Configuring Logosol CNC

In order to provide reliable operation setting up the software should be done by experienced personnel with detailed knowledge of the machine.

Logosol CNC is a CNC software package that allows users to load and execute G code files on CNC machines controlled by Logosol multi axis servo controllers. It supports integrated or external spindle drives, supervisor safety I/O controller, and general purpose inputs and outputs nodes. The software can be easily customized for specific machine configurations with up to 6 axes.

Logosol CNC consists of a system executable implementing a G code interpreter and user interface, motion driver and I/O driver, designed as separate DLL files. These files are located in the C:\MCTL folder:

Mctl_Logosol1.dll is a motion driver supporting up to 9 servo axes.

IO_Logosol1.dll and **IO_Logosol2.dll** are IO drivers each supporting a single Logosol I/O controller or safety module. Depending on the IO configuration, one or both can be used.

LS_Serial.dll is a communication driver providing the link between all of the drivers mentioned above and the actual hardware.

The following sequence should be followed in order to activate the advanced functions.
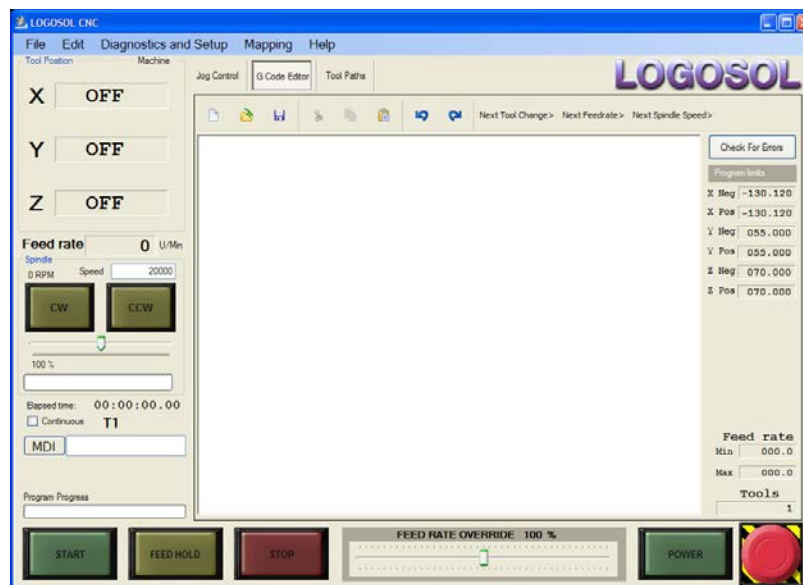
- Start the software by clicking the icon on the desktop. The following dialog box will appear on the screen.
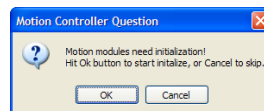
- Click the Cancel button. On the main menu, click File and select Login.
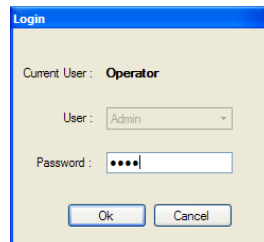
- Type the password "1234" and click OK. This will enable Diagnostics and Setup and Mapping menu items.

Setting up the machine limits and user units

Click Diagnostics and Setup on the main menu and select Machine. The Machine Setup dialog box will be displayed:

Enter the negative and positive machine travel limits for each axis. These limits are applied to the data entered in G code program and are measured from machine zero point. For linear axes specify the limits in the selected user units (inches or millimeters).

From the **User units** drop down box select the desired linear units – inches or millimeters. Enter the desired speeds for jogging linear axes in the specified units per second. The rotary axes speed should be specified in angular degrees per second.

Part approach speed parameter is used to determine the speed approach the detail after M6 TX command end.

Safe Z distance to approach the part is the safe distance to the part from where Z axis approaches the part with part approach speed.
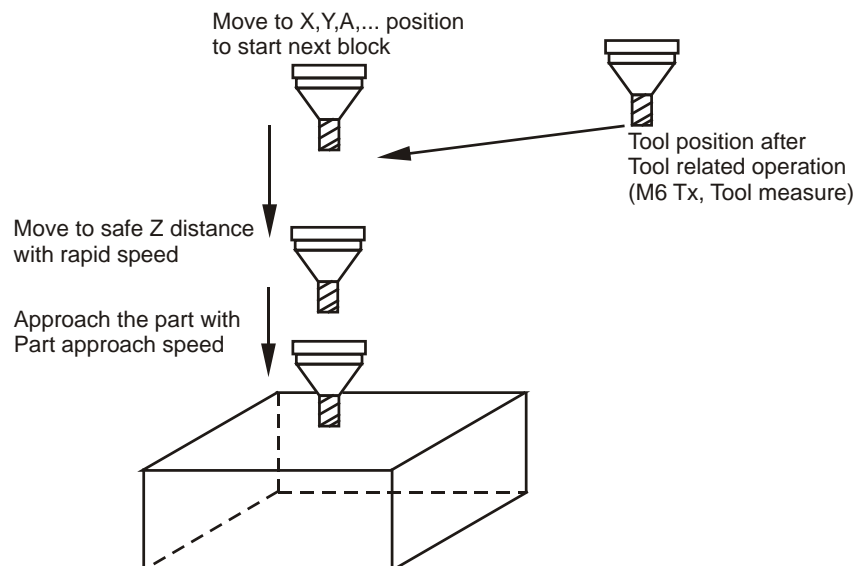
Part approach speed and Safe Z distance to approach the part

Click OK when done.

Spindle settings

If the spindle is controlled by the software, it should be set up as follows.
Click Diagnostics and Setup on the main menu and select Spindle. The Spindle Settings dialog box will be displayed:
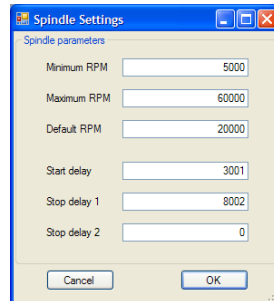
Enter the minimum and maximum spindle speed according to the spindle manual. Set the default spindle speed to a "standard" value according to the machine purpose.
Start delay is the time needed for the spindle to accelerate from 0 to the maximum rpm.

Spindle start sequence
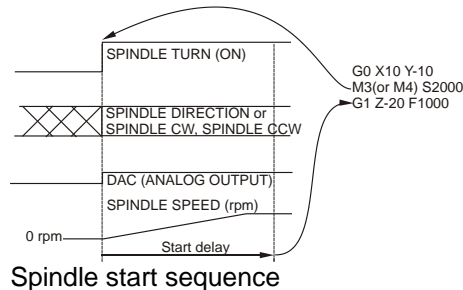
Stop delay 1 is the delay between zero speed command and disabling the spindle.
Stop delay 2 is the delay between disabling spindle output and starting the next command.

Spindle stop sequence
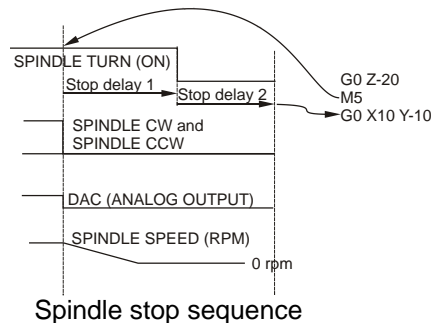
Set **Stop delay 1** and **Stop delay 2** to guarantee complete spindle stop when running at full rpm.
Click OK when done.

Setting up the Motion parameters

Click Diagnostics and Setup on the main menu and select Motion Control. The Motion Controller dialog box will be displayed:
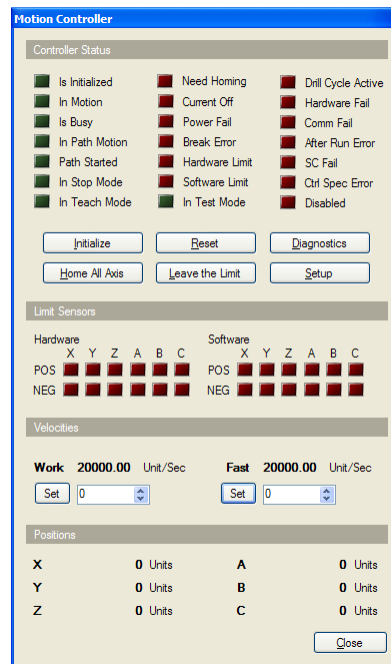
To Setup the motion driver click Setup button.
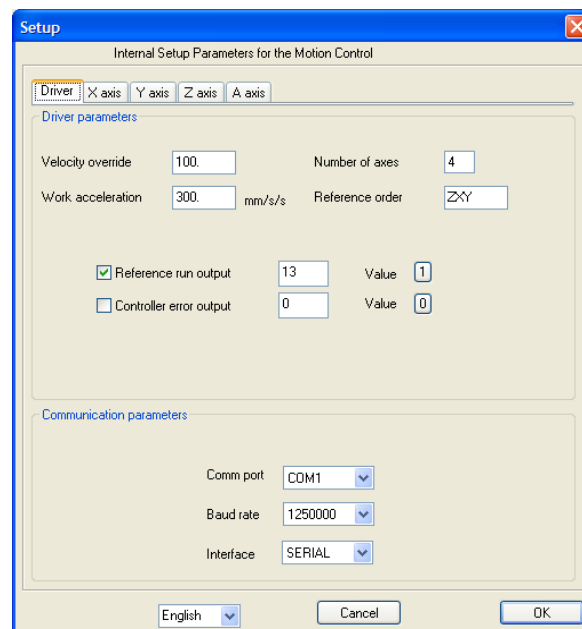
Under Driver parameters enter the following:
- **Velocity override** – default velocity override. Set to Velocity override=100.
- **Work acceleration** in machine units per second.
- **Number of axes.** Total number of axes installed on the machine.
- **Reference order**. According to the picture – Z axis will go to its reference point first, X – second, and Y – third.

- **Reference run output**. For machines controlled by Multi axis CNC servo controller such as CNC-1120/0680, this output should be enabled as output 13 and its active value should be 1 as shown above.
- **Controller error output**. This output, if used, will be turned on if any motion or hardware error happens during a working session of the software. It could turn off the power to some custom external devices or activate an alarm.

Under communication parameters enter the following:
- **Com port** used to communicate to the controllers.
- **Baud rate**. Depends on the type of communication port and on the controller. It is recommended to set this to the highest possible value for better throughput.
- **Interface**. The options are SERIAL, USB, or HS SERIAL.
  SERIAL should be selected if a standard PC communication port is being used, or high-speed PCI (ISA) LDCN plug-in card is used and frequency of 20 MHz is specified in COM port properties in Windows Device Manager[1].
  USB should be selected if an USB to LDCN adapter is used.
  HS SERIAL (high-speed) interface should be selected if a high-speed PCI or ISA LDCN plug-in card is used and LDCN communication port is configured with the standard PC crystal frequency of 1.8432 MHz[1].

[1] Note: For the Windows Device Manager right click on My Computer, select Manage, and click Device Manager.

After setting the Driver control parameters, the parameters for each axis should be set in their respective pages:

Most of these parameters are self explanatory. The servo parameters depend on many factors including the type of servo controllers and motor as well as the machine mechanics. The parameters shown above are example only and might not work well on all systems.

Positive software limit and **Negative software limit**. When the axis position is equal or greater than one of the software limits the axis will decelerate and stop. The motion opposite to the limit will be the only motion allowed at this point.

All motion parameters are stored in C:\MCTL\Mctl_LogosolX.ini. Up to 100 such files (X=0, X=1, … X=99) can be supplied. G code program can switch between parameters sets using M32 Px. By default X=1.

Click OK and Close the Motion Controller dialog when done.

I/O mapping

Click Mapping on the main menu and select Machine. The I/O Setup dialog box will be displayed.

In most cases, only one I/O controller should be configured (the first one on the list to the left). All others should be disabled – the Enable check box should be unchecked. The first I/O controller should be set to use IO_Logosol1.dll driver and IO_logosol1.ini configuration file. Other I/O controllers that may need to be used should be configured with IO_Logosol2.dll / IO_Logosol2.ini and so on. Note that you need to have the actual files. If you do not have some of the files you need, please contact Logosol Inc. support.

For each of the I/O modules up to 16 outputs and 16 inputs could be specified. The description of the Inputs/Outputs should be specified using the drop down boxes for selecting their purpose and/or button/LED number. NONE and Button/LED=0 should be used if the input or output is not used.

Inputs
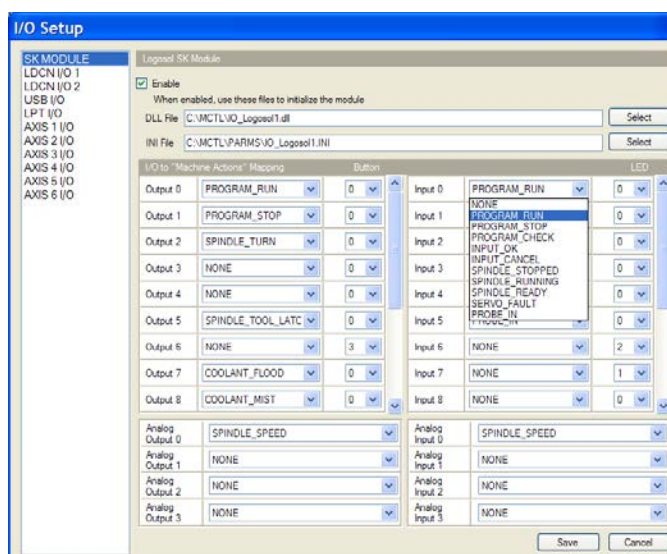
Program execution control:
- PROGRAM_RUN – Program start – same as main screen START;
- PROGRAM_STOP – When program is running – FEED HOLD. When FEED HOLD is active – STOP;
- PROGRAM_CHECK – reserved;
- INPUT_OK – reserved;
- INPUT_CANCEL – reserved.

Spindle status safety control:

During the program execution the spindle status inputs can be used to prevent the cutting when the spindle is not turning because of spindle or other hardware related failure.
- SPINDLE_STOPPED – The input should be "off" when spindle is running. PROGRAM_STOP command will be issued if this input is "on" when Spindle CW/CCW (M3 or M4) command is active. Spindle safety control circuit uses by default Input 2 and if used SPINDLE STOPPED function should be assigned to Input 2;
- SPINDLE_READY – Expected to be "on" during the program execution. The G-code execution program will be stopped when the input is "off" ;
- SPINDLE_RUNNING – When spindle is running "on" indicates that the spindle speed is within the limits of the spindle velocity command (according to the spindle drive settings). If spindle speed does not match to the spindle velocity command the output is turned "off" and the G-code program execution will be stopped.
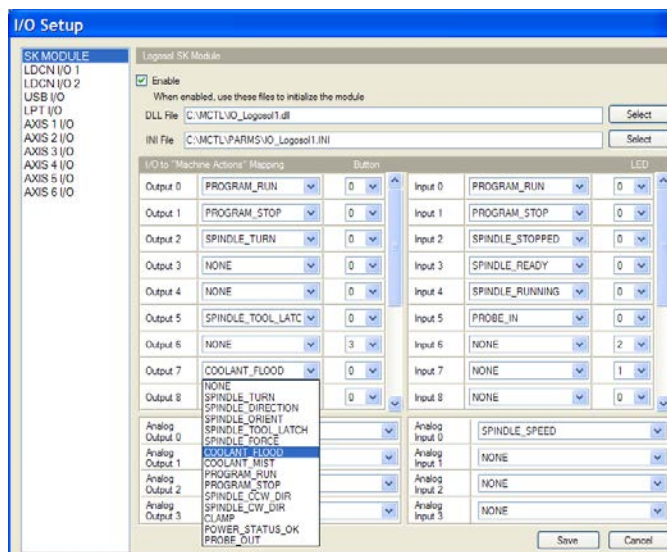
Servo safety control:
- SERVO_FAULT – If used this function should be assigned to Input 10 (connected to servo drives "FAULT" output). If the input is "on" the program execution will stop.

Probe interface:
- PROBE_IN – probe input. The transition to active (on) state causes the axes position to be transferred to the corresponding variables (#5061 to #5066).

Outputs

The following functions used by the CNC program should be assigned to control the outputs if needed.
- SPINDLE_TURN – active when spindle command is active (M3, M4). Cleared by M2, M5 and M30. The use of Output 2 (CNC-SK-2310) is strongly recommended because of the implemented spindle safety control circuit;
- SPINDLE_DIRECTION – on when M3 is active, cleared by M4;
- SPINDLE_ORIENT – reserved;
- SPINDLE_TOOL_LATCH – controlled by custom button. The output is disabled (force to off state) when spindle is running, or during G-code program execution, except during tool change sequence;
- SPINDLE_FORCE – reserved;
- COOLANT_FLOOD – on when M8 is active. Cleared by M9;
- COOLANT_MIST – on when M7 is active. Cleared by M9;
- PROGRAN_RUN – on when program is started;
- PROGRAM STOP – pulse when FEED HOLD is active.
- SPINDLE_CCW_DIR – on when M3 is active, cleared by M4, M5, M2, M30;
- SPINDLE_CW_DIR – on when M4 is active, cleared by M3, M5, M2, M30;
- CLAMP – axis clamp M26-on M27-off;
- POWER_STATUS_OK – on when homing is done and axes are on, off otherwise;
- PROBE_OUT – on during G38.2 execution, off otherwise;

The analog inputs and outputs can be specified the same way. Note that specifying an analog input to be used for feed rate override or spindle speed override will set it up to control the feed rate override or spindle speed override sliders (refer to the screen below). In this case, the sliders cannot be controlled using the mouse.

Customizing Jog Control screen
Selected inputs and outputs could be setup to appear on the Jog screen of Logosol CNC as buttons controlling the state of the outputs and labels showing the state of the inputs. For example, Output 6 (previous page) can be controlled by custom Button 3 on the Jog screen as shown bellow.
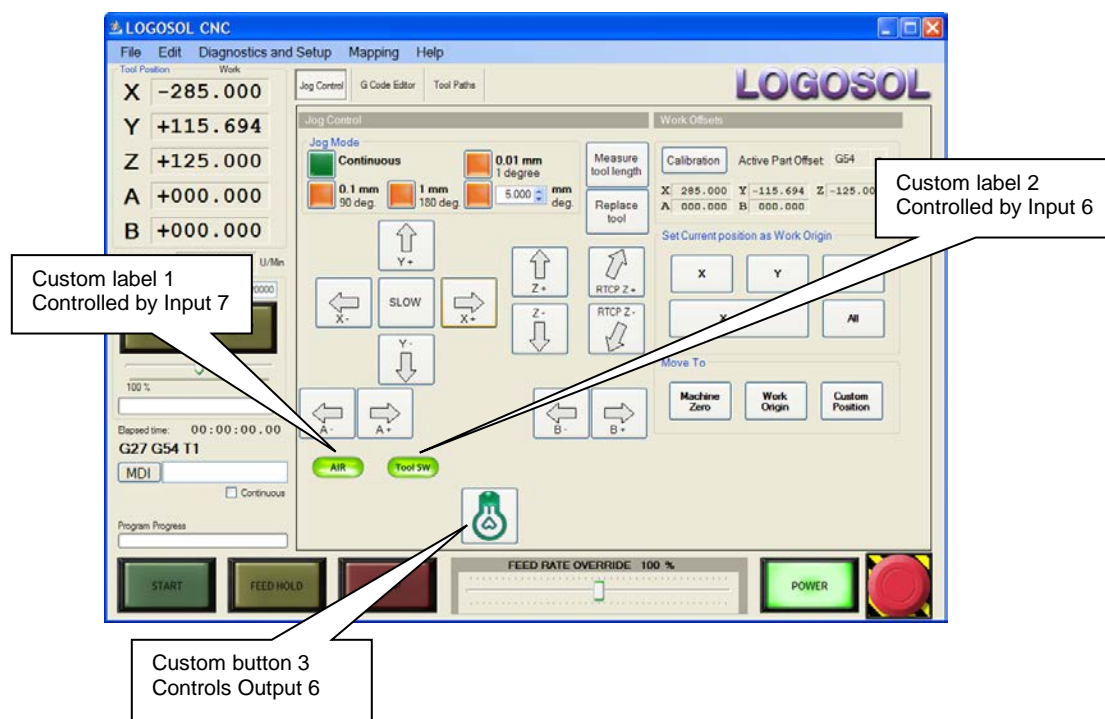
Customer supplied images can be placed on the buttons and labels on this screen. Supported image formats are Portable Network Graphics (PNG) and Bitmaps (BMP) sometimes called Device Independent Bitmap (DIB). All these images should be places in C:\MCTL\BITMAPS folder. Recommended size for the buttons images is 64 by 64 pixels. Recommended size for the labels images is 64 by 32 pixels.

For the jog buttons, the following file names are being used:
JOGXPOS.PNG (JOGXPOS.BMP), JOGXNEG.PNG (JOGXNEG.BMP),
JOGYPOS.PNG (JOGYPOS.BMP), JOGYNEG.PNG (JOGYNEG.BMP),
JOGZPOS.PNG (JOGZPOS.BMP), JOGZNEG.PNG (JOGZNEG.BMP),
JOGAPOS.PNG (JOGAPOS.BMP), JOGANEG.PNG (JOGANEG.BMP),
JOGBPOS.PNG (JOGBPOS.BMP), JOGBNEG.PNG (JOGBNEG.BMP),
JOGCPOS.PNG (JOGCPOS.BMP), JOGCNEG.PNG (JOGCNEG.BMP).

For the Fast speed and Slow speed buttons:
JOGFAST.PNG (JOGFAST.BMP) and JOGSLOW.PNG (JOGSLOW.BMP).
Two images should be supplied for each custom button - one for ON and one for OFF state:
OUTPUTxON.PNG (OUTPUTxON.BMP) and OUTPUTxOFF.PNG (OUTPUTxOFF.BMP), where x is the number of the button and should be between 1 and 8. For example, OUTPUT3ON.PNG and OUTPUT3OFF.PNG could be supplied for custom button 3.
Two images should be supplied for each custom label - one for ON and one for OFF state:
INPUTxON.PNG (INPUTxON.BMP) and INPUTxOFF.PNG (INPUTxOFF.BMP), where x is the number of the label and should be between 1 and 8.

I/O setup

Click Diagnostics and Setup on the main menu and select Inputs/Outputs. The I/O Diagnostics dialog box will be displayed:
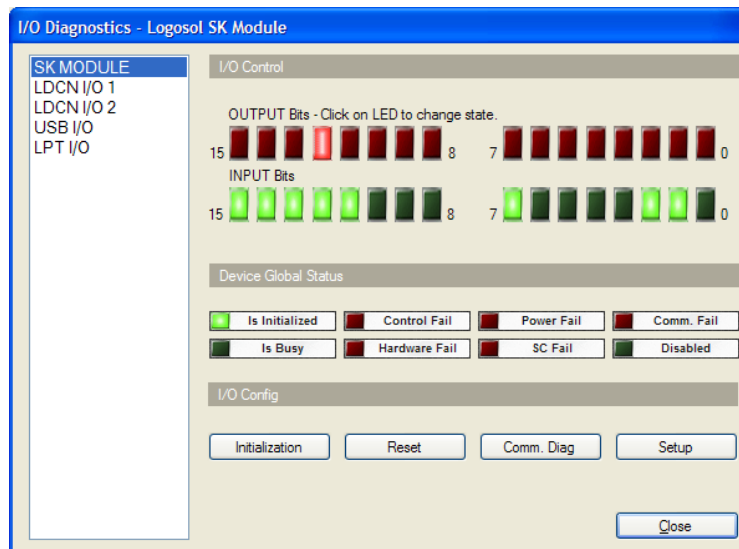
Click the Setup button. The Setup dialog of the motion driver will be displayed.
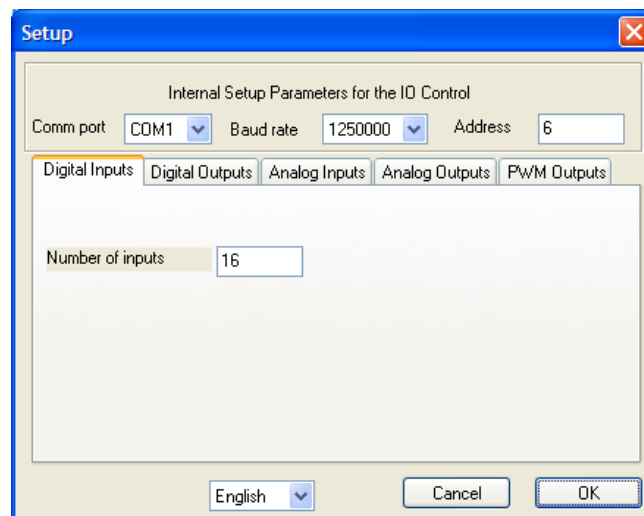
The communication parameters should not be changed here since they are already set in the motion driver setup. The address of the IO controller should be set accordingly to the order of all servo and I/O controllers connected to the PC.

In Digital Outputs page, the number of outputs of the IO controller should be set as well as their desired power-up state.

In Analog Inputs page, the analog inputs of the I/O controller should be enabled and their range of user values should be specified. For example, if an analog input is used for measuring the actual spindle speed, its range should be from 0 to the maximum spindle speed.
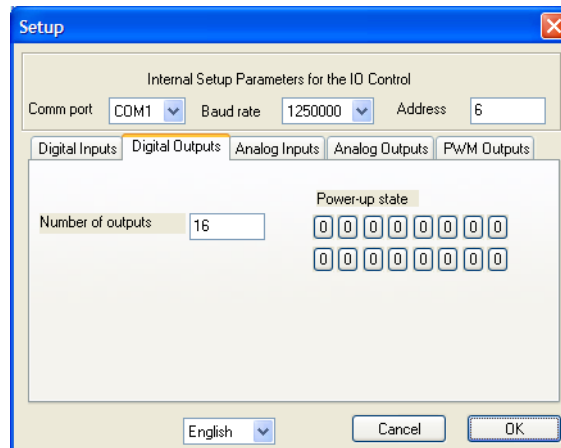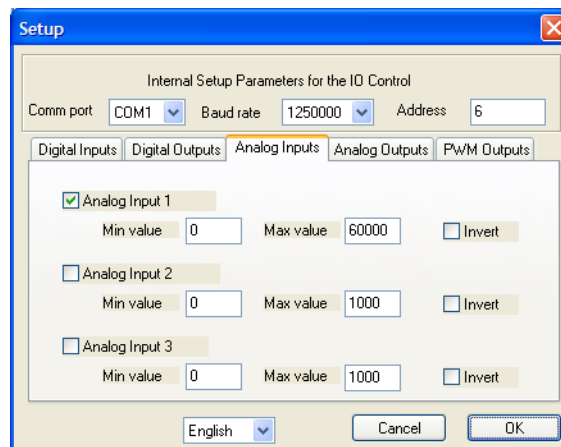
In Analog Outputs page, the analog outputs of the I/O controller should be enabled and their range should be specified. For example, if an analog output is used for controlling the spindle speed, its range should be from 0 to the maximum spindle speed.
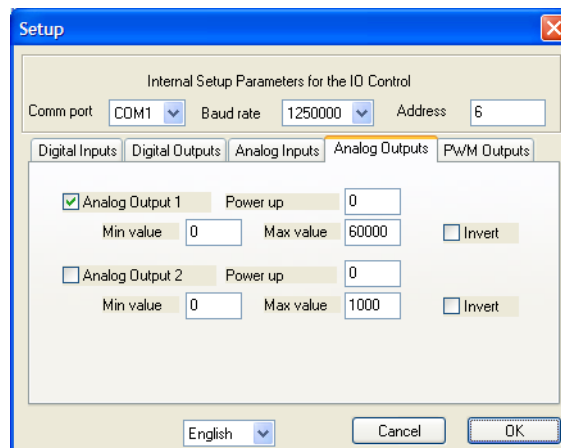
In PWM Outputs page (similar to Analog outputs page), the PWM outputs of the I/O controller should be enabled and their range should be specified.
Click OK on the Setup dialog and Close on the I/O Diagnostics dialog when done.

User defined procedures

User defined procedures are used to control the machine interface. All procedures must be included in USERPROC.SETUP file located in C:\MCTL\PARAMS directory.

Each procedure could be executed in one of the following modes:
- *<Proc_Mode>0</Proc_Mode>* – time interval mode. The procedure is executed based on time interval parameter (*<Interval>240000</Interval>)*;
- *<Proc_Mode>1</Proc_Mode>* – the procedure is executed every time when the G-code file is started;
- *<Proc_Mode>2</Proc_Mode>* – the procedure is executed after the G-code program successful end ;
- *<Proc_Mode>3</Proc_Mode>* – the procedure is executed when error occurs or G-code program is stopped during execution.

Example:
```
<?xml version="1.0" encoding="utf-8"?>
<USER_PROCSxmlns:xsi="http://www.w3.org/2001/XMLSchema-instance" xmlns:xsd="http://www.w3.org/2001/XMLSchema">
 <UserProcs>4</UserProcs>
 <Procs>
  <USER_PROC>
   <Proc_ID>0</Proc_ID>                        ; Procedure ID
   <Proc_Mode>0</Proc_Mode>                    ; Mode – time interval
   <Proc_Name>Name</Proc_Name>                 ; Procedure name
   <Enabled>true</Enabled>                     : Enabled
   <Interval>240000</Interval>                 ; Time interval in ms (260000=240sec=4min)
   <StopOnError>false</StopOnError>            ; False – the procedure will continue when error is detected
   <CanStopGCode>true</CanStopGCode>           ; True – the procedure will stop G-code execution if error occur
   <ErrMessage>Error</ErrMessage>             ; Procedure error message – displayed when error occur
   <Body>                                      ; Procedure body
     <string>OUT 4 1</string>
     <string>WAITIN 4 1 15000</string>        ; Generate an error when INPUT 4 remains 0 after 15 sec
     <string>OUT 4 0</string>
   </Body>
  </USER_PROC>
  <USER_PROC>
   <Proc_ID>3</Proc_ID>
   <Proc_Mode>3</Proc_Mode>                    ; Mode – This procedure is executed when error occurs or STOP
                                                 is pressed during G-code execution
   <Proc_Name>Name</Proc_Name>
   <Enabled>true</Enabled>
   <Interval>60000</Interval>                  ; Ignored but should be included
   <StopOnError>false</StopOnError>
   <CanStopGCode>false</CanStopGCode>
   <ErrMessage>Error</ErrMessage>
   <Body>
     <string>OUT 6 1</string>
   </Body>
  </USER_PROC>
 </Procs>
</USER_PROCS>
```
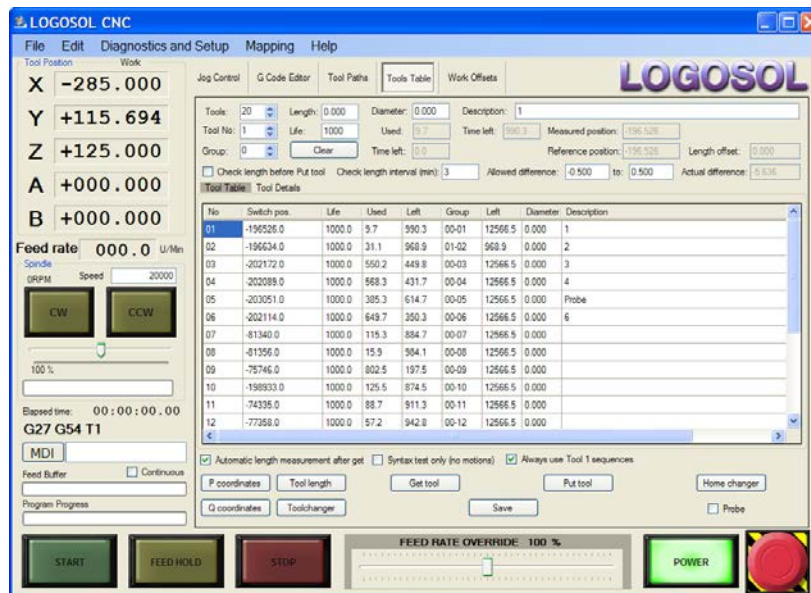
The procedures are executed in order they appear in the USERPROC.SETUP file. The procedures are executed only one at a time. The active procedure must be completed before starting the next.

Tools control

Open **Tools Table** to setup tools parameters and toolchanger control procedures.

Tool Life management

Each tool has **Life** and **Group** parameters assigned. The **Life** parameter is the time limit in minutes for active tool use (tool is selected and spindle is turning). The tools belonging to the same group are considered identical. Selected tool will be replaced automatically with tool from the same group when **Time Left** for first tool is 0. Contact the machine and tools manufacturers to set the proper **Life** value.

Tool length compensation

The following options are available:
- **None** – all compensations are turned off.
- **Standard** option will apply the tool length compensation only to Z axis.
- **RTCP** (**Rotating Tool Center Point**) could be used when the tool (spindle) is attached to one of the rotary axes. According to the angle position of the tool the length difference will be transformed into vertical and horizontal axis offset.
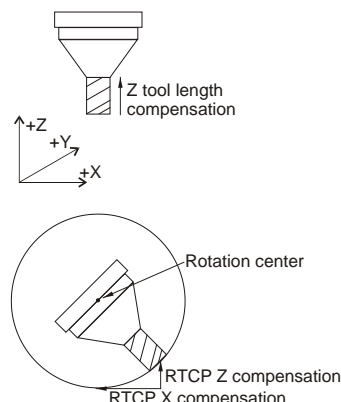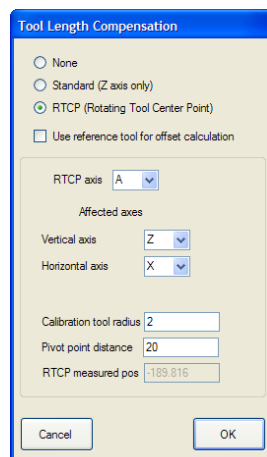
RTCP Tool length compensation

Select the axes according to the machine design. One of the tools is used to measure the RTCP position and to calculate the rotation radius. This data are used by the software to calculate the vertical and horizontal offsets during the machine operation. To measure the RTCP position – go to **Tools Table / Tool details** and run **RTCP calibration** sequence. The data for RTCP will be calculated automatically after the calibration. For specific machine designs the **Pivot point distance** could be entered. The value entered is used during the calculation of the actual pivot point distance and depends on the machine design.

Fixed pockets Toolchanger setup
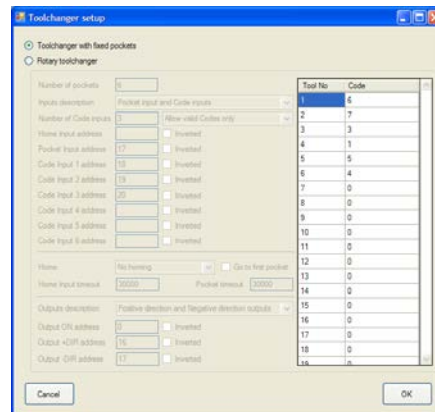
Toolchanger with fixed pockets should be selected when each tool pocket has fixed coordinates (position) or the tools are changed manually. All operations are performed using set of specific toolchanger I/O and axes movements commands. The tool change sequences should be entered in "Tools Table" screen. This setting should be used also when the tools are changed manually.
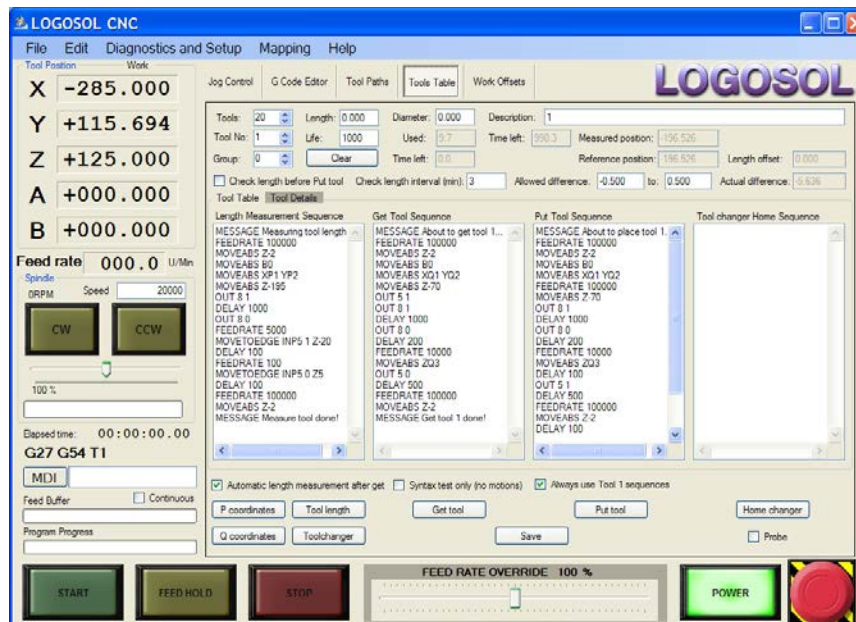
Logosol CNC supports up to 32 tools. For each tool one **Get Tool Sequence** and one **Put Tool Sequence** should be defined. There is an option for use of only T1 Put and Get tool sequences if **Always use Tool 1 sequences** is checked. If there is tool measurement hardware one **Length Measurement Sequence** should be defined. This sequence will be used for length measurement of all tools.

One of the tools could be assigned as a **Probe tool** (**Probe** should be checked). **Probe tool** is used for calibration. Refer to G38.2 description in Logosol CNC Operators manual and the machine manufacturer description. Probe tool has its own control sequences displayed when the **Probe tool** is selected (Tool No 5 below).
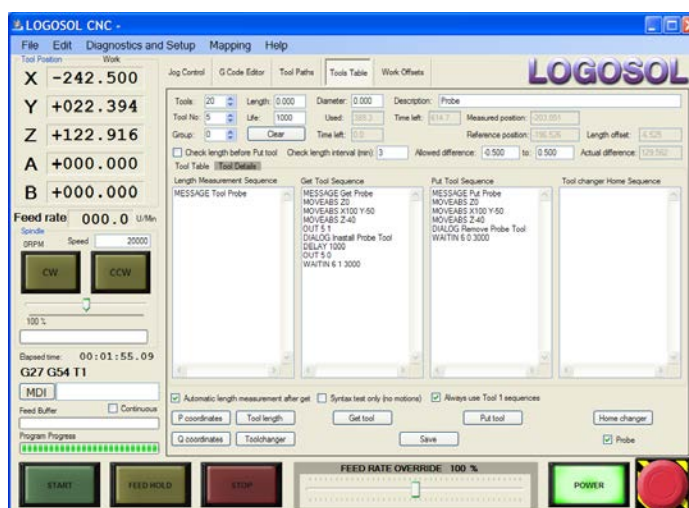
The **Tools Sequences** uses separate set of commands. Supported commands are:

MOVEABS – move one or more axes to the target machine position. This command accepts any number of pairs of axis name (X, Y, Z, A, B, or C) and a target position in machine coordinates. For example, **MOVEABS Z 10.5** will move the Z axis to machine position 10.5.

MOVETOEDGE –
- In **Length Measurement Sequence** - move axis Z a specified distance and stop the motion immediately if a specified input has a specified value. The main purpose is for measuring the tool length using a dedicated switch, which activates when touched with the tool. For example, **MOVETOEDGE INP 5 1 Z-10.0** will start relative (-10.0) motion of axis Z and will stop this motion immediately if input 5 becomes 1. If the input does not activate during the motion the whole sequence will be terminated.
- In **Put Tool Sequence** - move axis Z a specified distance and stop the motion immediately if a specified input has a specified value. The main purpose is to check if the tool is still in place. For example, **MOVETOEDGE INP 5 1 Z-10.0** will start relative (-10.0) motion of axis Z and if the value of input 5 is 0, **Put Tool Sequence** will continue without interruption. If input 5 becomes 1 during the motion the will stop the motion immediately and the sequence will be terminated.

FEEDRATE – set the feed rate for the next motion commands. The units are 1/1000 of the user units per second. For example, **FEEDRATE 10000** will set the speed for the next motions to 10 mm/sec or 600 mm/min.

OUT – set or clear a digital output. This command can be used to open or close the tool changer cover or the clamp. For example, **OUT 4 1** will activate output 4 and if output 4 controls the tool changer cover, it will open the cover.

WAITIN – wait for specified input to activate. This command can be used to wait until a sensor activates. For example, **WAITIN 4 1 2000** will wait until input 4 becomes 1 or the 2000 milliseconds timeout expires. If the timeout expires, the sequence will terminate. Otherwise, the next command in the sequence will be executed.

MESSAGE – display a message for the operator in manual mode and wait for input from the operator. For example, **MESSAGE Get Tool completed** will display on the screen a message box with the text "Get Tool completed" and will wait for the operator to click OK. If the operator clicks Cancel, the whole sequence will be terminated. Note that if the get/put tool sequence is activated by an **M** command from the G code file, there will be no message on the screen and the next command in the sequence will be executed without delay.

DIALOG - display a message for the operator. Similar as **MESSAGE** command but displays the dialog box even if activated from a G code file. For example, **DIALOG Please place tool 2** will display the text "Please place tool 2" on the screen and will wait for the operator to change the tool and click OK. Clicking Cancel will terminate the whole sequence.

DELAY – wait specified amount of milliseconds. For example **DELAY 2000** will delay execution of the next command with 2 seconds.

SPINDLE – enables the spindle with specified analog spindle command. **SPINDLE CW 50** (**SPINDLE CCW 50**) will enable and rotate the spindle CW (CCW) direction with analog output value equivalent to 50 rpm. **SPINDLE STOP** or **SPINDLE OFF** will stop and disable the spindle. This command can be used to rotate spindle when spindle orientation is necessary during the "Put Tool" and "Get Tool" sequences.
Before creating the sequences for put tool, get tool, and measure the tool length, make sure all tool positions are known as well as the inputs and outputs numbers and states that control the hardware related to any of the tool change operations.

Activate the Tools Table page and select tool number 1 or any other tool. Create the put tool sequence in the designated edit box.

> *MESSAGE Put tool*
> *FEEDRATE 10000*
> *MOVEABS Z0*
> *MOVEABS X50.0 Y20.0*
> *FEEDRATE 1000*
> *MOVEABS Z-10.0*
> *OUT 5 1*
> *DELAY 1500*
> *WAITIN 5 1 1000*
> *FEEDRATE 10000*
> *MOVEABS Z0*
> *OUT 5 0*
> *MESSAGE Put tool done!*

In the sequence above, it is assumed that activating output 5 releases the tool, input 5 detects the tool, and the machine coordinates of the tool holder are X50, Y20. Then click "Save" button. Check the sequence for syntax errors by checking "Syntax test only (no motions)" checkbox and click "Put tool" button. The software will display several dialog boxes on the screen with information about any existing errors. After fixing any problems, make sure the selected place is empty and perform a live test of the sequence without a tool. Correct any positions if needed and after everything is OK, test the sequence with a tool.
Create and test your "Get tool Sequence" and "Length Measurement Sequence" sequences step by step as described above. Make sure you click "Save" button each time you finish or change a sequence and before testing it.
After finishing with the first tool, create the "Get Tool" and "Put Tool" sequences for all other tools in the machine. You can copy an existing sequence, change the tool number and paste it for the new tool. Then change only the X, Y coordinates of the new tool and check if each sequence works correctly.
After finishing with these tests, test changing the tool using the M6Tx command in the G code editor.

Note: For some CNC machines the sequences for different tools are similar. Replacing the values used with data from Q Coordinates and P Coordinates tables allows Put tool/Get Tool sequences to be optimized.

P Coordinates

The data written in this table can be used as coordinate's value in tool changer sequences.

Example:

User Defined Coordinates

P variables   P10

	1	2	3	4	5	6	7	8	9	10
P1	10.0	20.0	30.0	0	0	40.0	50	0	0	60.0
P11	-10.0	70.0	0	0	0	0	0	0	0	0
P21	0	0	0	0	0	0	0	0	0	0
P31	0	0	0	0	0	0	0	0	0	0
P41	0	0	0	0	0	0	0	0	0	0
P51	0	0	0	0	0	0	0	0	0	0
P61	0	0	0	0	0	0	0	0	0	0
P71	0	0	0	0	0	0	0	0	0	0
P81	0	0	0	0	0	0	0	0	0	0
P91	0	0	0	0	0	0	0	0	0	0
P101	0	0	0	0	0	0	0	0	0	0
P111	0	0	0	0	0	0	0	0	0	0
P121	0	0	0	0	0	0	0	0	0	0
P131	0	0	0	0	0	0	0	0	0	0
P141	0	0	0	0	0	0	0	0	0	0
P151	0	0	0	0	0	0	0	0	0	0
P161	0	0	0	0	0	0	0	0	0	0
P171	0	0	0	0	0	0	0	0	0	0
P181	0	0	0	0	0	0	0	0	0	0
P191	0	0	0	0	0	0	0	15	0	0
P201	0	0	0	0	0	0	0	0	0	0
P211	0	0	0	0	0	0	0	0	0	0
P221	0	0	0	0	0	0	0	0	0	0

Description

Comments

[Cancel]   [OK]

The program:	Could be transformed using P coordinates:
MESSAGE Put tool	*MESSAGE Put tool*
FEEDRATE 10000	*FEEDRATE 10000*
MOVEABS Z0	*MOVEABS Z P4*
MOVEABS X50.0 Y20.0	*MOVEABS X P7 Y P2*
FEEDRATE 1000	*FEEDRATE 1000*
MOVEABS Z-10.0	*MOVEABS Z P11*
OUT 5 1	*OUT 5 1*
DELAY 1500	*DELAY 1500*
WAITIN 5 1 1000	*WAITIN 5 1 1000*
FEEDRATE 10000	*FEEDRATE 10000*
MOVEABS Z 0	*MOVEABS Z P4*
OUT 5 0	*OUT 5 0*
MESSAGE Put tool done!	*MESSAGE Put tool done!*

Q Coordinates

The data written in this table have similar application as P variables described above. The difference is that each set T1Q1 - T1Q11, T2Q1 – T2Q11…. is assigned to a particular tool number. The data written in T1Qx will be used only in "Put Tool Sequence" / "Get Tool Sequences" for T1, T2Qx will be used in Put Tool Sequence" / "Get Tool Sequences" for T2, etc.

Example:

	1	2	3	4	5	6	7	8	9	10
T1 Q1	10	20	50	-10	0	0	0	0	0	0
T1 Q11	0	0	0	0	0	0	0	0	0	0
T2 Q1	60	70	80	-15	0	0	0	0	0	0
T2 Q11	0	0	0	0	0	0	0	0	0	0
T3 Q1	0	0	0	0	0	0	0	0	0	0
T3 Q11	0	0	0	0	0	0	0	0	0	0
T4 Q1	0	0	0	0	0	0	0	0	0	0
T4 Q11	0	0	0	0	0	0	0	0	0	0
T5 Q1	0	0	0	0	0	0	0	0	0	0
T5 Q11	0	0	0	0	0	0	0	0	0	0
T6 Q1	0	0	0	0	0	0	0	0	0	0
T6 Q11	0	0	0	0	0	0	0	0	0	0
T7 Q1	0	0	0	0	0	0	0	0	0	0
T7 Q11	0	0	0	0	0	0	0	0	0	0
T8 Q1	0	0	0	0	0	0	0	0	0	0
T8 Q11	0	0	0	0	0	0	0	0	0	0
T9 Q1	0	0	0	0	0	0	0	0	0	0
T9 Q11	0	0	0	0	0	0	0	0	0	0
T10 Q1	0	0	0	0	0	0	0	0	0	0
T10 Q11	0	0	0	0	0	0	0	0	0	0
T11 Q1	0	0	0	0	0	0	0	0	0	0
T11 Q11	0	0	0	0	0	0	0	0	0	0
T12 Q1	55	65	75	80	0	0	0	0	0	0

User Defined Coordinates

Q variables   Tool No:2 Q4

Description
Other comments

Cancel

OK

For T1:
MOVEABS X Q1   =>   MOVEABS X 10.0

For T2:
MOVEABS X Q1   =>   MOVEABS X 60.0

This allows same sequence to be used for different tools.

Rotary Toolchanger setup

Two specialized commands are implemented to control the rotary Toolchanger in addition to the set of commands described above.

HOME ROTARY command moves the Toolchanger to fixed position (pocket) associated with tool No 1. It is intended to be used in "Toolchanger Home Sequence", but can also be implemented in "Put Tool Sequence" and "Get Tool Sequence".

POSITION ROTARY command moves the toolchanger to a position (pocket) determined by "Put Tool Sequnce" and "Get Tool Sequence".

In order to position Rotary Toolchanger at the specific tool pocket **HOME ROTARY** and **POSITION ROTARY** commands must be included in "Put Tool Sequence", "Get Tool Sequence" and "Tool Changer Home Sequence". To determine the exact sequences refer to machine and Toolchanger description.

Toolchanger Home Sequence will start automatically after power-up.

"Toolchanger setup" menu provides the ability to define the operation of **HOME ROTARY** and **POSITION ROTARY** commands according to the toolchanger design.

Rotary Toolchanger parameters

Number of pockets
Total number of pockets (carousel positions) could be any number between 2 and 50. This parameter should be always set.

Inputs description
This parameter describes all inputs available to control toolchanger position. It is related to the input data interpretation for **HOME ROTARY** and **POSITION ROTARY** commands. Refer to toolchanger design data to select the proper setting.

Home input and Pocket input

The toolchanger position is controlled by using one "Home input" and one "Pocket input". **POSITION ROTARY** command will determine the carousel position by counting "Pocket input" transitions.

The inputs parameters to be defined with this input description are "Home Input number" and "Pocket Input number".

Home input, Pocket input and Code inputs

Every pocket is associated with a specific code (combination of "code inputs"). **POSITION ROTARY** command compares the code corresponding to the current state of all code inputs with the target tool code. **POSITION ROTARY** command will end when the two codes match and the "Pocket input" is active.

In addition to the "Code Inputs"= (code assigned to Tool No1) and "Pocket input" active **HOME ROTARY** command will expect "Home Input" to be in its active state.

Home Input and Code Inputs

Every pocket is associated with a specific code (combination of code inputs). **POSITION ROTARY** command will end when the combination of "Code Input X" matches the target tool number code. This command does not control "Pocket Input" state.
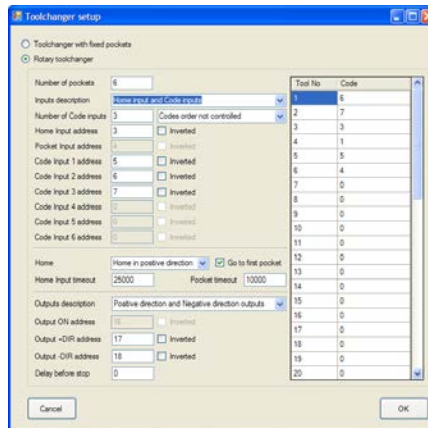
Pocket Input and Code Inputs

POSITION ROTARY command will end when the combination of the "Code Input X" matches the target code. By default **HOME ROTARY** command will position the Toolcanger to Tool No 1 pocket. In this configuration "Home Input" is ignored.
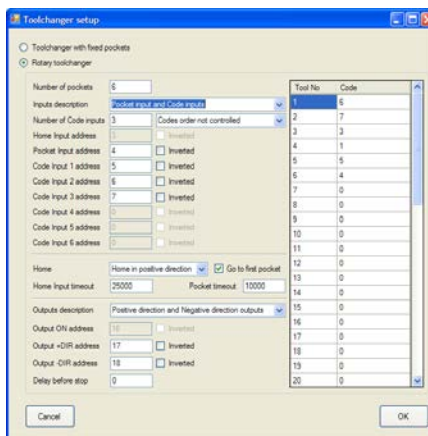
Code Inputs only

Carousel position is controlled only by "Code Input X". **POSITION ROTARY** command will end when the "Code Inputs" matches the target code. By default **HOME ROTARY** command will position the toolcanger to Tool No 1 pocket. In this configuration "Pocket Input" and "Home Input" are ignored.
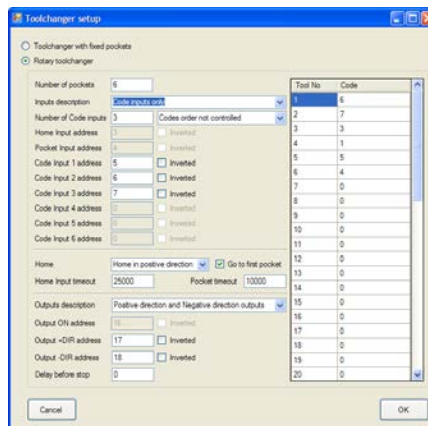
Number of Code inputs, Home Input address, Pocket Input address, Code Input address
The parameters below describe the relationship between **HOME ROTARY**, **POSITION ROTARY** commands and the actual inputs.

Number of inputs dedicated to control the tool changer position.
Note: Only when inputs description includes "Code inputs".

Code Inputs control level description.
Note: Only when Inputs description includes "Code Inputs"

"Home Input", "Pocket Input" and "Code Input X" I/O map location. The parameters describe the relation between HOME ROTARY and POSITON ROTARY commands and code inputs.

Any of the inputs values could be inverted by checking one of this boxes

Tool No / Code table
The table describes the relationship between tool number selected with M6Txx command and the combination of "Code Input X" at this pocket. This table is active when "Inputs description" includes "Code inputs".
The software converts the inputs values (current toolchanger position) into a number and compares it to the target tool code (selected with M6Txx command). According to the result and outputs description the command sets the toolchanger control outputs (move forward, reverse, or stop).

Inputs states are converted into a number by the formula:
(Code Input 1)x1 + (Code Input 2)x2 + (Code Input 3)x4+(Code Input 4)x8 + (Code Input 5)x16 + (Code Input 6)x32=Code
Example: Number of Code Inputs=3, Code Input 1=0, Code Input 2=1, Code Input 3=0 (Code inputs 4 to 6 are ignored)
        Code=1x0+0x2+1x4=4 => Toolchanger is positioned at the Tool No 6 pocket.

Homing

This parameter allows excluding the **HOME ROTARY** command. If **HOME ROTARY** command is active (one of the directions should be selected) the system will automatically perform toolchanger homing after power-up. The direction selected with this parameter is effective only when the toolchanger position or homing direction is not known.

Depending on inputs description **HOME ROTARY** command will control Home input, Home and Pocket inputs, and/or Code inputs.

Home Input timeout, Pocket Input timeout

Time interval between **HOMER ROTARY** or **POSITION ROTARY** command start and next "Home Input" or "Pocket Input" active transition. The timeout expiring will terminate any operations related to the commands.

Outputs description

This parameter describes toolchanger rotation control outputs. Starting the **HOME ROTARY** or **POSITION ROTARY** command will activate the outputs. They will remain active until the target inputs state is detected. If the inputs state matches the target state the outputs will be deactivated.

Single Positive direction output   Single Negative direction output

Output I/O map address. The output value will be inverted if the box is checked

Output I/O map address. The output value will be inverted if the box is checked

The output is active during the **HOME ROTARY** and **POSITION ROTARY** commands. The direction selected will not affects the tool counting. For Single Positive direction the tool pockets order is assumed to be: 1, 2, 3, 4,…N, 1, 2… For Single Negative direction the tool pockets order is assumed to be: N…4, 3, 2, 1,N…4, 3, 2, 1.
N is the Number of pockets parameter.

Outputs control (single direction toolchanger)

Output description		Output ON	Output +DIR	Output -DIR
Single Positive direction output	Stop	N.A.	OFF	N.A.
	Start	N.A.	ON	N.A.
Single Negative direction output	Stop	N.A.	N.A.	OFF
	Start	N.A.	N.A.	ON

Enable and Positive direction outputs   Enable and Negative direction outputs

"Output +DIR" will be ON for positive (positive) rotation and OFF for negative.

"Output –DIR" will be ON for negative (negative) rotation and OFF for positive.

HOME ROTARY and **POSITION ROTARY** commands always will activate Output ON.
Commands calculate the shorter distance between current and target positions (tool pocket number) before setting the "Output +DIR" or "Output –DIR".

Positive direction and Negative direction outputs

Two separate direction outputs. Commands will calculate the shorter distance between current and target positions (tool pocket number) and will activate one of the control outputs.
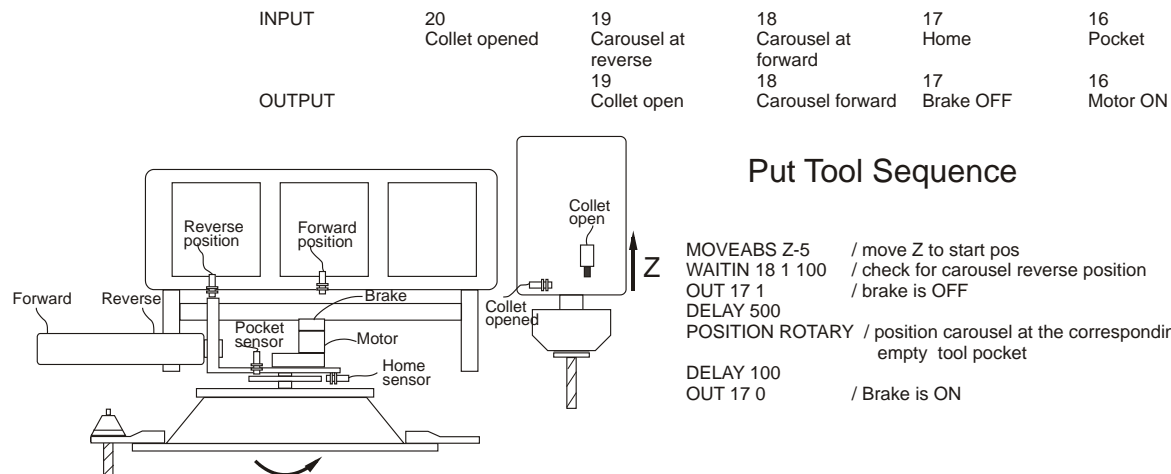
Outputs control (bidirectional toolchanger)

Output description		Output ON	Output +DIR	Output -DIR
Enable and Positive direction outputs	Stop	OFF	OFF	N.A.
	Start (+)	ON	ON	N.A.
	Start (-)	ON	OFF	N.A.
Enable and Negative direction outputs	Stop	OFF	N.A.	OFF
	Start (+)	ON	N.A.	OFF
	Start (-)	ON	N.A.	ON
Positive direction and Negative direction outputs	Stop	N.A.	OFF	OFF
	Start (+)	N.A.	ON	OFF
	Start (-)	N.A.	OFF	ON

Delay before Stop

If this parameter is different from zero then the outputs inactive state (Stop) will be delayed after the target position is detected.
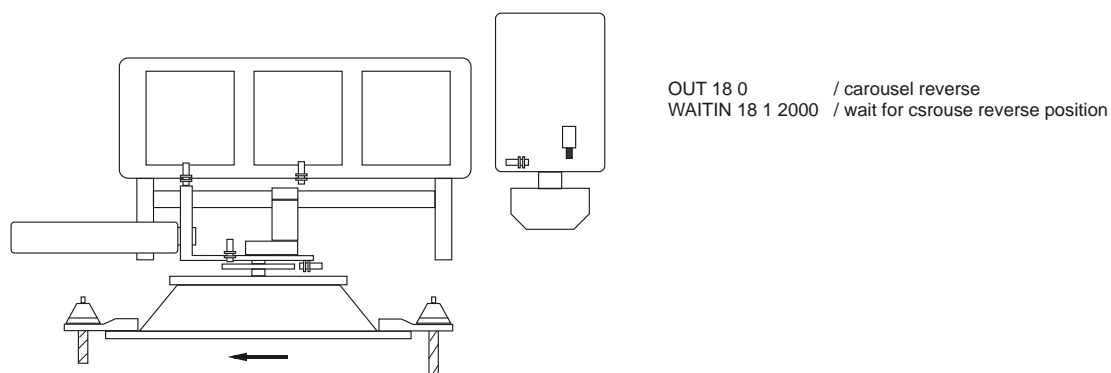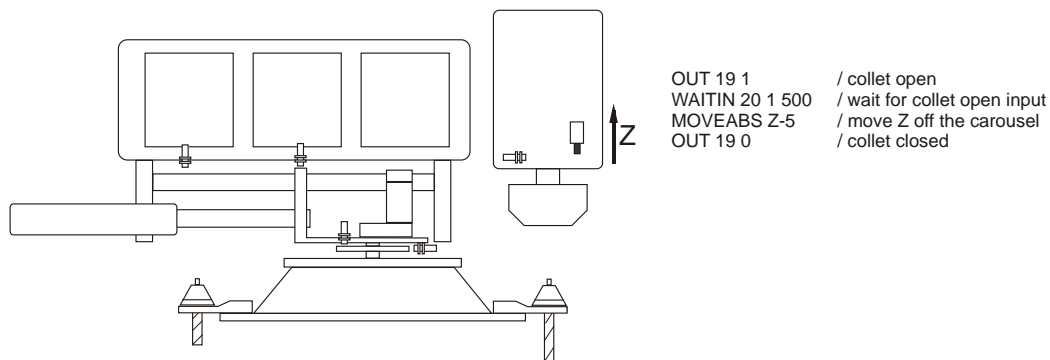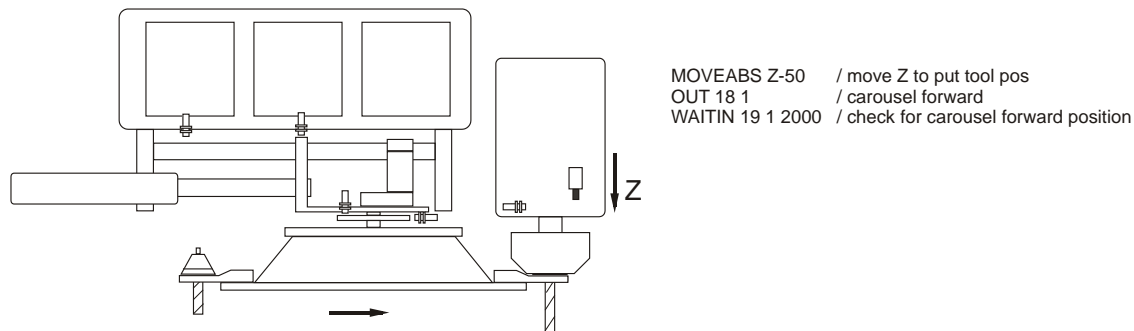

200ms delay

Rotary Toolchanger sample application with Enable and positive direction output

INPUT	20 Collet opened	19 Carousel at reverse	18 Carousel at forward	17 Home	16 Pocket
OUTPUT		19 Collet open	18 Carousel forward	17 Brake OFF	16 Motor ON

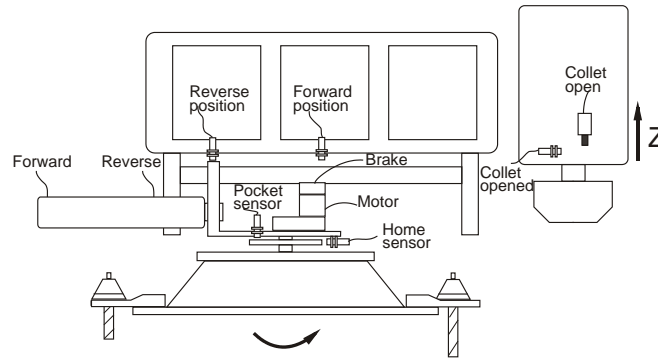### Put Tool Sequence

```
MOVEABS Z-5          / move Z to start pos
WAITIN 18 1 100      / check for carousel reverse position
OUT 17 1             / brake is OFF
DELAY 500
POSITION ROTARY      / position carousel at the corresponding
                        empty  tool pocket
DELAY  100
OUT 17 0             / Brake is ON
```

```
MOVEABS Z-50         / move Z to put tool pos
OUT 18 1             / carousel forward
WAITIN 19 1 2000     / check for carousel forward position
```

```
OUT 19 1             / collet open
WAITIN 20 1 500      / wait for collet open input
MOVEABS Z-5          / move Z off the carousel
OUT 19 0             / collet closed
```

```
OUT 18 0             / carousel reverse
WAITIN 18 1 2000     / wait for csrouse reverse position
```

Note: The sequence above is for sample purpose only and does not represent fully developed Put Tool sequence.

Rotary Toolchanger sample application (cont)

INPUT	20 Collet opened	19 Carousel at reverse	18 Carousel at forward	17 Home	16 Pocket
OUTPUT		19 Collet open	18 Carousel forward	17 Brake OFF	16 Motor ON

### Get Tool Sequence

```
MOVEABS Z-5        / move Z to startl pos
WAITIN 18 1 100    / check for carousel reverse position
OUT 17 1           / release the brake
DELAY 500
POSITION ROTARY    / position carousel at the new tool pocket
                     selected
DELAY 100
OUT 17 0           / brake is ON
DELAY 500
```
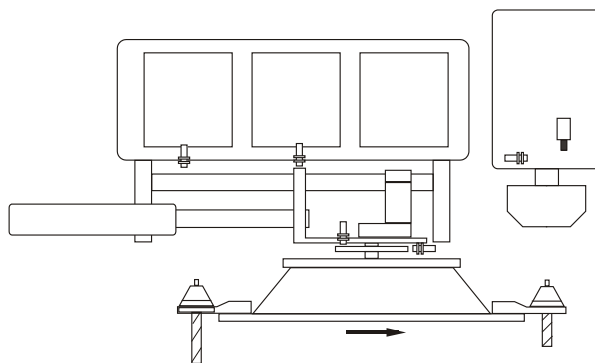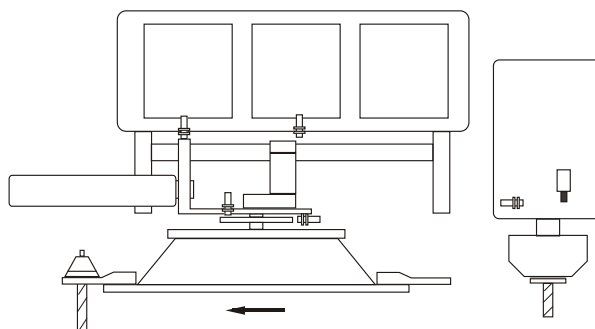
```
OUT 18 1           / carousel forward
WAITIN 19 1 2000   / check for carousel forward position
OUT 19 1           / collet open
WAITIN 20 1 200    / wait for collet open input
```
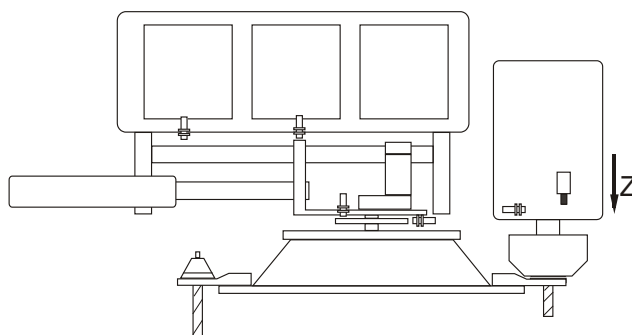
```
MOVEABS Z-50       / move Z off the carousel
OUT 19  0          / collet closed
WAITIN 20 0 200    / check for collet closed
```

```
OUT 18 0           / carousel reverse
WAITIN 18 1 2000   / wait for reverse position
```

Note: The sequence above is for sample purpose only and does not represent fully developed Get Tool sequence