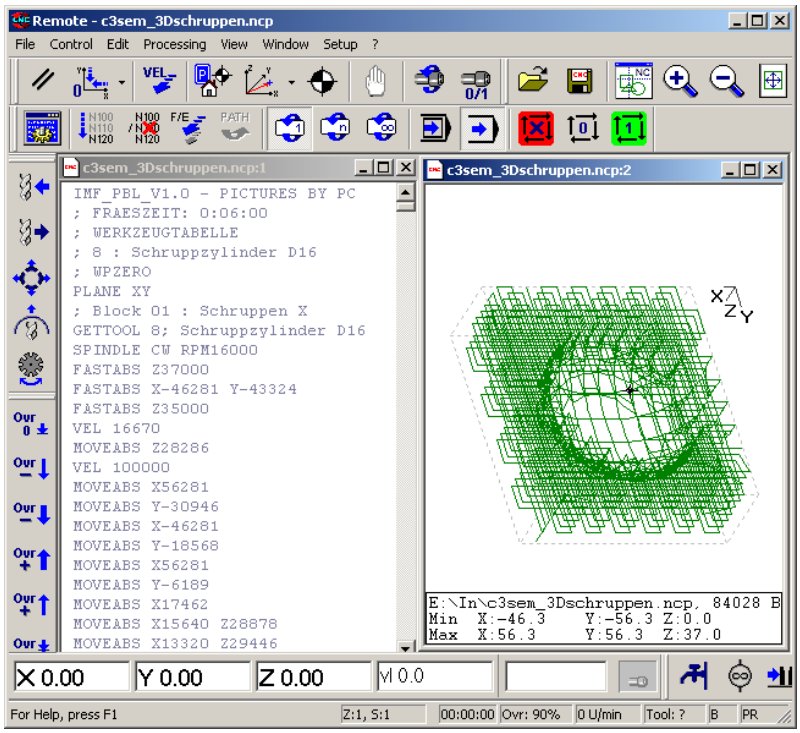
iselautomation KG




# *isel*-RemoteWin

## Operating Instruction

www.isel.com

isel®

To the Manual:

In this manual you find same symbols calling your attention to important informations quickly.

| **Caution:** | **Example:** | **Hint:** | **Information:** |
|---|---|---|---|
|  |  |  |  |



In spite of all care printing errors and mistakes cannot be ruled out completely.
Suggestions for improvement and notes on errors are always welcomed.



Produser: Fa. ***isel***automation KG
Bürgermeister-Ebert-Straße 40
D-36124 Eichenzell

Tel.: (06659) 981-0
Fax: (06659) 981-776
email: automation@isel.com
http://www.isel.com

Stand: 20/2003

# Content

# 1 Introduction

## 1.1 Product

**Product:**

**Remote** is a new interpreter program to output **isel-NCP files** or **isel-CNC files** for processing technologies drill, stick, jet cutting, laser cutting or -welding for PC based on the operating system **Windows**. Remote is the replacement of Remote based on DOS.

All **NCP files** generated **by the CAD/CAM-Program ISY 2.0 / ISY 3.0** or made with other methods can be started immediately without compiling or conversation.
In Remote you can also use all **CNC files**, compiled in **ProNC from an application program written in ISO-** or **PAL-syntax.**

Remote contains windows with particular buttons to start the desired function by mouse click.
Functions, often be used, like program start, program stop, reset, reference run, you can also activate by keyboard.
**Displays of positions, velocities or spindle speed** inform you about the **current machine status** during processing.

Optional you can do your inputs with an Operating Panel.

It is possible **to change the NCP file** with the **integrated text-editor** to optimise the processing. Following you can start without compiling or converting.

## 1.2 Installation

**System requirement:**

- **PC from Pentium II**, 333 MHz (at least)

- **32 MByte memory** (at least)

- Controller IMC4 / IMS6 or C142/4 executable by Win98 / WinNT / Win2000

- isel-Servo machines with UPMV4/12 executable by Win98

- isel-Servo machines with CAN-Controller executable by WinNT 4.x

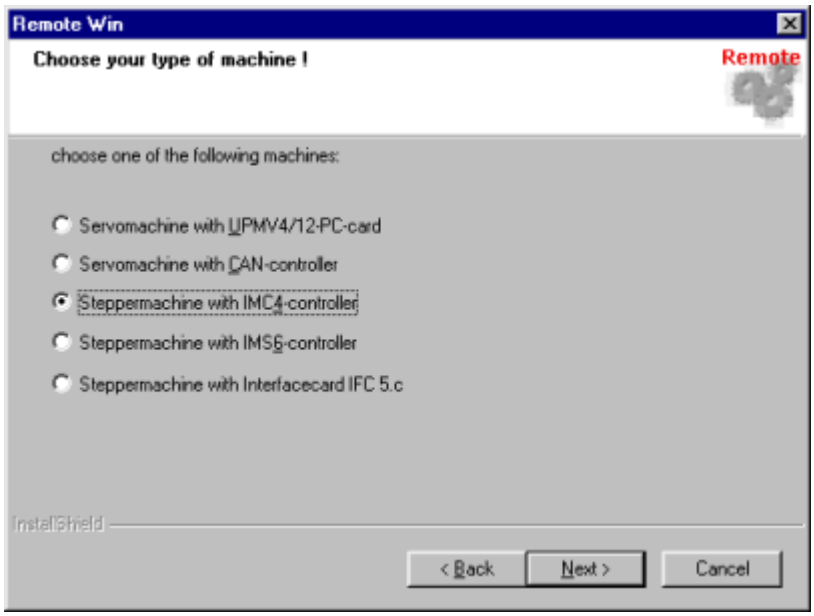
If not already factory-installed, you can install the software on your computer using the installation-CD.
The installation for Remote contains the **Remote Basic-Set up** und the **set up for your target controller**.
The following isel-controllers are supported:

- IMC4 in all CPMxxyy or GFMxxyy respectively
- IMS6 / IML4
- C142/4 with Interface card 5
- isel-Servo machines with UPMV4/12
- isel-Servo machines with isel-Power Unit for PVD on CAN-Bus

**Installation of Remote:**

Before starting the installation we recommend you, to quit all programs currently running.
Insert the installation CD into the appropriate drive of your computer. The installation will start automatically. Follow the directions on screen.

Please select within the installation the relevant type of your applied isel-machine.

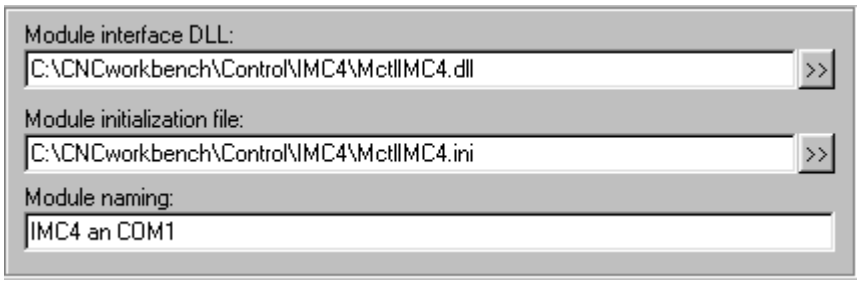



After a **successful setup** the program icon will appear on your desktop.

In a particular case (no CD-drive), it is also possible, to install the program Remote from disk. It is the same proceeding, additional you have to change the disk after request.

**Hint:**

Would you like to install Remote for an isel-machine with UPMV4/12-PC-Card, after deciding for a Servo controller you can select such an already created servo.ini-file.
The file Servo.ini can be created by two adjustment-tools PAREIN.exe and PARKON.exe. These programs you get, if you buy the isel-Controller UPMV4/12.
After creating the servo.ini-file must be assigned.
Please select the function "Setup" from the menu bar and then enter "Control". Move the cursor into the line below the text **"Modul-initialization-file",** press the button >>, select the directory containing the Servo.ini file and choose it with double click.




*please refer:* 2.8.3 Menu Control

## 1.3 Functions of Remote

**Overview:**

**Controlling und management** with the Software Remote:

- 1 **Axis System** with max. **6 axes** (X, Y, Z, A, B and C)
- 1 **Spindle**
- 2 **Free** Input- and Output Modules
- 1 **Tool Changer**
- 1 **isel-Operator Panel**, as an optional Operating Device

**Parameter machine:**

- Reset, Reference run
- **Teach-In** and manual motion of axes during running of program
- **Simple management of** machine positions (for example workpiece zero point, park position and home position)
- Rapid-, normal-, teach-, reference velocity adjustable
- Axis-Override
- Workpiece zero point set, workpiece zero point delete
- Spindle function, spindle speed override
- **Tool changer administration** for max. 128 tools
- **Machine periphery** (for example coolant, tool- and workpiece clamp, vacuum) can be activated by button or Operator panel in manual mode

**Parameter output file:**

- The last started files can be opened automatically
- Adjustment of output options
- **Program skip**, set skip, **rapid traverse**
- Repetition of outputs can be fixed
- Single step, automatic mode
- **Online path calculation** on / off
- **Applications** (CNC files), creating by **ProNC** , start immediately

# 2 Operating

## 2.1 Operating surface

**Operating surface:**

The operating surface consists of a section to display the **user program**, surrounded with symbol bars to start file, setting machine-parameters, machine-periphery, displaying of status, positions and velocities.
You can determine, how the windows are arranged on screen.

All functions are selected by menu and submenu.

To trigger off the selected functions **quickly** you can use **buttons for selected functions**. These buttons are located in the symbol bars. The symbol bars are classified in specific function groups. If some buttons are inactive, you will not apply these functions in this mode.

**Often repeated functions** like Start, Stop, Reset, Reference run can be selected by keyboard (e. g. the function "Reference run" from the menu "Control" you can start with the keys Ctrl + Z. You have to press both keys at the same time).

All windows using for decisions of the operator contain the buttons **OK** and **Cancel**.
Please select "**OK"**, if you want to save the new parameter or your decision.
If you want to leave the dialog without modifications please select "**Cancel**".

## 2.2 Menu File

Menu **File** - **Open...** (Ctrl+O)

Please select your NCP- or CNC-program, you want to start on the equipment, from the relevant directory.
With a double-click at the user program or by marking the user program and clicking at the button "Open" the program will be displayed in the working area of the operating surface Remote.

In the menu "File" there will be displayed the last four started files standing under the line "Properties".
With double click at one of these files the selected file will be opened immediately.

***Hint !***

In the menu **Setup - Options** you can fix, if the last used file will be opened automatic when Remote starts.

Menu **File** - **Close**

The **active** file is **closed**. You have to decide in a dialog, if you want to **save** your changes.

Menu **File** - **Save**

With this function the file is overwritten by all changes, you have done. The file will be saved with the same name.

Please make sure, that all modifications are not automatically saved in your file. Before starting the program after changing you have to save explicitly
in order to use the program in the correct version.

Menu **File** - **Save as...**

The "Save as" function enables the saving of the changes under a new file name.
Use the displayed window to enter the new name. Optional you can also define a new directory.

Menu **File** - **Properties**

The default property to open a file is "Write protected" (marked with the symbol ✓ in front of the text).
Please click on the text and the symbol ✓ will disappear and after opening again the file can be changed.
This parameter can also be defined in the menu **Setup - Options.**

Menu **File** - **Exit**

**Finish** the program Remote.

## 2.3 Menu control

### 2.3.1 Reset

Menu **Control - Reset** (Ctrl+R)

Select this function to configure the motion control.
The **Reset** function triggers a software reset of the driver and the controller.
A reset is particularly required if:

- the processing of a test program was interrupted
- the power supply of the controller has faulted
- the equipment was newly powered up
- an undefined error has happened

### 2.3.2 Reference run

Menu **Control - Reference run** (Ctrl+Z)

The **Reference run** function causes the driver to execute a reference movement of all connected axes.
The axes in **order Z, Y, X** move one after the other until reaching the reference switches.
If the control recognizes a limit switch, this point will become **the machine zero point for all following movements of the relevant axis.**

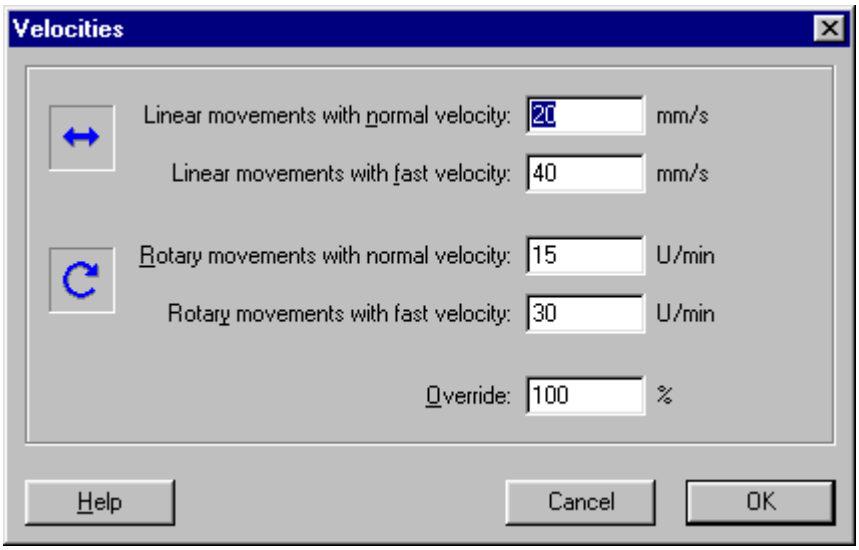
The **Reference run** enables the motion control **to assign** the **position value** (0,0,0) to the **mechanical zero point** of all axes.

**After a reference run the current activated workpiece zero point is deleted** (if the workpiece zero point was set with the function Menu **Control - Workpiece zero point activate/deactivate** or with corresponding buttons). Before starting or continuing the program please define the workpiece zero point new.

The function menu **Control - Reference run of axis...** allows, to run the axes separately.

### 2.3.3 Velocities

Menu **Control - Velocities ...**


The modification of the linear- and/or rotation velocity of the axes is possible. Please set the fast velocity (for positioning) and the normal velocity (for processing) in dependence of the processing task.
These velocities are default values; velocities, defined in your user program, have priority.

**Override:** Processing override should be set to 100%. Override is a speed factor in [%], set here or during processing with the buttons of the symbol bar "Override" .

**Hint:** The modification is possible during the run time of the CNC program. Therewith you can supplementary correct the processing velocity, defined too high or too low, during the work processing.

With **exception of stepper motor equipment** always the current velocity is modified.
At machines **with stepper motor** the **new velocity** is **effective** just in the **next NC-segment**.

**Change the velocity step by step:**


reduce Override at 1%

reduce Override at 10%

increase Override at 1%

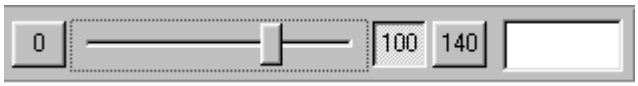
increase Override at 10%

Override = 0%

Override = 100%

Override = 140%

**Changing velocity with slide control:**

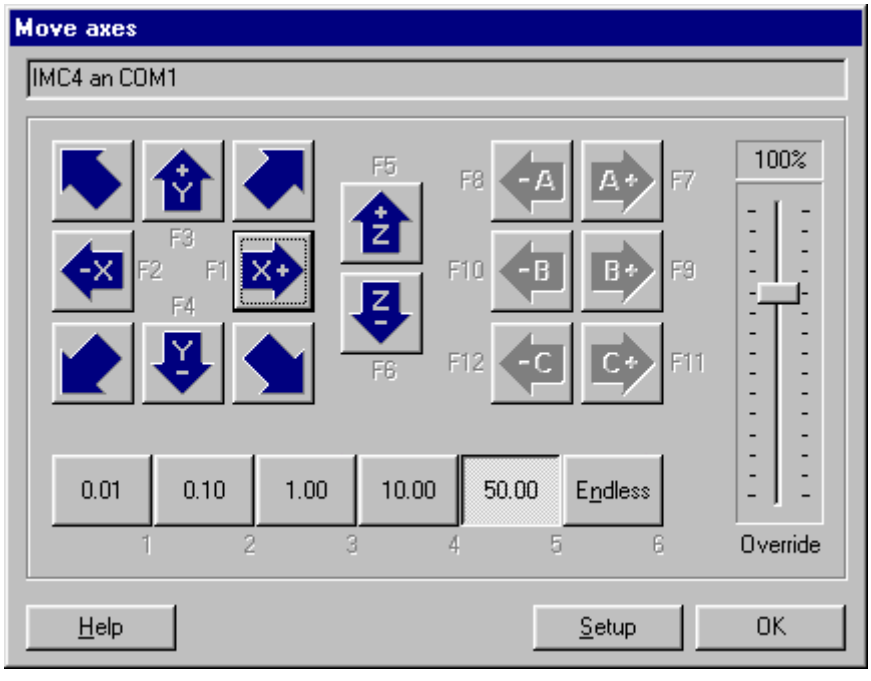



The installation of the desired display respectively of the buttons is done with the menu **View - Symbol bar "Override"**.

*please refer:* 2.6 Menu View

### 2.3.4 Manual movement

Menu **Control - Manual movement ...** (Ctrl+M)

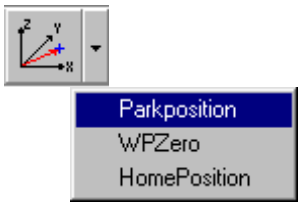


This function enables the manual movement of all activated numerical axes.
Please select step widths of the movement with the buttons (0,01, 0,10, 1,00 ...) or you can define it after choosing the button "Setup".
To move the Z-axis you should prefer small distances to touch the surface with the tool for defining the zero point.
For the movement of the axes please select one of these three possibilities:

1. **Mouse click on the dart** with pretended direction (e. g. -X, +Z); for a fast movement you can also use the diagonal movement.

2. **Move with help of the function keys** F1 ... F6 (for the axes X, Y, Z) respectively F8 ... F12 (for the axes A, B, C).

3. **Move** the axes with **keys** of the **Operating panel** (option).




Clicking on the dart of this buttons the axes will approach to the defined positions in the window "Machine positions":

- Parkposition
- WPZero (workpiece zero point)
- Homeposition

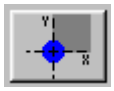
*please refer:* 2.3.5 Menu Control - Set up machine positions

### 2.3.5 Set up machine positions

Menu **Control - Set up machine positions**

To define the workpiece zero point and another particular machine positions please select this menu .
Following positions are default:

- ParkPosition
- Wpzero (workpiece zero point)
- HomePosition

The coordinate values can be detected by manual movement 
or can be inserted

by selecting the button "Edit position"  .

To update a machine position please click on the desired line. A beam will point you out to your selection.
All the following inputs refer to this machine position.

**Edit position list:** To correct, to construct or to delete machine positions you can use the buttons **New, Copy, Insert, Delete, Name.**

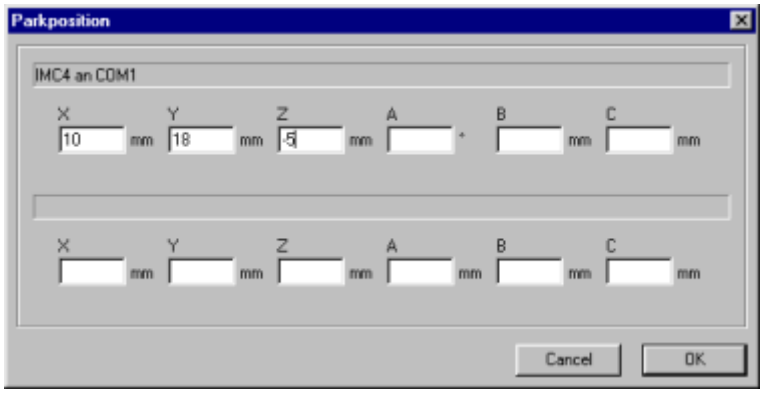
If you want to define a new machine position, please follow the next steps:
Mark with a mouse click e. g. the line "WPZero".

Select from the "Edit position list" the button 

Insert.
Above the marked line a empty line appears.
Please fix a name for the position and enter the coordinates or teach it, how it is described in the following both sections.

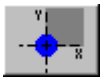
**Edit machine positions:**

With the button  or with double click at the marked machine position a window is opened. In this window it is possible to input the numerical values for the activated axes.
The second line is reserved for the extension of the equipment with two axis systems.




**Edit positions:**

To define a workpiece zero point by moving the axes manually, please select

- the button  **Teach all**
- the window **Manual movement** will be opened
- move the axes with mouse click, keys of keyboard or Operator panel to the position you will use as workpiece zero point

After exiting with OK the actual position values are entered in the highlighted line.

**Move to position:**

A movement to the marked machine position in the schedule will be carried out. Before starting this function you should examine whether the movement will be possible without danger.

**Options:**

- **Approach order**

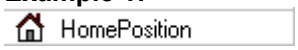
  Setting movement order

- **Special usage**

  Attributing a special usage to a machine position

**Examples:**

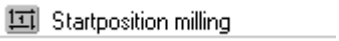
**Example 1:**




Defining this position as **Home position** please select the icon within the option "Special usage".

Activating e. g. the option „Move to Home position" the axes will approach to this position.

**Example 2:**




Activating with the menu Setup - CNC/NCP file processing the option "Before processing starts, move to position "Start of processing"„ the defined position of the axes will be approached before executing the first instruction in the user program.

### 2.3.6 Activate/deactivate workpiece zero point

Menu **Control - Activate workpiece zero point**

With this function the **current** or a **teached position** of the axes will be defined as the new workpiece zero point. The zero point will be deleted by click the button again, that means, the button is deactivated.
Starting the program this position is the basic position for all following movements of the axes.
This zero point is active until it will be defined and set new by the operator or within the program.

You can use this function to set a new workpiece zero point after a Reset or a Reference run or to correct a defined workpiece zero point.
After defining a certain position as a new zero point, the position-display will show the value zero for all axes.
**For a visual support the coordinate values are displayed in blue, if a workpiece zero point is set.**

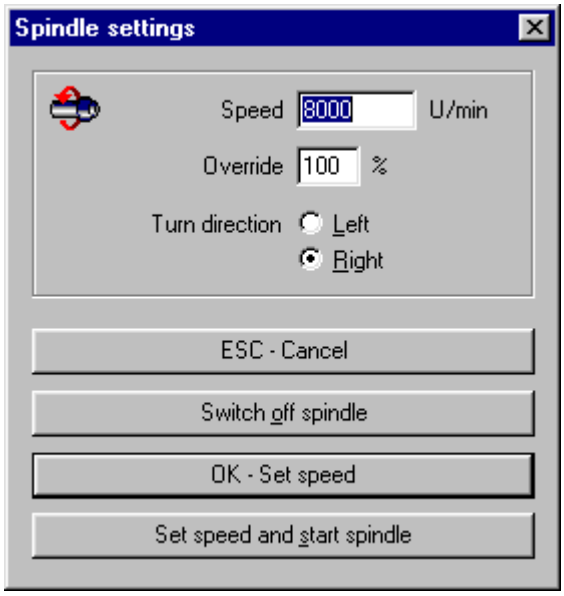
**Hint:**

A workpiece zero point, set by this function will not be taken on in the program enduringly.
If you want to change a workpiece zero point enduringly, teach the workpiece zero point within the menu **Control - Set up machine positions** and declare it as current workpiece zero point.

*please refer:* 2.3.4 Menu Control - Manual movement
2.3.5 Menu Control - Set up machine positions

### 2.3.7 Spindle

Menu **Control - Spindle speed** (CTRL+N)




A continuous setting of the spindle speed with nominal value (absolute instruction) or with **override** (percent instruction) is possible, if the processing spindle is controlled with a frequency transverter.

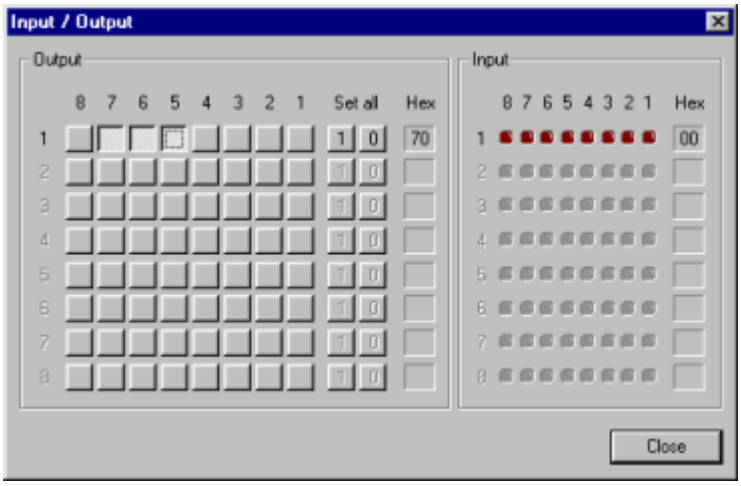
If the equipment does not have a spindle with a control by transverter the spindle speed should be fixed at the spindle by hand.

Menu **Control - Switch on spindle Switch off spindle**


With this function please switch on or switch off the spindle.

### 2.3.8 Input/Output

Menu **Control Input/Output** (Ctrl+I)




Setting or resetting the outputs **Bit 1** to **8** on the output port 1.
The function keys F1 to F8 can be used, the TAB key selects the actual port if more than one output ports are available.

*please refer:* 2.3.10 Accessories

### 2.3.9 Change tool

Menu **Control - Change tool** (Ctrl+T)

Tool change means get a new tool (from a tool changer)

Menu **Control - Put away tool**

Tool put away (categorizing in a tool magazine)

Menu **Control - Tool clamp open/close**

Tool clamp open / close

Menu **Control - Tool hood open/close**

Tool hood of a tool changer open / close

Menu **Control - Rotate tool magazine**

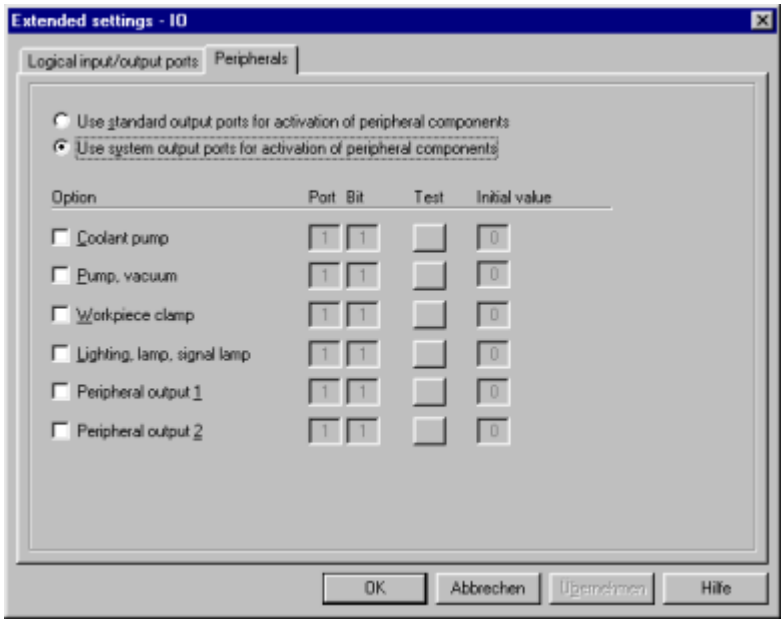
Tool magazine rotate

### 2.3.10 Accessories

Menu **Control - Accessories**

In the menu **Accessories** you find a selection of peripheral devices, you can use within your proceeding.
Before activating or deactivating these peripheral devices, the used devices and the port/bit must be configured in a dialog.
Please select the menu:

- Menu **Setup - Control**
- Line Extended **I/O-settings**
- Button **SETUP**
- Index file card **Peripherals**



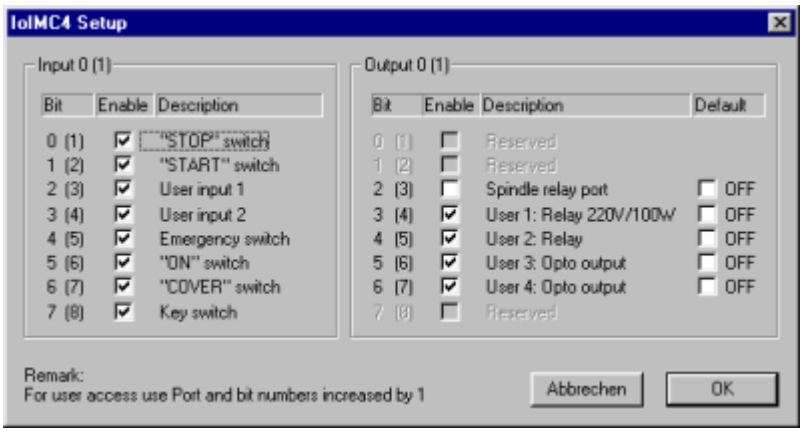

***Hint !***

Subject to the control environment not all output ports are useable.
E. g. using the Controller IMC4 you can apply the output port from bit 4.

Information about this matter you will get in the window **Input-/Output module Setup.**

To open this window, please select:

- Menu **Setup - Control**
- Line **IO module**
- Button **SETUP**




The bit 3 of the output port 1 is reserved for the spindle, the bits 4 to 7 are arbitrary.

**Utilization of the periphery:**

After defining the outputs according to your specific periphery the setting and resetting can be done with the menu **Control - Accessories** or direct with the corresponding buttons.
A further possibility to use this function is to insert the commands into the NCP program.

*please refer:* 3.10 Basic commands - Coolant
3.11 Basic commands - Workpiece clamp
3.12 Basic commands - Pump
3.13 Basic commands - Lamp
3.14 Basic commands - Periphery options

Menu **Control - Accessories - Coolant**

Coolant on/off

Menu **Control - Accessories - Workpiece clamp**

Open/close workpiece clamp

Menu **Control - Accessories - Vacuum/Pump**

Vacuum/Pump on / off

Menu **Control - Accessories - Lighting**

Lighting on / off

Menu **Control - Accessories - Accessory 1**

Accessory 1 on/off


Menu **Control - Accessories - Accessory 2**

Accessory 2 on/off


### 2.3.11 Options

Menu **Control - Options - Move axes out of limit switch**

After an **error situation** with the effect of reaching one or more limit switches and a following **stop of the equipment**, this function causes **a run of the axes out of limit switch**.

## 2.4 Menu Edit

**Edit a file:**

The menu **Edit** contains all functions to change a file and corresponds to the essential standard functions of Windows-Application for text editing e. g. WordPad.
If you want to change your file, please open the file in **no protected mode**.
please refer: 2.8.1 Menu Setup - Options

Menu **Edit** - **Undo**

The **last action** is not done.

Menu **Edit** - **Cut**

A **marked word** or a **marked area** is deleted (cut) and will be **saved** in the **clipboard**.

Menu **Edit - Copy**

A **marked word** or a **marked area** is saved in the **clipboard**.
The **marked text** survives in the file.

Menu **Edit - Paste**

The **saved text/graphic** will be inserted from the clipboard to the place in the file where the cursor is located.

Menu **Edit - Insert contents ...**

Insert the clipboard into the program, to work with the inserted contents e. g. **with Microsoft Word**.

Menu **Edit - Select all**

The **complete file** is highlighted.

Menu **Edit - Find...**

Please **prompt the text** you look for. The next appearance of the text will be displayed.

**Menu Edit - Find again...**

After prompting the input and finishing the command "**Find**" you can search the same word with the menu "**Find again**" without a new prompt of the text.

**Menu Edit - Replace**

Please prompt the text you want to look for.
Write the text, that shall replace the text found.
You can search **step by step** and decide after searching the text, if the replacement shall happen. (buttons „**Find again" + „Replace**")
Selecting the button „**Replace all**" it will be replaced at each agreement.

## 2.5 Menu Processing

### 2.5.1 Program skip, Set skip, Rapid traverse, Online path calculation

Menu **Processing - Program skip**


With the function **Program skip** user programs can continue at the break point, breaking off while the automatic mode.

To activate the Program skip, click the button  before starting the program again.
**A condition** for a perfect continuation is, the position of the workpiece did not change and the program abort occurred with the button  or with the menu **Processing - Abort**.

Menu **Processing - Set skip**


If you want to use the function **Set skip**, please mark the sets in your NCP file with **the character "?"** as first character of the set. All sets with this marking will be skipped during processing.

Menu **Processing - Rapid traverse**


The function **Rapid traverse** causes, that the linear movement will not run with programmed velocity, but always with rapid velocity.
This function can be activated and deactivated during the runtime of the user program.
To test a large program you can use this function, but please **pay attention to the following hint:**

**Attention:**

Please note, that a **meaningful processing is not possible** with an active **rapid traverse**. **Clamp** the tool **out** of the spindle, **correct** the workpiece zero point in **+Z** or **do not clamp a workpiece**.
At Laser- or Water jet equipments you should provide security for a deactivated Laser aggregate / High pressure pump.

Menu **Processing** - **Online path calculation**


The controller IMS6/UPMV4 and CAN have the ability of a **buffered continuous path calculation** (Online).

This ability can be activated with this button.

The button is deactivated, if the current motion control can not carry out a **buffered continuous path calculation** (Online).

### 2.5.2 Output repetition

**Fix the output repetitions:**


**No** output repetitions


**User defined number of outputs** (after selecting this button you can enter the number of replay)


User program (NCP file or CNC file) is processed in an **endless loop.**

**Hint:**

The defining of one of these three parameters is also possible in the menu **Setup - CNC/NCP file processing**, index card "Output repetitions".

As **default parameter** the endless loop is defined.

### 2.5.3 Operating mode, Start, Stop, Abort

Menu **Processing** - **Single step**

Select this function to process the program **step by step**. The running command is marked by a beam.
The continuation of the program carries out with click to the button start 

.
This **operating mode is used primary** in the **test phase**, generally to find bugs.

To change into automatic mode, please click the button  and then once on the button start 

.

Menu **Processing** - **Automatic mode**

The processing of the program starts in **automatic mode** in order of the program lines.

To change to the modus **step by step** please click once the button  .

Menu **Processing** - **Stop** (Ctrl+P)

To carry out e. g. a quality check of the processing, you can interrupt the current movement defined.

To continue please click the button start  .

Menu **Processing - Start** (Ctrl+RETURN)

The user program will start at the first command line of the main program. According to the parameters in menu **Setup - CNC/NCP file processing after** starting the processing file and **before** interpreting the first line in the CNC/NCP program e. g. a reference run will be executed.

*please refer:* 2.8.2 Menu Setup - CNC/NCP file processing

Menu **Processing - Abort** (Shift+ESC)

The user program is aborted. The spindle turns off. Finishing the program the valid parameters of this program are saved in the initialization file. Using the function **Program skip** and the data out of the initialization file the program will be continued at the same point, where the abort had happened.

## 2.6 Menu View

Menu **View:**

Select this menu, if you will fade in/out the symbol bars.

The visible symbol bars are marked with a token.
**To remove** one of the symbol bars or display window please click on it.

**→ Symbol bar "File"**
**→ Symbol bar "Machine"**
**→ Symbol bar "Override"**
**→ Symbol bar "Accessories"**
**→ Symbol bar "Tool change"**
**→ Symbol bar "Processing"**

**→ Status bar**

**→ Positions**
**→ Spindle speed**
**→ Machine status**

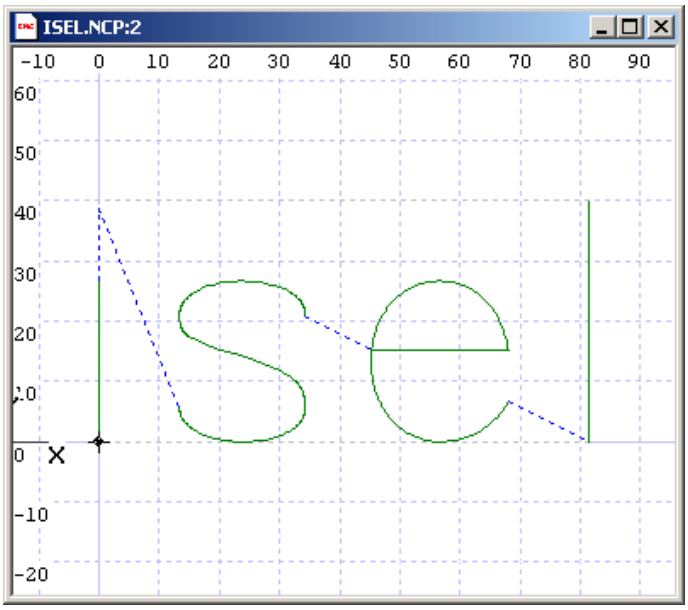
**→ Process variable monitor**

## 2.7 Menu Window

Menu **Window**

In this menu you can determine, if the opened user program shall be displayed graphically. Please choose for this purpose "Display graphic" or press the button  .
The options Cascade, Tile horizontally, Tile vertically are available to the arrangement of both windows (user program and graphic display).

**Display graphic:**




Pressing the right mouse key a menu is opened with the following functions:

- 3D view or in the planes XY, XZ, YZ
- Zoom
- Measuring
- Display grid, grid coordinates, dimensions, zero point, fast movement
- Colors
- File info
- Refresh

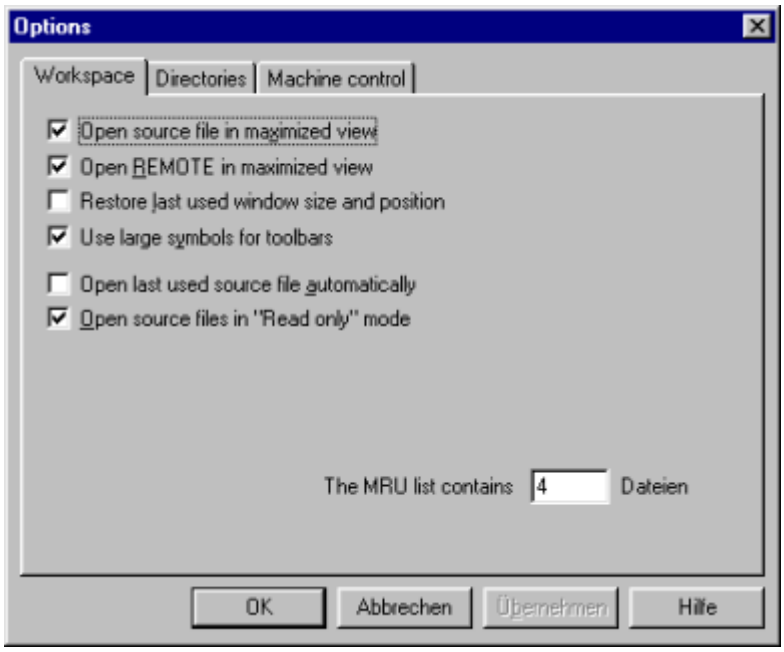
## 2.8 Menu Setup

### 2.8.1 Options

Menu **Setup - Options**

For a convenient operating please define your specific parameters for the file processing.

**Workspace:**




Menu **Setup - Options**
**Directories:**

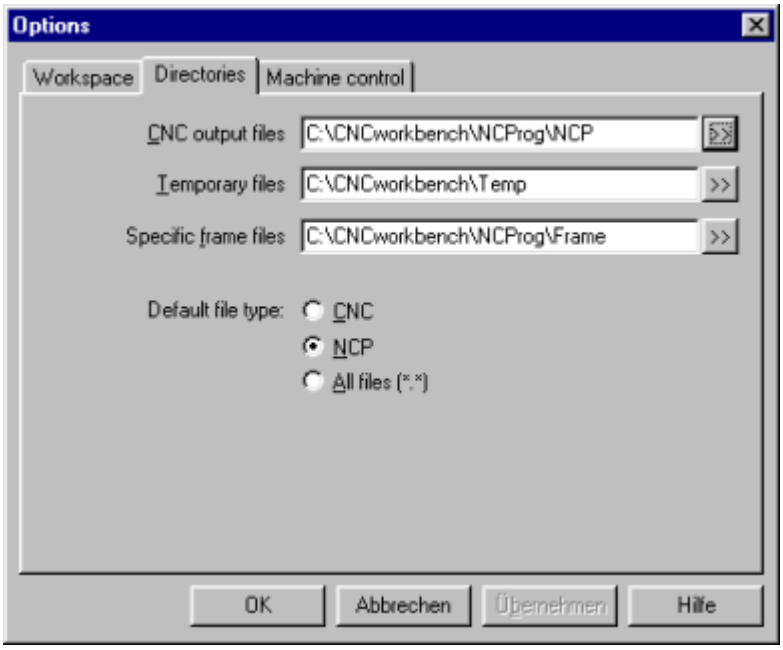
**To find** your user files **quickly** in this window you can designate the directory for:

**- User program** with file type **NCP** or **CNC**

Simultaneous you can define, what kind of **file type** you use favourite. Opening a file the selected **directory** and the **file type** are always default.

- **Frame files** ( specific geometry file, containing machine-positions, useable within a running program)

**Frame file** will be used in bearing on a processing of user programs generated and compiled with the NC-Software **ProNC**

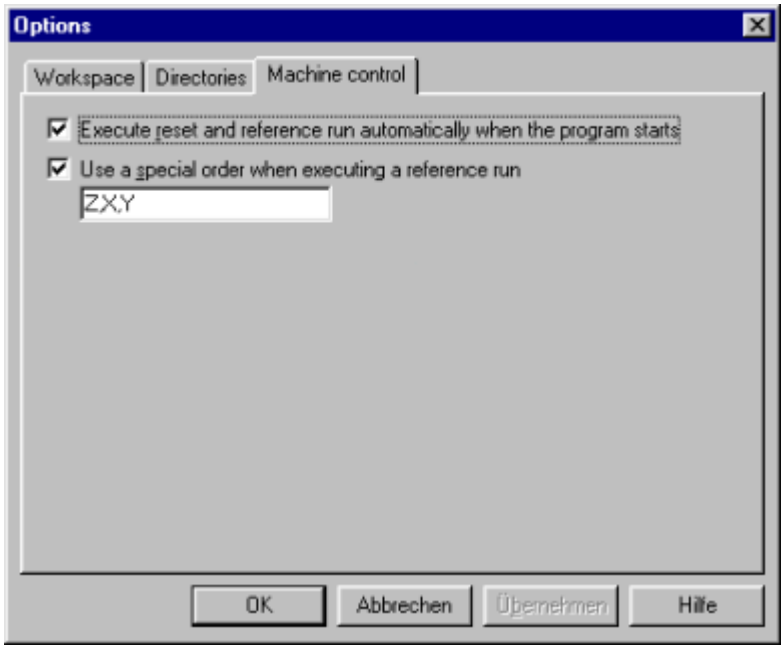



Menu **Setup - Options**

The selection of the option **Reset** and **Reference run** result a Reset and a Reference run on every start of Remote.
The second option definite the order of axes, when executing a Reference run.

**Machine control:**

Activating both options after the start of Remote a Reset und a Reference run in order of the axes Z,X,Y will be executed.

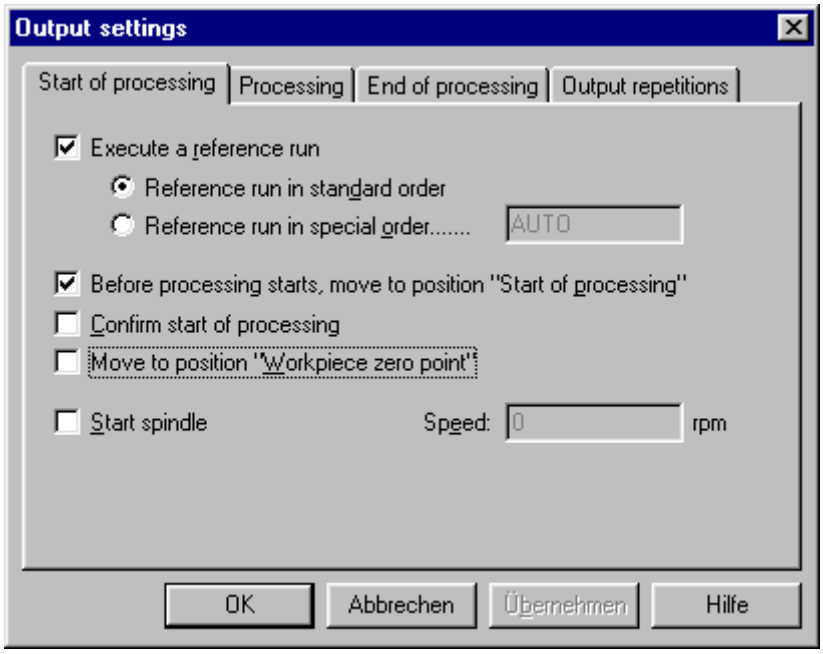


### 2.8.2 CNC/NCP file processing

Menu **Setup - CNC/NCP file processing**

With this menu can be designated, what kind of additional functions should be executed before beginning and after the end of processing.

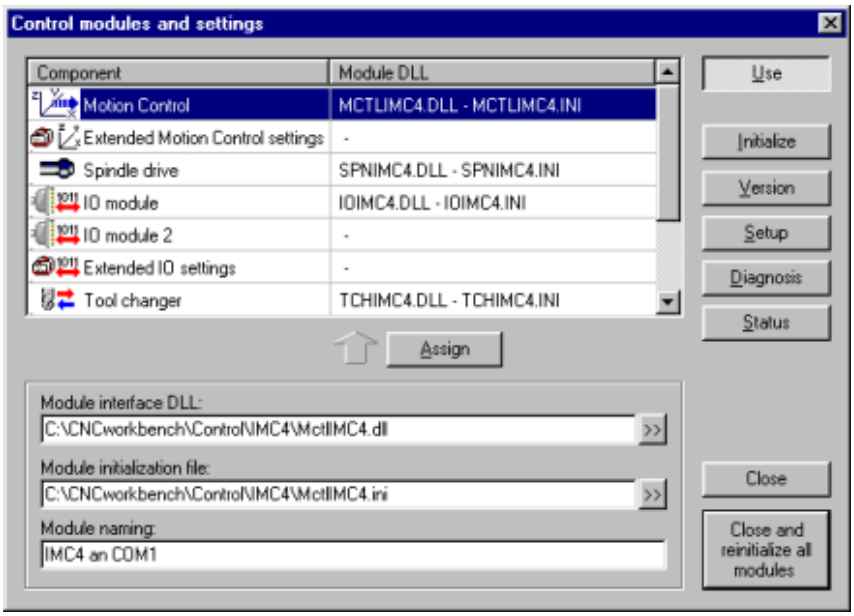
- Executing a Reference run with standard order (Z,X,Y) or user defined order
- Move to selective positions; the definition of these positions occurs in the menu **Control - Set up machine positions** with the button *special usage*  .
- Activating a dialog before processing the NCP file or CNC file; you decide in this dialog, if the process will be started or aborted.
- Move to the current workpiece zero point
- Switch on /off the spindle, definition spindle speed



### 2.8.3 Control

#### *2.8.3.1 Configuration*

Menu
**Setup - Control**




This window offers the dialog aided possibility alike the system controlling in Windows, to configure the special **Module-DLL** including the required initialization files.

In the rule the modules and settings will be factory-installed according to the customers target controller.

Changing or extending the equipment the user is able to configure the Hardware/Modules with the corresponding Motion Control, Spindle control, IO control and Tool changer (DLLs) himself.

**Remote is configured after a successful installation according to the selected target control.**

**Module parameter:**

A selected Motion control is defined by the following information:

- ***Module naming:***
  please enter a name of the equipment/module

- ***Module interface DLL:***
  please select the DLL-file according to the control in your equipment with a click at the button >>.

***Hint:***

**You find the relevant files e. g. in the directories:**
\CNCworkbench\Control\Imc4 for modules of IMC4
\CNCworkbench\Control\StdSV1 for modules of UPMV4

- ***Module initialization file:***
  select the relevant initialization file for the configured module with a click on the button >>.

**For more support** of the configuration you find the following functionalities for any module:

- Versions request with common information about the special module DLL
- Set up of relevant parameters e. g. the pitch of the current axes
- Diagnostics for checking the functions of the Module-DLL and the equipment
- Status to determination and checking the current state of the module

**Hint:**

For a temporary deactivating of a control module please utilize the button Use .

Please mark the line with the component you will activate or deactivate. With the button **"Use"** determine the state (active/inactive).

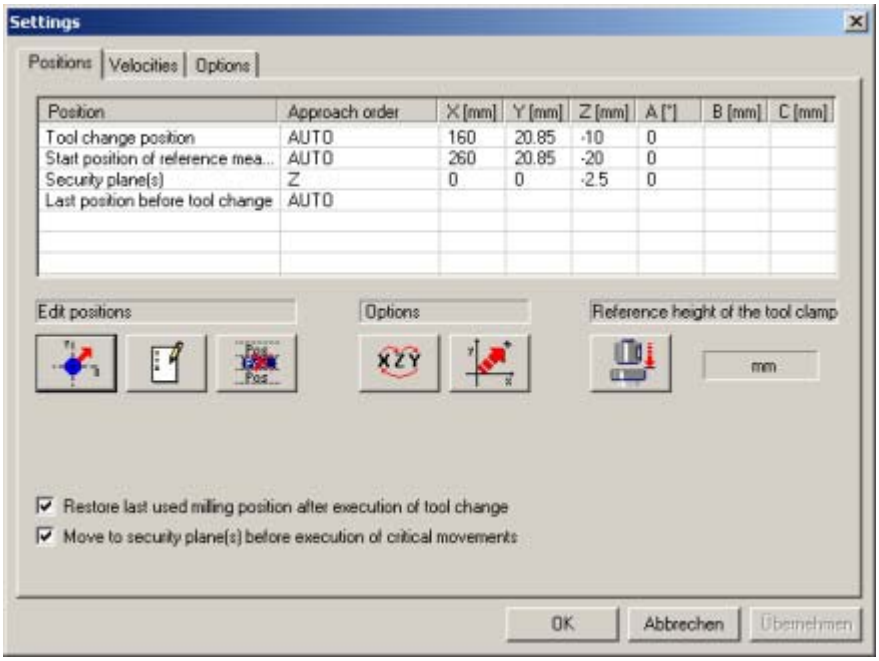
**Hint:**

After changing the parameters, finish always with the button Close and reinitialize all modules .

Therewith all resulted modifications become actively. A **exit/restart** of Remote is **not required**.

*2.8.3.2 Installation of tool changer*

Menu **Setup-Control**



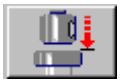
**Installation of the positions for tool change:**

To adapt the following positions for tool changer according to your equipment you can use the functions teach or edit:

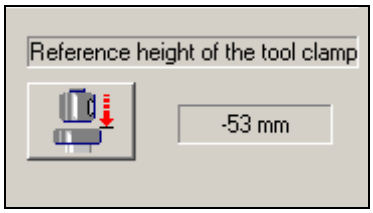
- **Tool change position** (an advantageous position of the axes to change the tool at manual tool change)
- **Start position** for referencing after tool change (position about the length measuring calliper)
- **Last position before changing** (optional)

*please refer:* Menu 4.5 Settings tool change

After defining the start position for referencing the tool clamp, that means for the tool length determination

Please push the button  .

After moving to this start position please clamp a teaching to carry out the referencing.
The measurement of the reference height will be executed automatically; the detected value as basic value for each tool change is displayed.




## 2.9 Operator Panel

**Operator panel -** optional device

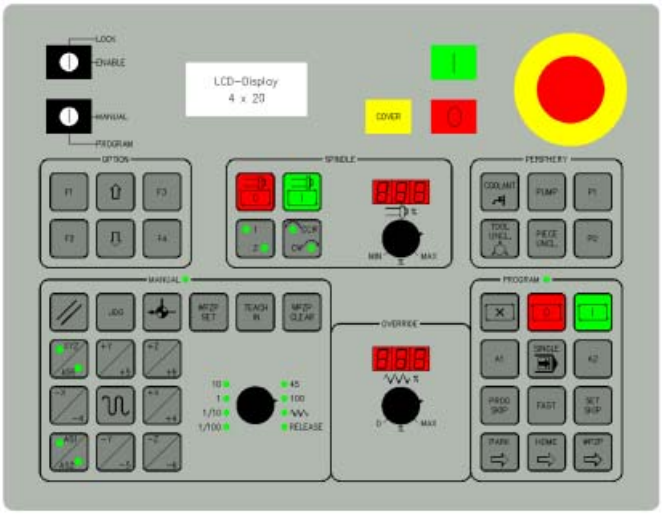
The Operator panel represents an user friendly interface to the CNC control with the following features:

- Communication with the CNC control based on the CAN field bus
- Data rates up to 1 Mbits per second
- Integration into the control software structure under Windows NT (DLL), using Windows Message concept
- Providing security functions like EMERGENCY, Power ON, Enable/Lock, Switching MANUAL/PROGRAM
- Layout with several operating sections

isel-CNC machines and - equipments can run in the **operating mode**:

- **Manual** (manually controlled)
  *or*
- **Automatic** (program controlled).

The modification of the operating mode is possible on the Operator panel using the key switch.
According to the selected operating mode the keys of the operating section MANUAL respectively PROGRAM are active respectively inactive.



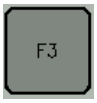

### 2.9.1 File selection

Operating section **OPTION:**

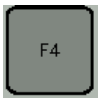
The keys of this operating section serve the following functions; a adaptation for user defined functions is possible at any time.

**Keys:**


Open a window to select the user output file.

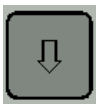



Selected output file is loaded. F3 carries out the function of the button "**OK**".




**The selection of the file will be aborted.** F4 carries out the function of the button "**Abort**".

Select the file, that shall be opened.



### 2.9.2 Spindle

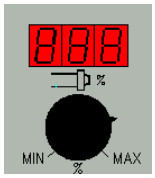
Operating section
**SPINDLE:**


**Switch off spindle**


**Switch on spindle**


**Selection spindle 1** or **spindle 2**


Direction of revolution
- **clockwise** (CW)
- **counterclockwise** (CCW)




**Override spindle speed** from min to max in percent

*please refer:* 2.3.7 Menu Control - Spindle

### 2.9.3 Accessories

Operating section
**PERIPHERIE:**


**Coolant on/off**


**Pump/Vacuum on/off**


**Tool clamp open/close**


**Workpiece clamp open/close**


**Accessory 1 on/off**


**Accessory 2 on/off**

*please refer:* 2.3.10 Menu Control - Accessories

### 2.9.4 Manual

Operating section **MANUAL:**

**Reset**

*please refer:* 2.3.1 Menu Control - Reset


**Move axis step by step**
Activating this button, a manual running of each current axis is carried out when pushing on the coordinate keys X, Y, Z or 4, 5, 6. The step width of the axis movement will be defined with the turn switch within this operating section.

*please refer:* 2.3.4 Menu Control - Manual movement

**Reference run**
Activate this button and then push one of the coordinate keys X, Y, Z respectively 4, 5, 6 a reference run of the current axis is carried out.

Activate this button and then the button  carries out the Reference run of all axes.

*please refer:* 2.3.2 Menu Control - Reference run

Operating section **MANUAL:**


**Set workpiece zero point**

*please refer:*
2.3.6 Menu Control - Activate/deactivate workpiece zero point


**Set up machine positions**

*please refer:* 2.3.5 Menu Control - Set up machine positions


**Clear workpiece zero point**

*please refer:* 2.3.6 Menu Control - Activate/deactivate workpiece zero point


**Select the axes** X.Y, Z **or** the axes 4, 5, 6
(the axes 4, 5, 6 may be rotary- or linear-axis.)

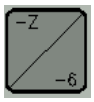

**Axis X** respectively axis **A** approach in positive direction from coordinate zero point


**Axis X** respectively axis **A** approach in negative direction from coordinate zero point


**Axis Y** respectively axis **B** approach in positive direction from coordinate zero point


**Axis Y** respectively axis **B** approach in negative direction from coordinate zero point


**Axis Z** respectively axis **C** approach in positive direction from coordinate zero point

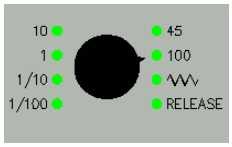



**Axis Z** respectively axis **C** approach in negative direction from coordinate zero point


**Select axis system 1** *or* **axis system 2**

**Rapid traverse**
- pressing this key simultaneous with a coordinate key (e. g. axis X):
  **Axis X is running in rapid traverse**

- pressing this key, when the reference key was activated
  before, results in a Reference run of all axes




**Turn switch**
- define the step width/continuous for the manual movement of the axes
- move out of the position "Limit switch" (RELEASE)

### 2.9.5 Axis-Override

Operating section
**OVERRIDE:**


**Override the actual feed rate from 0 to max percent**

*please refer:* 2.3.3 Menu Control - Velocities

### 2.9.6 Automatic mode (program controlled)

Operating section
**PROGRAM:**

**Abort processing**

*please refer:* 2.5.3 Menu Processing- Abort

**Stop processing**


*please refer:* 2.5.3 Menu Processing - Stop


**Start output file in automatic mode**

*please refer:* 2.5.3 Menu Processing - Start


**Program processing in single step mode**

*please refer:* 2.5.3 Menu Processing - Operating mode


User programs **continue at the last aborted point**, when restarted.

*please refer:* 2.5.1 Menu Processing - Program skip


**Rapid traverse** ( linear movements will not run with programmed velocity, but with rapid velocity)

*please refer:* 2.5.1 Menu Processing - Rapid traverse


Sets with the marking "?" will be **skipped** during processing.

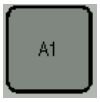
*please refer:* 2.5.1 Menu Processing - Set skip

Operating section **PROGRAM:**

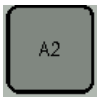

**Approach Park position**


**Approach Home position**


**Approach Tool change position**




**Reserved for user function**




**Reserved for user function**

# 3 Basic commands

## 3 Basic commands in the NCP program

**Definition NCP program:**

In this chapter the most important commands of a NCP file are explained. NCP files was created out of a vector file in a postprocessor run e. g. in isy 3.x.

To guarantee a faultless processing of NCP files by Remote, it must be secured, that the first line of the **NCP program** contains the significant string:

**IMF_PBL**

Before and behind of these letters may be any text. Please note, that a semicolon is not allowed, because it will be interpreted as comment.

The program isy2.x / 3.x and Rapid Mill create this signs automatically. If you edit the NCP program, pay attention, that **these seven signs will not be modified.**

**Units in the NCP program:**

| | | |
|---|---|---|
| Axis position values linear axes | micrometer | [µm] |
| Axis position values rotary axes | arc sec | ["] |
| Axis linear velocities | micrometer/second | [µm/s] |
| Axis pitch rate | arc sec/second | ["/s] |
| Spindle speed | revolutions per minute | [rpm] |
| | revolutions per second | [rps] |

**Example:**

A command e. g. the "Linear movement command" with the coordinates X= 10 mm and Y= 5 mm has as set in the program the following syntax:

N10 MOVEABS X**10000** Y**5000**

**Command description:**

In the following chapter a uniform structure is used for the commands in the program:

| Command | Short declaration |
|---|---|

**Syntax:**

The syntax defines, how the construction (WORD / INSTRUCTION) has to appear in the application program text. It is noted, which parameters, e.g. coordinates, variables or markers are allowed within the construction.

*Hints to the notation in the syntax:*

| **Notation** | **Meaning** |
|---|---|
| **[**construction**]?** | the defined construction in square bracket is optionally, i.e. it can be maximum programmed for once or not |

**Statement:** The purpose, the task, the speciality and / or the use of the construction are explained.

**Example:** The purpose, the task, the speciality and / or the use of the construction are explained with examples.

## 3.1 Movement with rapid velocity

| **FASTABS/ FASTREL** | Movement with **Rapid** velocity |
|---|---|

**Syntax:**

[set number]?

**FASTABS** *or* **FASTREL**
[target position values]{1,6}

**Statement:**

**XYZ:**
**Linear movement with rapid velocity:**

- at least one target position value has to be defined
- maximal six target position values can be defined in a NC-set
- **FASTABS** (absolute dimension): the target position value refers to the current zero point of the workpiece coordinate system
- **FASTREL** (incremental dimension): the target position value refers to the current start point

- the rapid velocity is defined in the initialization file of the motion control
- the rapid velocity can be also defined with the command FASTVEL in front of the movement command

*please refer:* 3.6 Rapid velocity

**Example:**

; absolute movement to the target point with the coordinates
; (100 mm, 200 mm, 300 mm) with rapid velocity:

N200 **FASTABS** X100000 Y200000 Z300000

; relative movement of the X-Axis about 10 mm, of the Y-Axis about
; 20 mm and of the Z-axis about 30 mm, from the actual start point,
; with rapid velocity:

N200 **FASTREL** X10000 Y20000 Z30000

## 3.2 Linear normal movement

| **MOVEABS/ MOVEREL** | **Linear normal movement** |
|---|---|

**Syntax:** [set number]?

**MOVEABS** *or* **MOVEREL**
[target position values]{1,6}

**Statement:** **XYZ: Linear movement carries out a linear interpolation with processing velocity**

- at least one target position value has to be defined
- maximal six target position values can be defined in a NC-set
- **MOVEABS** (absolute dimension): the target position values refers to the current zero point of the workpiece coordinate system
- **MOVEREL** (incremental dimension): the target position values refers to the current start point

- the processing velocity can be also defined with the command VEL in front of the current set with the movement command

*please refer:* 3.5 Processing velocity

**Example**:

```
; straight line in space to the absolute target point with the
; coordinates (100 mm, 200 mm, 300 mm) with
; processing velocity:

    N100 MOVEABS X100000 Y200000 Z300000

; straight line in space to the target point with the coordinates
; X-basic + 10 mm, Y-basic + 20 mm, Z-basic – 30 mm
; with processing velocity:

    N200 MOVEREL X10000 Y20000 Z-30000
```

## 3.3 Circular interpolation clockwise

| CWABS<br>CWREL | **Circular interpolation** clockwise |
|---|---|

**Syntax:** [set number]?

**CWABS** *or* **CWREL**
[target position values]{1,6}
[central point position values]{1,3}

**Statement:** **XYZ:**
**Circle / arc of a circle in the active interpolation plane clockwise with definition of the center coordinates**

- this instruction can only be used for Cartesian plants
- at least one target position value and the corresponding center coordinate have to be defined
  **X-> I, Y-> J, Z-> K**
- the definition of target position value can be absolute or relative
- center coordinate statements are always specified: **both absolute or both relative**
- the direction of rotation is defined so, that the third coordinate runs always from positive to negative, if you look on the interpolation plane:

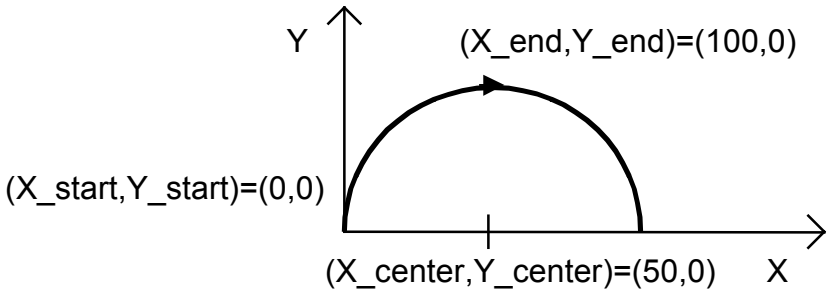
Hint:
The **X-Y-plane** as interpolation plane is selected; now please look into negative **Z-direction** on a "phantom-clock" in this plane, that direction of rotation agrees with the direction of rotation of the circle.

**Example:**

; Semicircle clockwise in the X-Y-plane:
; startpoint:
(X_start,Y_start)=(**0**,**0**)
; endpoint:
(X_end,Y_end)=(**100**,0)

; processing velocity: **50** mm/sec:

```
N10 PLANE XY                ; define interpolation plane
N20 FASTABS X0 Y0           ; approach start point
N30 VEL 50000               ; processing velocity 50 mm/sec
N40 CWABS X100000 I50000 ; run circle
```

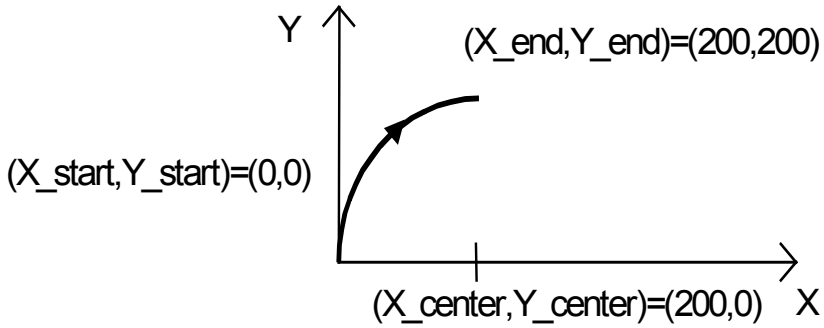



**Example**:

; **Circle arc** clockwise in the X-Y-plane:
; startpoint:
(X_start,Y_start)=(**0**,**0**)
; endpoint:
(X_end,Y_end)=(**200**,**200**)

; processing velocity **75** mm/sec:

```
N10 PLANE XY
N20 FASTABS X0 Y0
N25 VEL 75000
N30 CWABS X200000 Y200000 I200000 J0
```



## 3.4 Circle interpolation counterclockwise

| | |
|---|---|
| **CCWABS**<br>**CCWREL** | **Circle interpolation** counterclockwise |

**Syntax:**

[set number]?

**CCWABS** *or* **CCWREL**
[target position values]{1,6}
[central point position values]{1,3}

**Statement:**

**XYZ:**
**Circle / arc of a circle in the active interpolation plane counterclockwise with definition of the center coordinates**

- this instruction can only be used for Cartesian plants
- at least one target position value and the corresponding center coordinate have to be defined
  **X-> I, Y-> J, Z-> K**
- the definition of target position value can be absolute or relative
- center coordinate statements are always specified: both absolute or both relative
- the direction of rotation is defined so, that the third coordinate runs always from positive to negative, if you look on the interpolation plane:

Hint:
The **X-Y-plane** as interpolation plane is selected; now please look into negative **Z-direction** on a "phantom-clock" in this plane, that rotary course agrees with the direction of rotation of the circle.

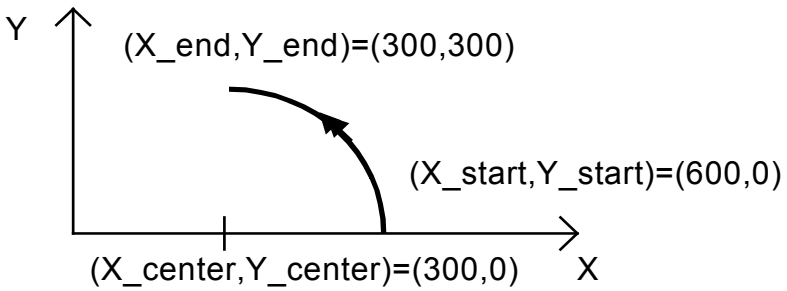
**Example:**

; **Quarter circle** counterclockwise in the XY-plane:
; startpoint:
(X_start,Y_start)=(**600**,**0**)
; endpoint:
(X_end,Y_end)=(**300**,**300**)

; processing velocity **66** mm/sec:

```
N10 PLANE XY
N20 FASTABS X600000 Y0                ; move to start point
N25 VEL 66000                         ; processing velocity
N30 CCWABS X300000 Y300000 I300000    ; approach circle
```

Because the center coordinate Y_center = 0 does not change opposite the start value Y_start = 0 , the definition of the J-position value in the NC-set can escape.

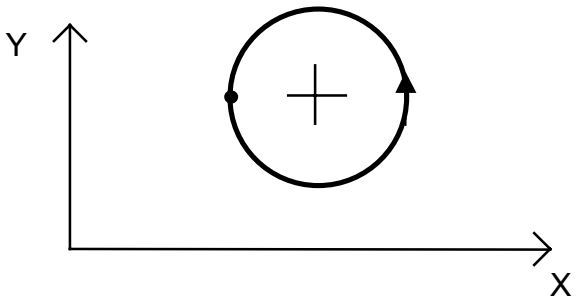



**Example:**

; Circle counterclockwise in the X-Y-plane:
; startpoint: (X_start,Y_start)=(**120**,**180**)
; radius. 50 mm
; endpoint: (X_end,Y_end)=(**120**,**180**)

; processing velocity **110** mm/sec:

```
N10 PLANE XY
N20 FASTABS X120000 Y180000
N25 VEL 110000
N30 CCWABS X120000 Y180000 I170000 J180000
```




(X_start,Y_start)=(X_end,Y_end)=(120, 180)

(X_center,Y_center)=(170,180)

## 3.5 Processing velocity

| VEL | Processing velocity |
|---|---|

**Syntax:** [set number]?

**VEL velocity**

**Statement:**
- velocity is a natural number
- with this command the processing velocity can be defined in the program
- the unit of the processing velocity is µm/sec

**Example:** ; define processing velocity of 100 mm/sec

```
N5  VEL 100000
```

## 3.6 Rapid velocity

| FASTVEL | Rapid velocity |
|---|---|

**Syntax:** [set number]?

**FASTVEL velocity**

**Statement:**
- velocity is a natural number
- with this command the rapid velocity can be defined in the program
- the unit of the rapid velocity is µm/sec

**Example:** ; define rapid velocity of 200 mm/sec

```
N5  FASTVEL 200000
```

## 3.7 Spindle command

| SPINDLE | Spindle command |
|---|---|

**Syntax:** [set number]?

**SPINDLE CW**, RPM [RPS] **speed**
**SPINDLE CCW**, RPM [RPS] **speed**
**SPINDLE ON**
**SPINDLE OFF**

**Statement:**

- **speed** is a natural number

- **RPM**: definition of speed unit = revolutions per minute
- **RPS**: definition of speed unit = revolutions per second

**SPINDLE CW**: **spindle on** clockwise
**SPINDLE CCW**: **spindle on** counterclockwise

**SPINDLE ON: switch on spindle** in the last declared mode (cw or ccw)

**SPINDLE OFF**: **switch off spindle**

**Example:** Spindle switch on clockwise with a number of revolutions 5000 n/min

```
N35 SPINDLE CW RPM5000
```

## 3.8 Tool change

| GETTOOL | Tool change |
|---|---|

**Syntax:** [set number]?
**GETTOOL tool number**

**Statement:**

- **tool number** {1-128}

**-** get the tool with the defined tool number

- with the tool change administration max. 128 tools can be configured for the available tool places

**Example:** ; Get new tool with number 3:

```
N10 GETTOOL 3
```

## 3.9 Program start, Program end

| ProgBegin<br>ProgEnd | Program start<br>Program end |
|---|---|

**Syntax:** **ProgBegin** *or* **ProgEnd**

**Statement:** **ProgBegin**: designate the start of the main program
**ProgEnd**: designate the end of the main program

## 3.10 Coolant

| | |
|---|---|
| **COOLANT ON/ COOLANT OFF** | **Coolant on**<br><br>**Coolant off** |

**Syntax:** [set number]?

**COOLANT ON/COOLANT OFF**

**Statement:** **COOLANT ON**: the pump for the coolant will be switched on
**COOLANT OFF**: the pump for the coolant will be switched off

## 3.11 Workpiece clamp

| | |
|---|---|
| **WPCLAMP ON/ WPCLAMP OFF** | **Workpiece clamp on**<br><br>**Workpiece clamp off** |

**Syntax:** [set number]?

**WPCLAMP ON/WPCLAMP OFF**

**Statement:** **WPCLAMP ON**: close workpiece clamp
**WPCLAMP OFF**: open workpiece clamp

## 3.12 Pump

| | |
|---|---|
| **PUMP ON PUMP OFF** | **Pump on**<br>**Pump off** |

**Syntax:** [set number]?

**PUMP ON/PUMP OFF**

**Statement:** **PUMP ON**: **Pump** (sucking off, compressor, vacuum) **on**
**PUMP OFF**: **Pump** (sucking off, compressor, vacuum) **off**

## 3.13 Lamp

| | |
|---|---|
| **LAMP ON**<br>**LAMP OFF** | **Lamp on**<br>**Lamp off** |

**Syntax:** [set number]?

**LAMP ON/LAMP OFF**

**Statement:** **LAMP ON**: signal lamp / working room lamp **on**
**LAMP OFF**: signal lamp / working room lamp **off**

## 3.14 Periphery option

| | |
|---|---|
| **POPTION1 ON/OFF**<br>**POPTION2 ON/OFF** | **Periphery option1 on/off**<br>**Periphery option2 on/off** |

**Syntax:** [set number]?

**POPTION1 ON/POPTION1 OFF**
**POPTION2 ON/POPTION2 OFF**

**Statement:** **POPTION1 ON**: Accessory 1 **on**
**POPTION1 OFF**: Accessory 1 **off**
**POPTION2 ON**: Accessory 2 **on**
**POPTION2 OFF**: Accessory 2 **off**

# 4 Working session

## 4.1 Start of Remote

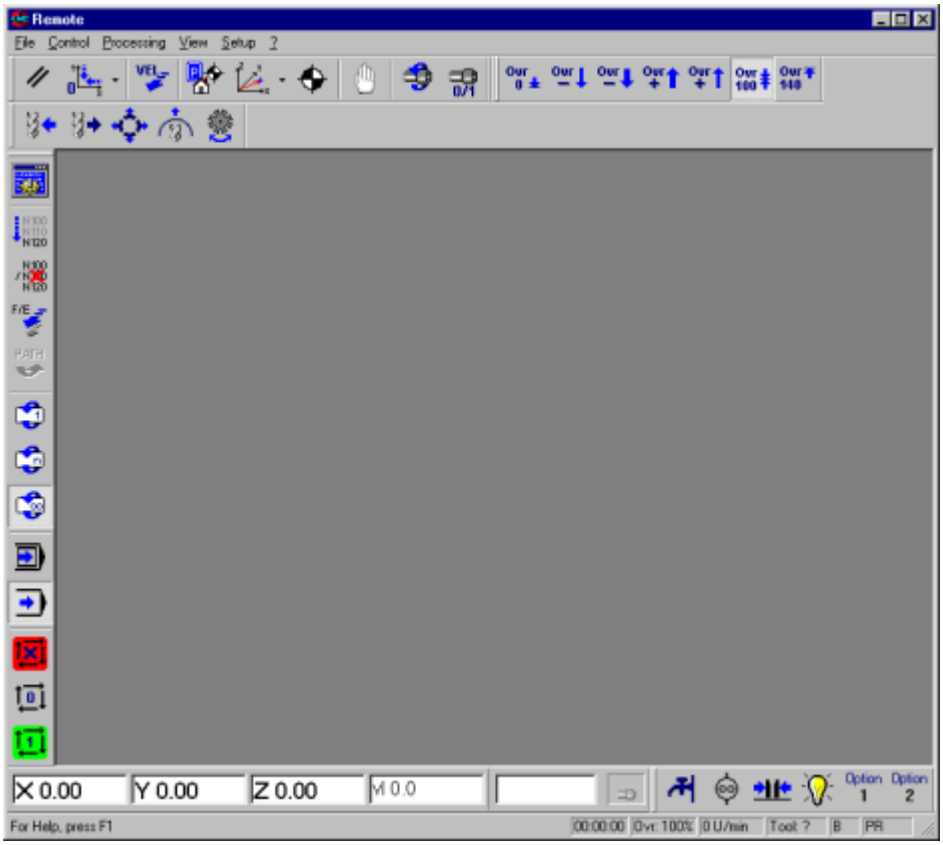
The example in this session is valid for a machine with the controller IMC4.

**Please unlock the emergency switch** at the controller, at the machine or at power supply.
**Switch on the power** in front of the equipment using the green key (POWER), close the cover.

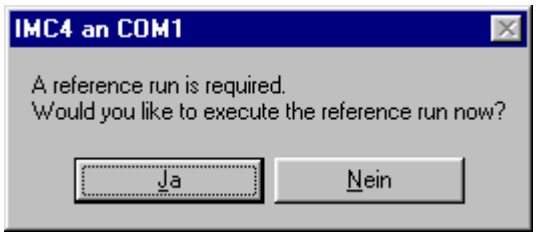
*Controller is powered up, PC boots up*.

**Please start** Remote with double click on the **program icon** .

*Remote will be loaded with the start screen.*
*All symbol bars can be placed on the screen arbitrary. Please click with the mouse in one of these windows, pull it with the pressed mouse key to the desired place on the screen.*
*Finishing the program the settings will be saved. By starting the program again the screen view will have the same appearance.*




If the machine is not ready for operation the program looks for the motion control and invites you to switch on the amplifiers on the machine.
A further window will inform you that a reference run is required.




The same effect you will reach, if you activate the button  **or** the menu **Control - Reset**

and then the button  **or** the menu **Control - Reference run.**

Please pay attention to a harmless possibility to approach the axes.

*If you will get a failure message e.g.*

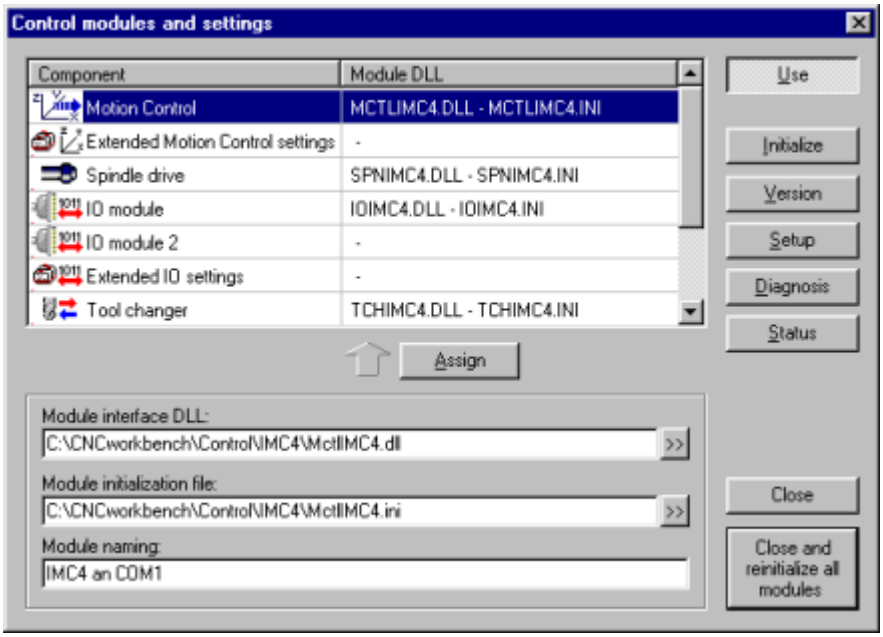
***„The module IMC4.DLL was not initalizated“***
***or***
***the axes do not approach not at all***

*is this a hint for you, that the Motion control and the settings are not correct.*

*In the dialog box "**Control modules and settings**" you have the possibility, to get an overview about the configured modules and the defined settings.*
*If necessary you can carry out changes.*

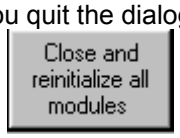
*please refer:* 2.8.3 Menu **Setup - Control**

A short example to execute modifications you will find in the next section. Select the menu **Setup - Control:**




Mark the component and then activate the button 

. The dialog box will be opened and modifications of parameter are possible (e. g. you can change the pitch of the axis X) .

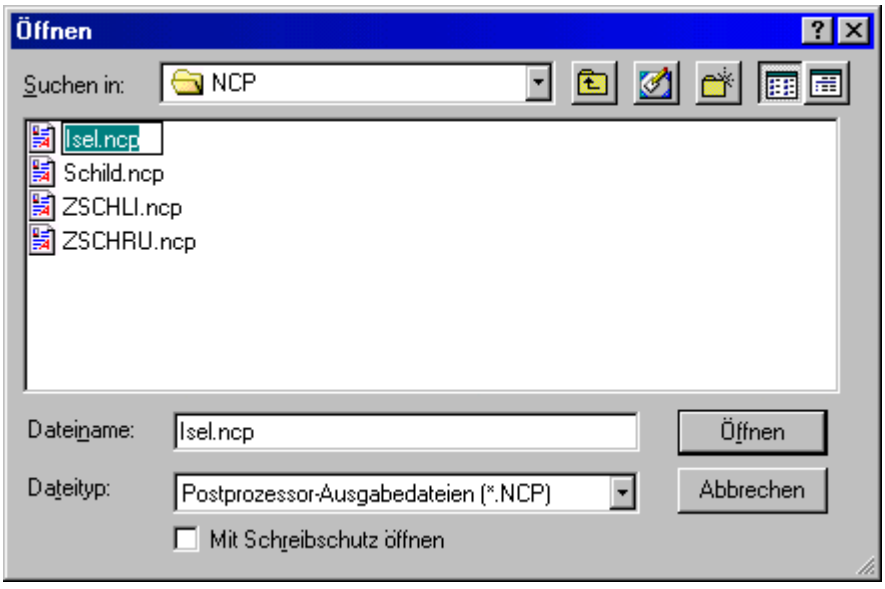
The modifications will be only effective, if you quit the dialog box "Settings" with "OK" and close the dialog box always with the button 

.

## 4.2 Open user program

With the program IsyCAM 3.0 the text "isel" was created with all necessary CAM-parameters. The workpiece zero point is marked by darts ⇨X and ⇧Y.

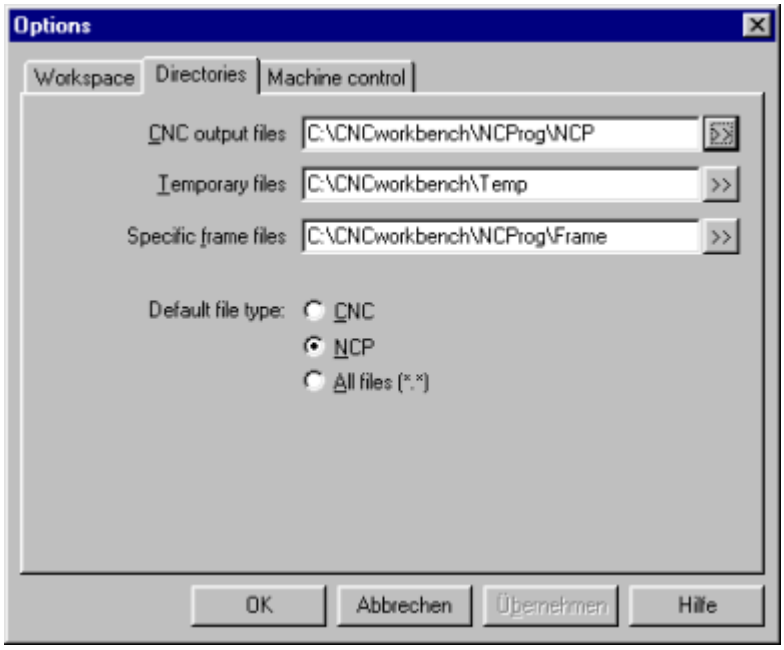

To open the program please **activate** the button  or the menu **File - Open** and load with a double click the file **isel.ncp** into the edit area of the screen.



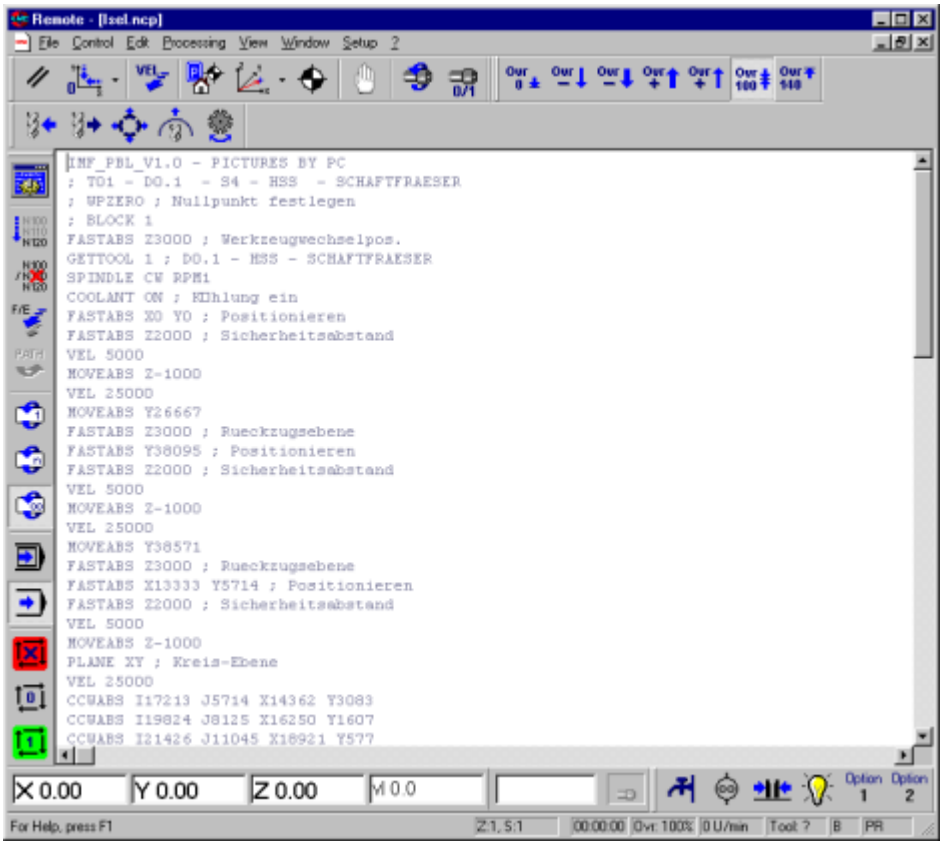

*please refer:* 2.2 Menu **File**

By selecting the index card "Directories" in the menu **Setup - options** please choose the file type you want to open mainly.

***Hint !***




Working favourite **with ISY** CAD/CAM, select **NCP**.
Working with programs created in ProNC, please select **CNC**.
Simultaneous it is meaningful, to install the directory containing the user programs.
Please select the button >> near the line output files and select the directory.

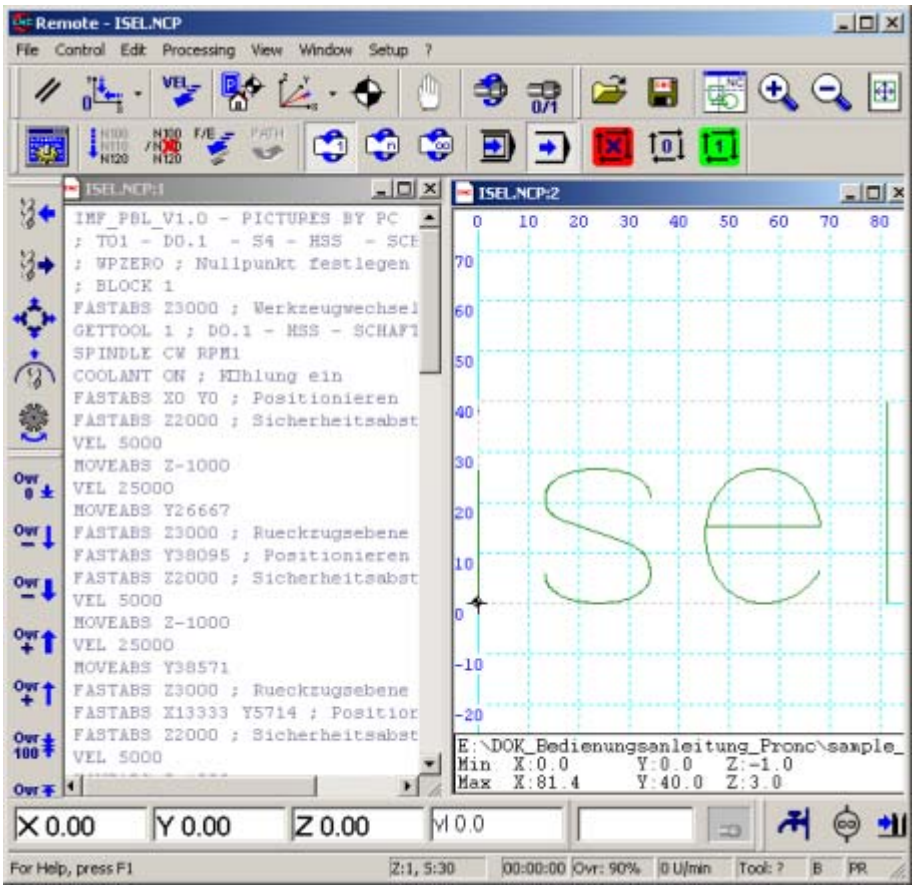



*The result of the successful loading should show the following picture.*

The light grey font in the user program is a note for you, that the file was opened in a write protected mode.
For changes in the NCP program, please deactivate the property "Write protected" or the parameter "Open source files in Read only mode" and open the file again.

For a additional graphical display of your user program, please choose in Menu Window "**Graphic display**" and then e. g. the option "**Tile horizontally**".

*please refer:* 2.2 Menu **File**
2.8.1 Menu **Setup - Options**
2.7 Menu **Window**

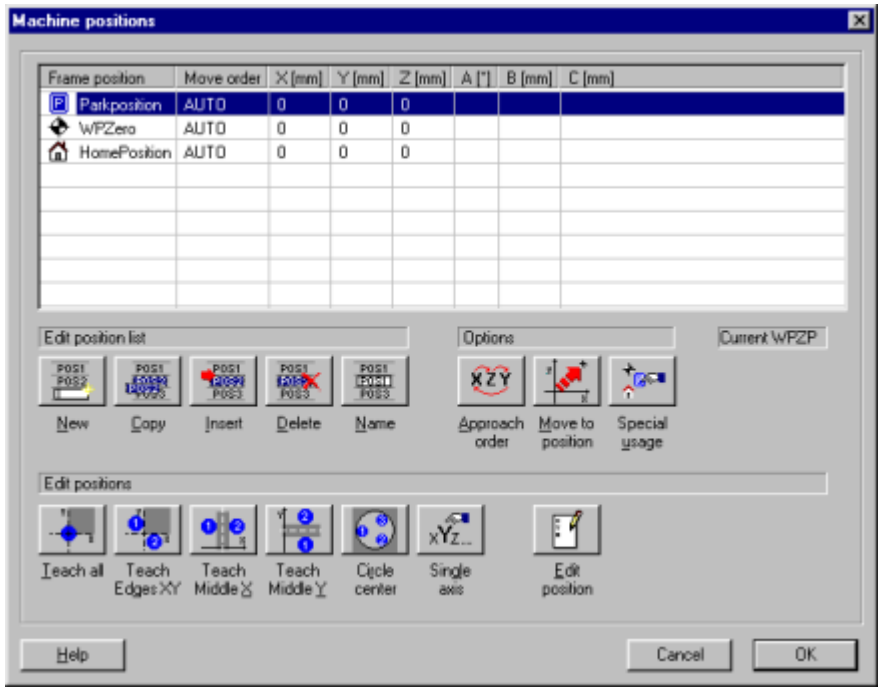



## 4.3 Determine workpiece zero point

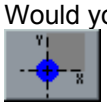
In result of the created file with ISY CAD/CAM 3.0 you know the dimension of the workpiece and the position of the zero point on your workpiece.
In this example you need a specimen with the dimensions at least 100 x 60 mm.
The zero point was defined in the lowest point of the letter i.
After clamping the workpiece, it is necessary to define this workpiece zero point at the equipment.

Select the menu **Control - Set up machine positions**.

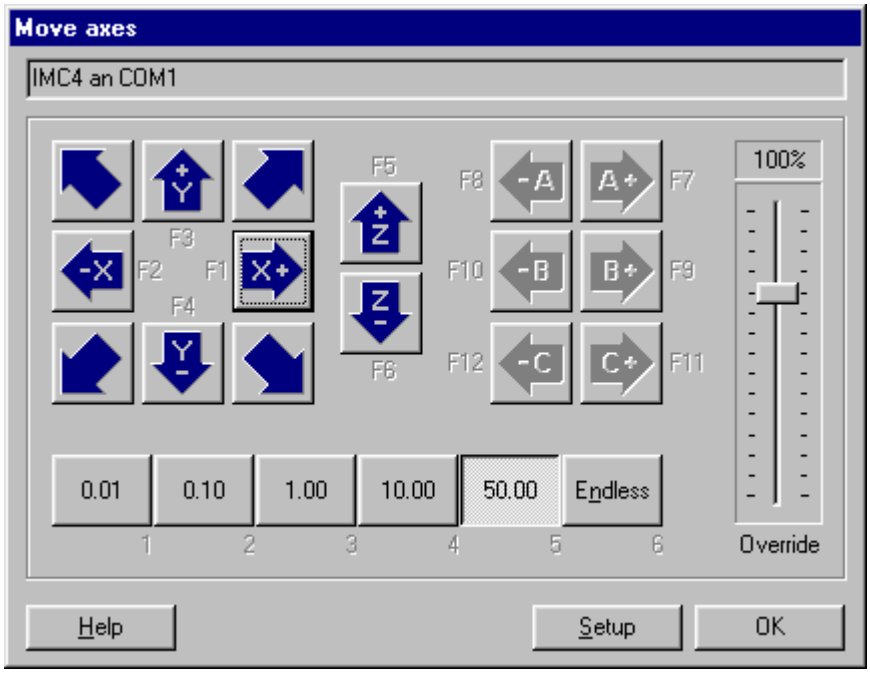
**Hint:** To display the values of the workpiece **in Remote,** please **move the mouse into the Graphic window**, press the **right** mouse-key and **choose** the option "File info".




To define the workpiece zero point please select with the cursor keys ↑ or ↓ respectively per mouse click the line WPZero.

Would you approach the workpiece zero point with hand (manually), push the button  or the keys Alt + T (Teach all).

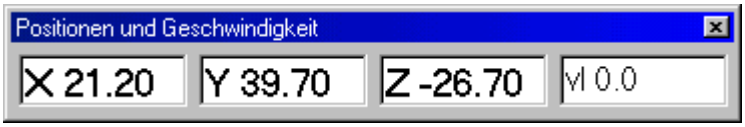
The window for the manual approach of the axes is displayed.



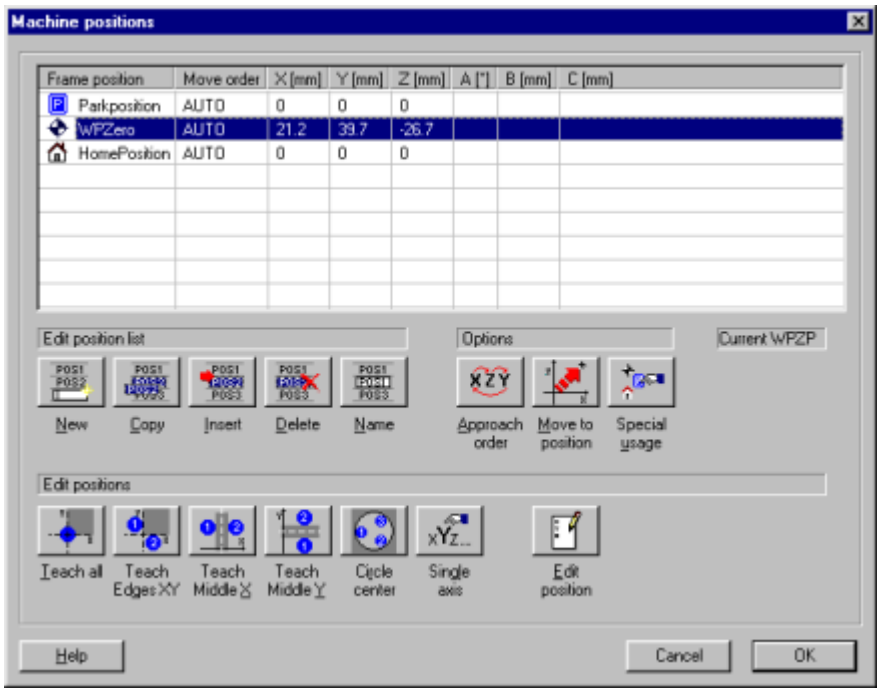
Adjust with help of the buttons:


the **step width** of the movement (unit = mm).

**Approach the axes**, that the tool peak touches the surface of the workpiece directly. Approach the X- and the Y-axis to the desired start position and sink the Z-axis carefully as long, as the workpiece is touched. Quit the dialog with "OK".
In the position window the approached coordinates are displayed e.g.:




Quit with "OK", the window machine position is displayed again and the position values are assumed.

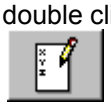


To define this position as zero point for all movements in your user program (NCP- or CNC file), the line with the position "Wpzero" must contain the token . If this token is not jet visible, please do the following:

Select the button  or the keys **Alt + V** (special usage) and assign the usage "Workpiece zero point". Now in front of WPzero the token is displayed to mark this position as workpiece zero point.

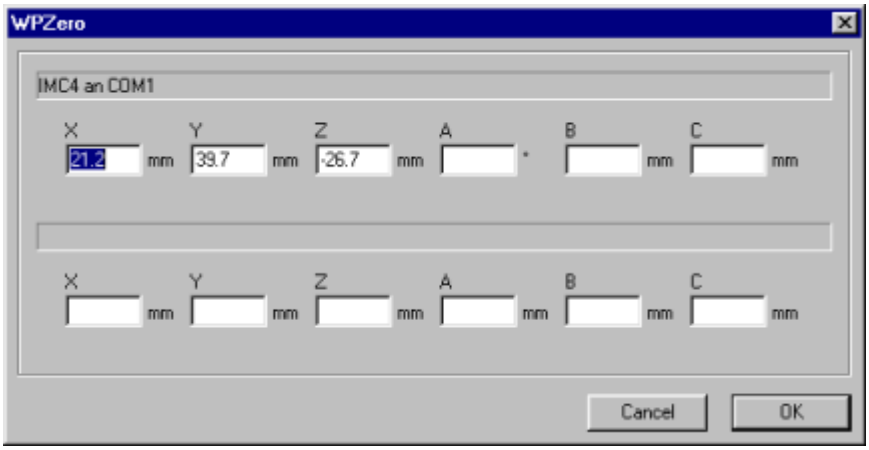
**Caution:** Only the marking **of the defined point as workpiece zero point** guarantees, that the options in the menu **Set up - CNC/NCP file processing** will be effective.

To define the approach order of the workpiece zero point, please activate the button  or the keys **Alt + S**
The selection "Move automatically" generates the option AUTO (default approach order X+Y (diagonal), Z).

If you don`t want to approach the workpiece zero point with hand (teach), but you want to enter the machine positions, please double click in the window "Machine positions" on the marking position *or* push the button  *or* use the keys **Alt + e** (Edit position).

In this window enter the position of the zero point with keyboard.

**Hint:**

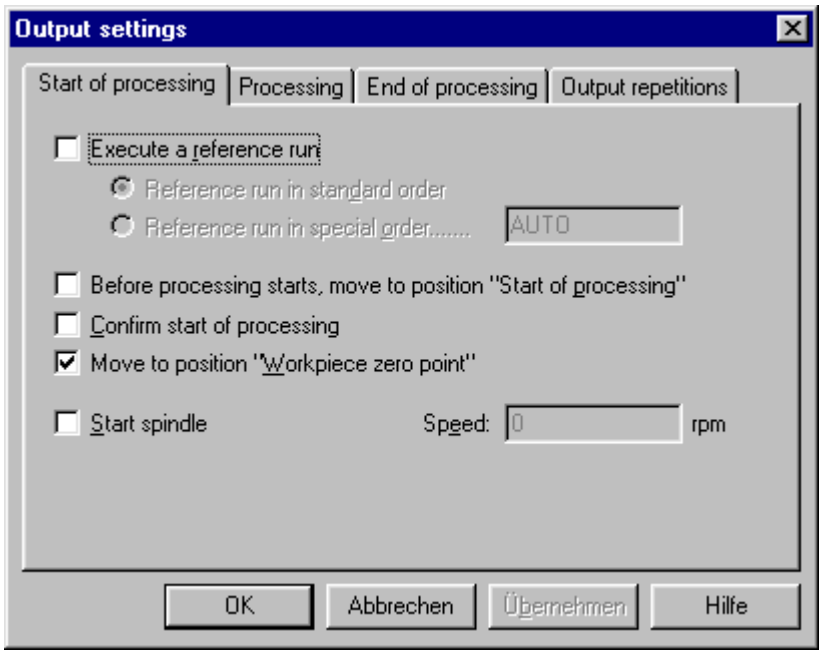



Quit with "OK", the window machine position is displayed again and the position values are assumed.
Please quit now the window "Machine positions" with the button "OK".

## 4.4 Set up to user file processing

Select the menu **Set up - CNC/NCP file processing** and define, which actions before/after the output of the user program shall be done.

**Select the index card "Start of processing":**



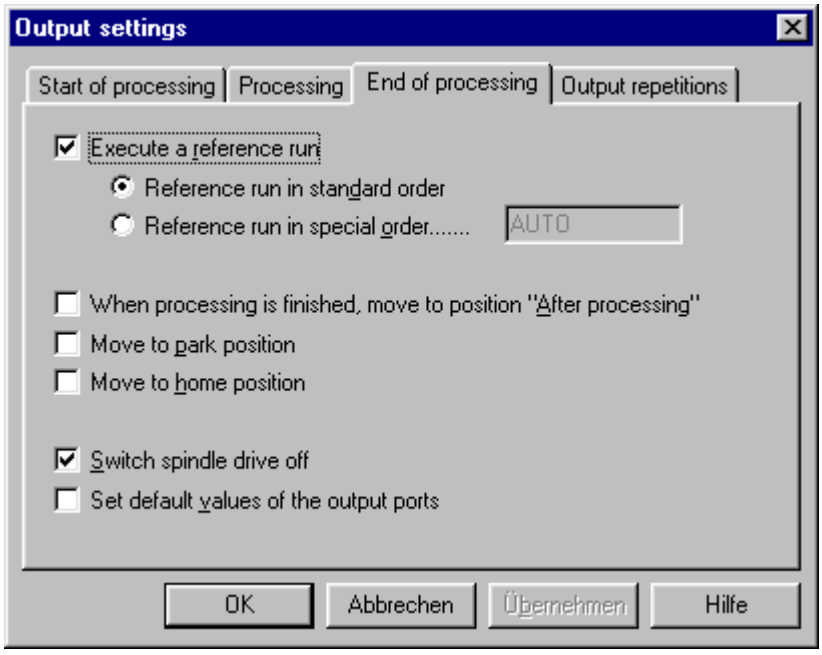

Activate the function:

- Move to position "Workpiece zero point".

This takes the following effect:

Before executing the first command in your NCP file by the interpreter the workpiece zero point, defined in "Machine positions", is approached.
If no workpiece zero point was defined in the window "Machine positions", the processing starts running out from the actual position of the tool.

**Select the index card "End of Processing":**



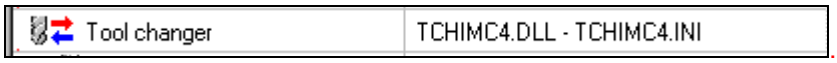

**Activate the functions:**

- Execute a reference run
- Switch spindle drive off

The defining of these functions is not connected to the respective user program.
If you start another user program you should check up, if the set ups can be used.

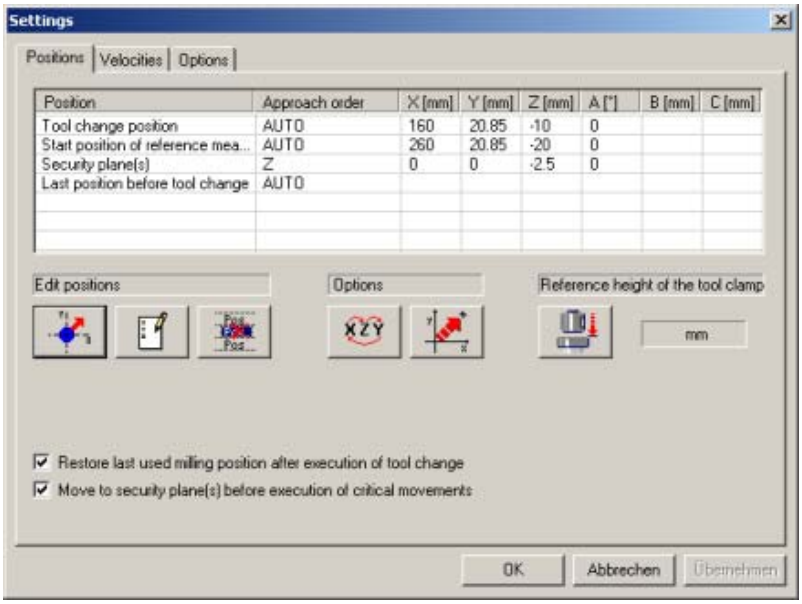
## 4.5 Settings tool change

Select the menu **Setup - Control** and click with the mouse on the line Tool changer:




After activating the button 

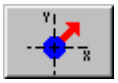
the input window for tool change- and reference positions will open.




Please define in according to the processing task the coordinates for the best position of the manual tool change by moving the axes with hand or entering with keyboard.
Please mark the line

| | | | | | |
|---|---|---|---|---|---|
| Tool change position | AUTO | 160 | 20.85 | -10 | 0 |

with the mouse.

Then please select the button  . The menu for manual movement of the axes is displayed.
Please do the same similar to define the workpiece zero point.

*please refer:* 2.3.4 Menu Control - Manual movement

After selecting the button  you can enter the coordinates with the keyboard.

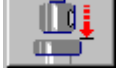
*please refer:* 2.3.5 Menu Control - Set up machine positions

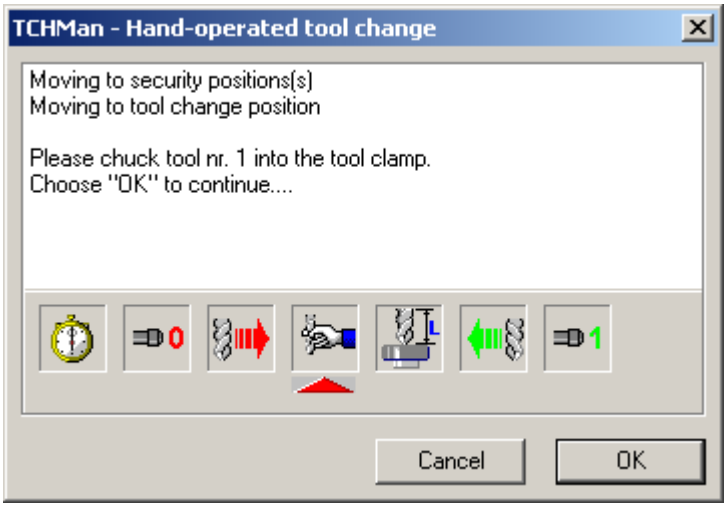
Please define the start point to reference the tool clamp by approaching the axes with hand. Choose position so, that the longest tool can be placed. During tool change a length measurement of the new clamped tool, starting from this position, will be executed automatically.

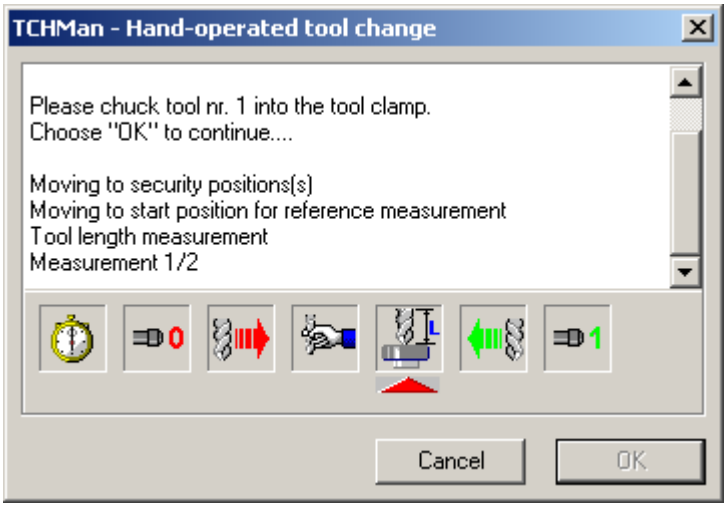
Select the line:

| | | | | | |
|---|---|---|---|---|---|
| Start position of reference measurement | AUTO | 0 | 149.225 | -33.675 | 0 |

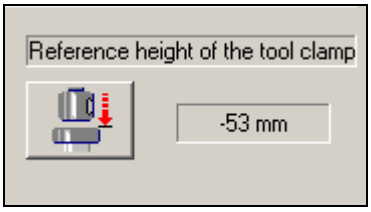
Teach the start position for referencing similar to the teaching of the tool change position.

Please determine the height of the clamp with a reference tool as base for the length measurement after tool change . Clamp a reference tool, push the button  and the measurement of the reference height will be executed automatically.




*After referencing the value will be enlisted and it is visible in this window.*
Example:




Independent of the number of tool changes in your user program, the NCP file created with ISY, contains always at least one GETTOOL- Command. If you intend to

deactivate the **dialog** for **tool change** generally, you have the possibility, to edit your user program as follows:

**A precondition** to edit the file is: Open the file **without the property: "Write protected"**.

*please refer:* 2.8.1 Menu **Setup - Options**

Please move the cursor into the line containing the command GETTOOL, write in front of the line a **semicolon**. This sign causes, that the program line will be interpreted as a commentary.
Following please save the file with the same name using the menu **File - Save** or clicking on the button  .

**Just thereby the changes will be effective.**

It is better, you do not deactivate the tool change; in order to execute a duly tool change after a disaster (e.g. the tool knocks off) or a manual tool change.

## 4.6 Program start

**We recommend** to use the mode „Single step mode“ (step by step) during the testing phase.

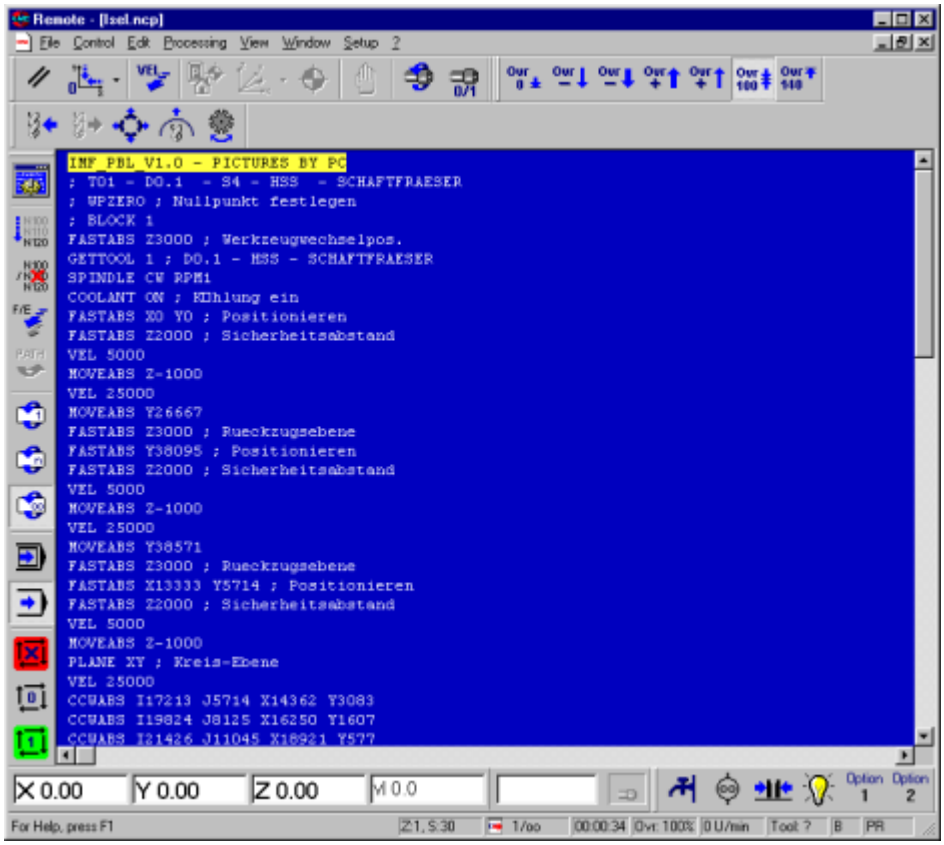
Please put in this mode with the button  or with the menu **Processing - Single step**.

This mode enables you, to follow the process step by step. After each command you can change to the mode "**Automatic**" (button  ).

**Start** the processing of the program with the button

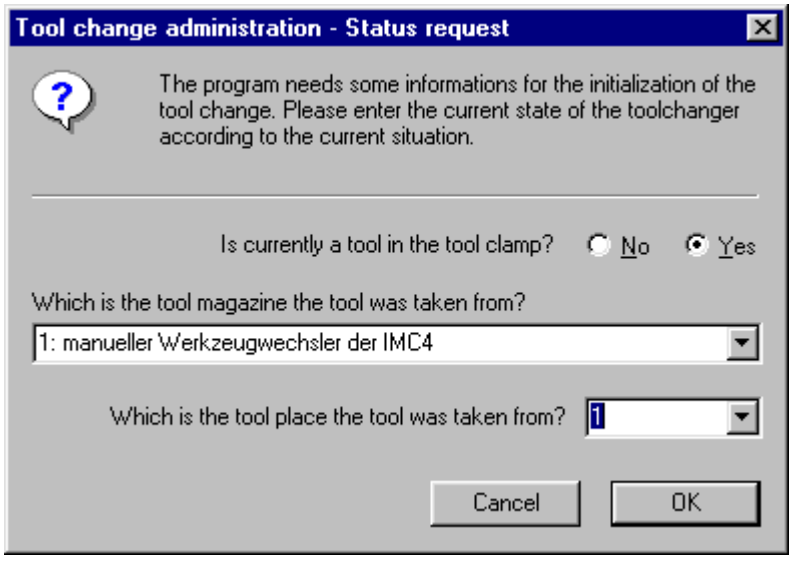
 **or** with the menu **Processing file in automatic mode**.

*The first line of the program is highlighted with a beam. After every click on the button start*  *the marked line will be executed.*
*In this way please follow also the annotations displaying in the status line of the screen.*

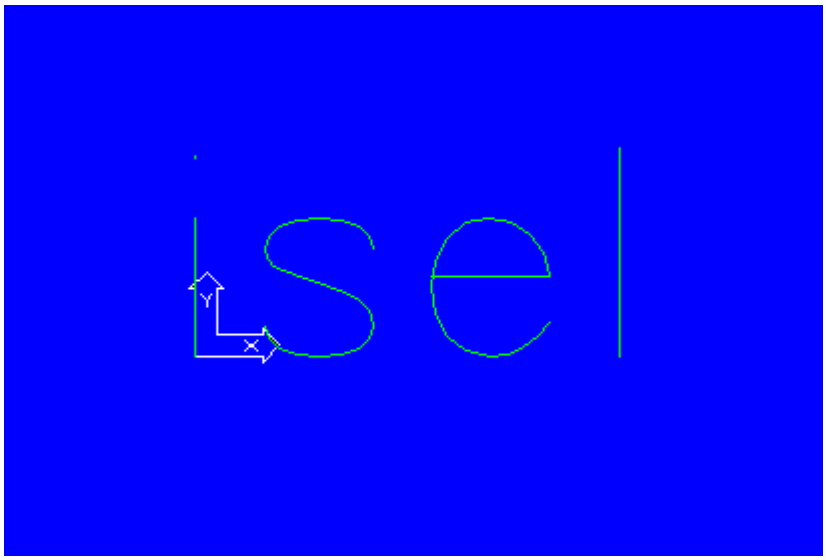



In dependence of the parameter to output the user file the defined workpiece zero point is approached before beginning the process.
The coordinates in the position window are shown with blue numbers to signal that the workpiece zero point is active.

At the **first** run of your program the GETTOOL-command causes a dialog. To define the current tool, please enter the relevant data in the window **Tool change management - Status request**.



After finishing the milling process, a reference run will be executed and the spindle drive will be switched off according to the setting "End of the processing".
The result of your processing should correspond to this picture.



# Glossary

**CNC file**

The CNC file is a in ProNC created and compiled file from a faultless source file in ISO/DIN- or PAL-Syntax. A CNC file can be immediately processed in Remote.

**DLL**

Dynamic-Link-Libraries are software modules, including one or more practicable functions. The DLLs are saved as independent files. They are called from the main program during running the main program, they are connected with the application at runtime and their functions are executed.

**Frame file**

A Frame file (as well Geometry file) can consist of some defined machine positions (as well called Frames). Each machine position is characterized with a frame name. In the simplest cause a Frame contains the current positions of maximum 6 axes.

**Home position**

The Home position is a exposed position of the axes. In the rule it is defined in the factory by assembling a special sensor.

This position ensures a maximum distance to the tool. The position is defined for a harmless opening of the cover e. g. within program running.

**IO**

Input/Output Module

**MCTL**

Motion Control Module

**Module**

Controller (e. g. a stepper motor-controller) or a device (e. g. a converter for a working spindle) or a controller card (e. g. the servo motor controller card or Multi-I/O-card) or a Hardware (e. g. a CAN-field bus interface)

**NCP file**

The NCP file is a user file generated by a postprocessor e. g. in the software ISY CAD/CAM 3.0. The NCP files own a specific syntax. To start the NCP file immediately in Remote without any conversions, the first line in the program must always contain the significant string IMF_PBL .

**SPN**

(Spindle)  Spindle module

**TCH**

Tool Changer

**Workpiece zero point**

Explained point (e. g. the bottom left corner of the workpiece in the X-Y-plane) is scratched and measured with the tool in set up mode

# Index