LICENCE AGREEMENT

1. Introduction

This agreement between the company PRIMUS DATA, located in CH-8840 Einsiedeln, Switzerland, called PRIMUS DATA in the following, and the client has as purpose setting rules for the use of the software package, called SOFTWARE in the following, delivered by PRIMUS DATA. This agreement becomes effective by opening the diskette/dongle seal.

2. Right of usufruct

PRIMUS DATA concedes to the client the nontransferable right to use precisily as many sets of the SOFTWARE as he purchased licenses. The right of usufruct may not be transfered to a third person. Parts of the SOFTWARE (for example postprocessors) may not be given to a third person as well.

3. Copyright

The SOFTWARE is the property of PRIMUS DATA and is protected by copyright law and international copyright treaty as well as the respective national legislation. Therefore it must be treated as all the other products that are protected by copyright. Copies of the SOFTWARE may be made exclusive for safety and archive purposes. The printed material that is delivered together with the SOFTWARE may only be reproduced with the explicit authorization from PRIMUS DATA. The client does not acquire the property for the SOFTWARE by buying it, but merely a right of usufruct.

4. Alterations

The client is not allowed to make any alterations or complements in the SOFTWARE or in parts of it, or to let it be made (specially alterations that enable the use of the SOFTWARE without the included protection hardware). In the same way neither the SOFTWARE nor parts of it may be used in other programs. Alterations in postprocessors, libraries, fonts, paramater macros and system configuration are exceptions to this designation, which the client may do for using the software.

5. Limited liability

The manufacturer has made every effort to produce a product (SOFTWARE and/or support material) to the best of his knowledge and believe. PRIMUS DATA specifically disclaims all warranties, expressed or implied, including but not limited to implied warranties of merchantability and fitness for a particular application, use or purpose. In no event will PRIMUS DATA or its dealers be liable to the client for any damages, including lost profits, lost savings, or other incidental or consequential damages arising from the use or inability to use the program even if PRIMUS DATA or its dealers have been advised of the possibility of such damages, or for any claim by any other party. The client is, for example, responsible to test programs generated by the SOFTWARE before he uses them.

6. Payment

In case the client comes to a delay in payment, PRIMUS DATA reserves the right to reclaim the SOFTWARE from the customer.

7. Warranty

With respect to the physical diskettes and physical documentation enclosed herein, PRIMUS DATA warrants the same to be free of defects in materials and workmanship for a period of 12 months from the date of purchase. In the event of notification within the warranty period of defects in material or workmanship, PRIMUS DATA will replace the defective diskettes or documentation.
The absence of defects in the SOFTWARE cannot be assured by PRIMUS DATA, although the manufacturer has taken the greatest care during its development. Its repairing or avoidance are a consequence of the producer’s technical and personal possibilities.Those defects, if they may occur, cannot have a warranty or any price reduction. If possible those defects or errors will be repaired in the following versions of the SOFTWARE.

8. Governing law

This agreement shall be costrued, interpreted and governed by the laws of Switzerland.

Prim CAM registration form

Please fill in this registration form and

- Fax it (0041-55-418 49 50) or
- Send it (fold and glue together)

Then you will receive

- Information about the new versions
- Technical support for questions about PrimCAM

Company:	
Name:	
Street, Number (or p.o. box):	
State, City:	
Country:	
Phone:	
Fax:	
Email address:	
Licence no. (see label on dongle):	

Suggestions: _____

_____

✍ We **don't** want our company to be on the reference list!

_____ ✍ _____

PRIMUS DATA
Kornhausstrasse 35
Postfach 413
CH-8840 Einsiedeln
Switzerland

Prim CNC®

User's Guide

Contents

Important information about PrimCNC

General

?? Make sure that you are logged in as a system administrator when installing PrimCAM, because that's the only way the parallel port driver canbe installed. You can also install the parallel port driver by calling PHDIoInstall.EXE.

?? If you use a controller with a serial port, make sure the dongle is not on the same serial port and the port is not in the dongle search path (see PrimCAM manual fordongle search path).

Preface

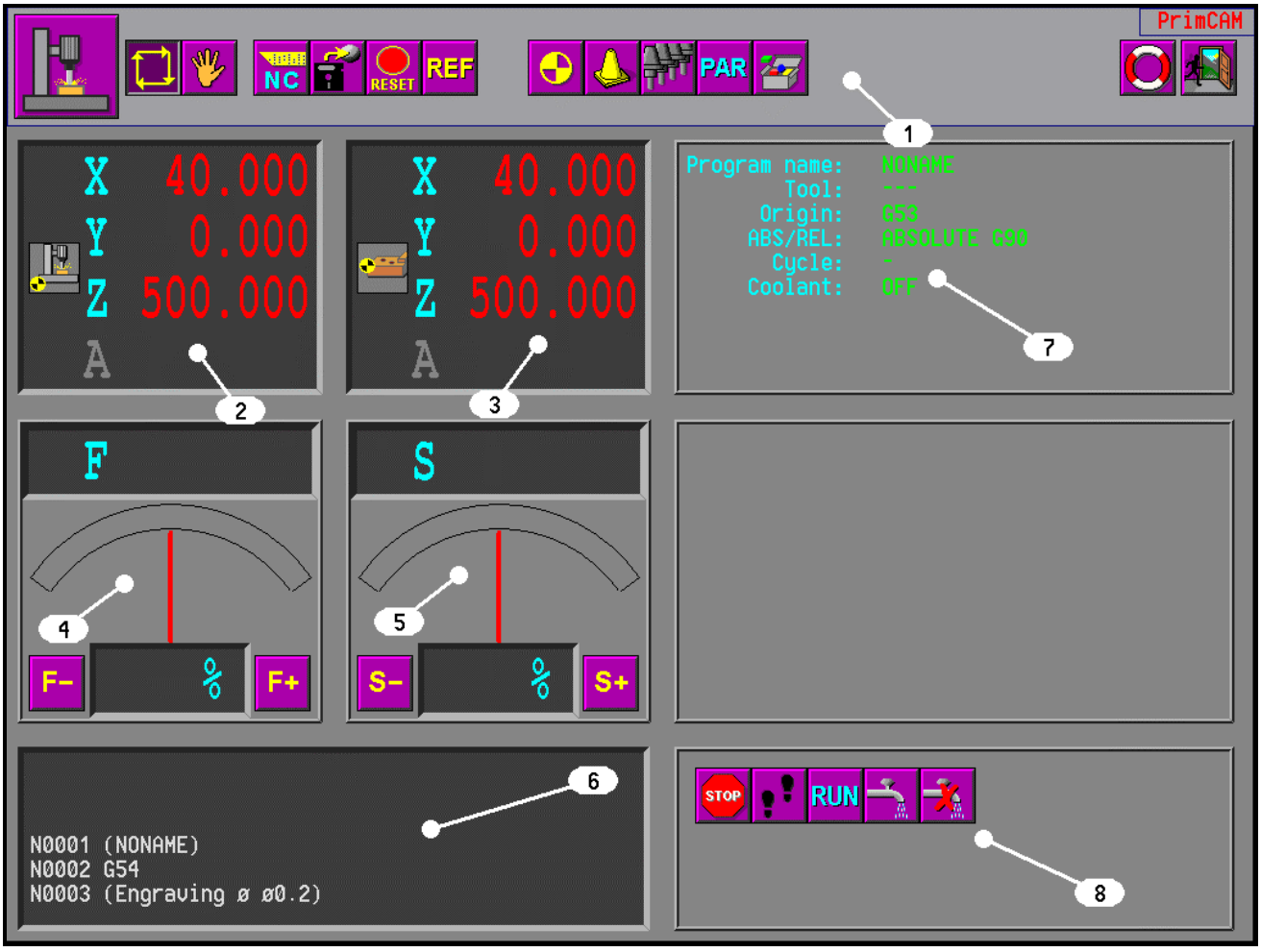
PrimCNC is an interface program for the isel controllers and makes a little CNC milling center out of your isel machine. You get similar options as with an intelligent CNC control. Because isel controllers don't implement all the features required by PrimCAM (Z height correction, surface measurement, tool length compensation, drill cycles etc.), the controller is operated in DNC mode and the missing commands can be emulated this way. You can also call the control interface directly from PrimCAM, without leaving the program.

1. System overview PrimCNC

PrimCNC is a supplementary program for the PrimCAM CAD/CAM/NC programming package and provides an integrated interface to various controllers. Its additional features increase the capabilities of the controllers by commands like Z height compensation, surface measurement, tool length measurement and drilling cycles. PrimCNC can also control the automatic tool changing units and set spindle speed and feed according to the values determined in PrimCAM.
PrimCNC reads DIN/ISO code generated by PrimCAM and communicates directly to the controller. Basically a machine can be applied to build up a complete "Mini CNC machining center". The combination of a simple machine + PrimCAM is particularly interesting for training centers. A very cost effective CNC machining center is provided. Operator mistakes are unlikely to cause damage.
The following keyword overview gives an impression of the possibilities of PrimCNC:

Introduction

The following diagram shows the graphical user interface to the control.



(1) Icons to select CNC functions.
(2) Absolute machine coordinates.
(3) Relative machine coordinates.
(4) Feed display and correction.
(5) Spindle speed display and correction.

(6) NC code window showing the DIN/ISO codes currently processed
(7) Display of the machine status.
(8) Selection of functions.

Setting origins

- Up to 6 origins can be set (G54-G59).
- Operation of the machine in steps of 1, 1/10, 1/100 mm and continuously to set origins.

Automatic length measurement of tools

The length measurement device consists of a precise end switch, which is activated when the tool moves in Z direction. In this way the exact tool length can be measured. The length measurement device is screwed on the operation area of the Isel machine.

Control of spindle, coolant and vacuum cleaner

- integrated control and automatic setting of spindle speed (RPM) with various spindle controllers
- Possibility to switch on/off non RPM controlled milling motors via relay.
- automatic delay till the spindle has reached its final rpm
- Fully automatic on/off control of cooling water, vacuum cleaner etc.

Automatic change of Tools

- Support of Isel tool changing unit for stepping and servo motor controls.
- Up to 24 automatic and 1 manual tool changing positions.

Machine cycles

- Support for drilling, peck, high speed peck, and surface measurement cycles.
- Parameters of cycles similar to those used in big CNC machining centers.

Surface measurement

- Engravings to surfaces that are not plane
- Measurement points manually selectable in the drawing
- Measurement by mechanical unit or laser

Hand operating unit

There are different hand operating units available. They are connected to the parallel port of the computer and have functions like start, stop, stepwise mode, emergency stop, spindle speed and feed control.

2. Installation PrimCNC

If PrimCNC was aquired together wit PrimCAM, it gets installed automatically when installing PrimCAM. The software is the same, the hardlock decides wheter to run PrimCAM, PrimCNC or both of them.

2.1 Hard- and software requirements

To run PrimCAM on your computer, the following hard- and software requirements have to be fulfilled:

- IBM-compatible PC with processor 486 or higher
- 16 MB RAM memory
- 20 MB free harddisk space
- Operating system Windows 95 / 98 / NT / 2000 / XP
- free RS232-interface (only for the serial transmission of programs to machine)
- Graphics card with resolution 1024x768 pixels in 256 colors (at least 1MB display memory)

2.2 Installation Hardware

Hardlock (dongle)

The black hardlock can only be used with the parallel port. It is found automatically by PrimCAM, searching the ports Hex 378, 278, 3BC in this order. This search order can be changed as described below.
The green-gray hardlock can be used on the parallel as well as on the serial ports. It is found automatically on parallel ports. To use them on a serial port, you have to set the address the dongle should be searched for by setting an environment variable in the file AUTOEXEC.BAT:

SET HL_SEARCH=*[Port], ...*

In Windows NT, you can set an environment variable in Settings/Control/System/Environment:
Variable: HL_SEARCH
Wert: *[Port]*

[Port] is composed of an I/O address in hexadecimal and a port identifier:

I/O address	Meaning
378	Parallel port
278	Parallel port
3BC	Parallel port
3f8	Serial port COM1
2f8	Serial port COM2
3e8	Serial port COM3
2e8	Serial port COM4

I/O address	Meaning
p=parallel	normal parallel port
s=seriell	normal serial port
e=ECP	parallel port in ECP mode

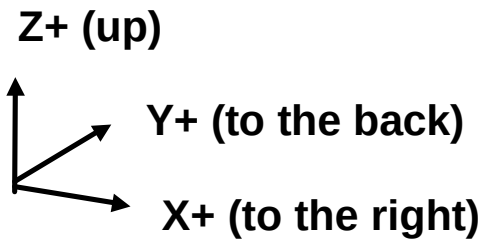
Examples:

SET HL_SEARCH=3f8s
The hardlock is only searched for on the parallel port COM1.

SET HL_SEARCH=378p
The hardlock is only searched for on parallel port 0x378.

Machine and controller

Install the machine and controller according to the instructions in the appendix.
Pay attention to the fact that PrimCAM uses a coordinate system that conforms to the CNC standard: Z+ up, X+ to the right, Y+ to the back, looking at the machine from the front.
If you create a drawing and a CNC program in PrimCAM, it gets milled exactly as you saw it on the screen, assumed that you stand in front of the machine.



2.3 Software installation

If you use an installation CD, the installation starts automatically. You can also manually call the file setup.exe.
If you download the program from Internet, execute the file install.exe. There are files setup.exe, setup.w02, … extracted. Then you can start setup.exe.
After selecting the language for the installation you are prompted for the destination drive.
If your operating system is Windows **NT** and you choose the installation directory for PrimCAM, don't put any spaces into the installation path (like c:\program files\primcam). This will cause an error message „Hardlock not found" because the installation program cannot install the hardlock driver into a path containing spaces. Also make sure that you are logged in as a system administrator when installing PrimCAM, because that's the only way the hardlock driver can be installed.
If PrimCAM has already been installed on the indicated directory, you are asked whether you would like to make a new installation or an update.

?? **Update**: The new version of PrimCAM will be installed, but all your old configuration files, tables, tool-, material- and part libraries will be kept.

- **New installation**: PrimCAM will be installed completely new. Thereby old tables, tool- and material libraries and so forth are overwritten with the factory settings.

Now PrimCAM is copied to the computer and decompressed. At the end of the process, the file README.TXT is presented, which contains the most actual information.

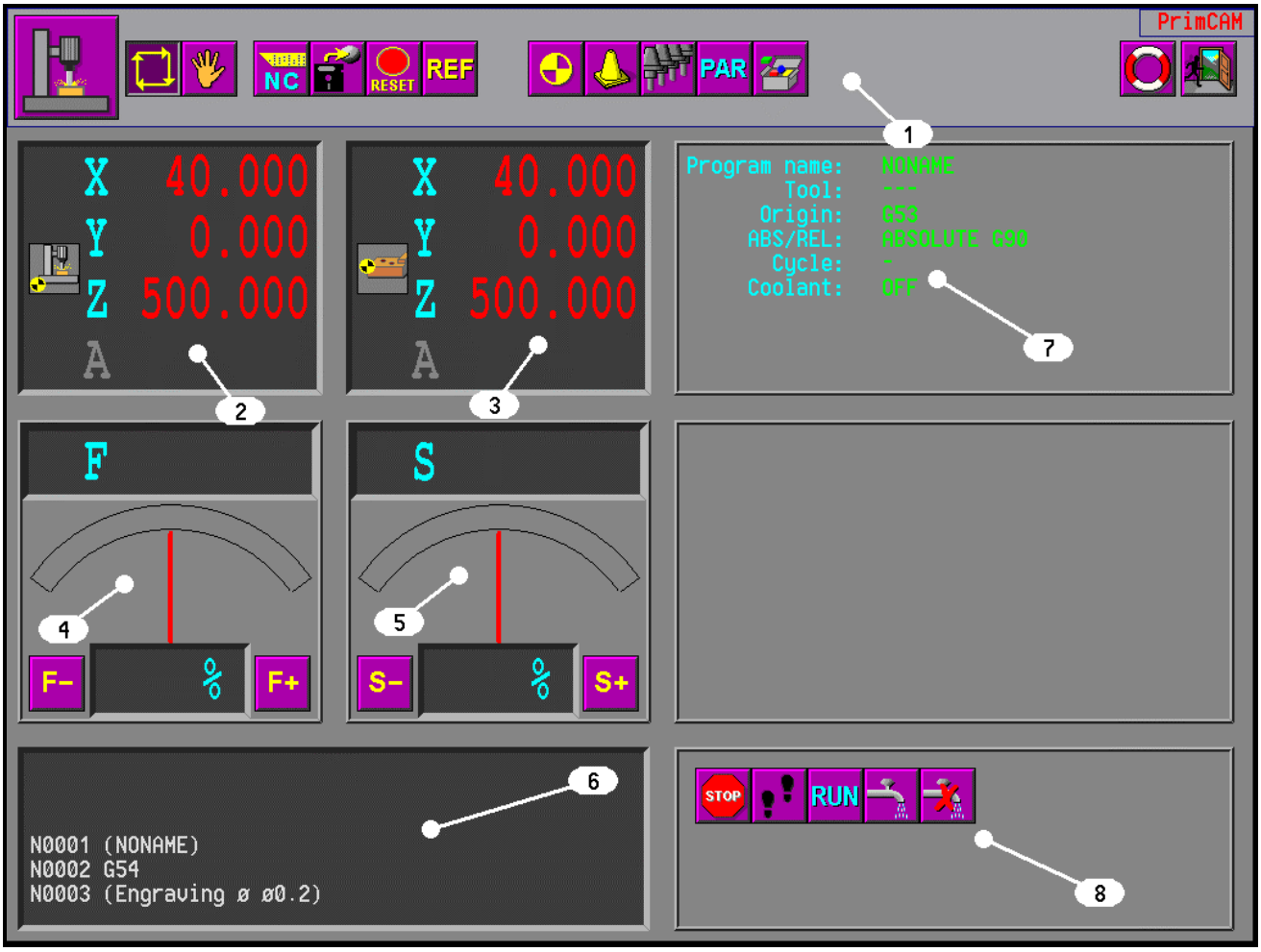
Do by all means execute the following **parameter configuration** before using the program and look at the **additional information to your controller in the appendix**. This is the only way proper function of the program is possible.

3. Introduction PrimCNC

3.1 User interface

Start the isel interface program in the menu **File functions** with the icon

Machine user interface. The following image appears on your screen:



(1) Icons to call CNC functions

The upper screen border shows the functions to call different CNC functions, which are:

The **automatic mode** serves to execute NC programs made by PrimCAM. The display shows actual relative and absolute machine coordinates, feed, spindle speed and status in different screen windows. The left bottom window shows the section of the whole NC program that is executed at the moment. The window at the left bottom features function icons to control the program flow.

Opposed to the automatic mode is the **manual mode**. In this mode, the machine can be moved using arrow keys or the according icons. It helps setting origins or tool change positions.

Normally, NC code is generated by the postprocessor of PrimCAM automatically. But this function allows to enter NC code manually line by line and execute it. All the commands of the machine can be carried out this way. This function is useful for example to insert another tool on a machine equipped with automatic tool changer. Consult the chapter **Format of the NC code** for detailed information

If you enter the machine interface, the NC code file of your actual drawing is loaded automatically, supposed you did make the according NC file using the postprocessor. This function allows you to load another NC file. A file window is opended that shows you the selection of available programs.

Program reset the NC program in memory to the first line. This function is used to process another piece with the same NC program. Active cycles are cancelled. This function is executed automatically after an emergency stop. If a NC program is finished, this function is also executed by the code M30.

The Function **reference drive** initializes the CNC controller and moves the axes to the reference position. This function must be executed every time PrimCAM is started up. If you forget the reference function, an error message is displayed as soon as you try to use the controller. Once the reference drive is executed, you can switch between PrimCAM and the machine interface as many times as you like without repeating the reference function.

Up to 6 **origins** can be defined here. They are accessed in a program by the codes G54..G59.

All over the working area of the machine there are obstacles like clamping tools, tool change unit, tool lenght measuring unit etc. This function lets you define **forbidden areas** that prevent the machine from driving into an obstacle.

Tool configuration lets you determine for every tool whether it should be changed automatically or manually. The tool length can also be defined if the automatic tool length measurement unit is not in use.

Here you set the **parameters** for machine and accessories. These adjustments have to be made only once when configuring the machine. They remain fix, except if you install new options for your machine.

The **Trickbox** provides special functions that are used rarely. Some of them allow to measure machine parameters automatically. They clamp and unclamp tools with the automatic tool changer or open and close the cover. You can also test inputs and outputs. The following list gives an overview:

Measure tool length
Set actual tool number
Test surface measurement laser

Calibrate laser for surface measurement
Clamp tool
Close tool cover
Test inputs

Calibrate tool length measurement unit
Measure Z origin with laser/tracing pin
Show surface measurement points
Calibrate tracing pin
Unclamp tool
Open tool cover
Test outputs

Activates the **Help function** with an overview of the CNC functions.

Quits the CNC machine interface

(2) Absolute machine coordinates

This window shows the absolute machine coordinates. The absolute zero point is at the left front bottom of the machine. The positive X direction goes to the right, the positive Y direction to the back of the machine and the positive Z direction up. After executing the reference function, the machine is at the position left-rear-top, showing coordinates like:

X= 0 mm
Y= 340 mm
Z= 155 mm

(3) Relative machine coordinates

shows the coordinates measured from the actually chosen origin. The origin is set with NC codes G54..G59 in the program. G53 switches to absolute machine origin, meaning absolute and relative coordinates correspond.

(4) Feed window and override

This window shows the actual feed in mm/min. The feed can be adjusted in 5% steps from 0..200% using the arrow icons or F5 and F6 on the keyboard. Most controllers consider a feed override with the next line or arc segment, not the current.

(5) Spindle speed window and override

This window shows the actual spindle speed [rpm]. The spindle speed can be adjusted in 5% steps from 0..200% using the arrow icons or F7 and F8 on the keyboard. Most controllers consider a feed override with the next line or arc segment.

(6) NC code window

This window shows the NC code line actually executed in purple. Preceding and following lines are shown in white.

(7) Status window

The status windows shows information about the program and it's status.

Program name:	Name of the NC program that is loaded
Tool:	No of tool inserted, lenght in mm
Origin:	Origin set (G53,G54-G59)
ABS/REL:	Absolute/relative coordinate mode
cycle:	Cycle, for example G79 for surface measurement
Coolant:	off/on

(8) Program function window

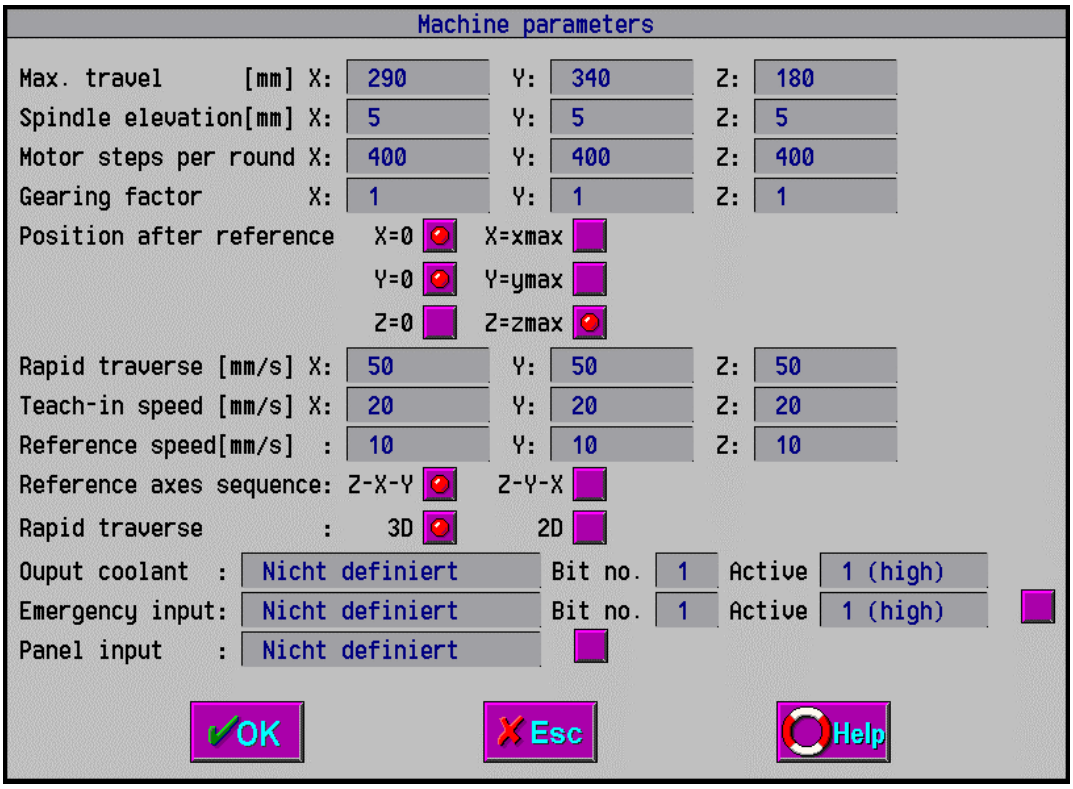
Here you see icons that are only valid in certain program modes. In the automatic mode, it shows icons to start and stop programs.In the manual mode, you see icons to move the machine in steps or continuously.

3.2 Parameters (machine, controller interface)

Klick the icon **parameters** to configure the machine and interface parameters. A windows with different possibilities appears.

Start with the **Selection of the controller**. Pay attention to leaving the machine interface (return to PrimCAM) and enter it again so that your changes become active.

Continue with setting the **general machine parameters**. This window may look a bit different according to the machine chosen.
The following picture shows the window for isel interface controller card 5.0:



Axis active: Set here if the axis is available.

Axis assignemnt: Assign here logical axes X,Y,Z,... to the physical axes 1,2,3...
This allows to change axes by software.

Type(linear/circular): Choose between linear and rotating axes

Max. travel: Max. travel of the machine is entered in mm for every axis. If a NC program goes beyond the values given here, an error message is displayed, showing the axis and direction that caused the error. The NC program is stopped and reset. Depending on the controller, the values can be set 0..MAX or MIN..0

Spindle elevation: Spindle elevation is entered in mm/round or °/round for every axis seperately. The axes for example have an elevation of 5mm.

Motor steps: Set the number of steps per round for the motors of your axes. They are for example:

Half step mode	0.9°/step	400 steps/round
Full step mode	1.8°/step	200 steps/round

Gear: The gearing factor is set to 1 if no gear is installed.

Axis direction: There are controllers that allow changing the axis direction.

Reference direction: For doing the reference, the machine moves to a reference switch. Some controllers allow moving here in positive or negative direction.

Pos. After ref.: Set here where the machine is after the reference for every axis.

Rapid traverse feed: The rapid traverse feed is entered in mm/s. You have to optimize this value experimentally. If an axis falls out of the step, the feed is too high.

Teach-In feed: Here you define the feed rate for the manual mode. It is normally set to about 20mm/s.

Reference in feed: Speed for the reference function in mm/s. This is the feed the axis moves into the switch. The reference speed can be adjusted seperately for every axis. If the speed is too high, there is a danger to damage the end switch of theaxis. If the setting is too low, the reference function takes a lot of time. A value of 10mm/s should be ok.

Reference out feed: Speed for moving out of the reference switch in mm/s. Some controllers allow setting this value. It should not be higher thanthe start-stop frequency for stepper motors.

Reference distance: After the reference, the axes can be moved out of the switch by this value. Then they are zeroed. This may be necessary if the reference switch is also used as an end switch.

Start/stop ramp: Start and stop ramp give the max acceleration / decceleration for an axis in mm/s^2.

Axis sequence for reference: Determines the sequence for the axis when doing a reference drive. You can minimize the danger of collisions during reference. If you have an automatic tool changer in the back part of your machine, you do the reference drive first in X, then in Y direction. The Z end switch should always be referenced first.

Rapid traverse 2D/3D Controllers normally execute movements with all 3 axis together (3D). Fore some controllers, it is recommended to switch to 2D for rapid traverse and position in XY with Z lifted. This should prevent from loosing steps. If you switch to 2D here, rapid traverses are executed in XY with Z axis up.

Configure the **CNC interface** now. Part of those settings are specific to the controller and described in the appendix for the actual controller.

Circle/helix interpolation: For circle/helix interpolation, you can set the max. bulge error in mm or a resolution in °. This is used for controllers that don't support circle or helix. PrimCAM then approximates them with small lines. A high degree value leads to circles that are processed very fast but consist of lines that you can see. If the degree value is very low, the transmisstion of data for all lines takes quite some time prior to every circle/bow segment, but the circle looks better.

Buffer mode: Defines if single elements (arcs and lines) are first transmitted and then executed all together without stops between elements (if the controller supports that).

If the parameters are set properly and all connections between motors and controller have been made, do a **reference drive**. The CNC controller is initialized and the axes moved to their reference position. This function has to be executed every time you restart PrimCAM. If you forget, you will get an error message.

Right after moving to the reference positions, the macro CNC_REFO.SUB in the directory \PRIMCNC gets called. This macro can be changed by the user. In the present form, it moves the machine to the absolute position Y=190mm, which means out of the forbidden area of the automatic tool changer located in the rear of the machine.

```
G53          Select absolute machine coordinates
G00 Y190     Move to  Y190
```

3.3 Configure tool change

Tool change can be acomplished manually or fully automatic by use of the isel tool changer. You can define up to 24 automatic plus 1 manual tool changing positions.

manual: If tool change is set to manual, the machine traverses to the point defined by TCH-point M and opens a dialog box requesting to insert the new tool. Choose a position for manual tool change that can be easily accessed, for example at X=0, Y=0, Z0=max.

automatic: For a fully automatic tool change, you need the equipment from Isel. The tool is brought back and the new tool is fetched. Depending on the configuration for tool lenght measurement, the tool is measured automatically afterwards.

TCH point: Here you set the coordinates for the tool change points. Set the red ball to the point you would like to configure. 1-24 are for automatic changing positions, M for the manual tool change position. To set the coordinates, use the manual mode to move the machine to the according position and copy the actual coordinates by clicking the field on the right of the coordnates. If the green hook is visible, the actual position is set as tool change coordinate. But you can also enter the coordinates manually using the keyboard. A description on how to set tool change points using the manual mode follows.

Output clamp+: Activated if the tool should be clamped in the automatic tool change unit. Set active to 1(high).

Output clamp-: Activated if the tool should be released from the automatic tool change unit. Set active to 0(low).

Output cover: opens cover of tool changing unit

Input cover opened: Test if cover is opened already

Input pressure ok: Input for pressure sensor

To move the machine manually, leave the tool change configuration window and select the **manual mode**. A gray rectangle appears, symbolizing the moving area of the machine in XY direction. A yellow cross shows the actual position of the machine. If you move the mouse cursor into the gray rectangle and click the left mouse button, the machine moves to the according position. The feed used is the one defined in parameters/teach-in feed. This function serves to approach a position roughly and use stepwise mode afterwards to approach exactly.
The program functions at the right bottom show icons to move the machine. You can use steps of 0.01mm, 0.1mm, 1mm or continuous mode. The ranges can be switched by clicking at the blue numbers. The machine moves as soon as you click to the icons(X+, X-, ...) or use the arrow keys/pg up/pg down. In continuous mode, the machine moves as long as you press the button. The following keys can be used:

X-	arrow left
X+	arrow right
Y-	arrow down
Y+	arrow up
Z+	page up
Z-	page down

Move to every tool position of your tool magazine and save it in the tool change configuration window. Approach a tool position so that the tool can be clamped when it sits inside the tool store.

The **automatic tool change** runs the following way:

1. Spindle is turned off
2. Cover of tool magazine is opened.
3. Z axis is lifted to maximum Z height.
4. Macro CNC_WWPI.SUB gets called. This macro can be changed by the user if the tool change unit is not installed in the back of the machine and the tool doesn't drive into the tool changer in Y+ direction. Normally the macro looks something like this:

G00 X0 Y-20	Machine moves in XY to a position 20 mm before the tool change point
G00 Z0	Machine moves in Z to the height of the tool change point
G01 Y0 F400	Machine moves in Y+ direction into the tool changer, with a feed of 400mm/min

5. Tool gets unclamped.
6. Macro CNC_WWPO.SUB is called (Move out of tool changer)
7. Macro CNC_WWGI.SUB is called (Move to position of new tool)
8. New tool is clamped.
9. Macro CNC_WWGO.SUB gets called. It moves the tool out of the tool magazine and looks like this:

G01 Y-20 F1000	Machine moves in Y-direction with 1000 mm/min.
G53	Absolute machine coordinate system is activated.
G00 Z155	Machine moves to Zmax; this value must be adjusted to the height of your machine.

G00 Y190 Machine moves out of the forbidden area of the tool changer to Y=190 mm.

10. Tool cover is closed
11. Z axis moves to maximal height.
13. Depending on the setting for tool length measurement, the tool lengthis measured.
14. The program continues.

If your tool changer is not installed at the back of your machine and the tools are not released to the front (Y- direction), you have to change the macros.

Now you should test all the positions. By entering *T1 M6* in the **NC code window** you can clamp the first tool. Continue with *T2 M6* etc. At the end, use *T0 M6* to bring the tool back without getting a new one.

3.4 Automatic tool length measurement

The automatic tool length measurement is one of the outstanding features of the PrimCNC interface software for isel machines. It functions the following way: After a tool change, the machine moves towards the end switch installed in the tool measurement unit. As soon as the switch is activated, the movement stops and the actual position of the Z axis is read from the control. The Z position is then used to calculate the lenght of the tool.

The following paragraph describes the calibration of the tool measurement unit.

To function properly, the **tool length measurement** unit has to be **configured** first. Here you tell PrimCNC when the length of a tool should be measured. Measurement is done automatically if the appropriate unit is installed. Otherwise, you have to enter the length of a tool manually (for manual tool change) or the value is taken from the tool library (automatic tool change).

Option off: Tool length is not measured automatically. The length value is taken from the tool library. For manual tool change, the system asks for the new length.

Length invalid: Tool is only measured if the length valid flag in the tool library is not set. After measurement, the flag ison and the tool doesn't get measured again.

1st use in program: At the start of a new NC program, the tool length valid flags are reset. So all the tools are measured when used the first time in the program. If the program is executed more than once, the tools are not measured anymore.

Always after tool change: The tool is measured after every tool change.

Measurement point: Set the coordinates of the automatic tool measurement point here.

When measuring the length of a tool, the machine first goes in rapid traverse to Z=zmax, then to the XY coordinate of the tool measurment point, finally, also in rapid traverse, to the Z value of the tool

measurement point. Then the tool goes down in Z very slowly until touching the measurement unit. For speed reasons you should set the Z value of the tool measurement point as low as possible, so that the slow movement in Z down is as short as possible. But do not set it too low because a long tool may hit the measurement unit in rapid traverse and damage it.
The best way to set the tool measurement point is to use the manual mode to drive the machine right over the measurement unit so that the tool stays right over the metal plate of the measurement device. Move in Z only so far down that even the longest tool you will ever use won't hit the metal plate yet. Go to the tool length measurement configuration then and copy the actual machine coordinates to the tool measurement point coordinates (by setting the cross in the box behind the values).

Z height: The tool length measurement unit must be calibrated before it can be used. This value is set automatically if you do the calibration steps described in the next paragraph.

Input: Input and logic used for tool measurement unit. The units delivered by PRIMUS DATA have a break contact. Therefore you need to select active 0(low). This guarantees that the tool doesn't hit the measurement unit if a cable is broken.

Calibration of the tool length measurement unit

Insert a tool and measure it's value manually with a ruler. The value doesn't have to be exact. Open the window **tool configuration**. Enter the manually measured tool length for tool no. 1 and set it's length to valid.

Go back to **set paramteters / configure tool measurement**. Set `Measure tool length` to `Length invalid`.

The **Trickbox** features functions that are used rarely. Activate the function

Set tool number. Set the actual tool number to 1.

Use the function **calibrate length measurement unit**. The tool drives down to the metal plate and, because the tool length is known, the height of the measuring unit can be calculated.

3.5 Spindle control configuration

Here you determine how PrimCAM sets the appropriate spindle speed. Use the appendix to configure specific spindle controllers.

manual: Opens a window for every spindle speed change indicating the new spindle speed. After confirmation, the program is continued.

1 bit on/off: Simple on/of control of a spindle with no automatic setting of spindle speed. PrimCNC uses the bit defined in the interface/output field. You can use the according output to drive a relais that switches the spindle on/off. You can

also set a waiting time in [s] that pauses the programm until the spindle has reached its final speed.

8 Bit digital: User configurable spindle controller with parallel output. The bits for start, stop and direction can be set arbitrary. If they are set to the same output byte as the speedbyte, the speedbits are distributed to the remaining bits, starting with the highest bit. You can optionally use inputs for rpm ok and error. If not used, set them to 'Not defined'. The speedbyte is divided linear to the region between rpm min. and rpm max.

Interface:

Rpm min./max.: The output speed byte is divided linear to the range between min. and max. rpm (8 Bit controller).

Output: Here you can set the Output used to control the spindle. Please notice that if you use a parallel port for spindle control, the spindle may be switched on when communication with the dongle takes place on another parallel port. The dongle is searched for in the order HEX 378, 278, 3BC. If you place the dongle before the spindle controller in search order, the dongle doesn't interfere with the control. Output start, stop, direction: special outputs for Kavo and 8 Bit.

Input rpm ok, error: special inputs for Kavo and 8 Bit.

COM/Baud/Adress: serial port, baud rate and modul address (FC1200,default 32) of the frequency converter.

Waiting time: Accelerating the spindle motor takes some time. Here you define how many seconds (On/Off controller) or how many seconds per 1000 rpm the program should wait before continuing.

Speed increasing ratio: example: spindle motor turns with 1000 rpm, spindle turns with 1500 rpm, meaning the factor is 1.5

3.6 Set origins

Now you have to set at least one **work piece origin**. Switch to the manual mode and move the tool so that the end of the tool is at that point of your workpiece where the origin should be. Pay attention that the length of the inserted tool has been measured.
Activate the function and choose the origin to set (1..6). The coordinates of the origin appear in the window where they can be edited. Now you can set the actual machine coordinates as the new origin by clicking the fields on the right. A green hook means that the actual machine value was inserted for the origin. The tool length is considered when calculating the z value for the origin.

3.7 Surface measurement

Surface measurement with mechanical unit

Here you configure the options for the **mechanical surface measurement**. The tool to do surface measurement is a tracing pin with mechanical switch.

Option scanning: is on if the green hook is set. Choose `tracing pin` or `laser` according to the equipment of your machine. Consider that a change in this setting only takes effect after leaving and reentering the machine interface.

Max. scanning points: sets the maximal count of scanning points that can be used in a program. If you choose this value too high, it takes a lot of memory. Calculation time is also higher if you have a lot of points in a program.

Z calculation: Choose the method for Z calculation with surface scanning

3 nearest points - plane: PrimCNC searches the 3 nearest points, puts a plane through the 3 points and takes as Z the value on the plane at position XY of the point to be moved to

3 nearest points - mean value: takes the mean value of the 3 nearest points

2 nearest points - mean value: takes the mean value of the 2 nearest points

nearest point = z value: take Z value of nearest point

Tracing pin:

X offset: Distance of tool scanning point from spindle axis in X direction

Y offset: Distance of tool scanning point from spindle axis in Y direction

Calibration point:

X: X coordinate of the calibration point

Y: Y coordinate of the calibration point

Z start: The tracing pin goes from Zmax to Zstart in rapid traverse and then in feed mode down for calibration. Zstart is set as deep as possible over the calibration point for time reasons.

z height: This value is set automatically by calibrating the mechanical tracing pin as described in the following section.

The tracing pin's length is measured before use like for every other tool. Because a fix surface is needed, it cannot be measured on the metal plate of the tool length measurement unit. The calibration point is normally set on the tool length measurement unit, but next to the metal plate.

Input: Input and logic used for tool measurement unit. For a isel 4.0/5.0 control you can use EA extension unit input 1. Default is bit 2. For the microstep controller, Input bit 5 is normally used, but you can use any input.

Output activate tracing pin: use this output if you have a switchable tracing pin. When used, the calibration of the tracing pin is done differently: the pin moves to the workpiece without having been measured in length before. The tracing pin length is never measured before doing surfacetracing because it is assumed to leave constant. Make sure you use a separate tool number for the tracing pin in the nc programm because it will otherwise overwrite the tool length of other tools. This tool number has also to be set in the macro cnc_reft.sub.

Tools can optionally be left in the spindle when doing surface tracing.

Calibrate mechanical tracing pin
To be useful, the mechanical tracing pin has to be calibrated first. This calibration does measure the effective distance from the calibration point to a known height on the working area of the isel machine. This distance is saved as Z height of the calibration point. The calibration procedure takes the following steps:

1. Insert a workpiece at the position of origin 1. Place the tracing pin to tool store 1 in the magazine. Place another tool to tool store 2 (if you have automatic tool changer).
2. Get tool No 2 by entering the command T2 M6. The tool length has to be measured if it is not valid.
3. Set now the Z value of origin 1 to the surface of the workpiece. To do this, switch to manual mode and go down in Z until the tool touches the workpiece. If the tool has touched the surface, go to Set origin and take the actual Z coordinate as Z origin 1. The tool length is automatically subtracted.

4. Enter the **Trickbox** and call the function

calibrate tracing pin. This function calls the macro CNC_REFT.SUB:

G54	Set origin 1
M35	Surface measurement follows
T1 M09	Insert mechanical tracing pin
M06	
G79 X20 Y10 Z20 F200	Measure Z height of point X=20,Y=10 with initial level of 20
G80	Reset surface measurement cycle
R0	Measured point is on Z=0mm, because we did set the origin there. We could also set R-1 to measure a point that is on Z=-1mm

The tracing pin is calibrated now. To get very exact values, you can mill a groove into the work piece (for example 0.4mm deep) and set R-0.4 in the macro.

Calibrate switchable mechanical tracing pin
If the machine has a switchable tracing pin, calibration is done by just moving to the workpiece without measuring the length of the pin before. The tracing pin length is never measured before tracing anymore. Take care that you use a separate tool number for the tracing pin, because else it would overwrite the tool length of other existing tools. This tool number has also to be written to CNC_REFT.SUB.

1. Set the parameters as described above.
- offset of tracing pin negative (-75 for example) if the tracing pin is installed left of the spindle
- configure input for tracing pin
- configure and activate output for zylinder of tracing pin
- Activate „Leave tool in spindle when tracing active“ if the milling tool should not be released for tracing

2. Insert milling tool by „T2 M6“. Measure tool length.

3. Set the milling tool to the surface of the workpiece by manual movement. Set the Z coordinate of the origin to 0 when the tool touches the surface.

4. Use the trickbox to „Calibrate the mechanical tracing pin“. The machine moves to the position defined in CNC_REFT.SUB and calibrates the tracing pin.
You probably have to set the z start of the measurement now so that it doesn't start at maximum z position, but a bit lower.

Surface measurement with laser

Here you configure the options for the **laser surface measurement**. The tool to do surface measurement is the laser distance sensor from isel, equipped with serial interface.

Option scanning: is on if the green hook is set. Choose `tracing pin` or `laser` according to the equipment of your machine. Consider that a change in this setting only takes effect after leaving and reentering the machine interface.

Max. scanning points: sets the maximal count of scanning points that can be used in a program. If you choose this value too high, it takes a lot of memory. Calculation time is also higher if you have a lotof points in a program.

Z calculation: Choose the method for Z calculation with surface scanning

3 nearest points - plane: PrimCNC searches the 3 nearest points, puts a plane through the 3 points and takes as Z the value on the plane at position XYof the point to be moved to

3 nearest points - mean value: takes the mean value of the 3 nearest points

2 nearest points - mean value: takes the mean value of the 2 nearest points

nearest point = z value: take Z value of nearest point

Laser:

X offset:	Distance of tool scanning point from spindle axis in X direction
Y offset:	Distance of tool scanning point from spindle axis in Y direction
Z offset:	Calibrated value of laser Z height

To measure X and Y offsets for the laser, switch to origin 1 by entering G54. Move manually in XY so that the red laser point is visible on the origin of the workpiece. Then you see the offsets in the display of the relative machine coordinates. All you have to do is change the signand enter the value for the offsets (Example:display: X=38.5 X offset=-38.5).

port: Serial port for the laser; COM1 and COM2 are normally used by mouse and cnc interface, so COM3 and COM4 can be used for the laser. Notice that some graphic cards in8514A standard cannot be used with COM4, leaving only COM 3 for the laser.

power: To switch off the laser when not in use, define the according output here.

Calibrate laser for surface measurements

The laser can be mounted on different heights of the spindle. This means the laser has to be calibrated before use, which is done the following way:

1. Take a workpiece that reflects well and gives exact measured values. Different plastics gave good results. Clamp the workpiece at the position of origin 1.
2. Insert a tool you can touch the surface with, for example a mill. Do this by entering the command T1 M6. You have to measure the length of the tool automatically (Trickbox) if it is not valid.
3. Set the Z component of the workpiece origin 1 exactly to the surface of the material. To do this, enter manual mode and touch the surface of the material. If the tool touched the surface, go to set origins and set the active machine Z coordiante as origin. The length of the tool is automatically considered when you do this.

4. Call the **trickbox** and choose the function

calibrate laser. This function calls the macro CNC_REFL.SUB:

```
G54                         Set origin 1
M35                         Surface measurement follows
T0 M09                      Choose laser
M06
G79 X20 Y10 Z20 F200        Measure Z height of point X=20,Y=10 with initial level 20
G80                         Reset surface measurement cycle
R0                          Measured point is on Z=0 mm, because we did set the origin there. We
                            could also set R-1 to measure a point that is on Z=-1mm
```

The laser is calibrated now. To get very exact values, you can mill a groove into the work piece (for example 0.4mm deep) and set R-0.4 in the macro.

Surface measurement procedure

Surface measurement allows to engrave on surfaces that are not plane. For surface measurement, you draw circles or points in PrimCAM whose Z height is measured afterwards. They can be measured using a tracing pin or a laser distance sensor. The Z value of the points is saved to a file named filename.PNT. If the workpiece is machined, the Z heigth of every point to be moved to is calculated by considering the measured points in the neighbourhood. The calculation formula is the following:

no traced points:	Origin is taken for Z height.
1 traced point:	Traced point is taken as Z value for origin
2 traced points:	Mean value of the two points is taken as Z height
3 or more points:	The three poins in the neighbourhood of the point to be calculated are searched. A plane is build by those 3 points. The X and Y coordinates of the point are inserted into the plane formula and give the Z height.

Write configuration to text file

Now you have done all the necessary configurations and calibrations. You can write the whole configuration of the CNC part to the readable text file PRIMCNC.TXT. This file can then be printed and stored for reference purposes.

3.8 I/O configuration

If you want to use additional digital **outputs** that are not supported by your control, you can configure them here. Parallel ports and the Isel multi I/O card are already configured. Just enter a name and the HEX address of your new card.
As soon as the card is set to „active“, it can be used in PrimCNC. You can configure a total of 10 cards, each with 8 input bits.

Do the same for your **input** cards.

Here you determine the Inputs / Outputs that are read/set for M codes in your NC program. An NC program generated by PrimCAM normally contains M codes for switching spindle / coolant on or off.
With PrimCNC, you can also use M codes to set arbitrary outputs or read inputs. M100-M195 are reserved for outputs at the moment, M200-M295 for inputs.
M100 sets output 1 to „1“, M101 resets it to „0“.
M200 waits for input 1 to be „1“, M201 waits for input 1 to be „0“.
These codes can be used to synchronize PrimCNC with external events from robots or other machines.

3.9 Using the interface program for the isel machine

To work with the isel machine, you have to build an NC program first. This is done with the icon

generate NC programm that you find in the **file functions**.

Also in the menu file functins, you find the icon for the **interface program**.

Normally you are already in automatic mode, where you can execute NC programs. But if you worked in manual mode before, switch to automatic mode now.

Now you are ready to execute the first NC program. If you built your program right before in PrimCAM, it is loaded automatically. But you can also use **load NC program** to select another program.

Run NC program starts the file located in memory. The currently executed NC code line is displayed in white in the NC code window. After every movement segment, coordinates, feed and spindle speed displays are updated. The flow of the program can be controlled by clicking the according icons or by using function keys (F2 = STOP, F3 = STEP).

The **Step mode** executes the next line of NC code and stops then. If you activate step mode during execution of a line, the line is finished and the program stopped then.

Stops a movement suddenly. A dialog window asks whether the movement should be breaked, finished or continued with the program. If you break a movement, you have to call the reference function.

The coolant functions serve to override the settings in the NC program. If you click again, you can terminate the override.

Function keys

PAUSE:EMERGENCY STOP Stops the machine immediately. Coolant and spindle are turned off. Because the isel control cannot return the correct axis position when this function is used, you must use the reference function.

If you pressed the red button on the isel control for emergency stop, you can use the PAUSE key to reset the program (the red button blocks communication with the PC and the program waits in an endless loop).

F1: HELP Calls the help system

F2: Program STOP Stops program execution immediately. Movements of all the axis are stopped. This function can be applied for example to clean the work piece during milling. The milling process can be continued by F3 (STEP) or F4 (RUN). PAUSE or the icon STOP abort the movement definitively, making execution of the reference function necessary. If you press F2 to stop and then F6 to finish the actual movement segment, you can move the tool manually. The tool returns automatically to the ending point of the last segment when you step or start again.

F3: Program STEP In step mode, the next line of the NC program gets executed with every key press (F3) or mouse klick. This can be useful to test the generated program and validate that it is error free. If you activate step mode during RUNning the program, the actual line is finished and the program stops.

F4: Program RUN starts the code located in memory. The line currently executed is displayed in white. Coordinates, spindle speed and feed are updated after every movement. By klicking the according icons or pressing function keys, you can stop (F2) or step (F3) through the program. The machinestops automatically when the program is finished.

F5: FEED- Decrements feed factor by 5%

F6: FEED+ Increments feed factor by 5%, up to 200%
F7: SPEED- Decrements spindle speed factor by 5%
F8: SPEED+ Increments spindle speed factor by 5%, up to 200%

Arrow keys,PgUp,PgDn Move stepwise or continuously in the manual mode

4. Reference PrimCNC

4.1 Commands and format of the NC code

NC code opens a window where you can enter NC code manually. The code is executed as if it came from an NC file loaded and run. This way you can force tool changes manually, move the machine or drill manually for example. The NC code used by the isel machines is based on the ISO code. The following inputs would be feasible:

G00 X0 Y0 Z100	Move to X=0, Y=0, Z=100 mm in rapid traverse measured from the origin actually set.
G55	Select origin 2 (G54=origin 1)
G01 X5 Y5 Z0	Move in feed mode to X=5, Y=5, Z=0
F1000	Set feed to 1000 mm/min
S3500	Set spindel speed to 3000 rpm and activate spindle
G02 ...	Arc clockwise
T10 M09 M06	Toolchange with new tool no. 10

Overview G functions

G00 Rapid traverse
G01 Move in feed mode
G02 Arc clockwise
G03 Arc counterclockwise
G04 Delay function
G17 spindle axis Z, circle plane XY, centre point I,J
G18 spindle axis Y, circle plane ZX, centre point K,I
G19 spindle axis X, circle plane YZ, centre point J,K

G22 Program jump
G28 Move to reference position

G53 Select absolute machine coordinates
G54 Select workpiece origin 1
G55 Select workpiece origin 2
G56 Select workpiece origin 3
G57 Select workpiece origin 4
G58 Select workpiece origin 5
G59 Select workpiece origin 6

G73 Drill cycle high speed peck
G76 Drill cycle ream
G79 Surface measurement cycle
G80 Cancle cycle
G81 Drill cycle feed/rapid traverse
G82 Drill cycle feed/rapid traverse with delay
G83 Drill cycle peck

G84 Tapping cycle
G85 Drill cycle feed/feed
G90 Absolute coordinates
G91 Incremental coordinates
G92 Set offsets to actual coordiante system
G93 Reset offsets of actual coordinate system
G99 Program jump back
G280 Set machine origin on controller

Overview M functions

M00 Program stop
M02 Program end: reset to line 1
M03 Spindle on (CW)
M04 Spindle on (CCW)
M05 Spindle off
M06 Tool change
M08 Coolant on
M09 Coolant off
M13 Spindle (CW) and coolant on
M14 Spindle (CCW) and coolant on
M20 Tool breakage test / tool length control (not implemented yet)
M28 Move to reference position
M30 Program end: coolant off, spindle off, drive to tool change point, reset to line 1
M35 Tracing pin is next tool

M100-M195 Set outputs

M100 output 1.1 = 1	M101 output 1.1 = 0 (output 1 bit 1)
M102 output 1.2 = 1	M103 output 1.2 = 0
M104 output 1.3 = 1	M105 output 1.3 = 0
M106 output 1.4 = 1	M107 output 1.4 = 0
M108 output 1.5 = 1	M109 output 1.5 = 0
M110 output 1.6 = 1	M111 output 1.6 = 0
M112 output 1.7 = 1	M113 output 1.7 = 0
M114 output 1.8 = 1	M115 output 1.8 = 0
M116 output 2.1 = 1	M117 output 2.1 = 0
M118 output 2.2 = 1	M119 output 2.2 = 0 …

M200-M295 Wait for inputs

M200 input 1.1 = 1	M201 input 1.1 = 0 (wait for input 1.1=?)
M202 input 1.2 = 1	M203 input 1.2 = 0
M204 input 1.3 = 1	M205 input 1.3 = 0
M206 input 1.4 = 1	M207 input 1.4 = 0
M208 input 1.5 = 1	M209 input 1.5 = 0
M210 input 1.6 = 1	M211 input 1.6 = 0
M212 input 1.7 = 1	M213 input 1.7 = 0
M214 input 1.8 = 1	M215 input 1.8 = 0
M216 input 2.1 = 1	M217 input 2.1 = 0
M218 input 2.2 = 1	M219 input 2.2 = 0 …

Description G functions

G00 rapid traverse
Moves in rapid traverse to the coordinates given. The coordinates are measured from the origin set (G54-G59). The speed is set in the parameters/rapid traverse.
G00 X20 Y35 Z5 Moves to X=20mm, Y=35mm, Z=5mm in rapid traverse

G01 Move in feed mode
Moves in feed mode to the coordinates given. The coordinates are measured from the origin set (G54-G59). The speed is set by the command F, multiplied by the factor in the feed override window
G01 X10 Y1 F300 Moves the machine to X=10, Y=1mm with a feed of 300 mm/min

G02 Arc clockwise
Mills an arc clockwise
I X coordinate of arc centre, measured from arc starting point
J Y coordinate of arc centre, measured from arc starting point
X X ending coordinate of arc
Y Y ending coordinate of arc
F [mm/min] feed for arc

G00 X50 Y100 Move to point X=50 Y=100 mm
G02 I0 J-50 X100 Y50 Arc from X=50,Y=100 to X=100,Y=50 with centre X=50-0, Y=100-50

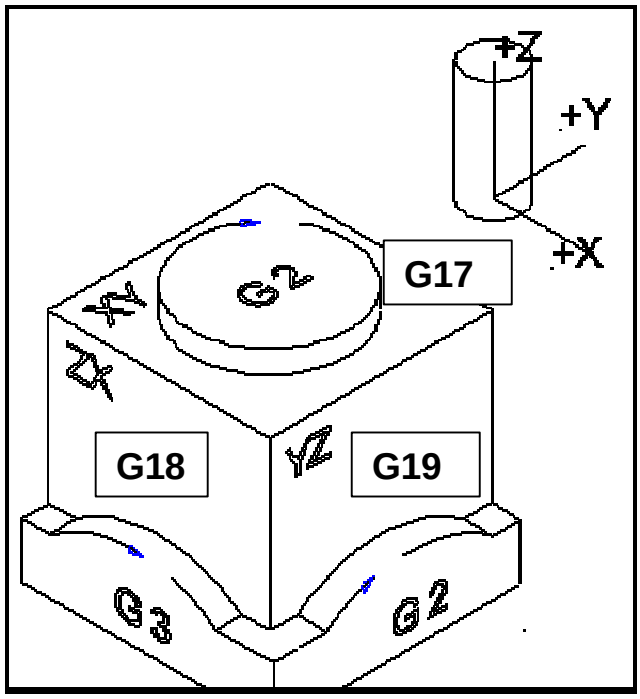
G03 Arc counterclockwise
Mills an arc counterclockwise
I X coordinate of arc centre, measured from arc starting point
J Y coordinate of arc centre, measured from arc starting point
X X ending coordinate of arc
Y Y ending coordinate of arc
F [mm/min] feed for arc

G00 X100 Y50 Move to point X=100 Y=50 mm
G03 I-50 J0 X50 Y100 Arc from X=100,Y=50 to X=50,Y=100 with centre X=100-50, Y=50-0

G04 Delay function
This function realizes a delay that lasts P milliseconds. It can be used for example to wait for valves to switch.
P Wartezeit in ms

G04 P300 Pause program for 300ms

G17 circle plane XY, spindle axis Z
G18 circle plane ZX, spindle axis Y
G18 circle plane YZ, spindle axis X
Sets the working plane for circles and radius compensation.



G22 Program jump

Allows conditional (on input) or unconditional jumps.The jumps are not cascadable

J Jump target line number N

(xx.nc) Jump to a subprogram xxx.nc.
The subprograms can't contain G22 (jumps) or G99 (jumps back)!

M optional: input for jump; corresponds with port configuration for m codes.

G22 M200 J55 If input 1.1=1 jump to line N55

G22 M201 J55 If input 1.1=0 jump to line N55

G22 M201 (prog1.nc) If input 1.1=0 call subprogram prog1.nc in the primcam directory. Please note that you can't place a comment (…) in a line like this!

G22 M201 (primcnc\\prog1.nc)

G22 M201 (c:\\primcam\\prog1.nc)

G99 Jump back from subprogram to the line following the calling line

G28 Reference drive

Moves the axes to the reference position. The sequence for the axis ist taken from parameters / machine parameters.

G53 Select absolute machine coordinates

This function resets the selected origin and switches to absolute machine coordinates. The machine zero point is adjusted with every reference drive. After the reference drive, the machine is on position X=0, Y=ymax, Z=zmax. The machine zero point is at X=0, Y=0, Z=0, which means in front-left-down.

G54-G59 Select workpiece origin

PrimCNC offers up to six different workpiece origins. It is possible to clamp several workpieces and mill them with the same program. The offsets for the different workpieces are taken from the workpiece origins.

G54 Selects origin 1

G73 Drill cycle high speed peck
In this cycle, the tool is lifted during the drilling process to break the chip.

Parameters:
R Clearance distance: The drilling tool starts moving in feed mode from here
Q Specifies the depth of cut at each time. Then the tool is retracted a bit to break the chip.
F drilling feed in mm/min

Example:

G00 X10 Y10 Z20	Prepositioning on initial level at X=10,Y=10
S3000 M13	Spindle and Coolant on, 3000 rpm
G73 X10 Y10 Z-15 R1 Q2 F240	Define and call cycle at X10,Y10
X20 Y20	Drill at X20 Y20
X30 Y30	Drill at X30 Y30
G80	Cancle drilling cycle

G76 Reaming / fine boring cycle
This cycle is not supported by the isel machine

G79 Surface measurement cycle
Look at surface measurement for a detailed description of surface measurement and calculation formulas used.

Parameters:
R Clearance distance: The tracing pin moves in rapid traverse rate to the clearance distance and then starts moving in slow feed. The clearance distance is relative to the surface of the workpiece, determined with the first measured point. You have to be sure that the workpiece is nowhere higher then 1st scanned point + clearance distance.

M35	Surface measurement follows (important to measure tool lenght with tracing pin)
T1 M09 M06	Insert tool no 1
S0 M3	(not considered for tracing pin)
G79 X10 Y10 Z20R1 D	efine and call cycle at X10,Y10 with initial level 20
X20 Y20	Measure at X20 Y20
X30 Y30	Measure at X30 Y30
G80	Cancle cycle

G80 Cancle cycle
Switches off the cycle that's active at the moment

G81 Drill cycle feed/rapid traverse
Drives into the hole with feed rate and out in rapid traverse.

R Clearance distance: The tool moves in rapid traverse rate to the clearance distance and then starts moving in slow feed.

G00 X10 Y10 Z20	Preposition to inital level at X10,Y10
S3000 M13	Spindle and coolant on, 3000 rpm
G81 X10 Y10 Z-15 R1 F240	Define and call cycle at X10,Y10
X20 Y20	Drill at X20 Y20
X30 Y30	Drill at X30 Y30
G80	Cancle cycle

G82 Drill cycle feed/rapid traverse with delay
Corresponds to the G81 Drill cycle feed/rapid traverse with a delay added at the bottom of the hole.

P Delay time at bottom of hole in ms
R Clearance distance: The tool moves inrapid traverse rate to the clearance distance and then starts moving in slow feed.

G00 X10 Y10 Z20	Preposition to inital level at X10,Y10
S3000 M13	Spindle and coolant on, 3000 rpm
G82 X10 Y10 Z-15 R1 P500 F240	Define and call cycle at X10,Y10
X20 Y20	Drill at X20 Y20
X30 Y30	Drill at X30 Y30
G80	Cancle cycle

G83 Drill cycle peck
For the peck drill cycle, the tool is boring in several steps by the value Q. Between the steps, it is retracted out of the hole to bring the chips out.

R Clearance distance: The drilling tool starts moving in feed mode from here
Q Specifies the depth of cut at each time. Then the tool is retracted out of the hole a bit to break the chip.

G00 X10 Y10 Z20	Preposition to initial level at X10,Y10
S3000 M13	Spindle and coolant on, 3000 Rpm
G83 X10 Y10 Z-15 R1 Q2 F240	Define and call cycle at bei X10,Y10
X20 Y20	Drill at X20 Y20
X30 Y30	Drill at X30 Y30
G80	Cancle cycle

G84 Tapping cycle
The tapping cycle is not available with isel machines.

G85 Drill cycle feed/feed
Corresponds to the drill cycle feed/rapid traverse G81, except that the tool is retracted from the hole in feed mode.

G90 Absolute coordinates
Coordinates are now interpreted as absolute values (I and J for arcs are always measured incrementally from the arc starting point).

G91 Incremental coordinates
Coordinates are now interpreted as incremental values measured from the current position.

G92 Set offsets to actual coordiante system
G92 allows to set manual offsets to the actual workpiece coordinates. The following movements in the program are done with the coordinates set by G92. It works like a additional offset that can be reset by G93.

G00 X100 Y20 Z0	Move to position 100,20,0
G92 X0 Y100 Z0	The x axis stands at 100 and is set to 0
	The y axis stands at 20 and is set to 100
	The z axis stands at 0 and is set to 0

G93 Reset offsets of actual coordinate system
The offsets set with G92 are reset to 0.

G99 Program jump back
Jumps back from a subroutine to the calling position.

G280 Set machine origin on controller (only for I332 controller)
Sets the absolute machine origins on the controller to new values.

G280 X100 Y20	Sets the machine origin so thatthe actual position gets X=100,Y=20

Description M functions

M00 Program stop
A dialog window appears on the screen, asking whether the program should be continued or canceled. The function can be used for example to clean the workpiece between different jobs.

M02 Program end
Opposed to M30, the NC program is only reset here to line 1.

M03 Spindle on (CW)
Starts the spindle clockwise, using the spindle speed given with S.., eventually overridden by the factor set in the speed override window.

M04 Spindle on (CCW)
Starts the spindle counterclockwise, using the spindle speed given with S.., eventually overridden by the factor set in the speed override window.

M05 Spindle off
Stops the spindle.

M06 Tool change
Executes, according to the equipement of the machine, a manual or automatic tool change.

For the manual tool change, the machine moves to the position set at the tool change point M and shows a dialog window, requesting you to insert a new tool.
If the automatic tool changer is installed, the old toolis returned to its place and the new tool is fetched.

M08 Coolant on
Switches the coolant on

M09 Coolant off
Switches the coolant off

M13 Spindle (CW) and coolant on
Switches spindle and coolant on.

M14 Spindle (CCW) and coolant on
Switches spindle and coolant on.

M20 Tool breakage test / tool length control (not implemented yet)
This function is only available if the automatic tool lenght measurement unit is installed. The tool can be tested for lenght or whether it's broken in the middle of the program. The system
test whether the actual tool length and the one measured before differ more than the value set in the parameters. If this is the case, tool breakage is assumed and the program stops.
This function can also be used to test the tool for abrasion. Therefore, you set the tolerance value in the parameters relatively small.

M28 Reference drive
Executes a reference drive. Spindle and coolant are stopped. The function is used to initialize or zero the axis when a stepping motor may have lost somesteps.

M30 Program end
Spindle and coolant are switched off, the machine moves to the tool length measurement point. The NC program is reset to line 1.

M35 Tacing pin is next tool
M35 is executed before surface measurement. If there exists data for the actual program name (*.PNT), PrimCNC asks whether the piece should be measured again. If the data is loaded from disk, the tracing pin is not inserted and the NC code for surface measurement is skipped.

M100-M195 Switch outputs
These codes switch the outputs of the EA extension unit on/off. They can be used for example to turn on a vacuum cleaner when moving from ZA to the workpiece and switch it off again when moving up from the piece to ZA.

M200-M295 Wait for inputs
These codes allow your NC program to wait for events from the outside. When M200 is placed in your program for example, it stopps and waits until EA input 1 goes to high. With STEP you can step over this waiting.

File names PrimCNC

PRIMCNC .CFG	configuration file for CNC
CNC_BLCK.CFG	definition of forbidden areas
CNC_END.SUB	macro that is called at the program end
CNC_TOCO.CFG	definition of tool change points
CNC_WPCO.CFG	definition of workpiece origins
CNC_INIT.SUB	macro for initialization, called before reference
CNC_REFL.SUB	macro for calibration of laser surface measurement unit
CNC_REFT.SUB	macro for calibration of mechanical tracing pin
CNC_REFO.SUB	macro to move out of forbidden area after too change
CNC_START.SUB	macro that is called before program start
CNC_WWPI.SUB	macro to drive into tool changer (place tool)
CNC_WWPO.SUB	macro to drive out of the tool changer (place tool)
CNC_WWGI.SUB	macro to drive into tool changer (get new tool)
CNC_WWGO.SUB	macro to drive out of the tool changer (get new tool)
PRIMCNC .TXT	text file containing configuration settings, in readable text form

5. Appendix

5.1 Axis controllers

5.1.1 SYSTEC MCM Step/Servo

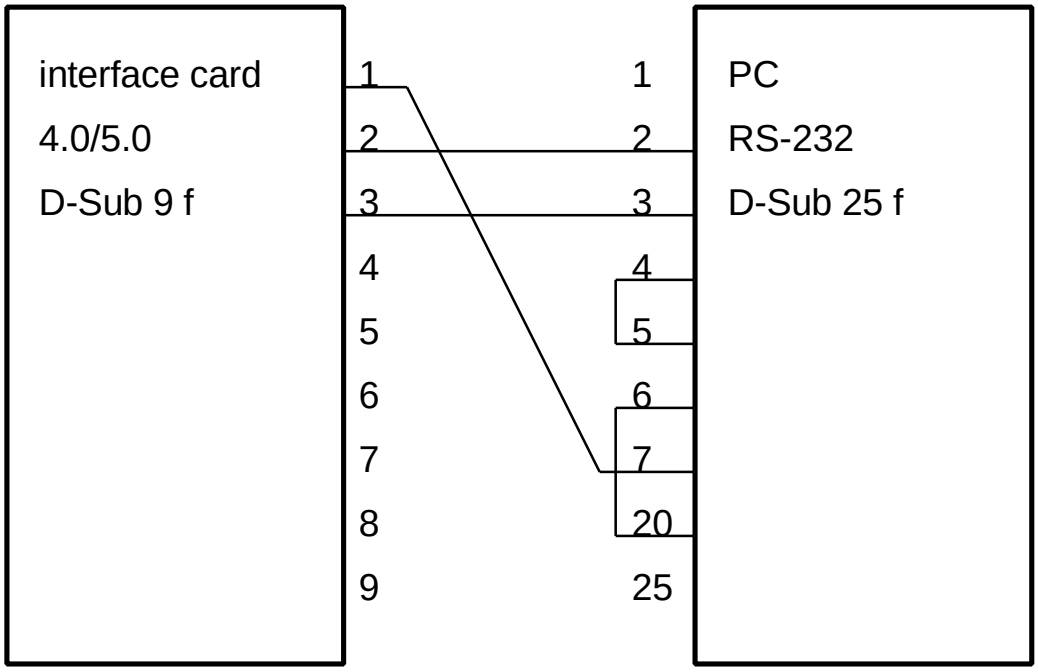
At the moment the systec MCM controller is only described in the german version of the manual.

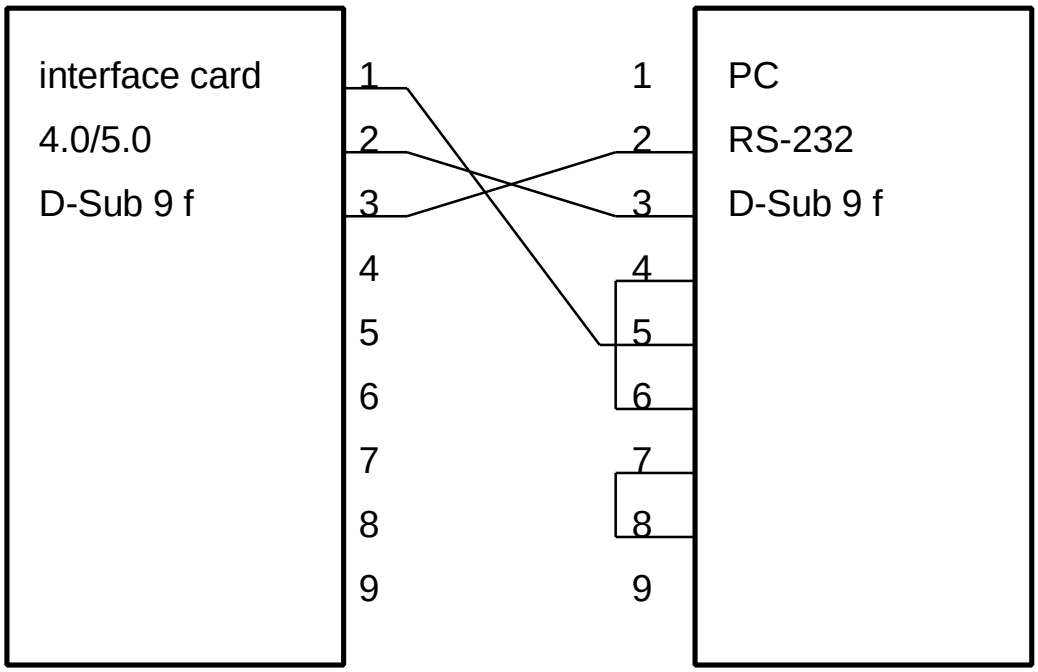
5.1.2 Isel 4.0/5.0/IMC4 Interfacekarte

Important information

- This controller does not need any drivers from isel.
- IMC4: tool length measurement / surface tracing with EA input is possible with PrimCAM 2.71 and later. It needs controller Eprom V.2.2.01/19.03.98 or later
- IMC4: If you see the message "Isel control: command to store incorrect" when doing reference, check the power switch on the controller (pressed and light on!)
- 4.0/5.0 controllers can use EA input for tool length measurement / surface tracing
- Inputs for emergency, start, stop and step can't be set to EA inputs because the controller does not check them while running a movement.
- For circles/arcs to be done correctly, the step width (calculated from steps/round, elevation and gear) must be the same for circle axes (X,Y)

Connection to serial port (isel 4.0/5.0 interface card)





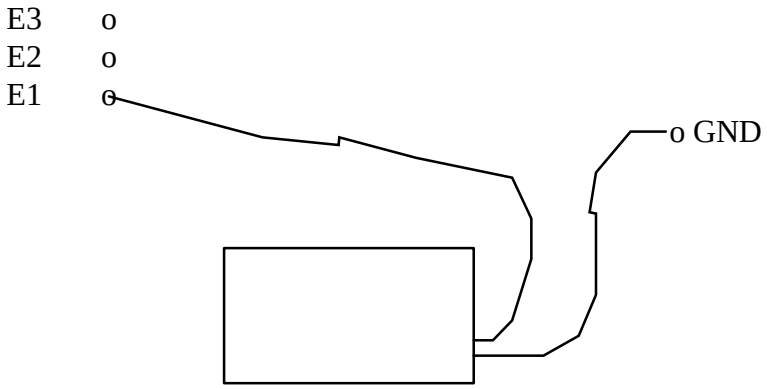
possible connection of inputs/outputs for EA Extension unit Isel 4.0/5.0

A1.1 Coolant
A1.2 Vacuum cleaner
A1.3 Spindle on/off
A1.4
A1.5 Laser on
A1.6 Tool changer cover
A1.7 Tool changer clamp
A1.8 Tool changer clamp (inverted)
A2 Spindle rpm control

E1 Tool length measurement unit
E2 Tracing pin
E3 Spindle stopped
E4
E5
E6
E7 Pressure ok
E8 Tool change cover open

Connecting the tool length measurement unit to EA extension unit

+Vs o——————————————————o +24V
E8 o
E7 o
E6 o
E5 o
E4 o

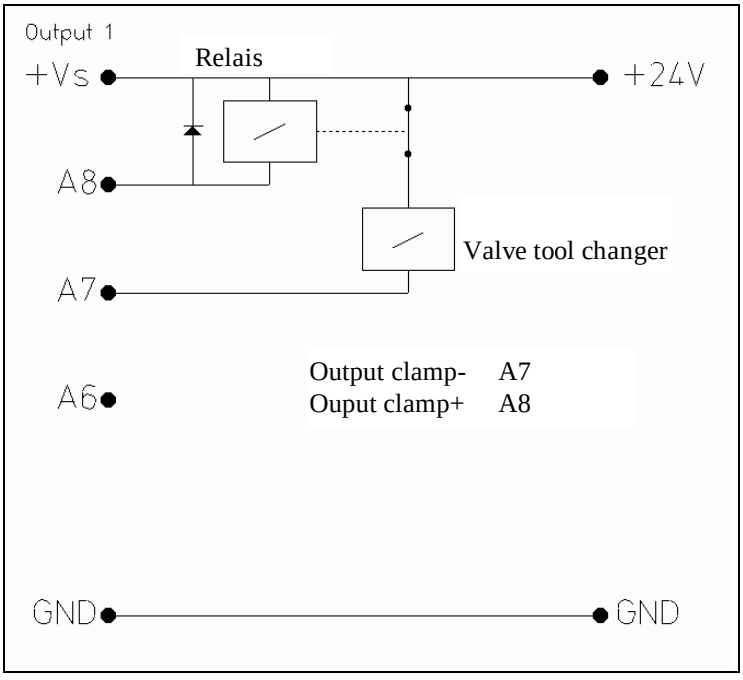


Spindle controller ISA300 with EA extension unit

If you use EA extension unit to control an ISA300 spindle controller, you must replace the integrated transistor array ULN2803A of the EA against a enclosed R-net 8*180R.

Relais circuit for automatic tool changer

If you drive the tool changer using Output 1 of the isel EA extention unit, there is the following problem: Either using the emergency stop button or switching the control on activates all outputs for a very short time. This time is long enough to release the clamped tool. To prevent this behaviour, the following circuit can be used:



This circuit causes the valve for clamping the tool is only activated if outputs A7 and A8 have different states, which is not the case when pressing the emergency stop or switching the controller on

5.1.3 Isel MPK3 microstep card

important information

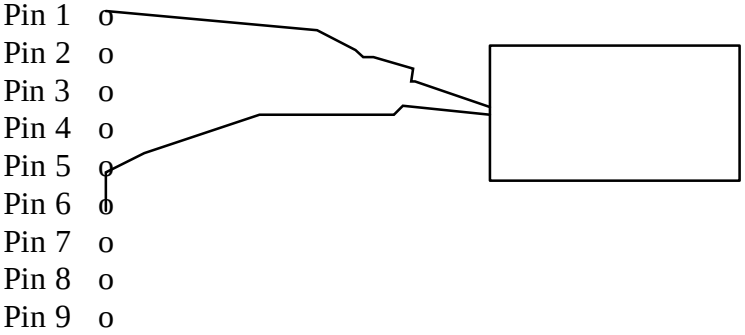
• This controller does not need any drivers from isel. Don't installMKP3DRV.
• Runs with the built in driver (Win95, Win98)
or an additional Mtasc driver license PP (Win98, WinNT, Win2000, WinXP)
• Machine can move in buffer mode(without stops between elements)
• Take care when installing that the installation path has directories with maximum 8 letters and that it doesn't contain ä, ö, ü or spaces. The driver MTDRV.VXD can not be written to the SYSTEM.INI if this is the case.
• Restart the PC after installation

Input/Outputs Isel Microstep

MS output relais	bit 2	Coolant
MS input	bit 4	Tool length measurement unit
	5	tracing pin
	6	free

Connection of the tool length measurement unit

DB9



Deleting unused drivers from the system (Win95/98)

• delete the line with mit device=..\mtdrv.vxd in windowsdirectory\system.ini

Deleting unused drivers from the system (Win98,NT,2000,XP with additional driver PP)

Win98
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete windowsdirectory \system\mtdrvx.vxd
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration
•:delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\VxD\MTDRV]

WinNT
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration

5.1.4 Movtec PCSM-300

important information

• Runs with the built in driver (Win95, Win98)
or an additional Mtasc driver license MM (Win98, WinNT, Win2000, WinXP)
• Machine can move in buffer mode (without stops between elements)
• Take care when installing that the installation path has directories with maximum 8 letters and that it doesn't contain ä, ö, ü or spaces. The driver MTDRV.VXD can not be written to the SYSTEM.INI if this is the case.
• Restart the PC after installation

Deleting unused drivers from the system (Win95/98)

• delete the line with mit device=..\mtdrv.vxd in windowsdirectory\system.ini

Deleting unused drivers from the system (Win98,NT,2000,XP with additional driver PP)

Win98
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete windowsdirectory \system\mtdrvx.vxd
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration
•:delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\VxD\MTDRV]

WinNT
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration

5.1.5 pulse direction output to parallel ports

important information

• This controller does not need any drivers from isel
• Runs with the built in driver (Win95, Win98)
or an additional Mtasc driver license CC (Win98, WinNT, Win2000, WinXP)
• Machine can move in buffer mode (without stops between elements)
• pulse and direction appear on the port at the same time. This must be supported by the motor dirvers (The isel adapter card for example needs direction 4us before pulse)
• The output of pulse/direction can go to any parallel port
• Take care when installing that the installation path has directories with maximum 8 letters and that it doesn't contain ä, ö, ü or spaces. The driver MTDRV.VXD can not be written to the SYSTEM.INI if this is the case.
• Restart the PC after installation

input/output bits (example base address 0x278, parallel port)

DB25	adress/bit	PC	PrimCNC	Description
1	0x27A/1	A ~Strobe	(z.B. coolant)	

2	0x278/1	A Data 1	Xstep	step X
3	0x278/2	A Data 2	Xdir	direction X
4	0x278/3	A Data 3	Ystep	step Y
5	0x278/4	A Data 4	Ydir	direction Y
6	0x278/5	A Data 5	Zstep	step Z
7	0x278/6	A Data 6	Zdir	direction Z
8	0x278/7	A Data 7	Astep	step A
9	0x278/8	A Data 8	Adir	direction A
10	0x279/7	E Ack	Zend	end switch Z
11	0x279/8	E ~Busy	(for ex. software emergency)	
12	0x279/6	E Pe	Yend	end switch Y
13	0x279/5	E Select	Xend	end switch X
14	0x27A/2	A ~Autofeed		
15	0x279/4	E Error	Aend/tool length m.	end switch A
16	0x27A/3	A Init		
17	0x27A/4	A ~Select	(for example spindle)	
18-25		GND	ground	

pulse / direction signals are always go always to the base address. End and ref switches are seperately defined in the parameters.

Example output from PC parallel to isel adapter card with C142

PC parallel	isel adapter card
2	19
3	18
4	17
5	16
6	15
7	14
10	3
12	5
13	4
25	1

Deleting unused drivers from the system (Win95/98)

• delete the line with mit device=..\mtdrv.vxd in windowsdirectory\system.ini

Deleting unused drivers from the system (Win98,NT,2000,XP with additional driver CC)

Win98
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete windowsdirectory \system\mtdrvx.vxd
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration
•:delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\VxD\MTDRV]

WinNT
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration

5.1.6 Isel UPMV4/12 servo controller with isel servo driver WinUPM

Important information

• Needs a license for isel driverWinUPMV4/12
• Machine can't do continuous movement (stops between elements)
• runs with Win95,98 (not NT,2000)
• Software emergency Stopp does not run during reference. Use the emergenca button on the controller
• X must be 1st axis, Y 2nd axis for circles to run

Installation

• install isel driver WinUPMV4/12
• use Parkon/Parein according to isel manuals to get machine running
• determine PID parameters using these programs✍ servo.ini
• install PrimCAM, controller „isel servo (Win95,98:needs isel WinUPMV4/12 License)"
• copy servo.ini to the PrimCAM subdirectory \primcnc
• copy an actual servo32.dll from the isel directory to the PrimCAM directory

Deleting unused drivers out of the system

• delete in the Windows directory c:\windows\winstart.bat the line c:\windows\system\iseldrv.exe

5.1.7 Isel UPMV4/12 servo controller with Trimeta driver MTasc

Important information

• This controller needs no drivers from isel.
• Needs a license for the Mtasc UPM driver from Trimeta
• Machine can do continuous movements (without stop between elements)
• Runs with Win98,NT,2000 (not 95)
• After installation, the PC has to be restarted

Installation

• install PrimCAM, controller „isel Servo (Win98,NT,2000:needs MTasc License UPM)"
• PAR / interface: set MTASC driver code= (license code from Trimeta)
• PAR / save to „primcnc.txt" ✍ drv_upm.ini is written
• PAR / machine parameters / reference switch:
    - Axis1: UPMV4/12 End. (BA+0E) BitNr. 1 Aktiv 0(low)
    - Axis2: UPMV4/12 End. (BA+0E) BitNr. 2 Aktiv 0(low)
    - Axis3: UPMV4/12 End. (BA+0E) BitNr. 7 Aktiv 0(low)
    - Axis4: UPMV4/12 End. (BA+0E) BitNr. 4 Aktiv 0(low)
    • Depending on the machine, you have to choose UPMV4/12 Ref. (BA+0C) or other bits.
• define if you have additional end switches and whether should be bridged during reference.
• if you want, you can change the parameters kp,ki,kd,li in primcnc\upminit.ts. These are determinded using parkon/parein.exe and can be set individually for every axis. But normally, the machine runs with the default values.

The values PAR / machine parameters / Isel Servo PID parameters should not be changed. Leave them at P=1'000'000, I=0, D=11'000'000.

Deleting unused drivers from the system (Win98,NT,2000,XP with additional driver UPM)

Win98
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete windowsdirectory \system\mtdrvx.vxd
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration
•:delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\VxD\MTDRV]

WinNT
• delete windowsdirectory \system32\drivers\mtdrv.sys
• delete [HKEY_LOCAL_MACHINE\System\CurrentControlSet\Services\Mtdrv] in registration

5.1.8 Knickmeier I332 controller card

important information

• supports up to 8 stepper/servo axes
• runs very quiet

Installation

• Windows 95/98/ME: the controller runs with the enclosed DLL.
For the PCI version of the I332, you can answer the questions of the hardware manager with ok
If the message „I332.DLL cannot be loaded“ appears when referencing, you have to install DCOM95.EXE or DCOM98.EXE (www.microsoft.com) first.

• Windows NT (Admin-Account): the controller runs with the enclosed driver
• Windows NT (Normaler Account): the driver for the controller must be installed first from an account with administator rights. This is done by calling Inst_332_NT.EXE in the PrimCAM directory

• Windows 2000 with PCI card: is recognized automatically by the hardware manager
- search for driver
- select GWIOPM.INF in PrimCAM directory
• Windows 2000 with ISA card: Install the driver first by calling Inst_332_2K.EXE in the PrimCAM directory

• Windows XP with PCI card: is recognized automatically by the hardware manager
- search for driver
- select GWIOPM.INF in PrimCAM directory
- „not MS signed“ can be ignored

• Consider the following when using circular axes:
- max. travel: Min. 0, Max. 360
- use the end switches ES+/- only as reference swiches for the circular axes.
- so you don’t have to bridge the switches during normal operation

5.1.9 vhf CNC750

important information

• www.vhf.de: more infos and manual
• please note: Origin in PrimCAM is left(X)– front(Y) – down(Z)
(Origin vhf is left(X)– back(Y) – up(Z))
• The reference always goes to the negative axis direction– changing the axis direction also changes the reference direction
• The 1st reference calculates the ramp parameters on the controller, taking about 7s. Subsequent references don't calculate them anymore, unless you changed the parameters for the ramp.

5.2 Spindle controllers

Here you determine how PrimCAM sets the appropriate spindle speed. Use the appendix to configure specific spindle controllers.

manual: Opens a window for every spindle speed change indicating the new spindle speed. After confirmation, the program is continued.

1 bit on/off: Simple on/of control of a spindle with no automatic setting of spindle speed. PrimCNC uses the bit defined in the interface/output field. You can use the according output to drive a relais that switches the spindle on/off. You can also set a waiting time in [s] that pauses the programm until the spindle has reached its final speed.

isel ISM 300: Isel ISM 300 spindle controller, speed regulated by a parallel output

isel FC 1.2: Isel FC 1.2 frequency converter, speed regulated by a parallel output

isel FC 1200: Isel frequency converter FC 1200/2200, speed regulated by a serial I/O card PA730 (20mA TTY).

isel FC 1200is: Isel frequency converter FC1200/2200is connected to a serial RS232 port.

Knickmeier DC3: Spindle controller driven by a parallel port. The speed increasing ration is 3 by default, because the spindle turns with 30000 rpm, driven by a cogged belt when the motor turns with 10000 rpm.

Kavo EWL4444: Kavo spindle controller, driven by an analog output of the isel multi-I/O card. Start- and stopbits can be realized with the isel multi-I/O output 1. Reading the inputs rpm ok and error is only possible when the multi-I/O card has 12V inputs. Otherwise you can also use the waiting time.

8 Bit digital: User configurable spindle controller with parallel output. The bits for start, stop and direction can be set arbitrary. If they are set to the same output byte as the speedbyte, the speedbits are distributed to the remaining bits, starting with the highest bit. You can optionally use inputs for rpm ok and error. If not used, set them to 'Not defined'. The speedbyte is divided linear to the region between rpm min. and rpm max.

Interface:

Rpm min./max.: The output speed byte is divided linear to the range between min. and max. rpm (8 Bit controller).

Output: Here you can set the Output used to control the spindle. Please notice that if you use a parallel port for spindle control, the spindle may be switched on when communication with the dongle takes place on another parallel port. The dongle is searched for in the order HEX 378, 278, 3BC. If you place the dongle before the spindle controller in search order, the dongle doesn't interfere with the control. Output start, stop, direction: special outputs for Kavo and 8 Bit.

Input rpm ok, error: special inputs for Kavo and 8 Bit.

COM/Baud/Adress: serial port, baud rate and modul address (FC1200,default 32) of the frequency converter.

Waiting time: Accelerating the spindle motor takes some time. Here you define how many seconds (On/Off controller) or how many seconds per 1000 rpm the program should wait before continuing.

Speed increasing ratio: example: spindle motor turns with 1000 rpm, spindle turns with 1500 rpm, meaning the factor is 1.5

Isel spindle controller ISA 300

Using the ISA 300 spindle controller, the rpm is set automatically. This is done by setting the appropriate bits at the output. If you use the ISM-300 with the ISEL EA extension unit, your dealer has to insert a R-net for OUT2 first.

The cable between the EA extension unit and the controller is wired 1:1 as follows:

DSUB 9 female Isel EA extension unit OUT2			DSUB 9 female ISE 300 Signal Input	
1	A1	_____	1 E0	speed Bit 0
2	A3	_____	2 E2	speed Bit 2
3	A5	_____	3 E4	speed Bit 4
4	A7	_____	4 E6	(valve control: optional)
5	GND	_____	5 GND	
6	A2	_____	6 E1	speed Bit 1
7	A4	_____	7 E3	speed Bit 3
8	A6	_____	8 E5	speed Bit 5
9	A8	_____	9 E7	direction

If you use the ISM300 with a parallel PC port (LPT), use the following cable:

DSUB 25 male Parallel port	DSUB 9 female ISE 300 Signal Input

2 Bit 1	———	1 E0 speed Bit 0
4 Bit 3	———	2 E2 speed Bit 2
6 Bit 5	———	3 E4 speed Bit 4
8 Bit 7	———	4 E6 (valve control: optional)
24 GND	———	5 GND
3 Bit 2	———	6 E1 speed Bit 1
5 Bit 4	———	7 E3 speed Bit 3
7 Bit 6	———	8 E5 speed Bit 5
9 Bit 8	———	9 E7 direction

Isel frequency controller FC 1.2

Using the FC 1.2 frequency controller, the rpm is set automatically. This is done by setting the appropriate bits at the output. If you use the FC 1.2 with the isel EA extension unit, there is no Rnet necessary as for the ISM 300.

The cable between the EA extension unit and the controller is wired as follows:

DSUB 9 female		DSUB 15 male
Isel EA extension unit		Frequency controller FC1.2 digital input
OUT 2 1 A1	———	1 bit 1
2 A3	———	2 bit 3
3 A5	———	3 bit 5
4 A7	———	4 bit 7 (direction bit)
5 GND	———	13=8 GND
6 A2	———	9 bit 2
7 A4	———	10 bit 4
8 A6	———	11 bit 6
9 A8	———	12 bit 8 (enable)

If you use the FC 1.2 with the isel microstep controller, you can use the isel multi i/o card to control the spindle speed:

DSUB 37 male		DSUB 15 male
Isel multi i/o card		frequency controller FC1.2 digital input
22 bit 1	———	1 bit 1
23 bit 3	———	2 bit 3
24 bit 5	———	3 bit 5
25 bit 7	———	4 bit 7 (direction)
10 GND	———	13=8 GND
4 bit 2	———	9 bit 2
5 bit 4	———	10 bit 4
6 bit 6	———	11 bit 6
7 bit 8	———	12 bit 8 (enable)

If you want to use other output cards for spindle control, you can configure parallel output cards here and make them available for PrimCNC. Just enter an arbitrary name and the HEX address of the card, then switch it to active. For the isel multi i/o card at base address 340

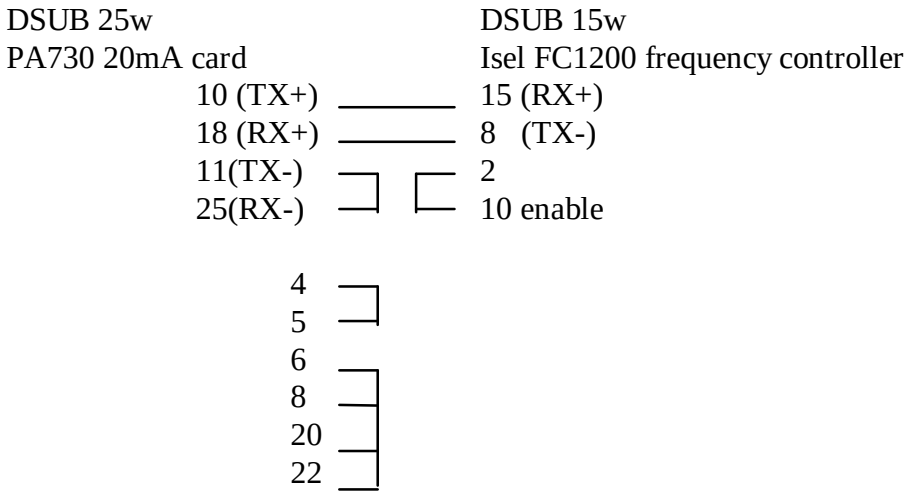
HEX, you would enter for example address 346 HEX for the output.

You can configure input cards the same way. For the isel multi i/o card at base address 340 HEX, you enter address 344 HEX for input1 and 346 HEX for input2.

Isel frequency controller FC 1200

The FC1200 is controlled by a 20mA serial port of the PC. Normal PC serial ports don't support 20mA current loops. You can order a compatible current loop serial port card (PA 730) from PRIMUS DATA. When using the Isel 5.0 controller (also using a serial port), we recommend the following configuration:

COM1:	PA730 20mA-Current loop card für FC1200
	Jumper 12-13: IRQ4
	Jumper 14 : open! (COM1)
	Jumper 4-5 : Current Loop
	Jumper 7-9 : +5V Power Supply for Current Loop
	Jumper 10 : Internal Ground to CL
	Jumper 15 : Terminal Resistor to RS422 Receiver Line
COM2:	Serial port for mouse, IRQ3
COM3:	Serial port for Isel 5.0 controller, IRQ4



Isel frequency converter FC1200is

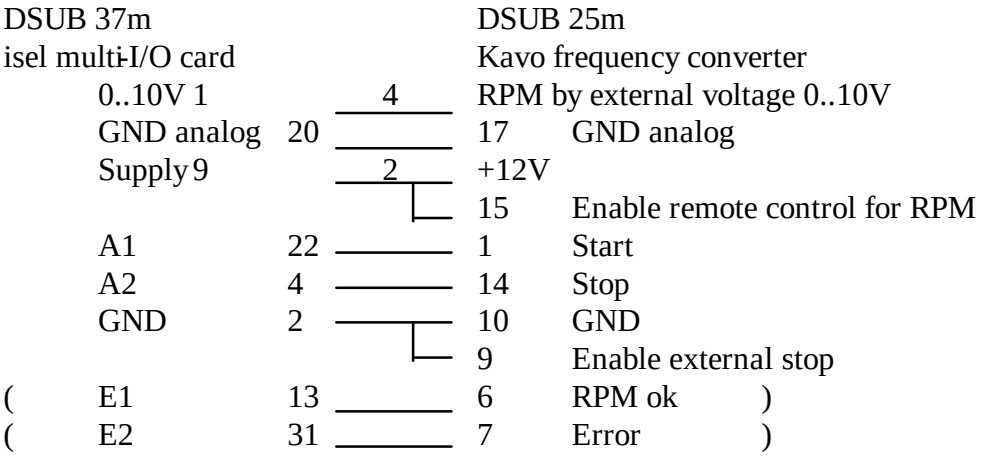
The isel frequency converter is connected to the PC by a serial RS232 port. Take care that the FC1200is and the mouse are not on the same interrupt (COM 1 and COM 3 have interrupt 4, COM2 and COM 4 both have interrupt 3.

Knickmeier DC3

The DC3 is connected to the PC the same way as the ISA300.

Kavo EWL4444

The Kavo is connected to the isel multi-I/O card:



Configure the outputs the following way:

Output multi-I/O AD	: multi-I/O Out1
Rpm min./max.:	Insert the values obtained by manually testing the min. and max. Rpm of the controller without using PrimCAM Default values: 5000 - 51000
Output Start	: multi-I/O out1 bit1 1(high)
Output Stop	: multi-I/O out1 bit2 0(low)
Input RPM ok	: not defined
Input Error	: not defined

5.3 I/O cards

Pins of the parallel PC port (example: parallel 0x278)

Pin	name	addr/bit	output/input	
1	strobe	27A/1	O	(inverted)
2	data0	278/1	IO	
3	data1	278/2	IO	
4	data2	278/3	IO	
5	data3	278/4	IO	
6	data4	278/5	IO	
7	data5	278/6	IO	
8	data6	278/7	IO	
9	data7	278/8	IO	
10	acknowledge	279/7	I	
11	busy	279/8	I	(inverted, but turned normal by software)
12	paper empty	279/6	I	
13	select	279/5	I	
14	autofeed	27A/2	O	(inverted)
15	error	279/4	I	
16	init	27A/3	O	
17	select input	27A/4	O	(inverted)
18-25	GND			

Please note: PrimCAM version before 2.0b37 shift the bits of input 0x279, 0x379, 0x3BD 4 bits to the left, meaning for example pin 11 is bit 4 (This was necessary for reading the panel with not bidirectional ports.) Later versions correspond to the bit assignment above.

Please note that the software protection key on the parallel port doesn't connect all the pins.
The twin-dongle (grey-green, can be used on parallel and serial ports doesn't connect pin 11 (BUSY) and pin 22 (GND).
The normal dongle (black, only usable on parallel port) doesn't connect pin 11 (BUSY) and pin 25 (GND).

Wasco OPTORE-PCI16

Install the card first in the PC. Install then the drivers according to the card manual. PrimCNC then sees the card and puts the I/O addresses in the inputs/outputs to be selected.

Isel multi i/o card

Pay attention to the pin assignment of the isel multi i/o card (erroreous in the isel documentation)

Input byte 1: 344 HEX (base address 340 HEX)

- Bit 1 pin 13
- Bit 2 pin 31
- Bit 3 pin 12
- Bit 4 pin 30
- Bit 5 pin 15
- Bit 6 pin 33
- Bit 7 pin 14
- Bit 8 pin 32

Input byte 2: 346 HEX (base address 340 HEX)

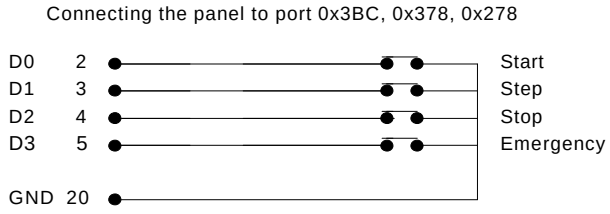
- Bit 1 pin 34
- Bit 2 pin 16
- Bit 3 pin 35
- Bit 4 pin 17
- Bit 5 pin 36
- Bit 6 pin 18
- Bit 7 pin 37
- Bit 8 pin 19

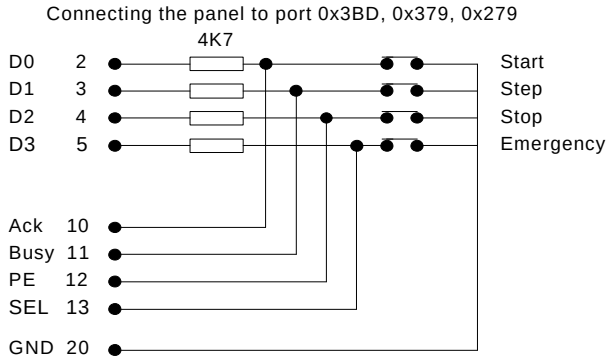
Output byte: 346 HEX (base address 340 HEX)

- Bit 1 pin 22
- Bit 2 pin 4
- Bit 3 pin 23
- Bit 4 pin 5
- Bit 5 pin 24
- Bit 6 pin 6
- Bit 7 pin 25
- Bit 8 pin 7

Connecting the panel to the parallel port

The control panel can be connected to the parallel port where the dongle is installed or to any other parallel port.
The parallel port has to be bidirectional, meaning it can read data from the outside.
If the port is not bidirectional, you can use adresses 0x3BD, 0x379 or 0x279 and make the connections according to the lower circuit. The signals are pulled up by 4K7 resistors. When PrimCAM for example reads input 0x279, it sets 0x278 to high. The resistors pull up the signals. Only the inputs connected with ground give a 0 this way.
The connections are as followed:





Those inputs are active low. As soon as you connect pin 3 (Step) to pin 20 (Gnd), the program steps one line further in the nc code.

6. Glossary

CNC The program is transferred to the controller first and then executed.

DNC The program is transferred part by part to the controller andevey part is executed immediately. PC and controller are in constant communication.

7. Index

