-7 加工轨迹窗口

## 第五章 导入加工文件

输入加工文件有四种方法:1 导入并装载、2 在系统内手动编写、3 复制粘贴到共享文件夹。第一种方法一般使用于常用的加工文件导入到系统, 第二种适用于比较简单的加工文件, 第三种适合批量导入到系统。

### 5.1 在程序管理导入

点击【程序管理】界面, 点击程序管理界面下方的“导入并装载”功能键或主界面下方的‘’进入文件选择窗口, 选择要导入的加工文件点击确定, 在程序管理界面可以看到加工文件已导入系统并且装载到系统。

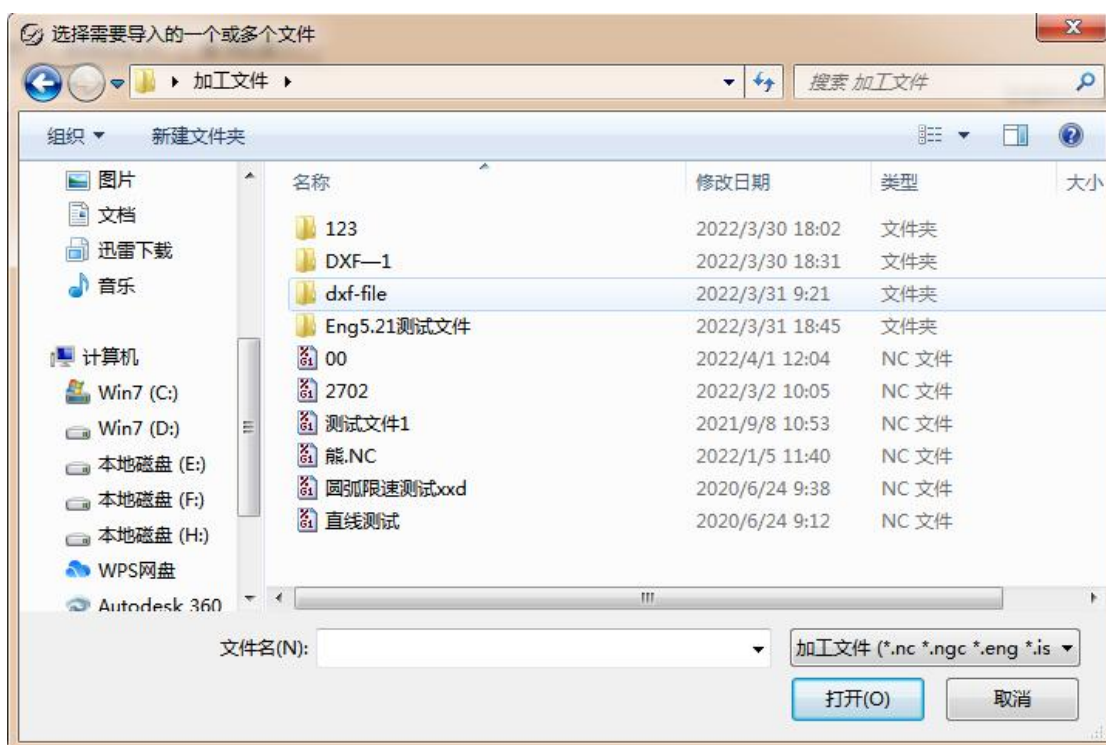


图 5-1 选择加工文件

### 5.2 在系统内手动编写

点击【程序管理】界面, 新建 nc 文件并选择新建 nc 文件, 然后点击窗口下方的“编辑”按钮或点击弹框的“OK”按钮, 即可对文件进行编辑。用户编辑完毕后, 点击鼠标右键选择保存。



图 5-2 编辑加工文件

### 5.3 复制批量粘贴到共享文件夹

打开共享文件夹，选择复制需要导入到系统的文件，粘贴到共享文件夹‘processfiles’文件夹路径下，然后在‘程序管理’中点击‘刷新’即可看到复制粘贴的加工文件。

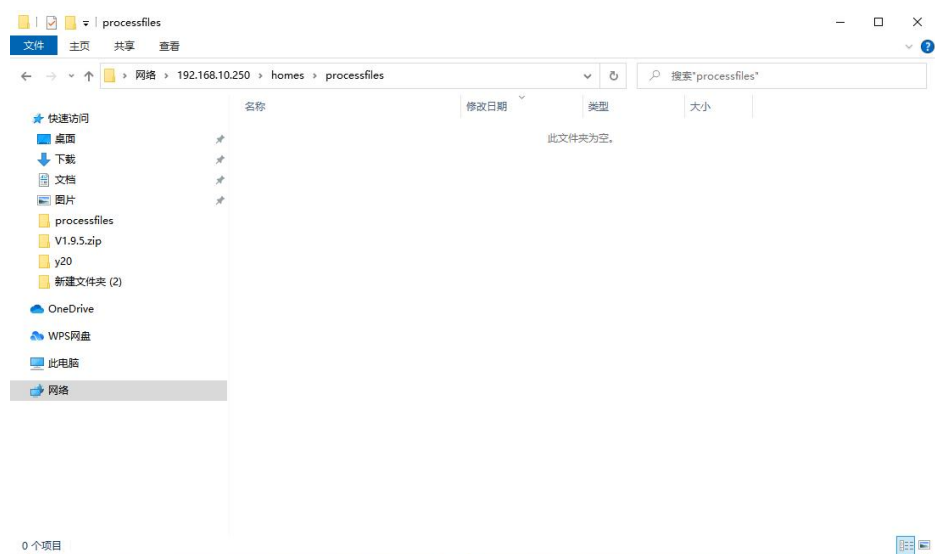


图 5-3 共享文件夹文件

---

## 第六章 回机械原点

### 6.1 回零前调试

回机械原点前需进行回零前调试，回零前调试主要包括调整脉冲当量、检查机床轴方向、设置工作台行程、原点信号限位是否有效。

#### 6.1.1 调整脉冲当量

脉冲当量：脉冲当量值越小，机床加工精度和工件表面质量越高；值越大，机床进给速率越大。

选择【参数管理】菜单中【参数设置】菜单项，点击厂商参数的轴参数设置 20101 脉冲当量。

#### 6.1.2 检查机床轴方向

设置好脉冲当量，确保轴移动的位置与实际情况一致。

选择【参数管理】菜单中【参数设置】菜单项，点击用户参数的轴参数查看 20100 轴方向设定值。然后点击【手动】窗口，选择连续或步长模式，移动轴，查看轴运动实际方向是否与参数设置一致。方向一致，则机床轴方向设置正确；方向不一致，则将轴方向参数值设与原来方向相反。

#### 6.1.3 设置工作台行程

根据机床实际大小设置工作台行程与使用工作台行程是否有效

选择【参数管理】菜单中【参数设置】菜单项，点击厂商参数的进给轴参数根据机床实际尺寸设置工作台行程下限和上限（20302 软限位使能、20303 软限位最小位置、20304 软限位最大位置）。

### 6.2 回机械原点

机械原点是机床的一个固定位置，由机械开关和电气系统共同确定，是机械坐标系的零点。执行“回机械原点”功能需要机床本身安装有原点开关，如果机床没有相关的硬件支持，则需要禁止该功能，详见 第十一章 参数管理 中的“原点参数”设置。由于机械原点是整个机床的基准，所以，该功能的重要作用在于校正当前点坐标。为防止断电或者是造成当前位置不正确，请在程序启动或发生急停之后执行回机械原点操作。

在系统上电启动后，将自动弹出回机械原点对话框，点击按钮，对应轴将自动回到机械原点，并且校正系统坐标。在 X 轴或 Y 轴回原点之前，请先将 Z 轴回到机械原点。

选择“手动”菜单中“回机械原点”菜单项系统将弹出回原点对话框如下图所示：



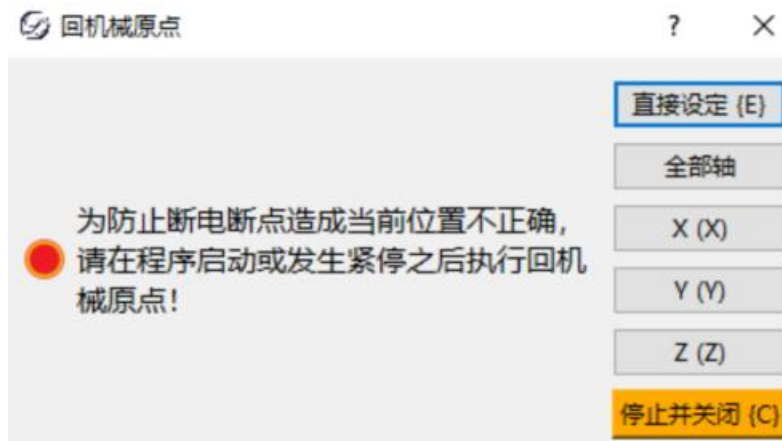


图 6-1 回机械原点功能画面

单轴各自回机械原点包括：

直接设定：如果确定当前位置与机械坐标一致，并且机床没有关闭过，机床没有执行过急停，可采用直接设定。

X 轴回机械原点：选择此命令，X 轴回机械原点。

Y 轴回机械原点：选择此命令，Y 轴回机械原点。

Z 轴回机械原点：选择此命令，Z 轴回机械原点。

全部轴回机械原点：选择此命令，全部轴回机械原点。

**注意：**

如果没有执行回机械原点，在手动操作时请先将 Z 轴尽量抬高，确保刀头不待加工工件不会发生碰撞。

本系统在退出时，都会自动保存当前坐标信息。如果在自动加工过程中，出现突然掉电情况，系统会自动将掉电前的相关信息保存到断点保护文件中（断点保护文件即掉电时将断点信息、文件名等保存到系统内存中，同一个加工文件只与一个断点保护文件对应）。电力恢复后，系统会弹出提示框，提示用户上次某加工文件发生掉电。用户需先人工进行回机械原点操作，然后可继续加工上次发生掉电的文件，也可重新选择新的加工文件：

1. 若用户想继续加工上次发生掉电的文件，可点击主界面左上方工具栏的“断点继续”按钮，机床将会返回掉电前的位置，点击“开始”后，机床将从断点处继续无缝加工掉电前尚未加工结束的文件。

2. 若选择新文件进行加工，在加工结束后，用户仍可继续加工之前发生掉电的文件，机床将会从对应文件的断点处继续无缝加工。

## 第七章 设置工件原点

加工文件前，用户需通过手动操作调整刀具与工具与工件的位置，以便从工件的预订位置开始加工。

Z 轴的工件原点设定有两种方式：1 手动设置、2 固定对刀

### 7.1 手动清原点

X、Y 轴工件原点设定：将 X 轴、Y 轴手动走到预定加工位置，通过点击工件坐标弹出对话框提示，根据提示确定将当前位置的 X 轴和 Y 轴坐标值清零（在坐标窗口点击 X 轴和 Y 轴坐标栏）。如下图所示：



图 7-1 设置工件原点

### 7.2 固定对刀

首先需要在参数管理中设置好对刀仪的机械坐标。执行固定对刀动作后，系统会自动移动到 X、Y 相应的机械坐标，然后开始 Z 轴对刀。

固定对刀分第一次对刀和换刀后对刀，用户在使用时需注意，第一次对刀是加工前的对刀，通过这次对刀确定刀尖与固定对刀块接触时的工件坐标。换刀后对刀，通过这次对刀，在刀尖与固定对刀块接触时恢复 Z 向工件坐标为第一次对刀动作中设定的值。固定对刀时，为了保护刀具，采用速度分段的方式，分为快速对刀速度和对刀速度两种，速度可以通过参数进行设置。当刀尖快要接近对刀仪时采用快速对刀速度，刀尖不对刀仪接触后采用正常的对刀速度。

### 7.3 浮动对刀

修改参数切换对刀模式，系统默认为固定对刀模式，固定对刀是否有效设置为‘否’，浮动固定对刀是否有效设置为‘是’，切换为浮动对刀模式。

浮动对刀可以使用户方便的确定工件表面高度，并设好 Z 轴工件原点。与手动对刀类似，由于通常 Z 轴的工件原点在旋转轴的中心，所以需要将刀块放置的位置离旋转轴中心的距离设置到公共偏置中。（或者将对刀块离旋转轴中心的距离加入到对刀块厚度中）具体操作如下：将对刀块放置于工件表面，通过手动操作将刀尖移动到工件原点上方，点击“测量工件表面”按钮，机床将进行

对刀动作，刀尖碰到对刀块后，自动上抬 10mm，再加上对刀块厚度,从而确定 Z 轴坐标。

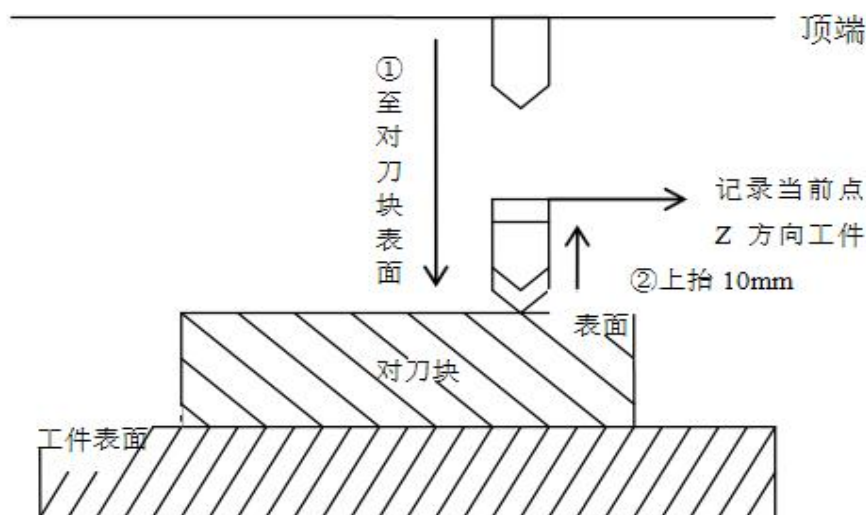


图 7-2 浮动对刀示意图

注意：

(1) 在对刀之前，用户必须保证刀尖位于对刀块的上方，即下刀后，刀尖能够碰到对刀块，否则会一直下刀，导致刀头与工件的毁坏。

(2) 对刀块厚度可以在厂商参数中设置，对刀后 Z 轴方向工件原点坐标会自动对其进行补偿。

(3) 由于工件表面与对刀块之间存在一定的间隙，在对刀完成后，可以在设置工件偏置中对间隙进行补偿。补偿差值视间隙大小而定，如果工件表面比较粗糙，则补偿差值可以大一点，如果工件表面比较光滑，与对刀块间隙较小，则补偿差值可设小一点。一般情况可以采用 0.1mm。

(4) 关于对刀速度，是在参数设置里进行的。对刀速度的范围在 60-1000mm/min，若对刀速度超过参数设置的最大值，则会磨损刀头或损坏对刀块。

注意：

#### 7.4 保存和选取工件原点

选择【手动】菜单中【保存工件原点】/【读取工件原点】菜单项，可以将当前工件原点保存，点击【读取工件原点】即可将保存的工件原点设置为当前工件原点。最多可保存 10 个工件原点。

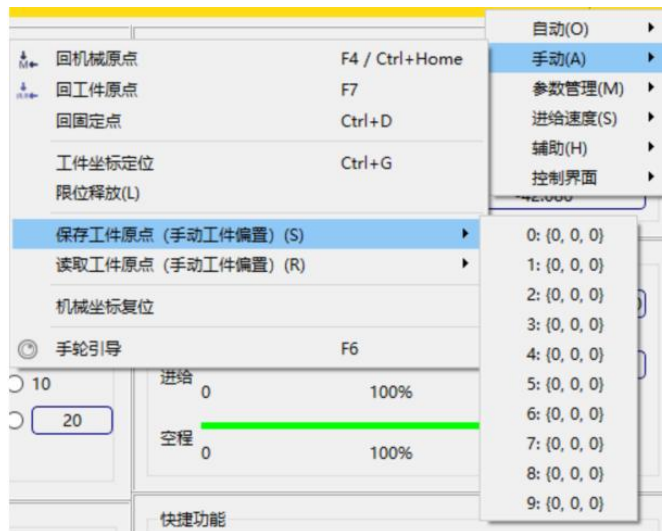


图 7-3 保存工件原点

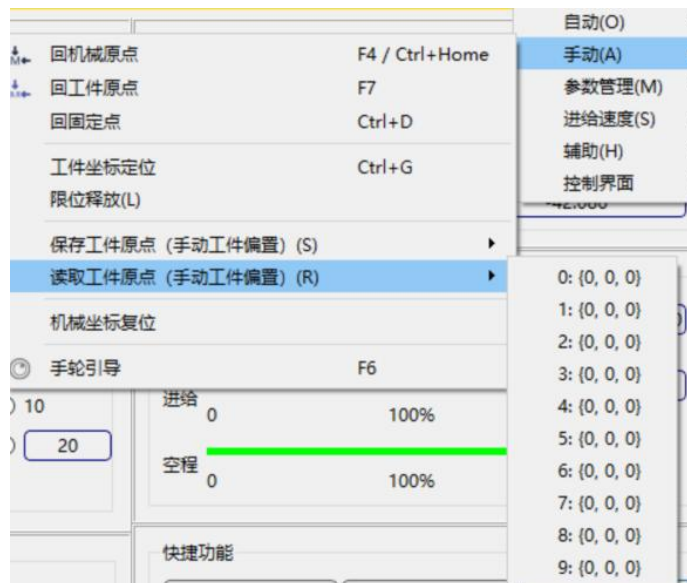


图 7-4 读取工件原点

**提示：**

用户可以选择“手动”菜单中“保存工件原点”菜单项，将当前工件原点保存到加工文件系统中。这样用户可以将频繁使用的工件原点保存起来，作为一个预置值。当用户第一次设定好工件原点，以后想要快速定位到这个工件原点，那么就可以使用这个功能来保存好这个工件原点坐标值。

## 第八章手动加工

手动加工指机床按用户所设置的参数对加工程序手动行行加工。手动操作机床有四种方式：手轮脉冲方式、连续点动方式、增量步进方式和自定义步长方式。

用户可选择手动操作模式对程序文件进行加工。您可以在该界面上行行相应的手动操作。在该窗口的手动按钮区包含六个手动按钮，分别对应 X、Y、Z 轴的正负方向。手动窗口为用户以手动方式操纵机床提供了一个交互式的操作环境。

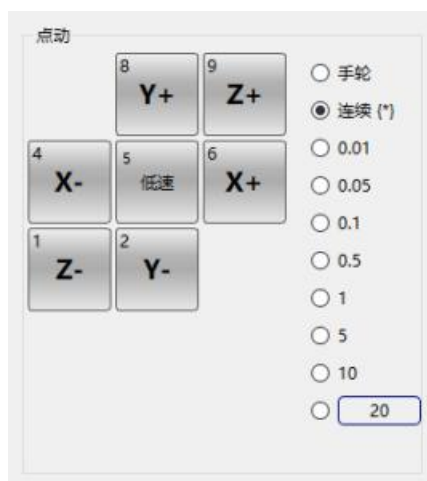


图 8-1 手动加工窗口

### 8.1 手轮脉冲方式

用户可以选择手轮脉冲方式进行连续加工。将手动加工窗口右侧的单选按钮切换到“手轮”选项时，机床的运动靠手轮输入来决定。

1. 手轮上有脉冲倍率选择：分别为 $\times 1$ 、 $\times 10$ 、 $\times 100$  档，表示手轮不同的脉冲倍数。

2. 手轮上有轴的选择：用户可选择需要进给的 X、Y、Z 轴。

3. 手轮上有步进方向选择：每个轴都有正、负向，在手轮上选择好步进轴后，可正、负向摇动手轮，对应手轮上方向旋钮“+/-”箭头所示。

4. 手轮 $\times 1$ 、 $\times 10$ 、 $\times 100$  档位时，手轮每格对应的距离可通过参数设定。

5. 手轮加速度可单独设置。

6. 手轮支持两种模式：1 严格脉冲计数、2 非严格脉冲计数。在严格脉冲计数的情况下，机床所走的距离不手摇脉冲数是严格对等的(注意：在此模式下，

---

如果手摇脉冲过快时，可能回导致缓冲的脉冲过多，当停止摇动手轮时，机床还会走相当长一段距离)。在非严格脉冲计数模式时，手摇脉冲数不机床所走的距离不严格对等，当手轮一停止，机床立即开始减速停止。

**注意：**

在执行操作前，先检查外部手轮设备已正确连接。该模式主要用于机床的快速定位。

## 8.2 连续点动方式

通过鼠标选中窗口右侧的“连续”单选按钮，进入连续点动加工方式。在这种方式下，可用鼠标点击手动按钮或。当鼠标按下相应轴的手动按钮时，机床动作；鼠标弹起时，机床停止动作。

在执行点动动作时，轨迹显示窗口显示相关的加工轨迹。

## 8.3 步进方式

与连续点动方式类似，步进方式是另一种手动操作机床模式，与连续点动方式不同的是，步进方式可以精确地控制机床运动轴的进给距离。

在使用该方式操作之前，必须设定合适的步长，通过修改点动步长，设定每次点动进给的距离。

步长设置可通过以下方式进行：

进入手动页面，鼠标点击  ，弹出对话框，填写步长值。如下图所示：



图 8-2 自定义步长窗口

**注意：**

要避免把 Z 方向的点动步长值设置过大，以免由于误操作而损坏机床。

## 第九章 加工操作

### 9.1 装载文件

点击“程序管理”切换到程序管理窗口，选中本次要加工的文件，点击“装载”或双击鼠标。装载完成后，主界面上方工具栏左侧将显示文件装载成功，主界面上方工具栏右侧显示已装载的文件名，此时主界面左下方显示加工文件内容。

```
N0000001 G00 X0.000 Y0.000 Z15.000
N0000002 T20 (攻石2.3 刀具 (平底)JD-13.00 )
N0000003 G01 Z15.000
N0000004 G00 X0.000 Y234.000 Z5.5
N0000005 Z0.8
N0000006 G01 Z0.3
```

图 9-1 “装载”加工文件

### 9.2 设置工件原点

详见第七章。如果工件原点已经设置好，无需重新设置。

### 9.3 自动加工

“自动”菜单中包含了与自动加工有关的各项。



图 9-2 “自动”菜单

#### 9.3.1 开始

用户选择了加工文件后，点击主界面下方工具和状态栏的“开始”按钮图



标：或是快捷键“F9”，机床将按所选加工文件从第一行开始进行自动加工。在加工轨迹窗口中可看到加工轨迹窗口根据刀具的移动显示出相对应的加工轨迹；在自动加工窗口中可看到程序逐行被加工，光标会自动跟踪到当前代码，红色的高亮显示正不停向下滚动，用户通过这个窗口可以查看当前加工程序代码信息。

**注意：**

如果要进行参数设置，加工前必须回机械零点，系统会提示先回机械原点。在未回机械原点的情况下不能执行自动加工指令

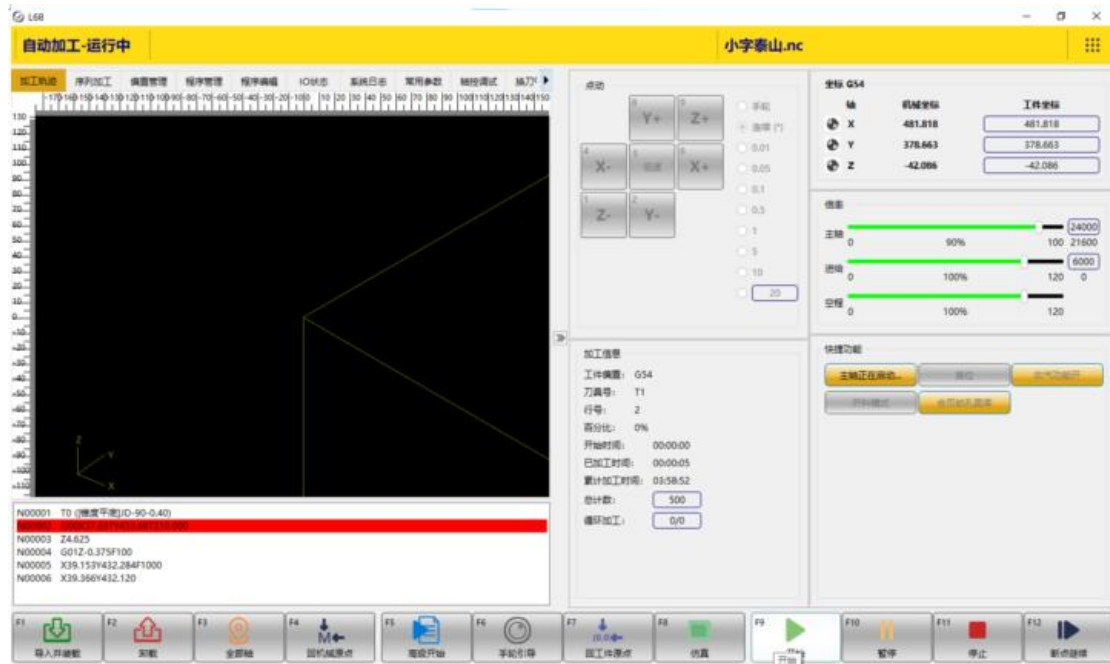


图 9-3 自动加工界面状态



图 9-4 工具栏中当前自动加工状态提示

注意：

系统会对自动加工文件边加工边进行语法检查，且语法检查比自动加工提早进行（即语法检查具有“前瞻”功能）。若系统检查出加工文件中某行程序语法出错，则在自动加工窗口中高亮显示出错语句并报警，同时自动加工停止。用户可对出错语句进行语法、语义检查和修改编辑，修改编辑后点击保存，然后再次点击主界面左上方工具栏的“断点继续”按钮，则程序将从被修改处继续自动加工。

自动加工过程中，不能加载新的加工文件。在状态栏“加工信息窗口”中会显示当前文件开始加工时间、已加工时间、当前正在使用的刀具号信息。方便用户查看自动加工的运行情况。

在进给速度区可调节滑杆或更改进给速度设定值来重新设定进给速度；主轴转速区可调节滑杆或更改主轴转速设定值来重新设定主轴转速，数值更改后将立即生效。

自动加工的开始和结束信息都保存在系统日志文件中。系统日志记录用户重要的操作和发生的事件，用户不仅可以从系统日志窗口浏览自从这次启动以来发生的日志信息，而也可以通过该窗口回顾曾经发生的历史信息的纪录。如果系统发生故障，该功能可以帮助您进行系统分析和诊断。



加工轨迹		序列加工	偏置管理	程序管理	程序编辑	IO状态	系统日志	常用参数
	时间	类型	模块	消息				
1	2024-06-13 10:19:41	错误	控制器: 脚本	脚本执行错误: ...				
2	2024-06-13 09:03:42	错误	控制器: 脚本	脚本执行错误: ...				
3	2024-06-13 09:03:34	提示	控制器: 默认	---- 启动程序: L68 2.1.16 ----				
4	2024-06-13 09:03:32	提示	客户端: 默认	---- 启动程序: L68 2.1.16 ----				
5	2024-06-12 17:48:11	错误	控制器: 脚本	脚本执行错误: ...				
6	2024-06-12 17:48:01	提示	控制器: 默认	---- 启动程序: L68 2.1.16 ----				
7	2024-06-12 17:47:59	提示	客户端: 默认	---- 启动程序: L68 2.1.16 ----				
8	2024-06-12 17:44:15	提示	控制器: 默认	---- 启动程序: L68 1.6.0 ----				
9	2024-06-12 17:44:13	提示	客户端: 默认	---- 启动程序: 山龙数控系统 L68 1.6.0 ----				
10	2024-06-12 16:02:48	提示	控制器: 默认	---- 启动程序: L68 2.1.16 ----				
11	2024-06-12 16:02:46	提示	客户端: 默认	---- 启动程序: L68 2.1.16 ----				
12	2024-06-12 16:01:44	提示	控制器: 默认	---- 启动程序: L68 2.1.16 ----				
13	2024-06-12 16:01:43	提示	客户端: 默认	---- 启动程序: L68 2.1.16 ----				
14	2024-06-12 16:00:07	提示	控制器: 默认	---- 启动程序: L68 2.1.16 ----				
15	2024-06-12 16:00:05	提示	客户端: 默认	---- 启动程序: L68 2.1.16 ----				

导出 清除 刷新

图 9-5 系统日志功能

系统当前纪录的日志信息包括：


- (1) 自动加工开始和结束信息；
- (2) 工件坐标变动；
- (3) 系统报警信息；
- (4) 文件的加工完成信息；
- (5) 其他一些系统信息。

提示：

当系统日志信息只能存储 1000 条。

### 9.3.2 暂停

在“开始”自动加工后，如需暂停加工，点击主界面右下方工具栏的“暂

停”按钮图标：或是快捷键“F10”，机床将从当前速度开始进行减速，直到速度为零。

### 9.3.3 停止

在“开始”自动加工后，如果用户想中止加工文件，可点击主界面右下方工具栏“停止”按钮图标：或是快捷键“F11”，机床将从当前速度开始进行减速，直到速度为零并且抬刀。系统停止时会自动保存断点。

自动加工时，如果系统处于仿真状态，点击“停止”按钮，系统停止仿真，但并且退出仿真状态，这时用户可以分析仿真结果。

### 9.3.4 微调

加工过程中按自动菜单中的“微调”，弹出如图所示的界面：

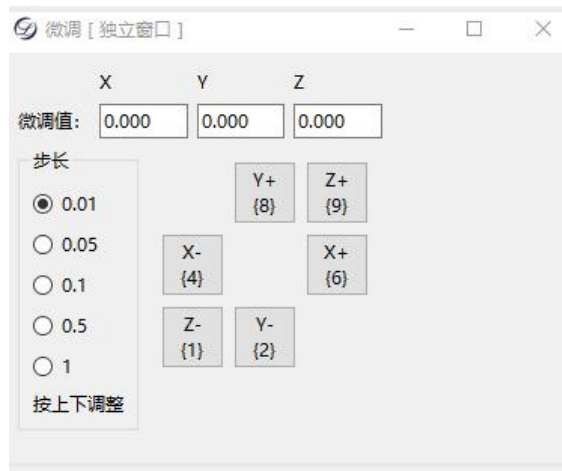



图 9-6 加工中微调

可选择步长，步长值为微调值，如界面所示数字键分别对应 X、Y、Z、A 轴不同方向上的微调。

### 9.3.5 断点继续


如果用户想从上次停止的地方继续加工该工件，点击主界面左上方工具栏的“断点继续”按钮图标：或是快捷键“F12”。如果此次断点继续是因为加工文件过程中发生掉电所致，则在断点继续之前必须先回机械原点。

加工中突然掉电可通过点击“断点继续”按钮，恢复现场，系统会从上次加工中断处开始继续加工。“断点继续”也可以用于仿真继续执行。

### 9.3.6 高级开始

若不需要对整个文件进行加工，只要从加工文件中指定的某行开始，到指定的行号加工结束，即自动加工中的“跳段执行”。也可用于检查加工文件中的某段程序是否正确。



点击：或是快捷键“F5”，或者在“自动”菜单中选择“高级开始”菜单项。该功能实现了程序跳段执行和最近点加工的功能。选择该功能，系统弹出“执行（高级选项）”对话框，如图 9-7 所示：

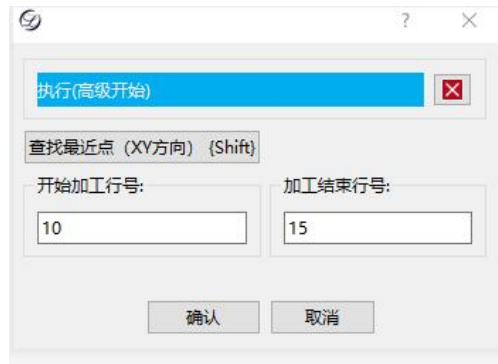


图 9-7 高级开始

用户在对话框中设定好文件开始位置行号、结束位置行号，然后点击“确定”按钮，机床将按您的要求只执行整个加工程序中的指定程序段。

**注意：**

若选择从文件开始到文件结束，则对整个程序文件进行加工。即是跳段执行的**最大范围**。

若停止加工后移动了 XY 轴，要继续从停止点加工，可移动到停止加工的地方，点击【**查找最近点**】继续加工

### 9.3.7 阵列加工

用户可以对文件进行阵列加工，点击菜单栏-自动-阵列加工、弹框输入文件选择需要进行阵列的文件。然后阵列行数、阵列列数、阵列行间距、阵列列间距输入完参数后，点击生成加工文件，即会在程序管理目录下生成阵列文件，阵列后的文件名采用以下规则命名：原文件名阵列行数×阵列列数。

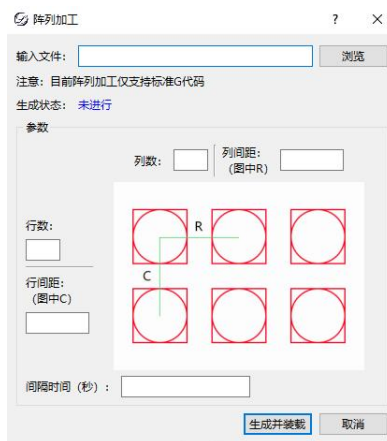



图 9-8 选择阵列文件

## 9.4 手轮引导加工

系统支持手轮引导加工，需要进入手轮引导模式，如下图所示：



图 9-9 手轮引导

选择界面下方工具栏里面的“手轮引导”选项，点击“开始”按钮图标：。由于当前是在手轮引导模式，机床并不会动。

摇动手轮，机床会根据你手轮摇动的快慢调整速度沿加工轨迹进行加工，当摇动手轮停止时，机床也停止。当继续摇动手轮时，机床会沿着原来的轨迹继续加工。整个加工过程由手轮进行控制。

**注意：**

手轮引导加工时，向手轮正方向摇动，才可以引导加工，手轮负方向摇动无效。

## 9.5 镜像加工

点击菜单栏参数设置【手动】【镜像旋转加工】进入以下界面，按方向键即可调整 XY 轴镜像以及±90, 180 度的旋转的加工方式，选择完毕点击确定键即可退出，此时点击【开始】加工的即为选择加工方式。

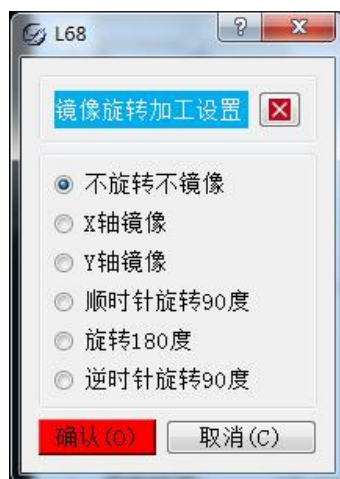


图 9-9 镜像加工

## 第十章 检查加工文件

当加工文件装载完毕，且当前系统状态为“空闲”，用户可选择“自动”菜单中的“仿真程序信息”选项（或者点击主界面左上方工具栏的“仿真”按钮图标：



或是快捷键：“F8”，对已装载加工文件进行高速仿真。

坐标信息	最小	最大	范围
工件X	0.000	87.250	87.250
工件Y	0.000	79.750	79.750
工件Z	0.000	30.000	30.000
机械X	-90.400	-3.150	87.250
机械Y	-90.000	-10.250	79.750
机械Z	87.800	117.800	30.000

图 10-1 仿真程序信息

仿真为用户提供了一个快速和真切的模拟加工环境。仿真开始后，系统不再发出脉冲驱动机床运动，仅仅只是在窗口中高速跟踪显示刀具加工后的实际效果。通过仿真，用户可以预先了解机床的运动情况和加工效果，防止编辑加工程序时的失误而造成机床的损坏，也可以了解其他一些附加信息。一旦仿真过程开始，该菜单项变成“停止仿真并退出仿真模式”，执行此功能，仿真将立即终止。

**提示：**

**仿真信息包括：**

(1) 当参数设定仿真限位有效时，系统在仿真过程中会检查是否会超行程。如果仿真时提示超行程，在不改变工件原点的前提下，实际加工也会超行程。

(2) 仿真过程中会对 G 代码进行语法检查，语法错误会报错。

在加工轨迹中，鼠标右键进入‘仿真设置’。



图 10-2 仿真设置

**启用控制器仿真：**启用控制器仿真时仿真会显示出‘文件总行号’、‘加工时长’、‘加工百分比’，不启用时则不显示。

**开始时弹出信息提示窗：**启用控制器仿真加工时，不能取消开始时弹出的信息窗口，只有取消控制器仿真加工时才能取消开始时弹出的信息窗口

**完成时弹出信息窗口：**启用控制器仿真加工时，会总是弹出，取消控制器仿真加工时，才能‘仅在超限时’弹出信息窗口

**绘图速度：**‘最快’取消时进度条设置速度才能生效

## 第十一章 铣底、铣边框操作

用户需要进行简单的铣底、铣边框时，无需手动编写 G 代码或者是用 CAM/CAD 软件生成加工文件，在【自动】-【执行加工指令】界面，采用系统提供的执行加工指令功能，输入几个参数即可完成。

执行加工指令当有矩形铣底、圆形铣底、铣矩形边框、铣圆形边框功能。

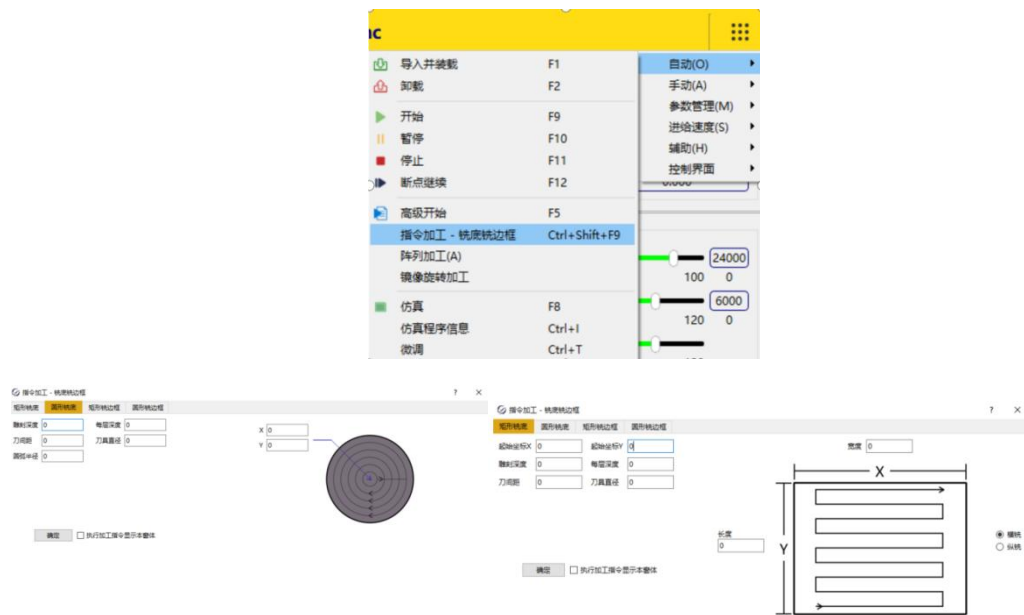


图 11-1 圆形铣底窗口

图 11-2 矩形铣底窗口

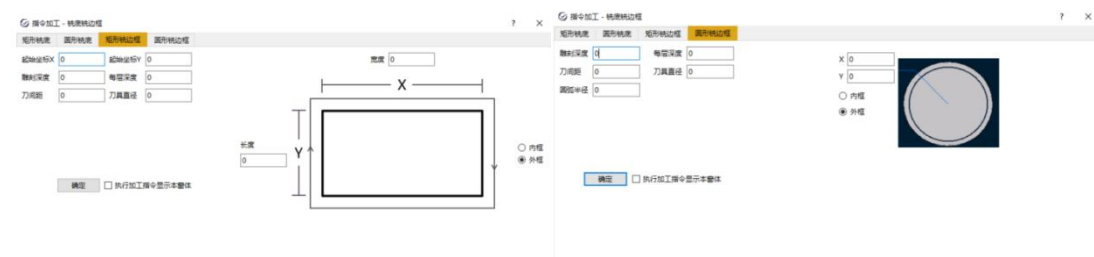


图 11-3 矩形铣边框窗口

图 11-4 圆形铣边框窗口

## 第十二章 程序管理

点击【程序管理】切换到程序管理窗口，可新建、编辑、删除、重命名、装载、卸载、导入、导入并装载、浏览文件夹、刷新加工程序文件。

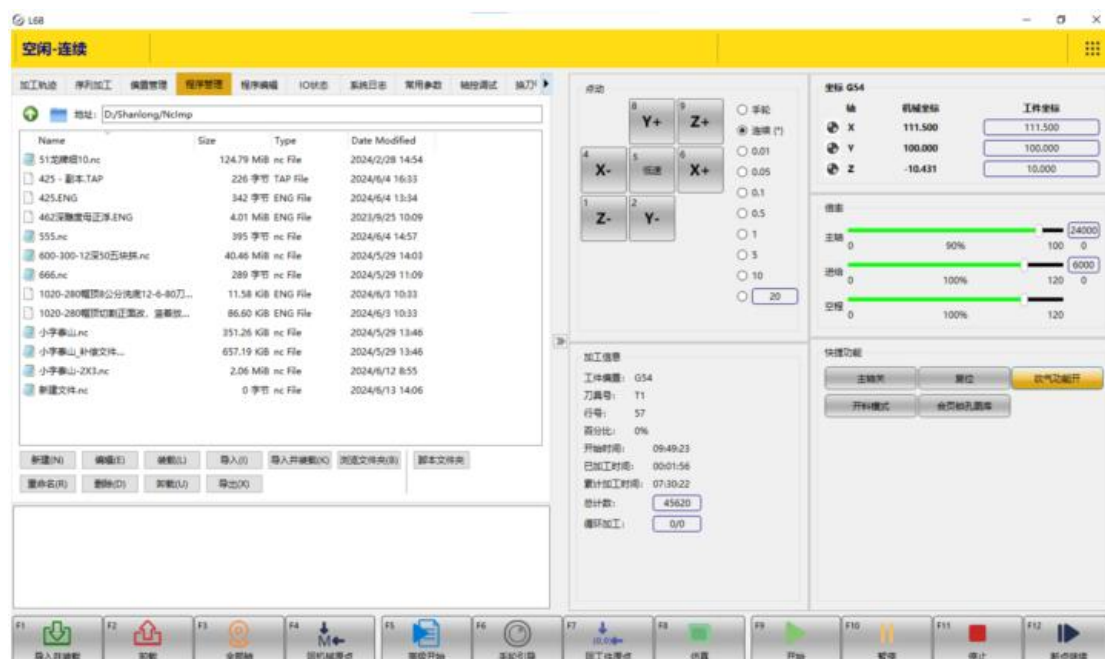


图 12-1 “程序管理”窗口

### 12.1 新建

点击【程序编辑】窗口下方“新建”按钮，在窗口中会新建一个默认命名（新建文件 1.nc）的空新文件。

### 12.2 编辑

选择已有文件或新建文件，点击窗口下方的“编辑”按钮，即可对文件进行编辑和修改。还可以点击【程序编辑】切换到程序编辑窗口，在编辑窗口中单击鼠标右键，会弹出上下文菜单，用户可以方便的进行复制，粘贴、剪切等功能快捷的实现程序的编辑和修改。用户编辑完毕后，点击鼠标右键选择保存或关闭再保存。



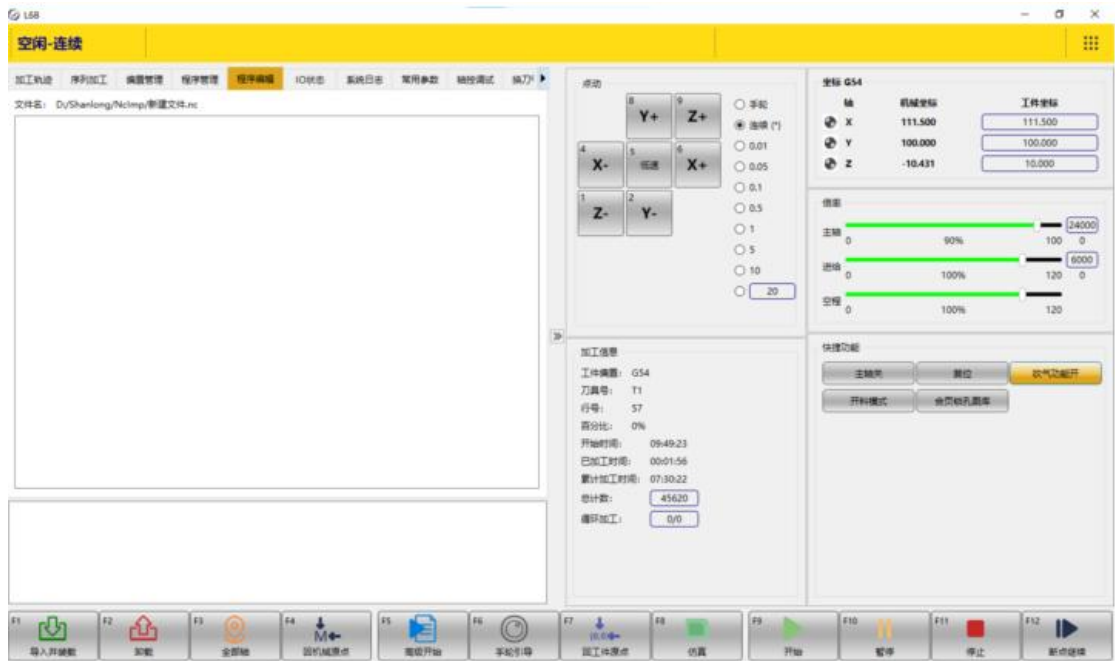


图 12-2 “程序编辑”窗口

对于新建或编辑过的文件，保存前系统将自动对其进行语法检查。用户必须按照我司的编程规范进行文件编辑，否则系统报错，详见《编程规范》部分。

#### 提示：

该编辑窗口可以编辑小于 10 兆字节大小的加工程序，如果大于 10 兆字节大小的文件，请在 PC 机使用与用编辑器进行编辑。

在编辑窗口中可由用户输入任意文本，输入完成后本系统会自动进行语法检查，从而保证机床不会执行错误指令而造成机床损坏。

### 12.3 删除

点击或移动键盘的向上、向下按键选定所要删除的文件，点击删除键，可对选定的文件进行删除。也可以选择中的某些文件进行批量删除。

### 12.4 装载

选中加工文件点击“装载”或双击鼠标进行装载。文件装载成功，在界面右上方也会显示装载的文件名。

## 第十三章 参数管理

【参数管理】菜单中包含了与参数有关的各项，如图：



图 13-1 参数管理菜单

### 13.1 设置参数

选择【参数管理】菜单中【设置参数】菜单项，弹出如下窗口，用于在参数权限下进行参数设置，分为用户参数和厂商参数两部分。

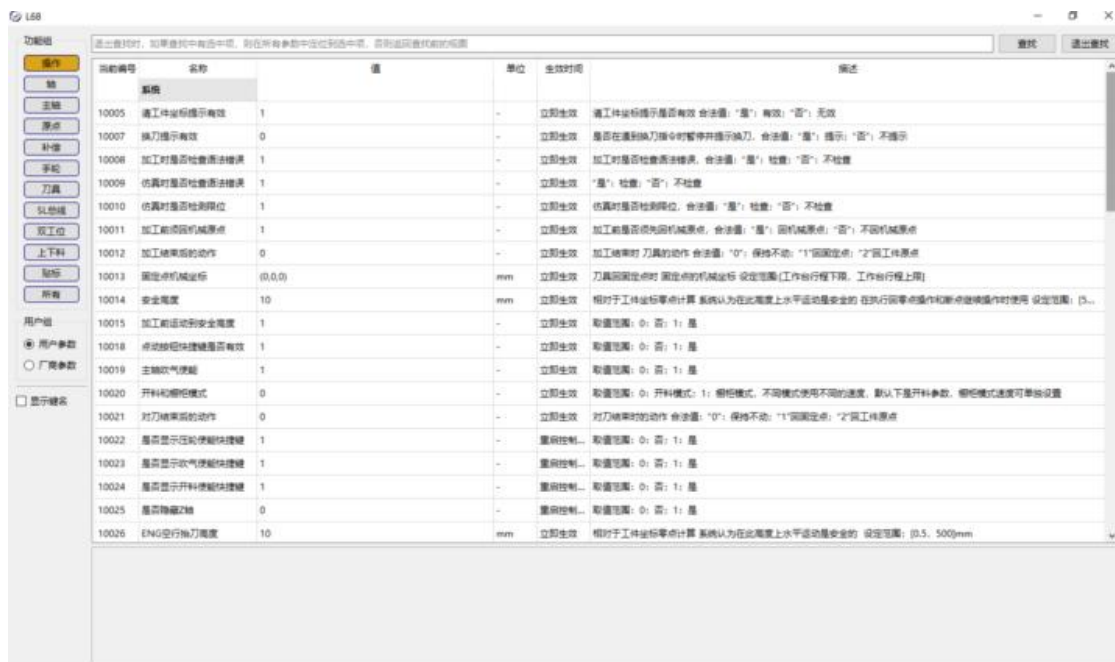


图 13-2 设置参数窗口

“用户参数”设置：如果对用户参数进行设置，则选择窗口左下角选项“用户参数”，然后分别对参数类别中的操作参数、进给轴参数、主轴参数、原点参数、补偿参数、刀具参数等进行设置。设置完成后，所有用户参数将生效。

“厂商参数”设置：首先选择参数权限，如果对厂商参数进行设置，则选择窗口左下角选项“厂商参数”，然后分别对参数类别中的操作参数、进给轴参数、主轴参数、原点参数、补偿参数、刀具参数等进行设置。设置完成后，所有厂商参数将生效。

注意：

一般情况下（默认状态），参数部分显示的是用户参数，供一般加工使用，若需要修改机器性能相关参数，如脉冲当量，主轴最大转速等参数，则要输入密码，开启厂商参数进行修改。

### 13.2 恢复厂商参数

用于将厂商参数恢复成刚出厂时的值。

选择【参数管理】菜单中【恢复厂商参数】菜单项，弹出如下窗口，点击“是”输入密码确定，用于将厂商参数恢复成刚出厂时的值。

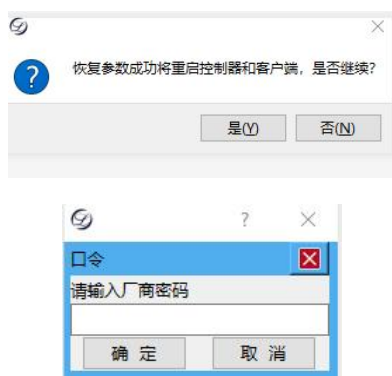


图 13-4 恢复厂商参数

### 13.3 备份参数

用于将系统的参数以文件的形式输出到 PC 机，以备数据恢复使用。

选择【参数管理】菜单中【备份参数到客户端路径】菜单项，弹出如下窗口，用于将系统的参数以文件的形式输出到 PC 机。

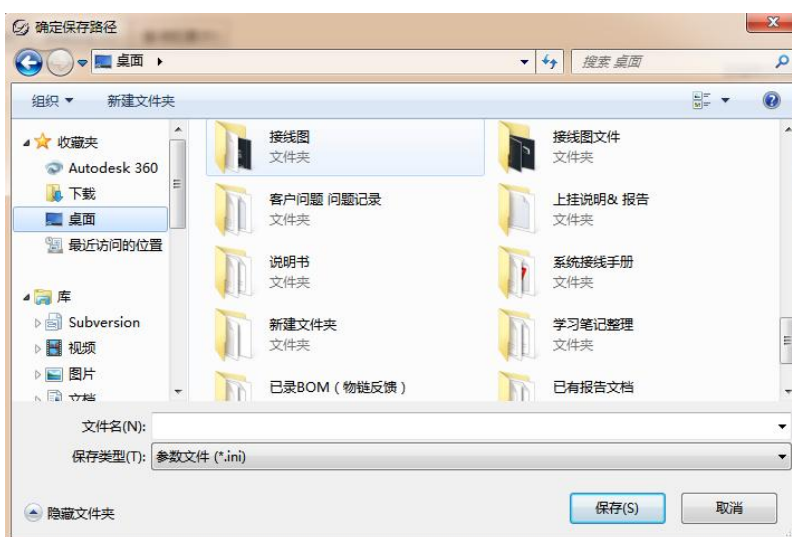


图 13-5 备份参数到客户端路径

### 13.4 恢复参数

用于将 PC 机保存的参数恢复到系统。

选择【参数管理】菜单中【从客户端路径恢复参数】菜单项，弹出如下窗口，用于将参数恢复成以前设置的值：在弹出的窗口中选择参数备份文件的名称，点击“确定”按钮，系统就会将参数重新恢复成当时设定的值。

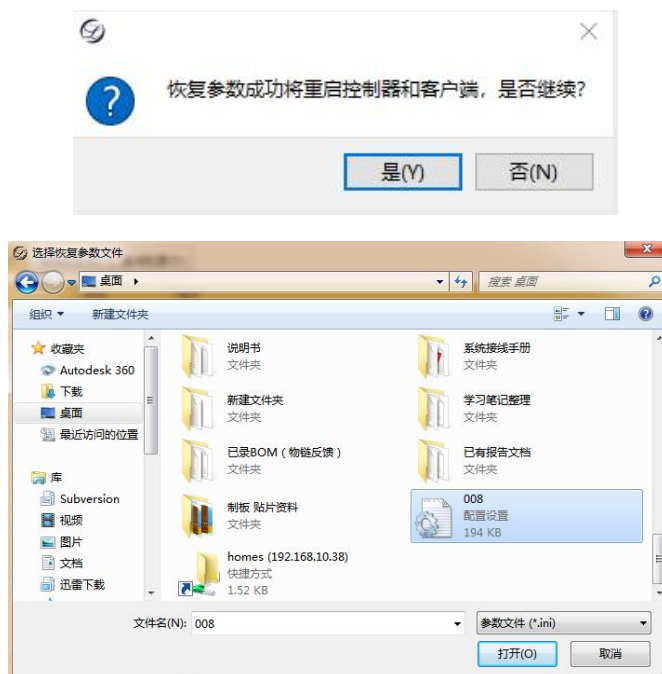


图 13-6 从客户端路径参数恢复

### 13.5 修改密码

选择【参数管理】菜单中【修改厂商密码】菜单项，弹出如下窗口，用于修改厂商密码，可以有效保护参数设置的安全性。

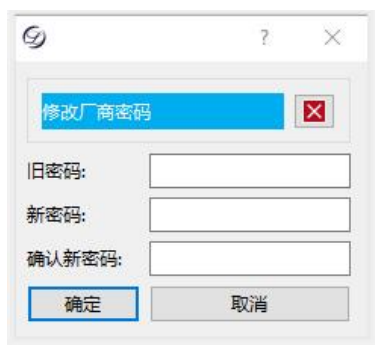


图 13-7 修改密码

出厂设备厂商密码为初始密码，修改密码时，‘旧密码’填写初始密码，在‘新密码’输入所要修改的密码，在‘确认新密码’再次输入所要修改的新密码，确认后，密码会立刻生效，再次进入厂商参数或恢复出厂参数时则需要输入新的密码

本系统中涉及到较多的参数，将参数分为用户参数和厂商参数两类。想要修改和查看某种类型的参数，必须要有查看和修改这种类型参数的权限。

### 13.6 客户端设置

选择【参数管理】菜单中【客户端设置】菜单项，弹出如下窗口，用于修改客户端界面布局、字体样式和大小，修改后，点击【辅助】菜单的【重启控制器系统】，重启修改生效。界面布局勾选的显示在 Table 界面，未勾选的显示在菜单的控制界面。

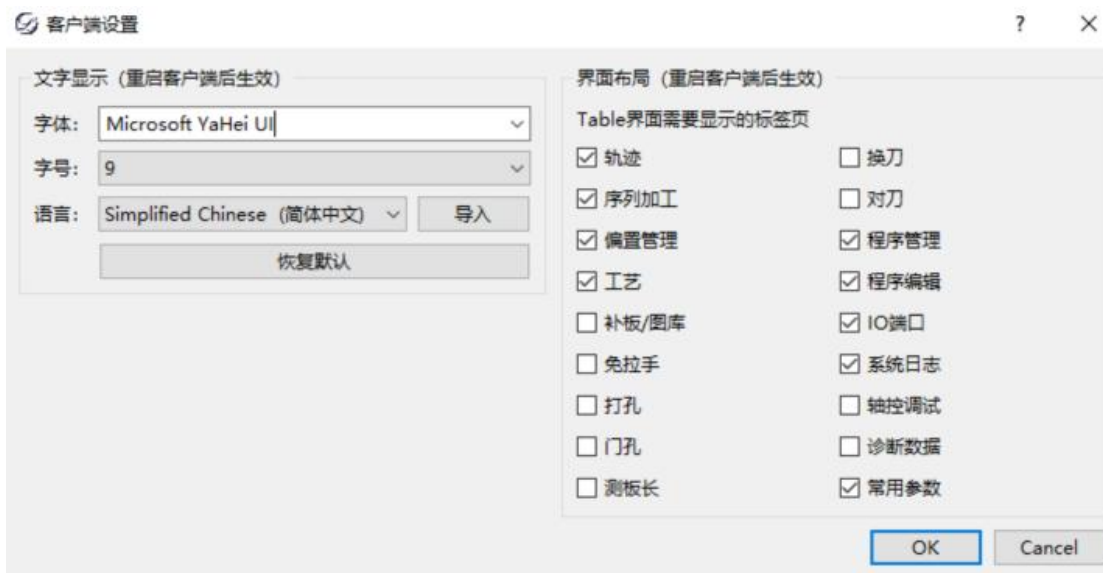


图 13-8 客户端设置

### 13.7 自定义快捷输出、自定义动作按钮、自定义 M 代码

点击自定义快捷输出，弹框点添加可以将冷却和真空吸附 1 添加到主界面右下角快捷功能中，

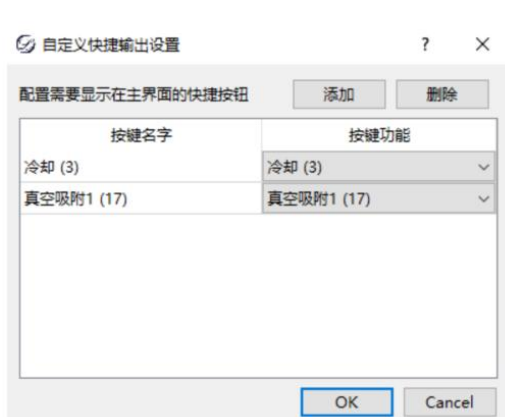


图 13-9 自定义快捷输出

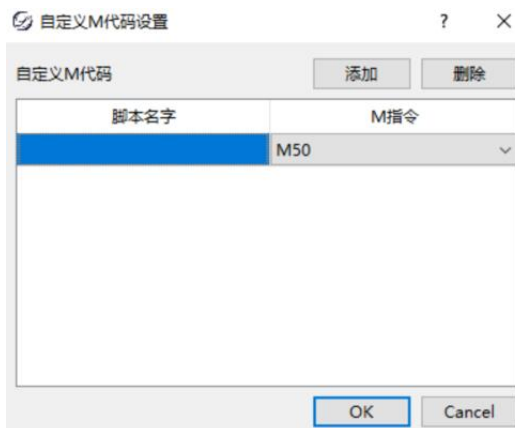


图 13-10 自定义 M 代码

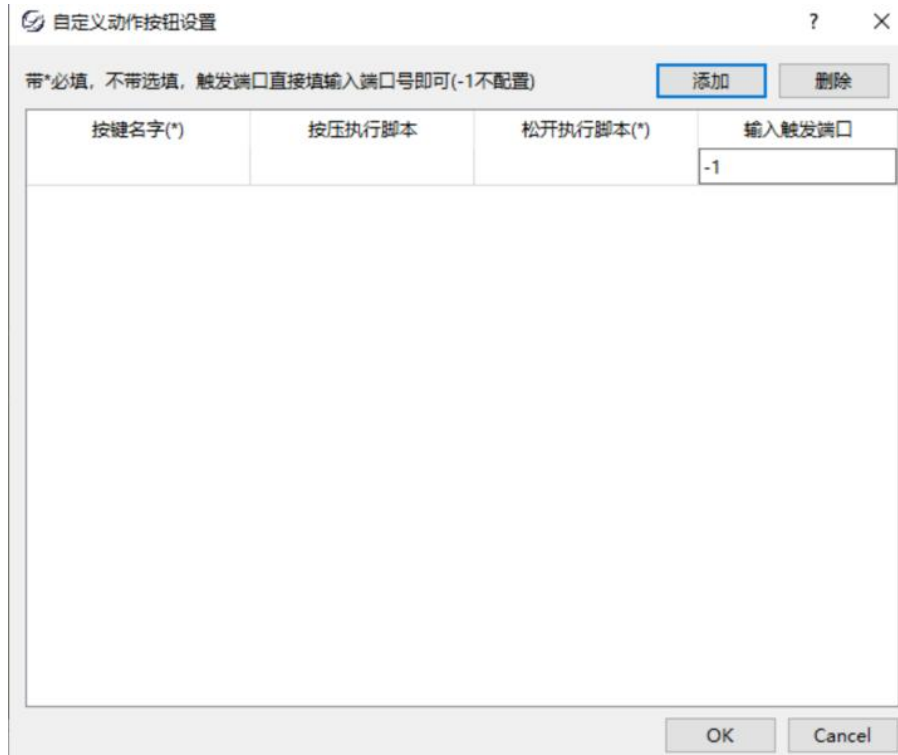


图 13-11 自定义动作按钮

### 13.8 刀具补偿设置

对各刀具的刀具直径、直径磨损、刀具长度、长度磨损进行设置。



图 13-12 刀具补偿

### 13.9 刀具偏置

对各刀具的 X 轴偏置、Y 轴偏置、Z 轴偏置进行设置。

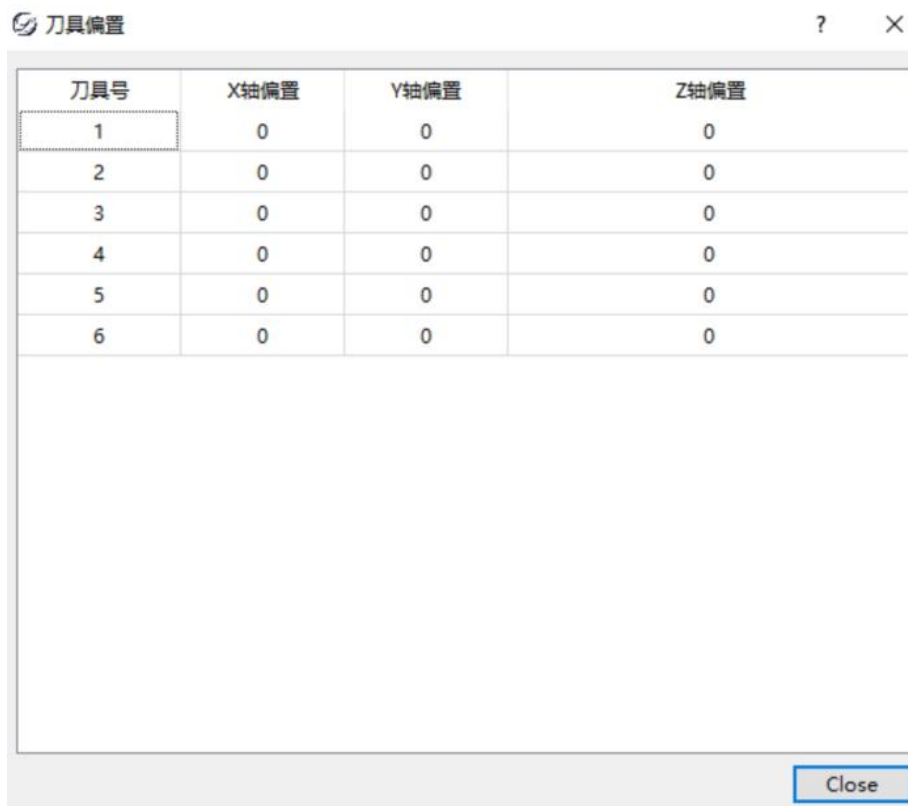


图 13-13 刀具偏置

### 13.10 参数修改方法

修改参数的方法是用鼠标双击参数所在行，弹出对话框，在参数区中输入数值，对于“是\否”型的参数，点击选择“是”或“否”。

参数生效时间分有立即生效和重启生效。参数生效时间为立即生效的参数修改，修改完成即可生效。参数生效时间为重启生效的参数修改，修改完成系统自动重启后才生效。

**注意：**

所有参数在加工状态下均不能修改，必须在加工完毕之后及下一个加工开始之前才可以修改。

### 13.11 参数总览

操作参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>1.1 . 系统</b>				
10005	清工件坐标提示有效	清工件坐标提示有效是否起作用	0 (否) : 无效 1 (是) : 有效	立即生效
10007	换刀提示有效	换刀提示是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
10011	加工前须回机械原点	加工前须回机械原点	0 (否) : 无效 1 (是) : 有效	立即生效
10012	加工结束后的动作	加工结束后的刀具动作	0 (否) : 无效 1 (是) : 有效	立即生效
10013	固定点机械坐标	刀具回固定点时固定点的机械坐标	软限位最小位置中的第 6 个值)~ (软限位最大位置中的第 1 个值, 软限位最大位置中的第 2 个值, 软限位最大位置中的第 3 个值, 软限位最大位置中的第 4 个值, 软限位最大位置中的第 5 个值, 软限位最大位置中的第 6 个值)	立即生效
10014	安全高度	相对于工件坐标零点计算系统认为在此高度上水平	5~500	立即生效
10015	加工前运动到安全高度	加工前运动到安全高度	0 (否) : 无效 1 (是) : 有效	立即生效
10016	弹窗报警是否有效	弹窗报警是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
10018	点动按钮快捷键是否有效	点动按钮快捷键是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
10019	主轴吹气使能	主轴吹气使能	0 (否) : 无效 1 (是) : 有效	立即生效
10020	开料和橱柜模式	橱柜模式可单独设置	0 : 开料模式 ; 1 : 橱柜模式	立即生效
10021	对刀结束后的动作	对刀结束后的动作	“0” : 保持不动 ; “1” 回固定点 “2” 回	立即生效



			工件原点	
10022	是否显示压轮使能快捷 键	是否显示压轮使能快捷 键	0 (否) : 无效 1 (是) : 有效	重启生效
10023	是否显示吹气使能快捷 键	是否显示吹气使能快捷 键	0 (否) : 无效 1 (是) : 有效	重启生效
10024	是否显示开料使能快捷 键	是否显示开料使能快捷 键	0 (否) : 无效 1 (是) : 有效	重启生效
10025	是否隐藏 Z 轴	是否隐藏 Z 轴	0 (否) : 无效 1 (是) : 有效	重启生效
10027	加工时是否打开吸尘 罩	加工时是否打开吸尘 罩	0 (否) : 无效 1 (是) : 有效	立即生效
10028	加工前 A 轴是否回 工件原点	加工前 A 轴是否回工件原 点	0 (否) : 无效 1 (是) : 有效	重启生效
10031	G00 和 G01 倍率是 否合并	G00 和 G01 倍率是否合 并	0 (否) : 无效 1 (是) : 有效	重启生效
10034	是否支持 3 轴和 4 轴切换	是否支持 3 轴和 4 轴切 换	0 (否) : 无效 1 (是) : 有效	重启生效
<b>1.2 相关文件</b>				
10008	加工时是否检测语法 错误	加工时是否检测语法错 误	0 (否) : 无效 1 (是) : 有效	立即生效
10009	仿真时是否检测语法 错误	仿真时是否检测语法错 误	0 (否) : 无效 1 (是) : 有效	立即生效
10010	仿真时是否检测限位	仿真时是否检测限位	0 (否) : 无效 1 (是) : 有效	立即生效
10029	G73 钻孔退刀类型	G73 钻孔退刀类型, 0 和 1 代表按退刀 量每次固定增量抬高, 2 代表按退刀表 面坐标每次固定走到设置的工件坐标	0~5	立即生效
10030	G73 钻孔退刀位置 变量	G73 钻孔退刀位置变 量	0~9999	立即生效
10026	ENG 空行抬刀高度	相对于工件坐标零点计算系统认为在此 高度上水平运动是安全的	0.5~500	立即生效
10033	G01 加工结束提前 输出 IO	进给结束多少长度前输出 IO,在下一段 G00 开始后关闭 IO,0 则表示功能无效,	0~1000	立即生效

		该值不影响		
<b>1.2 暂停/停止动作</b>				
10100	暂停时动作	暂停时刀具动作	“0”：保持不动； “1” 抬到安全高度 “2” 使用抬刀量	立即生效
10101	暂停时 Z 轴抬刀量	暂停时 Z 轴抬刀量	1~1000	立即生效
10102	停止时动作	停止时刀具动作	“0”：保持不动； “1” 抬到安全高度 “2” 使用抬刀量	立即生效
10103	停止时 Z 轴抬刀量	停止时 Z 轴抬刀量	1~1000	立即生效
10104	Eng 选刀加工	Eng 选刀加工	0 (否)：无效 1 (是)：有效	立即生效
<b>1.3 . 默认速度</b>				
20403	最大速度	XYZ 最大速度	XY：0~100000 Z:0~30000	立即生效
10200	轴起跳速度	加工过程中的最小进给速度	0~各轴最大速度中的最小值	立刻生效
10201	轴加速度	进给轴的加速度	0.01~100000	立即生效
10202	轴加加速度	进给轴加速度的变化率	0.01~300000	立即生效
10203	转弯加速度	进给运动发生在相邻轴上的最大加速度	1~100000	立即生效
10204	回零加速度	回零加速度	1~100000	立即生效
20401	手动低速	手动模式下的默认速度	起跳速度~手动高速	立即生效
20402	手动高速	手动模式下的高速运行时的速度	手动低速~最大速度	立即生效
10207	手动低速加速度	手动低速时的加速度	1~100000	立即生效
10208	手动高速加速度	手动高速时的加速度	1~100000	立即生效
10209	空行速度	刀具空行时的速度	轴起跳速度~各轴最大速度	立即生效
10210	空行加速度	加空行时的加速度	1~100000	立即生效

10211	空行加加速度	空行加速度的变化率	1~300000	立即生效
10213	默认进给速度	系统默认的进给速度	轴起跳速度~各轴最大速度	立即生效
10214	使用默认进给速度	加工文件中指定的速度将无效	0 (否) : 无效 1 (是) : 有效	立即生效
10215	Z 轴下刀速度	刀具沿 Z 方向向下落刀运行的速度	轴起跳速度~最大速度的第三个值	立即生效
10216	Z 轴抬刀速度	刀具沿 Z 方向向上抬刀运行的速度	轴起跳速度~Z 轴最大速度	立即生效
10217	接近速度	定位过程中刀具快接近工作时的进给速度	轴起跳速度~100000	立即生效
10218	接近距离	定位过程中机床开始减速时离目标位置的距离	0~50	立即生效
<b>1.4 . 圆弧</b>				
10300	弧 IJK 增量是否有效	弧 IJK 增量是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
10301	圆弧限速是否有效		0 (否) : 无效 1 (是) : 有效	立即生效
10302	圆弧加工最小速度	圆弧加工最小速度	0.01~100000	立即生效
10303	圆弧加工最大速度	圆弧加工最大速度	圆弧加工最小速度 ~100000	立即生效
10304	圆弧半径公差	圆弧半径公差	0~10	立即生效
10305	参考圆半径	参考圆半径	0.01~+∞	立即生效
10306	参考圆速度	参考圆速度	0.01~+∞	立即生效
<b>1.5 . 櫥柜速度</b>				
10307	轴起跳速度	加工过程中的最小进给速度	0~各轴最大速度中的最小值	立刻生效
10308	轴加速度	进给轴的加速度	0.01~100000	立即生效
10309	轴加加速度	进给轴加速度的变化率	0.01~300000	立即生效
103010	转弯加速度	进给运动发生在相邻轴上的最大加速度	1~100000	立即生效

10311	默认进给速度	系统默认的进给速度	轴起跳速度~各轴最大速度	立即生效
10312	Z轴下刀速度	刀具沿Z方向向下落刀运行的速度	轴起跳速度~最大速度的第三个值	立即生效
10313	Z轴抬刀速度	刀具沿Z方向向上抬刀运行的速度	轴起跳速度~Z轴最大速度	立即生效
10314	参考圆速度	参考圆速度	0.01~+∞	立即生效
<b>1.6 . 加工效果</b>				
10400	插补算法选择	插补算法选择	"0" : "高效率" ; "1" : 品质优先, "2" : 兼顾品质与效率	立刻生效
10401	转角容差	转角容差	0~0.1	立刻生效
10405	平滑算法选择	平滑算法选择	1 : 加权平均 0 : 移动平均	重启生效
10402	平滑时间	平滑时间	0~0.6	立刻生效
10403	短线段加速度	短线段加速度	1~1000	立刻生效
10404	短线段长度	短线段长度	0.001~100	立刻生效
<b>1.7.对刀</b>				
10500	对刀模式	对刀模式	1 : 浮动对刀 0 : 固定对刀	立刻生效
10501	对刀安全Z点机械坐标	对刀安全Z点机械坐标	+∞~-∞	立刻生效
10502	对刀速度	在对刀过程中,接近对刀仪表面时的速度	0~1000	立刻生效
10503	固定对刀仪厚度	固定对刀仪厚度	0~1000	立刻生效
10504	固定对刀快速速度	固定对刀时从最高点移动到对刀起始高度时的速度	0~z轴下刀速度	立刻生效
10505	固定对刀仪机械坐标	固定对刀仪机械坐标	-9999.~软限位最大位置	立刻生效
10506	浮动对刀块厚度	请准确测量对刀块厚度	0~500	立刻生效

### 13.11.2 轴参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>2.1 . 轴—基础</b>				
20100	方向（手动方向）	进给轴方向	（正）1~-1（反）	重启生效
20101	脉冲当量	系统每发一个脉冲对应机床所移动的实际距离	0.0001~9999	重启生效
<b>2.2.轴-限位</b>				
20302	软限位使能	软限位是否有效。即工作台行程范 检是否有效	0（否）：无效 1（是）：有效	立即生效
20303	软限位最小位置	指定机械坐标在各轴方向上的下限	-9999~软限位最大位置	立即生效
20304	软限位最大位置	指定机械坐标在各轴方向上的上限	软限位最小位置~9999	立即生效
20305	限位减速速度	设置硬限位减速值	100~+∞	立即生效
20306	软限位最小位置 2	刀库类型为固定盘刀和固定直排的 时候指定机械坐标在各轴方向上的 下限	-9999~软限位最大位置	立即生效
20307	软限位最大位置 2	刀库类型为固定盘刀和固定直排的 时候指定机械坐标在各轴方向上的 上限	软限位最小位置~9999	立即生效
<b>2.2.4 轴/旋转轴</b>				
10700	旋转轴工作台配置	0:通用 3 轴；1：3 轴转台；2：4 轴	0~2	重启生效
10701	旋转轴下标（旋转轴配 置）	0:X 为旋转轴；1：Y 为旋转轴	0~1	重启生效
10702	旋转轴的编程轴	0:A 轴；1：B 轴	0~1	重启生效
10703	旋转轴的编程单位	0:长度；1：角度	0~1	重启生效
10704	4 轴旋转台类型	0:旋转轴为 A 轴；1：旋转轴为 B 轴	0~1	立即生效
10705	4 轴圆雕非线性补偿使能	1:打开；0：关闭	0~1	立即生效
10706	旋转轴最大速度		0~9999	重启生效
10707	旋转轴加速度		0~9999	重启生效

10708	旋转轴控制半径			重启生效
10709	是否使用轴心距	是否使用轴心距 0 不使用, 1 使用	0~1	重启生效
10710	轴心距	A 轴方向上顶针的半径	1~9999	重启生效
10711	是否互换 XY 的点动按钮位置	0 不换, 1 交换	0~1	重启生效
10712	是否启动 4 轴联动轨迹画图	是否启动 4 轴联动轨迹画图	0 (否): 无效 1 (是): 有效	立即生效

### 13.11.3 主轴参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>3.1 . 主轴</b>				
10900	是否使用默认主轴速度	使用系统默认主轴转速, 加工文件中指定的进给速度将无效	0 (否): 无效 1 (是): 有效	立即生效
10901	暂停时停转	停止时主轴是否停转	0 (否): 无效 1 (是): 有效	立刻生效
10902	停止时停转	暂停时主轴是否停转	0 (否): 无效 1 (是): 有效	立即生效
10903	最大转速	置主轴的最大转速	0~100000	立即生效
10904	默认转速	设置主轴的默认转速	0~最大转速	立即生效
10905	主轴启动延时	设置主轴接收到启动命令后的延时时间	1~300	立即生效
10906	主轴停止延时	设置主轴接收到停止命令后的延时时间	1~300	立即生效
10907	加工时主轴启动模式	无特殊说明时, 启动后等待启动完成才进行下一步动作	0: 执行加工文件时启动 (默认); 1: 执行 G00 或 G01 等加工指令前启动; 2: 执行 C00 时启动但不等待完成, 执行 G01 等加工指令前等待完成; 3: 执行 M03 指令时启动	立即生效

### 13.11.4 原点参数

参数编号	参数名称	含义及作用	取值范围	生效时间
------	------	-------	------	------

4.1 . 轴-回零				
20200	回零定位方向	回机械原点的运动方向	(正) 1~-1 (反)	立即生效
20201	回零粗定位速度	在回机械原点过程中, 粗定位阶段的进给速度	-∞~最大速度	立即生效
20202	回零精定位速度	在回机械原点过程中精定位阶段的进给速度	0.1~回零粗定位速度	立即生效
20203	回零回退距离	在回机械原点精定位阶段结束后、附加的移动距离	-1000~1000	立即生效
4.1 . 轴-限位				
20300	原点限位使能	原点限位是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
20301	原点限位时允许运动方向	原点限位时各轴运动方向	(正) 1~-1 (反)	立即生效

### 13.11.5 补偿参数

5.1 . 补偿				
11000	丝杠误差补偿有效	丝杠误差补偿是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
11001	反向间隙补偿有效	反向间隙补偿是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
11002	刀具补偿有效	刀具补偿是否有效	0 (否) : 无效 1 (是) : 有效	立即生效
20105	反向间隙	当丝杠间由于长时间使用产生间隙时, 为了到达设置的加工精度而进行补偿	0~1	立即生效
20106	公共偏置		-9999~9999	立即生效

### 13.11.6 手轮参数

参数编号	参数名称	含义及作用	取值范围	生效时间
6.1 . 手轮				
11100	手轮方向	手轮摇动时运动的方向	-1 : 负向 1 : 正向	重启生效
11101	手轮加速度	值越小, 速度越平稳	0~6000	立即生效

11102	严格手轮脉冲计数	如果采用手轮严格计数,系统将会运动手轮所指定的距离;反之,机床只在手轮摇动时才运动	0(否):无效 1(是):有效	立即生效
11103	手轮倍率 X1 挡	在手轮倍率 X1 挡,每转到手轮 1 刻度移动的距离	0~1000	重启生效
11104	手轮倍率 X10 挡	在手轮倍率 X10 挡,每转到手轮 1 刻度移动的距离	0~1000	重启生效
11105	手轮倍率 X100 挡	在手轮倍率 X100 挡,每转到手轮 1 刻度移动的距离	0~1000	重启生效

### 13.11.7 刀库参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>7.1 . 刀库基础</b>				
20700	刀具容量	刀具容量 : [1,30], 轮动换刀 [1,6]	1~30	重启生效
<b>7.2 . 刀库-刀具</b>				
20701	位置	XYZA 代表刀具机械位置。B 代表如果是固定直排刀库时,设置 0:刀具处在机床前边,设置 1:刀具处在抓布后边,设置 2.刀具处在机床左边,设置 3 : 刀具处在机床右边	(-∞..~(+∞..))	立即生效
20702	偏置		(-∞..~(+∞..))	立即生效
20704	主轴转速		0~最大转速	立即生效
20706	长度		0.001~500	立即生效
20709	动吹气刀号	"0" : 不使用; "1" : 使用	0~1	立即生效
20710	刀具类型	0 : 普通刀具 1 : 锯片刀具	0~1	立即生效
<b>7.2 . 刀库-设置</b>				
11400	进出刀库速度	换刀时进出刀库速度	1~9999	立即生效
11401	换刀起点到换刀点距	换刀起点到换刀点距离	-9999~9999	立即生效



	离			
11402	抓刀接近速度	抓刀接近速度	1~9999	立即生效
11403	换刀起点 Z 坐标	换刀起点 Z 坐标	-9999~9999	立即生效
11404	是否回换刀起点	换刀完成后是否回换刀起点	-9999~9999	立即生效
<b>7.3 . 多工序换刀</b>				
10910	换刀是否平移	汽缸换刀时, 是否平移	0~1	立即生效
10911	换刀延时	汽缸换刀时, 延时时间	0~99	立即生效
10912	换刀点 Z 坐标	汽缸换刀时, Z 轴抬到该坐标, 再切换其他刀具	-999~0	立即生效
<b>7.4 . 多工序预启动</b>				
10908	使用多工序预启动功能	通过设定的行号, 达到预启动作用目前只对刀库 4 有效	0 (否) : 无效 1 (是) : 有效	立即生效
10909	预启动行号	提前多少行启动主轴 排钻	1~100	重启生效
<b>7.5 . 排钻</b>				
11405	排钻号		2~4	立即生效
11406	排钻数量		0~刀具容量	立即生效
<b>7.5 . 锯片</b>				
11407	锯片上抬的刀具号	0:不需要到固定刀刀具号; 其他值: 需要回到的固定号	0~刀具容量	立即生效
11408	上下锯片切换延时	上下锯片切换延时	0~60	立即生效
11409	锯片换刀点 Z 坐标	锯片换刀时, Z 轴抬到该坐标, 再切换其他刀具	-999~0	立即生效

### 13.11.8 SL 总线参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>8.1 . SL 总线</b>				

10800	轴控 IO 板类型	0 : 单轴脉冲板, 1 : 3 轴脉冲板 2 : 5 轴	0~2	重启生效
10801	总线伺服个数		0~12	重启生效
<b>8.2 . 附属组(X2Y2..)</b>				
10802	附属组的轴使能	0:不使用 ; 1 : 使用	(0... )~(1... )	重启生效
10803	附属组的轴同步类型	0:独立轴 ; 1 : 同步轴	(0... )~(1... )	重启生效
10804	附属组的轴输出端口	1~10 : 非总线轴输出端口 ; 11~20 : 总线轴输出端口	(0... )~(20... )	重启生效
<b>8.3 . 拓展配置/工艺</b>				
10600	使能夹具避让	使能夹具避让	0 ( 否 ) : 无效 1 ( 是 ) : 有效	重启生效
10601	设置刀库类型	0 : 不换刀 2 : 固定直排刀库 3 : 龙门架直排 4 : 轮动换刀 5 : 固定式圆盘刀库 6 : 跟随式 圆盘刀库 7 : 气换刀 +排钻 8 : 直排+锯片 9 : 多工 序+锯片	0~30	重启生效
10602	插补联动轴个数		3~5	重启生效
<b>8.3 . 轴—基础</b>				
10004	使用绝对值编码器	“0” : 不使用 ; “1” : 使用	0 ( 否 ) ~1 ( 是 )	重启生效
20103	绝对值编码器方向	-1 : 负向 ; 1 : 正向	-1 : 负向 1 : 正向	重启生效
20104	轴输出端口	1~10 : 非总线轴输出端口 ; 11~20 总线轴输出端口	0~20	重启生效

### 13.11.9 双工位参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>9.1 . 双工位</b>				
11200	双工位是否有效	“是” 表示使用双工位 , “否” 表示单工位	0 ( 否 ) ~1 ( 是 )	重启生效
<b>9.2 . 压轮</b>				

11201	压轮功能是否有效	“是”表示使用压轮功能, “否”表示不使用压轮功	0 (否) ~1 (是)	重启生效
11212	空闲状态压轮功能是否有效	“是”表示空闲状态使用压轮功能 “否”表示空闲状态不使用压轮功能	0 (否) ~1 (是)	重启生效
11203	是否使用 X 轴作为压轮方向	是否使用 X 轴作为压轮方向	0 (否) ~1 (是)	立即生效
11204	压轮间距	压轮间距	0~9999	立即生效
11205	台面间距	台面间距	0~9999	立即生效
11206	G54 行程下限	G54 行程下限	-9999~9999	立即生效
11207	G54 行程上限	G54 行程上限	-9999~9999	立即生效
11208	G55 行程下限	G55 行程下限	-9999~9999	立即生效
11209	G55 行程上限	G55 行程上限	-9999~9999	立即生效
11210	是否使用压轮高度	“是”表示使用压轮高度控制压轮升降, “否”表示不使用压轮高度控制压轮升降	0 (否) ~1 (是)	重启生效
11211	压轮高度	当系统加工时,Z 轴工件坐标低于这个高度值时,压轮打下,相反抬起来	-9999~9999	立即生效
<b>9.2 . 定位</b>				
11213	双工位定位功能	“是”表示加工过程中定位汽缸可以自动动作或者手动控制 “否”表示不使用该功能	0 (否) ~1 (是)	立即生效
11214	定位提前和延后动作距离	定位汽缸在台面起始点提前动作,或者在台面终点延后动作的距离	0~台面距离	立即生效
11215	双工位是否暂停	是”表示机器在工位之间会停等待汽缸缩回,再前进 “否”表示不使用该功能	0 (否) ~1 (是)	立即生效
11216	双工位暂停坐标	机器暂停的位置	-9999~9999	立即生效
<b>9.3 . 大小板</b>				
11217	是否使用大板模式	是否使用大板模式	0 (否) ~1 (是)	重启生效

11218	大板模式下 G54 行程 下限	大板模式下 G54 行程下限	-9999~9999	立即生效
11219	大板模式下 G54 行程 上限	大板模式下 G54 行程上限	-9999~9999	立即生效

### 13.11.10 上下料参数

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>10.1 . 上下料</b>				
11300	上料功能使能	上料功能使能	0 (否) : 无效 1 (是) : 有效	重启生效
11301	下料功能使能	下料功能使能	0 (否) : 无效 1 (是) : 有效	重启生效
11302	上料起点位置	上料起点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11303	上料终点位置	上料终点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11304	下料起点位置	下料起点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11305	下料终点位置	下料终点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11306	右推料位置	右推料位置	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11307	工位 2 下料起点	工位 2 下料起点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效
11308	工位 2 下料终点	工位 2 下料终点 Y 坐标	软限位最小位置的第二个 值~软限位最大位置的第 二个值	立即生效

11309	上料速度	上料速度	0~最大速度的第 2 个值	立即生效
11310	下料速度	下料速度	0~最大速度的第 2 个值	立即生效
11311	右推料速度	右推料速度	0~最大速度的第 2 个值	立即生效
11312	当前位置运动到上下料起点速度	当前位置运动到上下料起点速度	0~最大速度的第 2 个值	立即生效
11313	上料汽缸延迟	上料汽缸延迟	0~60	立即生效
11314	吸附延迟	吸附延迟	0~60	立即生效
11315	下料汽缸延迟	下料汽缸延迟	0~60	立即生效
11316	上下料辅助延迟 1	上下料辅助延迟 1	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11317	上下料辅助延迟 2	上下料辅助延迟 2	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11318	上下料辅助延迟 3	上下料辅助延迟 3	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11319	上下料辅助延迟 4	上下料辅助延迟 4	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11320	上下料辅助延迟 5	上下料辅助延迟 5	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11321	上下料辅助位置 1	上下料辅助位置 1	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11322	上下料辅助位置 2	上下料辅助位置 2	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11323	上下料辅助位置 3	上下料辅助位置 3	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效
11324	上下料辅助位置 4	上下料辅助位置 4	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效

			二个值	
11325	上下料辅助位置 5	上下料辅助位置 5	软限位最小位置的第二个值~软限位最大位置的第二个值	立即生效

### 13.11.11 贴标

参数编号	参数名称	含义及作用	取值范围	生效时间
<b>11.1 . 贴标</b>				
11500	侧推预留距离	侧推预留距离	默认为“0”，范围(0~9999) mm	立即生效
11501	后推预留距离	后推预留距离	默认为“0”，范围(0~9999) mm	立即生效
11502	往前推料距离	往前推料距离	默认为“0”，范围(0~9999) mm	立即生效
11503	贴标进出延时	贴标进出延时	默认为“0”，范围(0~60) S	立即生效
11504	贴标旋转气缸延时	贴标旋转气缸延时	默认为“0”，范围(0~60) S	立即生效
11505	贴标升降气缸延时	贴标升降气缸延时	默认为“0”，范围(0~60) S	立即生效
11506	真空吸附延时	真空吸附延时	默认为“0”，范围(0~60) S	立即生效
11507	打印标签延时	打印标签延时	默认为“0”，范围(0~60) S	立即生效
11508	标纸宽度	标纸宽度	默认为“60”，范围(10~300) mm	立即生效
11509	标纸高度	标纸高度	默认为“40”，范围(10~300) mm	立即生效
10017	是否使用贴标机 IO	是否使用贴标机 IO	默认为“0”，范围(使用贴标机 IO “1”，不使用此功能“0”)	重启生效

## 第十四章 辅助功能

【辅助】菜单中包含了不辅助功能相关的各项，用鼠标单击【辅助】菜单，出现下拉菜单项：



图 14-1 菜单栏上的“辅助”菜单

### 14.1 当前版本

选择【辅助】菜单中【当前版本】菜单项，将给出当前系统软件版本等相关信息



图 14-1 版本信息

### 14.2 重启控制器软件

选择【辅助】菜单中【重启控制器软件】菜单项，执行客户端重启。



图 14-5 重启控制器软件

选择【辅助】菜单中【重启控制器系统】菜单项，执行控制器重启。

### 14.3 程序负载监测

选择【辅助】菜单中【程序负载监测】菜单项，将给出当前系统通信周期相关信息

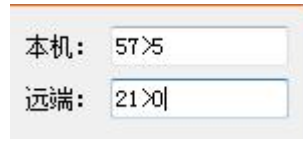


图 14-6 负载监测

### 14.4 打开/关闭软件开机自启动

选择【辅助】菜单中【打开软件开机自启动/关闭软件开机自启动】菜单项，修改开机是否自动启动 L68 软件。

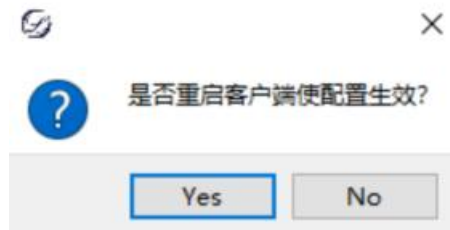


图 14-7 重启生效

### 14.5 软键盘

选择【辅助】菜单中【软键盘】菜单项，将弹出软键盘。

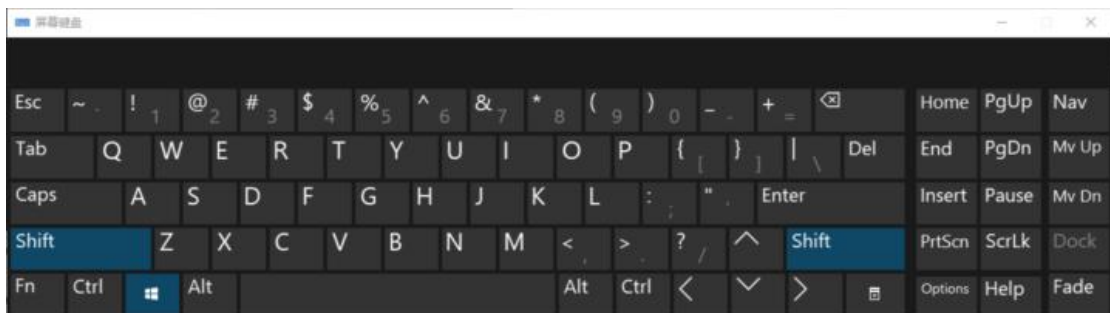


图 14-8 软键盘

### 14.6 定时暂停

选择【辅助】菜单中【定时暂停】菜单项，到设定的时间暂停程序。





图 14-9 定时暂停

### 14.7 导出安装包

选择【辅助】菜单中【导出安装包】菜单项，将安装包导出。

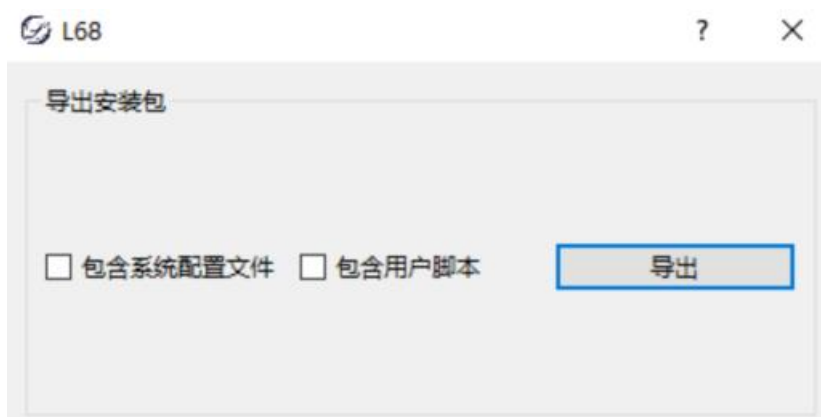


图 14-10 导出安装包

### 14.8 升级主站

选择【辅助】菜单中【升级主站】菜单项，升级主站。

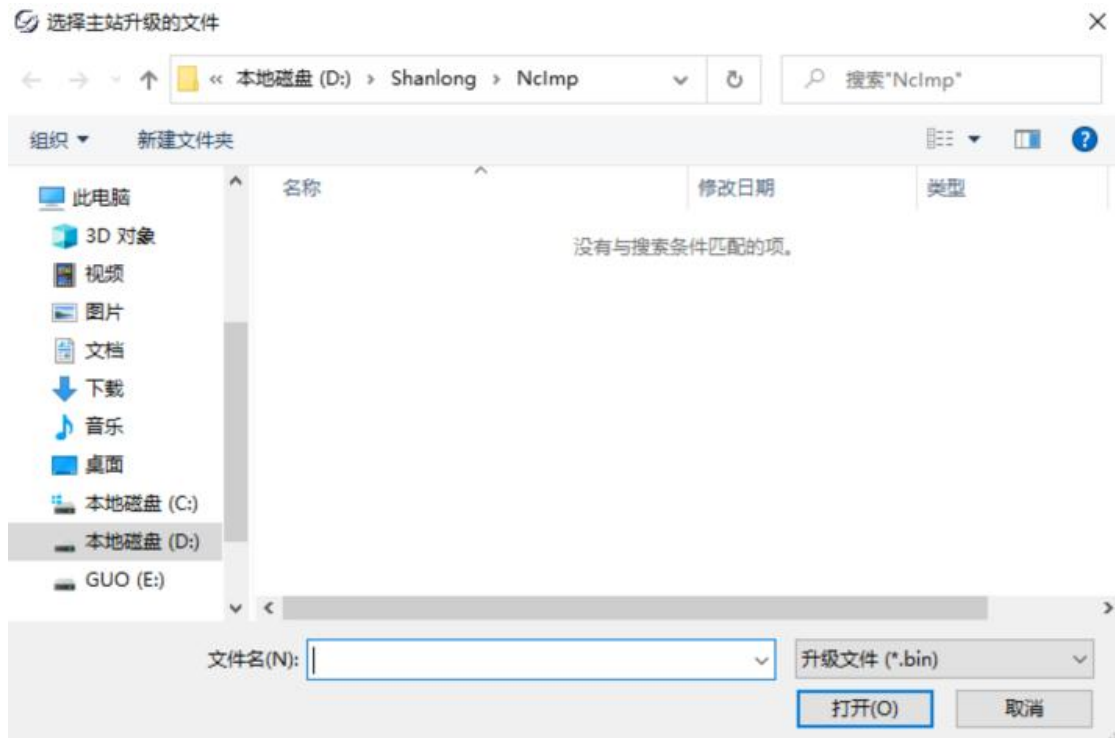


图 14-11 升级主站

## 第十五章 IO 状态

输入 IO 有 27 路，输出 IO 有 17 路，可以自由设置；主控 IO:1-8，扩展板 IO:33-48，配置了就能使用。配置 IO 端口为 '0'，则为未配置（不使用）。

IO 状态窗口，显示了输入/输出端口描述、端口号、极性，以及当前 IO 状态。IO 打开时，IO 状态中该 IO 变成红色；IO 关闭时，IO 状态中该 IO 变成绿色。用户可配置端口，IO 状态窗口查看有无输入信号或信号输出。



图 15-1 IO 状态窗口

### 15.1 输出 IO 测试

用户可以勾选界面下方【测试输出】框，对输出端口 IO 进行测试有无信号输出。选择输出端口，点击“状态”，进行测试。测试完成，需去掉勾选“测试输出”退出测试状态。

### 15.2 输入/输出 IO 配置

选中输入/输出端口的一项双击或是点击主界面下的【设置端口号】，输入端口号点击“确定”。点击“确定”，弹出对话框，选择“是”，重启系统置，即可完成端口的配置。首次修改端口号或是取反极性需要输入厂商密码。

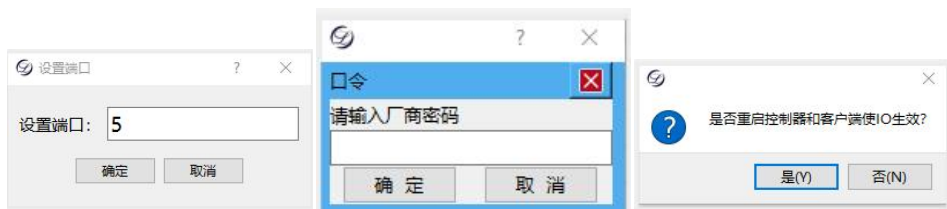


图 15-2 配置 IO

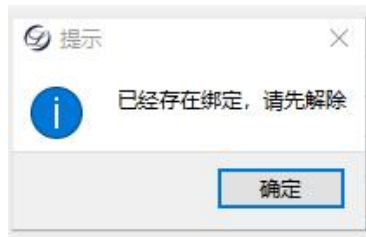
图 15-3 默认输入 I0

极性	端口	描述	极性	端口	描述
N	2	急停	N	4	停止
N	16	X 轴原点限位	N	65	X 轴伺服报警
N	13	Y 轴原点限位	N	66	Y 轴伺服报警
N	10	Z 轴原点限位	N	67	Z 轴伺服报警
N	15	X 轴正向限位	N	79	手轮输入 X 轴
N	12	Y 轴正向限位	N	80	手轮输入 Y 轴
N	9	Z 轴正向限位	N	81	手轮输入 Z 轴
N	14	X 轴负向限位	N	85	手轮输入 X1
N	11	Y 轴负向限位	N	86	手轮输入 X10
N	8	Z 轴负向限位	N	87	手轮输入 X100
N	77	主轴报警	N	71	X 轴 z 信号
N	1	对刀	N	72	Y 轴 z 信号
N	3	开始加工	N	73	Z 轴 z 信号

图 15-3 默认输出 I0

极性	端口	描述	极性	端口	描述
N	5	抱闸	N	79	主轴调速
N	4	红灯	N	6	拓展输出 2
N	3	绿灯	N	78	主轴使能_反转
N	2	冷却	N	65	X 轴伺服使能
N	1	自动加工完成	N	66	Y 轴伺服使能
N	77	主轴使能	N	67	Z 轴伺服使能
N	8	黄灯	N	71	X 轴伺服报警清除
N	7	拓展输出 1	N	72	Y 轴伺服报警清除
			N	73	Z 轴伺服报警清除

如果配置了重复的端口号，会提示如下图：



### 15.3 端口极性修改

选中输入/输出端口下的一项，点击输【取反极性】，弹出对话框，点击“确定”，弹出对话框，选择“是”，重启系统置，即可完成端口极性的修改。



图 15-3 修改端口极性

### 15.4 SL 总线配置

本系统可以适配总线通讯及脉冲通讯，本系统在出厂时默认为脉冲通讯，如要使用总线通讯：

在“参数管理”“参数设置”“SL 总线”内将：“总线伺服个数”更改为当前机床伺服数例为“3”，如果使用绝对值编码器就“使用绝对值编码器”更改为“1”否则不改动，“轴输出端口”将目前脉冲端口更改为总线端口。

当前编号	名称	值	单位	生效时间	描述
<b>SL 总线</b>					
10800	轴控IO板类型	1	-	重启控制...	取值范围: 0: 单轴脉冲板, 1: 3轴脉冲板 2: 5轴脉
10801	总线伺服个数	3	-	重启控制...	取值范围: 0~12
<b>附属组 (X2Y2...)</b>					
10802	附属组的轴使能	(0, 0, 0, 0, 0, 0)	-	重启控制...	0: 不使用; 1: 使用
10803	附属组的轴同步类型	(0, 0, 0, 0, 0, 0)	-	重启控制...	0: 独立轴; 1: 同步轴
10804	附属组的轴输出端口	(0, 0, 0, 0, 0, 0)	-	重启控制...	取值范围: 1~10: 非总线轴输出端口; 11~20: 总线
<b>拓展配置/工艺</b>					
10600	使能夹具避让	0	-	重启控制...	使能夹具避让
10601	设置刀库类型	4	-	重启控制...	设置刀库类型, 0: 不换刀 2: 固定直排刀库 3: 龙门
10602	插补联动轴个数	3	-	重启控制...	
<b>轴-基础</b>					
10004	使用绝对值编码器	1	-	重启控制...	取值范围: "0": 不使用; "1": 使用
20103	绝对值编码器方向	X:1 Y:1 Z:1	-	重启控制...	取值范围: -1: 负向; 1: 正向
20104	轴输出端口	Y:11 Y:12 Z:13	-	重启控制...	取值范围: 1~10: 非总线轴输出端口; 11~20: 总线

图 15-6 总线参数配置

参数配置完来到“I/O 状态”将输入的“伺服报警”，输入端口的“伺服使能”从脉冲伺服改成总线端口，以上更改完成重启即可

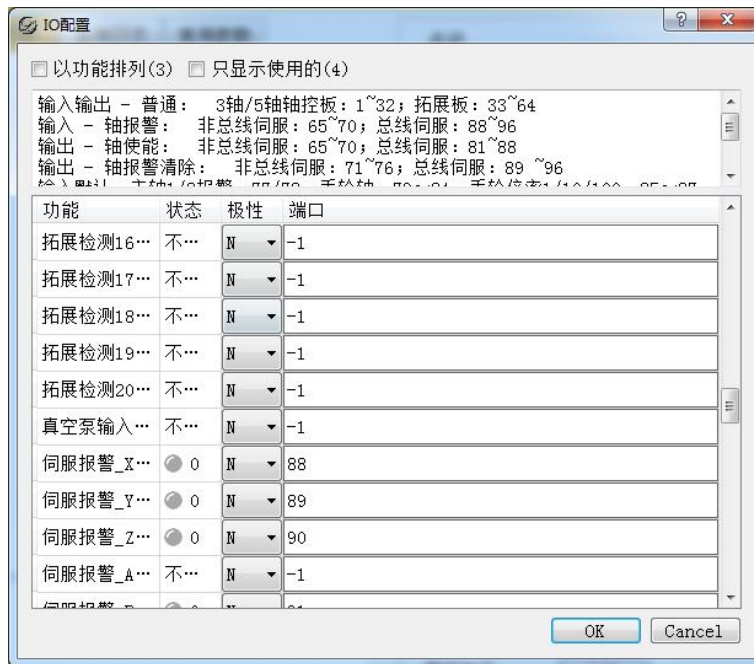


图 15-7 输入端口更改

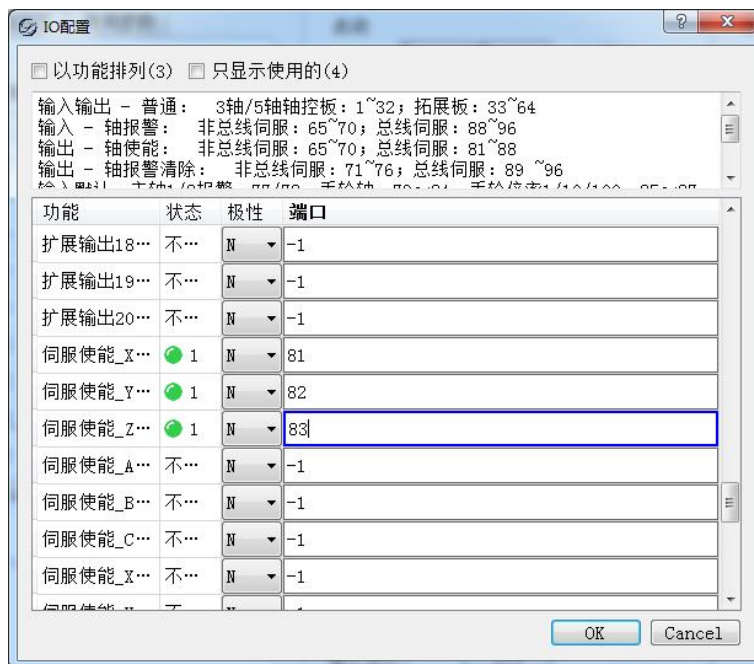


图 15-4 输出端口更改

## 更新日志

版本	更新内容	日期
V2.1.0	L68 V2 初始版本	2024.05.19

### 辅助指令 M 代码

M代码	含义	M代码	含义
M01/M84	程序暂停	M21	刀具汽缸暂停
M03/M04	主轴启动, 正转/反转	M22	主轴启动延时无需等待
M05	主轴停止	M24	真空吸附1开
M06	换刀	M25	真空吸附1关
M08	吸尘开	M30	程序结束, 真空气缸关闭
M09	吸尘关	M31/M41	扩展输出1开/关
M12	推料气缸打开	M32/M42	扩展输出2开/关
M13	推料气缸关闭	M33/M43	扩展输出3开/关
M14	关闭当前汽缸刀具	M34/M44	扩展输出4开/关
M15	左定位OUT7打开	M80	汽缸自动换刀
M16	左定位OUT7关闭	M81/M86	系统执行下料工序
M18	定位1打开	M99	循环加工
M19	定位1关闭		

---

## Contents

Chapter I Overview .....	75
1.1 Safety precautions for use .....	75
1.2 Machining process .....	75
Chapter II Installation and Safety Precautions .....	75
2.1 System configuration .....	75
2.2 Safety reminder and system installation .....	75
2.2.2 Installation environment of carving machine .....	76
2.2.3 Precautions storage and handling .....	76
2.2.4 General notes .....	76
2.2.5 Safety precautions for use .....	77
2.2.6 Prohibitions .....	78
2.2.7 Precautions for disposal .....	78
2.3 Cable requirements .....	78
2.4 Common EMC problem-solving suggestions .....	79
Chapter III System Wiring Instructions .....	81
3.1 Definition of terminal .....	81
3.2 Wiring example .....	84
Chapter IV System Operation Interface .....	85
4.1 System interface .....	85
4.2 Toolbar .....	85
4.3 Status bar .....	86
4.4 Machining path window .....	88
Chapter V Machining File Import .....	90
5.1 Import in program management .....	90
5.2 Manually write in the system .....	90
5.3 Copy and paste in bulk to a shared folder .....	90
Chapter VI Return to Mechanical Origin .....	91
Chapter VII Set the Workpiece Origin .....	93
7.1 Manual origin removal .....	93
7.2 Fixed tool setting .....	93
7.3 Floating tool setting .....	93
7.4 Storage and selection of workpiece origin .....	94



---

Chapter VIII Manual Machining .....	96
8.1 Handwheel pulse mode .....	96
8.2 Continuous inching mode .....	97
8.3 Stepping mode .....	97
Chapter IX Machining Operation .....	97
9.1 Load file .....	97
9.2 Set the workpiece origin .....	98
9.3 Automatic machining .....	98
9.3.1 Start .....	99
9.3.2 Pause .....	100
9.3.3 Stop .....	101
9.3.4 Fine tuning .....	101
9.3.5 Breakpoint continuing .....	101
9.3.6 Advanced start .....	101
9.3.7 Array machining .....	102
9.4 Handwheel guided machining .....	103
Chapter X Check Machining File .....	104
Chapter XI Milling Bottom and Frame Operations .....	106
Chapter XII Program Management .....	107
12.1 Create .....	107
12.2 Edit .....	107
12.3 Delete .....	108
12.4 Load .....	108
Chapter XIII Parameter Management .....	109
13.1 Setting parameters .....	109
13.2 Restore manufacturer parameters .....	110
13.3 Backup parameters to internal controller .....	110
13.4 Backup parameters to client path .....	110
13.5 Restore parameters from the controller .....	111
13.6 Restore parameters from client path .....	111
13.7 Controller connection .....	111
13.8 Modify the password .....	112
13.9 Modify the controller IP .....	112
13.10 Client settings .....	113

---

13.11 Parameter modification methods .....	113
13.12 Overview of user parameters .....	114
13.13 Overview of manufacturer parameters .....	128
Chapter XIV Miscellaneous Function .....	149
14.1 Current version .....	149
14.2 Restart of controller software .....	149
14.3 Program load monitoring .....	149
Chapter XV IO Status .....	150
15.1 Output IO test .....	150
15.2 Input/output IO configuration .....	150
15.3 Port polarity modification .....	152

---

## Chapter I Overview

Welcome to use our L68 control system. The Manual introduces the functions of the control system in details and is accompanied by a large number of examples and charts to illustrate. Before using the carving machine or cutting machine, please read the Operating Manual carefully to ensure correct use and prevent accidents. Please keep the Manual properly for easy reference at any time.

The system adopts an industrial control host + L68 controller mode, so a PC needs to be configured. The system is easy to operate, easy to learn and understand, easy to install, and has a small volume, which is suitable for stone carving and aluminum plate cutting.

### 1.1 Safety precautions for use

Do not use this product in the environments with strong interference and magnetic field;

Do not plug or unplug the power supply of the operation box with electricity;

Pay attention to waterproofing, dust prevention, and fire prevention;

Prevent conductive substances (such as metals, etc.) from entering the shell;

Do not disassemble this product without authorization. There are no user repairable components inside;

Plug and unplug other cables with moderate force;

If not used for a long time, please power off and store it properly;

Turn off the power during the machine repair or adjustment;

Operators and maintenance personnel must undergo training.

### 1.2 Machining process

The machining process of the system is as follows:

1. Debug before returning to zero
2. Return to mechanical origin
3. Set the work origin
4. Load the machining program
5. Select the machining mode
6. Implement machining

Refer to the following function introduction for specific operations.

## Chapter II Installation and Safety Precautions

### 2.1 System configuration

**3C-IO11A master control, industrial personal computer, power pack, display, and expansion board**

**Industrial network cable**

### 2.2 Safety reminder and system installation

Please read the Manual carefully before operating the control system.

Carefully read the Operating Manual and the User Safety Instructions. Users should take corresponding protection and safety measures before the operation. Before operating the system for the first time, operators should understand the correct usage of the corresponding functions. It is strictly prohibited to operate or change the unfamiliar system functions or parameters at will.

The safety labels and safety related contents in the Manual are very important and must be followed. Failure to follow the requirements can result in hazardous situations, even result in minor or moderate injuries, as well as equipment damage.



**Note: The controller cannot be used for heavy machinery and equipment that can easily cause personal safety accidents. 2.2.1 Precautions for system**

---

## **installation**

- 1) Wiring operations must be carried out by professional electricians.
- 2) Confirm that the power supply is disconnected before the operation.
- 3) Install it on metal or other flame retardants and keep it away from combustible materials.
- 4) Ensure the safe grounding during use.
- 5) If there is an abnormality in the external power supply, the control system may malfunction. To ensure the safe operation of the entire system, please make sure to set up a safety circuit outside the control system.
- 6) Be familiar with the contents of the Manual before installation, wiring, operation and maintenance; know relevant mechanical and electronic knowledge and all safety precautions during use.
- 7) The electrical cabinet for installing the controller should have good ventilation, oil proof, and dust proof conditions. If the electrical cabinet is sealed, it is easy to cause the temperature of the controller to be too high, which affects normal operation. Therefore, it is necessary to install a fan. The suitable temperature inside the electrical cabinet is below 40°C, and it should not be used in areas with condensation or freezing.
- 8) The installation position of the controller should try to avoid being too close to the arrangement of AC accessories such as contactors and transformers, etc., to avoid unnecessary surge interference.

### **2.2.2 Installation environment of carving machine**

- 1) Solid ground;
- 2) Avoid direct sunlight;
- 3) Leave a certain space for maintenance and repair;
- 4) Space temperature: 5-40°C;
- 5) Relative humidity: 30-95%;
- 6) Horizontal installation of equipment;
- 7) Good ventilation.

### **2.2.3 Precautions storage and handling**

**\*Note: Do not store or place it in the following environment; otherwise, it may cause fire, electric shock, or machine damage.**



- 1) Places with direct sunlight, places where ambient temperature exceeds storage temperature, places where relative humidity exceeds storage humidity, and places with large temperature differences and condensation.
- 2) Places close to corrosive and combustible gases, places with high levels of dust, salt, and metal dust, places with dripping water, oil, and drugs, and places where vibration or impact can be transmitted to the main body.
- 3) Do not hold the cable for handling; otherwise, it may cause machine damage or fault.


### **2.2.4 General notes**

- 1) Do not stack this product too much together; otherwise, it may cause damage or fault during use.
- 2) This product is a general industrial product and is not intended to harm any life or health.
- 3) If applied to devices that may cause major accidents or damage due to the fault of this product, please install the safety devices.
- 4) If applied in environments with high concentrations of sulfur or sulfurized gases, please note that the chip resistor may be disconnected or have poor contact due to sulfurization reactions.

- 5) If the input voltage exceeds the rated range of the power supply of this product, smoke and fire may occur due to damage to internal components. Please pay close attention to the input voltage.
- 6) Please note that this product cannot guarantee use beyond the product specification range.
- 7) If there is any inconsistency or incompleteness with the system function in the Manual, the system software function shall prevail.
- 8) The control system functions are subject to change or improvement (upgrade) without prior notice. In case of any other needs, please contact our Company.

### 2.2.5 Safety precautions for use

 <p>Note</p>	<p>Do not use this product in the environments with strong interference and magnetic field.</p> <p>Pay attention to waterproofing, dust prevention, and fire prevention. Plug and unplug USB disk and other cables with moderate force. If not used for a long time, please power off and store it properly. Prevent conductive substances (such as metals, etc.) from entering the shell.</p> <p>Encoders must use shielded wires, and the shielding layer must ensure reliable grounding of a single end!</p> <p>Do not install transformers or other devices that generate electromagnetic waves or interference around the servo drive; otherwise, it may cause the fault of servo drive. If such devices need to be installed, a shielding plate should be installed between the devices and the servo drive.</p> <p>Please protect the branches, short circuits, and loops in accordance with local standards. If the protection measures for branches, short circuits, and loops are not appropriate, it may cause damage to the servo driver.</p> <p>Do not share the grounding wire with welding machines or power machines that require high current; otherwise, it may cause poor operation of the servo driver or machine.</p> <p>Do not connect or operate the machine if there is obvious damage or missing components. Wiring, inspection, etc. should be carried out by professional personnel.</p> <p>Do not disassemble this product without authorization. There are no user repairable components inside;</p> <p>The rotating motor will feed electrical energy to the servo driver, which will cause the servo driver to become energized even when the motor stops and the power is cut off. Before maintaining the servo drive, please ensure that the motor servo drive is safely disconnected.</p>
 <p>Danger</p>	<p>Do not perform wiring operations while the power is on; otherwise, there will be a risk of electric shock. Please cut off the power of all the equipment before inspection. Even if the power supply is cut off, there is residual voltage in the capacitor. After cutting off the power, please wait at least 10 min.</p> <p>The carving knife is very sharp and should not be touched by hand during operation to prevent injury. Do not contact it with handkerchiefs or scarves to prevent injury or damage to the equipment caused by entanglement;</p>

	<b>After power on</b>
	<p>Do not open the control box cover after power on; otherwise, there will be a risk of electric shock!</p> <p>Do not plug or unplug the cable of the operation box with electricity.</p> <p>Do not remove the cover of the servo drive or touch the printed circuit board while it is powered on; otherwise, there will be a risk of electric shock.</p>
	<b>During operation</b>
	<p>Non professionals cannot detect signals during operation; otherwise, it may cause personal injury or equipment damage!</p> <p>Do not touch the cooling fan and discharge resistor to test the temperature; otherwise, it may cause burns!</p>
	<b>During maintenance</b>
	<p>Operators and maintenance personnel must undergo training</p> <p>Turn off the power during the machine repair or adjustment.</p> <p>Personnel without professional training cannot repair or maintain the servo drives; otherwise, it may cause personal injury or equipment damage!</p> <p>Do not repair or maintain the equipment with electricity; otherwise, there will be a risk of electric shock!</p> <p>All pluggable plug-ins must be plugged and unplugged under the power outage!</p> <p>Parameters shall be set and checked after replacing the servo driver.</p> <p>Do not power on or run damaged machines; otherwise, it may further damage the machine.</p>
	<p>Some systems may experience mechanical self action when powered on, please be careful; otherwise; it may cause death or serious injury.</p> <p>Non electrical construction professionals cannot install, maintain, check, or replace components; otherwise, there will be a risk of electric shock.</p> <p>Do not disassemble this product without authorization. There are no user repairable components inside.</p>
 Warning	

**\*Note: Improper handling may cause hazards, including personal injury or equipment accidents.**

### 2.2.6 Prohibitions

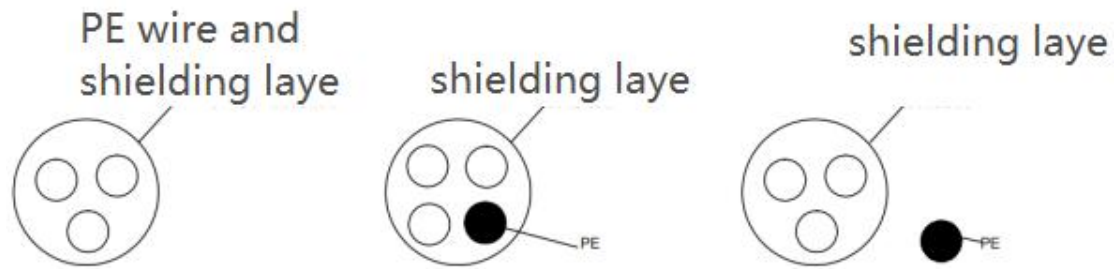
Do not disassemble or repair this product except for our staff.

### 2.2.7 Precautions for disposal

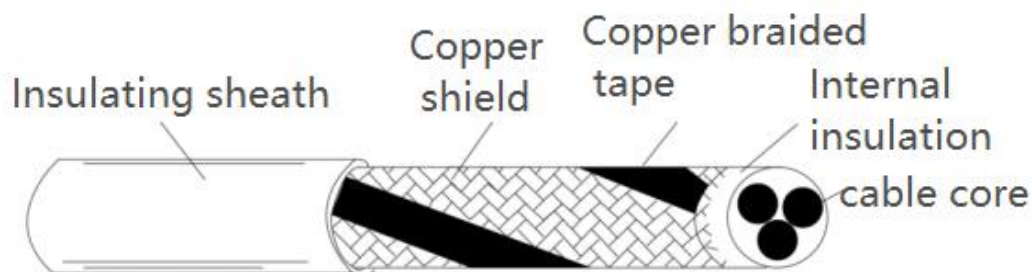
When the product needs to be disposed of as waste after normal use, please comply with the legal regulations of relevant departments regarding the recycling and reuse of electronic information products.

### 2.3 Cable requirements

In order to meet the requirements of EMC, encoder cables must use shielded cables with shielding layers. It is recommended to use shielded cables with shielding layers for power lines. The shielded cables include three phase-conductor shielded cables and four phase-conductor shielded cables, one of which is a PE line, as shown in the following figure:



In order to effectively suppress the emission and conduction of radio frequency interference, the shielding layer of the shielded wires is composed of coaxial copper woven belt. In order to increase shielding effectiveness and conductivity, the weaving density of the shielding layer should be greater than 90%. As shown in the figure below:



Installation precautions:

- (1) It is recommended to use shielded balanced cables for all shielded cables, and to use four-core cables for input cables;
- (2) The motor cable and its PE shielded wire (twisted shield) should be as short as possible to reduce electromagnetic radiation, stray current and capacitive current outside the cables;
- (3) It is recommended to use shielded cables for all control cables;
- (4) It is recommended to use shielded cables or steel pipe shielded power lines for the output power lines of the driver, and to use twisted pair shielded control wires for the leads of the affected equipment, and the shielding layer should be reliably grounded.

#### 2.4 Common EMC problem-solving suggestions

The driver belongs to equipment with strong interference, which may still cause interference due to the wiring, grounding, etc. during use. When there is mutual interference with other devices, the following methods can also be used for rectification.

Interference type	
Earth leakage circuit breaker switch tripped	Reduce carrier frequency; Reduce the length of the drive line; Add the magnetic ring on the input drive line (without PE wire); Once tripped immediately upon power on, disconnect the large ground capacitance at the input end; (Disconnect the grounding terminal of the external or internal filter, and the grounding terminal of the input port to the ground Y capacitor) Once tripped during operation or enabling, take leakage

	current suppression measures at the input end (leakage current filter, safety capacitor + magnetic ring, magnetic ring)
Communication interference	<p>Connect the motor shell to the PE end of the driver;  Connect the PE end of the driver to the power grid PE;  Add a magnetic ring on the input power line;  Add build-out resistors to the communication line source and load ends;  Add a common communication grounding wire to the differential line of the external communication line;  Shielded wire is used for communication lines, and the shielding layer is connected to the common communication grounding wire;  The communication wiring needs to be twisted pair;</p>

Table 2-1 Common EMC Interference Problems and Solutions



## Chapter III System Wiring Instructions

Figure 3-1 Bus Wiring Diagram

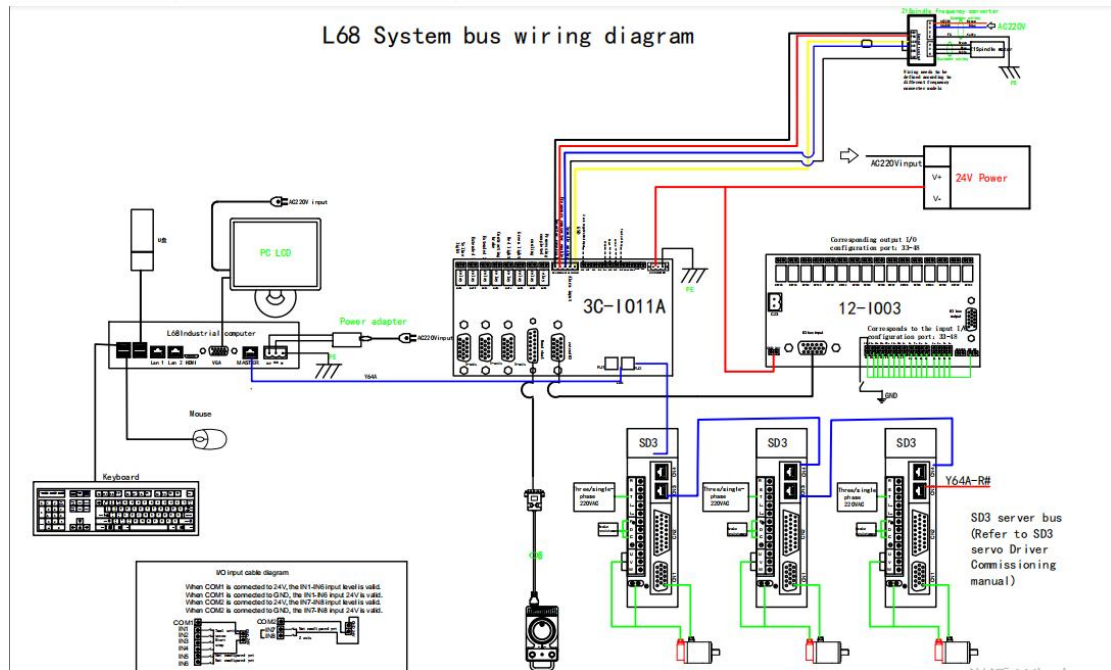
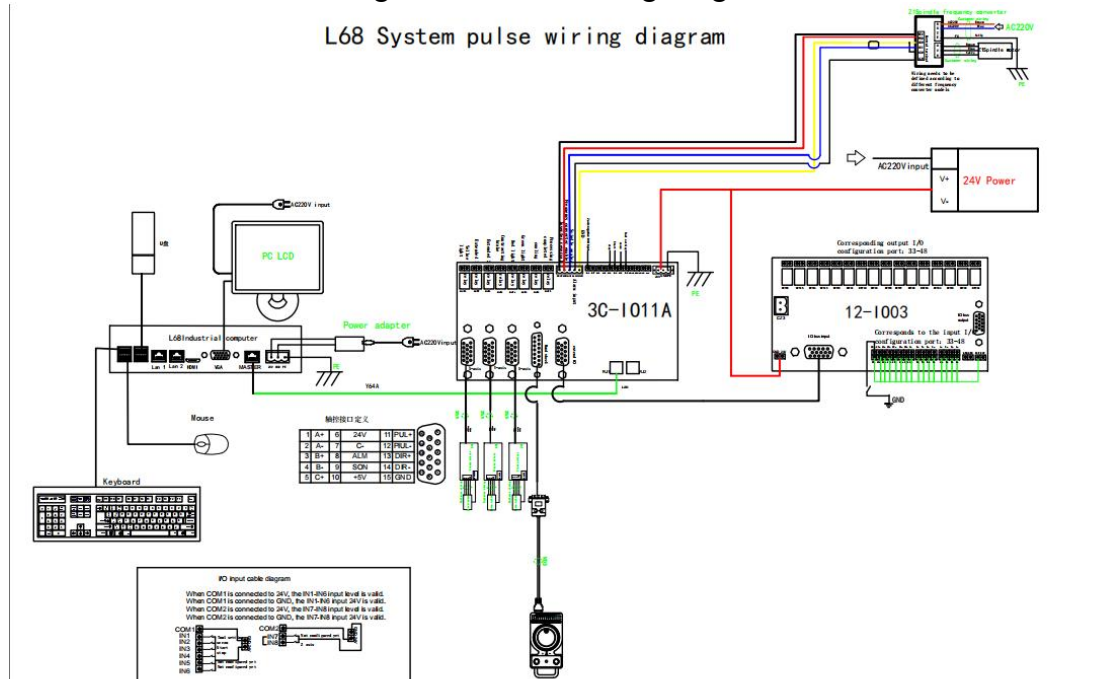


Figure 3-2 Pulse Wiring Diagram



### 3.1 Definition of terminal

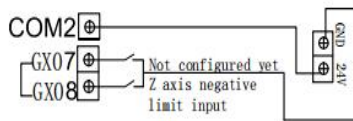
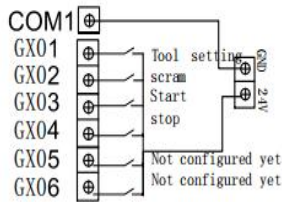
The master control input IO 1-8, output IO 1-8, and expansion board input and output each have 16 channels, and the corresponding input IO 33-48 and output IO 33-48 can freely set port functions, which can be used once configured.

Note: There are two ways to connect input IO, as shown in the figure

Figure 3-3 Input IO Wiring Diagram

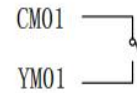
### I/O input cable diagram

When COM1 is connected to 24V, the IN1-IN6 input level is valid.  
 When COM1 is connected to GND, the IN1-IN6 input 24V is valid.  
 When COM2 is connected to 24V, the IN7-IN8 input level is valid.  
 When COM2 is connected to GND, the IN7-IN8 input 24V is valid.



### I/O Output wiring diagram

CM01 and YM01 are a set of switch signals, with on output signal in the normally open state and output in the normally closed state. CM01-CM08 the principle is the same.



Classification	Port	Definitions	Description
Power input	24V	24V power input end	DC 24V input, supplying power for system operation.
	GND	24V GND	
	PE	Grounding wire end	
IO input	24V	24V power input end	24V power input, supplying power to the optoelectronic switch.
	GND	Common terminal	Power ground and switch common terminal.
	COM1	Common terminal	See Figure 3-3
	COM2	Common terminal	See Figure 3-3
	GX01	Tool setting	Switch input, able to be connected to normally open or normally closed.
	GX02	Emergency stop	Switch input, able to be connected to normally open or normally closed.
	GX03	Start machining	Switch input, able to be connected to normally open or normally closed.
	GX04	Pause machining	Switch input, able to be connected to normally open or normally closed.
	GX05	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX06	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX07	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX08	Z-axis negative limit	Switch input, able to be connected to normally open or normally closed.
Axis control	X axis	X-axis control interface	Connect to external drives, see Figure 3-4 for details
	Y axis	Y-axis control interface	Connect to external drives, see Figure 3-4 for details
	Z axis	Z-axis control interface	Connect to external drives, see Figure 3-4 for details
Handwheel input	Handwheel	Electronic handwheel interface	Handwheel pin, see Figure 3-5
IO output	OUT1	Automatic machining completed	Transistor output
	OUT2	Cooling	Transistor output

	OUT3	Green light	Transistor output
	OUT4	Red light	Transistor output
	OUT5	Band-type brake	Transistor output
	OUT6	Expansion output 2	Transistor output
	OUT7	Expansion output 1	Transistor output
	OUT8	Yellow light	Transistor output
Spindle	GND	Common terminal	Power ground and switch common terminal.
	FWD	Spindle enable output end	Spindle on, usually connected to the positive rotation signal of the frequency converter.
	REV	Spindle enable reverse	Spindle on, usually connected to the negative rotation signal of the frequency converter
	ALM	Spindle alarm	Spindle alarm
	AVI	Spindle speed control	Spindle speed control output
Internet access	RJ1	Internet access	SLBUS protocol interface
	RJ2	Internet access	SLBUS protocol interface

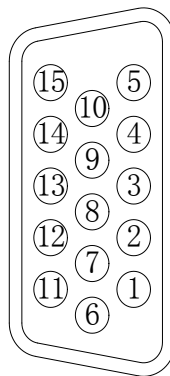


Figure 3-4 System Axis Control Interface

Port (IN)	Definitions	Description	Port (IN)	Definitions	Description
01	A+	A-phase feedback +	09	SON	Servo ON
02	A-	A-phase feedback -	10	ALM_RST	Alarm cleared
03	B+	B-phase feedback +	11	PUL+	Pulse output +
04	B-	B-phase feedback -	12	PUL-	Pulse output -
05	C+	C-phase feedback +	13	DIR+	Direction output +
06	24V	24V output	14	DIR-	Direction output -
07	C-	C-phase feedback -	15	GND	Common terminal
08	ALM	Alarm input			

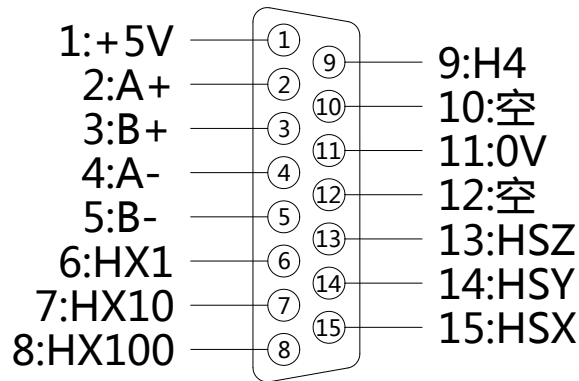


Figure 3-5 Handwheel Interface Definition

Port (IN)	Definitions	Description	Port (IN)	Definitions	Description
1	+5V	Supply power for the handwheel	9	H4	Select 4 axes
2	A+	Encoder A signal	10	Null	Null
3	B+	Encoder B signal	11	0V	Digital ground
4	A-	Encoder A signal ground	12	Null	Null
5	B-	Encoder B signal ground	13	HSZ	Select Z axis
6	HX1	Select X1 magnification	14	HSY	Select Y axis
7	HX10	Select X10 magnification	15	HSX	Select X axis
8	HX100	Select X100 magnification			

Table 3-1 System Terminal Definition Description

### 3.2 Wiring example

L68 system adopts DC power supply, with a rated power of 24V/2.2A, as shown in Figure 3-2. Please provide a switch power supply with sufficient power according to the actual use of external relays, solenoid valves, and other external accessories. It is recommended to use a waterproof power supply.

## Chapter IV System Operation Interface

### 4.1 System interface

The system interface consists of a title bar, menu bar, status bar, toolbar, machining path window, and function window. As shown in Figure 4-1 below:

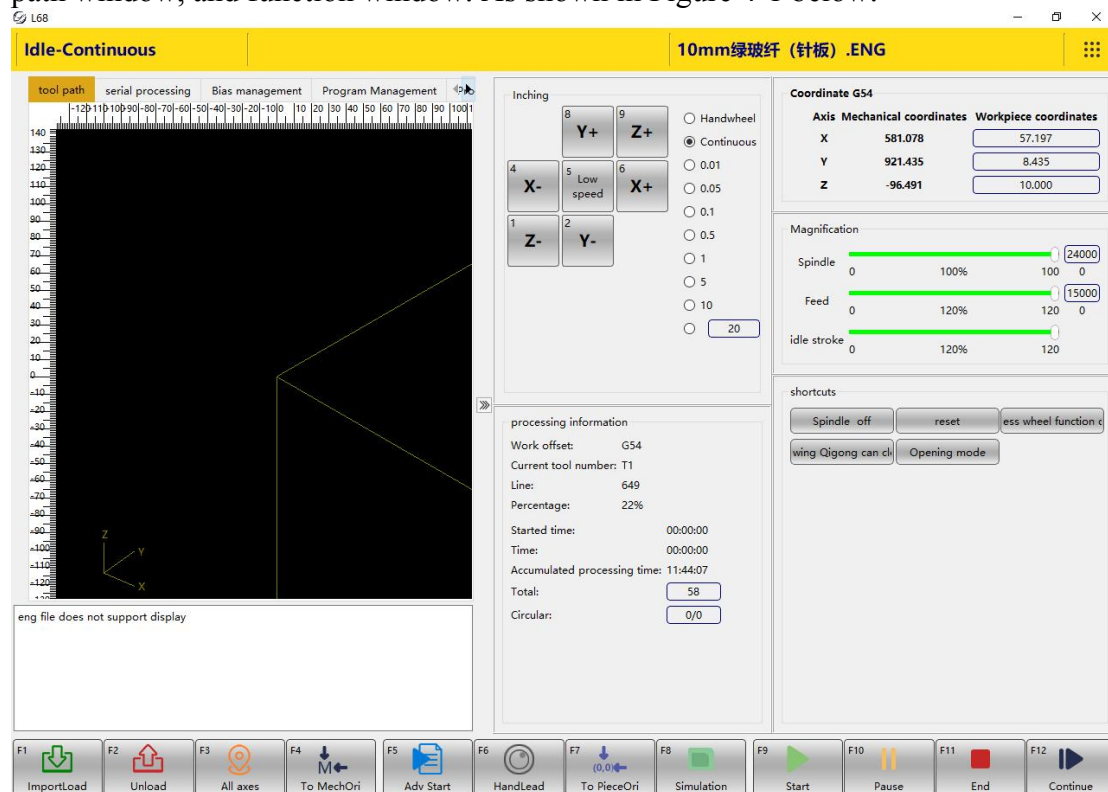


Figure 4-1 System Interface

**Menu bar:** It contains multiple drop-down submenus, representing four main operations: "Automatic", "Manual", "Parameter Management", and "Auxiliary". Users can select the corresponding menu item to perform an action or implement a function through the [Menu Bar].

**Status bar:** It displays the running status, alarm or prompt messages, loaded file names, or process names.

**Monitoring bar:** It includes four monitoring windows: "Coordinates", "Machining", "Feed", and "Spindle", mainly used to display some status information during the machining. By clicking the display button, the corresponding coordinates or speed parameters can be changed.

**Toolbar:** The shortcut button performs the corresponding operation through the button.

**Machining path window:** It displays 3D path images for simulation or machining, which can be zoomed in, out, moved, and viewed for machining details.

**Multi-function window:** It includes: "Automatic", "Manual", "Tool Setting", "Program Management", "Program Editing", "IO Status", "System Log", "Pulse Feedback", "Test Data", and each window represents a classification function.

### 4.2 Toolbar

The toolbar is located below the menu bar, consisting of some operation buttons that correspond to certain menu commands or option functions. Users can directly click these buttons through mouse to achieve the corresponding specified functions.

The information prompt box is on the right side of the toolbar, which displays alarm or prompt information, facilitating human-computer interaction.



Figure 4-2 Toolbar

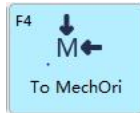
Toolbar button function:



: Import files and load them onto the system



: Unload the loaded files



: Return to the workpiece origin



: Set the current point (all axes) as the workpiece origin



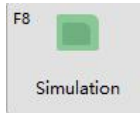
: Return to mechanical origin



: Advanced start, select line number for machining



: Handwheel guided



: Simulation



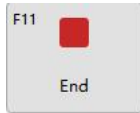
: Start



: Pause



: Breakpoint continuing



: Stop

### 4.3 Status bar

Status bar: It includes four information display windows, as shown in the following figure:

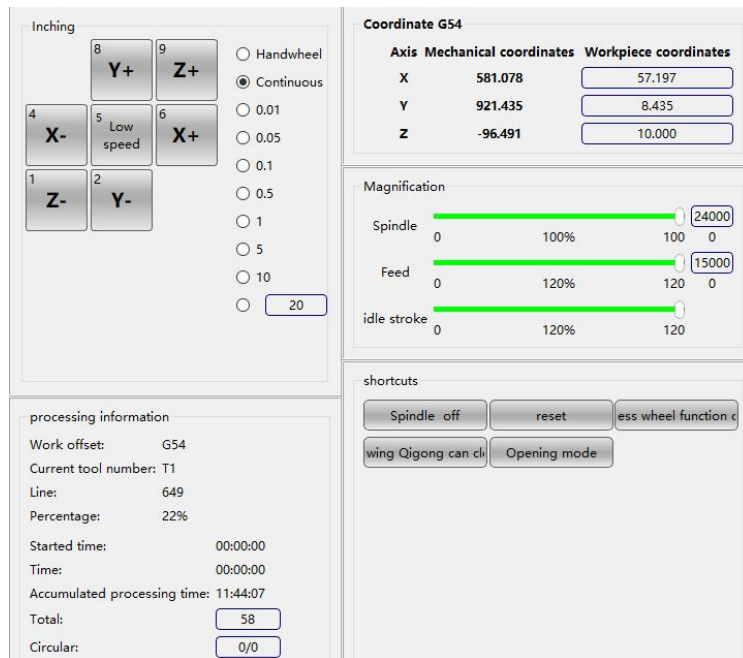


Figure 4-3 Status Bar

"Coordinates" are coordinate information

It displays the current mechanical coordinates and workpiece coordinates. To set the current point position as the workpiece origin at any time, simply click on the workpiece coordinate with the mouse to pop up a dialog box, and then click "OK" to set the corresponding axis coordinate position as the workpiece origin.

"Feed" is the feed rate area

Users can set the feed rate, adjust the feed rate, display the rate, and the actual value of the feed rate. In addition, it also displays manual high and low speeds, and G00 fixed speeds.

During the automatic machining, users can adjust the machining speed by pulling the slider with the mouse to adjust the feed rate. Alternatively, users can also select to adjust the feed rate through the submenu of [Feed Rate].

Users can adjust the current motion speed multiplier within the range of 0-120% by pulling the slider with the mouse. The feed rate is displayed as a percentage, and the actual maximum speed = set speed value \* speed multiplier.

When the system is idle, click on the speed box of the set value to pop up the "Modify Speed" dialog box. Enter the value and click "OK" to complete the modification of the manual high speed and manual low speed values. The setting of the machining speed cannot exceed the maximum single axis speed set in the parameters; otherwise, the system will report an error.

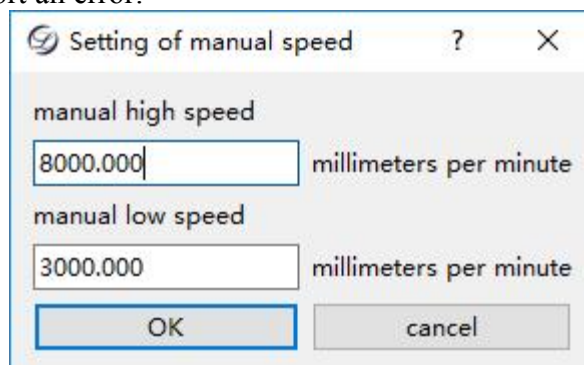


Figure 4-4 Adjust the Feed Rate

"Spindle" is the spindle speed area

Users can set the spindle speed, adjust the spindle rate, display the rate, and the actual value of the spindle speed. It can also start/stop the rotation of the spindle. Users can adjust the spindle speed by pulling the slider with the mouse and modify the spindle speed parameters.

When the system is idle, click on the speed box of the set value to pop up the "Modify Speed" dialog box. Enter the value and click "OK" to complete the modification of the machining speed. The setting of the machining speed cannot exceed the maximum spindle speed set in the parameters; otherwise, the system will report an error.

"Machining" is the machining information area

It displays the current workpiece coordinate system, current line number, start machining time, machined time, completion percentage, and the number of the currently used tool, T1, T2... respectively.

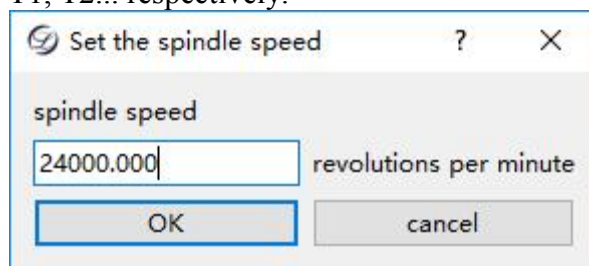


Figure 4-5 Adjust the Spindle Speed

#### 4.4 Machining path window

When the machine tool executes machining programs or simulations, the machining path window can track the tool machining path in real-time. Passed The 3D real-time display function for tracking machining path allows users to visually detect the path of the tool to confirm correct machining.

In 3D tracking mode, click on the left view function icon of the machining path window to switch between different views, including front view, bottom view, top view, back view, left view, right view, southwest isometric view, southeast isometric view, northeast isometric view, and northwest isometric view, making it convenient for users to view graphics from different angles with appropriate zoom scales. As shown in Figure 4-6.

Users can zoom in/out through the mouse wheel, hold down the left mouse button, and drag the currently displayed machining path. When performing a second or manual machining, if users need to clear the previous machining path, users can right-click and click [Clear] to avoid confusion.

In addition to the machining path window, there are also windows for program management, program editing, IO status, etc. Users can switch through the buttons "Program Management", "Program Editing", "System Log", "Pulse Feedback", and "Test Data" on the right side of the screen.



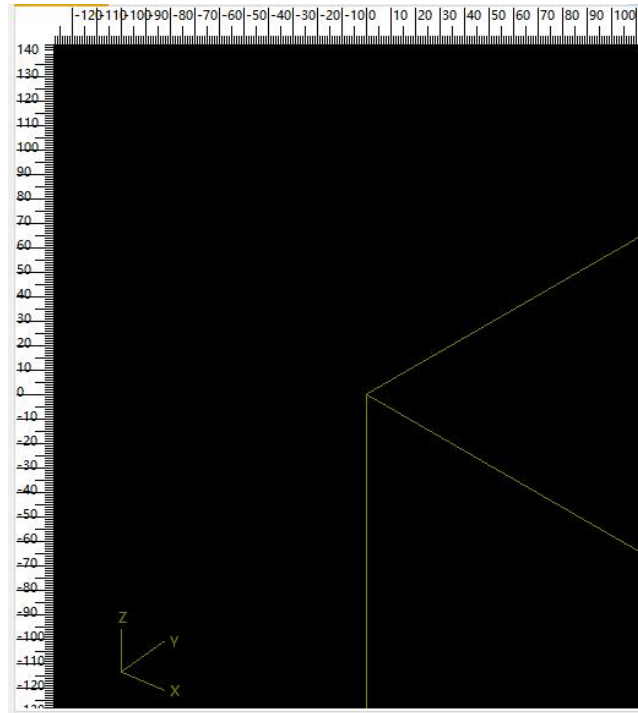


Figure 4-6 Machining Path Window


---

## Chapter V Machining File Import

There are three methods for inputting machining files: 1. Import and load, 2. Manually write in the system, and 3. Copy and paste to a shared folder. The first method is generally used for importing commonly used machining files into the system; the second method is suitable for relatively simple machining files; the third method is suitable for batch importing into the system.

### 5.1 Import in program management

Click the [Program Management] interface, then click the "Import and Load" function

key at the lower part of the program management interface or the "  " at the lower part of the main interface to enter the file selection window, select the machining file to import and click "OK". In the program management interface, users can see that the machining file has been imported and loaded into the system.

### 5.2 Manually write in the system

Click on the [Program Management] interface to create a new nc file, then select New nc File, and click the "Edit" button at the bottom of the window to edit the file. After the user completes editing, right-click and select "Save".



Figure 5-2 Edit Machining Files

### 5.3 Copy and paste in bulk to a shared folder

Open the shared folder, select to copy the files that need to be imported into the system, paste them into the path of the shared folder "processfiles" folder, and then click "Refresh" in "Program Management" to see the copied and pasted machining files.

---

## Chapter VI Return to Mechanical Origin

### 6.1 Debug before returning to zero

Before returning to the mechanical origin, it is necessary to carry out debugging before returning to zero, which mainly includes adjusting pulse equivalent, checking the direction of the machine tool spindle, and setting the table travel.

#### 6.1.1 Adjust the pulse equivalent

Impulse equivalent: The smaller the pulse equivalent value is, the higher the machining accuracy of the machine tool and the surface quality of the workpiece will be; The higher the value is, the higher the machine tool feed rate will be.

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the manufacturer parameter to set the pulse equivalent for the feed shaft parameter.

#### 6.1.2 Checking the direction of the machine tool spindle

Set the pulse equivalent to ensure that the position of the spindle movement is consistent with the actual situation.

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the operating parameters of the user parameters to view the spindle direction setting value. Then click on the [Manual] window, select continuous or step length mode, move the spindle, and check whether the actual direction of spindle movement is consistent with the parameter settings. If the direction is consistent, it indicates that the direction of the machine tool spindle is set correctly; if the direction is inconsistent, it is necessary to set the spindle direction parameter value to be opposite to the original direction.

#### 6.1.3 Set the table travel

Set the table travel according to the actual size of the machine tool

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the feed shaft parameters of the manufacturer parameters to set the lower and upper limits of the table travel based on the actual size of the machine tool.

### 6.2 Return to mechanical origin

The mechanical origin is a fixed position of the machine tool, which is jointly determined by the mechanical switch and electrical system, and is the zero point of the mechanical coordinate system. To execute the "Return to Mechanical Origin" function, the machine tool itself needs to be equipped with an origin switch. If the machine tool does not have relevant hardware support, this function needs to be disabled. Refer to the "Origin Parameter" setting in Chapter XI "Parameter Management" for details. Since the mechanical origin is the benchmark of the whole machine tool, the important role of this function is to correct the current point coordinates. To prevent power outage or incorrect current position, please perform the operation of returning to mechanical origin after program startup or emergency stop.

After the system is powered on and started, a "Return to Mechanical Origin" dialog box will automatically pop up. After clicking the button, the corresponding axis will automatically return to the mechanical origin and correct the system coordinates. Before returning the X-axis or Y-axis to the origin, please return the Z-axis to the mechanical origin first.

After selecting the menu item "Return to Mechanical Origin" in the "Manual" menu, the system will pop up the "Return to Origin" dialog box, as shown in the following figure:

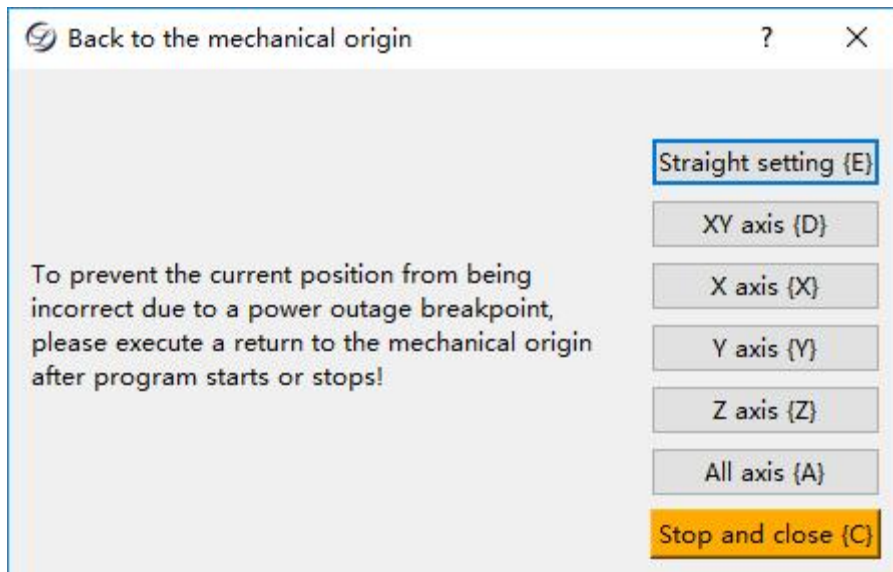


Figure 6-1 Mechanical Origin Function Screen

Single axis return to mechanical origin includes the following methods:

**Direct setting:** If it is determined that the current position is not consistent with the mechanical coordinates, and the machine tool has not been turned off and has not performed an emergency stop, direct setting can be used.

**X-axis return to mechanical origin:** Select this command to return the X-axis to the mechanical origin.

**Y-axis return to mechanical origin:** Select this command to return the Y-axis to the mechanical origin.

**Z-axis return to mechanical origin:** Select this command to return the Z-axis to the mechanical origin.

**All axes return to mechanical origin:** Select this command to return all axes to the mechanical origin.

**Note:**

If the return to the mechanical origin is not performed, please raise the Z-axis as high as possible during manual operation to ensure that the tool head will not collide with the workpiece to be machined.

When exiting, the system will automatically save the current coordinate information. If there is a sudden power outage during the automatic machining, the system will automatically save the relevant information before the power outage to the breakpoint protection file (that is, save the breakpoint information, file name, etc. to the system memory during the power outage. The same machining file only corresponds to one breakpoint protection file). After power restoration, the system will pop up a prompt box to prompt the user that the last machining file suffered from power outage. Users need to manually perform the operation of returning to the mechanical origin first, and then continue machining the file with the power outage, or choose a new machining file:

1. To continue machining the file with power the outage, users can click the "Breakpoint Continuing" button on the toolbar at the top left of the main interface, and the machine tool will return to the position before power outage. After clicking "Start", the machine tool will continue to seamlessly machine the file that was not finished before power outage from the breakpoint.
2. To select a new file for machining, after the machining is completed, users can still continue machining the file with the power outage, and the machine tool will continue seamlessly machining from the breakpoint of the corresponding file.

---

## Chapter VII Set the Workpiece Origin

Before machining the file, users need to manually adjust the position of the tool and workpiece to start machining from the predetermined position of the workpiece.

There are two ways to set the workpiece origin of the Z-axis: 1. Manual setting, 2. Fixed tool setting

### 7.1 Manual origin removal

X-axis and Y-axis workpiece origin setting: Manually move the X-axis and Y-axis to the predetermined machining position, and click on the workpiece coordinate to pop up a dialog box prompt. Based on the prompt, confirm to reset the X-axis and Y-axis coordinate values of the current position to zero (click on the X-axis and Y-axis coordinate columns in the coordinate window). As shown in the figure below:

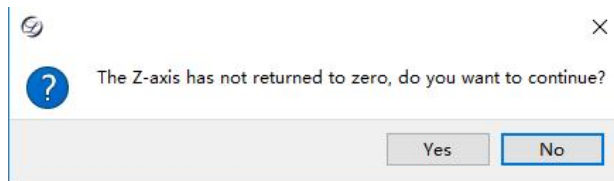


Figure 7-1 Set the Workpiece Origin

### 7.2 Fixed tool setting

Firstly, it is necessary to set the mechanical coordinates of the tool setting gauge in parameter management.

After performing a fixed tool setting, the system will automatically move to the corresponding mechanical coordinates of X-axis and Y-axis, and then start the Z-axis tool setting.

Fixed tool setting can be divided into the first tool setting and the tool setting after tool changing. Users should pay attention when using it. The first tool setting is the tool setting before machining, through which the workpiece coordinates when the tool nose contacts the fixed feeler block can be determined. Through the tool setting after tool changing, the Z-axis workpiece coordinate can be restored to the value set in the first tool setting action when the tool nose contacts the fixed feeler block. During the fixed tool setting, in order to protect the tool, a speed segmentation method is adopted, which is divided into two types: fast tool setting speed and tool setting speed. The speed can be set through parameters. When the tool nose is about to approach the tool setting gauge, a fast tool setting speed can be used. After the tool nose does not come into contact with the tool setting gauge, a normal tool setting speed can be used.

### 7.3 Floating tool setting

The tool setting mode can be switched through parameter modification. The system defaults to fixed tool setting mode. If the fixed tool setting is "No" and floating fixed tool setting is "Yes", it will be switched to the floating tool setting mode.

Floating tool setting allows users to easily determine the surface height of the workpiece and set the workpiece origin of the Z-axis. Similar to manual tool setting, since the workpiece origin of the Z-axis is usually at the center of the rotation axis, it is necessary to set the distance between the position of the feeler block and the center of the rotation axis to a common offset (alternatively, add the distance between the feeler block and the center of the rotation axis to the thickness of the feeler block). The specific operation is as follows: Place the feeler block on the workpiece surface, and manually move the tool nose above the workpiece origin. After clicking the "Measure Workpiece Surface" button, the machine tool will perform the tool setting

action. After the tool nose touches the feeler block, it will automatically lift up 10mm, and add the thickness of the feeler block to determine the Z-axis positioning.

**Note:**

(1) Before tool setting, users must ensure that the tool nose is located above the feeler block, that is, after lowering the tool, the tool nose can touch the feeler block; otherwise, it will continue to lower the tool, causing damage to the tool head and workpiece.

(2) The thickness of the feeler block can be set in the manufacturer parameters, and the workpiece origin coordinate in the Z-axis direction after tool setting will automatically compensate for it.

(3) Due to the existence of a certain gap between the workpiece surface and the feeler block, after the tool setting is completed, the gap can be compensated in the workpiece offset settings. The compensation difference depends on the size of the gap. If the workpiece surface is rough, the compensation difference can be larger. If the workpiece surface is smooth and the gap with the feeler block is smaller, the compensation difference can be set smaller. Generally, 0.1mm can be used.

(4) The tool setting speed can be determined in the parameter settings. The range of tool setting speed is 60-1000mm/min. If the tool setting speed exceeds the maximum value set by the parameter, it will wear the tool head or damage the feeler block.

**Note:**

**7.4 Storage and selection of workpiece origin**

Select the menu items [Save Workpiece Origin]/[Read Workpiece Origin] in the [Manual] menu to save the current workpiece origin. Click [Read Workpiece Origin] to set the saved workpiece origin as the current workpiece origin. Up to 10 workpiece origins can be saved.

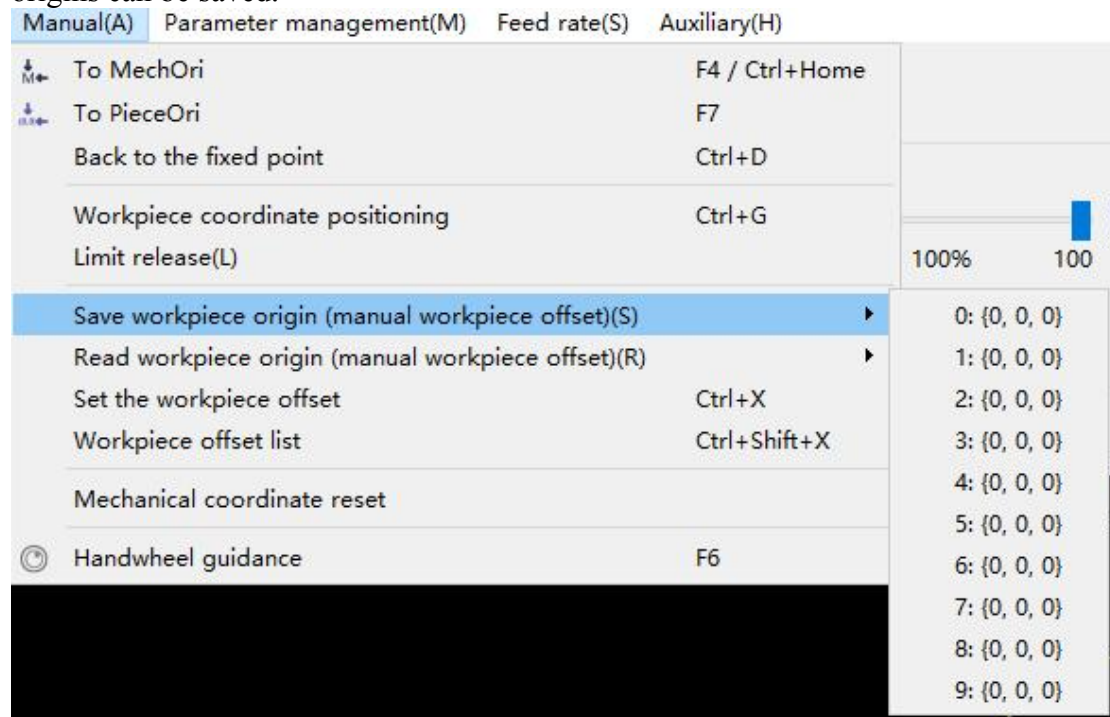


Figure 7-3 Save the Workpiece Origin

**Note:**

Users can choose the menu item "Save Workpiece Origin" in the "Manual" menu to save the current workpiece origin to the machining file system. In this way, users can save the frequently used workpiece origin as a preset value. When users set the

---

workpiece origin for the first time and want to quickly locate it in the future, they can use this function to save the coordinate value of the workpiece origin.

---

## Chapter VIII Manual Machining

Manual machining means that the machine tool manually machines the machining program according to the parameters set by the user. There are four ways to manually operate a machine tool: handwheel pulse mode, continuous inching mode, incremental stepping mode, and custom step length mode.

Users can choose the manual operation mode to machine program files. After clicking the "Manual" button on the multi-functional window at the bottom right of the main interface, the window will display a manual operation interface, where you can perform corresponding manual operations. The manual button area in this window contains six manual buttons, corresponding to the positive and negative directions of the X, Y, and Z axes. The manual window provides an interactive operating environment for users to manually manipulate the machine tool.

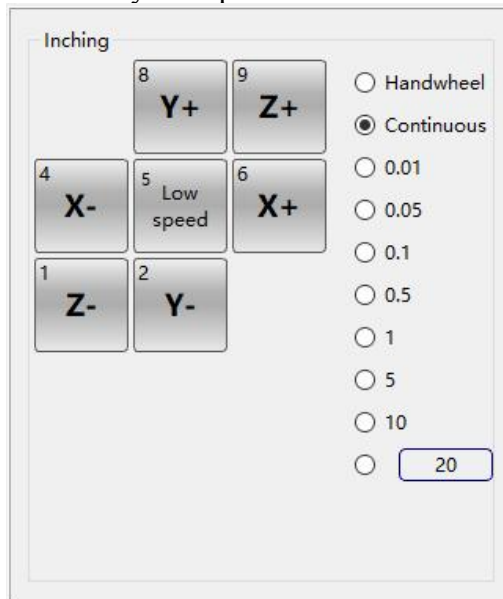


Figure 8-1 Manual Machining Window

### 8.1 Handwheel pulse mode

Users can choose the handwheel pulse method for continuous machining. When the radio button on the right side of the manual machining window is switched to the "handwheel" option, the movement of the machine tool is determined by the handwheel input.

1. There is a pulse rate option on the handwheel: X1, X10 and X100 gears, which represent different pulse multiples of the handwheel.
2. There is an axis option on the handwheel: Users can select the X, Y, and Z axes to feed.
3. There is a step direction option on the handwheel: Each axis has positive and negative directions. After selecting the stepping axis on the handwheel, the handwheel can be moved in both positive and negative directions, as indicated by the "+/-" arrow on the corresponding direction knob on the handwheel.
4. When the handwheel is in gears X1, X10, and X100, the distance corresponding to each grid of the handwheel can be set through parameters.
5. The acceleration of the handwheel can be set separately.
6. The handwheel supports two modes: 1. Strict pulse counting, 2. Non-strict pulse counting. In the strict pulse counting, the distance traveled by the machine tool is strictly equivalent to the number of manually operated pulses (note: in this mode, if the manually operated pulses are too fast, it may cause too many buffered pulses, and when the handwheel is stopped, the machine tool will also travel a considerable



---

distance). In the non-strict pulse counting mode, the number of manually operated pulses is not strictly equivalent to the distance traveled by the machine tool. When the handwheel stops, the machine tool immediately begins to slow down and stop.

**Note:**

Before performing the operation, check that the external handwheel device is properly connected. This mode is mainly used for rapid positioning of machine tools.

**8.2 Continuous inching mode**

Select the "Continuous" radio button on the right side of the window with the mouse to enter the continuous inching machining mode. In this mode, users can click the manual button with the mouse. When the manual button is pressed on the corresponding axis with the mouse, the machine tool moves; once released, the machine tool stops moving.



In the inching mode, the path display window will display the relevant machining path.

**8.3 Stepping mode**

Similar to the continuous inching mode, the stepping mode is another manual operation mode of the machine tool. Unlike the continuous inching mode, the stepping mode can accurately control the feed distance of the machine tool's motion axis.

Before using this mode for operation, a suitable step length must be set, and the distance for each inching feed must be set by modifying the inching step length.

The step length can be set in the following ways:

Enter the manual page, click   with the mouse to pop up a dialog box, and fill in the step length value. As shown in the figure below:

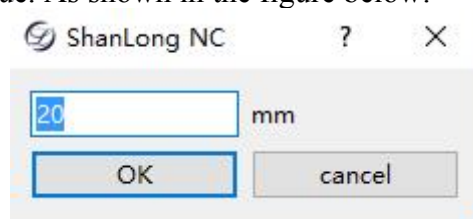


Figure 8-2 Custom Step Length Window

**Note:**

**It is necessary to avoid setting the inching step length value in the Z direction too large to prevent the damage to the machine tool due to misoperation.**

## Chapter IX Machining Operation

### 9.1 Load file

Click "Program Management" to switch to the program management window, select the file to be machined, and click "Load" or double-click the mouse. After loading, the file loading success will be displayed on the right side of the toolbar above the main interface, and the loaded file name will be displayed in the title bar below the main interface. At this time, it will automatically switch to the automatic machining window state.

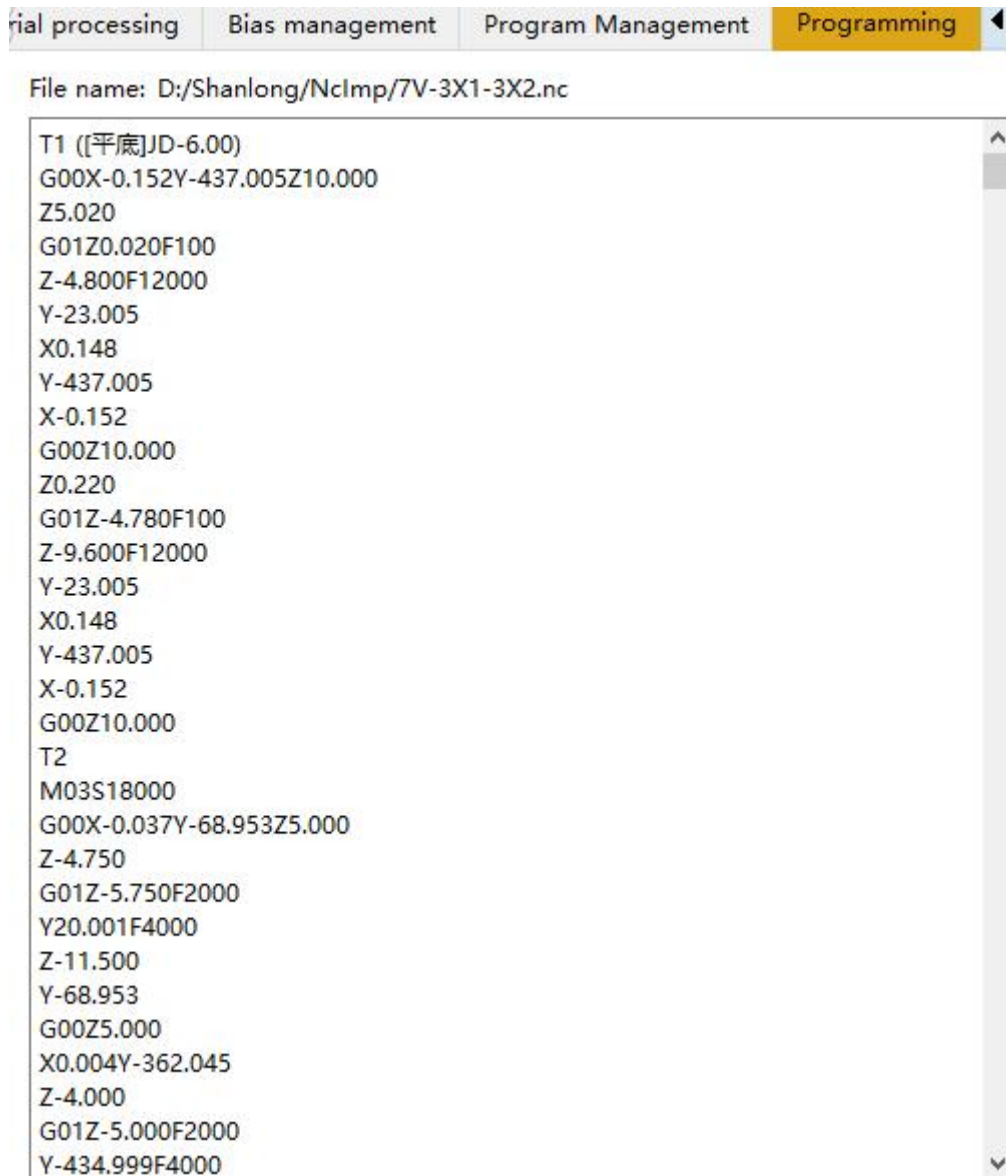


Figure 9-1 "Load" Machining Files

## 9.2 Set the workpiece origin

Refer to Chapter VII for details. If the workpiece origin has been set, there is no need to reset it.

## 9.3 Automatic machining

The "Automatic" menu contains items related to automatic machining.

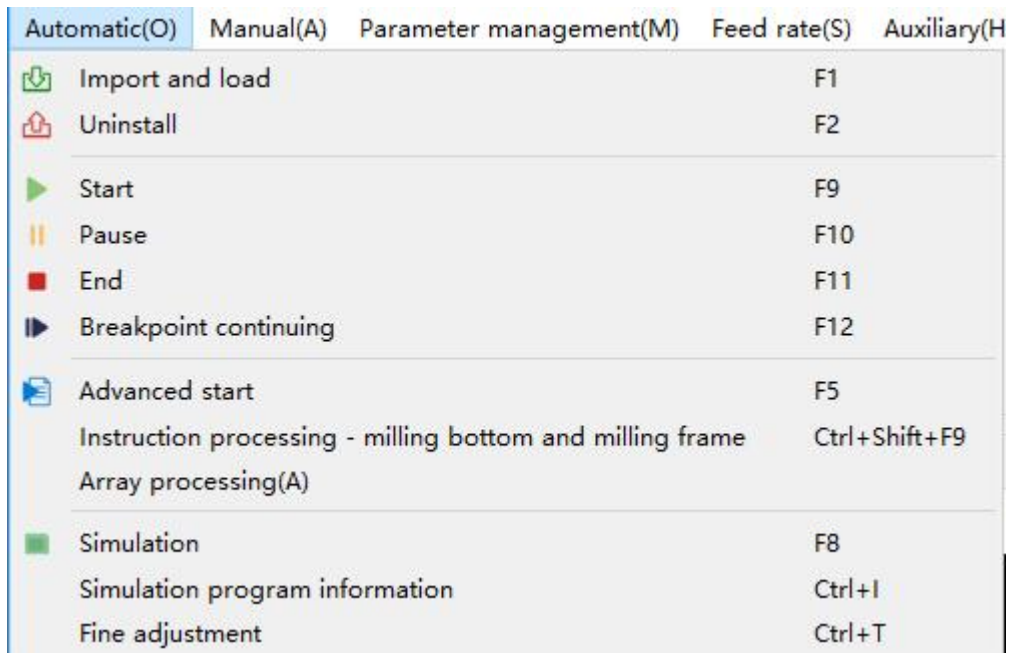



Figure 9-2 "Automatic" Menu

### 9.3.1 Start

After selecting the machining file, users can click on the "Start" button icon in the tool and status bar at the bottom of the main interface,  or the shortcut key "F9", and the machine tool will start automatic machining from the first line according to the selected machining file. In the machining path window, it can be seen that the corresponding machining path is displayed based on the tool movement; In the automatic machining window, it can be seen that the program is machined line by line, and the cursor will automatically track to the current code. The highlighted red code is continuously scrolling downwards, and users can view the current machining program code information through this window.

**Note:**

To set parameters, users must perform the operation of returning to the mechanical origin before machining, and the system will prompt you to return to the mechanical origin first. Automatic machining instructions cannot be executed without returning to the mechanical origin

```

N00023 G01Z-5.750F2000
N00024 Y20.001F4000
N00025 Z-11.500
N00026 Y-68.953
N00027 G00Z5.000
N00028 X0.004Y-362.045
N00029 Z-4.000

```

Figure 9-3 Automatic Machining Interface Status

Figure 9-4 Current Automatic Machining Status Prompt in the Toolbar

**Note:**

The system will perform syntax check on automatically machined files while machining them, and syntax check is performed earlier than automatic machining (i.e. syntax check has a "forward-looking" function). If the system detects a syntax error in a certain line of the program in the machining file, the error statement will be

highlighted in the automatic machining window and an alarm will be given, and the automatic machining will stop at the same time. Users can perform syntax and semantic checks on erroneous statements, modify and edit them. After editing, click "Save", and then click the "Breakpoint Continuing" button on the upper left toolbar of the main interface again, at this time, the program will continue automatic machining from the modified area.

During the automatic machining, new machining files cannot be loaded. The status bar "Machining Information Window" will display information on the start machining time, machined time, and tool number currently in use of the current file. It is convenient for users to view the operation status of automatic machining.

In the feed rate area, the slider can be adjusted or the feed rate setting value can be changed to reset the feed rate; In the spindle speed area, the slider can be adjusted or the spindle speed setting value can be changed to reset the spindle speed. The value change will take effect immediately.

The start and end information of automatic machining is saved in the system log file. The system log records important actions and events that occur for users, who can not only browse the log information that has occurred since this startup from the system log window, but also review the historical information records that have occurred through this window. If a system fault occurs, users can conduct system analysis and diagnosis through this function.

	Time	Type	Module	Message
1	2023-06-16 16:27:37	Error	Controller:Sports	Interpolation error
2	2023-06-16 16:27:37	Prompt	Controller:Sports	Stop <8-8-4X4.nc> on line 45
3	2023-06-16 16:27:37	Prompt	Controller:Sports	Stop spindle completion
4	2023-06-16 16:27:34	Prompt	Controller:Sports	Stop spindle start
5	2023-06-16 16:27:21	Prompt	Controller:Sports	Start spindle completed
6	2023-06-16 16:27:19	Prompt	Controller:Sports	Start spindle started
7	2023-06-16 16:27:18	Prompt	Controller:Sports	Start<8-8-4X4.nc> on line 1
8	2023-06-16 16:27:18	Note	Client:Default	User triggered processing start/continue
9	2023-06-16 16:27:01	Prompt	Controller:Sports	Stop spindle completion
10	2023-06-16 16:26:59	Error	Controller:Sports	X轴软件限位正向超限!
11	2023-06-16 16:26:59	Prompt	Controller:Sports	Stop <8-8-4X4.nc> on line 2

Figure 9-5 System Log Function

The current log information recorded by the system includes:


- (1) Start and end information of automatic machining;
- (2) Changes in workpiece coordinates;
- (3) System alarm information;
- (4) Completed machining information of the files;
- (5) Other system information.

**Note:**

**The system log can only store 1000 pieces of information.**

### 9.3.2 Pause

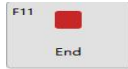
After "Start" automatic machining, in order to pause machining, users can click the

"Pause" button icon on the upper left toolbar of the main interface,  or the

shortcut key "F10", and the machine will slow down from the current speed until the speed reaches zero.

### 9.3.3 Stop

After "Starting" automatic machining, in order to stop machining the file, users can click on the "Stop" button icon in the upper left toolbar of the main interface,



or the shortcut key "F11", and the machine will slow down from the current speed until the speed is zero and the tool is lifted. When the system stops, breakpoints will be automatically saved.

During automatic machining, if the system is in simulation mode, click the "Stop" button to stop the simulation, and exit the simulation mode. At this time, users can analyze the simulation results.

### 9.3.4 Fine tuning

During the machining, press "Fine Tuning" in the automatic menu to pop up the interface as shown in the figure:

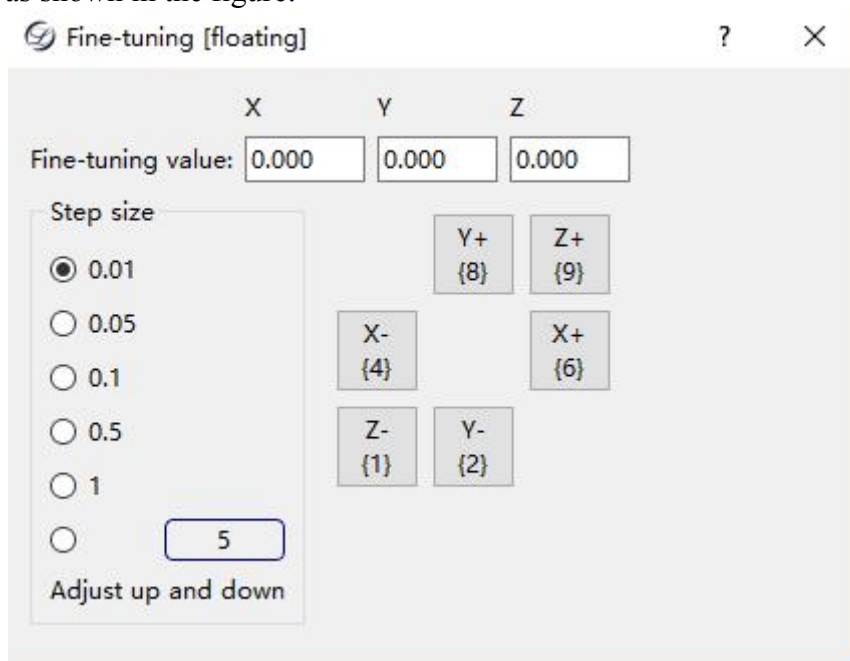


Figure 9-6 Fine Tuning during Machining

The step length can be selected, and the step length value is the fine tuning value. As shown in the interface, the number keys correspond to fine tuning in different directions of the X, Y, Z, and A axes.

### 9.3.5 Breakpoint continuing

In order to continue machining the workpiece from the last stop, users can click on the "Breakpoint Continuing" button icon in the upper left toolbar of the main interface,



or the shortcut key "F12". If the breakpoint continuing is caused by a power outage during the machining of the file, it must be returned to the mechanical origin before continuing.


If there is a sudden power outage during machining, users can click the "Breakpoint Continuing" button to restore the scene. The system will continue machining from the point where the last machining interruption occurred. "Breakpoint Continuing" can also be used to continue the simulation.

### 9.3.6 Advanced start

---

If the entire file does not need to be machined, as long as it starts from a specified line in the machining file and ends at the specified line number, it is the "Skip Segment Execution" in automatic machining. It can also be used to check whether a certain program in the machining file is correct.



Click , or use the shortcut key "F5", or select the "Advanced Start" menu item from the "Automatic" menu. This function implements the functions of "Skip Segment Execution" and "Nearest Point Machining". After selecting this function, the system will pop up the "Execute (Advanced Options)" dialog box, as shown in Figure 9-7:

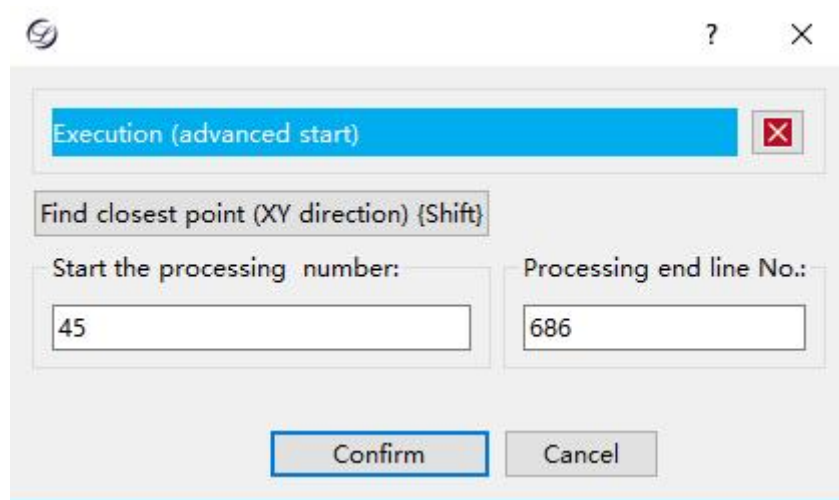


Figure 9-7 Advanced Start

Users can set the starting and ending line numbers of the file in the dialog box, and then click the "OK" button, and the machine tool will only execute the specified program segments in the entire machining program according to the requirements.

**Note:**

**If users choose to start from the file to the end of the file, the entire program file will be machined, which is the maximum range of "Skip Segment Execution".**

If the X-axis and Y-axis are moved after stopping machining, to continue machining from the stop point, users can move to the stop point and click [Find the Nearest Point] to continue machining

**9.3.7 Array machining**

Users can perform array machining on files, and click to select files that require array machining. After entering the parameters of array rows, array columns, array row spacing, and array column spacing, click to generate the machining file, which will generate the array file in the program management directory. The file name after the array is named according to the following rules: Original file name array rows \* Number of array columns

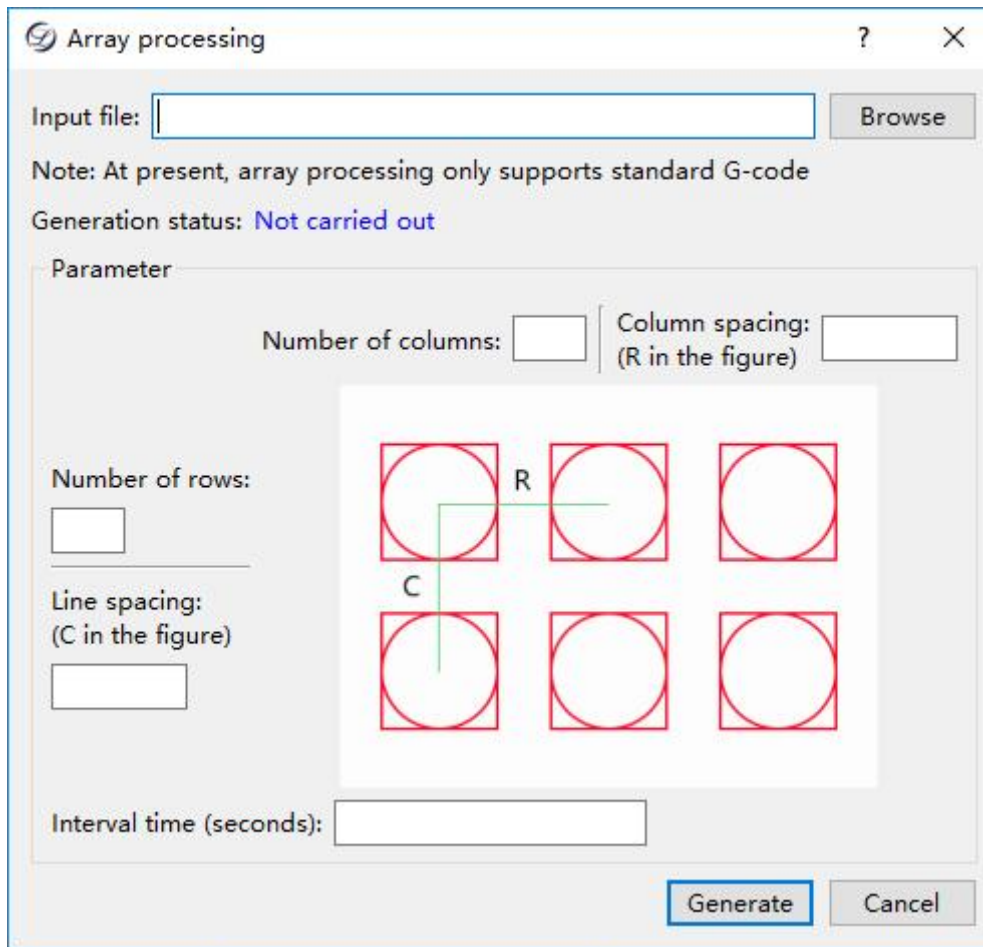


Figure 9-8 Select Array Files

#### 9.4 Handwheel guided machining

The system supports handwheel guided machining and needs to enter handwheel guided mode, as showing in the following figure:



Figure 9-10

Select the "Handwheel Guided" option in the toolbar at the bottom of the interface, and click the "Start" button icon. As it is in handwheel guided mode, the machine tool will not move.


After shaking the handwheel, the machine tool will adjust the speed according to the speed of the handwheel movement along the machining path. When the handwheel is stopped, the machine tool will also stop. When continuing to shake the handwheel, the machine tool will continue machining along the original path. The entire machining process is controlled by a handwheel.

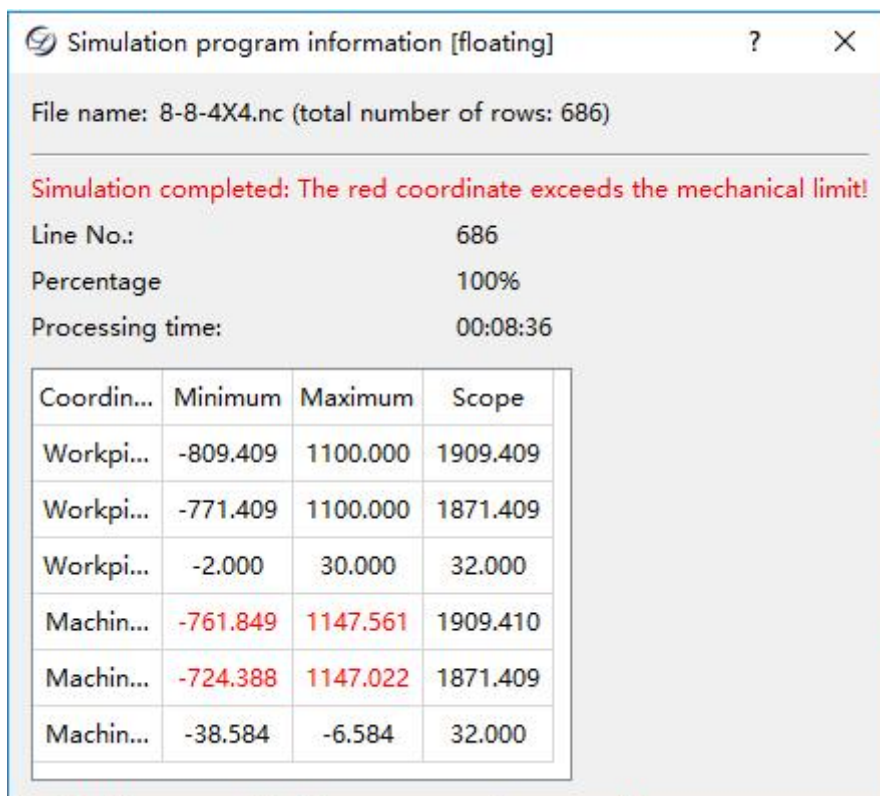
**Note:**

During the handwheel guided machining, it is only by shaking it in the positive direction of the handwheel that the machining can be guided; otherwise, it is invalid.

## Chapter X Check Machining File

When the machining files have been loaded and the current system status is "idle", users can choose the "Simulation Program Information" option in the "Automatic" menu (or click the "Simulation" button icon in the upper left toolbar of the main

interface,  or the shortcut key "F8") to perform high-speed simulation on the loaded machining files.



Coordin...	Minimum	Maximum	Scope
Workpi...	-809.409	1100.000	1909.409
Workpi...	-771.409	1100.000	1871.409
Workpi...	-2.000	30.000	32.000
Machin...	-761.849	1147.561	1909.410
Machin...	-724.388	1147.022	1871.409
Machin...	-38.584	-6.584	32.000

Figure 10-1 Simulation Program Information

Simulation provides users with a fast and realistic simulation machining environment. After the simulation starts, the system no longer emits pulses to drive the machine tool movement, but only tracks and displays the actual effect of tool machining in the window at a high speed. Through simulation, users can understand the movement and machining effect of the machine tool, as well as other additional information in advance to prevent errors in editing the machining program from causing damage to the machine tool. Once the simulation process begins, the menu item will change to "Stop Simulation and Exit Simulation Mode", through which the simulation will immediately terminate.

### Note:

The simulation information includes:

- (1) When the simulation limit set in the parameters is valid, the system will check whether it will exceed the travel during the simulation. If there is an overtravel prompt during simulation, the actual machining will also overtravel without changing the workpiece origin.
- (2) During the simulation, syntax check will be performed on the G code, and syntax errors will be reported.

In the machining path, right-click to enter "Simulation Settings".



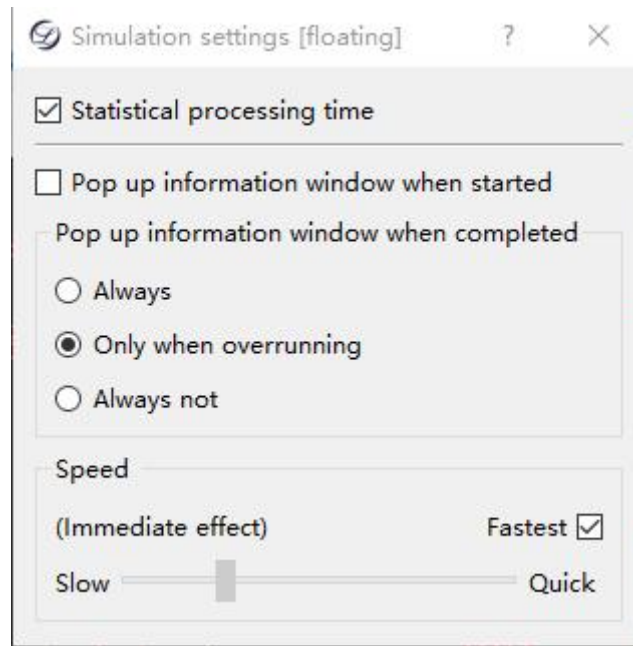


Figure 10-2 Simulation Settings

**Enable controller simulation:** When the controller simulation is enabled, it will display "total file line number", "machining time", and "machining percentage". If it is not enabled, it will not be displayed.

**A message prompt window will pop up after starting:** When the controller simulation machining is enabled, the information window that pops up after starting cannot be cancelled, which can only be cancelled after canceling controller simulation machining

**A message window will pop up after completing:** When the controller simulation machining is enabled, it will always pop up. When the controller simulation machining is cancelled, the information window can only pop up "only when exceeding the limit"

**Plotting speed:** The speed setting of the progress bar will only take effect when the "fastest" is cancelled

## Chapter XI Milling Bottom and Frame Operations

When users need to perform simple milling bottom and frame operations, there is no need to manually write G code or generate machining files using CAM/CAD software. On the [Automatic] - [Execute Machining Instructions] interface, the system provides the function of executing machining instructions, and it can be completed by inputting a few parameters.

"Execute Machining Instructions" have functions such as rectangular milling bottom, circular milling bottom, milling rectangular frame, and milling circular frame.

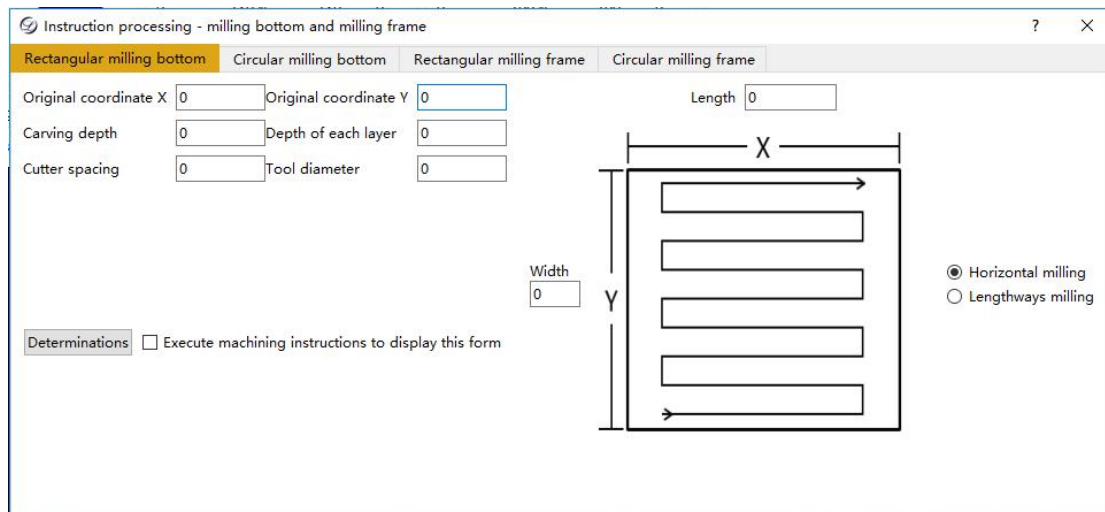
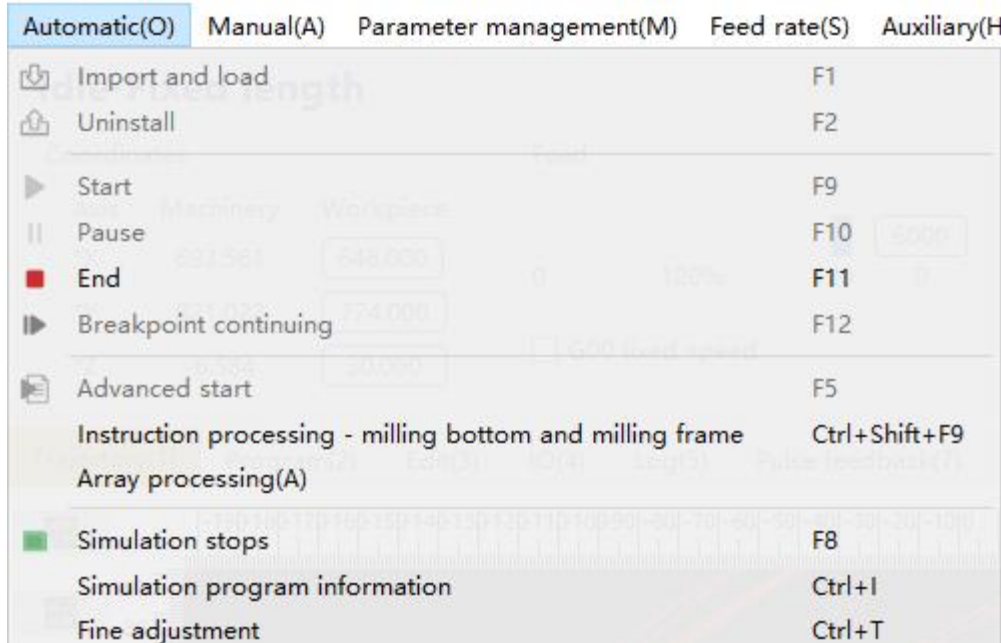


Figure 11-1 Circular Milling Bottom Window

## Chapter XII Program Management

Click [Program Management] to switch to the program management window, where you can create, edit, delete, rename, load, uninstall, import, import and load, browse folders, and refresh machining program files.

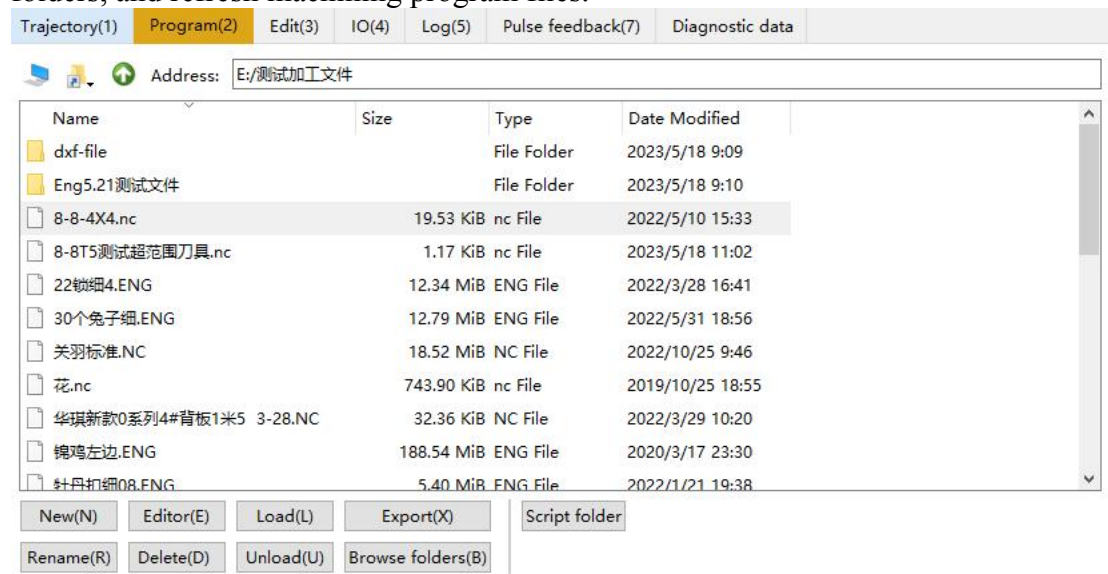


Figure 12-1 "Program Management" Window

### 12.1 Create

After clicking the "Create" button at the bottom of the [Program Editing] window, a blank new file with the default name (New File 1. nc) will be created in the window.

### 12.2 Edit

Select an existing file or create a new one, and click the "Edit" button at the bottom of the window to edit and modify the file. Alternatively, click [Program Editing] to switch to the program editing window, and right-click in the editing window to pop up a context menu. Users can easily copy, paste, cut, and other functions to quickly edit and modify the program. After editing, users can right-click and select "Save" or "Close before Saving".

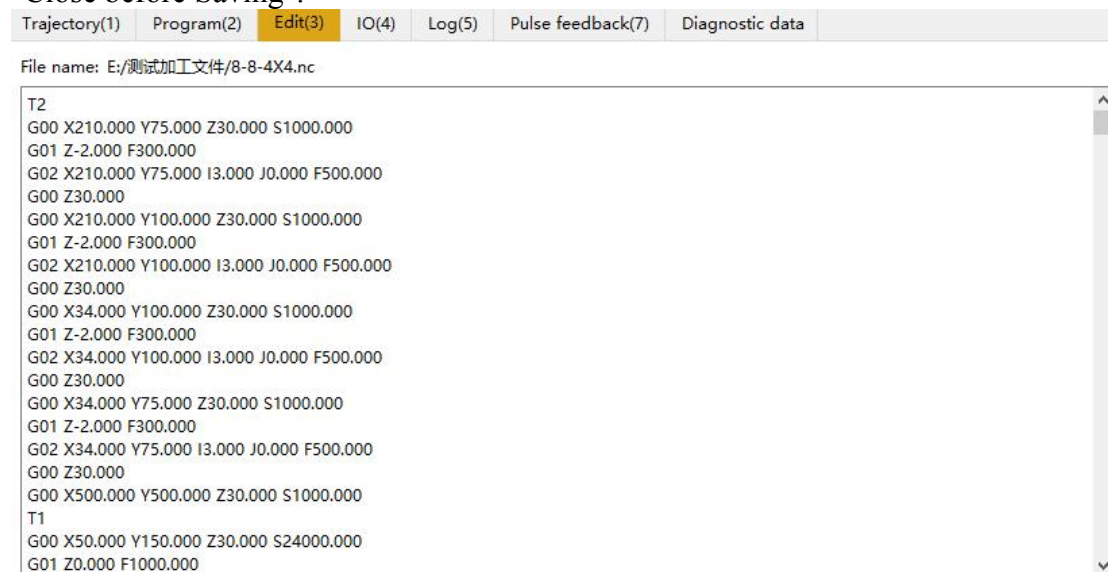


Figure 12-2 "Program Editing" Window

For newly created or edited files, the system will automatically perform a syntax check before saving. Users must edit files according to our programming standards;

---

otherwise, the system will report an error. Refer to *Programming Specification* for details.

**Note:**

In this editing window, users can edit machining programs smaller than 10 megabytes in size. In case of files larger than 10 megabytes in size, please use an editor on your PC.

In the editing window, users can input any text. After the input is completed, the system will automatically perform a syntax check to ensure that the machine tool will not execute incorrect instructions and cause damage to the machine tool.

**12.3 Delete**

Click or move the "Up and Down" keys of the keyboard to select the file to be deleted, and click the "Delete" key to delete the selected file. Certain files can also be selected for batch deletion.

**12.4 Load**

Select the machining file and click "Load" or double-click the mouse to load it. After the file is successfully loaded, the loaded file name will be displayed in the upper right corner of the interface.

## Chapter XIII Parameter Management

The [Parameter Management] menu includes items related to parameters, as shown in the figure:

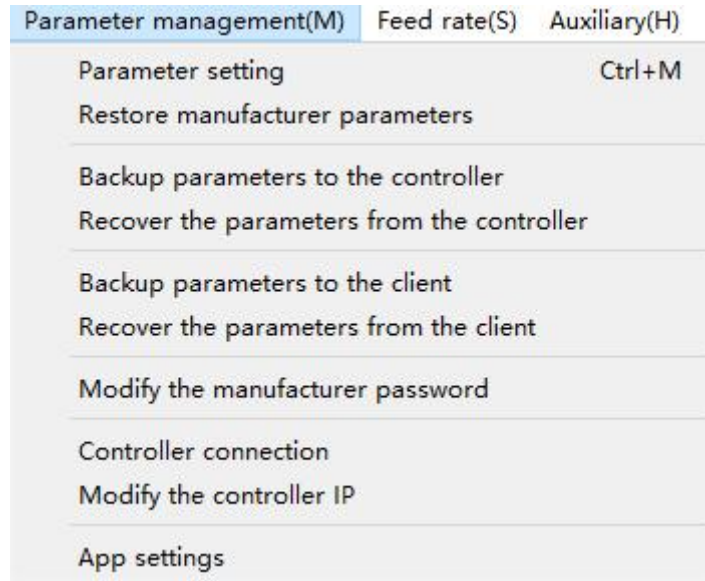


Figure 13-1 Parameter Management Menu

### 13.1 Setting parameters

Select the menu item [Set Parameters] in the [Parameter Management] menu to pop up the following window for parameter settings under parameter permissions. It is divided into two parts: user parameters and manufacturer parameters. The opening of tool parameters depends on whether [N81661] supports wheel driven tool change settings. Yes: it supports wheel driven tool change and opens tool parameters; No: it does not support wheel driven tool change, and tool parameters are shielded.

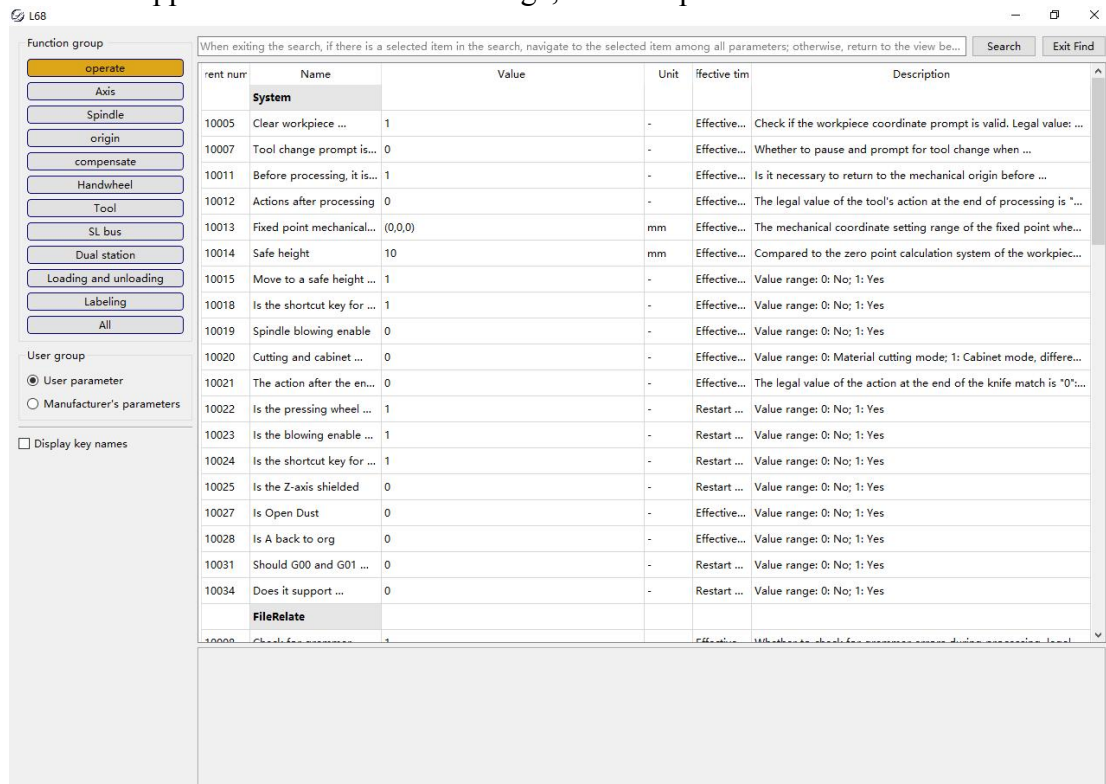


Figure 13-2 Setting Parameter Window

---

"User parameter" setting: To set user parameters, select the option "User Parameters" in the bottom left corner of the window, and then set the operation parameters, feed axis parameters, spindle parameters, origin parameters, compensation parameters, and tool parameters in the parameter categories. After setting, all user parameters will take effect.

"Manufacturer parameter" setting: Firstly, select the parameter permissions. To set manufacturer parameters, select the option "Manufacturer Parameters" in the bottom left corner of the window, and then set the operation parameters, feed axis parameters, spindle parameters, origin parameters, compensation parameters, and tool parameters in the parameter categories. After setting, all manufacturer parameters will take effect.

**Note:**

In general (default state), the parameter section displays user parameters for general machining. In order to modify machine performance related parameters, such as pulse equivalent, maximum spindle speed, etc., users need to enter a password and enable the manufacturer parameters for modification.

**13.2 Restore manufacturer parameters**

Used to restore the manufacturer parameters to their default values.

Select the menu item [Restore Manufacturer Parameters] in the [Parameter Management] menu to pop up the following window. Click "Yes" to enter the password (default is blank) and confirm to restore the manufacturer parameters to their default values.

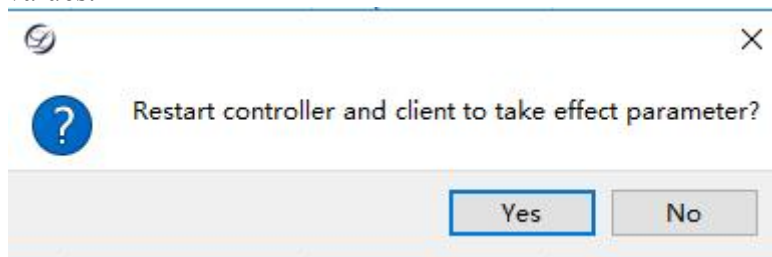


Figure 13-4 Restore Manufacturer Parameters

**13.3 Backup parameters to internal controller**

Used to back up system parameters internally for future use.

Select the menu item [Backup Parameters to Internal Controller] in the [Parameter Management] menu to pop up the following window, which is used to save system parameters in the form of a file to the internal system.

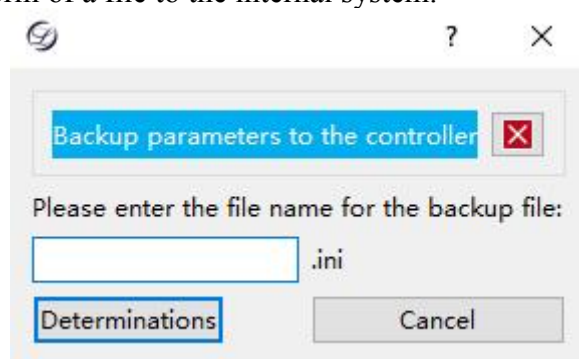


Figure 13-5 Backup Parameters To Internal Controller

**13.4 Backup parameters to client path**

Used to output system parameters in the form of files to a PC for data recovery.

Select the menu item [Backup Parameters to Client Path] in the [Parameter Management] menu to pop up the following window, which is used to output system parameters to a PC in the form of a file.

### 13.5 Restore parameters from the controller

Restore the parameters to the previously set values.

Select the menu item [Restore Parameters from the Controller] in the [Parameter Management] menu to pop up the following window, which is used to restore the parameters to the previously set values: Select the name of the parameter backup file in the pop-up window, and click the "OK" button. The system will restore the parameters to the values set at the time.

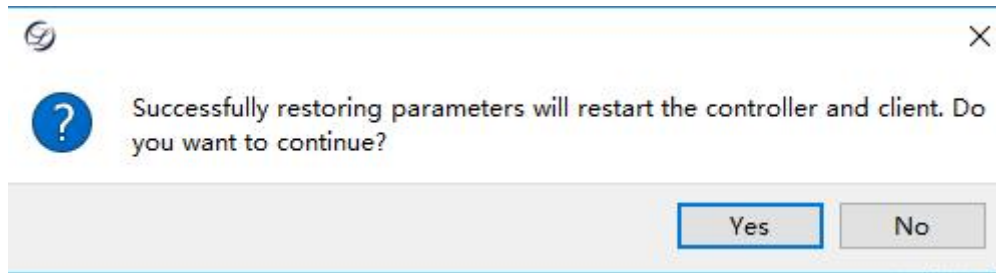


Figure 13-7 Restore Parameters from the Controller

### 13.6 Restore parameters from client path

Used to restore parameters saved on the PC to the system.

Select the menu item [Restore Parameters from Client Path] in the [Parameter Management] menu to pop up the following window, which is used to restore the parameters to the previously set values: Select the name of the parameter backup file in the pop-up window, and click the "OK" button. The system will restore the parameters to the values set at the time.

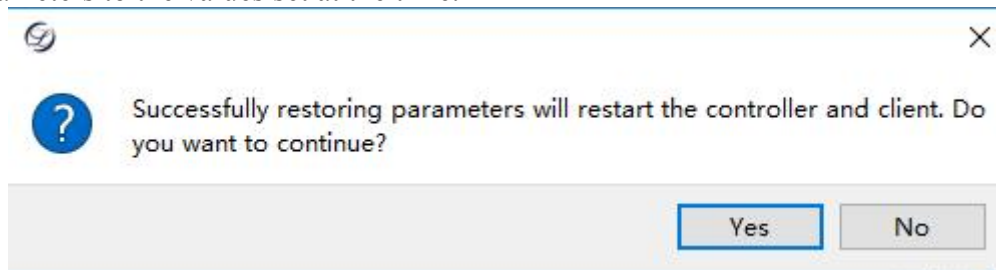


Figure 13-8 Restore Parameters from Client Path

### 13.7 Controller connection

If the controller is disconnected, users can click on [Controller Connection] in the [Parameter Management] menu to reconnect the controller. Click "Cancel" to exit the system

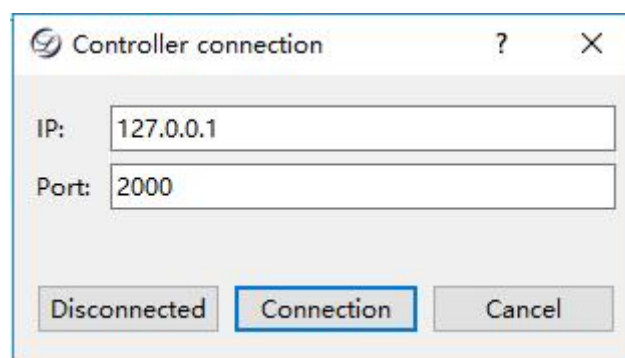


Figure 13-9 Controller Connection

---

### 13.8 Modify the password

Select the menu item [Modify the Manufacturer Password] in the [Parameter Management] menu to pop up the following window for modifying the manufacturer password, which can effectively protect the security of parameter settings.

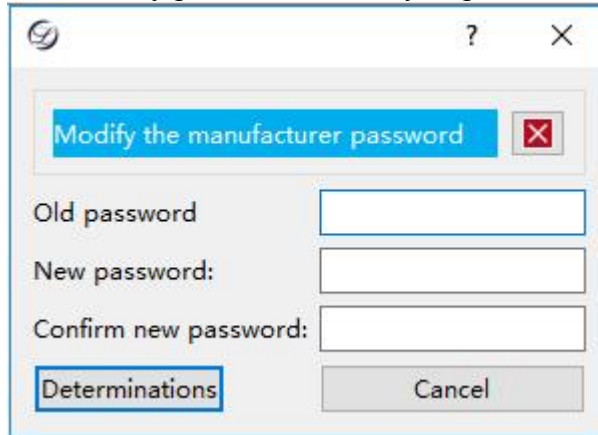


Figure 13-10 Modify the Password

The manufacturer password for the delivered equipment is blank. To modify the password, the "Old Password" does not need to be filled in. Enter the password to be modified in the "New Password", and then enter the new password to be modified again in the "Confirm New Password". After confirmation, the password will take effect immediately. When entering the manufacturer parameters again or restoring the factory parameters, a new password needs to be entered

There are many parameters involved in this system, which are divided into two categories: user parameters and manufacturer parameters. To modify and view a certain type of parameter, it is necessary to have the permissions to view and modify this type of parameter.

### 13.9 Modify the controller IP

Select the menu item [Modify the Control IP] in the [Parameter Management] menu to pop up the following window for modifying the controller IP, which can be modified to any IP (legal: compliant with TCP/IP protocol). After modification, click on [Restart Controller System] in the [Auxiliary] menu to restart, and make the modification take effect.



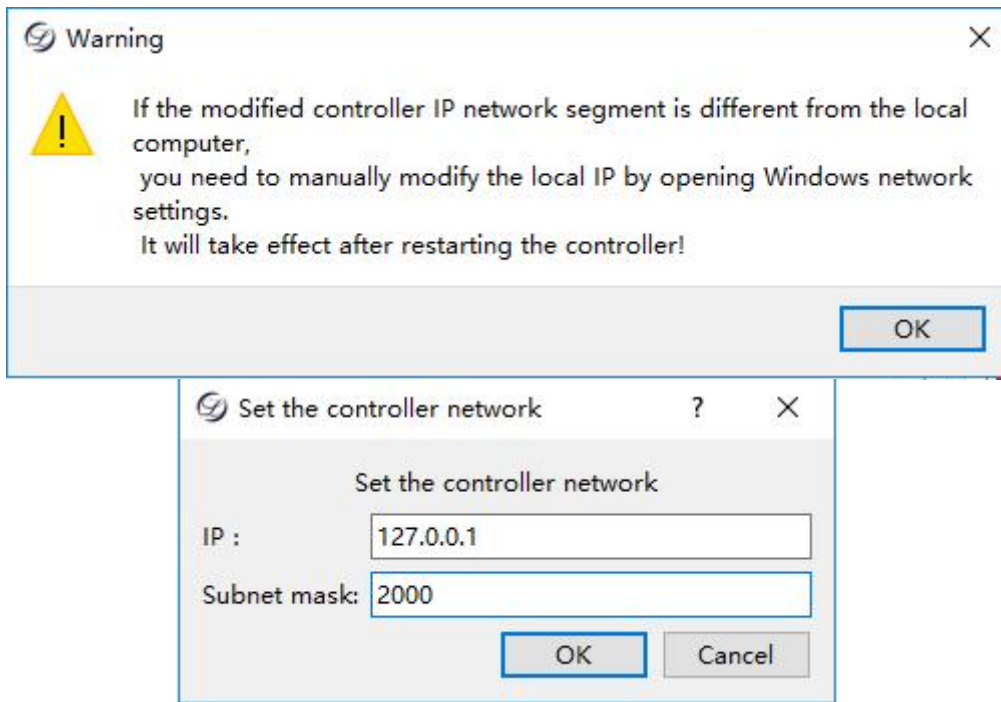


Figure 13-11 Modify the Controller IP

**Note:**

After modifying the controller IP, the PC IP must also be modified to be in the same network segment as the modified controller IP.

**13.10 Client settings**

Select the menu item [Client Settings] in the [Parameter Management] menu to pop up the following window for modifying the client interface layout, font style, and size. After modification, click [Restart Controller System] in the [Auxiliary] menu to restart, and made the modification take effect.

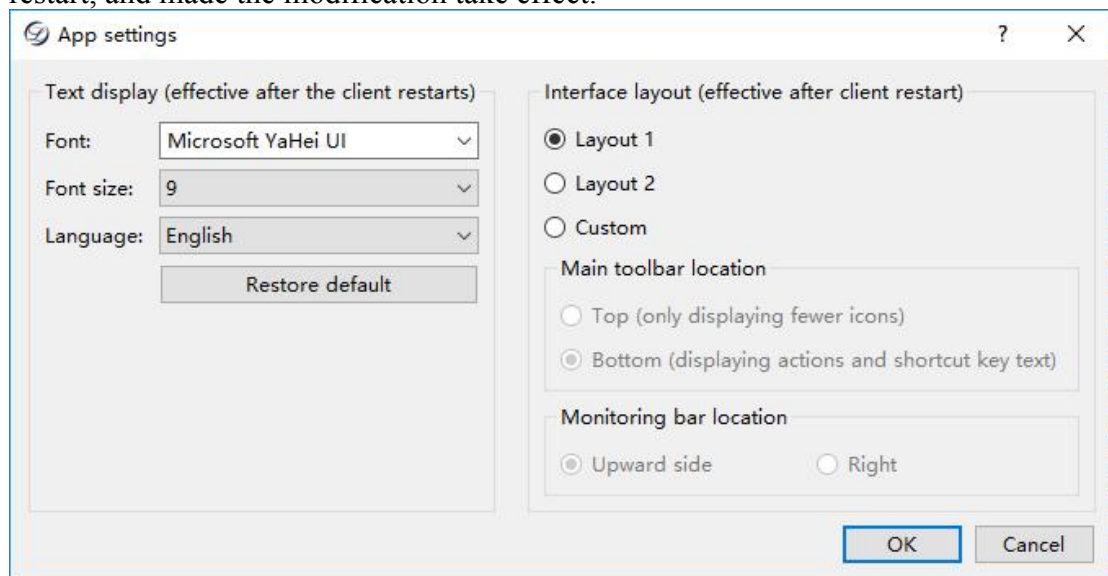


Figure 13-12 Client Settings

**13.11 Parameter modification methods**

The method to modify parameters is to double click on the row where the parameter is located with the mouse, pop up a dialog box, enter numerical values in the parameter area, and click to select "Yes" or "No" for "yes/no" type parameters.

The effective time of parameters can be divided into immediate effective and effective after restart. For parameter modifications that take effect immediately, the modification will take effect once completed. For parameter modifications that take effect after restart, the modification will only take effect after the system automatically restarts.

**Note:**

All parameters cannot be modified in the machining state, and can only be modified after the machining is completed and before the next machining begins.

**13.12 Overview of user parameters**

**Operating parameters**

No.	Parameter name		Meaning	Setting range	Permission
1.1 Manual					
N11000	Low speed under manual mode		When manually moving the machine tool at low speed, directly use the direction keys to operate	Default to 3000.000mm/min, range (takeoff speed, high speed under manual mode)	User/manufacturer
N11001	High speed under manual mode		When manually moving the machine tool at high speed, usually using CTRL + direction keys	Default to 6000.000mm/min, range (takeoff speed, 30000)	User/manufacturer
N11002	Workpiece coordinate clear prompt is valid		Whether a prompt dialog box pops up when setting the workpiece origin	Default to "Yes", range (Yes, No)	User/manufacturer
N11003	Exclude Z-axis during workpiece origin operation		Whether the Z-axis coordinate is not restored when reading the workpiece origin	Default to 0, range (Yes, No)	User/manufacturer
N81579	Whether the origin is valid	X axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to	Default to "Yes", range (Yes, No)	User/manufacturer

			"No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
N81580		Y axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "Yes", range (Yes, No)	User/manufacturer
N81581		Z axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "Yes", range (Yes, No)	User/manufacturer
N81582		A-axis	Set to "Yes". The origin serves as a mechanical	Default to "Yes", range (Yes, No)	User/manufacturer

			origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
1.2 Automatic					
N12000	Actions after machining		The next actions performed by the machine tool after the machining is completed	Default to 0, range (0: Hold still, 1: Return to fixed point; 2: Return to workpiece origin)	User/manufacturer
N12001	Mechanical coordinates of fixed points	X axis	Set fixed point X-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12002		Y axis	Set fixed point Y-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12003		Z axis	Set fixed point Z-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12004	Interpolation algorithm selection		Selection of machining effect and efficiency	Default to 2, range (0: High efficiency; 1: Quality first; 2: Balancing quality and efficiency)	User/manufacturer
N12005	Safety height		Reference height for machine tool stop and machining idle line	Default to 10mm, range (5, 500)	User/manufacturer

		elevation		
N12006	Idling speed	G00 command motion speed	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12007	Default feed speed	Command motion speed for G01, G02, G03, etc.	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12008	Use default feed speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N12009	Z-axis cutting speed	Speed limit of the Z-axis downward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12010	Z-axis cutter lifting speed	Speed limit of the Z-axis upward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12011	Approach speed	To protect the tool, the speed at which it is prepared to come into contact with the workpiece surface	Default to 300.000mm/min	User/manufacturer

		during machining		
N12012	Approaching distance	To protect the tool, the distance starting to use the approach speed, which is the distance from the workpiece surface.	Default to 5.000mm/min	User/manufacturer
N12013	Tool change prompt is valid	Whether to pause and prompt for tool change when encountering a tool change command during the machining.	Default to "Yes", range (Yes, No)	User/manufacturer
N12014	Whether to use cycle machining	Whether to run the cycle machining. If Yes, the cycle parameters on the main interface can be set. If No, the cycle parameters on the main interface cannot be set	Default to "No", range (Yes, No)	User/manufacturer
N12015	Whether to check syntax error during machining	Whether to report errors when encountering unrecognized codes during machining. Set to "Yes" to stop machining and prompt. Set to "No" to ignore the unrecognized code and	Default to "No", range (Yes, No)	User/manufacturer

		continue machining.		
N12016	Whether to check syntax error during simulation	Whether to report an error when encountering unrecognized code during the simulation. Set to "Yes" to stop simulation and prompt. Set to "No" to ignore the unrecognized code and continue simulation.	Default to "No", range (Yes, No)	User/manufacturer
N12017	Idling acceleration	Acceleration from the takeoff speed to the maximum speed during idle motion.	Default to 650.000mm/s <sup>2</sup> , range (1, 3000)	User/manufacturer
N12018	Idle acceleration jerk	Set the change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 10000.000/s <sup>3</sup> , range (1, 30000)	User/manufacturer
N81578	Whether G00 fixed speed is valid	Set whether G00 fixed speed is valid	Default to "No", range (Yes, No)	User/manufacturer
N13000	Action parameter selection during pause	Action of the machine tool when machining	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2:	User/manufacturer

		pauses.	Lift to the set tool lifting height)	
N13001	Z-axis tool lifting amount during pause	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during pause.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N13002	Tool lifting method when stopping	Action of the machine tool when machining stops.	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2: Lift to the reference point)	User/manufacturer
N13003	Z-axis tool lifting when stop	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during stop.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N14500	Floating feeler block thickness	Set the thickness of the floating feeler block, and automatically compensate for the thickness value when performing floating tool setting	Default to 0.000mm, range (0, 500)	User/manufacturer
N14501	Whether the floating tool setting is valid	Set to "Yes". The floating tool setting on the interface is operable and executable. Set to "No". The floating tool setting on the	Default to "No", range (Yes, No)	User/manufacturer



			interface cannot be operable and executable.		
N14502	Fixed tool setting gauge position	X axis	Set the fixed tool setting position and X-axis value	Default to 0.000mm	User/manufacturer
N14503		Y axis	Set the fixed tool setting position and Y-axis value	Default to 0.000mm	User/manufacturer
N14504		Z axis	Set the fixed tool setting position and Z-axis value	Default to 0.000mm	User/manufacturer
N14505	Whether the fixed tool setting is valid		Set to "Yes". The fixed tool setting on the interface is operable and executable. Set to "No". The fixed tool setting on the interface cannot be operable and executable.	Default to "No", range (Yes, No)	User/manufacturer
N14506	Fixed tool setting gauge thickness		Set the height of the fixed tool setting gauge relative to the table	Default to 0.000mm, range (0, 1000)	User/manufacturer
N14507	Tool setting speed		Speed at which the machine tool moves from the set fixed tool setting gauge position to the tool setting gauge surface.	Default to 60mm/min, range (0, 1000)	User/manufacturer
N14508	Fast fixed tool setting speed		Speed at which the Z-axis of the machine tool moves to the	Default to 300mm/min, range (0, Z-axis cutting speed)	User/manufacturer

		Z position set by the fixed tool setting gauge.		
Others				
N17000	Whether the limit is detected during simulation	Select "Yes" to consider whether the file exceeds the software limit during the simulation. If it exceeds the limit, stop the simulation and report an error; Select "No" to not consider the file machining range during the simulation.	Default to "No", range (Yes, No)	User/manufacturer

### Spindle parameters

No.	Parameter name	Meaning	Setting range	Permission
N31000	Use the default spindle speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N31001	Stall during pause	Set to "Yes". The spindle stops rotating when encountering a pause command; Set to "No". The spindle does not stop rotating when encountering a pause command.	Default to "Yes", range (Yes, No)	User/manufacturer
N31002	Stall during stopping	Set to "Yes". The spindle stops rotating when encountering a stop	Default to "Yes", range (Yes, No)	User/manufacturer

		command; Set to "No". The spindle does not stop rotating when encountering a stop command.		
--	--	--	--	--

### Origin parameters

No.	Parameter name		Meaning	Setting range	Permission
N41000	Before machining, it is necessary to return to the mechanical origin first		Set to "Yes". The machine must return to the mechanical origin before machining. If not executed, the system will prompt an error during machining. Set to "No". Regardless of whether to perform a return to the mechanical origin, the system can perform machining.	Default to "No", range (Yes, No)	User/manufacturer
N41503	Coarse positioning stage direction	X axis	Set to 1. The X-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The X-axis quickly searches for the mechanical origin	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			direction as the negative direction.		
N41504		Y axis	Set to 1. The Y-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Y-axis quickly searches for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41505		Z axis	Set to 1. The Z-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Z-axis quickly searches for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41506	Coarse positioning stage speed	X axis	Operating speed of the X-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer

N41507		Y axis	Operating speed of the Y-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41508		Z axis	Operating speed of the Z-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41509	Fine positioning stage direction	X axis	Set to 1. The X-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The X-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41510		Y axis	Set to 1. The Y-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Y-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

N41511		Z axis	Set to -1. The Y-axis searches again for the mechanical origin direction as the negative direction. Set to -1. The Z-axis searches again for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41512	Fine positioning stage speed	X axis	Operating speed of the X-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41513		Y axis	Operating speed of the Y-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41514		Z axis	Operating speed of the Z-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41515	Retrace distance	X axis	Distance to retreat after the X-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41516		Y axis	Distance to retreat after the Y-axis completes searching for the mechanical	Default to 4.000mm, range (-1000, 1000)	User/manufacturer

			origin		
N41517		Z axis	Distance to retreat after the Z-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41518	Permissible movement direction during origin limit	X axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41519		Y axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41520		Z axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

### 13.13 Overview of manufacturer parameters

#### Operating parameters

No.	Parameter name		Meaning	Setting range	Permission
1.1 Manual					
N11000	Low speed under manual mode		When manually moving the machine tool at low speed, directly use the direction keys to operate	Default to 3000.000mm/min, range (takeoff speed, high speed under manual mode)	User/manufacturer
N11001	High speed under manual mode		When manually moving the machine tool at high speed, usually using CTRL + direction keys	Default to 6000.000mm/min, range (takeoff speed, 30000)	User/manufacturer
N11002	Workpiece coordinate clear prompt is valid		Whether a prompt dialog box pops up when setting the workpiece origin	Default to "Yes", range (Yes, No)	User/manufacturer
N11003	Exclude Z-axis during workpiece origin operation		Whether the Z-axis coordinate is not restored when reading the workpiece origin	Default to 0, range (Yes, No)	User/manufacturer
N81579	Whether the origin limit is valid	X axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to	Default to "1", range (Yes, No)	User/manufacturer



			"No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
N81580		Y axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "1", range (Yes, No)	User/manufacturer
N81581		Z axis	Same as above	Default to "1", range (Yes, No)	User/manufacturer
N11500	Manual direction	X axis	Manual direction 1: Positive direction -1: Negative direction	Default to "-1", range (positive, negative)	User/manufacturer
N11501		Y axis	Manual direction 1: Positive direction -1: Negative direction	Default to "-1", range (positive, negative)	User/manufacturer
N11502		Z axis	Manual direction 1: Positive direction -1: Negative direction	Default to "1", range (positive, negative)	User/manufacturer
1.2 Automatic					

N1200 0	Actions after machining		The next actions performed by the machine tool after the machining is completed	Default to 0, range (0: Hold still, 1: Return to fixed point; 2: Return to workpiece origin)	User/manufacturer
N1200 1	Mechanical coordinates of fixed points	X axis	Set fixed point X-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N1200 2		Y axis	Set fixed point Y-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N1200 3		Z axis	Set fixed point Z-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N1200 4	Interpolation algorithm selection		Selection of machining effect and efficiency	Default to 2, range (0: High efficiency; 1: Quality first; 2: Balancing quality and efficiency)	User/manufacturer
N1200 5	Safety height		Reference height for machine tool stop and machining idle line elevation	Default to 10mm, range (5, 500)	User/manufacturer
N1200 6	Idling speed		G00 command motion speed	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N1200 7	Default feed speed		Command motion speed for G01, G02, G03, etc.	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N1200 8	Use default feed speed		Whether to use parameter settings for	Default to "Yes", range (Yes, No)	User/manufacturer

		speed, "Yes" to use parameter speed, "No" to use file internal speed.		
N12009	Z-axis cutting speed	Speed limit of the Z-axis downward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12010	Z-axis cutter lifting speed	Speed limit of the Z-axis upward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12011	Approach speed	To protect the tool, the speed at which it is prepared to come into contact with the workpiece surface during machining	Default to 300.000mm/min	User/manufacturer
N12012	Approaching distance	To protect the tool, the distance starting to use the approach speed, which is the distance from the workpiece surface.	Default to 5.000/min	User/manufacturer
N12013	Tool change prompt is valid	Whether to pause and	Default to "No", range (Yes, No)	User/manufacturer

		prompt for tool change when encountering a tool change command during the machining.		
N12014	Whether to use cycle machining	Whether to run the cycle machining. If Yes, the cycle parameters on the main interface can be set. If No, the cycle parameters on the main interface cannot be set	Default to "No", range (Yes, No)	User/manufacturer
N12015	Whether to check syntax error during machining	Whether to report errors when encountering unrecognized codes during machining. Set to "Yes" to stop machining and prompt. Set to "No" to ignore the unrecognized code and continue machining.	Default to "No", range (Yes, No)	User/manufacturer
N12016	Whether to check syntax error during simulation	Whether to report an error when encountering unrecognized code during the simulation. Set to "Yes" to stop simulation	Default to "No", range (Yes, No)	User/manufacturer

		and prompt. Set to "No" to ignore the unrecognized code and continue simulation.		
N12017	Idling acceleration	Acceleration from the takeoff speed to the maximum speed during idle motion.	Default to 650.000mm/s <sup>2</sup> , range (1, 3000)	User/manufacturer
N12018	Idle acceleration jerk	Set the change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 10000.000/s <sup>3</sup> , range (1, 30000)	User/manufacturer
N81578	Whether G00 fixed speed is valid	Set whether G00 fixed speed is valid	Default to "No", range (Yes, No)	Manufacturer
N12500	Takeoff speed	Starting speed, which can reach the set speed at the moment of startup, but if it is too high, the machine tool will vibrate	Default to 200.000mm/min, range (0, the minimum value among the maximum speeds of each axis)	Manufacturer
N12501	Individual axis acceleration	Acceleration or deceleration of the axis when moving in a straight line. The	Default to 500.000mm/s <sup>2</sup> , range (0.01, 2000)	Manufacturer

			greater the acceleration is, the shorter the time to reach the same speed will be.		
N1250 2	Turning acceleration		Maximum acceleration of the feed axis on adjacent axes during curve interpolation.	Default to 800.000mm/s <sup>2</sup> , range (1, 100000)	Manufacturer
N1250 3	Acceleration jerk		Change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 12000.000mm/s <sup>3</sup> , range (0.01, 300000)	Manufacturer
N1250 4	Maximum speed of each axis	X axis	Maximum running speed of X-axis	Default to 6000.000mm/min	Manufacturer
N1250 5		Y axis	Maximum running speed of Y-axis	Default to 6000.000mm/min	Manufacturer
N1250 6		Z axis	Maximum running speed of Z-axis	Default to 6000.000mm/min	Manufacturer
N1250 7	Minimum speed for arc machining		Minimum speed when performing arc machining	Default to 250.000mm/min	Manufacturer
N1250 8	Maximum speed for arc machining		Maximum speed when performing arc	Default to 9000.000mm/min	Manufacturer

		machining		
N12509	Arc radius tolerance	In the IJK increment representation of G02 and G03, the radius of the circle is calculated twice, but generally, the values obtained from these two calculations are different. The difference between them is the arc radius tolerance, which is generally not significant. The recommended error is 0.1	Default to 1.000mm, range (0, 10)	Manufacturer
N12510	Whether the arc speed limit is valid	Select "Yes" to limit the speed during arc machining; Select "No" to not limit the line during arc machining.	Default to "Yes", range (Yes, No)	Manufacturer
N12511	Allowable chord height error during arc machining	During the machining, the points on the curve are calculated and finally fitted to determine the error value of the chord height	Default to 0.010mm, range (0, 0.1)	Manufacturer

		calculated between points.		
N1251 2	Reference circle radius	Reference for machining circular workpieces by machine tools when referring to circles	Default to 5mm	Manufacturer
N1251 3	Reference circle speed	Reference speed when performing arc machining	Default to 3000.000mm/min	Manufacturer
N1251 4	Z-direction slow deceleration speed	During the machining, the speed at which each contact occurs to protect the workpiece.	Default to 300mm/min	Manufacturer
N1251 5	Corner tolerance	For the overall smoothness of the workpiece, there is a slight accuracy error between every two segments of the program, and for the error within the set value range, the program can be operated.	Default to 0.020mm, range (0, 0.1)	Manufacturer
N1251 6	Smoothing time	The larger the parameter is, the smoother the machining surface will be. However, too large a	Default to 0.020 second, range (0, 0.06)	Manufacturer



		parameter can easily cause significant changes in the workpiece size. It is recommended to set the high-precision of molds to 0.01 and woodworking machines to 0.03		
1.3 Pause				
N13000	Action reference selection during pause	Whether to lift to a safe height during pause, 0: Hold still 1: Lift to a safe height 2: Lift to the set tool lifting height	Default to 1, range (0, 2)	User/manufacturer
N13001	Z-axis tool lifting amount during pause	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during pause.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N13002	Tool lifting method when stopping	Action of the machine tool when machining stops.	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2: Lift to the reference point)	User/manufacturer
N13003	Z-axis tool lifting when stop	Parameter set for the use of tool lifting when the tool is lifted according to	Default to 10.000mm, range (1, 1000)	User/manufacturer

			the tool lifting amount during stop.		
N14500	Floating feeler block thickness		Set the thickness of the floating feeler block, and automatically compensate for the thickness value when performing floating tool setting	Default to 0.000mm, range (0, 500)	Manufacturer
N14501	Whether the floating tool setting is valid		Set to "Yes". The floating tool setting on the interface is operable and executable. Set to "No". The floating tool setting on the interface cannot be operable and executable.	Default to "No", range (Yes, No)	Manufacturer
N14502	Fixed tool setting gauge position	X axis	Set the fixed tool setting position and X-axis value	Default to 0.000mm	User/manufacture r
N14503		Y axis	Set the fixed tool setting position and Y-axis value	Default to 0.000mm	User/manufacture r
N14504		Z axis	Set the fixed tool setting position and Z-axis value	Default to 0.000mm	User/manufacture r
N14505	Whether the fixed tool setting is valid		Set to "No". The fixed tool setting on the interface cannot be	Default to "No", range (Yes, No)	User/manufacture r

		operable and executable. Set to "No". The fixed tool setting on the interface cannot be operable and executable.		
N14506	Fixed tool setting gauge thickness	Set the height of the fixed tool setting gauge relative to the table	Default to 0.000mm, range (0, 1000)	User/manufacturer
N14507	Tool setting speed	Speed at which the machine tool moves from the set fixed tool setting gauge position to the tool setting gauge surface.	Default to 60mm/min, range (0, 1000)	User/manufacturer
N14508	Fast fixed tool setting speed	Speed at which the Z-axis of the machine tool moves to the Z position set by the fixed tool setting gauge.	Default to 300mm/min, range (0, Z-axis cutting speed)	User/manufacturer
N15000	Whether the arc IJK increment is valid	Legal value, "Yes": Valid; "No": Invalid	Default to "Yes", range (Yes, No)	
Others				
N17000	Whether the limit is detected during simulation	Select "Yes" to consider whether the file exceeds the software limit during the simulation. If it exceeds the limit, stop the	Default to "No", range (Yes, No)	User/manufacturer

		simulation and report an error; Select "No" to not consider the file machining range during the simulation.		
--	--	---	--	--

### Feed shaft parameters

No.	Parameter name		Meaning	Setting range	Permission
N21500	Impulse equivalent	X axis	Distance of machine tool X-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21501		Y axis	Distance of machine tool Y-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21502		Z axis	Distance of machine tool Z-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21503	Whether the table travel range check is valid	X axis	Set to "Yes". The table travel on the X-axis is valid. Set to "No". The table travel on the X-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer
N21504		Y axis	Set to "Yes". The table travel on the Y-axis is valid. Set to "No". The table travel on the Y-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer
N21505		Z axis	Set to "Yes". The table travel on the Z-axis is valid. Set to "No". The table travel on the Z-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer

N21506	Lower limit of table travel (mechanical coordinates)	X axis	Minimum value for X-axis table operation.	Default to 0.000mm	Manufacturer
N21507		Y axis	Minimum value for Y-axis table operation.	Default to 0.000mm	Manufacturer
N21508		Z axis	Minimum value for Z-axis table operation.	Default to -100.000mm	Manufacturer
N21509	Upper limit of table travel (mechanical coordinates)	X axis	Maximum value for X-axis table operation.	Default to 2500.000mm	Manufacturer
N21510		Y axis	Maximum value for Y-axis table operation.	Default to 2500.000mm	Manufacturer
N21511		Z axis	Maximum value for Z-axis table operation.	Default to 0.000mm	Manufacturer

#### Spindle parameters

No.	Parameter name	Meaning	Setting range	Permission
N31000	Use the default spindle speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N31001	Stall during pause	Set to "Yes". The spindle stops rotating when encountering a pause command; Set to "No". The spindle does not stop rotating when encountering a pause command.	Default to "Yes", range (Yes, No)	User/manufacturer
N31002	Stall during stopping	Set to "Yes". The spindle stops rotating when encountering a stop command; Set to "No". The spindle does not stop rotating when encountering a stop command.	Default to "Yes", range (Yes, No)	User/manufacturer

N31500	Maximum speed	Set the maximum speed corresponding to the spindle at 10V analog output	Default to 24000.000rpm, range (0, 100000)	Manufacturer
N31501	Default speed	Actual speed of the spindle	Default to 24000.000rpm, range (0, maximum speed)	Manufacturer
N31502	Spindle start delay	Time from spindle startup to maximum spindle speed	Default to 10.000 seconds, range (0.5, 300)	Manufacturer
N31503	Spindle stop delay	Time from maximum spindle speed to spindle stop	Default to 5.000 seconds, range (1, 300)	Manufacturer

### Origin parameters

No.	Parameter name	Meaning	Setting range	Permission	
N41000	Before machining, it is necessary to return to the mechanical origin first	Set to "Yes". The machine must return to the mechanical origin before machining. If not executed, the system will prompt an error during machining. Set to "No". Regardless of whether to perform a return to the mechanical origin, the system can perform machining.	Default to "No", range (Yes, No)	User/manufacturer	
N41503	Coarse positioning stage direction	X axis	Set to 1. The X-axis quickly searches for the mechanical origin direction as	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			the positive direction; Set to -1. The X-axis quickly searches for the mechanical origin direction as the negative direction.		
N41504		Y axis	Set to 1. The Y-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Y-axis quickly searches for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41505		Z axis	Set to 1. The Z-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Z-axis quickly searches for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer

N41506	Coarse positioning stage speed	X axis	Operating speed of the X-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41507		Y axis	Operating speed of the Y-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41508		Z axis	Operating speed of the Z-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41509	Fine positioning stage direction	X axis	Set to 1. The X-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The X-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41510		Y axis	Set to 1. The Y-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Y-axis searches again for the	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer



			mechanical origin direction as the negative direction.		
N41511		Z axis	Set to 1. The Z-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Z-axis searches again for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41512	Fine positioning stage speed	X axis	Operating speed of the X-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41513		Y axis	Operating speed of the Y-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41514		Z axis	Operating speed of the Z-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41515	Retrace distance	X axis	Distance to retreat after the X-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer

N41516		Y axis	Distance to retreat after the Y-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41517		Z axis	Distance to retreat after the Z-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41518	Permissible movement direction during origin limit	X axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41519		Y axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41520		Z axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			positive direction and -1 as the negative direction.		
--	--	--	--	--	--

**Compensation parameters**

No.	Parameter name		Meaning	Setting range	Permission
N51500	Effective compensation for screw error		This parameter can be set to determine whether to perform screw error compensation, including reverse clearance compensation and pitch compensation	Default to "No", range (Yes, No)	Manufacturer
N51501	The reverse clearance compensation is valid		Set the reverse clearance of each axis screw	Default to "No", range (Yes, No)	Manufacturer
N51502	The tool compensation is valid		Set to "No". The tool compensation parameters are invalid. Set to "Yes". The tool compensation is valid.	Default to "No", range (Yes, No)	Manufacturer
N51503	Reverse clearance	X axis	The clearance value in the X-axis direction will be compensated multiple times if it exceeds 0.2	Default to 0.000mm, range (0, 1)	Manufacturer
N51504		Y axis	The clearance value in the Y-axis direction will be compensated multiple times if it exceeds 0.2	Default to 0.000mm, range (0, 1)	Manufacturer
N51505		Z axis	The clearance value in the Z-axis direction	Default to 0.000mm, range (0, 1)	Manufacturer

			will be compensated multiple times if it exceeds 0.2		
N71002	Handwheel	Handwheel acceleration	The smaller the value is, the smoother the speed will be	Default to 200.000m, range (1, 6000)	Manufacturer
N71006		Strict handwheel pulse calculation	If strict handwheel counting is used, the system will move the distance specified; otherwise, the machine tool will only move when the handwheel is shaking	Default to 1, range (0, 1)	Manufacturer
N71007		Handwheel direction	Set the handwheel to control the moving direction, -1: Negative; 1: Positive	Default to 1, range (-1, 1)	Manufacturer

---

## Chapter XIV Miscellaneous Function

The [Auxiliary] menu contains items related to non auxiliary functions. Click on the [Auxiliary] menu with the mouse to pop up the drop-down menu:

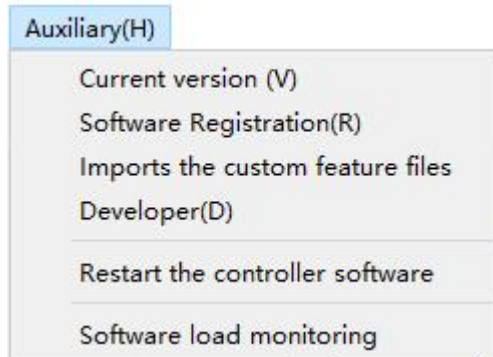


Figure 14-1 "Auxiliary" Menu on the Menu Bar

### 14.1 Current version

Select the menu item [Current Version] in the [Auxiliary] menu to provide relevant information such as the current system software version

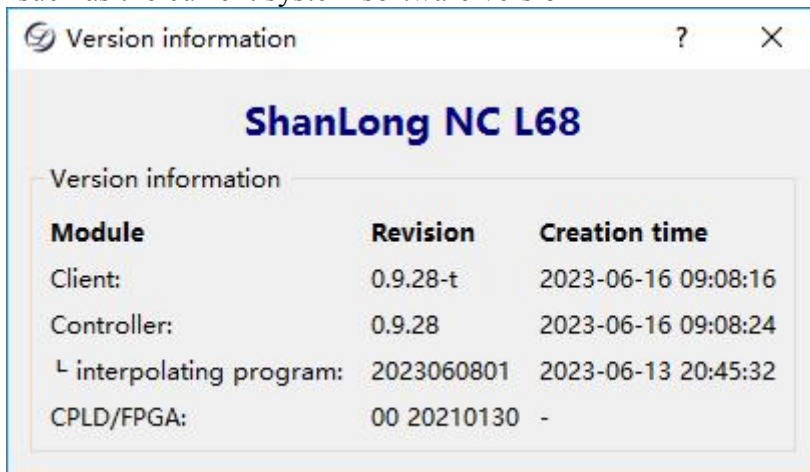


Figure 14-1 Version Information

### 14.2 Restart of controller software

Select the menu item [Restart Controller Software] in the [Auxiliary] menu to execute a client restart.

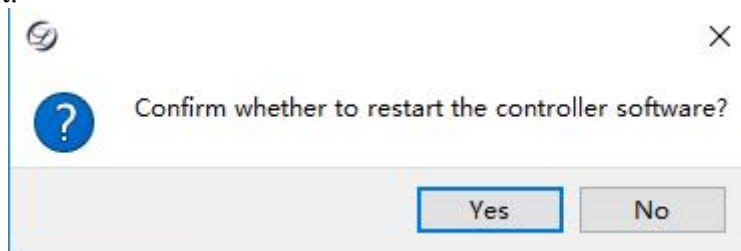


Figure 14-5 Restart Controller Software

Select the menu item [Restart Controller System] in the [Auxiliary] menu to execute the controller restart.

### 14.3 Program load monitoring

Select the menu item [Program Load Monitoring] in the [Auxiliary] menu to provide relevant information about the current system communication cycle

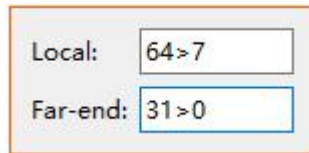


Figure 14-7 Program Load Monitoring

### Chapter XV IO Status

There are 27 input IO channels and 17 output IO channels, which can be freely set; Master control IO:1-8 and expansion board IO:33-48 can be used once configured. If the IO port is configured as "0", it is not configured (not used).

The IO status window displays the input/output port description, port number, polarity, and current IO status. When IO is turned on, the IO turns red in the IO status; When IO is turned off, the IO turns green in the IO status. Users can configure ports and view the IO status window for input signals or signal outputs.

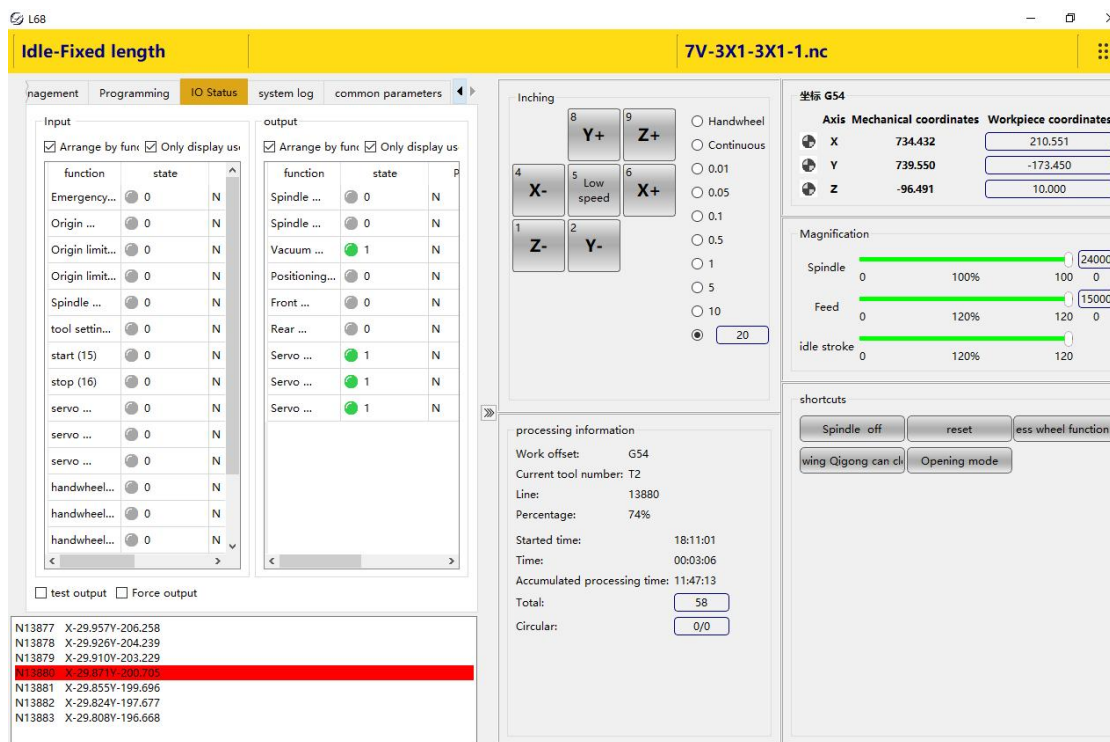


Figure 15-1 IO Status Window

### 15.1 Output IO test

Users can click on [Start Test] at the bottom of the interface to test the output port IO for signal output. Select the output port and click on the "Test Switch" to perform the test. Click "Cancel All" to cancel the testing of the selected port. After the test is completed, click "Exit Test" to exit the testing status.

### 15.2 Input/output IO configuration

Select an input/output port and double-click it or click [Set Port Number] under the main interface, enter the port number and click "OK". Click "OK" to pop up a dialog box, select "Yes" and restart the system to complete the port configuration. To modify the port number or reverse polarity for the first time, users need to enter the manufacturer password: "666666".

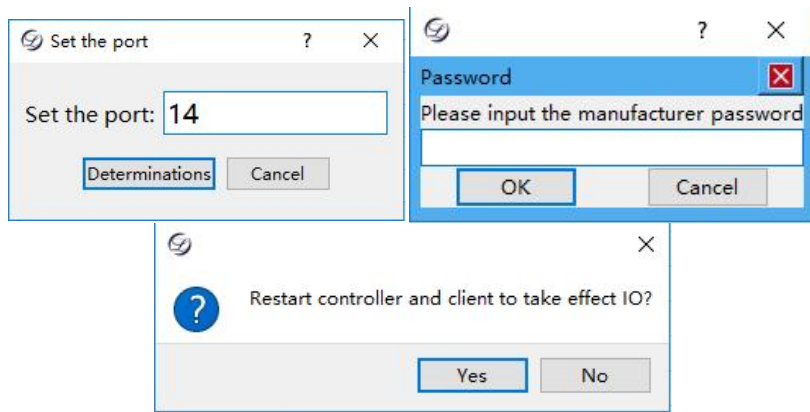


Figure 15-2 IO Configuration

Figure 15-3 Default Input IO

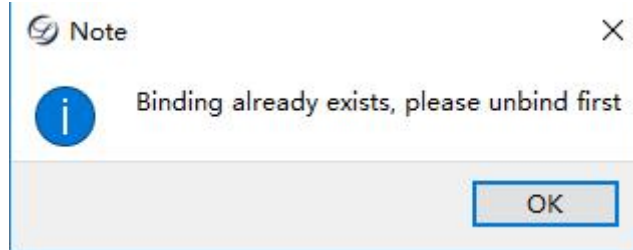
Polarity	Port	Description	Polarity	Port	Description
N	2	Emergency stop	N	4	Stop
N	16	X-axis origin limit	N	65	X-axis servo alarm
N	13	Y-axis origin limit	N	66	Y-axis servo alarm
N	10	Z-axis origin limit	N	67	Z-axis servo alarm
N	15	X-axis positive limit	N	79	Handwheel input X-axis
N	12	Y-axis positive limit	N	80	Handwheel input Y-axis
N	9	Z-axis positive limit	N	81	Handwheel input Z-axis
N	14	X-axis negative limit	N	85	Handwheel input X1
N	11	Y-axis negative limit	N	86	Handwheel input X10
N	8	Z-axis negative limit	N	87	Handwheel input X100
N	77	Spindle alarm	N	71	X-axis z-signal
N	1	Tool setting	N	72	Y-axis z-signal
N	3	Start machining	N	73	Z-axis z-signal

Figure 15-3 Default Output IO

Polarity	Port	Description	Polarity	Port	Description
N	5	Band-type brake	N	79	Spindle speed control
N	4	Red light	N	6	Expansion output 2
N	3	Green light	N	78	Spindle enable _ reverse
N	2	Cooling	N	65	X-axis servo enable
N	1	Automatic machining completed	N	66	Y-axis servo enable
N	77	Spindle enable	N	67	Z-axis servo enable
N	8	Yellow light	N	71	X-axis servo alarm cleared
N	7	Expansion	N	72	Y-axis servo alarm

		output 1			cleared
			N	73	Z-axis servo alarm cleared

If duplicate port numbers are configured, a prompt will appear as shown in the following figure:



### 15.3 Port polarity modification

Select an input/output port, click [Reverse Polarity] to pop up a dialog box, and click "OK" to pop up another dialog box. Select "Yes", restart the system, and complete the modification of port polarity.

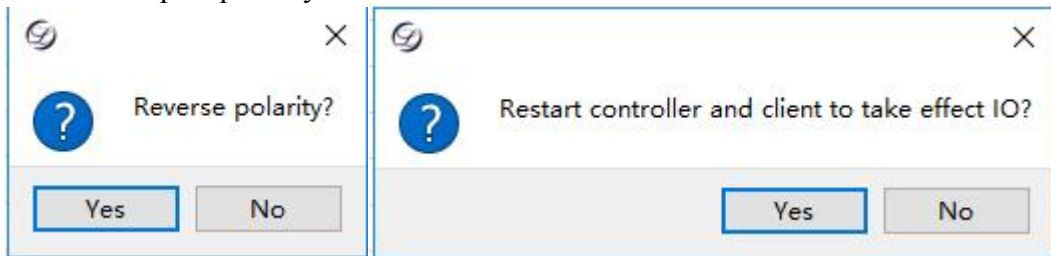


Figure 15-3 Modify Port Polarity