

ARTISAN®
TECHNOLOGY GROUP

Your **definitive** source for quality pre-owned equipment.

Artisan Technology Group
(217) 352-9330 | sales@artisantg.com | artisantg.com

Full-service, independent repair center
with experienced engineers and technicians on staff.

We buy your excess, underutilized, and idle equipment
along with credit for buybacks and trade-ins.

Custom engineering
so your equipment works exactly as you specify.

- Critical and expedited services
- In stock / Ready-to-ship
- Leasing / Rentals / Demos
- ITAR-certified secure asset solutions

Expert team | Trust guarantee | 100% satisfaction

All trademarks, brand names, and brands appearing herein are the property of their respective owners.

Find the *LTI Motion Andronic 3060S* at our website: *Click HERE*

LTI MOTION
KÖRBER SOLUTIONS



andronic 3060 CNC
XPanel Operating Instructions

andronic 3060 Multiprocessor CNC

andronic 3060 XPanel Operating Instruction

Id.-No.: 3060.200B.0-01 ▪ Stand: 07/2017

The German version is the original of this Operating Instruction

All rights are reserved with respect to the content of this documentation and the availability to the product.

Copyright ©
All content of the documentation, in particular the text, photographs and graphics it contains are protected by copyright. The copyright lies, unless otherwise expressly stated, with LTI Motion GmbH.

We reserve the right to make technical changes.
The content of this documentation was compiled with the greatest care and attention, and based on the latest information available to us. We should nevertheless point out that this document cannot always be updated in line with ongoing technical developments in our products. Information and specifications may be subject to change at any time. For information on the latest version please visit www.lti-motion.com.

LTI Motion GmbH
Schlätterstraße 2
88142 Wasserburg/Bodensee
Germany
Fon +49 8382 9855-0
Fax +49 8382 9855-50
info@lti-motion.com
www.lti-motion.com

Table of contents

Table of contents 3
Notice 4
General safety instructions and warnings 5
Use as directed 6
Introduction 6
Field of applications 6
Not used as directed 6
Switch on, start-up phase, turn off 7
Switch on 7
Boot phase 7
Switch off / Shut down 8
Operating system setting, system additions, user profiles (Windows 7) 9
Virus scanner 9
Firewall 9
Microsoft Windows 7 Updates/Security Patches 9
User administration of the CNC (User Profile/User Accounts) 10
Data back-up 10
Operating panel keys 11
Virtual machine operating panel (ANM007) 14
The XPanel user interface 15
Introduction 15
XPanel structure 15
XPanel pictograms of the status area 16
Overview of the eight main groups 17
The status display 19
The main menu of the status display 19
Zeropoints, Manage Zeropoint offsets 24
The main menu of the zero points 24
Tool management 26
Overview of functions 26
The main menu of the tool management 26
Cycles 33
The main menu of the cycles 33
Programs 37
The main menu "Programs" 37
andronic NC Editor 39
Service 43
The Service main menu 43
Error and Messages 49
The main menu Error and Messages (E/M) 49
Revisions 50

Notice

In the creation of this manual, we have made the greatest effort and have taken the greatest care. We reserve the right to make changes to this manual and to the controller or the programs, which are made necessary by technical progress, without previous notice. In later versions, additional pages may be inserted. We would appreciate your information as to errors in the manual or how the manual may be improved.

We accept no responsibility for damage resulting from neglect of the instructions contained in this manual.

We are neither liable for nor responsible for actual or alleged loss or damage, whether direct or indirect, which is caused by the operated or delivered equipment or the software programs, and which are claimed by a customer or another natural person or a legal entity. This clause includes faulty service, lost business transactions, loss of expected profit or consequential damages which occur due to the use of the equipment sold by us or software generated by us.



We accept absolutely no responsibility for damage due to faulty installation!

This instruction manual, the program descriptions belonging to it as well as other objects sold or supplied with the controller, are protected by copyright. All rights are reserved. This handbook, the programs, and all other copyrighted objects may be neither completely or partially copied nor in any other manner duplicated, without the previous written consent of LTI Motion GmbH. Legally created copies, made with the permission of LTI Motion GmbH, of a part of or the complete manual, the computer programs or other copyrighted objects, must have the same copyright notice as the original material.



Before the connection and the start-up of the control, it is imperative that the documentation is carefully read!

General safety instructions and warnings

Hazards may emanate from our devices. For this reason it is imperative you follow the safety instructions and warnings in this document.

DANGER!	**Risk of injury due to electrical power!**
	• **Carelessness will result in serious injuries or death.** Follow safety instructions and warnings in this document and on the device.

WARNING!	**Risk of injury due to electrical power!**
	• **Carelessness may result in serious injuries or death.** Follow safety instructions and warnings in this document and on the device.

CAUTION!	**Risk of injury or damage to the device due to incorrect operation!**
	• **Carelessness may result in minor injuries or damage.** Follow safety instructions and warnings in this document and on the device.

WARNING!	**Damage due to electrostatic discharge!**
	• **Electrostatic discharge can destroy components.** Do not touch electronic components and contacts! Follow safety instructions and warnings in this document and on the device.

DANGER!	**Risk of injury due to rotating parts on the motor!**
	• **Carelessness will result in serious injuries or death.** Follow safety instructions and warnings in this document.

Pay attention to **special safety instructions and warnings** that are given here in the document before a specific action and that warn the user about a **specific hazard**!



NOTE:
The pictograms may also be used on their own with the signal word, e.g. in the connection diagrams, however they have the same function as in the complete warning.

DANGER	**WARNING**	**CAUTION**
		

Use as directed

Introduction

LTI Motion products are developed and produced according to the latest technologies. They are not delivered unless they have been tested for service reliability.
The products may only be used as directed. If they are not used as directed, material damage and personnel injury may result.



LTI Motion assumes no liability for damage due to inappropriate use. LTI Motion is not liable for payment of damages. The user is liable at his own risk if the products are not used as directed.

The following requirements must be met before using LTI Motion products to ensure proper use:

- The corresponding safety instructions for use as directed must be read and understood by all who operate one of our products.
- If the products include hardware, the original condition must not be changed. Software products must not be de-compiled and the source codes must not be changed.
- Damaged or faulty products must not be integrated or put into operation.
- It must be guaranteed that the products have been installed according to the instructions specified in this manual.

Field of applications

The control is used for control panel integration, integration into a housing or the door of the switch cabinet or for machine tool housing integration.
It must be ensured that the required mounting, installation and environmental conditions are fulfilled.
The control can only be used with the configurations described in this manual. Furthermore, the use of an LTI Motion software or firmware is necessary.
Each control system must be parameterized and programmed by competent service personnel before operation.

Not used as directed

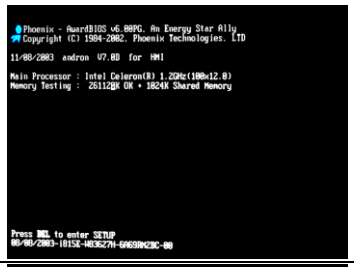
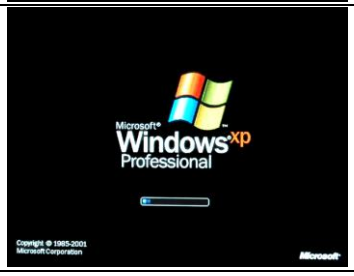
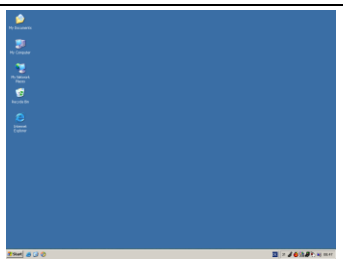
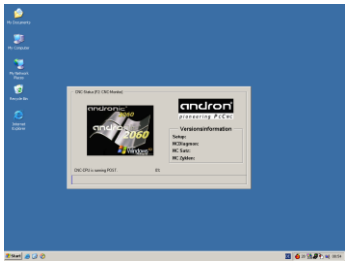
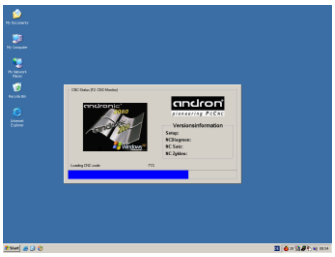
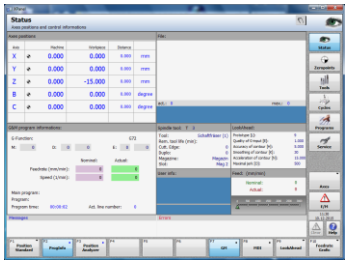
The control is "not used as directed" if it is used in a field of application not specified or if it is used under operating conditions or with technical data not specified in this manual.
The control must not be used if it is exposed to operating conditions which do not fulfill the determined environmental conditions, e.g. use with water or with extreme temperature differences or extreme maximum temperatures is not allowed.

Switch on, start-up phase, turn off

Switch on

The switching on of the control as well as the entire system can be realized in different ways, therefore observe the information of the machine manufacturer!

Boot phase

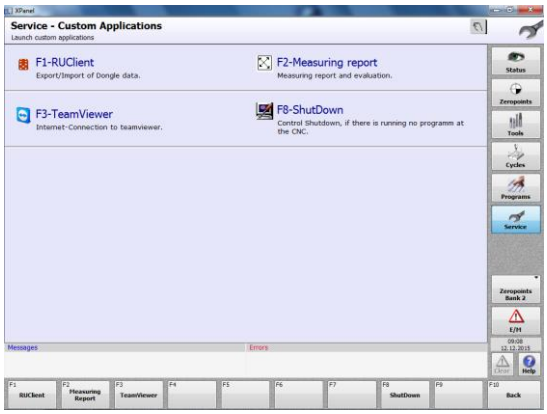
1.	Switch on the control Boot phase of the HMI CPU		If the control was started without connecting or switching on a display, it may happen that the **display** remains **dark** after switch-on. In this case, press the key combination **CRTL + ALT + F1 (for VGA)** **CRTL + ALT + F3 (for DVI)** in order to switch to the correct output of the graphics card. (only with SHB210 HMI CPU)
2.	Loading of the HMI operating system	 → 	
3.	Boot phase of the NC CPU Loading of the NC operating system	 → 	
4.	Initialization of the operating system Start-up of the sercos phases from 0 to 4		
5.	Control is ready for operation		

Switch off / Shut down

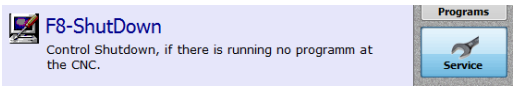
For switching off the control as well as the whole machine pay attention to the following notes:
To avoid <u>data loss</u> or <u>hard disk damage</u> the control must be shut down always in the correct way.
That means the operating system must be completely finished before switching off the control or the whole machine.

There are different possibilities to shut down the operating system, depending on how you have logged in to the control (→ depended on the user profile).

→ **See user management** (page 10)

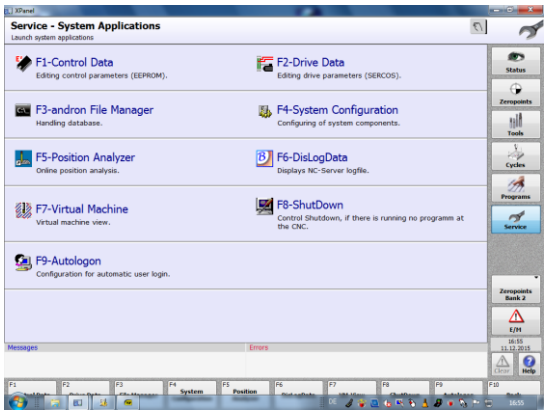


<u>CNC User:</u>
"CNC User" has access only to the andronic user interface, but no access to standard Windows functions, which is why he can shut down the control only via the *Service/Custom Applications* menu.



These actions will shut down the operating system, so that you can surely switch off the power supply of the control.

<u>admin / CNC Admin:</u>
The *"admin"* and *"CNC Admin"* users can shut down the system either via *Service/System Applications* or via *Service/Custom Applications*.



CAUTION

If you have logged in as user "admin" or "CNC Admin", you can click on "Shut Down" in the Windows start menu.

However, you are strongly advised not to do so, because the system is not monitored as to whether the machine is still in the machining phase or other important functions are still active!

Observe also the information of the machine manufacturer!

Operating system setting, system additions, user profiles (Windows 7)



Microsoft® Windows® 7 Ultimate SP1 is the installed operating system on the HMI computer of the andronic 3060.
In order to guarantee the highest possible system performance, all unnecessary animation, superfluous software additions and other optical gadgets are deactivated. Changes in these settings can slow down the system considerably.



Virus scanner

The controls are submitted to a virus scan prior to delivery.
If the CNC control is integrated into a company network by subsequent installation of additional software using USB memory sticks, or via a previously set up internet access of the control, it is **recommended** installing a virus scanner on the CNC.

However, please make sure that a "lean" virus scanner is installed, i.e. on that uses as few resources as possible and does not perform a complete hard disk scan with top priority, especially not unscheduled during machine processing.

We cannot and are not allowed to make any recommendations to that effect.



Firewall

In the default setting the Windows firewall is activated. The use of other firewalls is **dissuaded urgently**.



Microsoft Windows 7 Updates/Security Patches

The software installed on andronic controls prior to delivery is updated at regular intervals to the most current version of the Microsoft security updates.
However, it is not possible to install all security updates at approximately the same time as the Microsoft Security Bulletins, since extensive software tests have to be carried out after each update installation, in order to guarantee proper functioning of the entire system.
andronic 3060 CNC with Windows 7 Ultimate SP1 will be delivered ex works with deactivated automatic Windows updates.
Even though automatic installation of the patches would in most cases not cause any problems for the CNC function, but it could lead to errors.
In any case, patches / updates should not be installed automatically and especially not during the automatic restart of the operating system, nor manually during machine operation.

Known problems:

a) Replacement of network drivers with standard drivers, resulting, for example, in the EtherCat interface being disabled.
b) Stopping essential CNC services, resulting in an EMERGENCY STOP reaction during operation, with the inevitable consequences for a machining process in progress.
c) Installation of the KB3035583 patch – Windows 10 update reminder
This patch must be deinstalled and deactivated, since otherwise the operating system may eventually be updated to Windows 10 and moreover constant reminders for the update will appear.

It is absolutely prohibited to install Microsoft security updates or other updates subsequently (e.g. as part of the Windows function "automatic updates") without express approval by LTI Motion.

In this case, the warranty for the control will become void, since it is operated with non-released software.



User administration of the CNC (User Profile/User Accounts)

If andronic is integrated into a domain, please note that for proper operation of the control the user to be logged in automatically must have been entered in the "CNC Users" group, to ensure that CNC data access to their system files is working.

Automatic login is activated or changed by means of the "netplwiz.exe" system program. It is accessed via the start menu by entering netplwiz.exe under Search

andronic 3060 with Windows 7 Ultimate has the following predefined users with the corresponding user profile:

a) **"CNC User"** with the password **"cnc user",** who is logged in automatically (see below) This user can only use the andronic user interface, since he has no access to Windows functions. (No start menu, no taskbar, no Windows keys) In XPanel the "CNC User" only sees the User databases and can only load limited system functions (no SysConfig, ...).
b) **"CNC Admin"** with password, who has access to Windows functions, unless a UAC query with administrative rights is required (UserAccessControl active from Windows 7).
When making UAC queries (e.g. right click on "run as administrator"), no administrator can be selected since they are not visible.
In XPanel the "CNC Admin" also sees the system database paths and can start all system functions.
c) "**admin**" with password as previously under Windows XP for the "administrator" with administrative rights for the entire system.
d) The "administrator" account is still available, but has been deactivated under Windows 7 as standard, as has been the user "guest".

Predefined user profiles	**CNC User**	**CNC Admin**	**admin**
Access to the andronic user interface	•	•	•
Access to Windows functions		•	•
Administrative rights for the entire CNC system			•

Each user who wants to use the CNC functionality must have been entered in the "CNC Users" group.
The user who requires advanced rights in XPanel must be in the "CNC Admin" group.
This is why for CNC setup you have log out and log in as "CNC Admin".
To administer the system (install software, ...), you have to log out and log in as "admin".

Auto Login, how to set up automatic login to Windows 7:
Such an "**Autolog**" can be set up via the "netplwiz.exe" system program. It is accessed via the start menu by entering netplwiz.exe under Search.

It is absolutely necessary to deliver the control to end customers only with "autolog" on "CNC User" (delivery status ex works)

It is recommended changing the password prior to delivery to the end customer. The password for "admin" must not be passed on to the end customer.

For LTI Motion, there is an additional administrative user: "andron-admin" as emergency administrator, if the password for the "admin" user is no longer known.

If this "andron-admin" account is deleted, deactivated or manipulated, LTI Motion reserves the right to set up a new user administration in case of repair or on-site service.



Internet access

In the standard configuration the **machine user has <u>no</u>** internet access. Only with full access and administrative rights can you connect the control to the Internet

The andronic software offers the possibility of remote diagnosis via internet with the program NetViewer. To use this function it is necessary to make a restriction on administrator level so that the machine user has no general internet access.



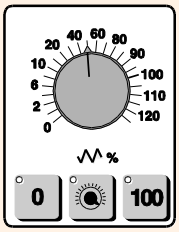
Data back-up

It is recommended performing a back-up of the contents of the CNC HD/SSD at regular intervals.
When using Acronis TrueImage, you must not install this tool on the CNC, since its integration into a virtual drive often results in data access problems and even blue screens.
Instead this tool should only be used off-line (booting from USB stick, followed by data back-up).

Operating panel keys

LTI Motion supports the control with a basic PLC program for the operating and control functions described below. It is possible that the machine manufacturer delivers his own PLC program with different functions and extensions. If there is no further information the following description is valid.

Feed step switch



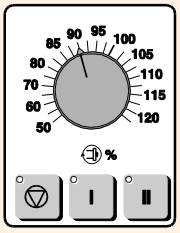
With this step switch you can modify the feed rate in manual and automatic mode between 0% and 125%. A switchover between the speed keys "0", "Feed step switch" and "100" is always possible within the operating modes.

0 **Rapid traverse/Creep feed 0%:** The rapid traverse or the creep feed rate is set to 0% for all operating modes (manual, automatic or MDI mode), which leads to an immediate stop of the selected axes (depending on the brake reaction of the drives). No more positioning command is carried out. For example, the execution of started NC block programs with traversing instructions is stopped until the key "100" or the key "Feed step switch" (observe switch position) is pressed.

Feed step switch: The rapid traverse or the creep feed rate can be set from 0% to 125%, according to the feed step switch position, for all operating modes (manual, automatic or MDI mode). The step switch has a multiple function. In position "0%" the axes stop; in position "100%" the axes move at the defined rapid traverse or creep feed rate. From position 100% to 125% you can raise the defined speed in percent up to a maximum of 125%.

100 **Rapid traverse/Creep feed 100%:** The rapid traverse or the creep feed rate is set to 100% for all operating modes (manual, automatic or MDI mode).

Spindle speed step switch

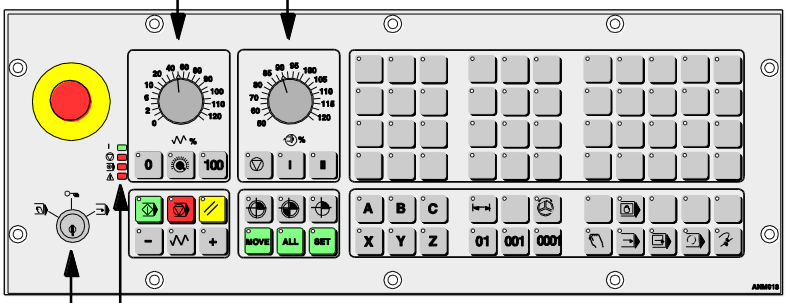


With this step switch you can modify the spindle speed between 50% and 125%.

Spindle stop: The selected spindle is stopped. A restart of the corresponding spindle is only possible by the key "NC START" (e.g. MDI mode: S10000 M3).

I **Spindle I:** The key "Spindle 1" is free programmable. It can be used for the activation of the spindle control 1, for example.

II **Spindle II:** The key "Spindle 2" is free programmable. It can be used for the activation of the spindle control 1, for example.

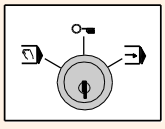




Zustands-LEDs

- I Supply voltage 24 V DC is applied at the operating panel.
- Feed release is missing or or was removed.
- Keyboard lock is shown.
- ⚠ A PLC error message is active.

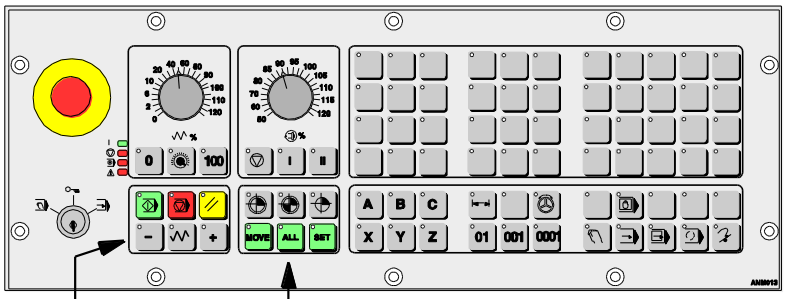
Keylock switch



The keylock switch has 3 positions which are provided for the following functions:

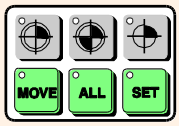
- Hand-, automatic-, MDI and set up mode: In the set up mode it is possible to work with reduced speed at opened machine cabinet.
- Keyboard lock: All keys are locked, with exception of the key **NC Stop** and **NC Reset**.
- Hand-, automatic-, MDI mode:

panel_keys_1_e.eps



Absolute, relative and reference point keys

The control takes three different types of position for each axis into account.



The reference points of the axes are already defined from the machine manufacturer. The absolute and relative positions can be set by the machine operator (SET) according to requirements. They can be moved (MOVE) or set (SET) at another position for each single axis, i.e. each preselected axis, or for several axes (ALL), i.e. the axes defined in the EEPROM in the defined succession. The reference points can only be moved.

MOVE With this key you can move single or several axes to the relative zero, absolute zero or reference point .

SET With this key you can set the relative or absolute zero for single or several axes.

Reference point

All axes have a machine-related reference point or zero. These can be automatically moved in succession in the operating mode HAND either for each single axis or for several axes (axes and succession definable in the EEPROM).

Procedure for one axis:

>> Preselect the axis >> Push the botton REFERENCE POINT >> Push the botton MOVE

Procedure for all defined axes:

>> Push the botton REFERENCE POINT >> Push the botton ALL >> Push the botton MOVE

Relative position

The relative position can be automatically set or moved in the operating mode HAND either for each single axis or for several axes. For the setting or the motion of several axes, the axes and the succession are defined in the EEPROM. You can move a relative position only after it has been set. With NC STOP, the relative position is set by the control. Previous relative positions were deleted.

Procedure "Set" for one axis:

>> Preselect the axis >> Push the botton RELATIVE ZERO POINT >> Push the botton SET -> the relative zero point is set

Procedure "Set" for all defined axes:

>> Push the botton RELATIVE ZERO POINT >> Push the botton ALL >> Push the botton SET -> the relative zero points are set

Procedure "Move" for one axis:

>> Preselect the axis >> Push the botton RELATIVE ZERO POINT >> Push the botton MOVE -> the relative zero point is moved

Procedure "Move" for all defined axes:

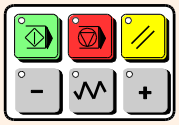
>> Push the botton RELATIVE ZERO POINT >> Push the botton ALL >> Push the botton MOVE -> the relative zero points are moved

Absolute zero point

The absolute zero can be automatically set or moved in the operating mode HAND either for each single axis or for several axes. For the setting or the motion of several axes, the axes and the succession are defined in the EEPROM. The setting of the absolute zero point works like SETPOS = 0 (all axes).

Procedure "Set or Move" for one axis or for all defined axes: (see RELATIVE POSITION)

Machine keys



With the NC-Start key a program is started in automatic mode. For this purpose, the following conditions must be fulfilled:

- The axes must be referenced
- A program has been loaded
- The operating mode "AUTOMATIC" is selected

With the NC-Stop key you stop the automatic execution of a program and simultaneously the relative position of all axes were stored. The stop becomes active at the end of the current program line. This state is signaled by the LED which is constantly on. At the same time the LED of the NC-START key flashes, which allows to continue the program from the stopped point. If you move one or more axes after actuating the key NC STOP in manual mode, the position on NC STOP can be reached again with the key REPOS.
With change of the automatic operating mode, the program can be continued with NC Start.

The NC-RESET key terminates the automatic processing of a program. It is no more possible to continue the processing of the program from the stopped point. For the control unit, the program has been definitely terminated. You can restart the program by NC-START.

This key switches between rapid traverse and creep feed rate in the set-up and standard mode. The appertaining speed values for set-up or standard mode are filed in the EEPROM of the control unit: after the control unit has run up, the set-up mode (parameter set 2) is active with its EEPROM parameters **Set up feed rate** (creep feed) and **Set up rapid traverse**. After a search for reference, the set-up mode is left and the standard mode becomes active. In the standard mode, the following EEPROM parameters are available: **Manual feed rate** (creep feed) and **Manual rapid traverse**. The rapid traverse is indicated by the LED which comes on.

Travel and positioning keys

+ With the trevel key "+" you can move a preselected axis in the positive direction in the manual mode.

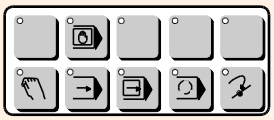
Procedure:
- Step mode due to the preselected feed or fixed path key.
- Constant traveling, if no feed or fixed path key is activated.

– With the travel key "-" you can move a preselected axis in the negative direction in the manual mode.

Procedure: (see travel key "+")

panel_keys_2_e.eps

Machine keys



The operating mode MANUAL MODE is automatically active after the run-up of the control unit and is signaled by the LED which comes on. In the manual mode it is possible to:

- to start the reference move
- traverse or set the absolute or relative zero point
- move the axes manually

The operating mode AUTOMATIC MODE can be activated only after a reference move has been carried out. In the automatic mode it is possible to process a loaded program. The automatic mode is started by operating the NC-START key.

The key SINGLE BLOCK processes an automatic program block by block. After having operated the key, a stop is carried out at the following points:

- In the expert mode after each positioning and I/O command of the program
- In the NC mode at block end of each NC program line. If the residual path display is activated in the EEPROM, the residual path is displayed to the block end.

The automatic program can be further processed in single block by the NC-START key. If the single block is deactivated, the processing is continued to the end of the automatic program.

The key OPTIONAL STOP stops the processing of a G&M code program. The G&M code automatic program can be processed to the next M01 function by the NC-START key. If the function OPTIONAL STOP is deactivated, the processing is continued to the end of the automatic program.

With this key (REPOS) the interruption position of an automatic program can be approached again. In this case all individual movements which can be executed in manual mode are summed up to a one travel distance. This travel distances are moved back to the interruption position in inverse order.

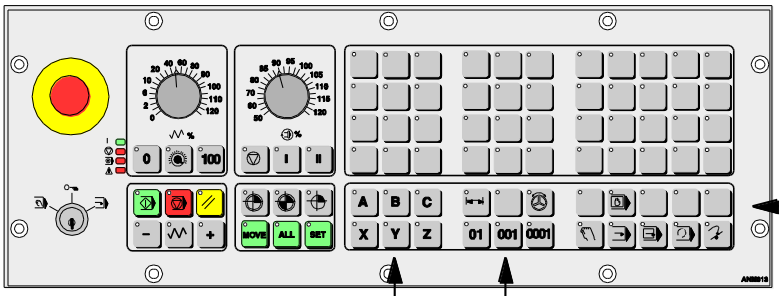
Proceeding:

>> Program abort in the operating mode automatic by actuating the key NC stop. >> Change of operating mode to manual mode.
>> Travel in manual mode by pre-selection of the desired axis and acknowledging the traverse keys "+" or "-" (RePos LED is blinking).
>> Actuating the key REPOS: Re-approaching of the interruption position in inverse order of the individual travel distances.
>> Change of operating modes to the automatic mode. >> Actuating the NC start key: the automatic program is continued from the interruption position on.

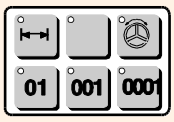
In the operating mode MDI (manual data input) it is possible to carry out a up to three G&M codes via an overlaid edit window.
The key cannot be selected, because it is activated over the operating interface.

Proceeding:

>> Press the function key F7 (MDI function in the andronic operating interface) >> Enter a G&M code line. The input must be closed with ENTER.
>> After having operated the NC-START key, the entered program line is executed.



Travel keys



With this key you can carry out a defined linear or rotational feed. The feed value is stored in the file "Parametersettings.ini" and can be changed in the program SYSCONFIG. In order to carry out a feed, activate the fixed path key and an axis. Now you can cover the entered feed value via a jog direction key (+/-)

By selecting the handwheel key, you can carry out a motion of the axis via handwheel in the desired resolution (handwheel increment), having first preselected the axis and operated the feed key. This is only possible in the operating mode HAND.

01
001
0001

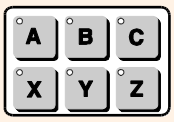
These keys allow a movement in the dimensions 0.1, 0.01 and 0.001 mm or inch. The activation of a feed key is confirmed by a light emitting diode. By selecting another feed key, the feed selected before becomes inactive. This is only possible in the operating mode HAND.

Vorgehensweise:

>> Select the desired feed >> Preselect the desired axis >> Press the corresponding travel direction key (+/-)
>> Every time you press a travel key, a jogging is carried out at the feed value

To use this function it is necessary to activate the incremental width in the EEPROM.

Manual travel keys



The manual travel keys consist of the axis keys A, B, C, X, Y and Z. The preselection of the required axis takes place by operating the corresponding key and is indicated by a light emitting diode (diode comes on). You can deactivate an axis by operating the key once again (light emitting diode goes off). You can move an axis by preselecting it and operating a travel key. The axis moves as long as the jogging key is pressed.
If a limit switch is approached when moving an axis, the corresponding axis drive stops immediately. The LED of the axis selection key and the LED of the jog direction key, by which the limit switch has been approached last, flash. This indicates that you can depart the limit switch by the jog direction key which is not flashing. Once the limit switch is free again, both light emitting diodes go off.
If an axis is preselected, which is not defined in the EEPROM of the control unit, the LED of the selected axis flashes and an error message is displayed, which disappears after correct axis preselection.

panel_keys_3_e.eps

Virtual machine operating panel (ANM007)

As an alternative to a true machine operating panel connected to the fieldbus, an easy-to-configure virtual operating panel of andronic can be used, which can be operated by mouse or touch screen.

CAUTION

LTI Motion supports the control with a basic PLC program for the operating and control functions with the ANM007 described below. It is possible that the machine manufacturer delivers his own PLC program with different functions and extensions.

"The virtual machine operating panel is a pure software solution. The applicable EU directives such as the Machinery Directive and their implementation into national law must be observed. The machine manufacture is responsible for this. LTI Motion cannot be held responsible for non-compliant use of the virtual machine operating panel with directives."



Illustration: Virtual machine operating panel page 1

The first page cannot be configured and contains the operating keys of the ANM hardware in the top section. In the bottom section, a keypad and the editing keys have been added. For text characters and symbols, a PC keyboard is required. The bottommost line contains the two keys for page forward and backward and LEDs as in the hardware version of the ANM013.

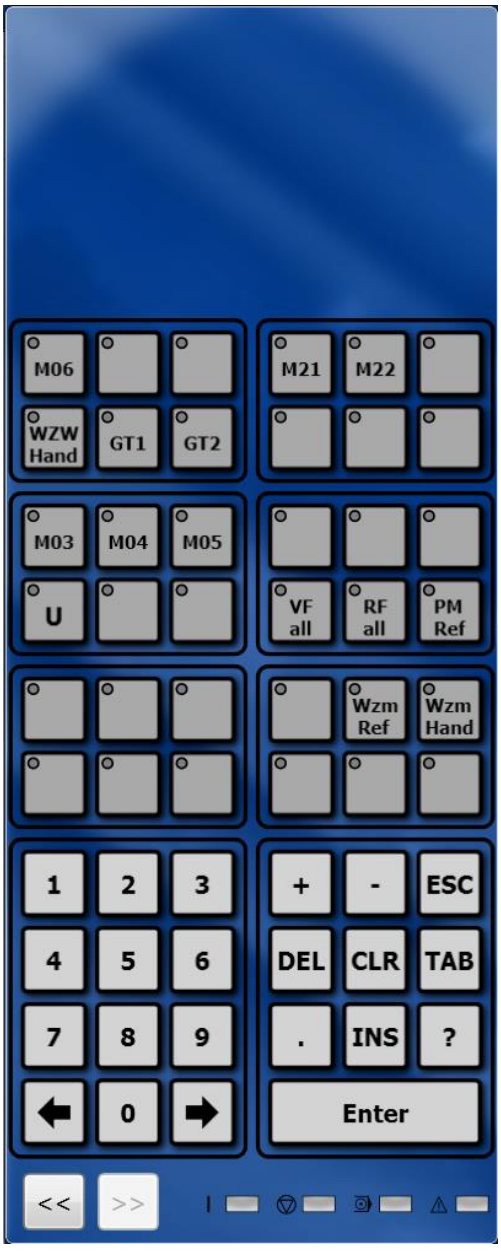


Illustration: Virtual machine operating panel page 2

The second page contains six freely assignable keypads for programming, see chapter "Components operating panel" of the technical manual. As on page 1, on page 2 a keypad and editing keys have been added.

The XPanel user interface

Introduction

The pleasantly designed and clearly structured operating interface can be operated intuitively. The Microsoft **.NET** based panel has various configuration possibilities and can be adjusted problem-free to the needs of the end user.
Our andronic CNC controls has a wide range of applications, ranging from classical machine applications like milling, honing, grinding and eroding to highly dynamic processing. For this reason we put a lot of value during the design of the new panel in individual adjustment options and optimal usability.

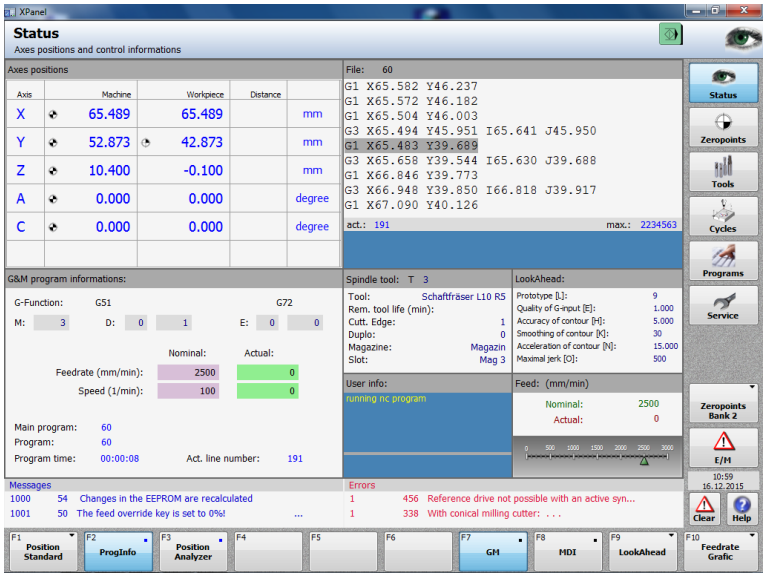


Illustration: Structure and functional areas of the XPanel interface

XPanel structure

Despite the extensive configuration options, the different areas of the interface are clearly defined. The outer frame of the XPanel always has the same appearance, allowing the user to become quickly familiar with it. The basic information always appears at defined locations:

Status area

At the top, extending from left to right over the entire screen, is the status area in which on the left the name of the active main group and underneath a description of the current action appear. This is followed by pictograms reproducing the status of the remote mode, key-operated switch, active transformations and of the current CNC operating mode.
On the very right in the status area, the pictogram of the active main group (MG) is shown. The background color of the status area depends on the main group.

Main group area

The right-hand side of the screen features eight MG keys.
They can be activated by pointer devices (mouse, touch, etc.) or the key combinations Ctrl+Shift+Alt+F1 – Ctrl+Shift+Alt+F8.
Main group 7 cam be configured. This is indicated by the small inverted black triangle in the right upper corner. It is configured by clicking this triangle or by a right mouse click on this button. From the popup menu that appears, a function can be assigned.
Note: If the function is active, no other function can be assigned.
Remedy: End the active function.
Below MG 8, the local time and the current date are shown as tooltip. Underneath are two smaller buttons.
"Clear" deletes the accumulated errors within the application. The key is only active if any errors are active. The next time XPanel is started again, these errors will reappear. "Help" (Ctrl+Shift+F1) provides context-dependent help.

Function keys area

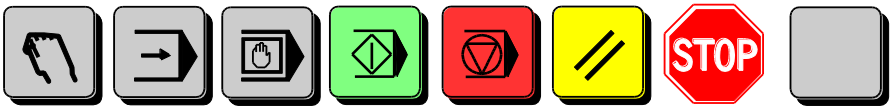
At the bottom of the screen are the function keys F1 – F10. They are assigned according to main groups and can be several levels deep.
Above them on the left is the area for displaying messages (blue) and on the right the area for error messages. Error messages can be deleted by pressing the "Clear" key. To delete the messages, the message bits must be removed by the PLC or CNC.

The attempt was made to place the same functions in all bars on the same function key. Context-dependent, function keys become enabled or disabled.

XPanel pictograms of the status area

Operating modes

Stop is shown in the event of an Emergency Stop, the empty pictogram is shown when the CNC could not initialized correctly. For the meaning of the remaining pictograms, see description of the control panel.



Transformations



Compensations



- Spindle pitch error compensation
- Cross compensation
- Grid compensation
- Calibration offsets
- Parametric axes of rotation
- Parametric volume compensation

Spindle Offset



Handwheel Offset



Key-operated switch

Control panel key-operated switch: controls data input and remote mode



Remote mode

In NC block mode, the control is remote-controlled. Remote mode and remote access are mutually exclusive.



Remote access

The control (programming station) access the data of a remote control. The tooltip provides the computer name.



Overview of the eight main groups

The XPanel user interface is divided into eight main groups, which can be accessed directly from the right-hand menu bar.



- **Status** (Axes positions and control information)
- **Zeropoints** (Manage zeropoint offsets)
- **Tool Management** (Manage tool data and magazine assignment tables)
- **Cycles** (Execute setup cycles, edit or create cycle parameter sets or configure cycle settings)
- **Programs** (Manage and edit programs)
- **Service** (Basic control configuration and diagnostic)
- **Configurable main menu**
- **Error and Messages E/M** (Displays error and messages)

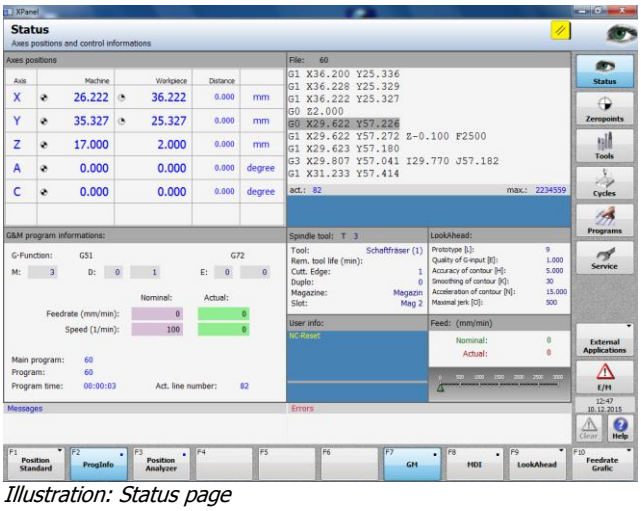


Illustration: Status page

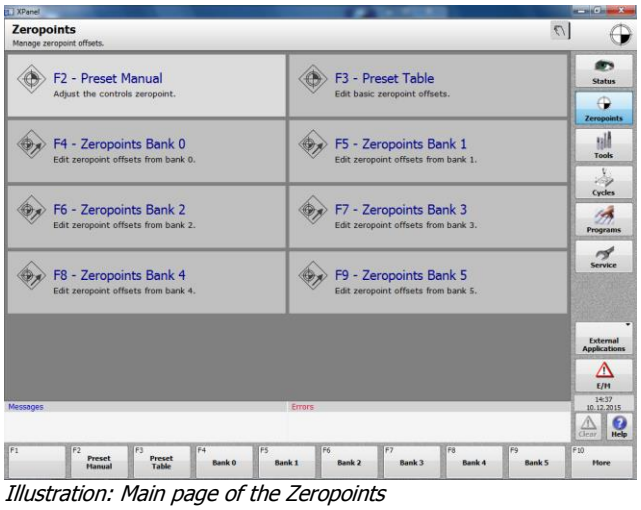


Illustration: Main page of the Zeropoints

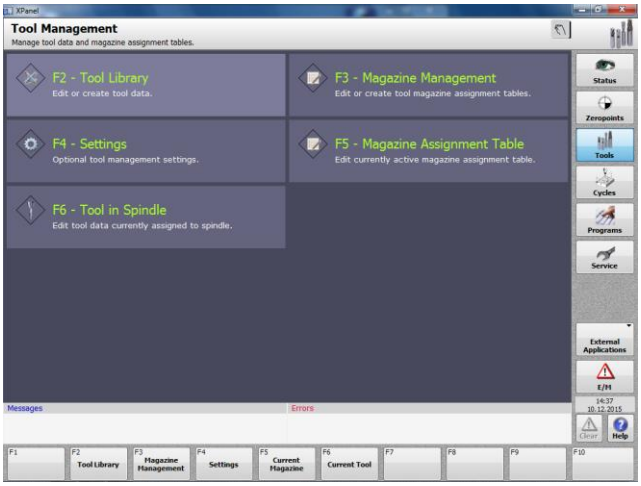


Illustration: Main page of the Tool Management

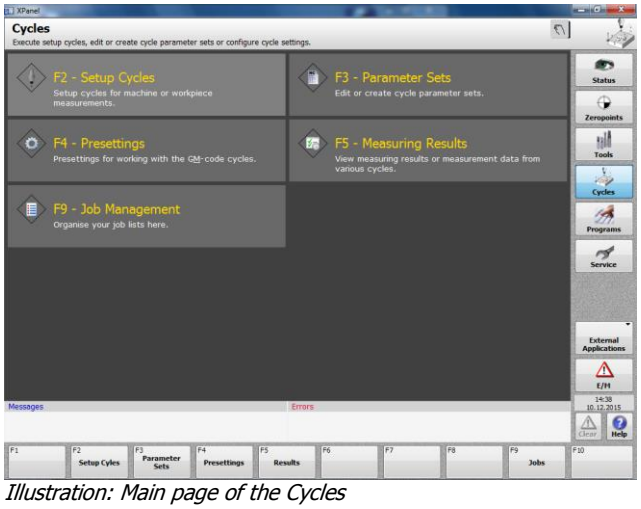


Illustration: Main page of the Cycles

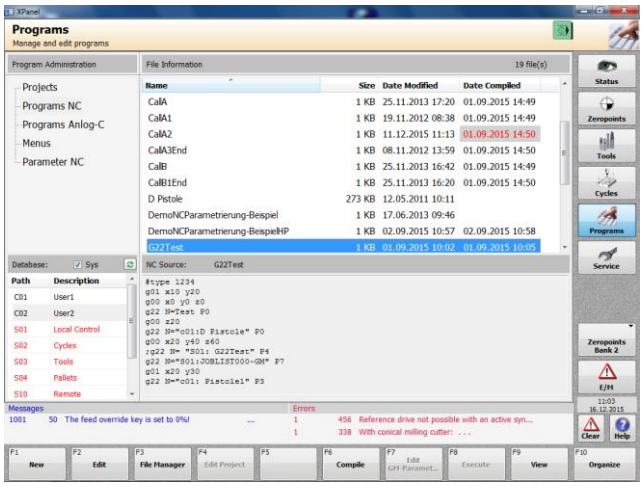


Illustration: Main page of the Programs

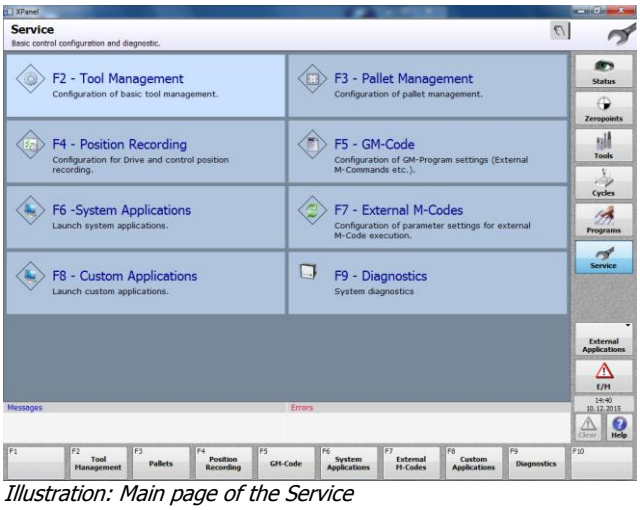


Illustration: Main page of the Service

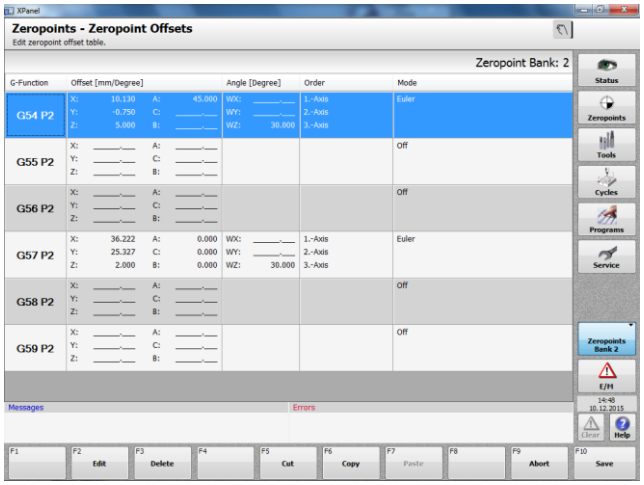


Illustration: Configurable main menu (here Zeropoint Offsets)

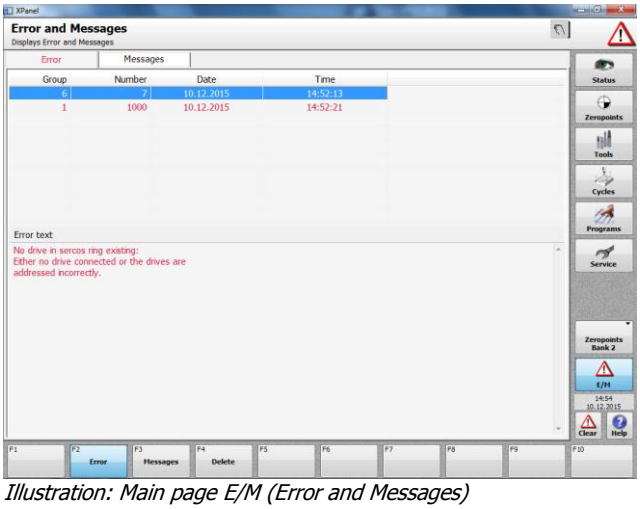


Illustration: Main page E/M (Error and Messages)

The status display

The main menu of the status display

The status display is divided into 16 fields of equal size. These fields can be configured for each language via the XML file ...\andron\Panel\Plugins\State.Config.

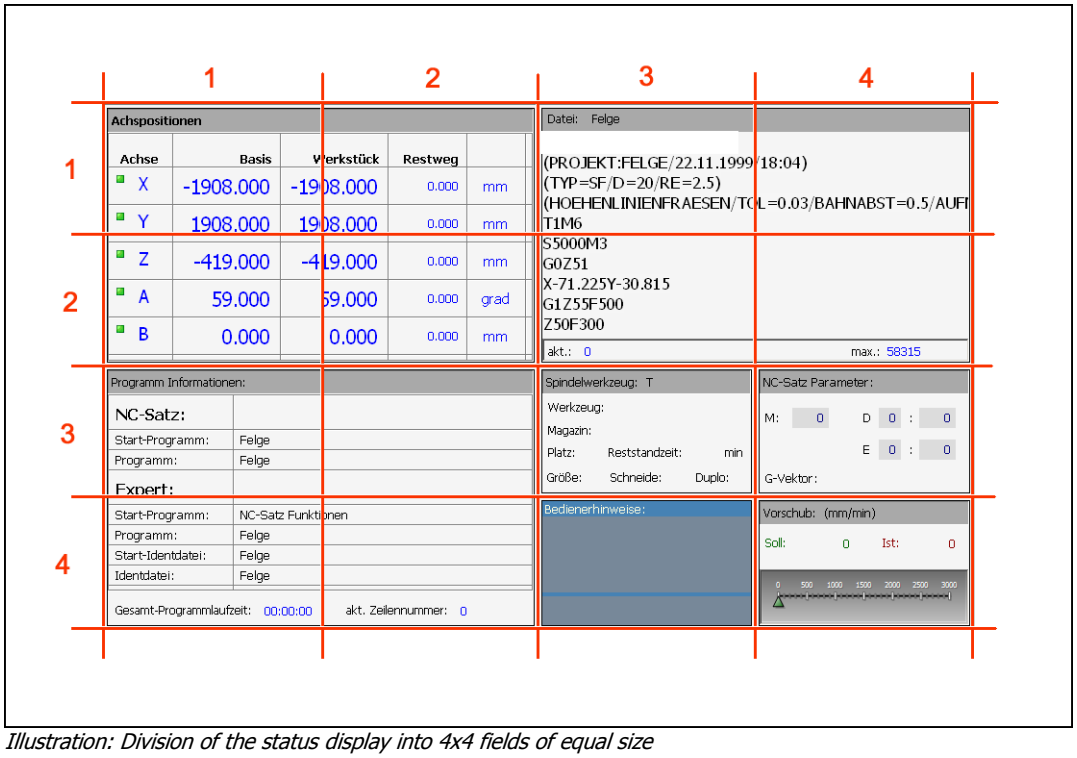


Illustration: Division of the status display into 4x4 fields of equal size

For these fields, **controls (display and input windows)** of different size (multiple of the basic field size) are available. The user can change their arrangement, but not the controls themselves.
For the configuration of the fields, a separate description is available: StateConfig

Group affiliation of the function keys

To let the user know which function keys have toggle functionality or which keys belong to the same group, graphic elements are shown on the function keys.

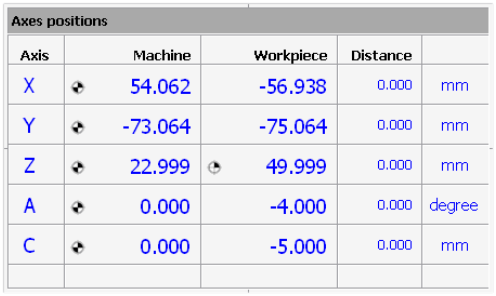
- **A black triangle with the tip pointing downward** is a toggle key. At least two controls are incorporated in them. You can toggle between these controls by pressing the key repeatedly.
- **A square** means that the key belongs to a group.
- **The same color** means the same group. The controls of a group are always shown in the same area of the status display.



Illustration: Function keys of the status page

At present, the following controls are available.

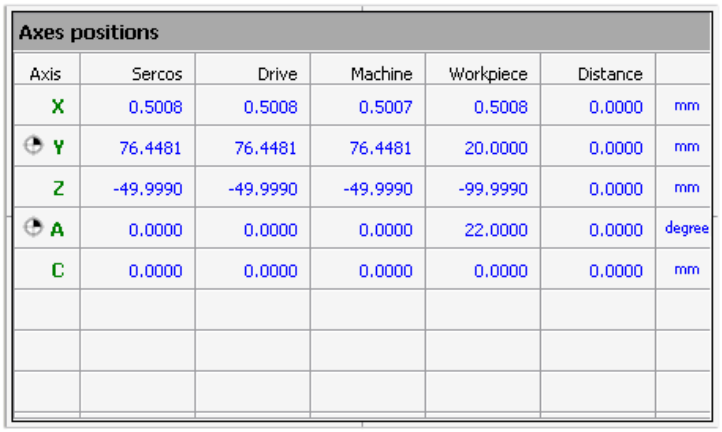
StatePosit2x2 (2x2):



(Standard display)
Metric display of a max. of 5 axes. The axis designator, machine with reference status, workpiece with manual preset offset, remaining path and dimension are displayed. Which axes and in which order they are shown, can be defined via the configuration file XPanel.exe.Config. By default, the axes XYZAC are shown. The two figures in parentheses behind the name of the control indicate its size in field rows and columns for which the control was designed.
In this illustration, all axes have been referenced, and the Z axis has been assigned a manual preset offset.
The tool tip of the graphics provides the offset value.

A8Setter (2x2):

(Setter display)
Metric display of a max. of 8 axes. The axis designator with reference status and manual preset offset, Sercos, drive, machine, work piece, remaining path and dimension are displayed.
For reasons of space, both the manual preset offset and the reference status are shown in the axis designator column.



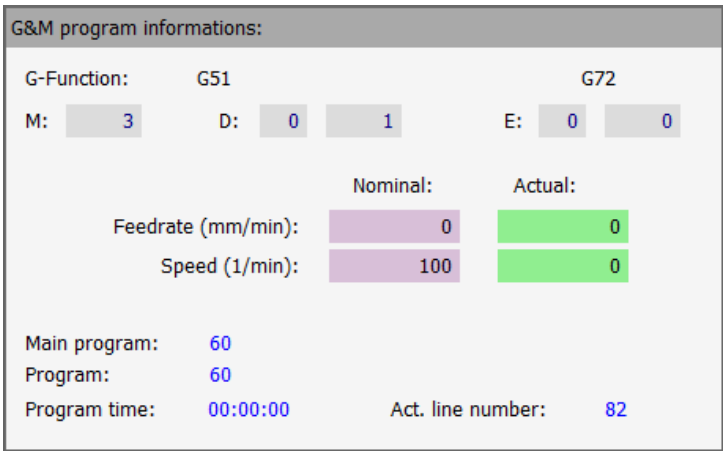
The reference status was color-coded in the axis designator:

green → referenced
red → not referenced

Manual preset offsets are signaled by a graphic in the Axis column. The tool tip of the graphics provides the offset value.

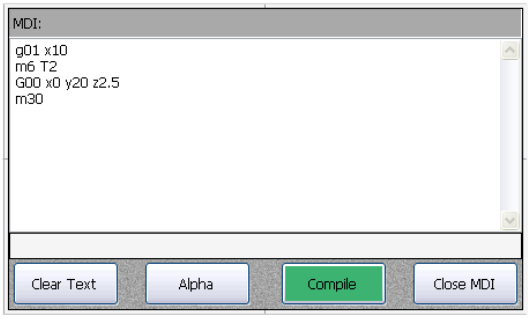
StateProgInfoGM (2x2):

Visualization of the NC block program information
Modal G functions, M-D-E words, federate, spindle speed (Nominal/Actual), start and current NC program, previous program time and line number of current program.



StateMDIAlpha (2x2):

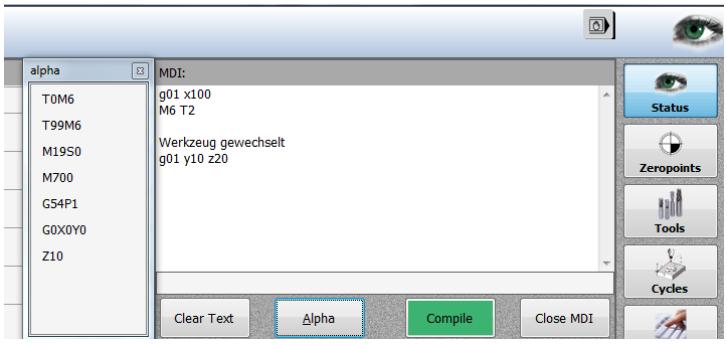
MDI Control:
Input of multi-line NC block instructions.
The length of the MDI source is limited by the size of the transfer buffer.



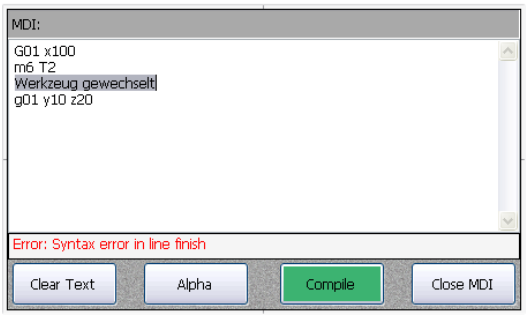
MDI is enabled by pressing the function key F8 MDI or by clicking into the text input field. The background color of the text input field changes and becomes focused, while the status line and the CNC buffer are deleted.
MDI is disabled by pressing the "Disable MDI" key or by clicking on a different control. The text input field turns gray and loses the focus.

The status line above the four keys is used both for user guidance and for displaying compilation error messages.

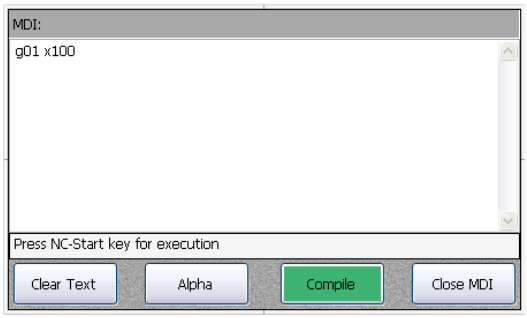
- Key **"Delete All"** or Ctrl+A: Text input field is deleted, CNC buffer is empty.
- Key **"Alpha":** another window containing predefined strings is displayed and becomes focused if it was already visible. The strings in this window can be transferred to the text input field and the selected text position via Drag & Drop, double-click or by pressing the Enter key. Upon transfer, the text input field becomes focused, allowing you to continue writing immediately behind the inserted string. The text strings have been defined in the MDIKeys-config file and are not language-specific.
  The file is located in the directory: ...\andron\Panel\Plugins
  Note: a string can only be transferred by pressing the Enter key if the string was selected and the Alpha window is focused.
- Key **"Compile":** The contents of the text input field is compiled, transferred to the CNC buffer, and a status message is output. As the Enter key will also trigger compiling, Ctrl+Enter is used for the line feed.
- Key **"Disable MDI":** MDI is disabled. If MDI is still being executed, it cannot be disabled.



MDI with Alpha keys window



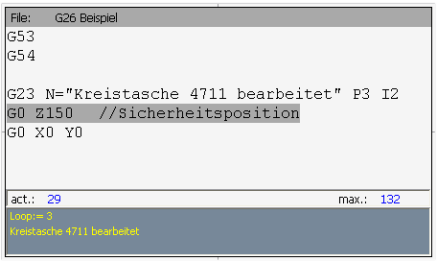
In the event of faulty compilation, an error message is output in the MDI status line, and the cursor is positioned in the faulty NC block line.



The MDI status line signals that the MDI instruction was compiled correctly and transferred to the CNC buffer. Execution can start.

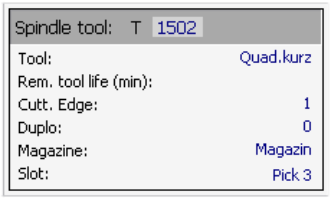
StateGMSrcView1 (2x2):

Display of the NC block currently to be executed in the CNC, including the filename, the current and maximum line number and a max. of 3 single-line texts.
The texts are defined in the NC block via G23.
The NC block can only be displayed if it is not more recent than the code executed in the CNC.



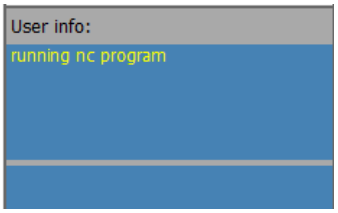
StateTool (1x1):

Display of the data relevant to the spindle tool.



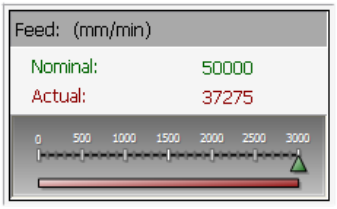
StateUser1 (1x1):

Display of user notes.
A maximum of 2 multi-line texts are possible. The display area for the texts can be moved by means of a pointer device. (SplitWindow)



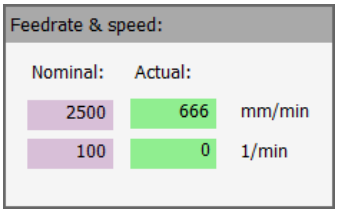
StateFeedGaug1 (1x1):

Explicit and graphic display of the nominal/actual feedrate The green triangle represents the nominal value, and the actual value is shown as a red bar.



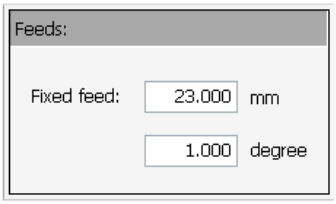
StateFeedSpindle (1x1):

Display of nominal/actual values for federate and speed.



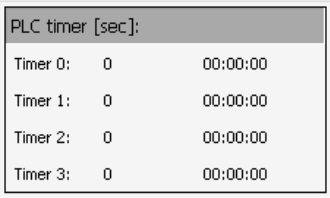
StateFeeds (1x1):

Editing of the fixed feed both for linear and rotary axes.



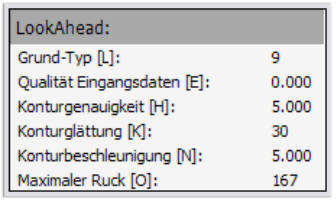
StatePLCTimer (1x1):

Display of the first 4 PLC timers. They can be started, stopped and reset via the PLC.
Resolution: 1 sec



StateLookAhead (1x1):

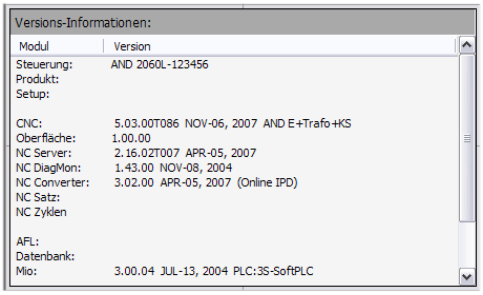
Display of the currently used LookAhead data.



Controls not used at the present time

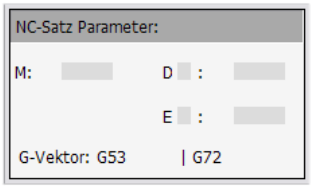
StateVersionInfo (2x2):

Display of the installed versions of the individual modules. In contrast to the control version, not the entire information is available for the demo versions.



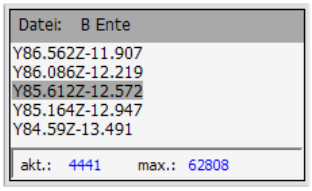
StateGM (1x1):

Display of the MDE NC block parameters and of the active G commands.



StateGMSrcView (1x1):

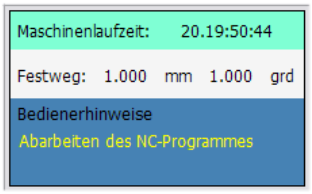
Display of the NC block currently to be executed in the CNC, including filename, current and maximum line number. The NC block can only be displayed if it is not more recent than the code executed in the CNC.



If this control is used in 2x2 field or larger, the size of the font will be changed.

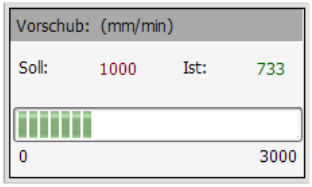
StateUser (1x1):

Display of user-specific information, such as machine runtime, current fixed paths and user information.



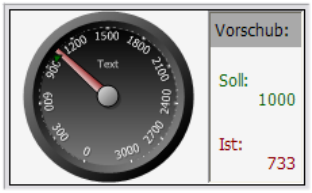
StateFeed (1x1):

Display of the nominal/actual feed. The actual feed is also shown on a progress bar. The upper range limit (here 3000) can be configured jointly for all 3 controls in the XML file Posit.Views.dll.config.



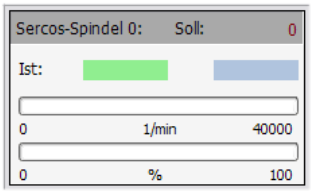
StateFeedGaug (1x1):

Here the same data as in **StateFeed** are shown.
Instead of the progress bar, an analog display instrument is used.



StateSercosSpindle_0 (1x1):

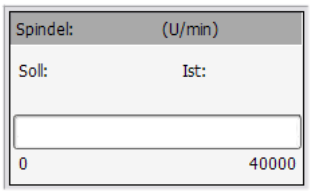
Display of the Sercos spindle "0". In addition to the nominal speed, the actual speed and actual torque of the Sercos spindle "0" are shown both explicitly and graphically by means of a progress bar. The upper range limit for the actual speed (here 40,000) can be configured jointly for all 3 controls in the XML file Posit.Views.dll.config.



The torque value is displayed in percent. Negative values are shown in both displays in the color lavender blush.

StateSpindle (1x1):

This control is used for displaying an analog spindle.
Except for the torque, the same data as in **StateSercoSpindle** are visualized.



Zeropoints, Manage Zeropoint offsets

The main menu of the zero points

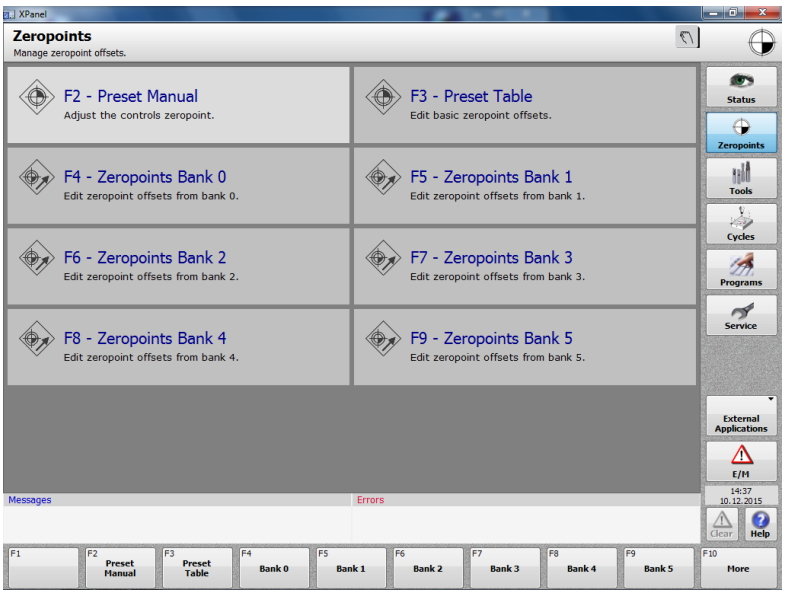
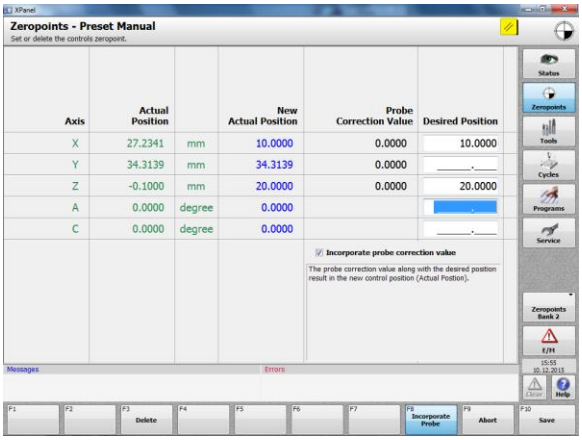


Illustration: The main menu of the zeropoints

F2 **Preset Manual:**

Definition of the workpiece zero point

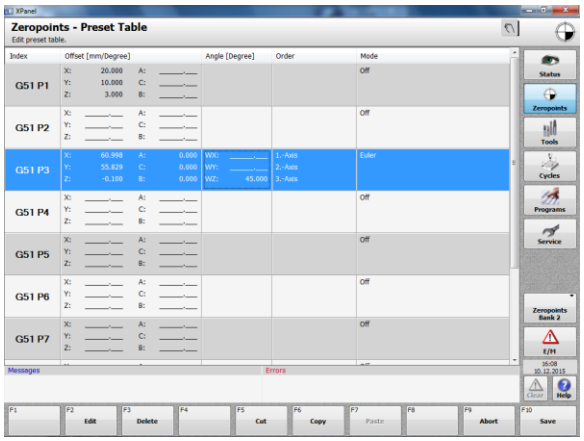
This input screen can be used to offset the workpiece zero point axis-specifically or to delete an offset again. The correction value of a button can also be included in the calculation. An offset is shown axis-specifically in the status display.



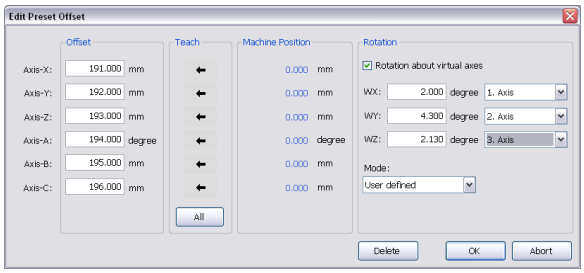
F3

Preset Table:

Saved preset offset
In the same way as the zero point offsets, a preset offset (G51) can also be stored. In contrast to the zero points, the preset table contains all 10 possible groups.
Preset offsets can be stored as the result of measurement cycles or entered directly by the user.



F2: Edit selected preset group

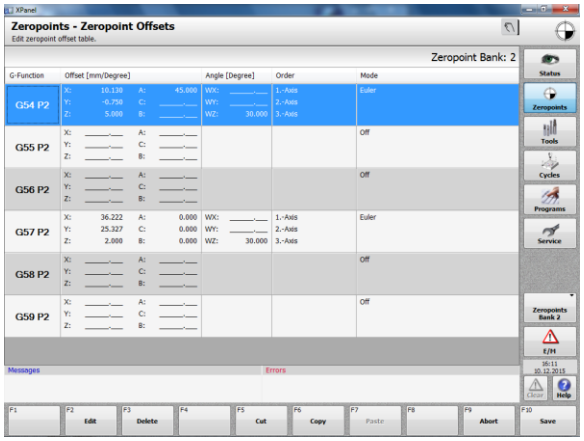


F3: Delete a preset group
F5-F7: Cut & Paste functions for a group
F9: Discard changes
F10: Save changes

F4-F10

Zero point Offsets
Groups 0 to 99

The groups 0 - 5 are directly addressable. All other groups are selected by explicitly specifying the group in an input dialog.
Each group contains the translatory and rotatory offsets for the G functions G54-G59. The parameter modification of a zero point offset group is done in the same way as that of a preset group.



NOTE:
At any time, only one application or only the CNC is allowed to open the preset table or a zero offset table. Otherwise a pop-up window containing information to that effect will open.

Tool management

Overview of functions

- any number of tool magazines
- 100 tool magazine slots per tool magazine
- clear overview of the magazine slots
- tool status information in the magazine table
- any number of tools per tool type
- input forms tailored to tool types
- variable or fixed coding of slots
- duplo tools
- tool life control / wear control
- 3 different types of status indication
- limitation of maximum speed
- speed control
- 9 pairs of correction values
- Tool-dependent measuring data input for the laser tool measurement system

Numerous adaptation possibilities at the machine:

- chain or disk magazine, pick-up, manual change in any combination up to a total of 100 locations,
- correction of position for every single magazine location,
- programming of the tool change procedures in G&M code format

Different options:

- tool can be changed, despite tool life has expired ,
- interruption of the NC program during speed reduction of the tool management,
- in case of tool magazine problems, all tools can be changed manually without intervening within the tool management,
- worn-out tools can be disabled by the breakage control, but NC program is not aborted when an error occurs.

The main menu of the tool management

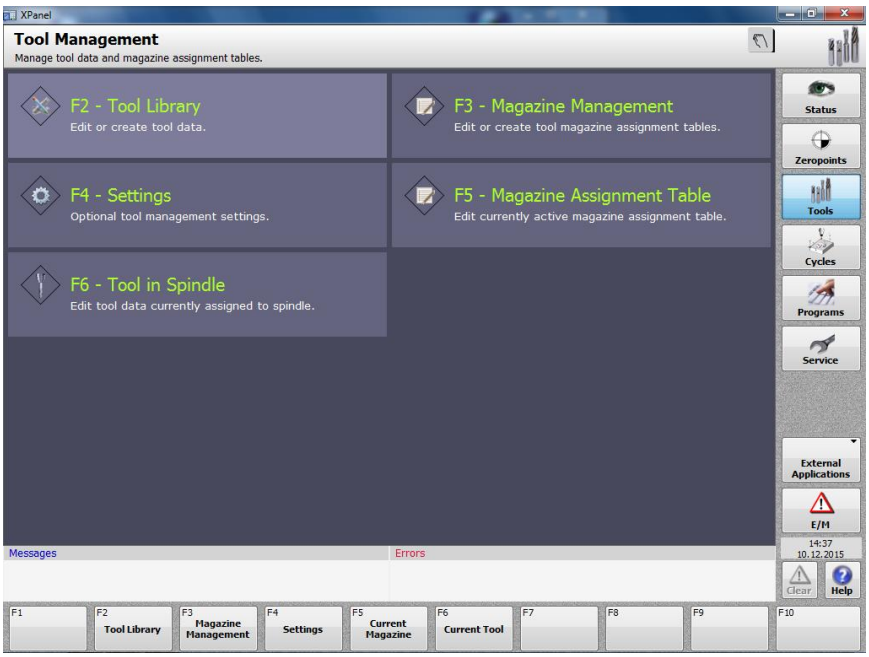
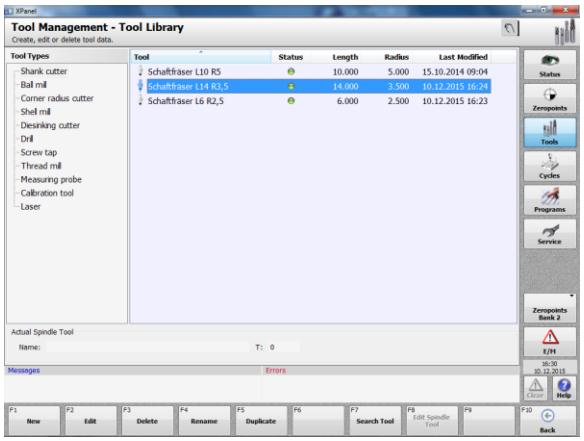


Illustration: The main menu of the tool management

F2 Tool Management / Tool Library

As a result of the two-level tool type / tool display, a tool can be quickly created, edited, deleted, renamed and duplicated. Moreover, all tool magazines can be searched for the selected tool.

Since the spindle tool must be often modified, it has been placed on a separate function key (F8).



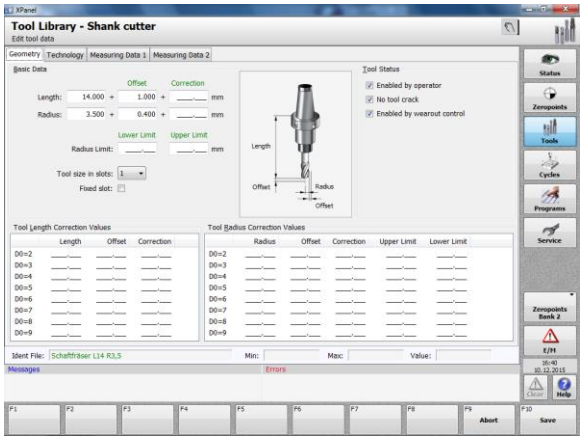
After selecting the tool type, all tools of this type will be shown in an overview. As concise information, status, length, radius and modification date are available. Moreover, the name and T number of the spindle tool are shown. An hourglass in the tool column identifies the spindle tool.
The Tool and Modification date columns can be sorted. The status display immediately shows whether the tool can still be used or must be replaced.
The selected tool can be edited by pressing "Edit" function key, the Enter key or by double-click.
For each tool type, an input form tailored specifically to that type is available.

Note:
There are situations in which the CNC does not allow a tool file to be edited. Conversely, the program sequence is stopped when the magazine assignment table is edited by the user.

Tool input form
Example shank cutter:

The forms of the individual tool types all have the same structure. One page for the geometry definitions, one page for the technology data and, depending on the complexity of the tool, 1-2 measuring data pages.

Geometry:

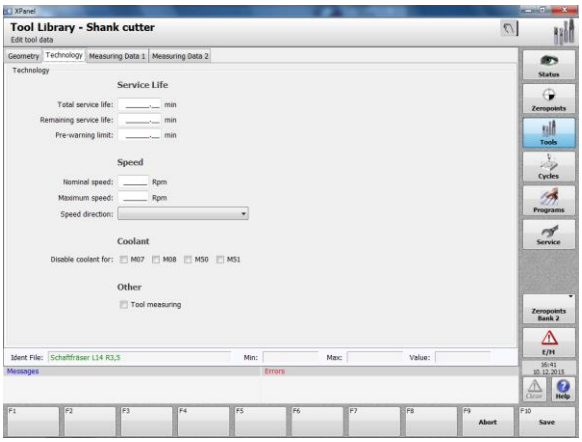


- Length, offset and correction are added by the control and the result will be set as active tool length when changing the tool.
- Radius, offset and correction are added by the control system and the result will be taken into consideration for compilation.
- The radius dimension limit will be checked when loading the tool. If the tool to be loaded is outside the range limits (wear), the duplo tool will be loaded, if one has been defined and is valid. If this tool does not meet the requirements either, machining will be aborted and an error message output.
- The tool size in slots defines the size in magazine slots (see magazine assignment)
- Fixed slot defines whether the tool is always managed on the same slot in the magazine. If the slot coding is variable, the CNC will search a suitable magazine slot, depending on the tool size.

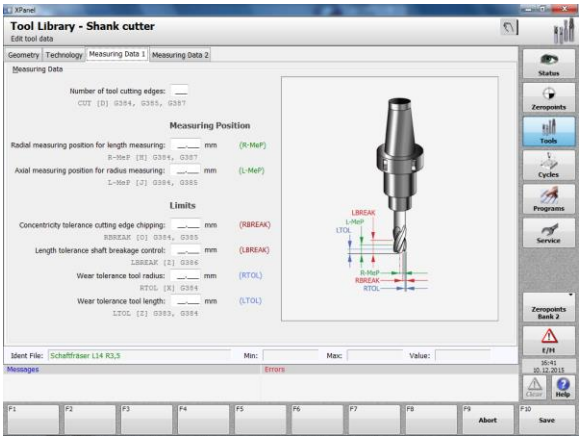
- Additional correction values: Apart from the length and the radius of the tool, it is possible to call up from the NC program the length and radius data for 8 further pairs of correction values. To this end, the NC address "D" or "D0=" is used with the number of the desired pair of correction values. When calling the tool, the first pair of correction values will automatically become active if no other pair has been activated with D2 (or D0=2) to D9 (or D0=9).
  Tool status: To be able to use a tool, all three check boxes must be checked. Tool breakage can be changed by measurement (load cell or laser), while wear control is influenced by the tool life and the radius dimension limit. If one check box is not checked, the status of the magazine assignment table (ST column) will be set to red.

Technology:

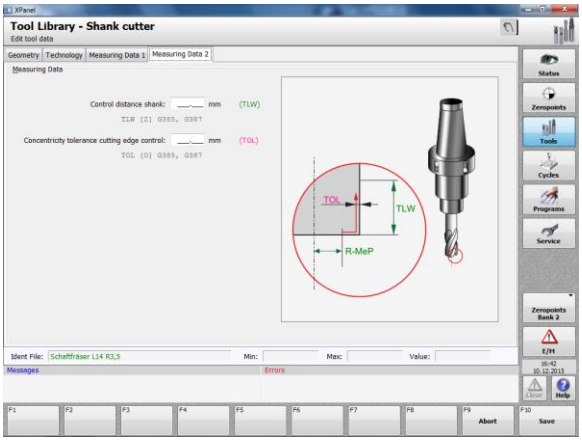
This page contains stored data on the tool life, speeds and cooling.
If no speed has been defined in the NC program, the nominal speed will be used. If too high a speed has been defined, the maximum speed will be used.



Measuring Data 1:
Here the input parameters for measuring the tool are defined.

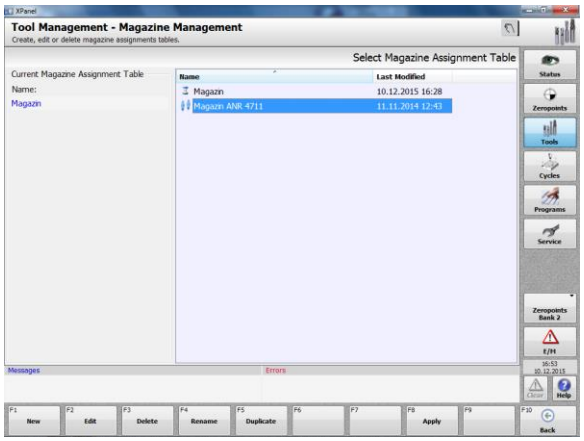


Measuring Data 2:
Depending on the complexity of the tool, further tool parameters are defined.



see BLUM cycles operating instructions

F3 Magazine Management



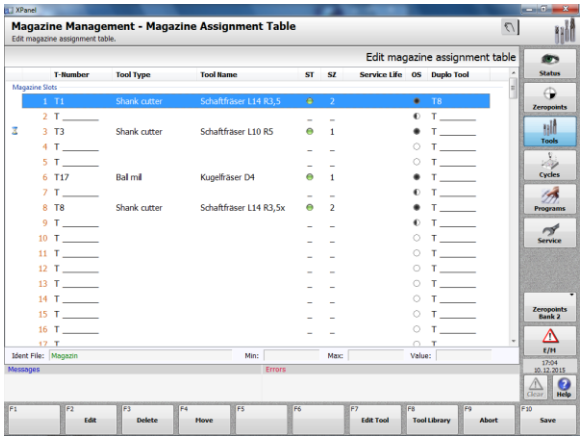
Magazine selection:

The control can manage any desired number of magazine assignment tables. Since these tables are stored as files, only characters also allowed in the NT file system (NTFS) can be used. The name of magazine currently used in the CNC is shown on the left-hand side of the window. The right-hand list box shows all assignment tables, the current table being identified by an hourglass.

- **F1 New:** a new empty table is created.
- **F2 Edit:** the selected table is opened and can be edited.
  Instead of F2, it is also possible to use the Enter key or a double click on the selected table.
- **F3 Delete:** the selected table is deleted.
- **F4 Rename:** the selected table is renamed.
- **F5 Duplicate:** the selected table is copied
- **F8 Apply:** the selected table becomes the current magazine assignment table
- **F10 Back:** return to the previous level

Note:
The current magazine assignment can be neither deleted nor renamed.
There are situations in which the CNC or other applications does not allow machining. Conversely, the program sequence can be stopped when the magazine assignment table is edited by the user.

F5 Magazine Assignment Table



Per assignment, up to 100 slots are available, depending on the tool sizes. They can be divided via the configuration (see Service/Tool Management) into magazine, pickup and manual slots.

An entry in the magazine assignment table consists of the T-number, the tool type, the tool name, the tool status, the tool size, the slot assignment and the duplo T-number.

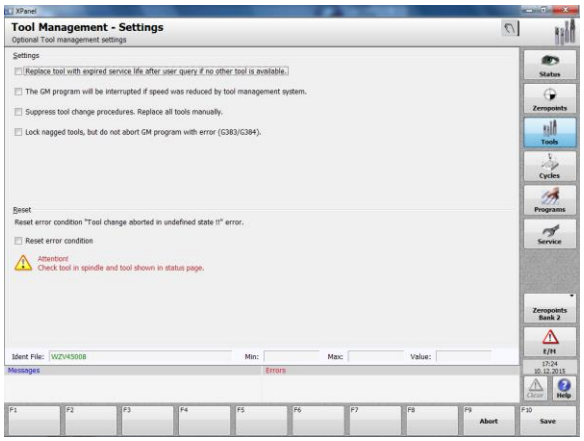
If a slot contains faulty entries or if the tool has not been defined sufficiently, an exclamation mark will appear in the unnamed column (on the far left). The spindle tool is identified by an hourglass in the 1st column as in the current magazine assignment.

- **F2 Edit:** edits the selected slot entry. Instead of F2, it is also possible to use the Enter key or a double-click. The T-number may have a max. of 8 digits and only appear once in the assignment. The tool type is selected from all available types via a list. The tool is also selected according to tool type. If the tool, due to its size, does not fit in this slot, a message is output during the selection. The next 3 columns are set automatically, i.e., the data from the selected tool file are shown. ST is green if all three check boxes of tool status have been checked in the tool file. GR indicates the size of the tool and BE the slot assignment. In the above illustration, slot three contains the tool T2 of the shank cutter type. The tool name is "Shank cutter L14 R3,5". The tool is ready for use has a size of 2 slot unit, which is why it requires half of its adjacent slots. The duplo tool is T8.
- **F3 Delete:** deletes a slot entry from the table. The spindle tool must not be deleted.
- **F4 Move:** marks a slot entry in the table for moving. Pressing this key produces a red frame around the selected slot. The entry can be moved using the UP/DOWN cursor. The moving cycle is finished by pressing the Enter key. Moving by means of the pointer device (mouse, touch) is also possible. The slot to be moved is selected with left mouse button and moved while holding down mouse button. While it is being moved, the slot also has a red frame. The cycle is finished by releasing the left mouse button.
- **F7 Edit Tool:** the tool of the selected slot can be edited directly.
- **F8 Tool Library:** the tool library is opened.
- **F9 Abort:** editing is finished without applying the changes.
- **F10 Save:** editing is finished, applying the changes.

Note:
The tool names are unique and type-independent. For example, if there is a drill "abc", there can be no shank cutter "abc".
There are situations in which the CNC or another application does not allow machining. Conversely, the program sequence can be stopped when the magazine assignment table is edited by the user.

F4 Tool Management - Settings

Optional tool management settings.



1. Tool with expired tool life is inserted after inquiry, if no other tool is available.

This option is intended to facilitate that a tool is used longer than the entered tool life.

☐ If the tool management does not find a tool with remaining tool life, the execution of the G&M code program is cancelled.

☒ If the tool management does not find a tool with remaining tool life, the G&M code program is only interrupted. This fact is indicated by a message. The execution of the G&M code program can be continued with the first released tool without remaining tool life. The NC program can be canceled with NC-Reset.

2. G&M code program is interrupted by the tool management in case of speed reduction

If a value for the maximum speed of the current tool is entered and the speed programmed in the G&M code program is higher than the maximum speed the speed of the spindle is reduced automatically.

☐ The spindle speed is reduced to the maximum speed of the tool without interruption of the G&M code program.

☒ In case of a speed reduction the G&M code program is interrupted with a message to provide the possibility to decide whether to cancel the program or to continue at reduced speed. A new NC start continues the G&M code program at reduced speed.

3. Suppress tool change sequences. All tools are changed manually.

To guarantee the operating function of the machine in case of damage of the mechanics for the tool change, all tool change sequences can be forced to manual change without need to change the magazine configuration and the magazine assignment table.

☐ Standard operation
☒ All tool changes are executed as "Manual change".

4. Lock worn-out tools, but do not abort GM program with error (G383/G384).

Cycles G383 and G384 can be used to carry out a wear test of the length and radius. To do so, a wear tolerance is specified. If the measured wear exceeds this wear limit, the tool will be locked by the cycle, and an error code will be returned to the CNC.

☐ The NC program will be finished with an error message if a cycle identifies the tool as "worn-out".
☒ The NC program will be continued with a worn-out tool. The next tool change of this tool will load the duplo tool, if available, or finish the NC program with an error message.

RESET

RESET of the error state of the tool change has not been canceled in the defined state!

The tool change is monitored internally by the control. If, due to a power cut or similar events, an undefined state should occur during the tool change, the following tool change is blocked. In this case, a tool change leads to a program abort and the error message „`Tool change has not been canceled in defined state!! Check tool management and reset error`".
The machine operator must check the tool management, compare it with the ACTUAL state in the tool magazine and in the spindle, and reset this error state. After this, a tool change is possible again.

☐ Standard operation
☒ Reset error.

CAUTION



It is absolutely necessary to check the tool in the spindle and the tool in the position menu!

Cycles

The main menu of the cycles

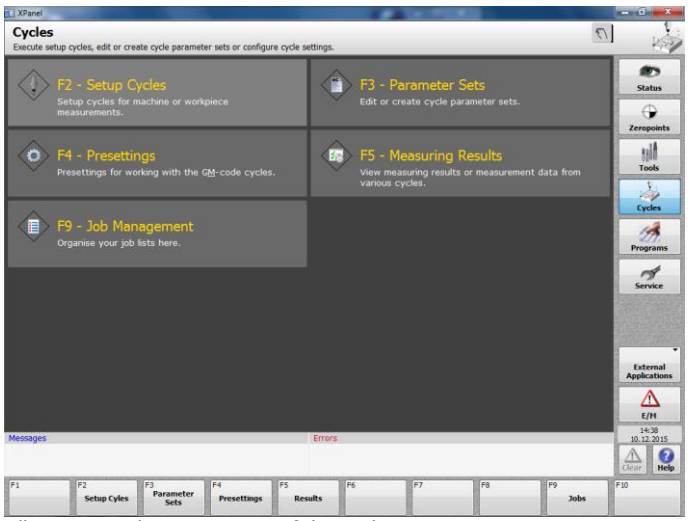


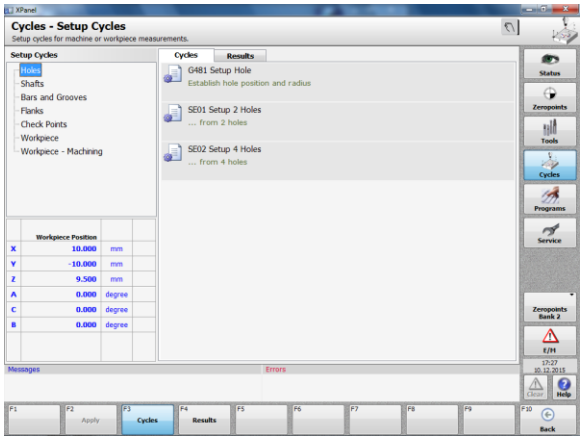
Illustration: The main menu of the cycles

Here it is possible to execute setup cycles, edit the parameter sets cycle, establish presettings, display measuring data and organize job lists.

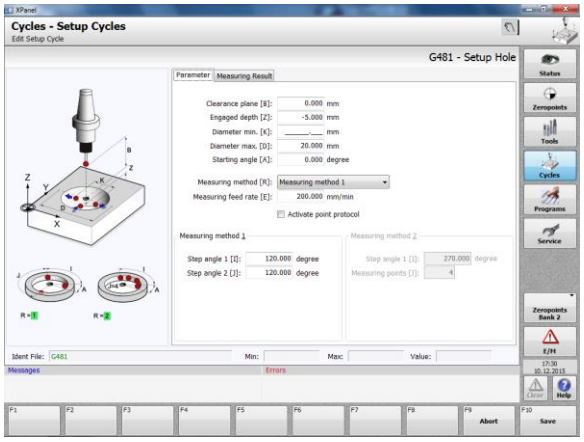
F2 Setup Cycles

All setup cycles can be executed from here, and the measurement results can be saved. The setup cycles are divided into groups:

- Drilling operations
- Shafts
- Ligaments and grooves
- Sides
- Space points
- Workpiece
- Workpiece - Machining



For all available setup cycles, input screens containing parameters for determining the measurement results are available.
There is one page each for the setup parameters and for the measurement results.

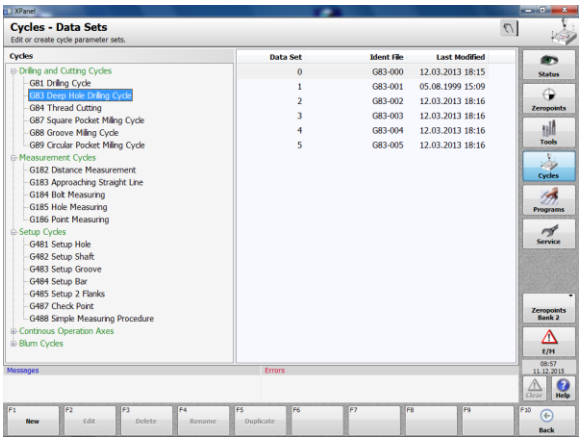


Further information on all setup cycles can be found in the "G&M code programming instructions".

F3 Cycles – Data Sets

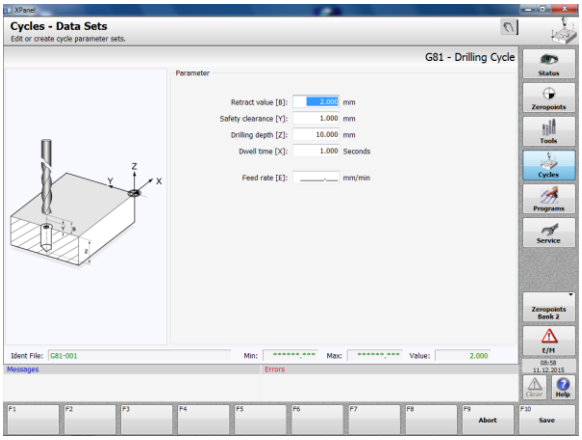
For all cycles, a max. of 1000 (0-999) parameter sets can be created, edited and saved. The cycles are divided into five groups:

- Drilling and milling cycles
- Measuring cycles
- Set-up cycles
- Continuous operation axes
- Blum cycles



Edit or create cycle parameter sets:

Here data set 1 is given as an example of the G81 drilling cycle.



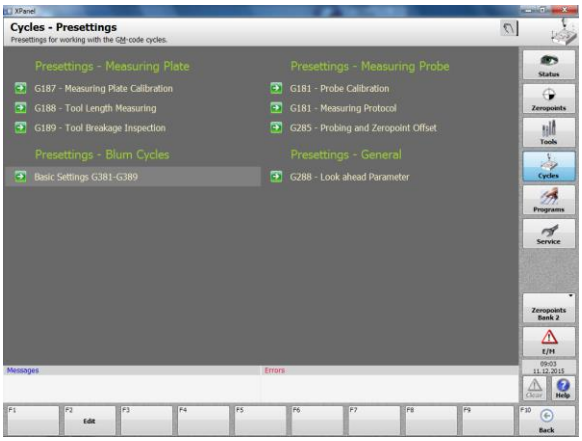
Further information on the individual cycles can be found in the "G&M code programming instructions" or in the Help

F4 Cycles - Presettings

For some NC block cycles, special presettings can be established and saved.

Here is the main menu for the following cycle groups:

- Measuring plate (G187, G188, G189)
- Measuring probe (G181, G285)
- Blum cycles (G381-G389)
- General (G288 Look ahead parameter

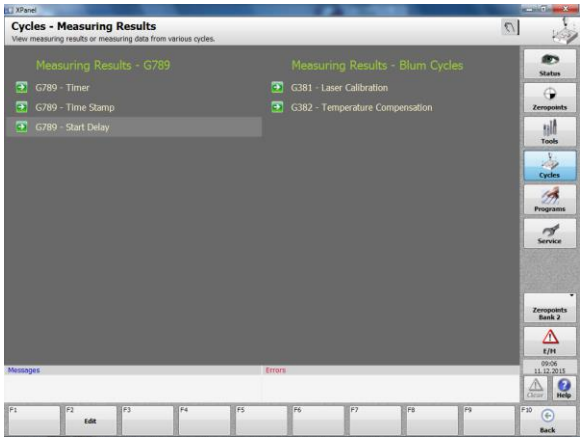


F5 Cycles – Measuring Results

View measuring results or measuring data from various cycles.

Here is the main menu for the following measuring results:

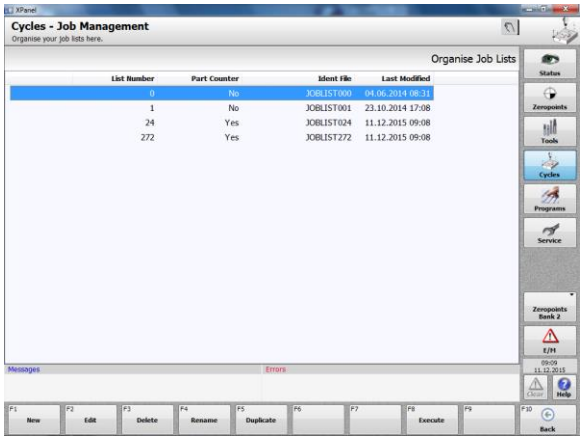
- G789 – Timer
- G789 – Time Stamp
- G789 – Start Delay
- G381 – Laser Calibration
- G382 – Temp. Compensation



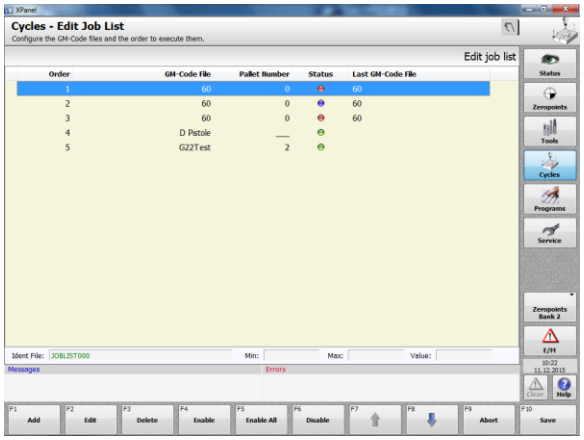
F9 Cycles – Job Management

Here it is possible to define job lists (000-999), which allow the following functions:

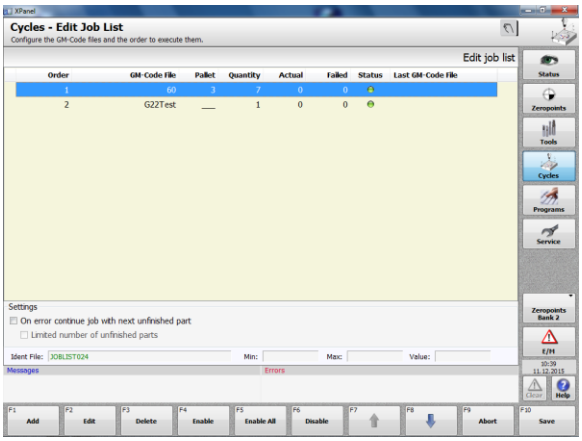
- to start more than one NC program in a row,
- to allocate a pallet number to every program
- use a part counter
- to restart an NC program by the PLC after being aborted by, e.g. ERROR Exit.
  The execution of the job list is continued with the next released NC program.



Note:
Part counter = No



Note:
Part counter = Yes



Further information on the job lists can be found in the online help "Ncsatz_Bedienung.chm".

A parameter description and further information on the individual cycles can be found in the "G&M code programming instructions" or in the Help

Programs

The main menu "Programs"

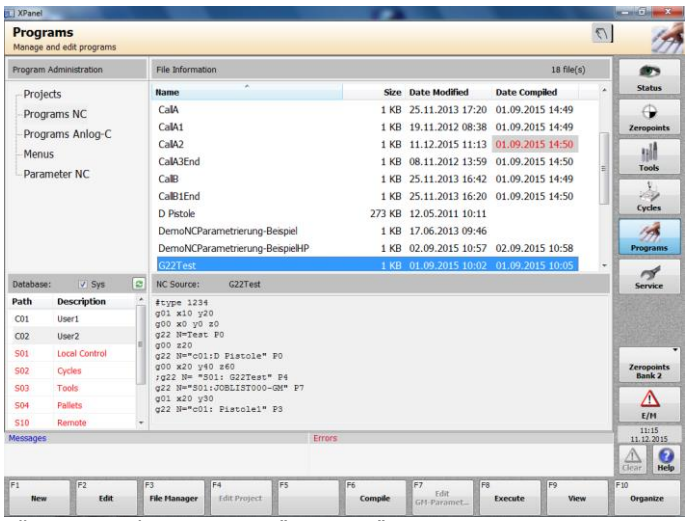


Illustration: The main menu "Programs"

The program-specific screen is divided into four areas.
The two left-hand windows are always the same, while the two right-hand windows depend on program management and the current database path, which can be set in the left lower section.

In the upper section **File Information**, projects, NC block, expert (anlog-C) programs and menu filenames are shown.

In the lower section, either ID files of anlog-C programs or menus are selected or NC blocks or project files are displayed.

Sorting can take place both in B1 and B2 according to column. Since are designed as split screen, the display ratio of B1 and B2 can be adjusted.

General: the lists in B1 and B2 can be sorted. The column in which sorting takes place and whether it is done ascending or descending order, can be seen from the triangle in the column heading.

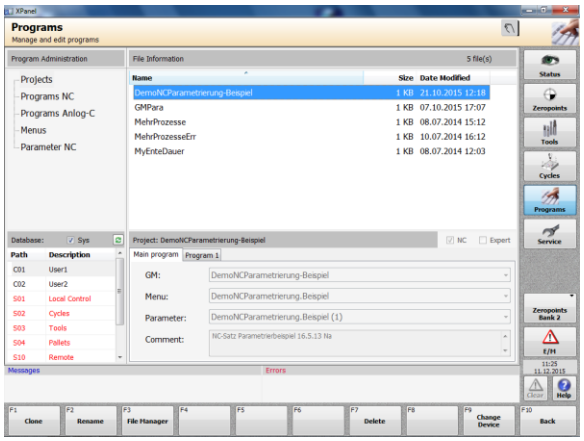
 This Refresh button is used to read in new database information.

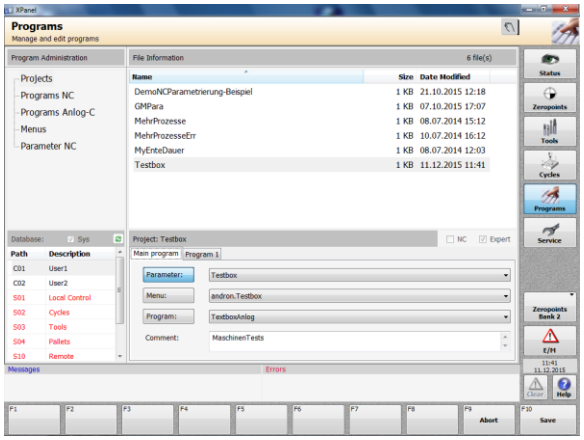
Array File Information

Projects

Here both NC block and Expert Mode projects can be created. A new project is created by pressing the function key F1. The project is edited by pressing Edit Project (F4). In an Expert Mode project, first the menu is selected, and then IDN and the program are selected. Only those files are displayed that fit the menu in terms of type. Optionally, as with the NC block project, a commentary can be included.

The existing NC block or Expert Mode project is then displayed in the right upper window.





An Expert Mode project is advantageous when starting up a program. Once the project has been selected, the ID file can be modified by means of the menu by pressing F2 and then transferred to the CNC together with the relevant program by pressing F4. In the absence of a project, in order to edit the IDN, first the item Menu would have to be selected in Program Administration, then the menu and IDN would have to be selected and then IDN modified. To load the program and IDN, the item anlog-C programs and then the program and IDN would have to be selected.

F1 Creates a new project
F2 Edits the project NC block or the project ID file (depends on the project type)
F3 File Manager: Import/Export of database files
F4 Edits the project
F5 Edits the PRKON file of the project NC block (option in xpanel.exe.config)
F6 Compiles the project NC block
F7 Edits the NC parameters if the project NC block is parameterizable
F8 Compiles the project NC block, if necessary, and transfers it to the CNC or, in the case of an Expert Mode project, loads IDN and program to the CNC.
F9 Visualization of the NC block by means of 3Division Basic (requires activation in dongle)
F10 Behind organization, a new function bar is hidden. It allows NC block projects and IDNs to be renamed and also to be deleted. When deleting, a multiple selection is possible as with the Explorer. Upon pressing F3, the file manager can be started and, upon pressing F9, the database directory can be changed.

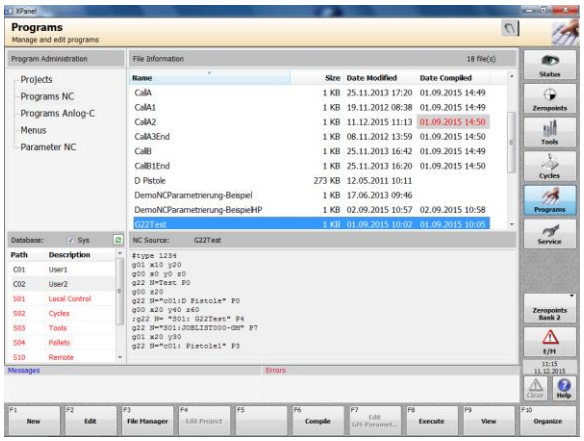


Note:
When renaming an NC block or an IDN, the project files are not renamed automatically.

Array
File Information

NC Programs

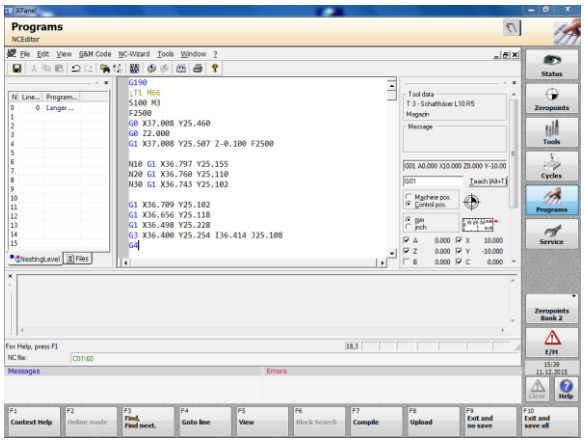
Here new NC programs can be created, modified, visualized, compiled and transferred to the CNC.



The function keys described below only refer to this main window

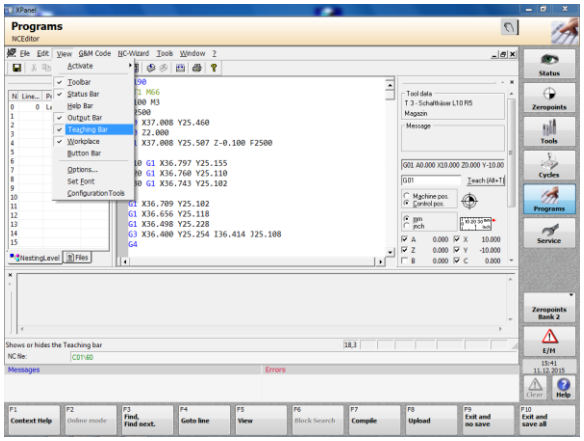
F1 **New:**
Creates a new NC block

F2 **Edit:**
Edits an existing NC block using a convenient NC editor



andronic NC Editor

The **andronic NC Editor** is an optimal tool for editing NC block files of any size. It contains a configurable NC syntax with tooltips and integrated help. An NC wizard and the option of data transfer from the clipboard complete the software package.



F3 File Manager: makes Import/Export of database files easy for the user

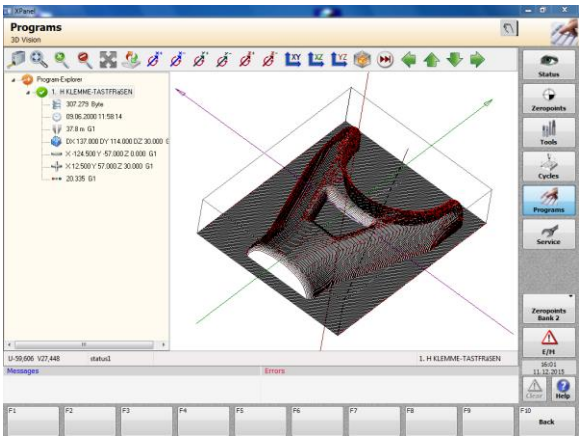
F5 Edits the PRKON file of the NC block (option in xpanel.exe.config)

F6 Compiles an NC block

F7 Edits the NC parameters within a project if the NC block is parameterizable

F8 Compiles an NC block, if necessary, and transfers it to the CNC.
Note: By entering "CompileAlways" in the XPanel.exe-Config, you can control whether the NC block is to be compiled always or only if it was modified. Default: false.

F9 Visualization of the NC block by means of 3Division Basic
(requires activation in dongle)



F10 Behind organization, a new function bar is hidden. It allows NC blocks to be renamed and also to be deleted. When deleting, a multiple selection is possible as with the Explorer. Upon pressing F3, the file manager can be started and, upon pressing F9, the database directory can be changed.

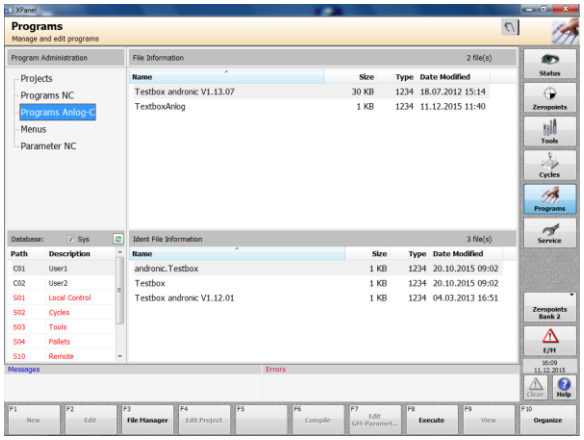
In *File Information*, the existing NC blocks are visualized. If an entry is missing in the "Compiled on" column, the program was not yet compiled. If the entry is shown in red, the NC block source was modified after compiling.

In *File Information*, the first lines of the selected NC block are shown.

Array
File Information

anlog-C Programs

Here program and IDN are selected and transferred to the CNC for execution.



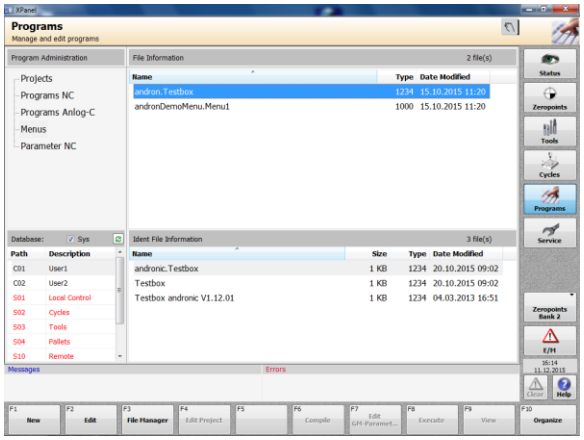
F8 Transfers program and IDN to the CNC

F10 Behind organization, a new function bar is hidden. It allows the ID files that match the selected program to be renamed or deleted. Upon pressing F3, the file manager can be started and, upon pressing F9, the database directory can be changed.

Array
File Information

Menus

Here new ID files can be created or existing ones edited. Depending on the selected menu, only those IDNs can be selected that fit this menu.



F1 Creates a new IDN. The name offered for the IDN is the menu name. If this name already exists, the name is formed from the menu name + index.

F2 Edits an NC block or an ID file (depends on the project type)

F3 File Manager: makes Import/Export of database files easy for the user

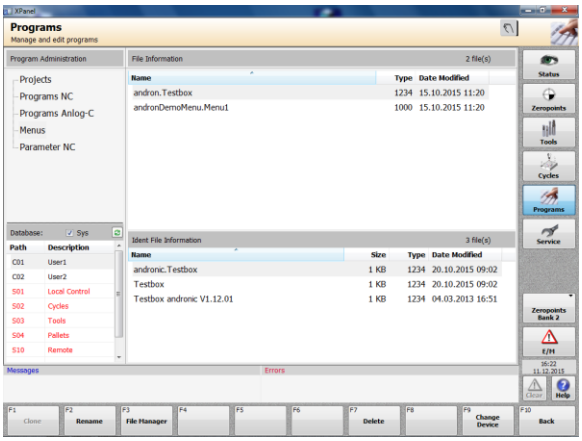
F10 Behind organization, a new function bar is hidden. It allows the ID files that match the selected menu to be renamed or deleted. Upon pressing F3, the file manager can be started and, upon pressing F9, the database directory can be changed.

Note: F2 is only enabled if an IDN was selected and is focused.

Array Database

In the **Database** area, the existing valid databases are displayed.
If the user has administrator rights, the system database paths are also displayed. By default, they are shown in red. The color can be changed by making a "SystemPathColor" entry in XPanel.exe.Config.

If the user has administrator rights, but does not want the system paths to be listed, he can show/hide them via the "Sys" check box at the top right in the database area. This setting remains valid after switching off the control. Moreover, all database paths are updated by pressing the Sys key. This is why the key is also valid if the user has no administrator rights.



 This button is used to read in new database information.

Sometimes, the reaction times may be quite long. This is caused by analysis of physical directories that are not present.

The *Database* area can be activated either via a pointer device or by pressing F9 *Change Directory* in the *Organization* toolbar. The data area cannot be activated by pressing the Tab key, but it can be left.

The Path column contains the short name for the database path.
Cxx (custom) stands for a user database path, xx indicating the path number.
Sxx (system) stands for the system database path, xx indicating the path number.

The Path column also shows the respective database directory as tooltip. Should the text in the Description column be too long, the complete description can also be shown as a tooltip.



Note:
Database paths can be created and changed via the system configuration (in Service\Systemprogramme). This must be followed, however, by reading in the data again by pressing Sys key or the Refresh button.

Service

The Service main menu

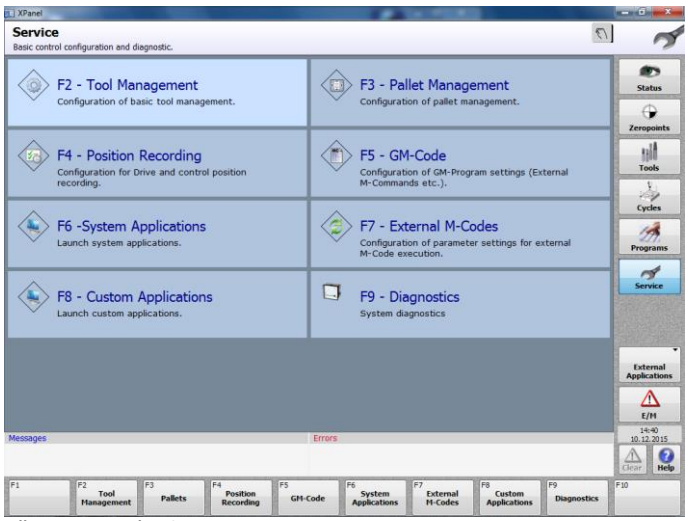


Illustration: The Service main menu

Here the basic configuration of the control and of the drives is established.

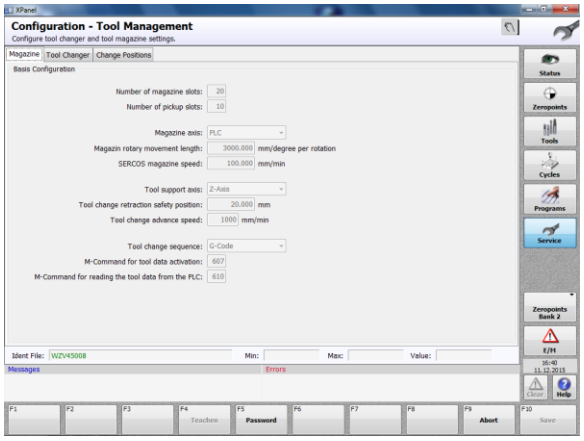
Changes to the control and drive configuration may only be made by trained skilled personnel!

Note: If the user has no administrator rights, he cannot used the functions F2 - F6 (possibly F7).

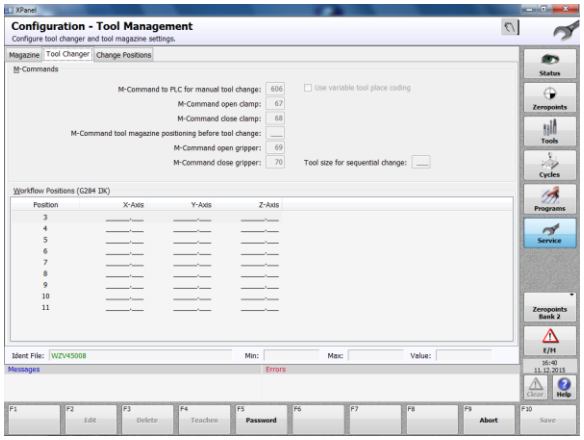
F2 Tool Management

Configure tool charger and tool magazine settings.

→ Magazine and change sequence



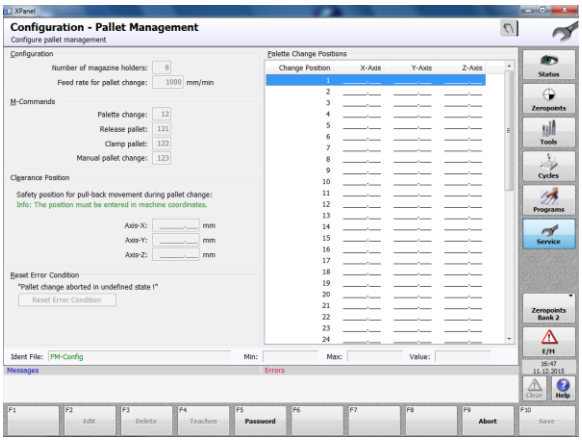
→ Tool Charger



Further information can be found in the online help file "Ncsatz_Konfiguration.chm".

F3 Pallet Management

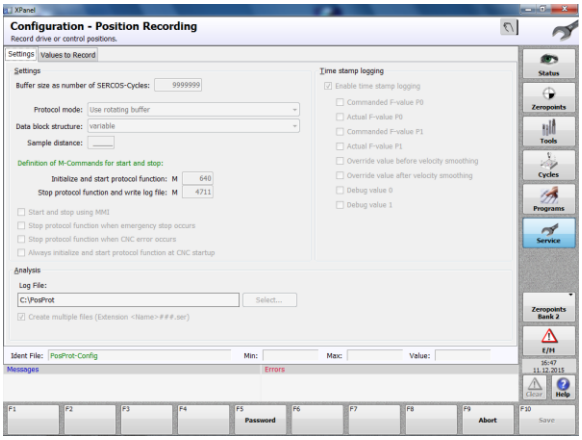
Configure pallet management



Further information can be found in the online help.

F4 Position Recording

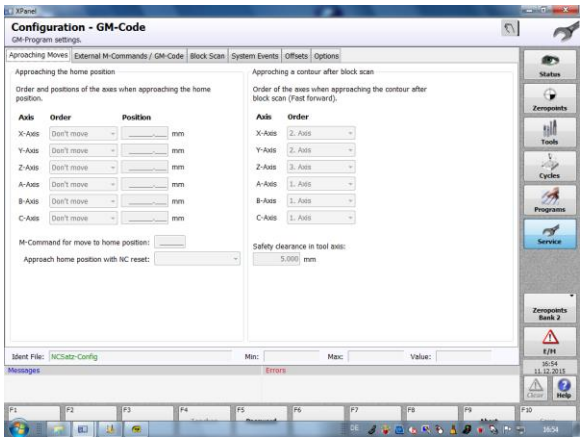
Record drive or control positions.



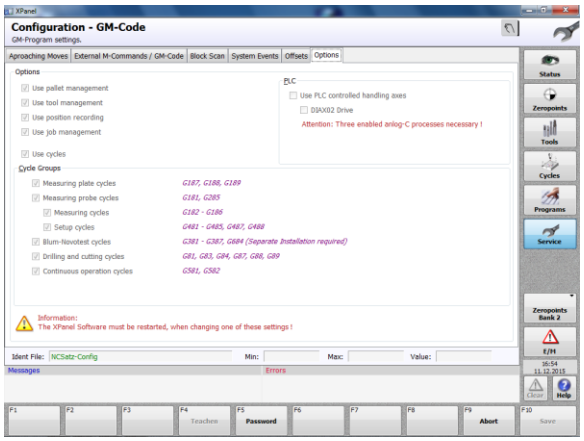
Further information can be found in the online help.

F5 Configuration – GM-Code

GM-Program settings (approaching moves, external M-Commands, Block Scan, special system events, offsets, options)

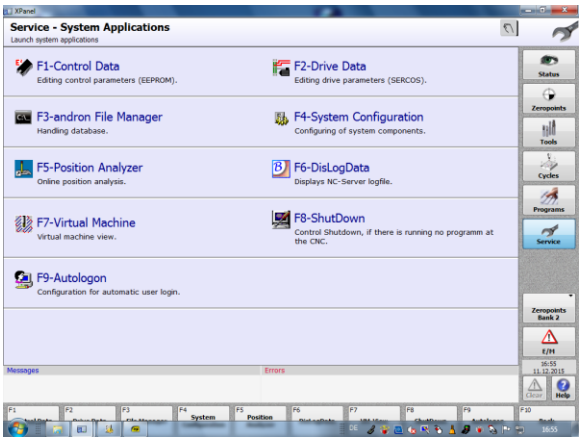


The pages External M functions and Options describe the functionality of the HMI in the main groups Cycles and Service.
Changes will not become effective until the XPanel is restarted.



Further information can be found in the online help file "Ncsatz_Konfiguration.chm".

F6 System Applications

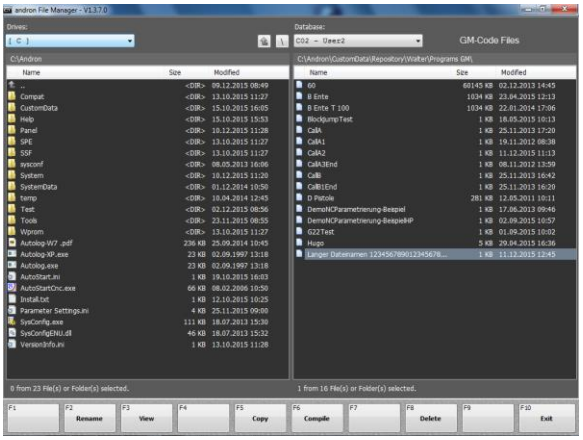


The system programs are used for configuring the control and the machine. The control should always be shut down by pressing F8 ShutDown, because this function first checks whether a program is still being executed on the machine. If this is the case, the actual shutdown is prevented.

F1-Control Data:
Editing control parameters (EEPROM)

F2-Drive Data
Editing drive parameters (SERCOS)

F3-andron File Manager
Handling database



F4-System Configuration
Configuring of system components

F5-Position Analyser
Online position analysis

F6-DisLogData
Displays NC-Server logfile

F7-Virtual Machine
Virtual machine view

F8-ShutDown
Control Shutdown, if no program is running at the CNC.

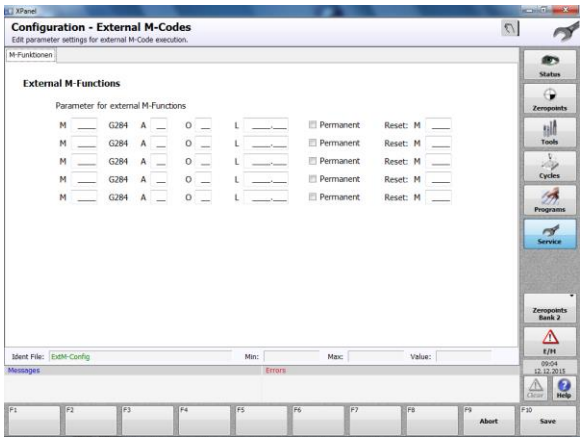
F9-Autologon
Configuration for automatic user login.

CAUTION



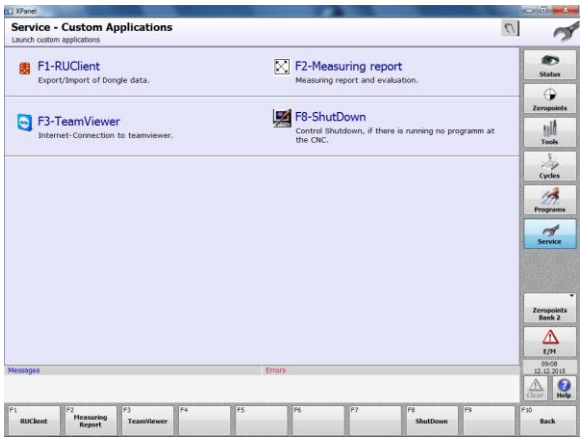
The general rule is:
Changes to the control and drive configuration may only be made by trained skilled personnel!

F7 External M-Functions



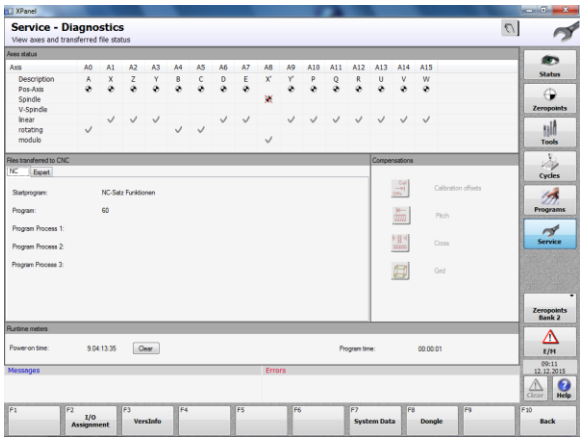
Note:
If there are no external M functions (no DLL installed), the function will be shown in the overview, but it cannot be selected.

F8 Service – User programs



Apart from the page of system programs, there is also a page on which user-specific programs can be stored. By default, 4 programs are installed. However, additional applications can be added via a configuration file.

F9 Diagnostics

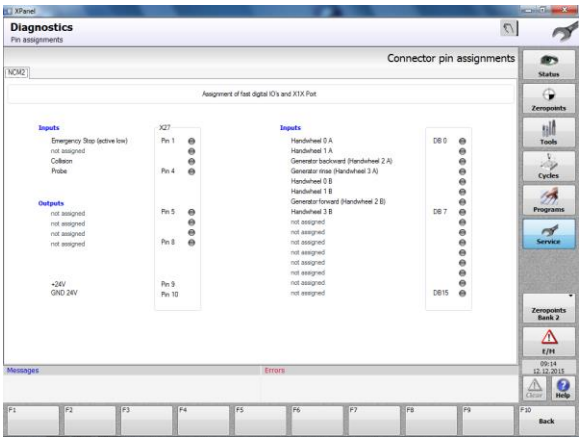


Diagnostics is designed not only to provide the user with information on the control and the machine, but also assist him with problems.
This may include axis status, compensations, I/O assignments, program and version information and dongle data. Under System Data, the user can view the assignment CMOS, manual, PLC and FastPLC parameters. As it is impossible to visualize all parameters at the same time, due to their large number, the user has the option of displaying the areas of interest.

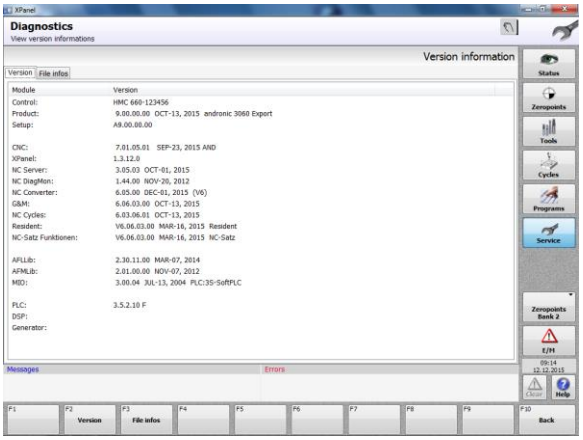
The start page provides information on

- **Axes status:**
  Existing axes, axis designator, type and reference status. Red denotes not referenced, green denotes referenced. The axis type - linear, rotating, modulo - is signaled by a check mark.
- **Files transfered to CNC:**
  This lists loaded programs, NC blocks and IDNs. In the execution of cycles, the information of the Expert Mode can also be of assistance.
- **Compensations**
- **Runtime meters:**
  Machine time is the time during which the control was switched on.
  Program time is the elapsed time of the NC block to be executed.

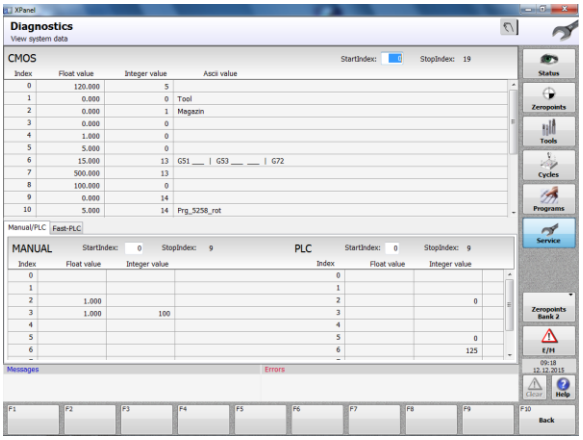
F2 **I/O assignment**
provides all the information on the signals of the 37-pin Sub-D socket, on the InterBus-S, the NCIO card and NCM X1X port.



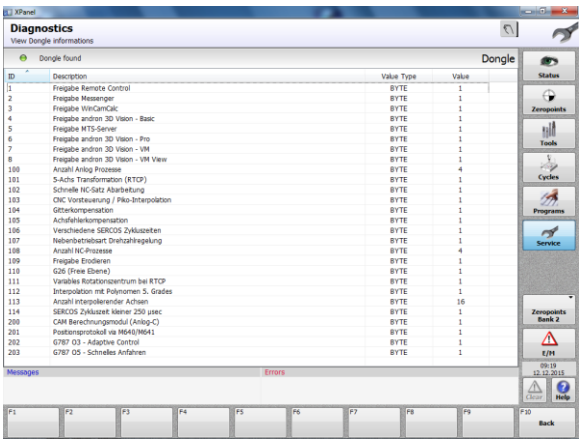
F3 **VersInfo**
provides the version information of individual control components, but also on another page an overview of the previous product versions (History).



F7 **System data**
provides system data. They include CMOS-RAM, manual, PLC and FastPLC data. The areas to be displayed can be defined by means of start and stop indices.



F8 **Dongle**
View dongle information



Error and Messages

Errors and messages that have occurred during operation are visible in the compressed Error/Message display under the main window. In contrast, in the E/M main menu, an entire page each is available for displaying error and message texts.

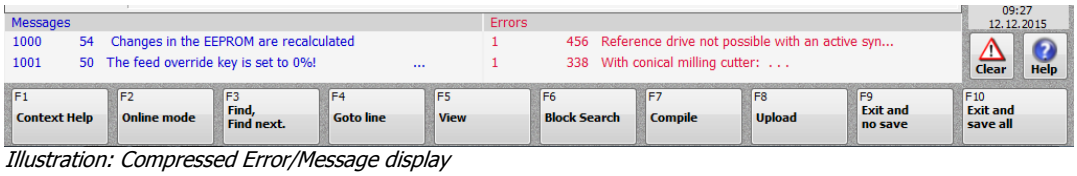


Illustration: Compressed Error/Message display

The error messages can be reset by using the ⚠ CLEAR function.

The main menu Error and Messages (E/M)

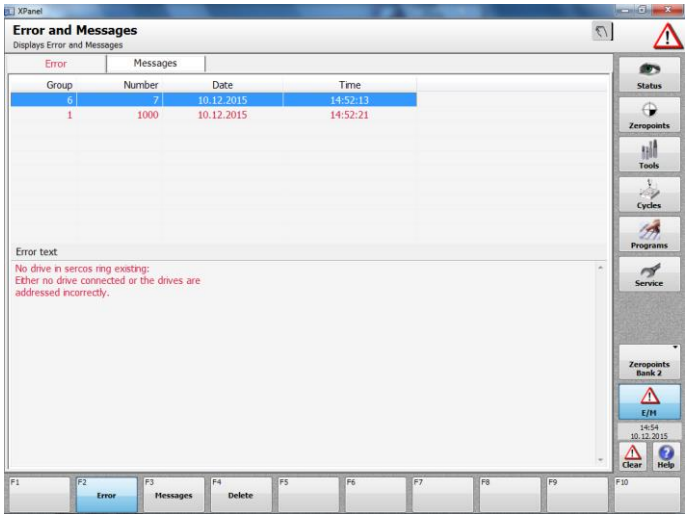


Illustration: The main menu E/M (Errors and messages)

Here errors and messages of the control and of the machine are displayed for the operator.

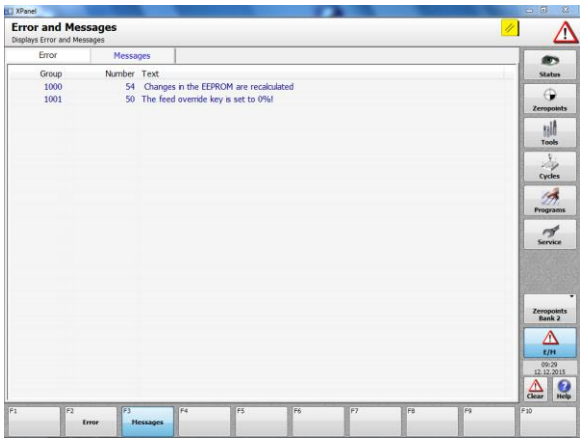
Errors are always shown in RED and messages in BLUE.

F2 Errors

To switch between errors and messages, press the function keys F2 and F3 or click on the tab at the top of the window.

F3 Messages

In contrast to error messages, message texts cannot be deleted by the user! This must be handled by the CNC or PLC.



Revisions

Version	Date	Additions and changes	Initials
V 0.9	07.05.2009	First edition	Vol/Na/Pa
V 1.0	14.11.2011	Revised edition	Vol/Na/Pa
V 1.1	08.12.2011	Revised edition	Vol/Na/Pa
V 1.2	08.03.2016	Complete revised edition: • The renaming to LTI Motion • Windows 7: New operating system, policies and user profiles • Virtual machine operating panel ANM07 • New general safety instructions and warnings • Small additions and changes	Na, Pa
V 1.3	03.07.2017	Additions of the compensations: ▪ Parametric volume compensation ▪ parametric rotations axes Different small modifications	Pa
V 1.4	In preparation	NEW: Windows 10 operating system settings, user rights, etc.	

LTI Motion GmbH
Schlätterstraße 2
88142 Wasserburg · Germany
Tel.: +49 8382 9855-0
Fax: +49 8382 9855-50
E-Mail: info@lti-motion.com
www.lti-motion.com

Artisan Technology Group is an independent supplier of quality pre-owned equipment

Gold-standard solutions

Extend the life of your critical industrial, commercial, and military systems with our superior service and support.

We buy equipment

Planning to upgrade your current equipment? Have surplus equipment taking up shelf space? We'll give it a new home.

Learn more!

Visit us at **artisantg.com** for more info on price quotes, drivers, technical specifications, manuals, and documentation.

Artisan Scientific Corporation dba Artisan Technology Group is not an affiliate, representative, or authorized distributor for any manufacturer listed herein.

We're here to make your life easier. How can we help you today?

(217) 352-9330 | sales@artisantg.com | artisantg.com

ARTISAN® TECHNOLOGY GROUP