Welcome to

FAGOR

Fagor Automation S. Coop.

the workshop on the CNC 8055 MC

Workshop: Operating and programming the CNC8055 MC

FAGOR

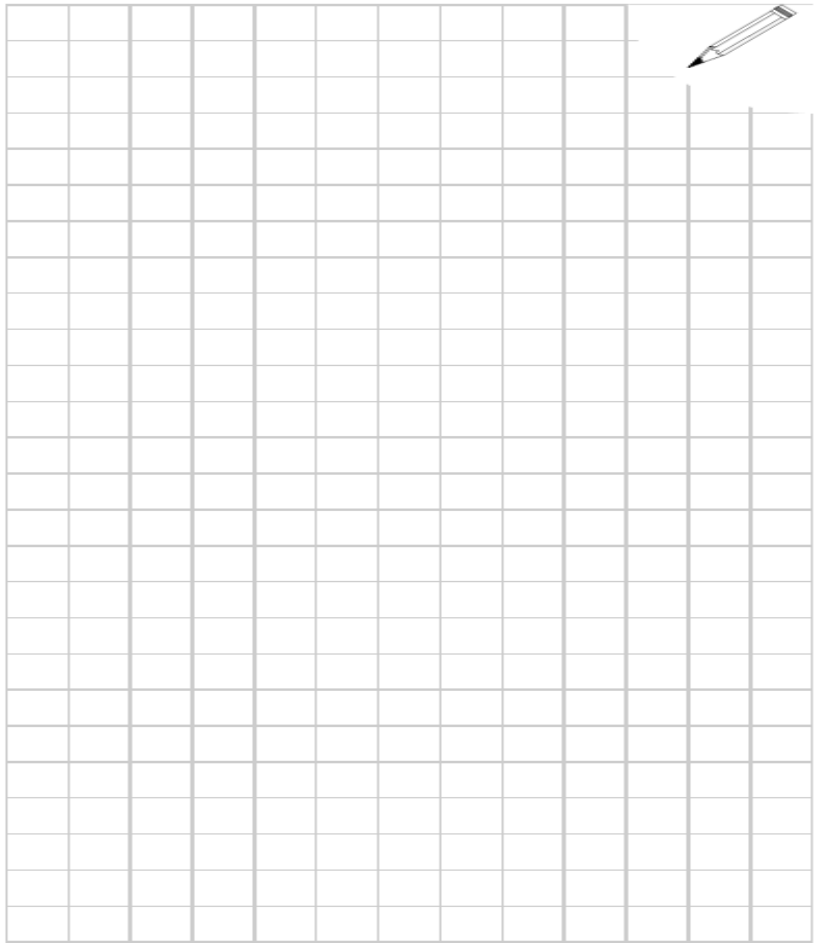
CNC 8055
CNC 8055i

This manual is part of the course for the **FAGOR CNC 8055MC**
which you can book at the company
FAGOR AUTOMATION.
It includes examples and explanations additional to
FAGOR **documentation for this** CNC **system.**
We also refer to these manuals
and to the additional manuals of the machine builder.

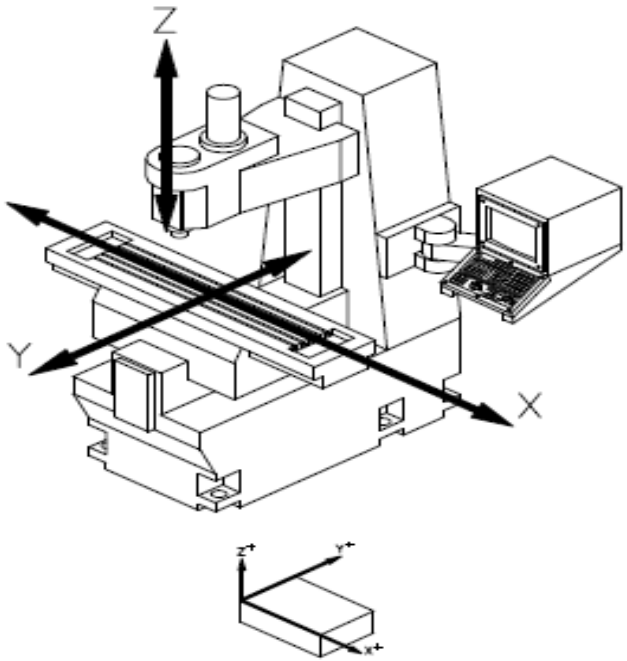
Overview seminar contents:

- **Theory on milling machines**
  - Defining the axes
  - Spindle
  - Axis feed rate
  - Tool changer
- **Zero points**
  - Home position
  - Machine reference zero
  - Part zero
- **Machine power up**
- **Operating panel CNC8055**
  - Display
  - Keyboard
  - Control elements
- **8055 CNC structure**
  - Basic system M
  - Conversational M system
- **Home position**
- **Tools**
  - Tool calibration
  - Tool parameters
  - Tool table
- **Programming cycles**
  - Principle
  - Overview
  - Important keys
- **Cycles**
  - Training and examples
  - Tool calibration
  - Facing
  - Boring
  - Grooving
  - Center punching and drilling
  - Tapping
- **Program management**
  - Save, delete, etc.
- **Cycles**
  - Training and examples
  - Profiles
  - Profile pockets
  - Rectangular boss
  - Circular boss
  - Grooving
  - Positioning
- **ISO functions**
  - ISO in MC mode
  - Part zero presetting cycle
  - work zones
  - Labels and jumps
  - Mirror image function
  - Scaling factor
  - Multiple positioning
  - Connection with cycles
  - Types
- **Examples**
- **Functions in M mode**
- **WINDNC**
- **Backup**
- **Diagnose**
- **Examples**

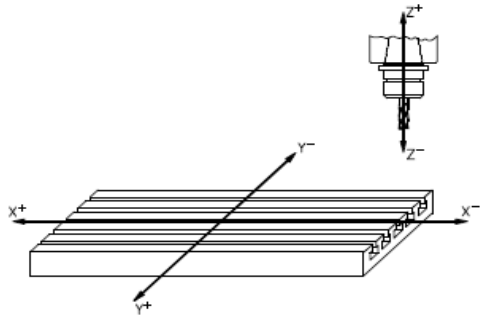
Notes



Milling machines – Definition of axes and coordinate systems

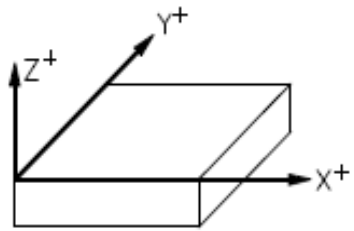


Typical 3-axis machine



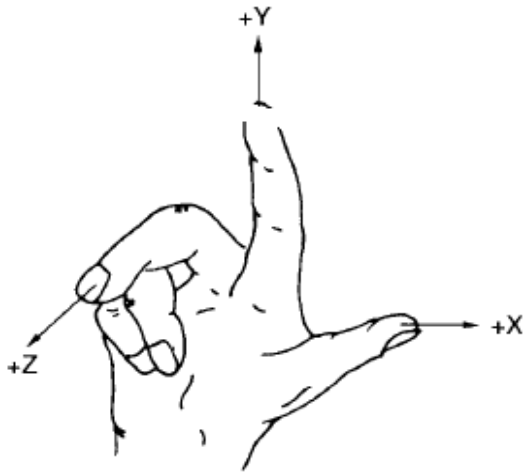
Basically , both the table and the tools can be moved.

Definition when programming:
the tool is moved !!

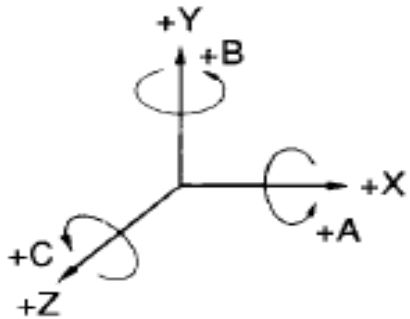


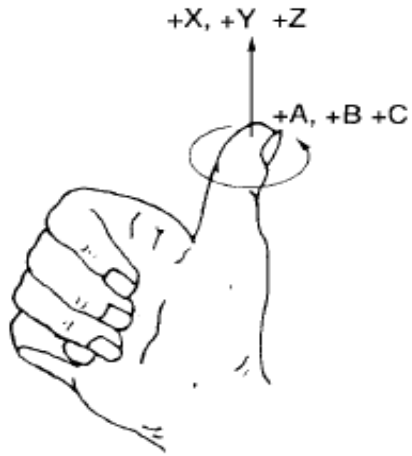
Orientation of the machine axes

The orientation of the axes depends on the type of machine.
"right hand rule"



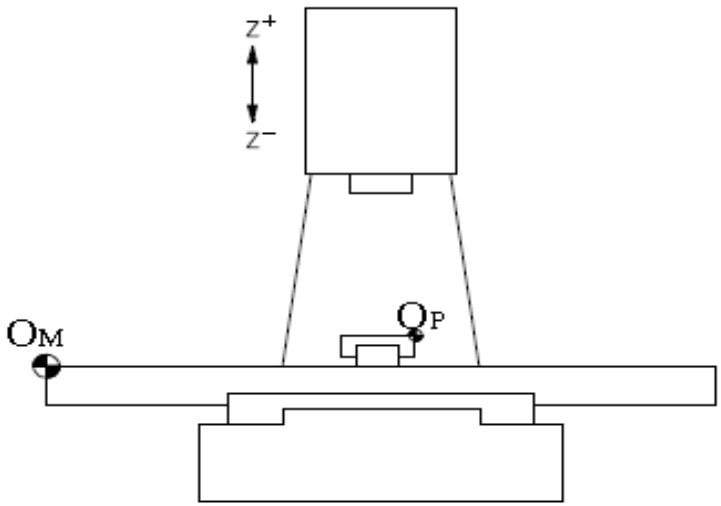
Linear axis orientation





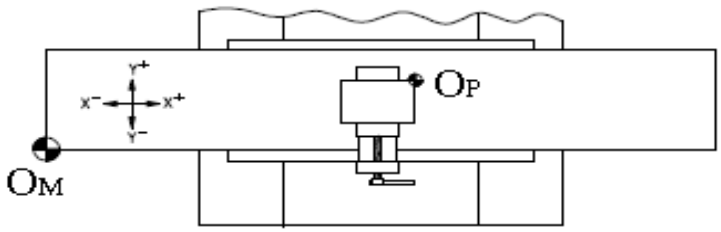
Rotary axis orientation

Machine zero and part zero



O_M – machine zero point

The machine builder defines the machine zero for all axes. This is the reference for all dimensions at the machine.

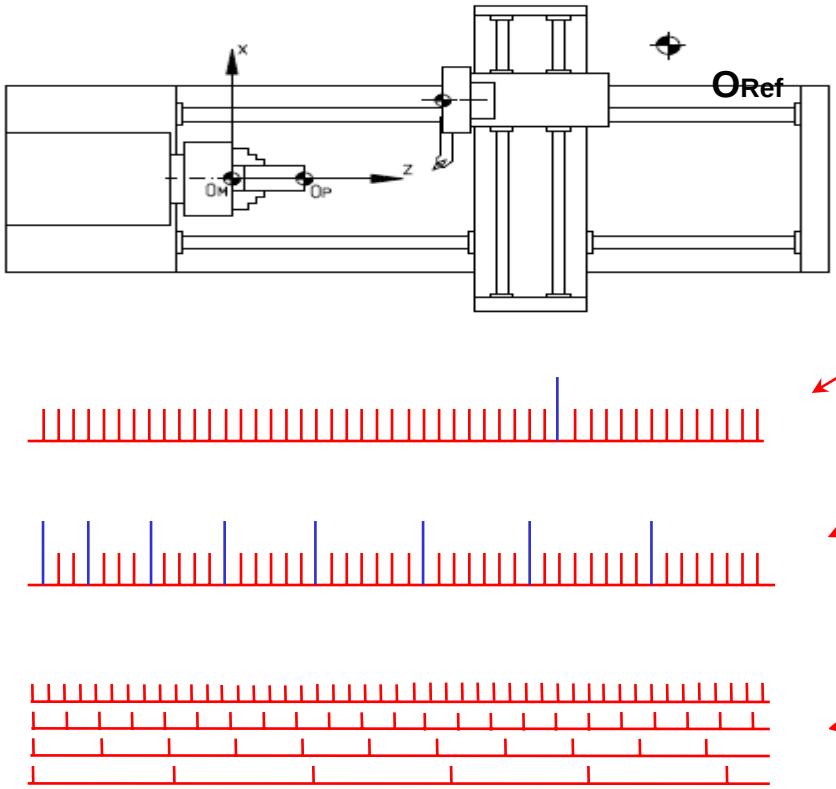


O_P – part zero point

The CNC sets the part zero.
This can be anywhere on the part.
He will normally use a point that relates to the blueprint.

Axis feedback

Each axis of the machine is equipped with a feedback system, so that the CNC can control the position and the speed of the axis.



The machine can be equipped with different feedback systems.
The feedback systems of each axis must be homed after power-up (*"HOME"* key).

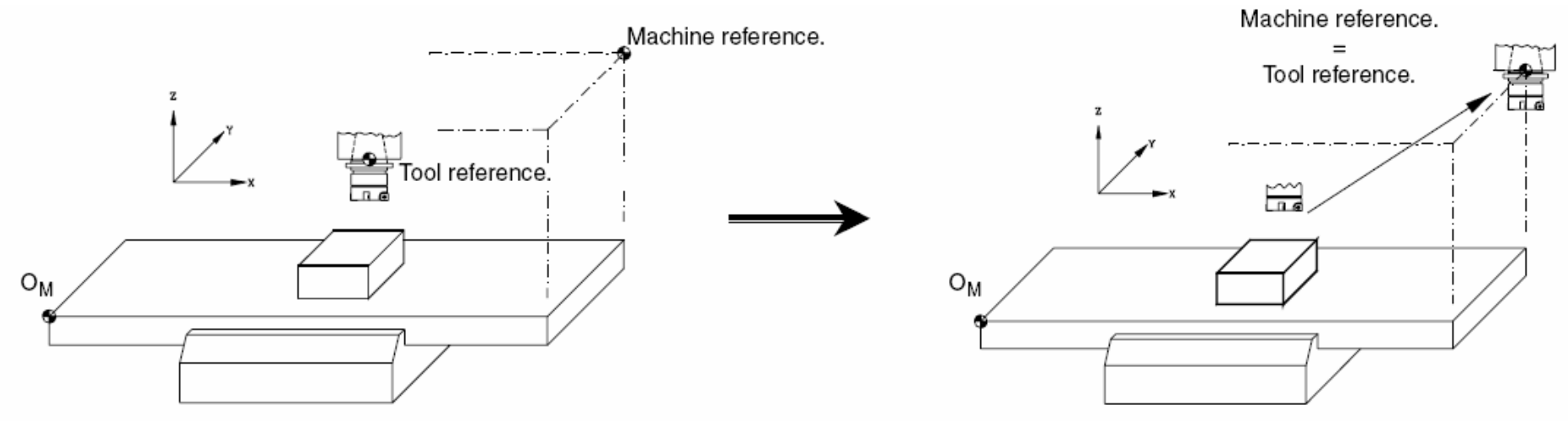
Incremental feedback system:
To initialize (home) the feedback system, the axis has to move over the reference point **0Ref**

Distance-coded feedback system:
The axis only has to move over two reference marks to find the initial reference position.

Absolute measuring system:
Multi-channel measuring system that indicates the exact position directly.
No home search needed.

Home position

All axes that are not equipped with absolute feedback systems must be homed on power-up. For this, the axes are moved to a special point at the machine that the machine builder has defined as home position **ORef**
After this the CNC knows the exact position of the axis and can calculate the machine zero point **OM.**
Then, the operator can define a part zero point **OP.**



Check on your machine how the homing will be carried out!!.

Milling machines – feedrate of the axes

The axis feedrate is defined by the manufacturer.
Each axis has its own rapid feedrate and maximum machining feedrate

Feedrate setting:

- mm/minute — Axis feedrate is independent from the spindle speed. (ISO code: G94)
- mm/rev — Axis feedrate varies depending on the spindle speed. (ISO code: G95)

The start-up condition is defined by the machine builder
Normally, milling machines start up in mm/minute

Feedrate is programmed with Fxxx.xx

Milling machines - main spindle

Depending on the requirements, the main spindle can have different configurations.

Main functions:

Open loop control – simplest variant

The CNC cannot react to spindle rpm changes because it does not have an encoder (no feedback)

Closed loop control – the most often used variant

The spindle motor is equipped with an encoder. The CNC can control the speed and the position of the spindle.
This function is needed for spindle positioning and also for tapping.

Closed loop control with C axis functions

The spindle motor is equipped with a high resolution encoder and a high resolution servo drive.
This allows exact control of the spindle position. In this case the spindle may be controlled like an axis.
It is possible to interpolate the C axis with the X and Z axes.

Milling machines – tool changer

Machines may be equipped with a wide range of tool holders or tool changers.
Fagor CNC´s can handle a lot of tool changers. The machine builder implements the right functionalities at the CNC and the PLC programs.

Manual tool changer:
The operator changes the tool manually like on a conventional machine.

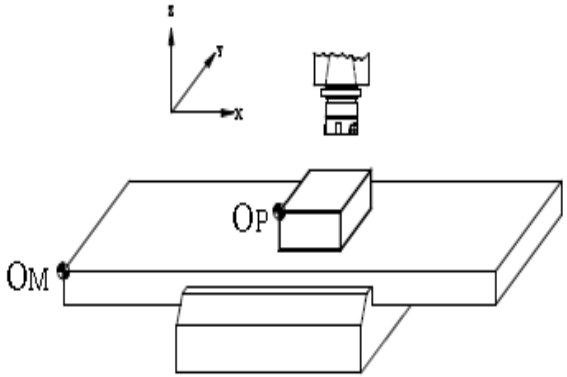
Automatic tool changer:
The tool is changed automatically.

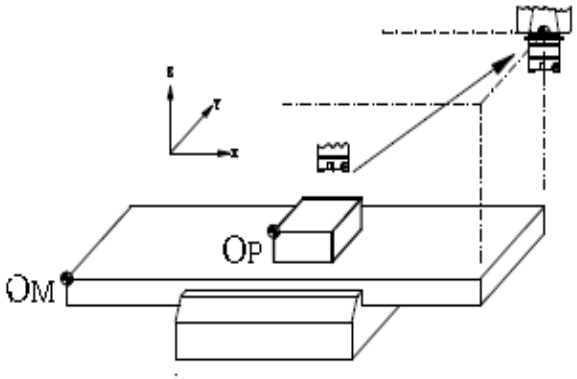
Home position and machine zero

The feedback systems must be homed right after power-up.

There are two ways to do it.

1. Automatic home search





The position of the axes may be unknown on power-up

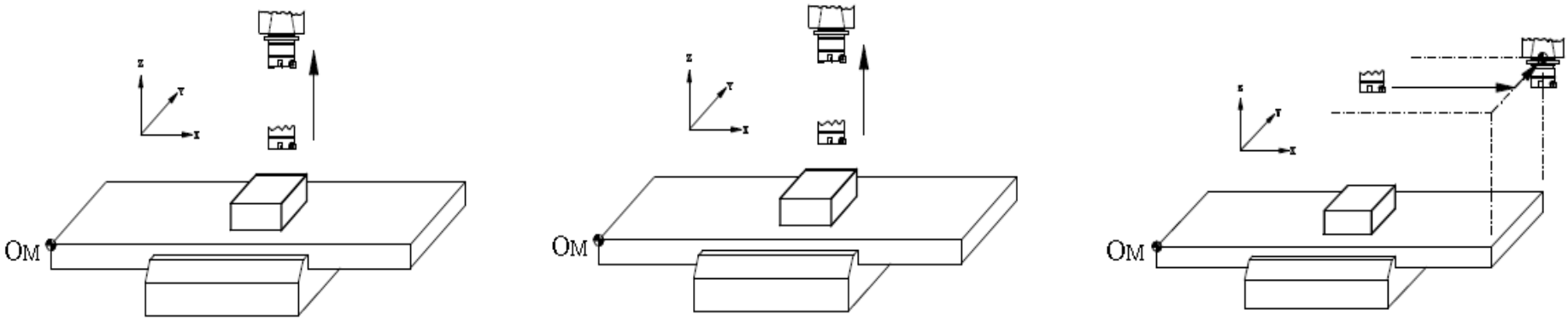
All axes move to the home position depending on a special subroutine defined by the machine builder.

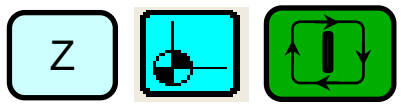
In this mode the CNC restores a part zero or zero offset that is active at the time.

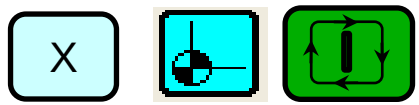
Home position and machine zero

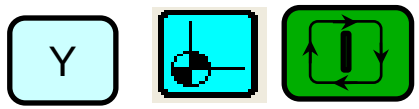
2. Manual home search

Homing is done axis per axis controlled by the machine operator.





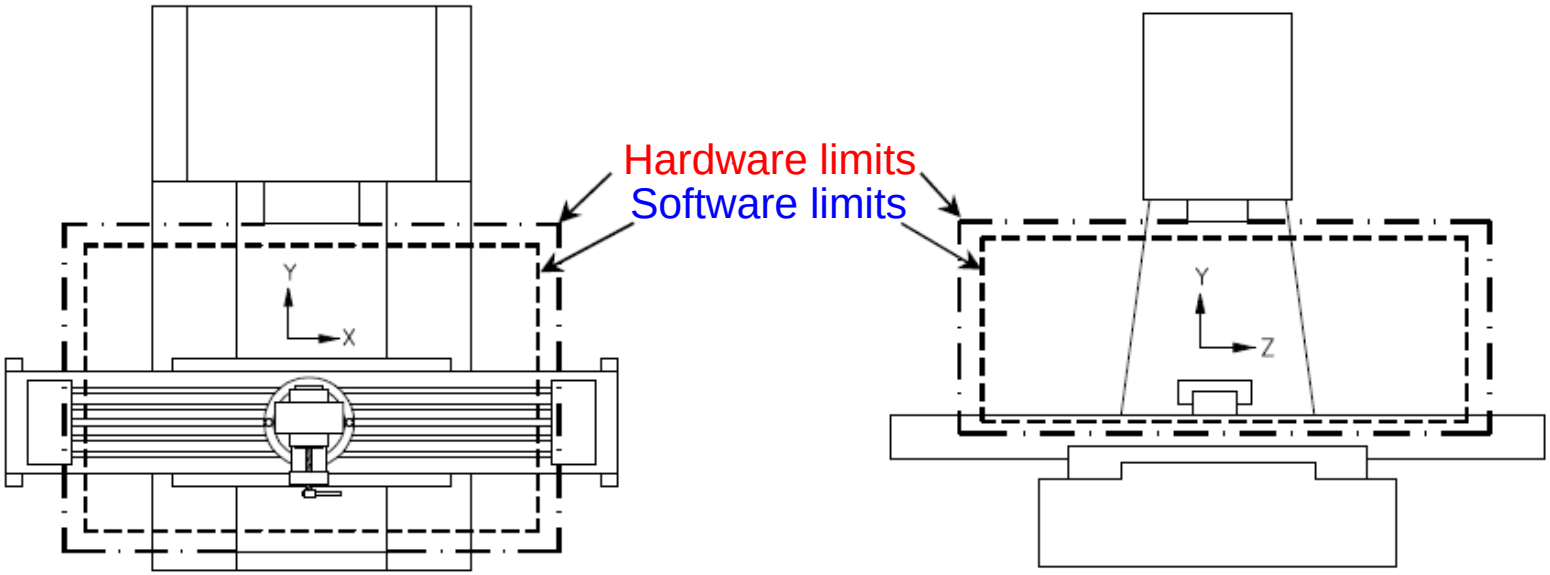




In this mode, the CNC resets any active part zero or zero offset.

Travel limits

There are two types of travel limits to save the machine from collisions



Hardware limits: Electrical switches installed at the ends of the way to stop the table before mechanical collision.

Software limits: CNC parameters, set by the manufacturer to prevent the table from running into the machine´s hardware travel limits.

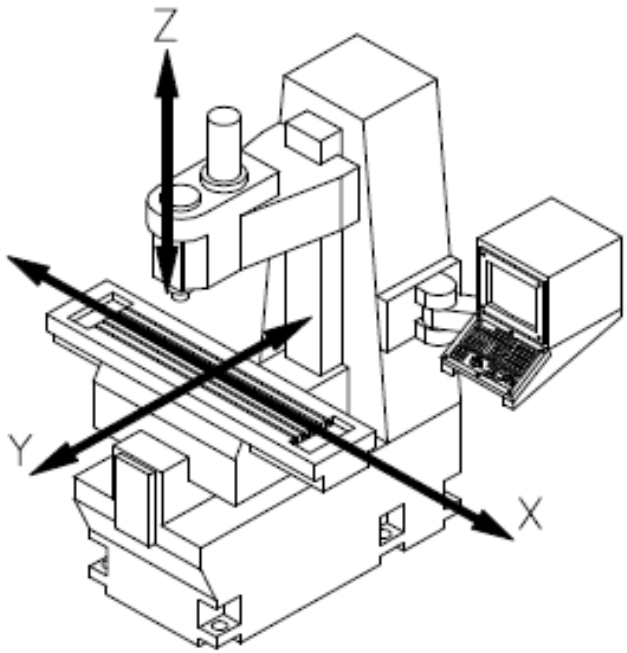
Soft limits are active after homing.

Machine power up

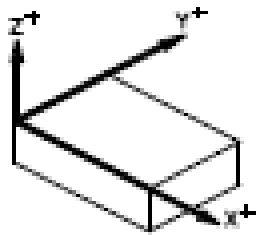
Power-up, maintenance and use of the machine must be done following the instructions of the machine builder.

Programs and examples of this course have to be carefully checked before using them on a real machine.

Coordinate system of a 3-axis machine

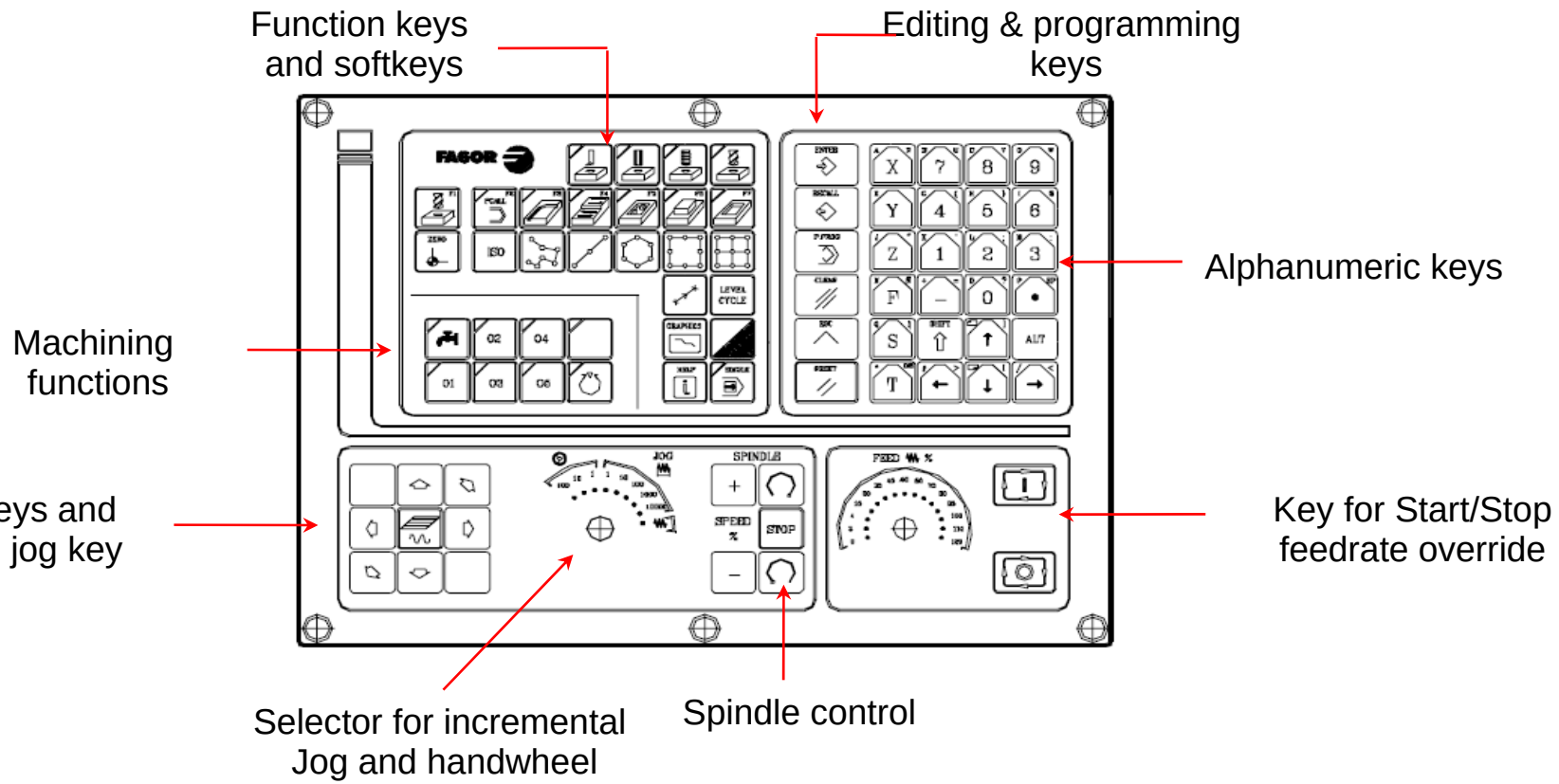


Part coordinates system



Operator panel 8055 MC

The CNC can be equipped with different operator panels.
In the course we use a panel with an "11" TFT monitor



HUMAN MACHINE INTERFACE - HMI

Two HMI´s have been implemented in the CNC 8055MC

- **Basic HMI: M**
  Offers full functionality to control complex milling machines and machining centers.

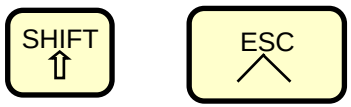
  Maintenance, CNC programs, PLC program, parameters, servos, diagnosis and communication lines.

- **Conversational HMI: MC**
  full functionality of the M HMI

  +

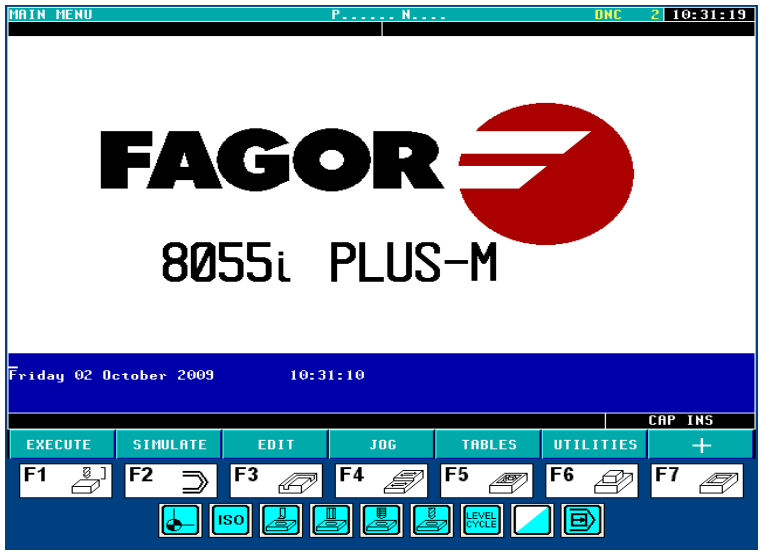
  Intuitive operation and programming of milling machines.
  Conventional operation of the machine.
  Programming with cycles without ISO knowledge.

To toggle between the HMI´s press the keys

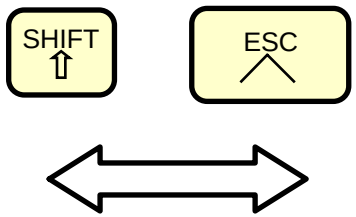


Displays of the M and MC type HMI's

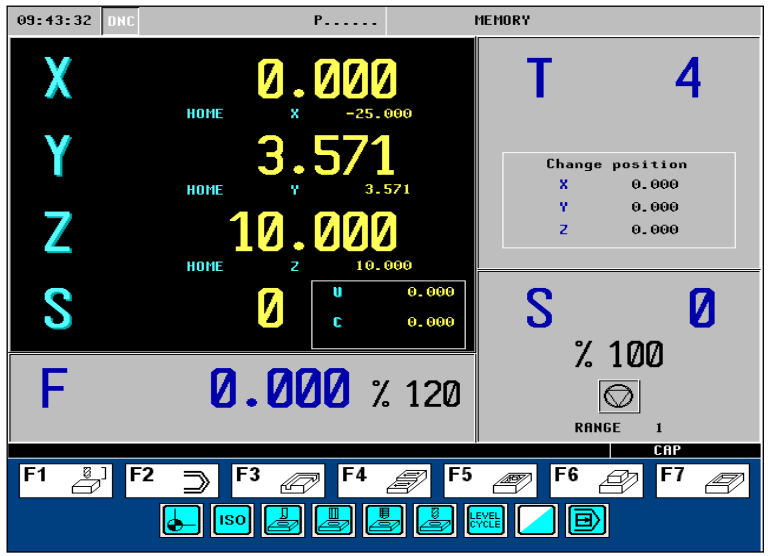
Basic system M



toggle



MC conversational mode

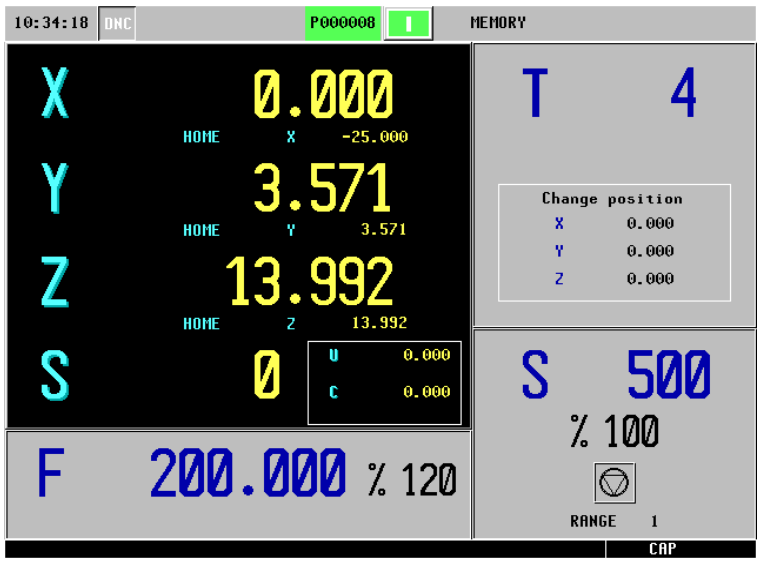


Screens of the MC mode

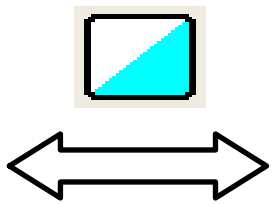
The TC mode shows two screens for displaying information

- Auxiliary screen
- Standard screen

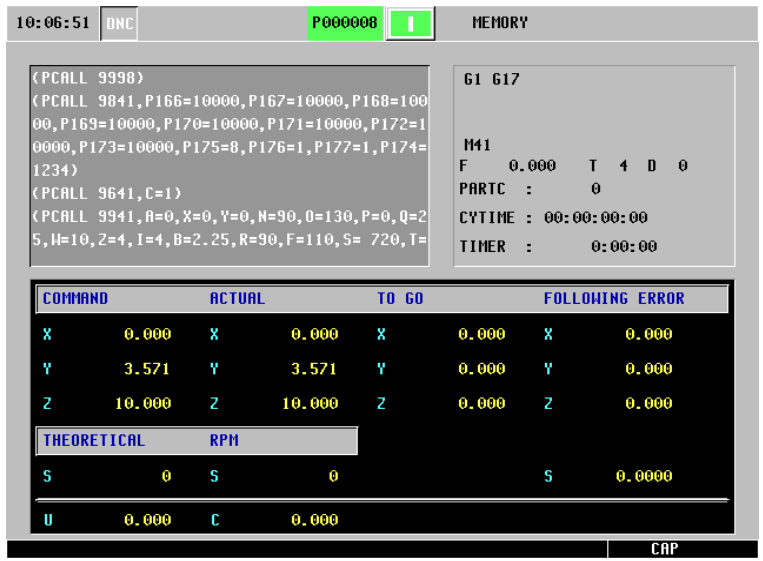
Standard screen



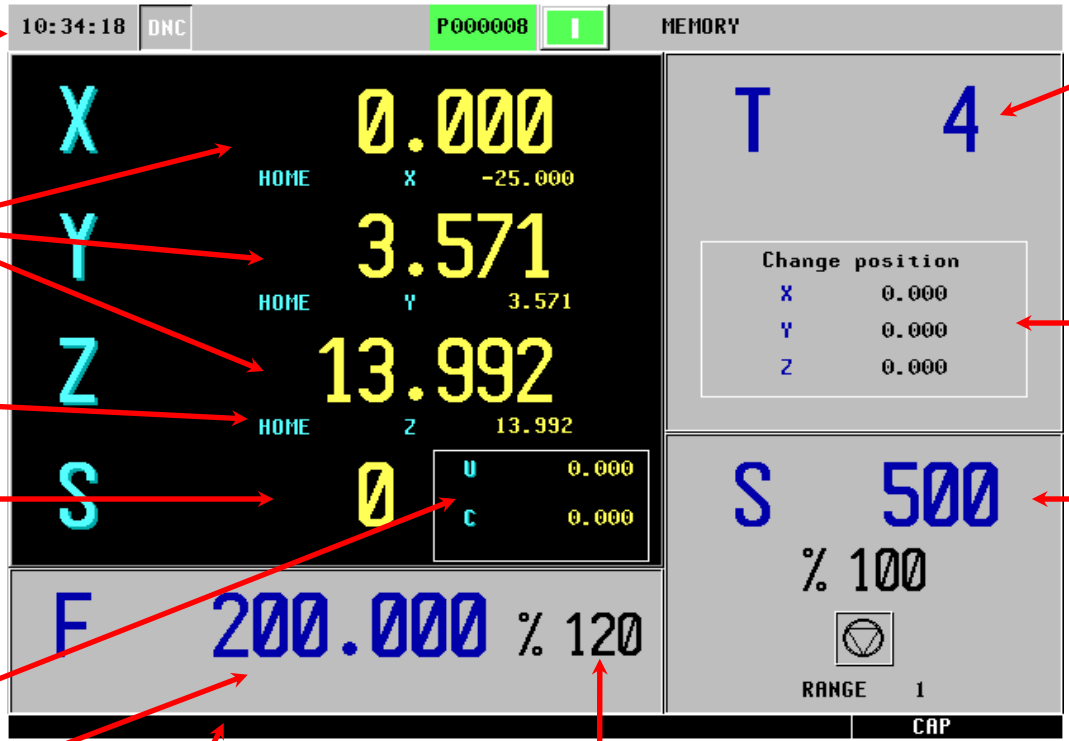
toggle



Auxiliary screen



Display: MC standard



Status line

Current axis position

Home position

Current spindle speed

Example of rotary axis and auxiliary axis

Current feedrate

info and messages

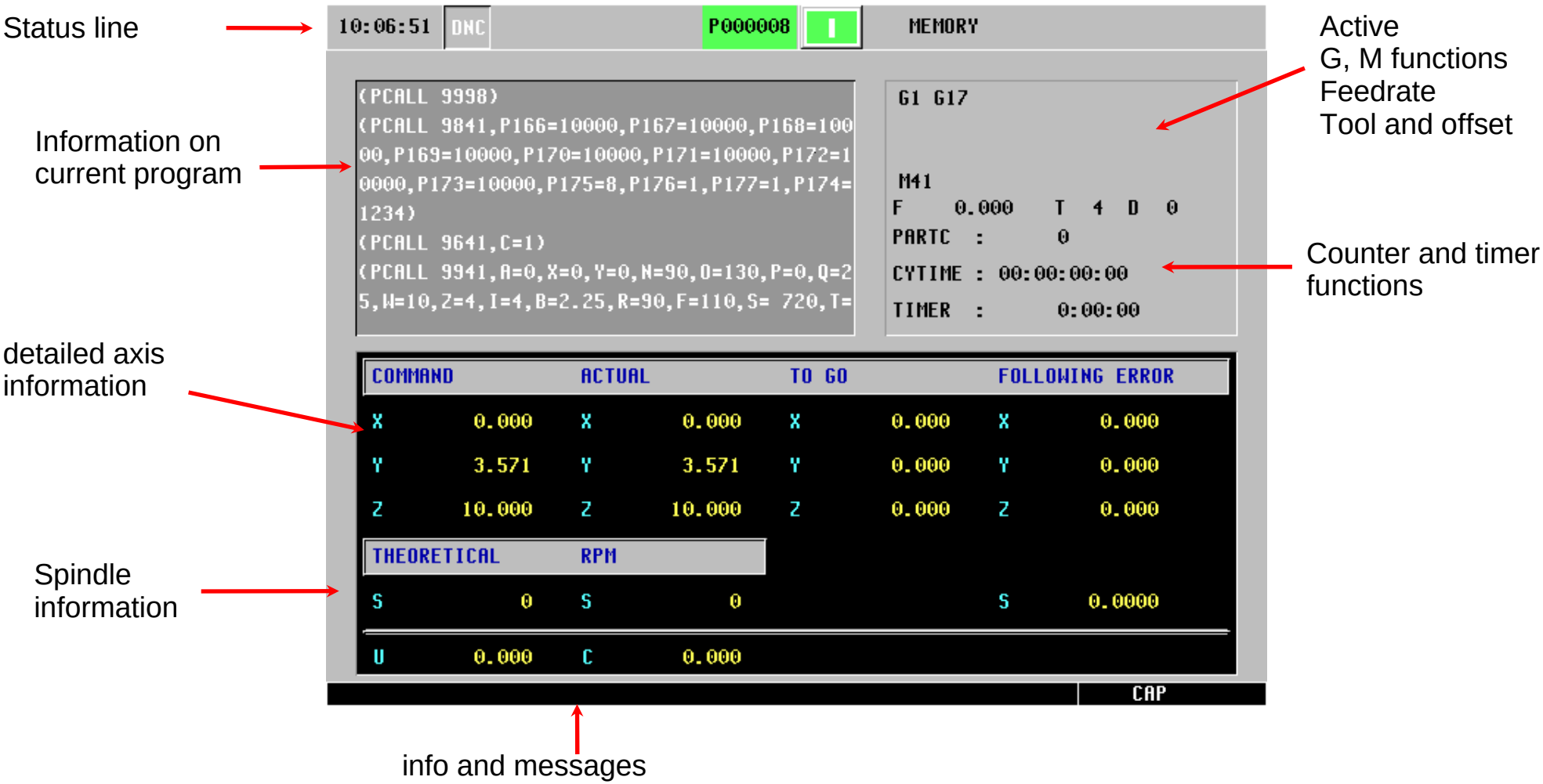
Feedrate override

Active tool

Tool change position

Spindle parameters:
Theoretical speed
Turning direction
Speed override
Active gear

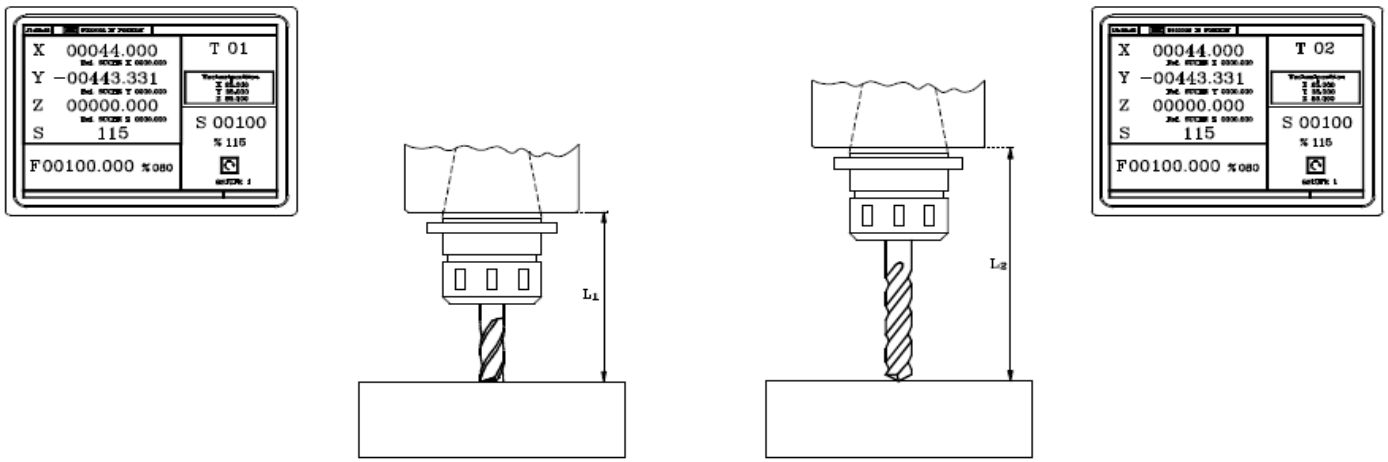
Display: auxiliary TC mode



Tool calibration

- Mainly the CNC program describes the profile of the part.
- The result must be the same regardless of the tool selected by the operator.

- For this reason, the CNC has to know the dimensions of the tool
- The operator has to calibrate the tools.
- The results of the calibration are stored in the tool offset table.



Different tools touch the part at the same point

Premise: all tools are exactly calibrated

Tool Offset table

The CNC can handle up to 255 tool offsets in the tool offset table.

Each tool T is assigned a D offset, by default with the same number.
Every D offset can be assigned to each tool T by CNC program.

TOOL OFFSET TABLE P..... N.... DNC 2 10:33:50

OFFSET	RADIUS		LENGTH		RADIUS WEAR		LENGTH WEAR	
D001	R	20.0000	L	0.0000	I	0.0000	K	0.0000
D002	R	12.5000	L	0.0000	I	0.0000	K	0.0000
D003	R	5.0000	L	0.0000	I	0.0000	K	0.0000
D004	R	0.0000	L	0.0000	I	0.0000	K	0.0000
D005	R	4.0000	L	0.0000	I	0.0000	K	0.0000

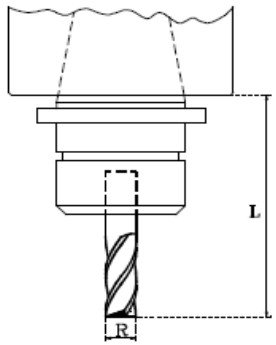
tool offset D

tool radius R

tool length L

radius offset L

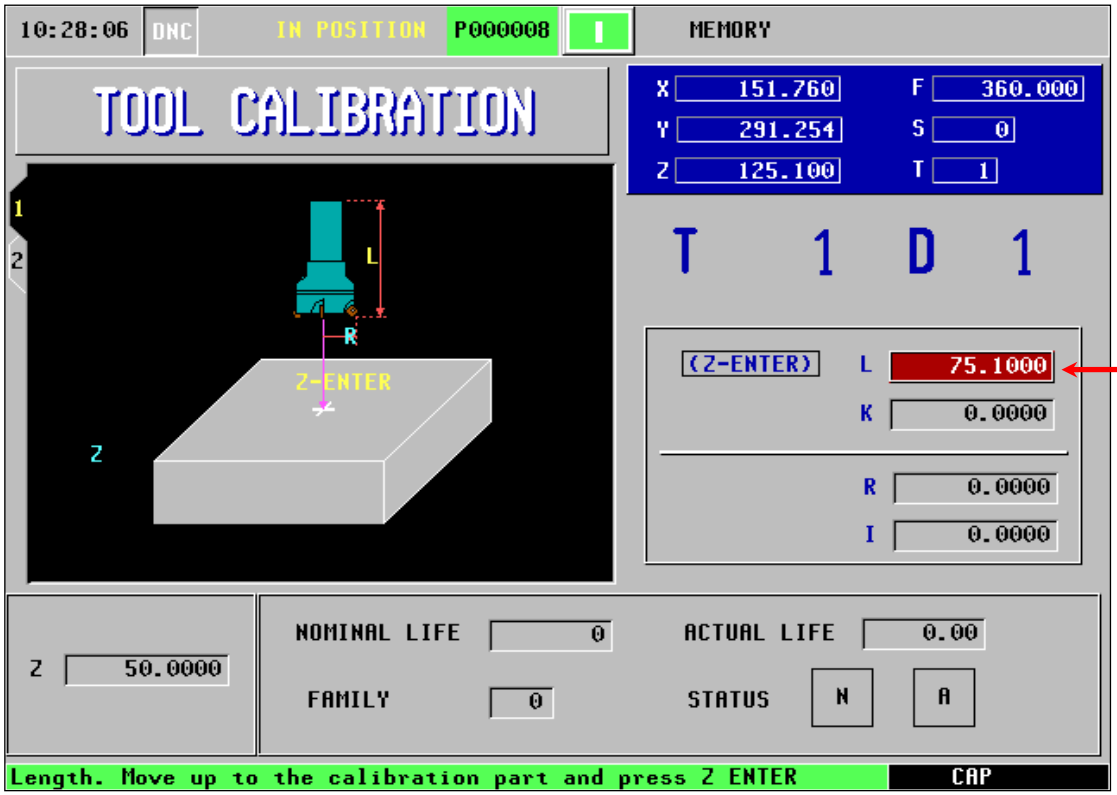
length offset K



Tool calibration cycle

The CNC8055 MC makes it easy to calibrate tools directly on the machine

Call cycle with the F1 key 



All axes must be homed

How does it work?

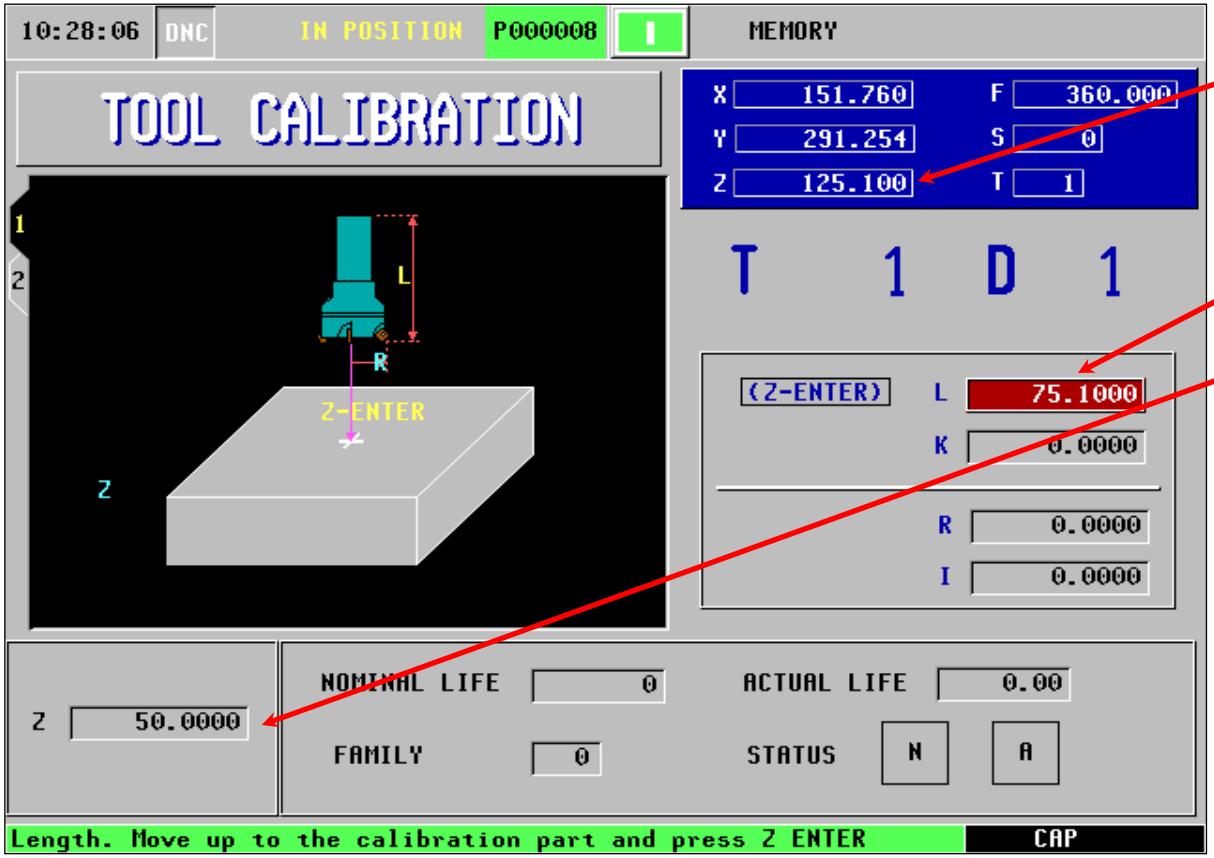
Touch a part with known dimension in Z.
Press the [Z] and [Enter] keys.

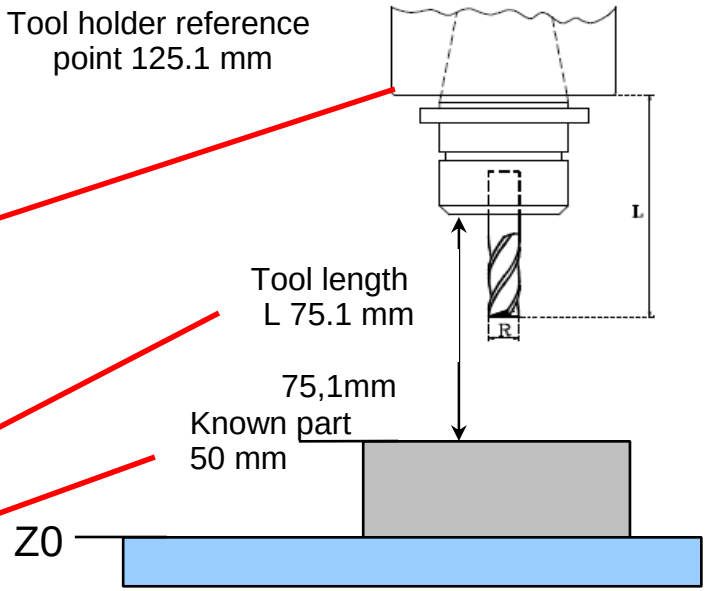
The CNC calculates the length of the tool and puts the value into the tool offset table.

Select the R field and insert the radius of the tool.
Select the other fields and insert the other values if necessary.

Tool calibration cycle

Example with tool T5





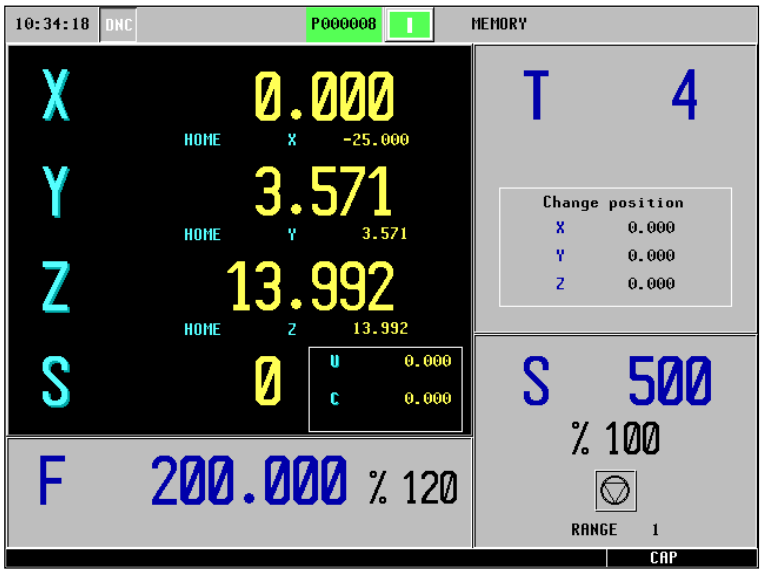
Touch known part 50mm.

Press the [Z] and [Enter] keys.
The CNC calculates tool length.

Insert Radius in field R.

Define tool change position

The machine builder defines the strategies of the tool change.
This feature allows tool change without collision with the part.
It depends on the machine builder.



Tool change position

T X [value]

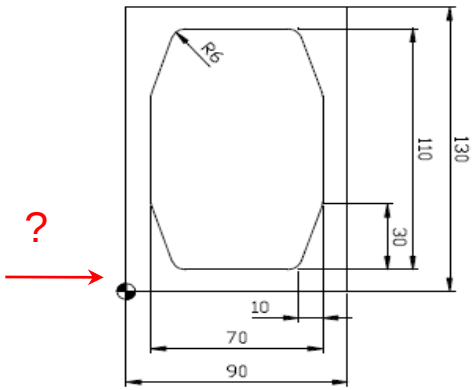
T Y [value]

T Z [value]

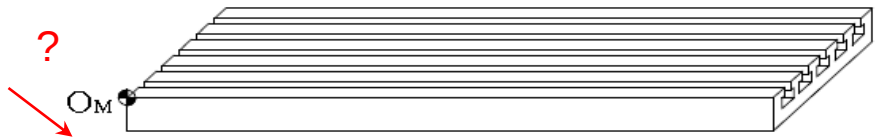
Define part zero

It means that the zero point of the blueprint and the zero point of the part are the same position.
It is easy to create a CNC program, because the values may be entered direclty from the blueprint into the program.

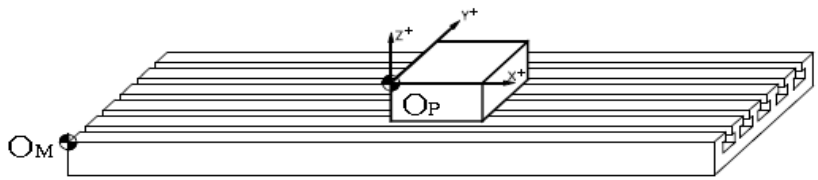
blueprint



machine with 0_M



machine with part



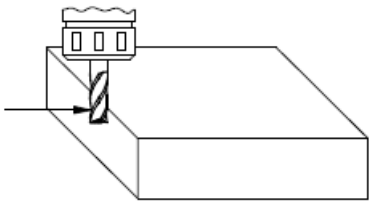
How can one connect the blueprint zero and the part zero?

Solution: preset the coordinates

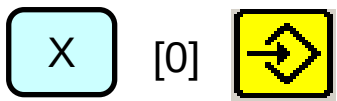
Setting part zero – different methods

Depending on how you use the machine you have different methods to set the part zero

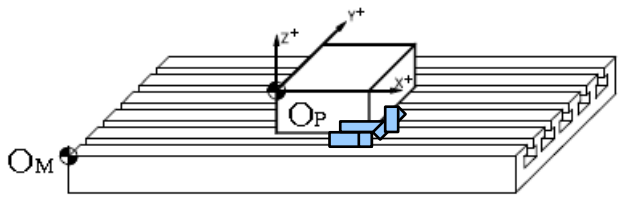
Single part production



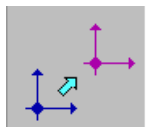
Set the part on the machine
Start the spindle
Touch the part with the tool
set part zero at this point



Series production

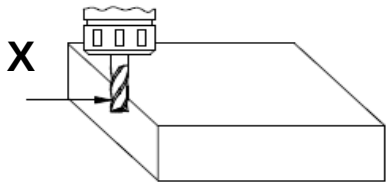


Install the part holder
Measure the fixture
write values in zero offset table
Call part zero in the program

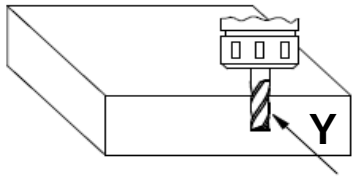


Define part zero through touching

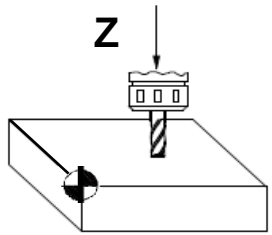
Touch the part in all 3 axes to set the zero points



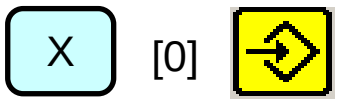
Touching on X

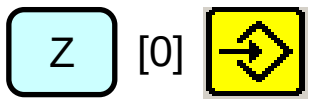


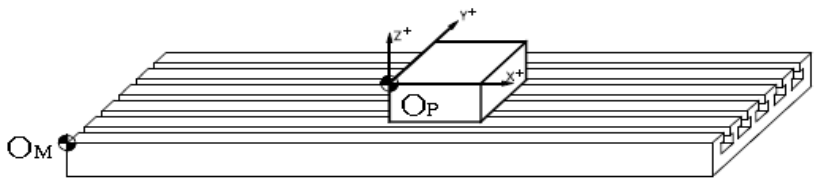
Touching on Y



Touching on Z







Now you have set the part zero on X0, Y0, Z0

General description Normal keys and function keys

SHIFT + RESET — System start-up

SHIFT + ESC — Toggle MC < - > M mode

SHIFT + CLEAR — Screen saver

+ — Homing all axes

X + + — Home X axis

Y + + — Homing axis Y

Z + + — Home Z axis

+ — Zero offset table

Change displayed symbols

LEVEL CYCLE — Select cycle level

PROG — Program management

GRAPHICS — Simulation mode

RECALL — Recall system data

ENTER — Enter key

Cycles

Call the cycles with the function keys


Boring.


Reaming.


Tapping.


Drilling and center punching.

Profile milling.

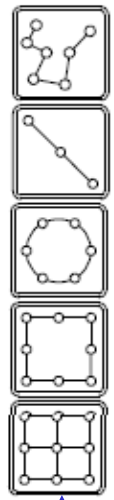
Surface milling and slot milling.

Pocket with profile (2D and 3D).

Rectangular and circular boss.

Rectangular and circular pocket.


Positioning.



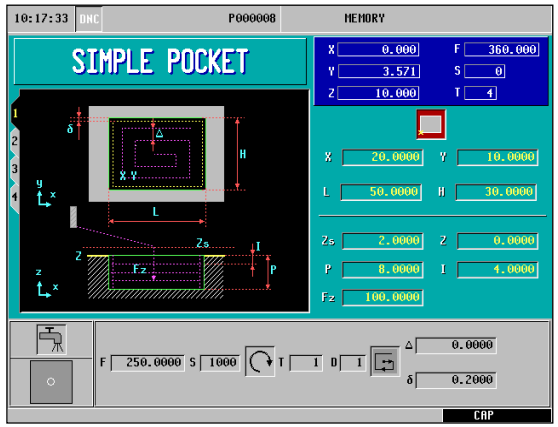
Call multiple execution of cycles

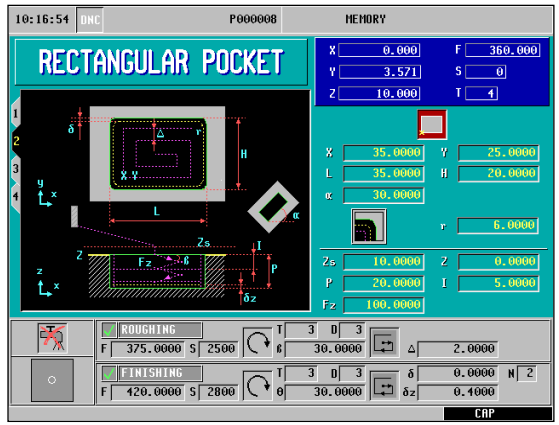
Cycles

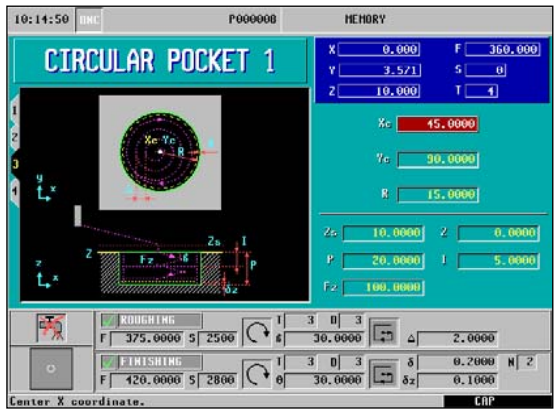
Call the different cycles with function keys

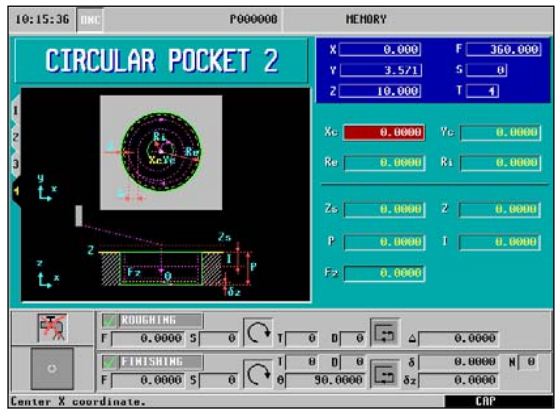
Example pocket cycles

Select different levels with these keys

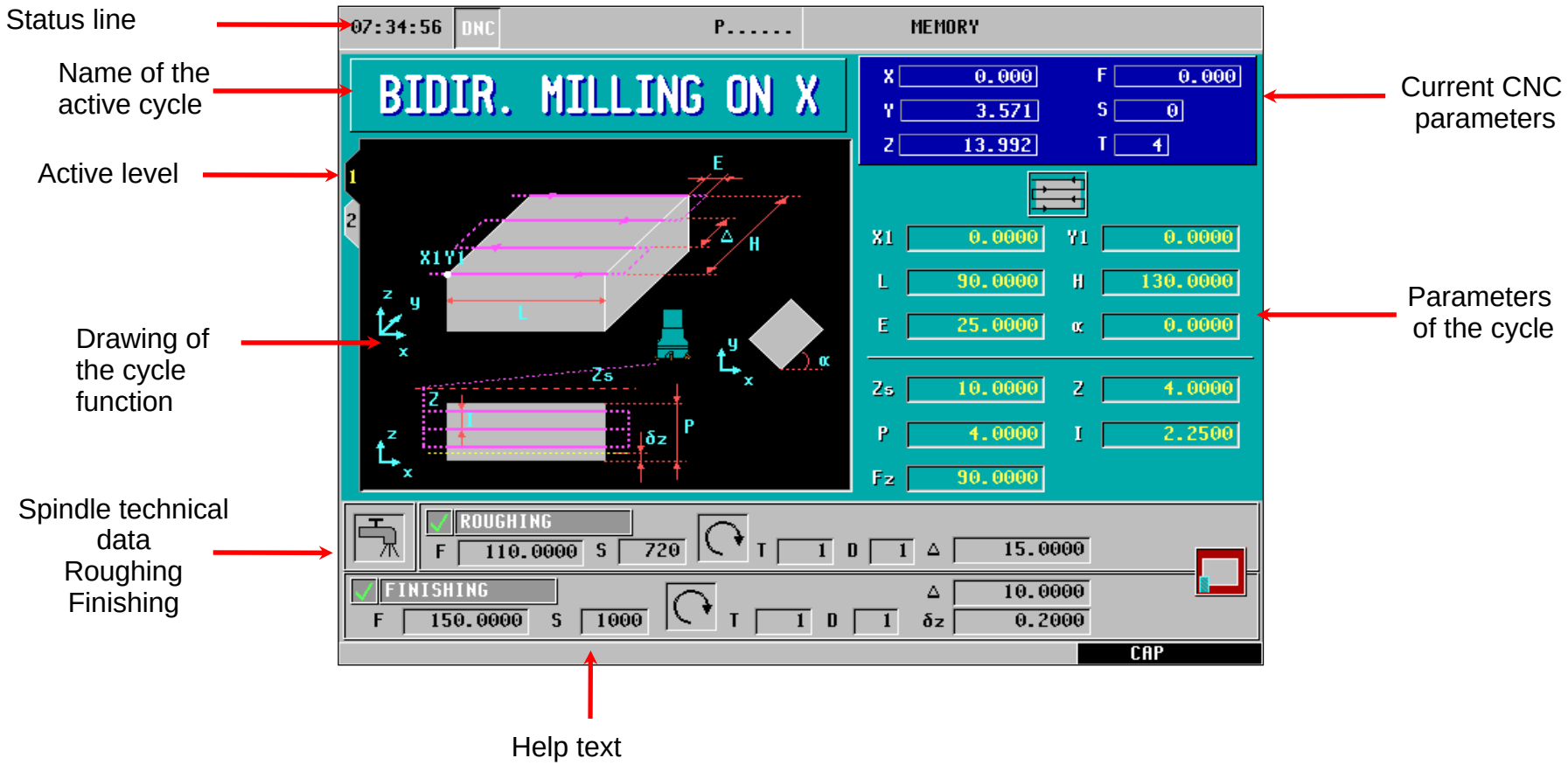








Basic functionality of the cycles

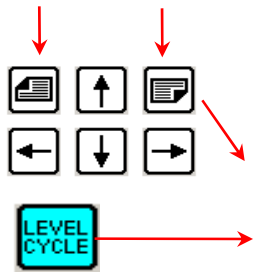


Basic functionality of the cycles

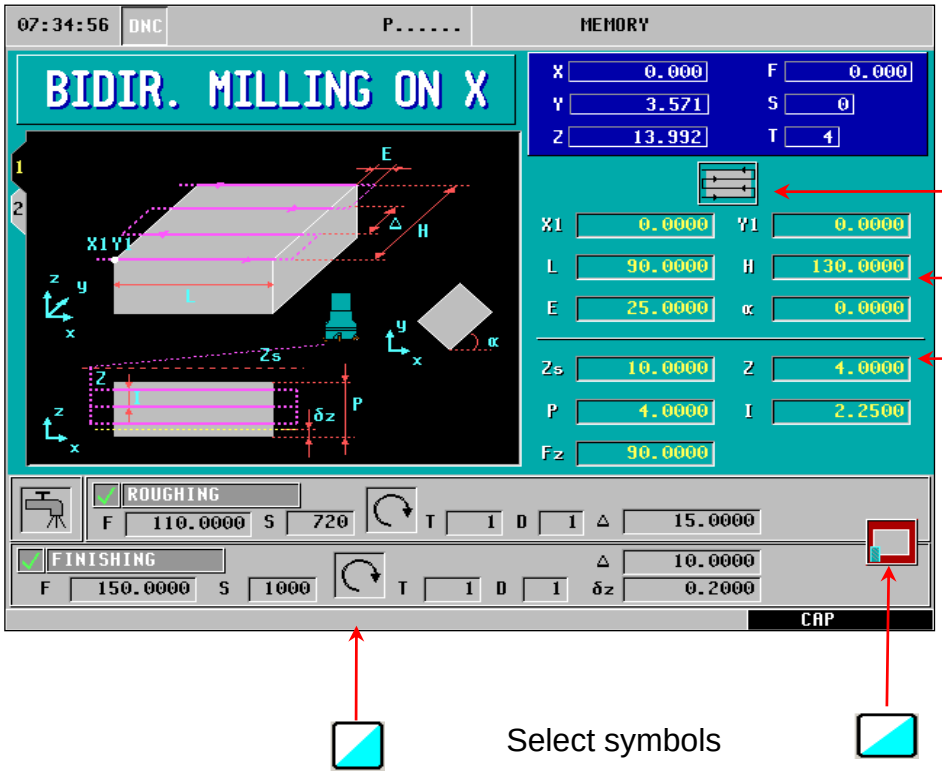
Function keys

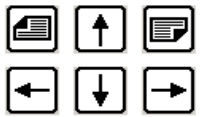
F4 Call cycle with function keys

Select level
Back forth



Select
next level





Cursor
control

Select symbols

ENTER key

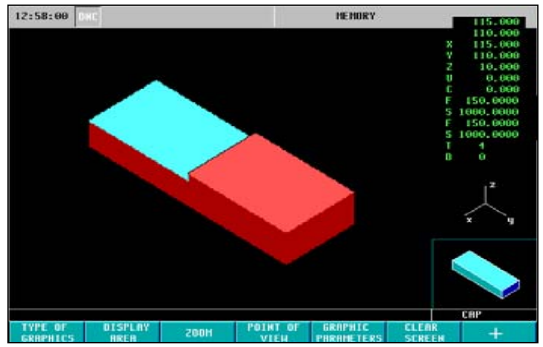
Restore CNC values

Escape

Select symbols

Execution modes of the cycles

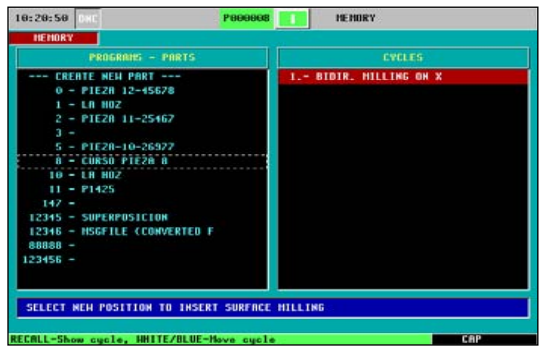
Simulation



Edit mode

Operation

Programming

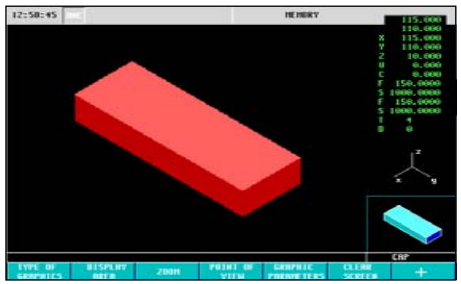


Start machine

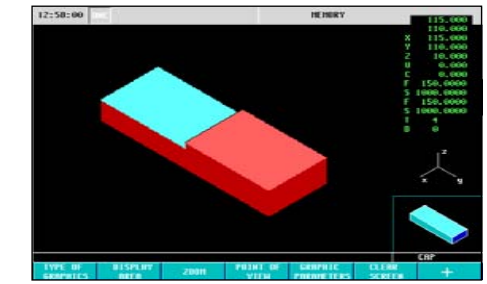
Execution modes of the cycles

Simulation

Edit cycle



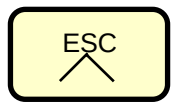
Call simulation



start simulation

A cycle can be simulated independently

- Edit cycle
- Press GRAPHICS key – call simulation
- Press START key – start simulation
- Press ESC key – back to edit mode

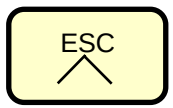


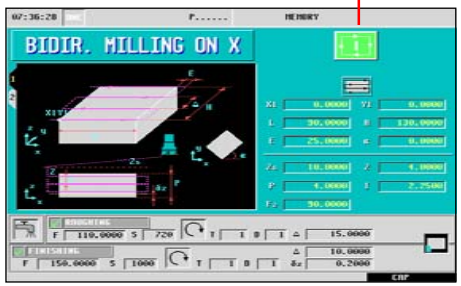
back to edit mode

Execution modes of the cycles

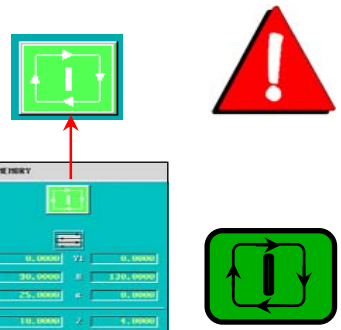
Operating mode

Edit cycle

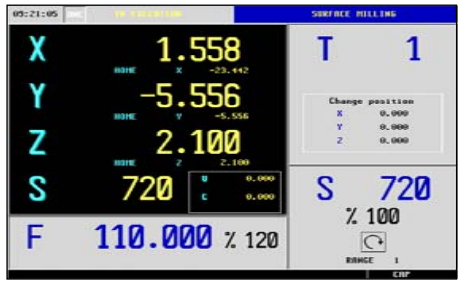




Activate operation



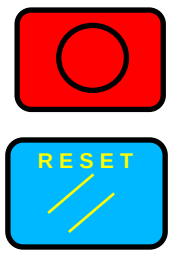
Start machine

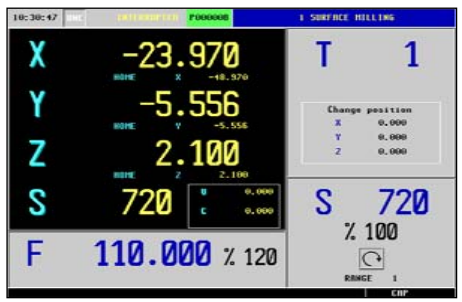


start operation

A cycle can be executed independently in operation mode

- Edit cycle
- Press ESC key – activate operation
- Press START key – operation starts
- Press STOP key – operation stops
- Press RESET key – resets operation



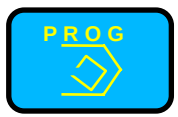


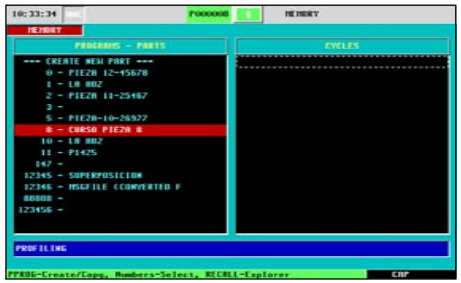
Stops operation

Execution modes of the cycles

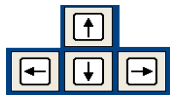
Programming mode

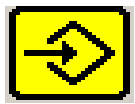
Edit cycle

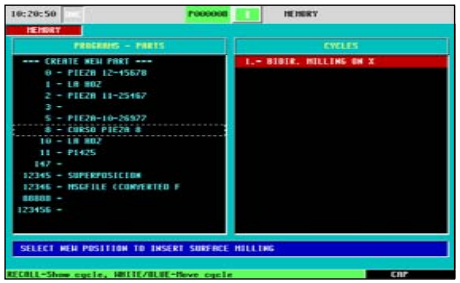




Call program manager



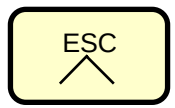




Save cycle

Store cycles in a cycle program

- Edit cycle
- Press P.PROG key – call program manager
- Cursor keys – select insert position
- Press ENTER key- save cycle in the program
- Press ESC – back to edit mode

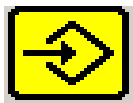


back to edit mode

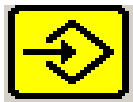
Open a new program

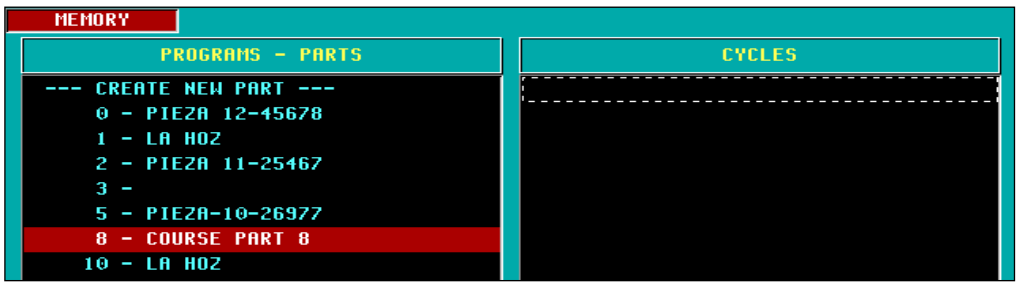
Cursor at
--- CREATE NEW PART---

8



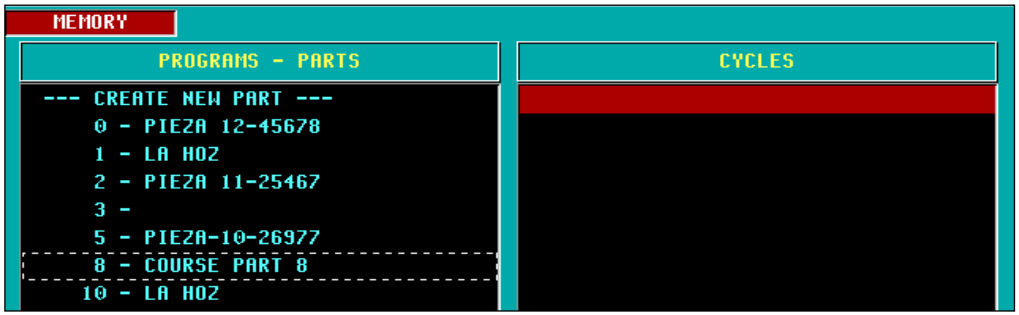
PART 8 COURSE



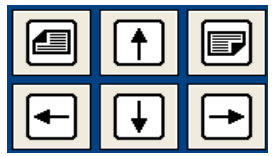


New, empty program created

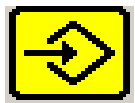
Save the last cycle edited



Place cursor at CYCLES column



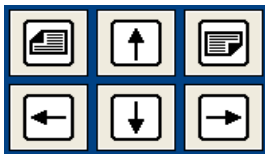
Insert last cycle into the program



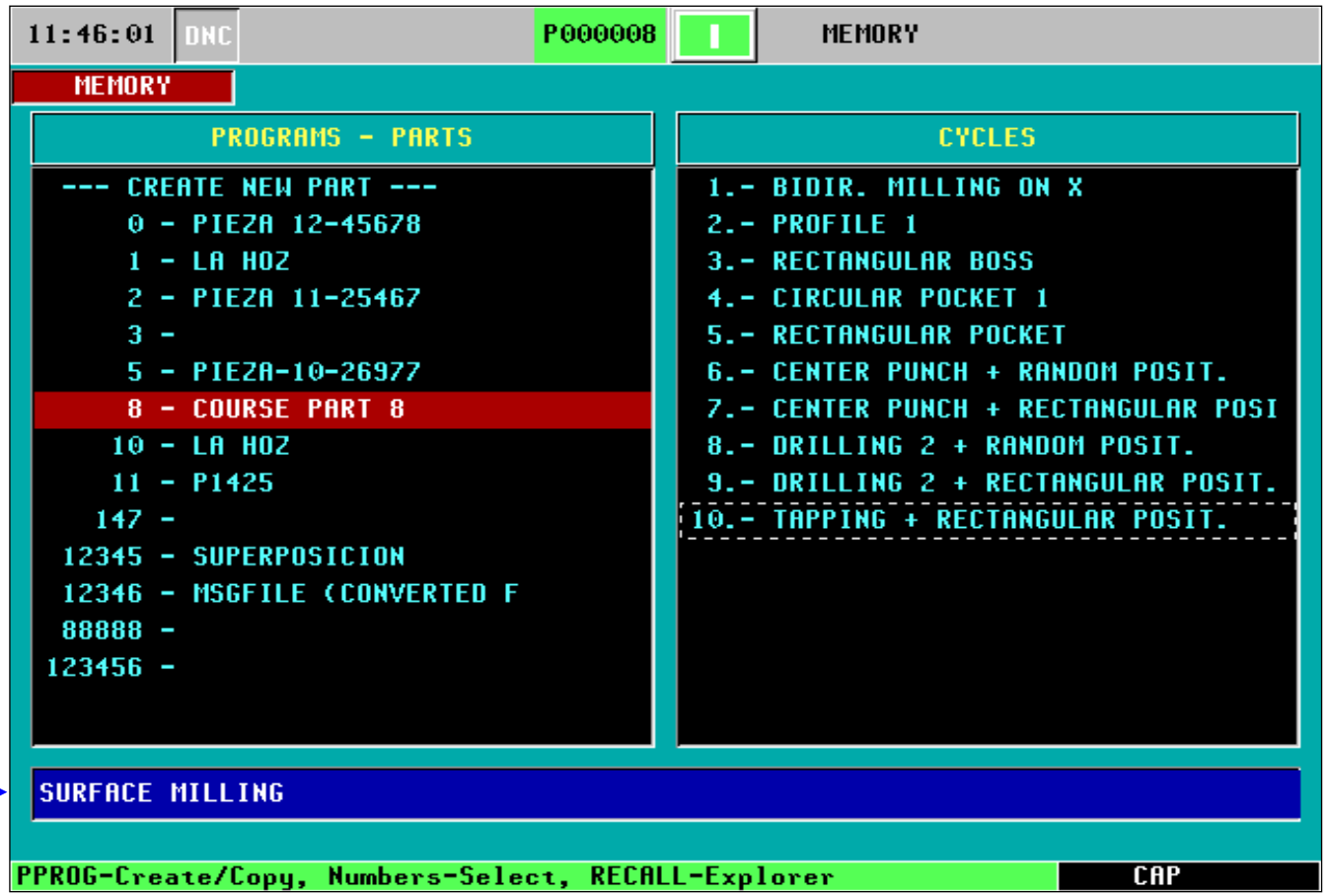
Recall the selected cycle

Program manager

Cursor control keys



Help and info line

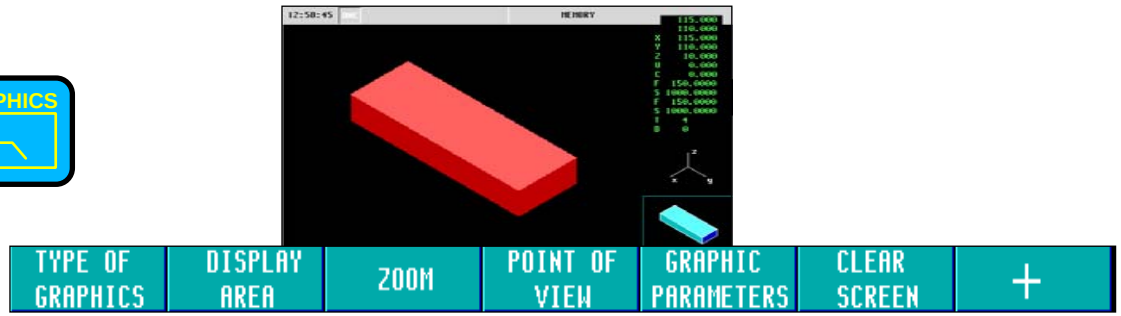


Graphic simulation

Edit cycle

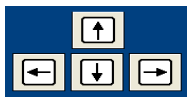
Call graphic simulation

Control via softkeys



Change display area:

Select field with

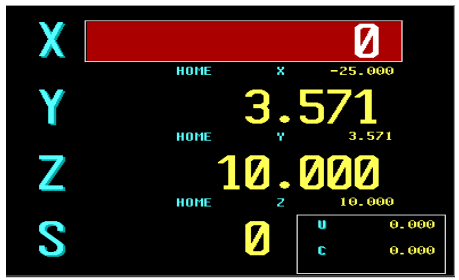


Put in values
confirm with

at the end

Part zero setting

Touch the part



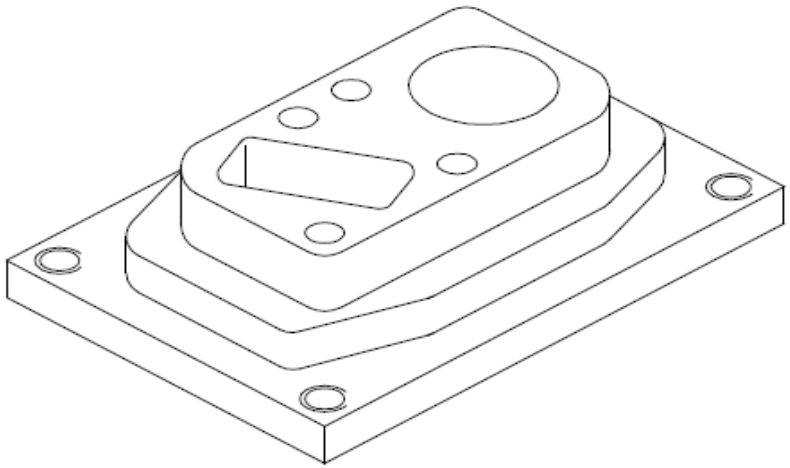
 + Edit part zero table

Part zero preset by cycle

Fagor high level language
With MDI or ISO

```
(ORGX54=0,ORGZ54=-64)
G54
```

Programming example



Tool listing

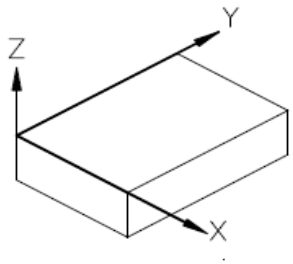
T1: End mill Ø40. T2: End mill Ø25.

T3: End mill Ø10. T4: Center punching drill bit.

T5: Drill bit Ø8. T6: Drill bit Ø5.

T7: Tap M-6.

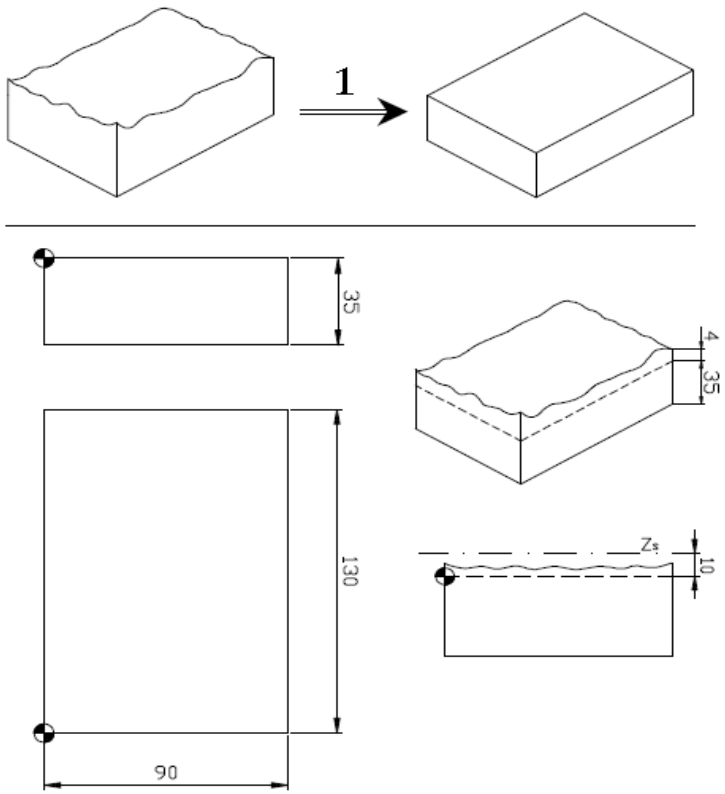
Coordinate system



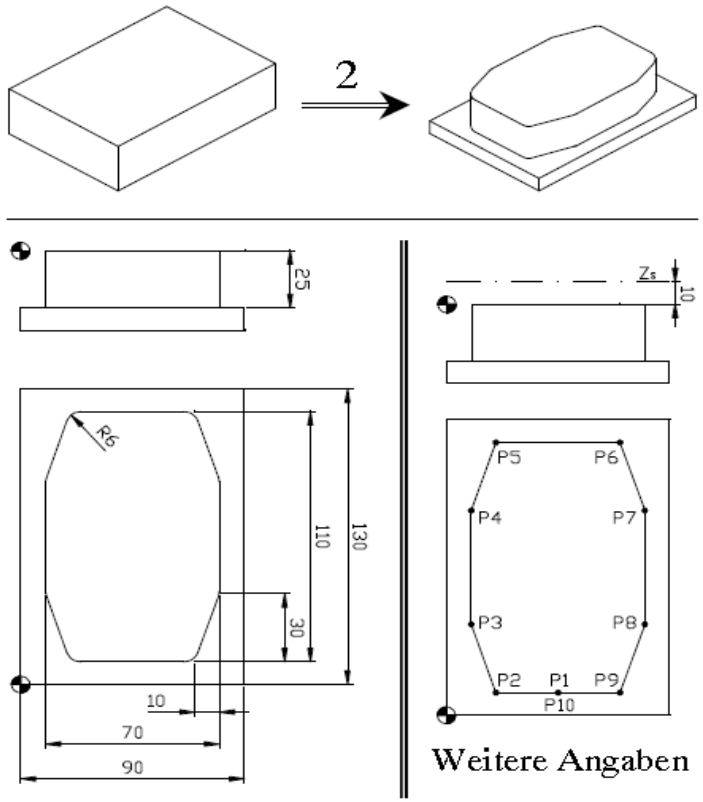
Part zero icon

Programming example step by step

Step 1 surface milling

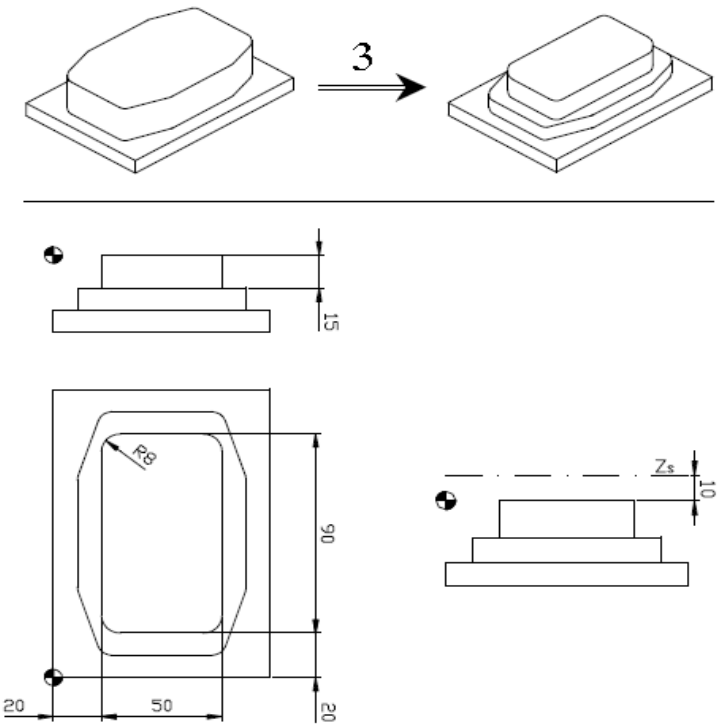


Step 2 profile milling

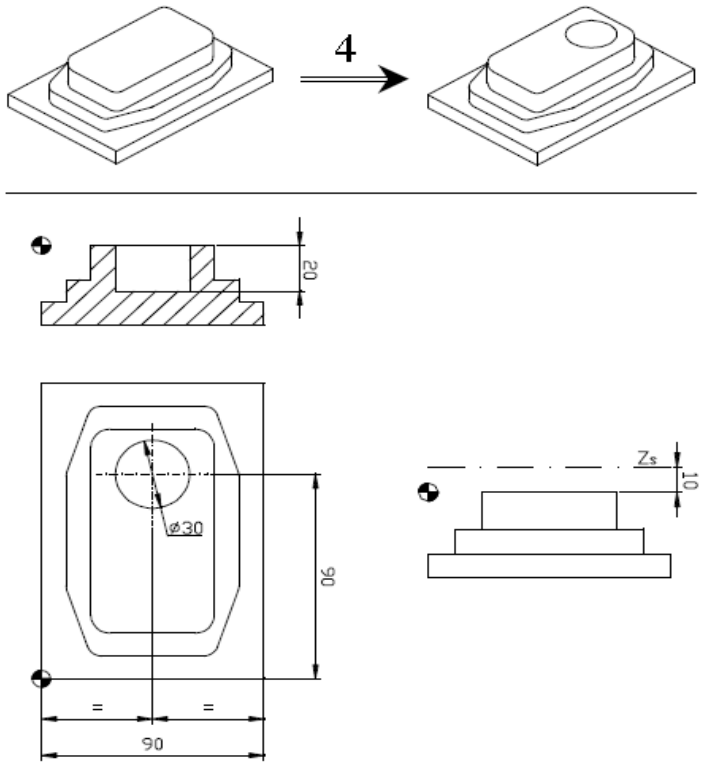


Programming example step by step

Step 3 rectangular boss

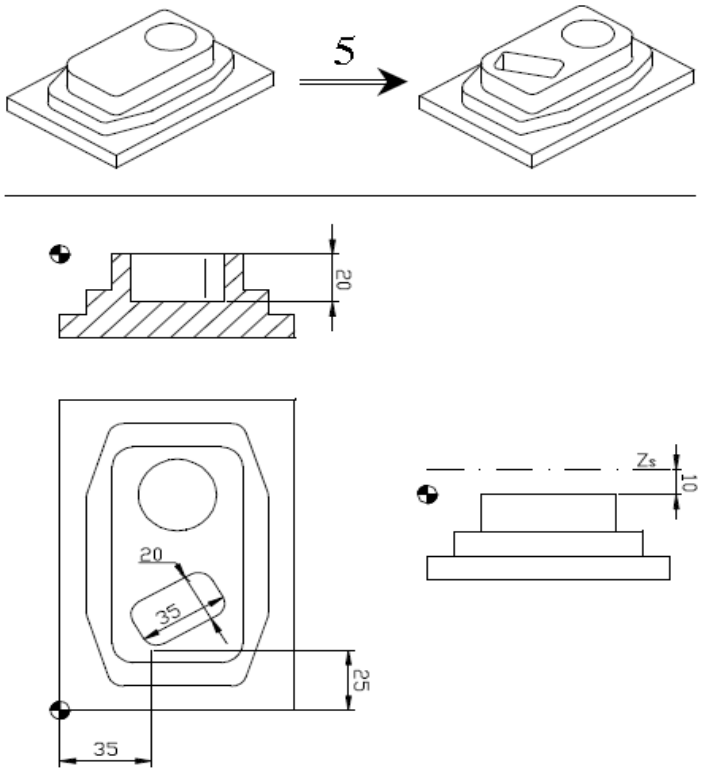


Step 4 circular pocket

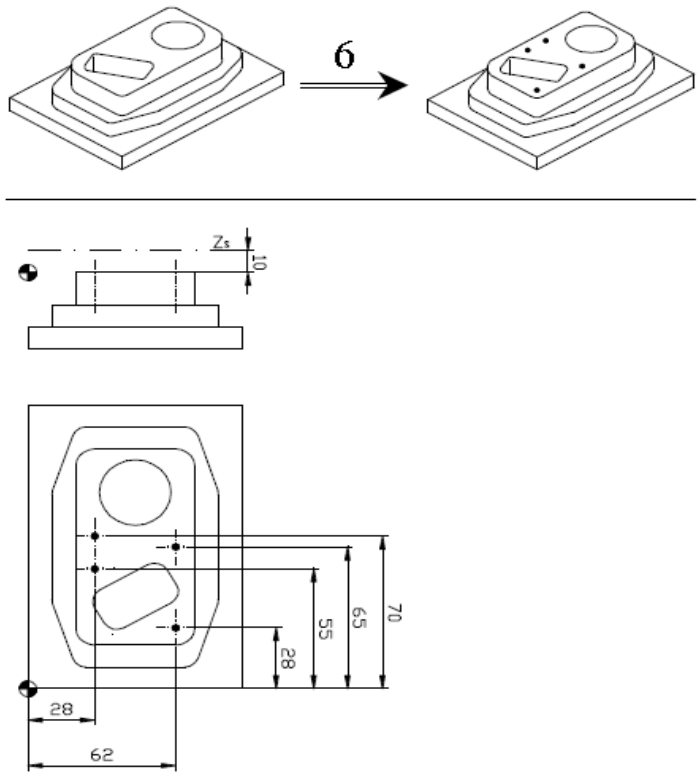


Programming example step by step

Step 5 rectangular pocket

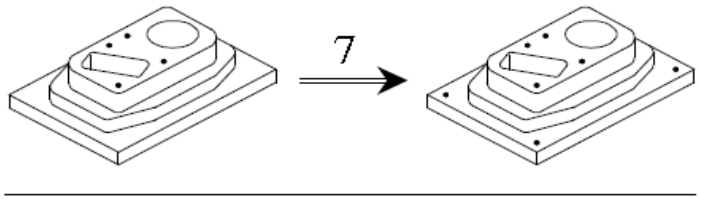


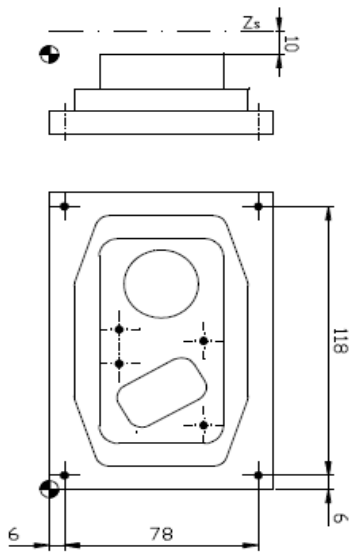
Step 6 center punching + multi-point positioning



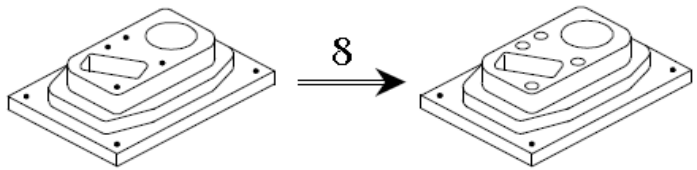
Programming example step by step

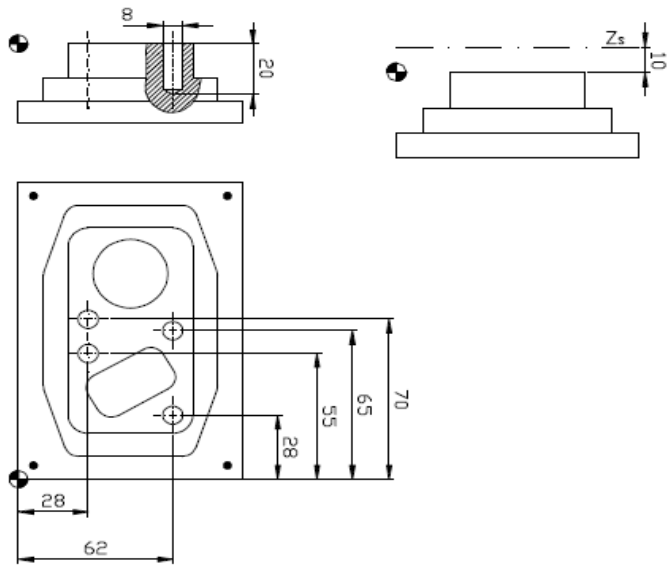
Step 7 center point + rectangular multi-point





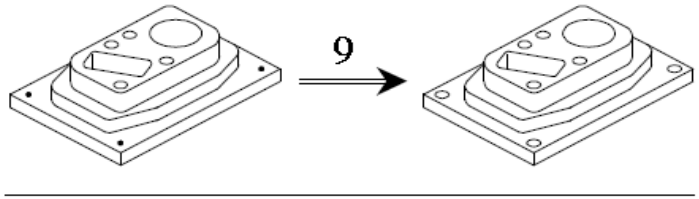
Step 8 drilling + multi-point positioning

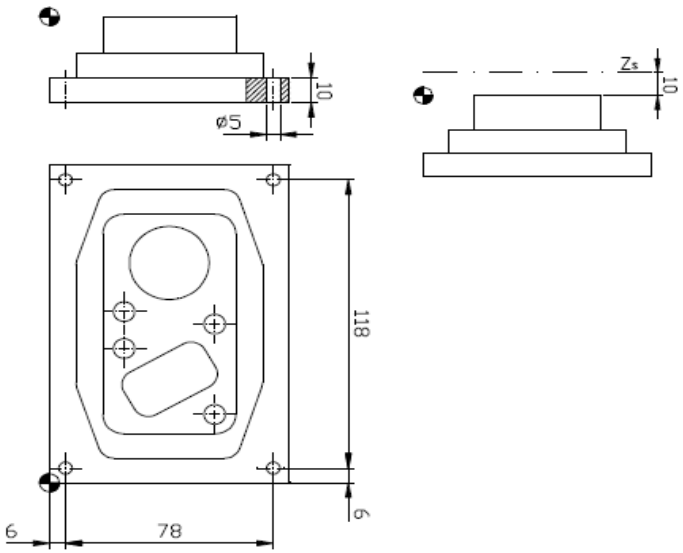




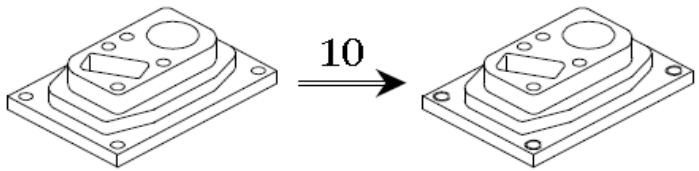
Programming example step by step

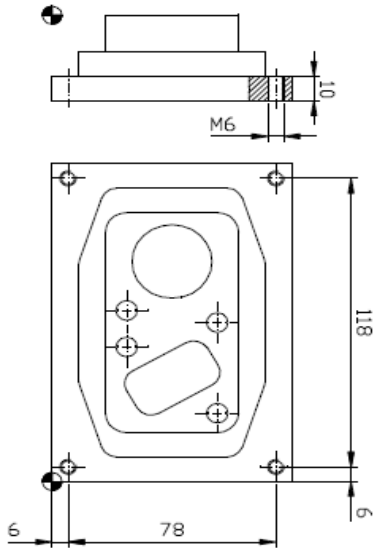
Step 9 drilling + rectangular multi-point





Step 10 tapping + rectangular multi-point



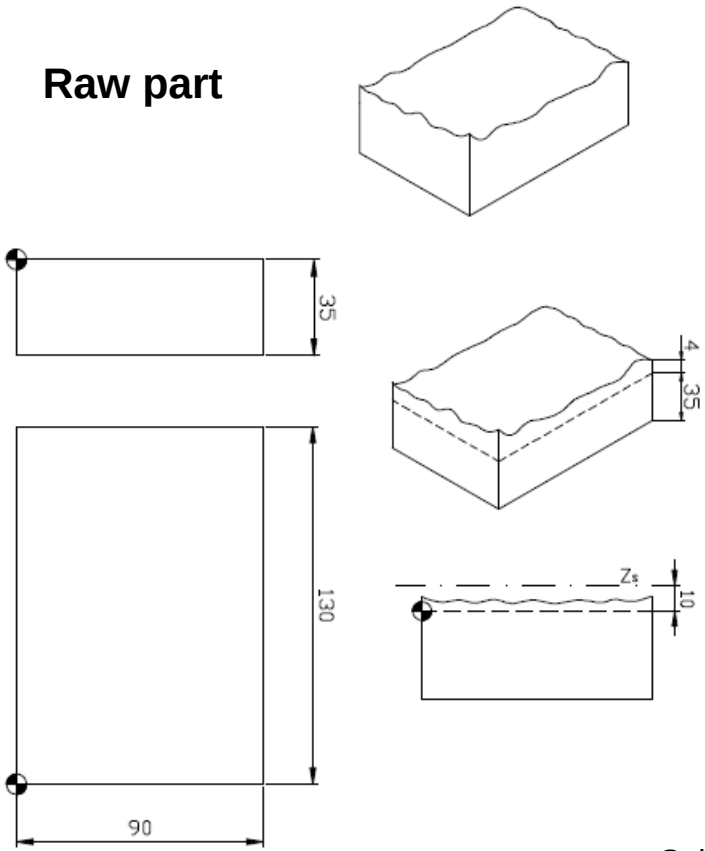


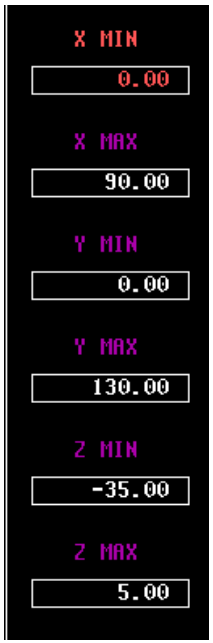
Programming example

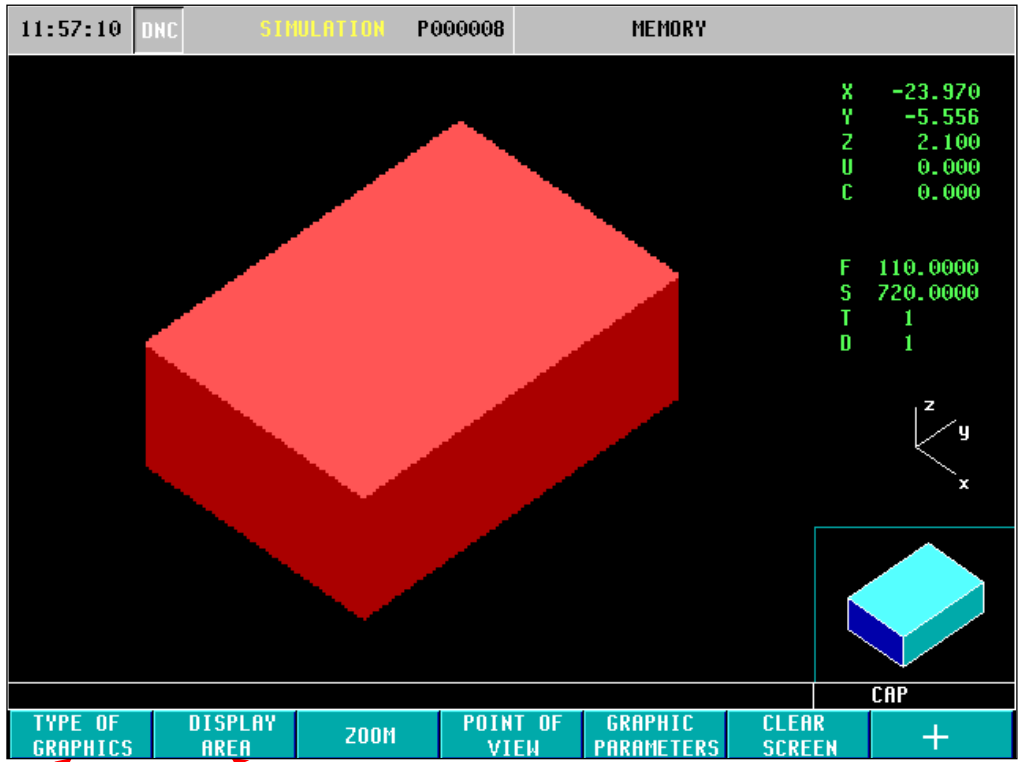
Adjust display area for simulation

GRAPHICS

Raw part





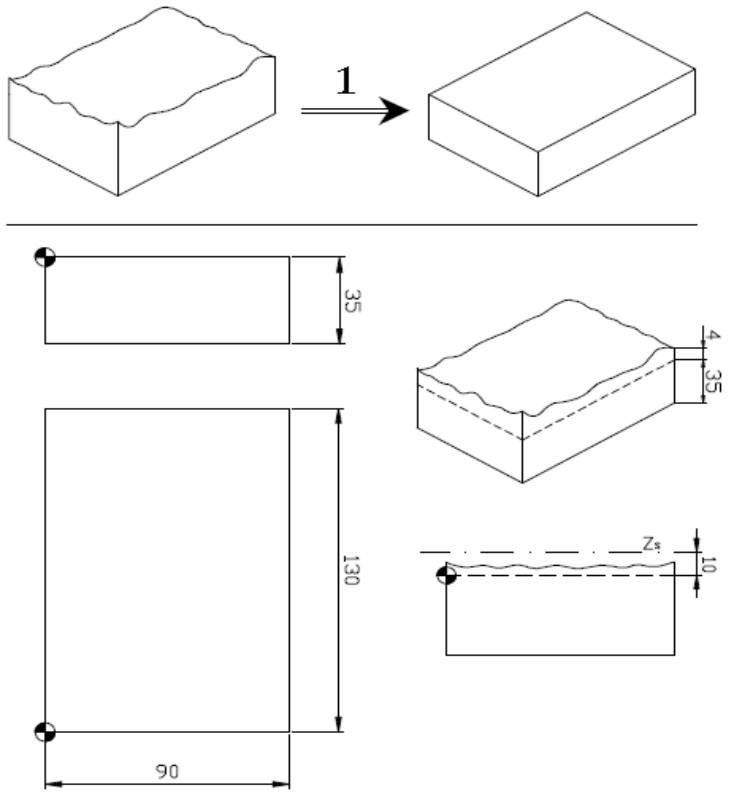


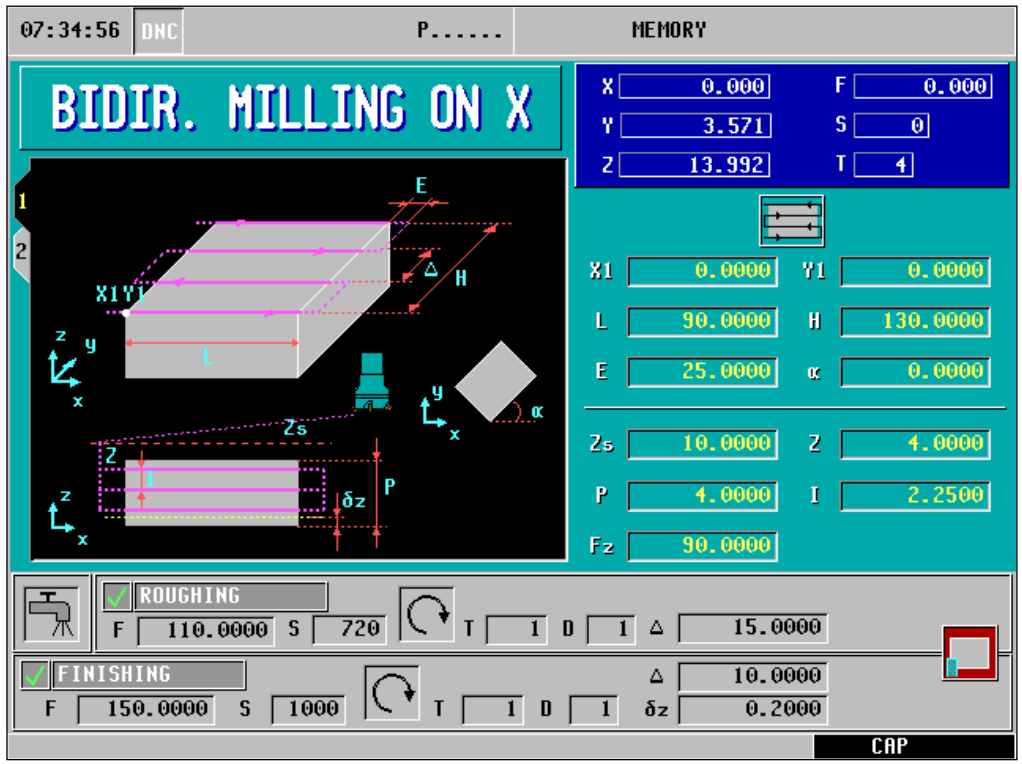
Select graphic type

Adjust display area

Programming example

Step 1 surface milling

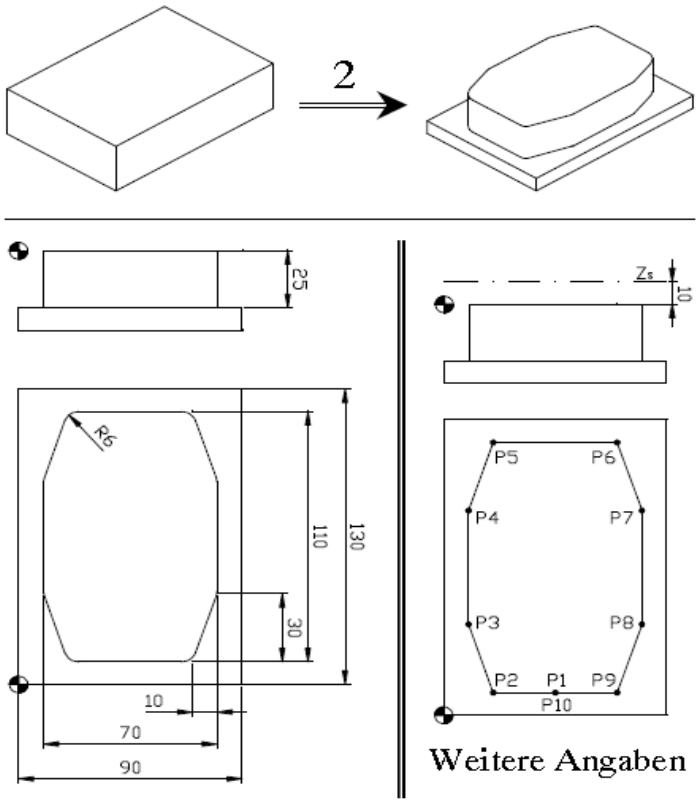


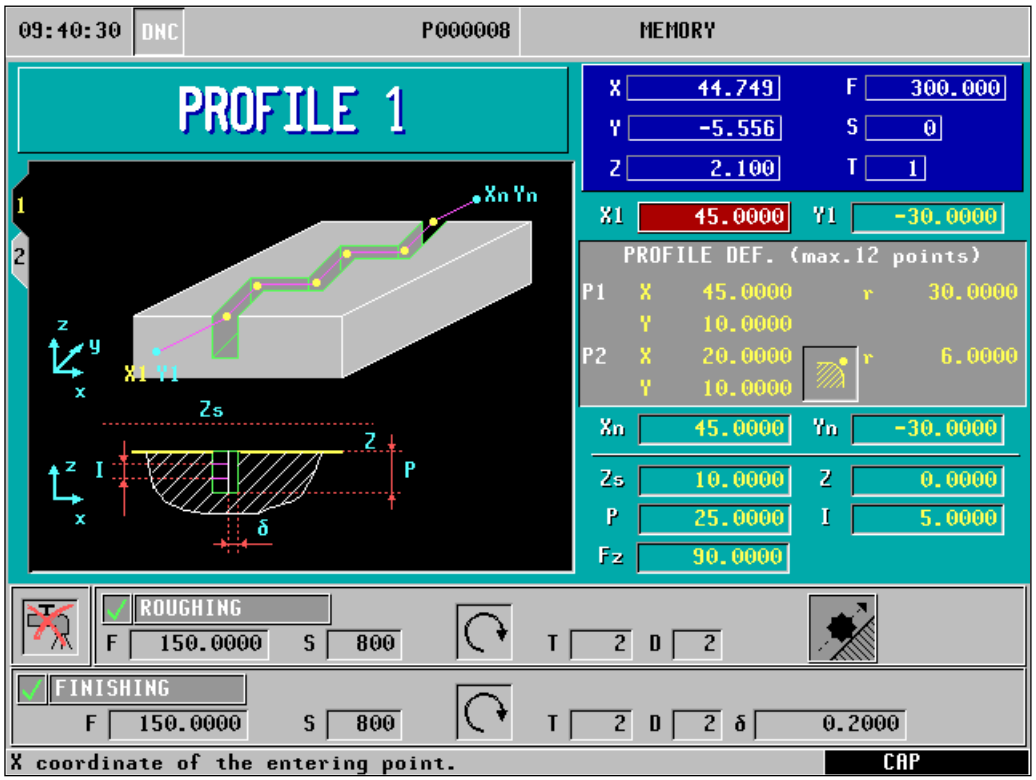


PROG Save cycle

Programming example

Step 2 profile milling

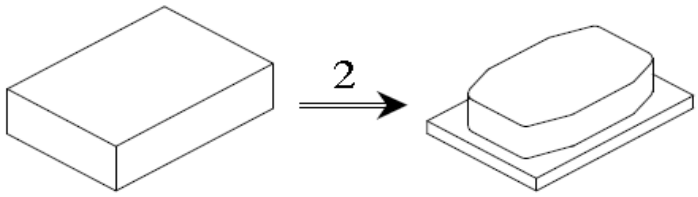


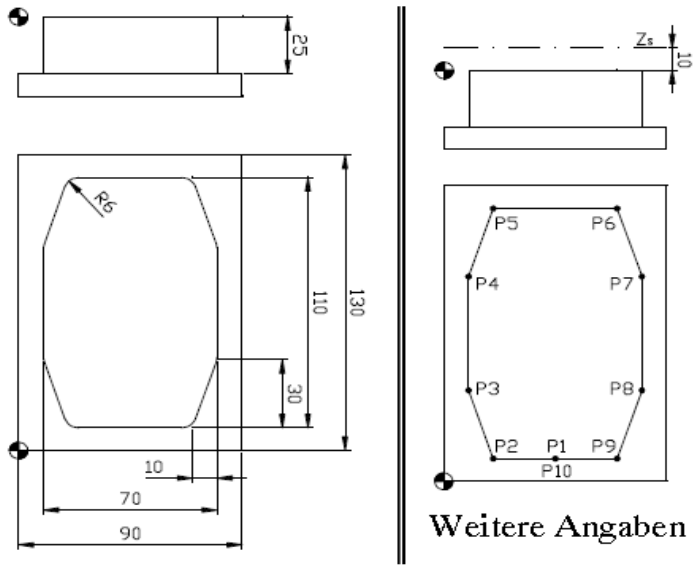


PROG Save cycle

Programming example

Step 2 profile milling

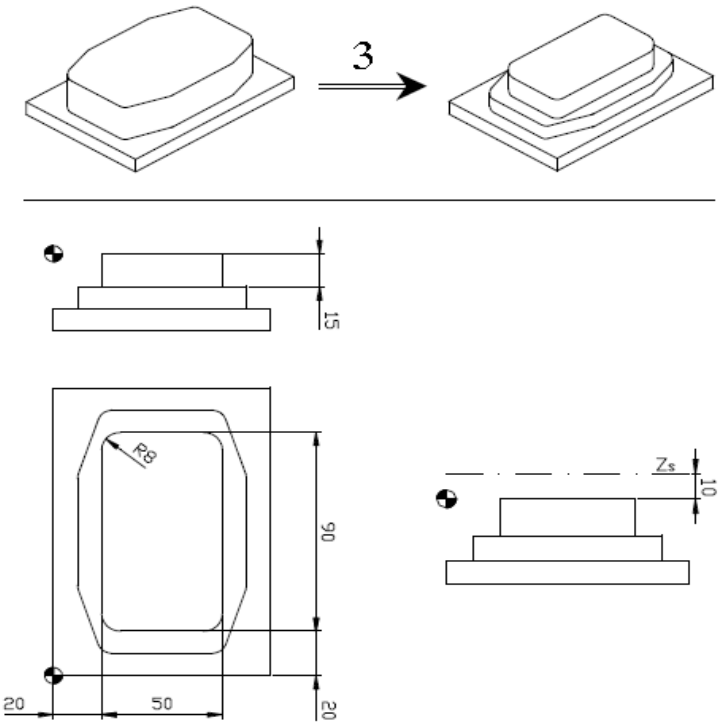


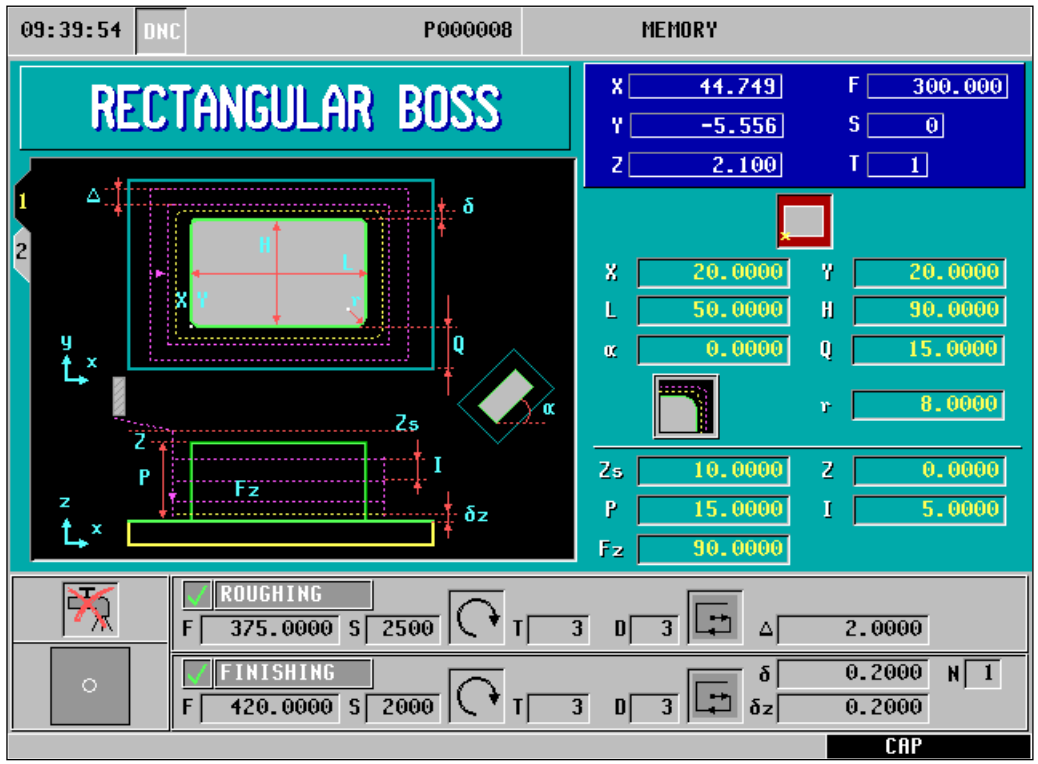


PROFILE DEF. (max.12 points)					
P1	X	45.0000	r	30.0000	Tangential entry
	Y	10.0000			
P2	X	20.0000	r	6.0000	
	Y	10.0000			
P3	X	10.0000	r	6.0000	
	Y	40.0000			
P4	X	10.0000	r	6.0000	
	Y	90.0000			
P5	X	20.0000	r	6.0000	
	Y	120.0000			
P6	X	70.0000	r	6.0000	
	Y	120.0000			
P7	X	80.0000	r	6.0000	
	Y	90.0000			
P8	X	80.0000	r	6.0000	
	Y	40.0000			
P9	X	70.0000	r	6.0000	
	Y	10.0000			
P10	X	45.0000	r	6.0000	Last point ? Repeat point.
	Y	10.0000			
P11	X	45.0000	r	6.0000	
	Y	10.0000			
P12	X	0.0000	r	30.0000	Tangential exit
	Y	0.0000			

Programming example

Step 3 rectangular boss

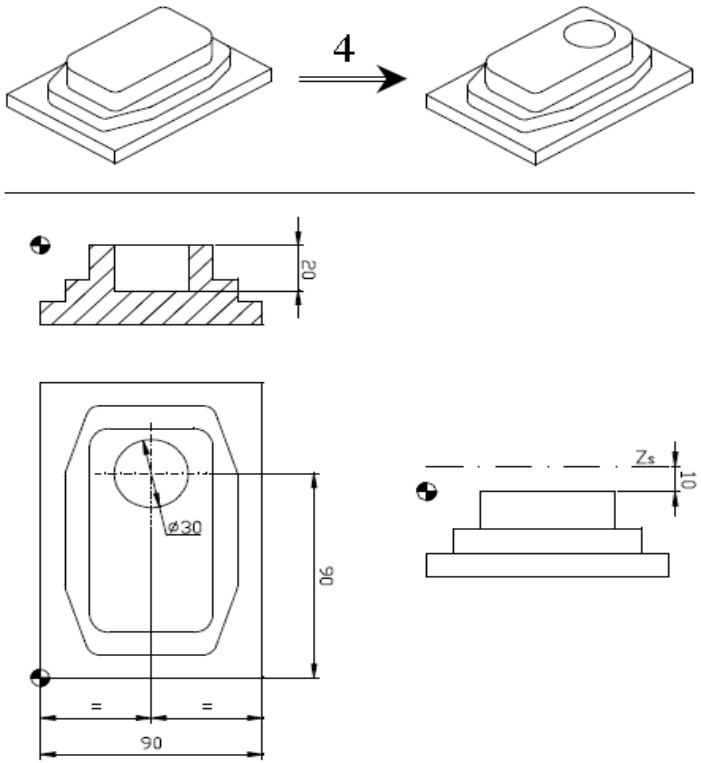


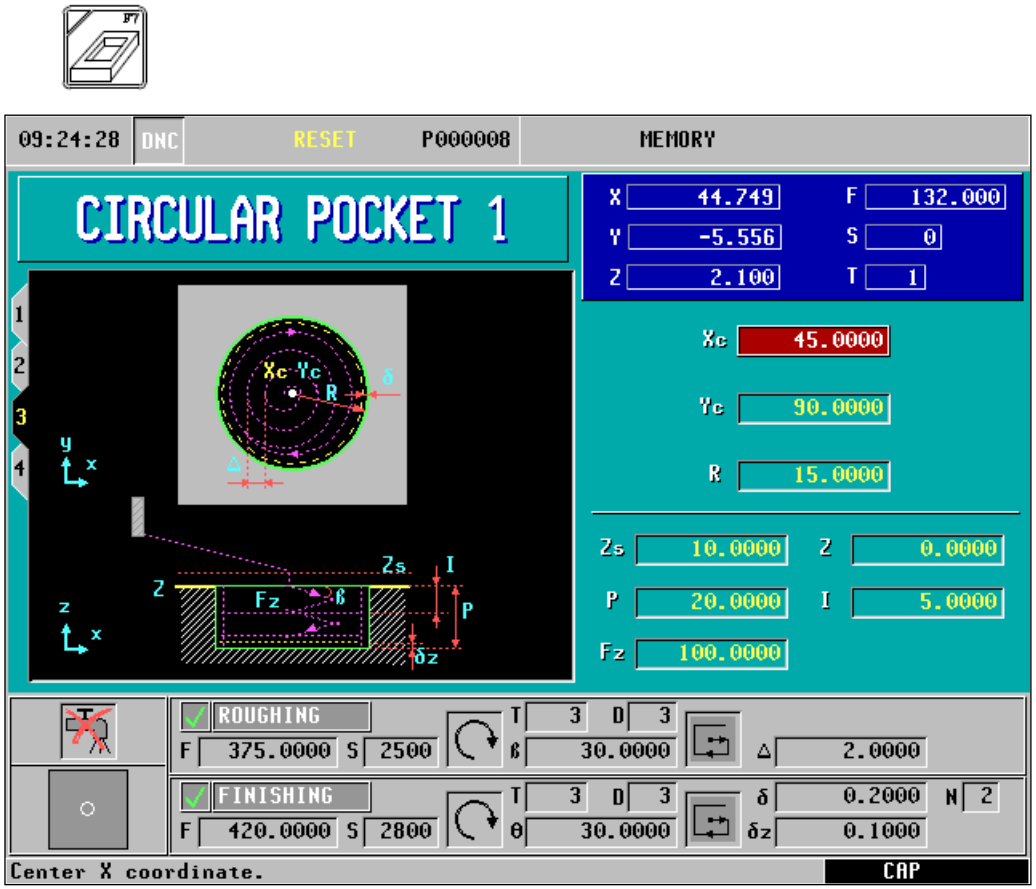


PROG Save cycle

Programming example

Step 4 circular pocket

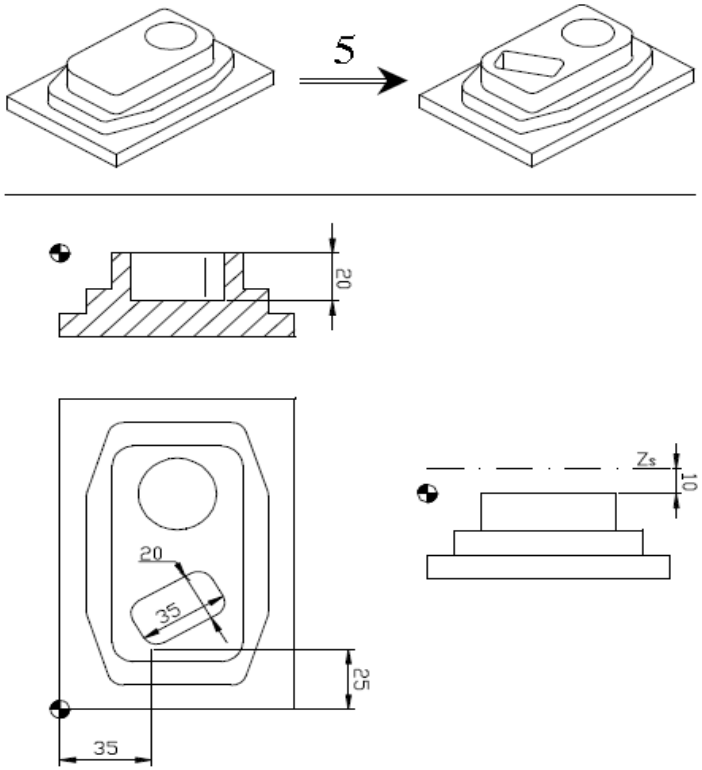


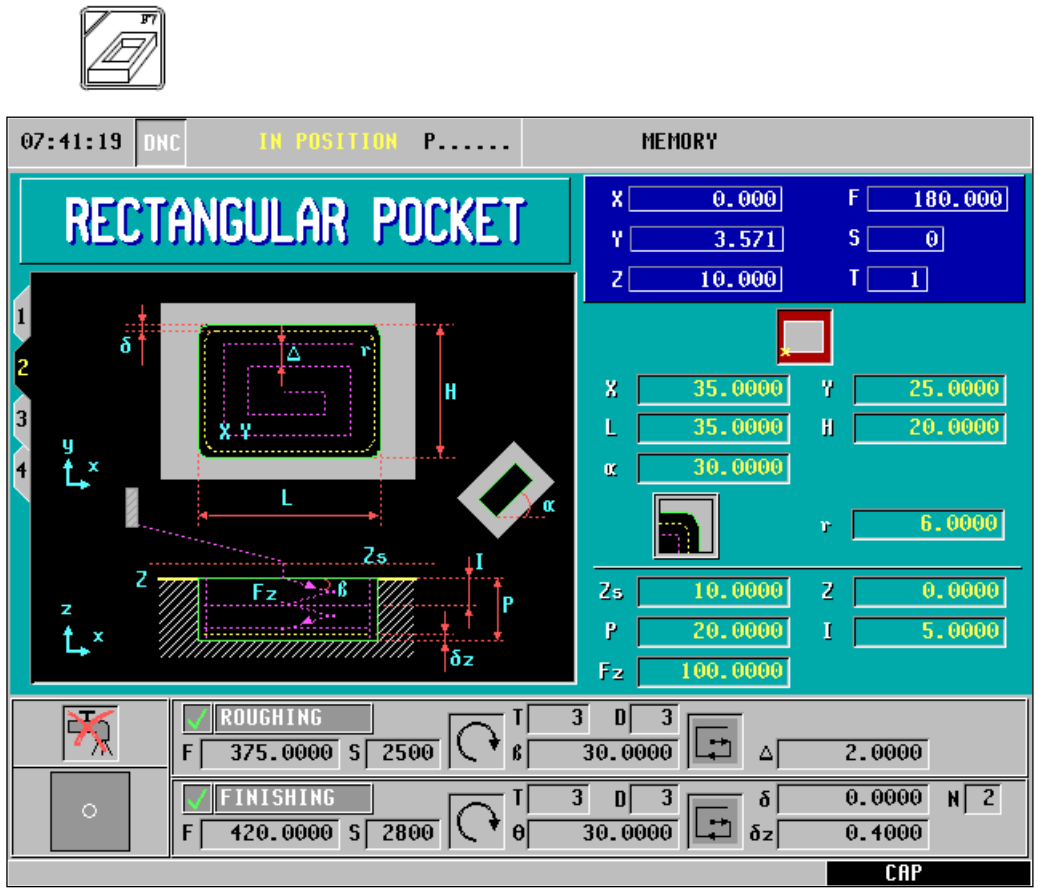


PROG Save cycle

Programming example

Step 5 rectangular pocket

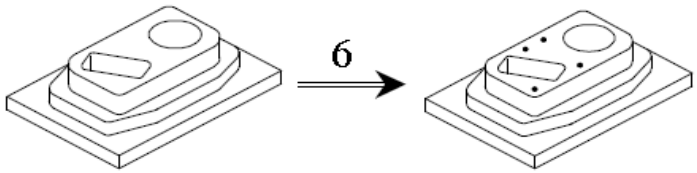


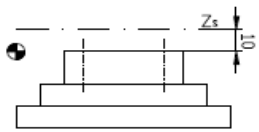


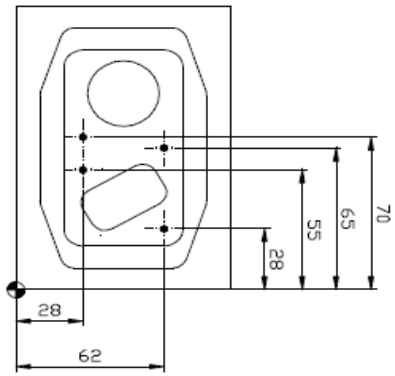
PROG Save cycle

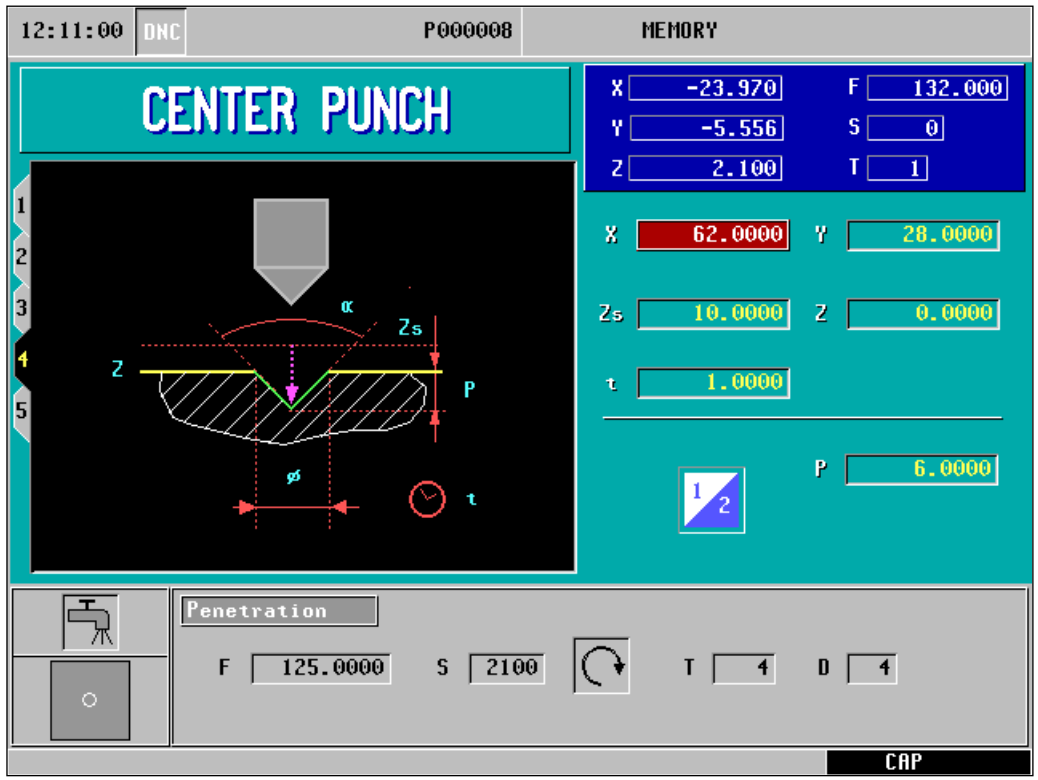
Programming example

Step 6 center punching + multi-point positioning



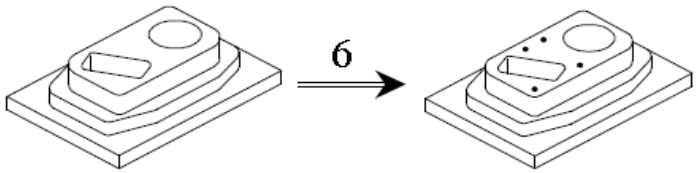


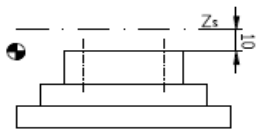


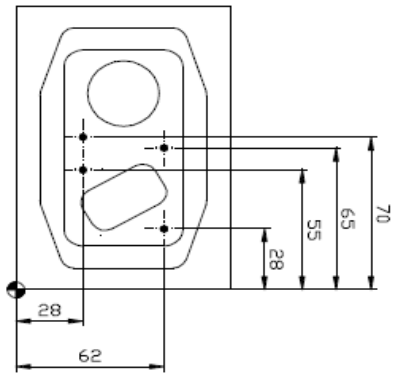


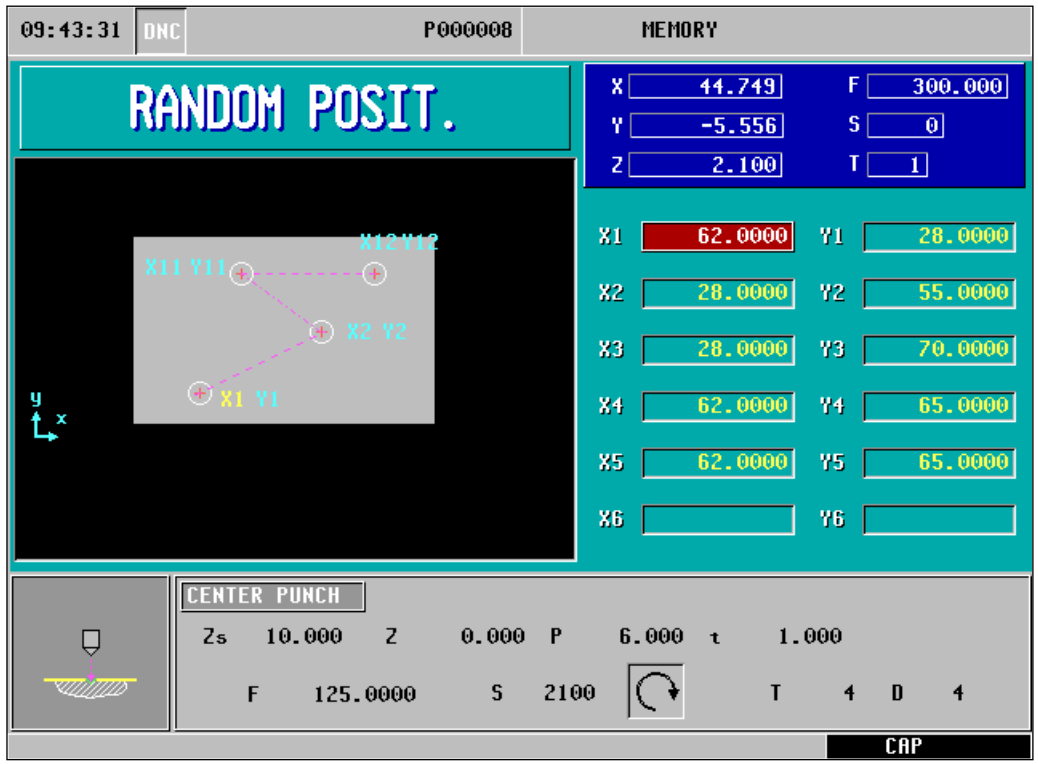
Programming example

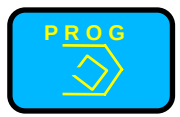
Step 6 center punching + multi-point positioning







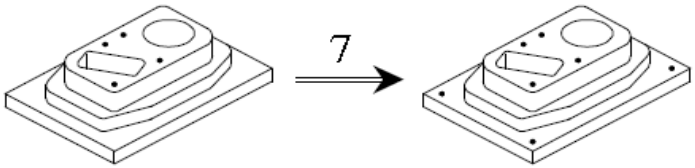


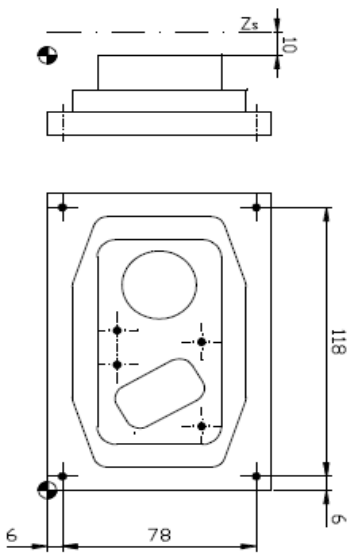


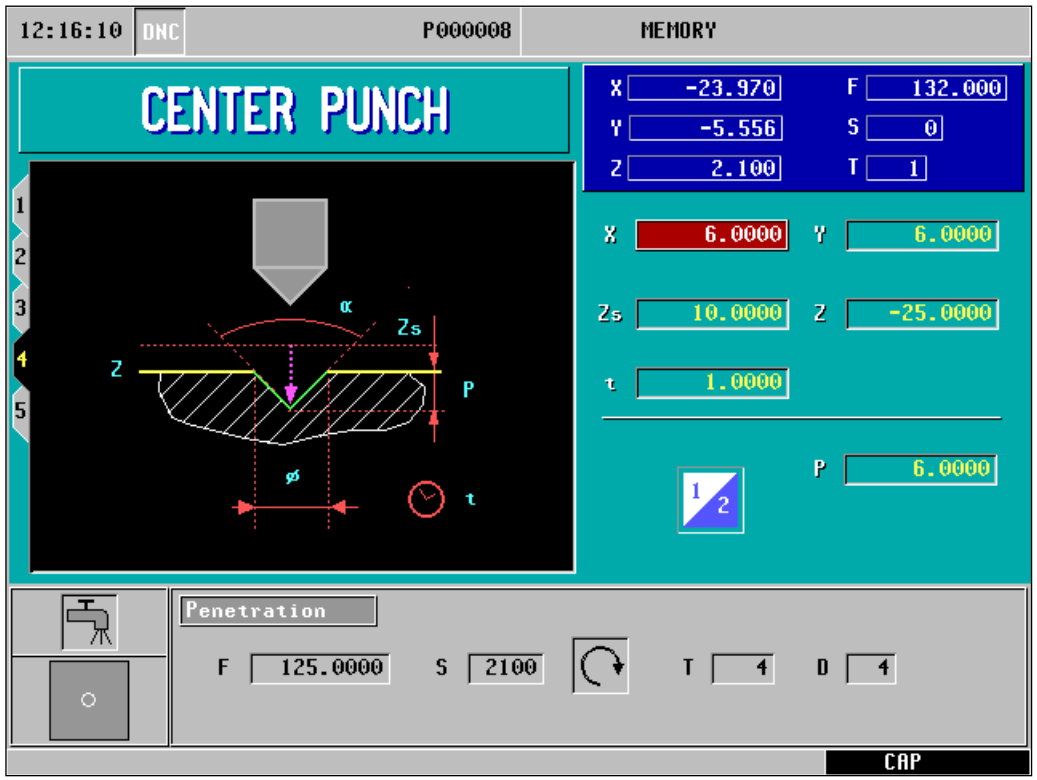
Save cycle

Programming example

Step 7 center point + rectangular multi-point

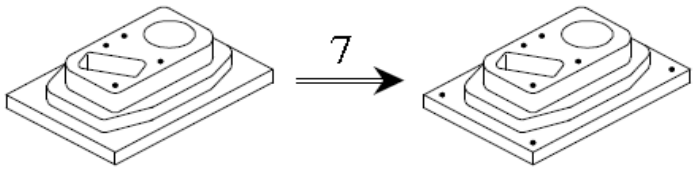


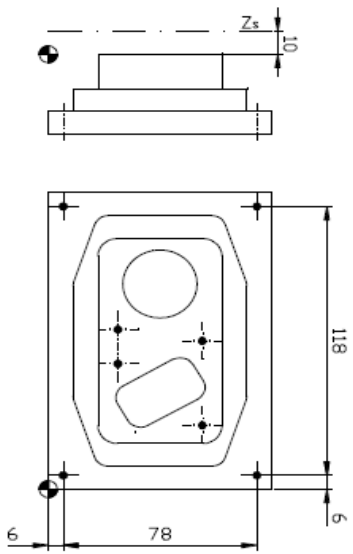


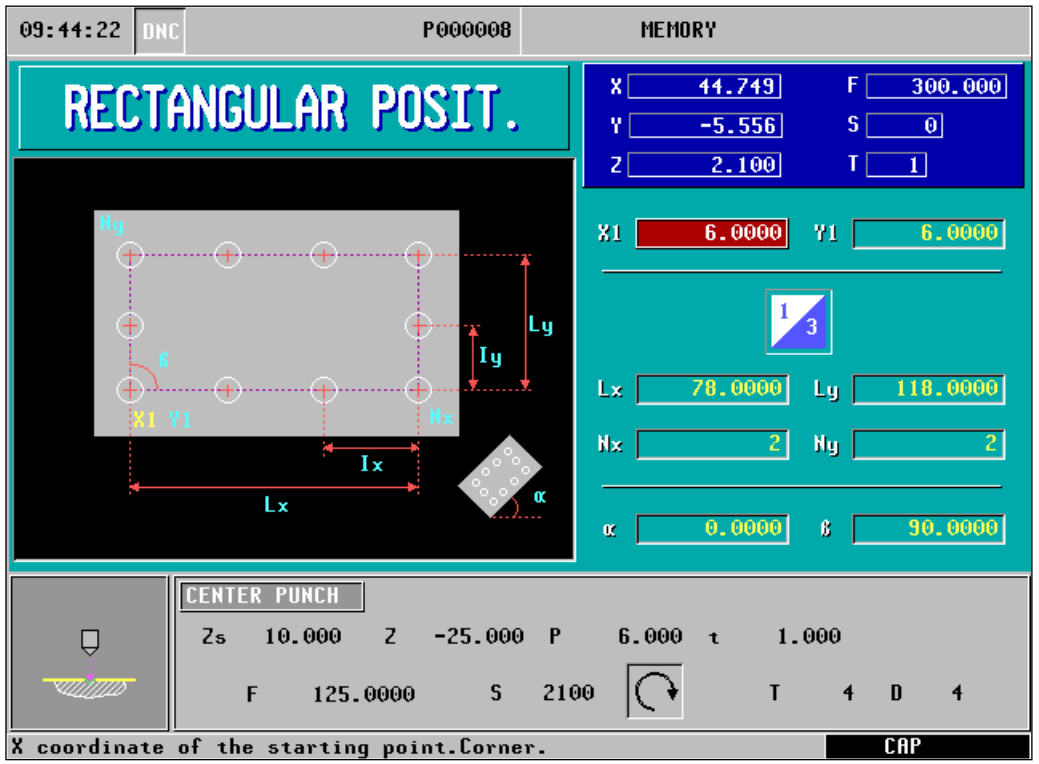


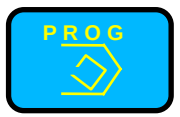
Programming example

Step 7 center point + rectangular multi-point





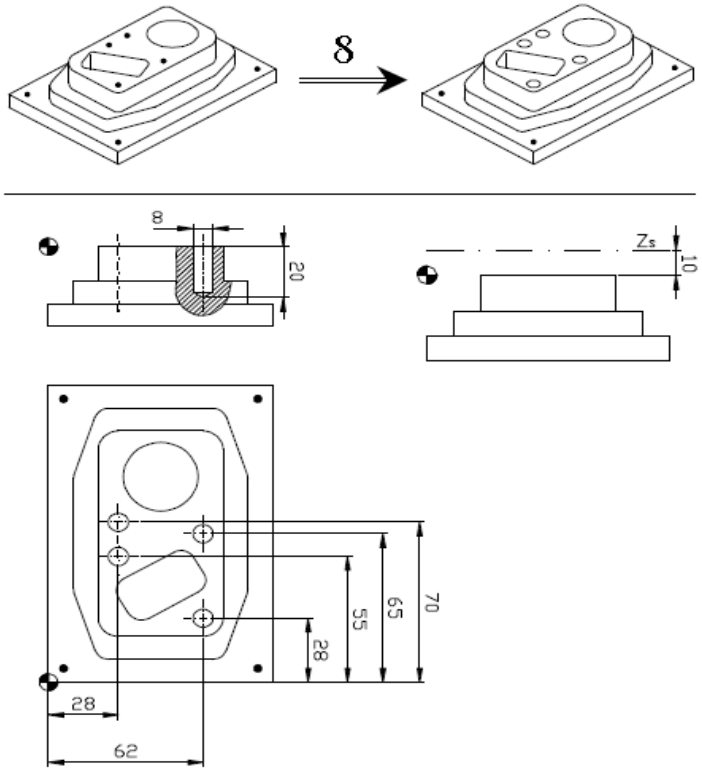


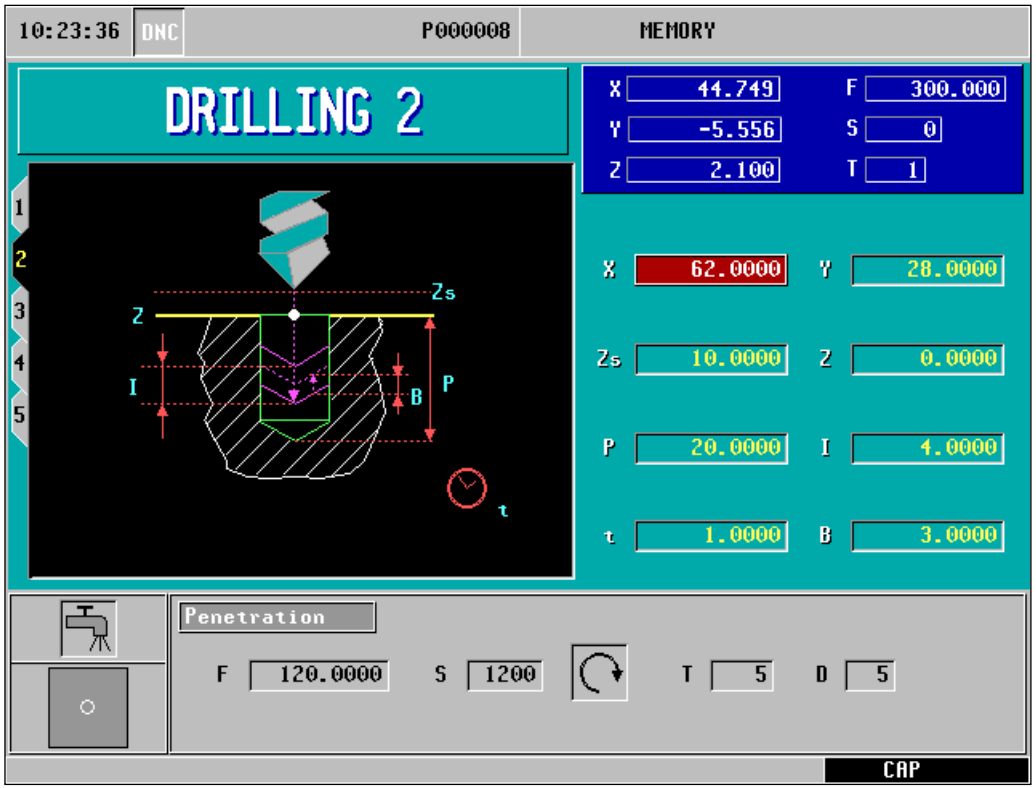


Save cycle

Programming example

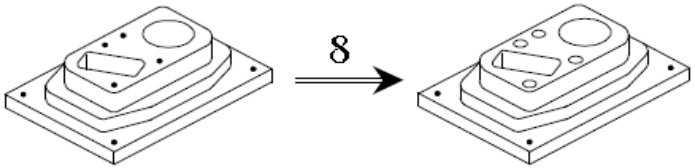
Step 8 drilling + multi-point positioning

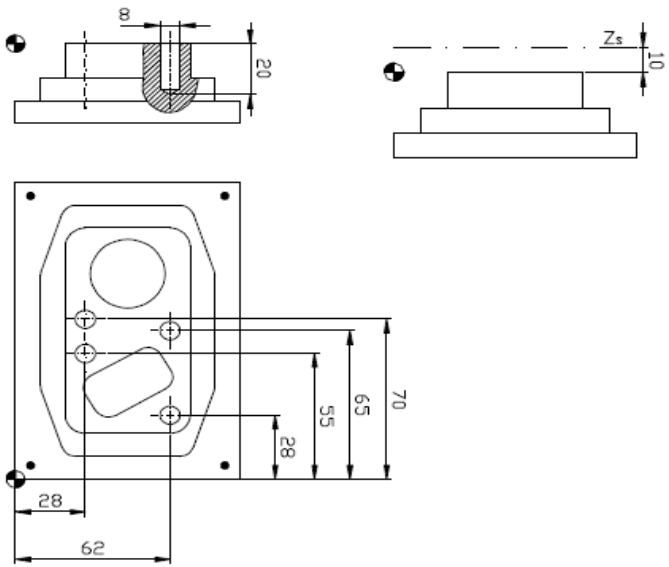


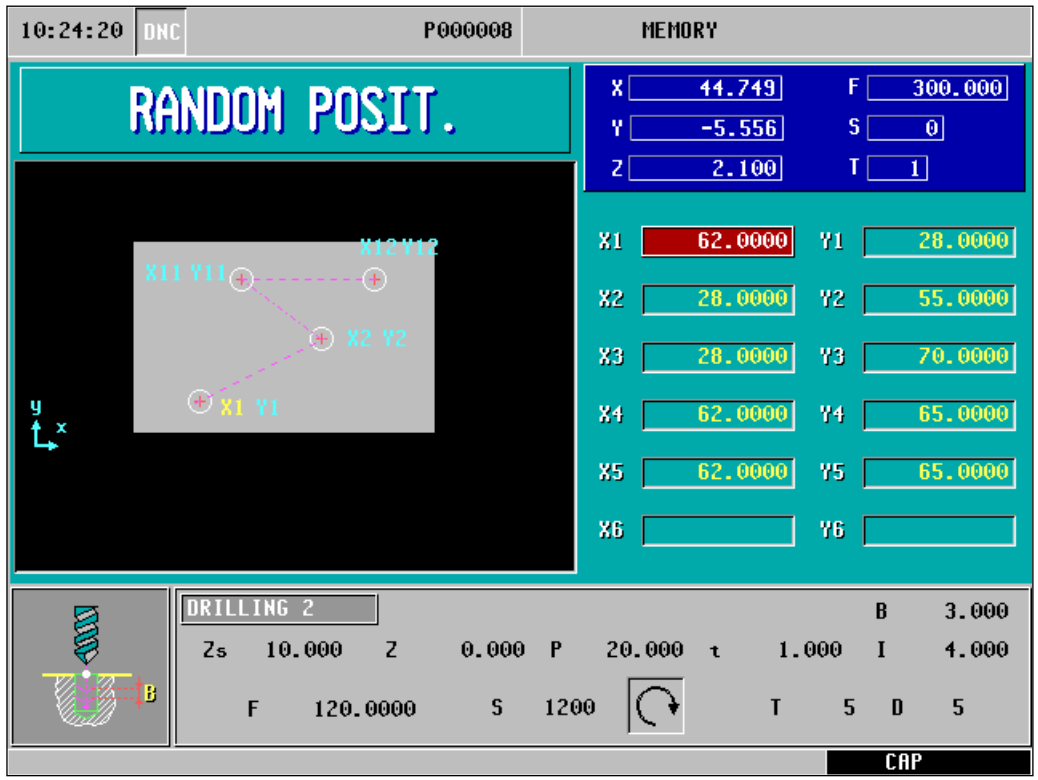


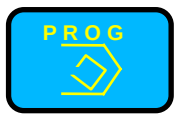
Programming example

Step 8 drilling + multi-point positioning





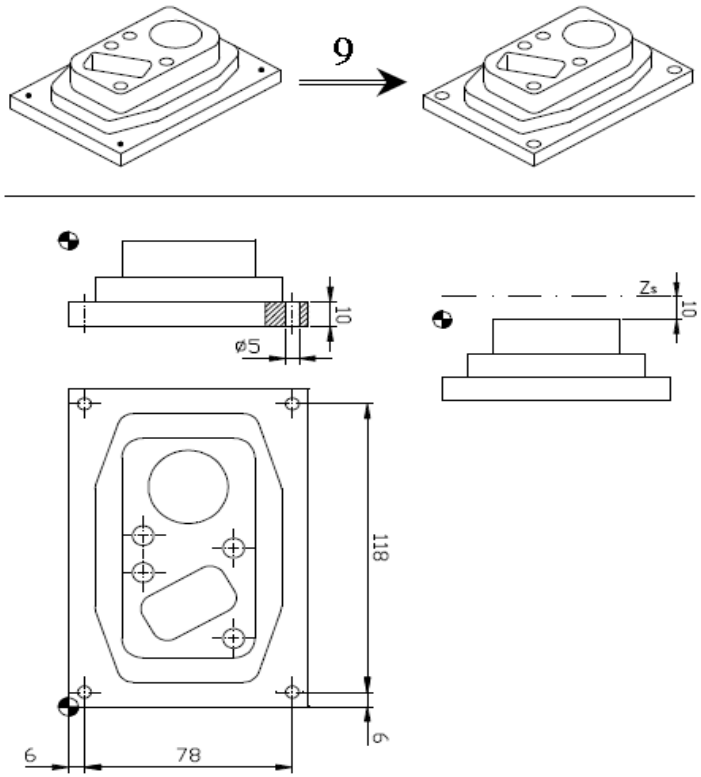


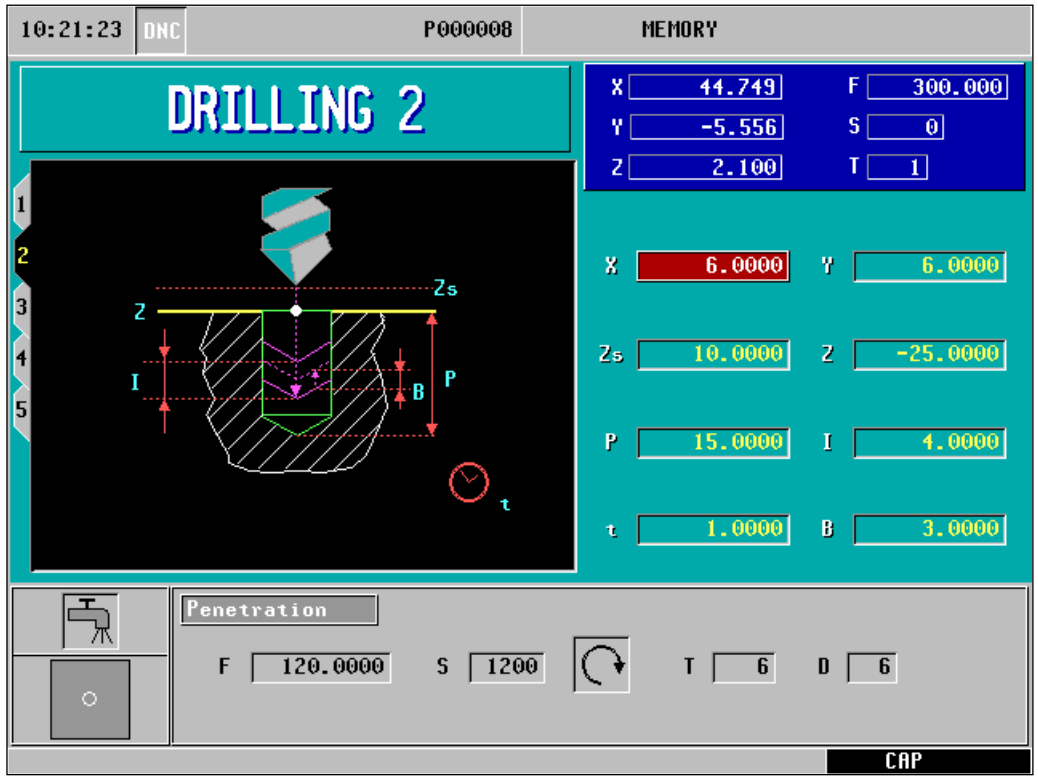


Save cycle

Programming example

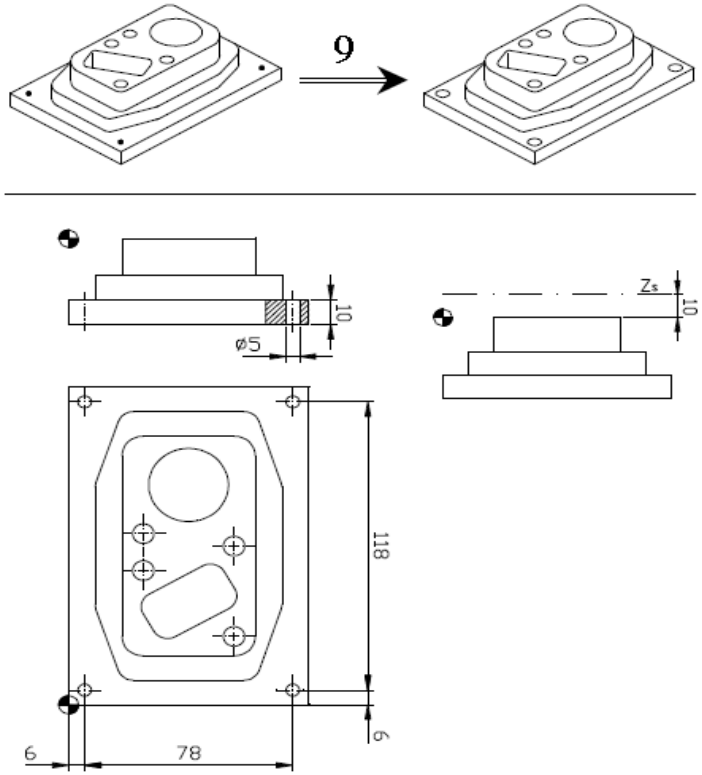
Step 9 drilling +
rectangular multi-point

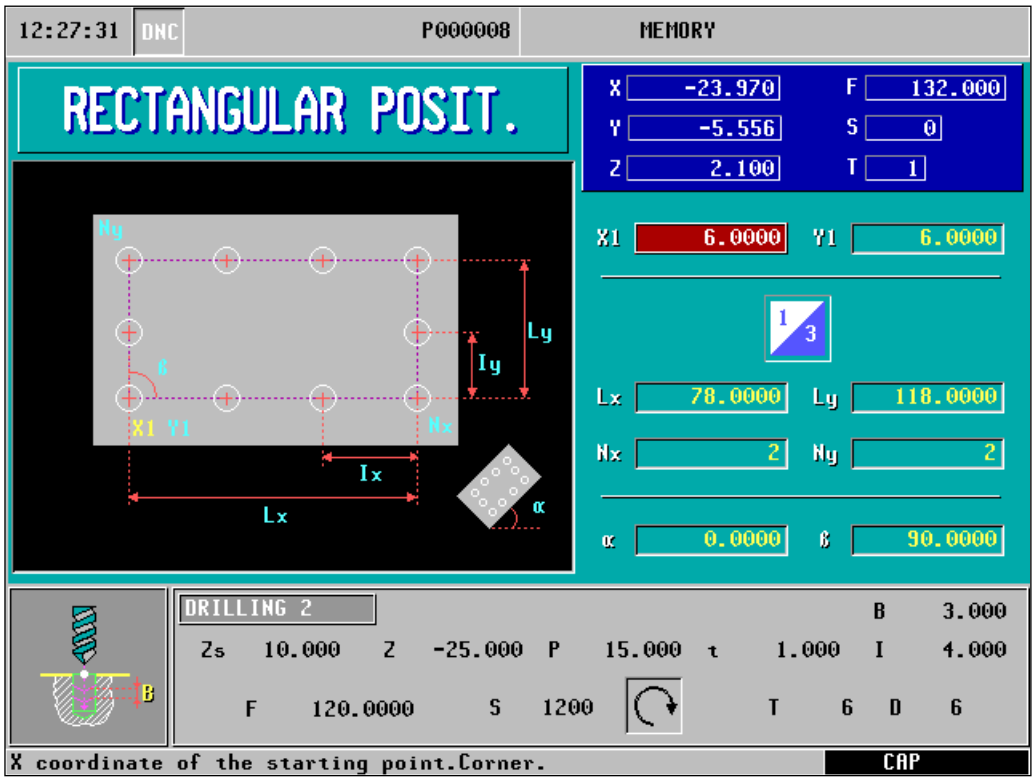




Programming example

Step 9 drilling + rectangular multi-point



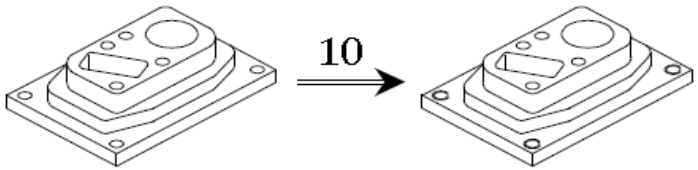


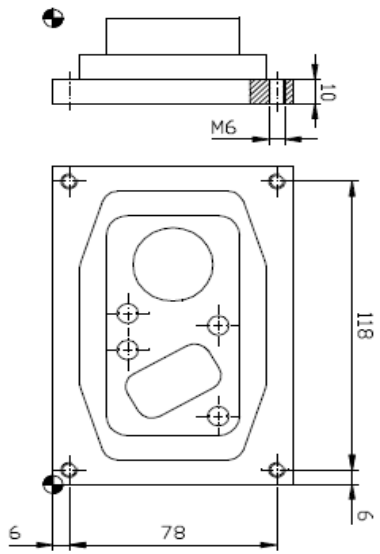
PROG

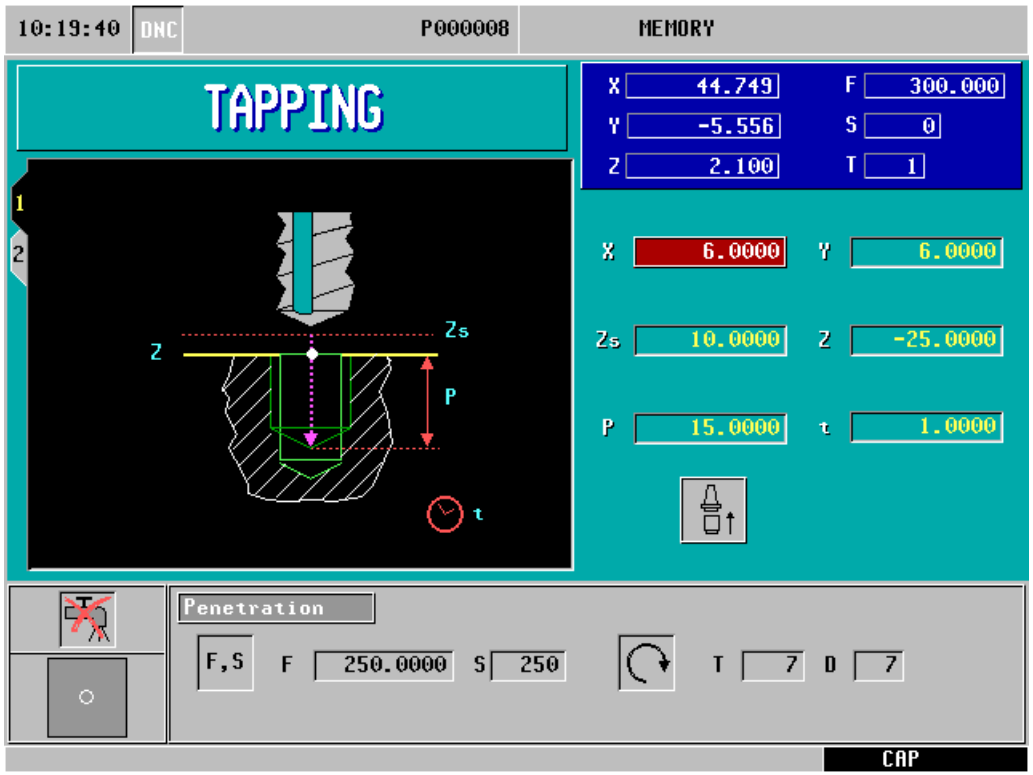
Save cycle

Programming example

Step 10 tapping + rectangular multi-point

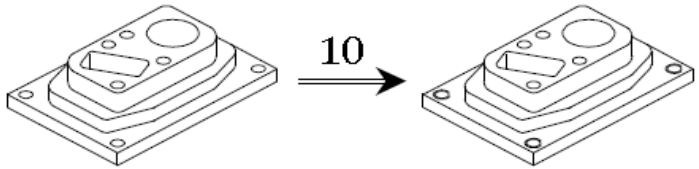


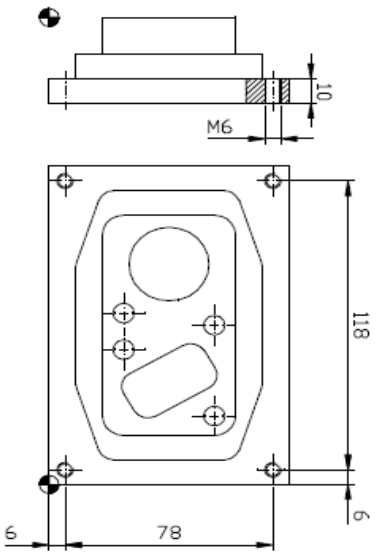


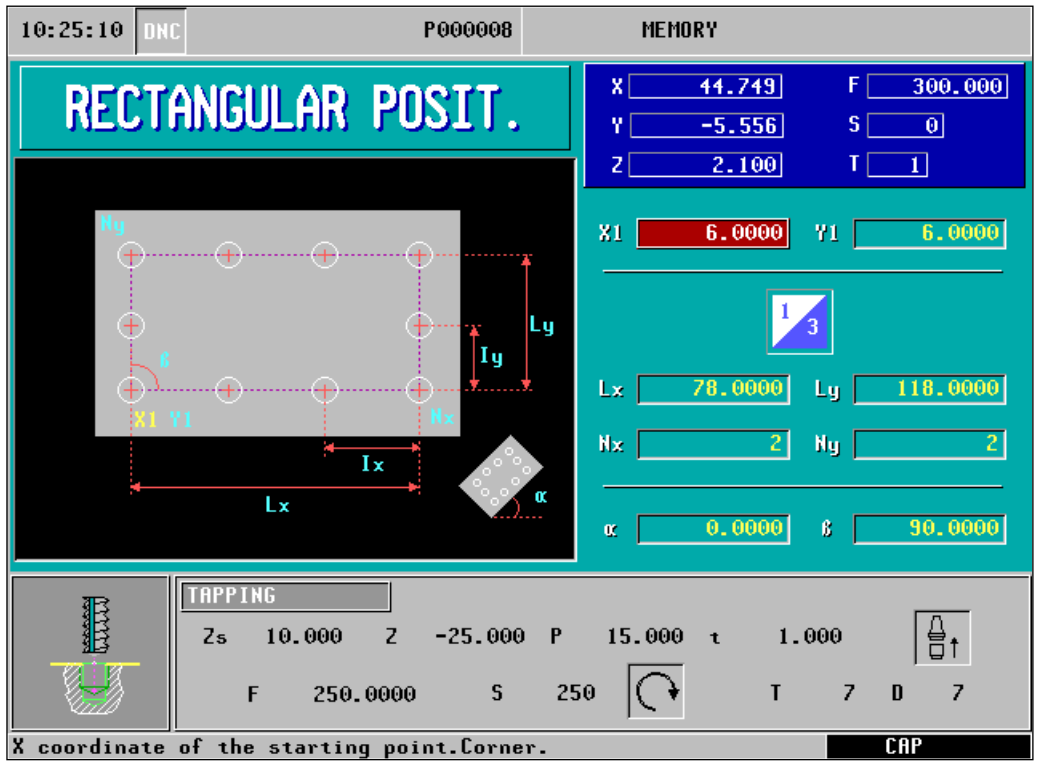


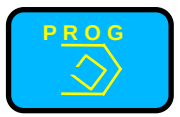
Programming example

Step 10 tapping + rectangular multi-point





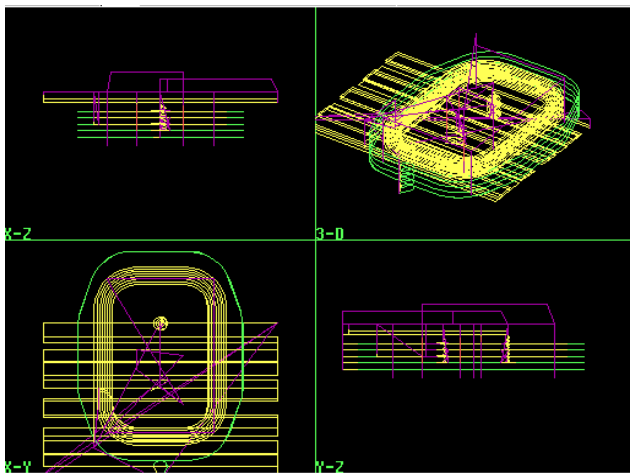




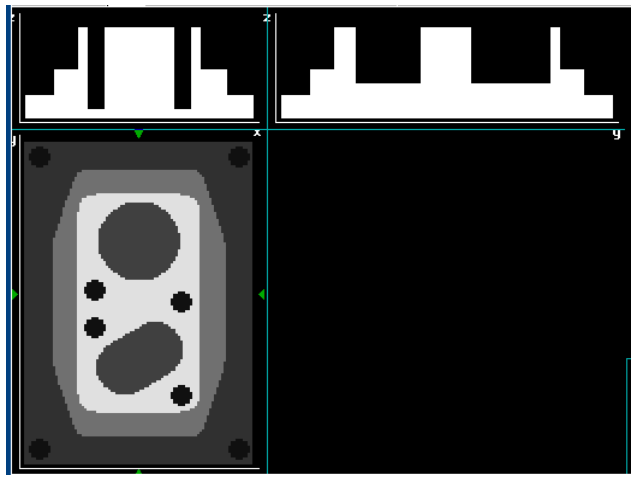
Save cycle

Screens of graphic simulation

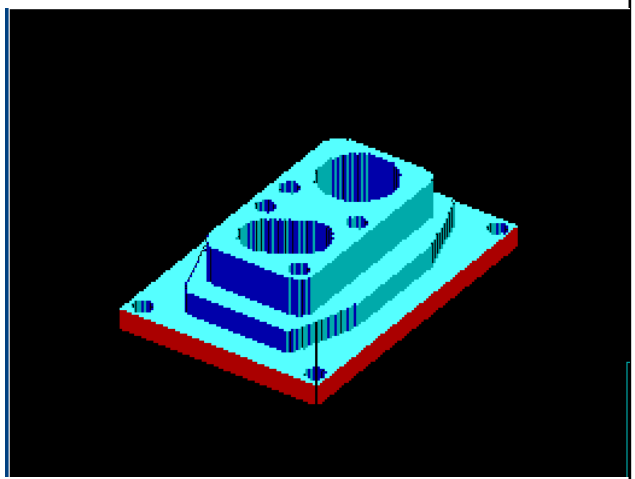
Line graphics



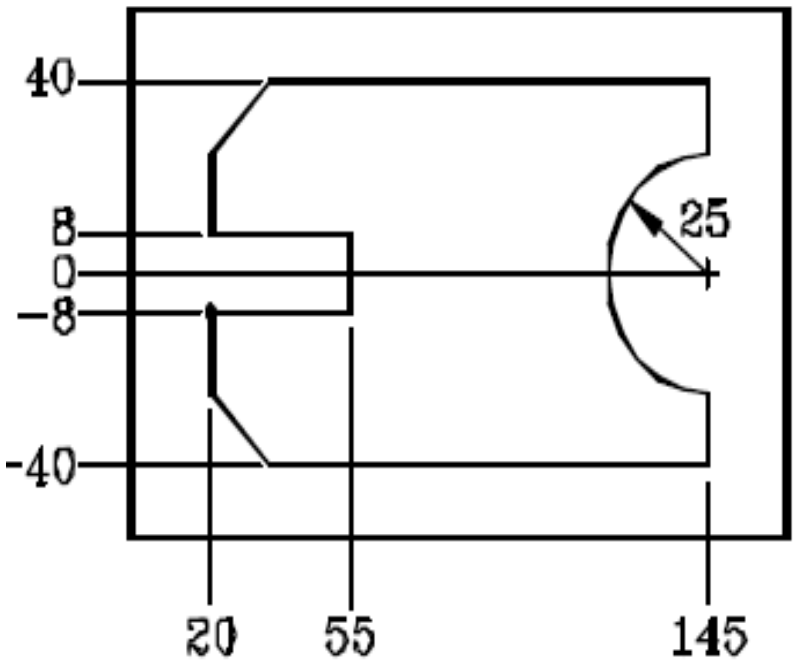
2D graphics



3D graphic



Exercise: Profile



Profile 100

Starting point	X20	Y -8		Validate
Straight	X20	Y -40		Validate
Straight	X145	Y -40		Validate
Straight	X145	Y -25		Validate
Clockwise arc	Xf145	Yf25	R25	Validate
Straight	X145	Y40		Validate
Straight	X20	Y40		Validate
Straight	X20	Y8		Validate
Straight	X55	Y8		Validate
Straight	X55	Y -8		Validate
Straight	X20	Y -8		Validate

Corners **Chamfer**

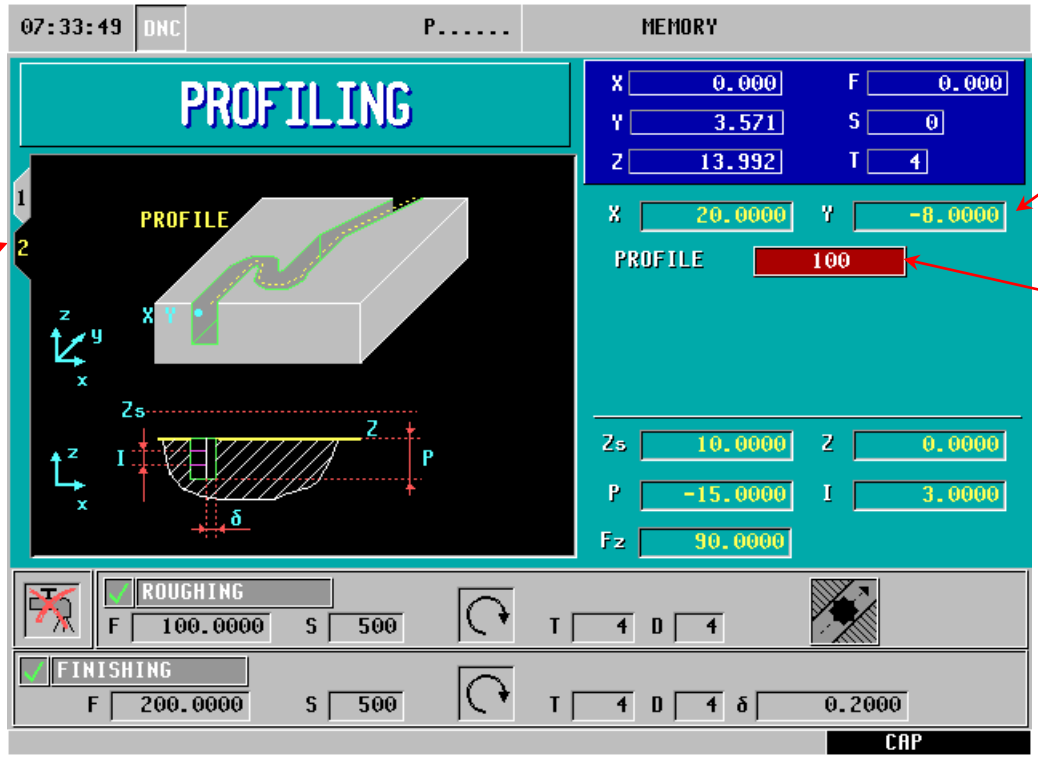
Select the lower left corner Chamfer 15 ENTER
Select the upper left corner Chamfer 15 ENTER
Escape

Finish **Save profile**

Profile editor

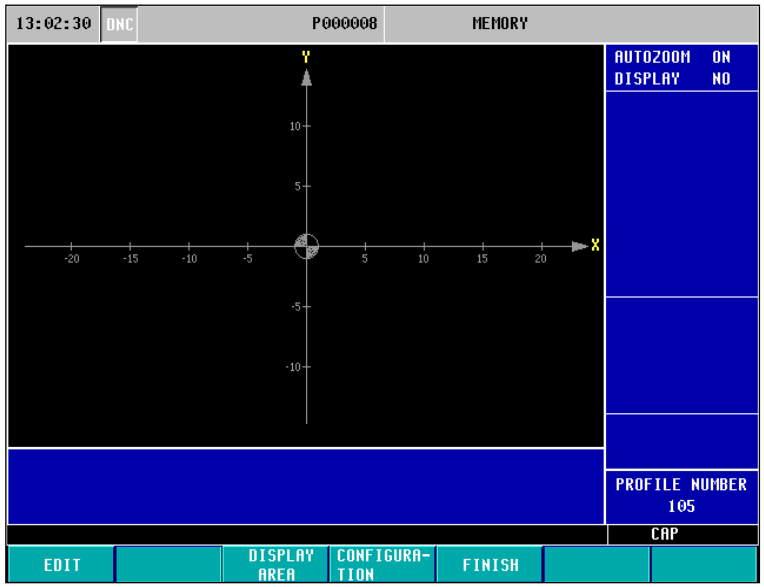
Define profiles with profile editor

F3

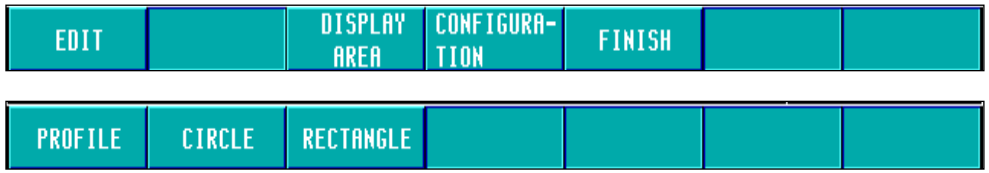


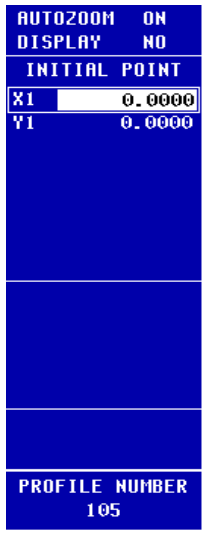
Profile editor

Use the softkeys to work with the profile editor



softkeys



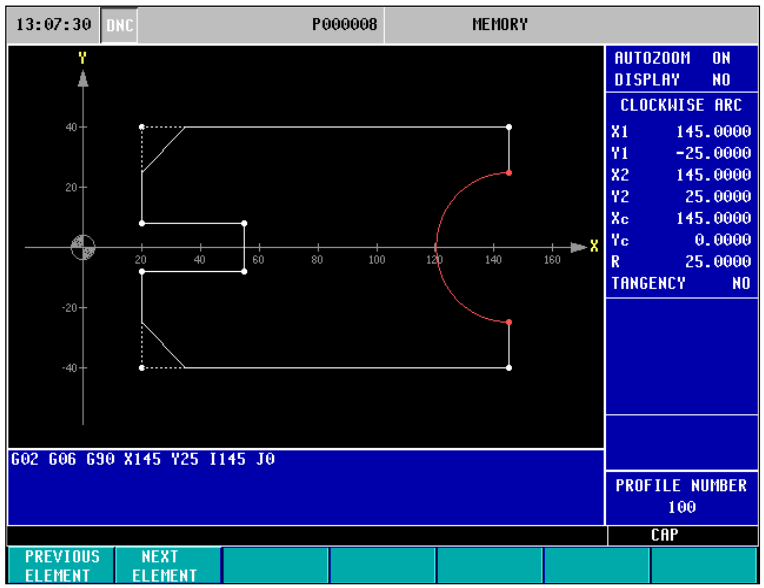


Windows for coordinates

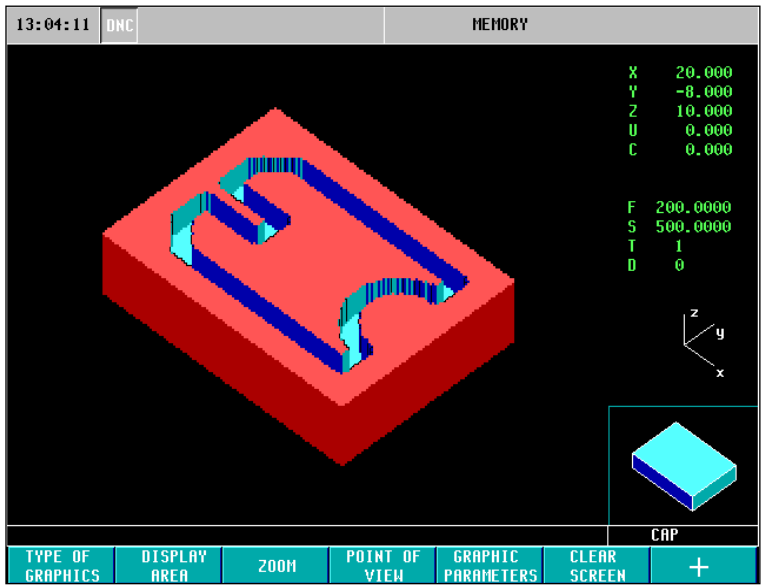
After inputing all values
Press [Validate]

Exercise: Profile

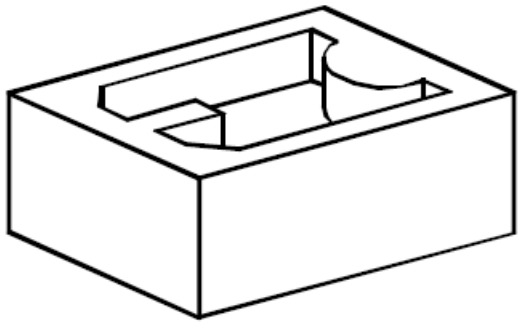
Edit screen

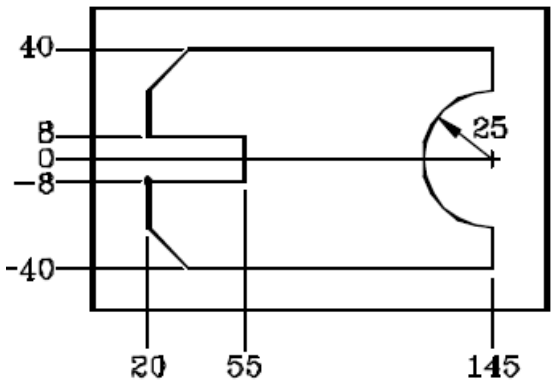


3D simulation screen



Exercise: Profile 2D pocket





Profile 101

Starting point	X20	Y -8		Validate
Straight	X20	Y -40		Validate
Straight	X145	Y -40		Validate
Straight	X145	Y -25		Validate
Clockwise arc	Xf145	Yf25	R25	Validate
Straight	X145	Y40		Validate
Straight	X20	Y40		Validate
Straight	X20	Y8		Validate
Straight	X55	Y8		Validate
Straight	X55	Y -8		Validate
Straight	X20	Y -8		Validate

Corners **Chamfer**

Select the lower left corner Chamfer 15 ENTER

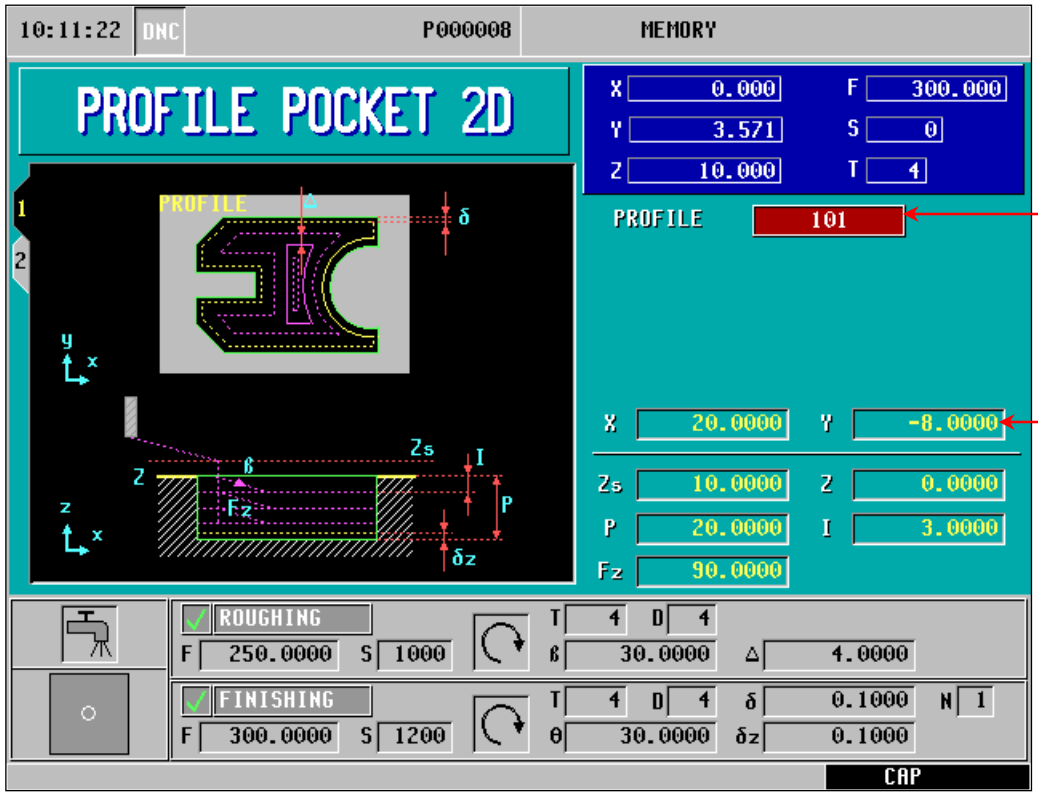
Select the upper left corner Chamfer 15 ENTER

Escape

Finish **Save profile**

Profile 2D pocket

F5



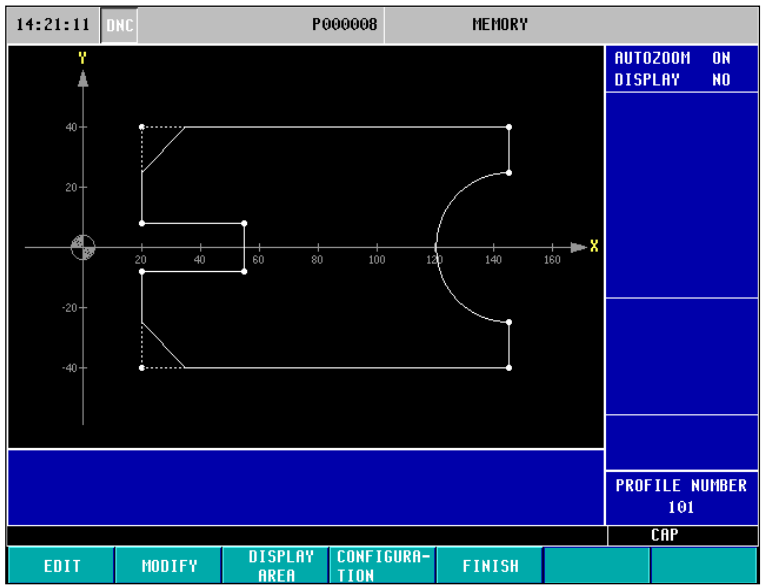
RECALL

Enter the profile editor

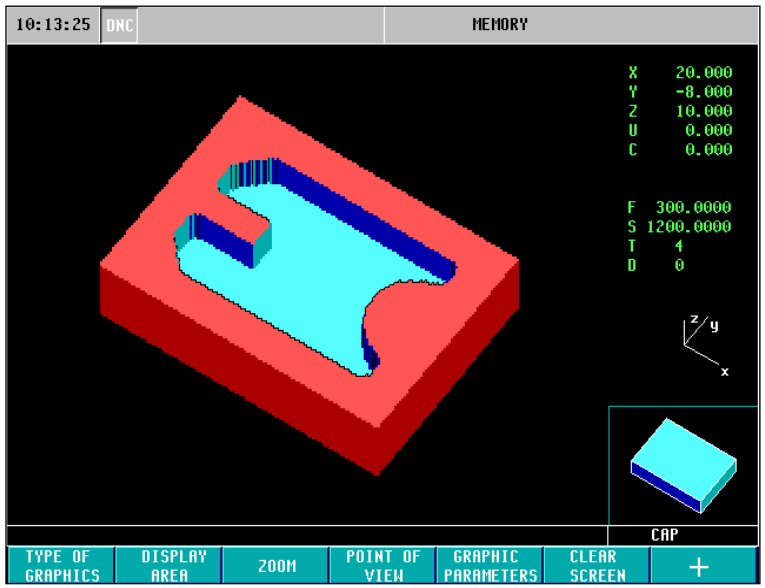
Entry point

Exercise: Profile 2D pocket

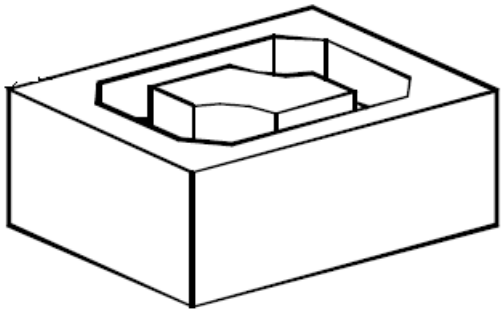
Edit screen

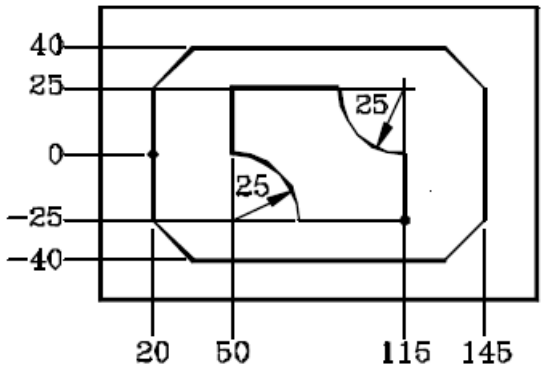


3D simulation screen



Exercise: Profile 2D pocket with island





Profile 102

Starting point	X20	Y0		Validate
Straight	X20	Y -40		Validate
Straight	X145	Y -40		Validate
Straight	X145	Y40		Validate
Straight	X20	Y40		Validate
Straight	X20	Y0		Validate

Corners **Chamfer**

Select the lower left corner	Chamfer 15 ENTER
Select the lower right corner	Chamfer 15 ENTER
Select the upper right corner	Chamfer 15 ENTER
Select the upper left corner	Chamfer 15 ENTER
Escape	

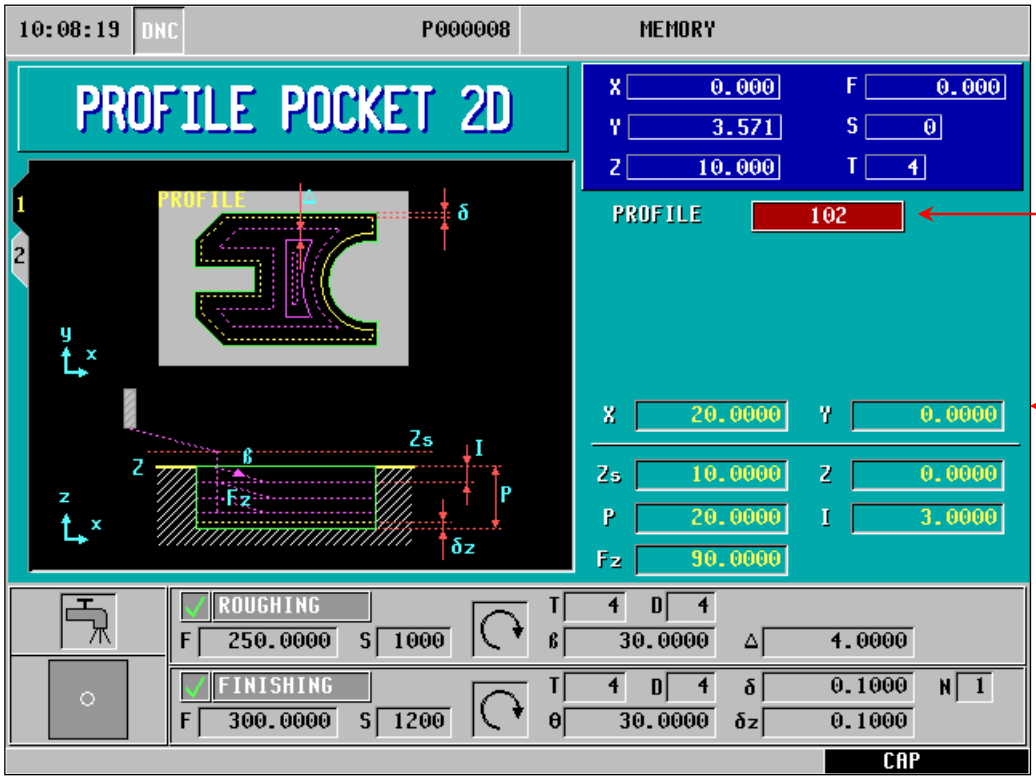
New profile **(Island)**

Starting point	X115	Y -25		Validate
Straight	X115	Y0		Validate
Clockwise arc	Xf90	Yf25	XR25	Validate
Straight	X50	Y25		Validate
Straight	X50	Y0		Validate
Clockwise arc	Xf75	Yf -25	XR25	Validate
Straight	X115	Y -25		Validate

Finish **Save profile**

Profile 2D pocket with island

F5



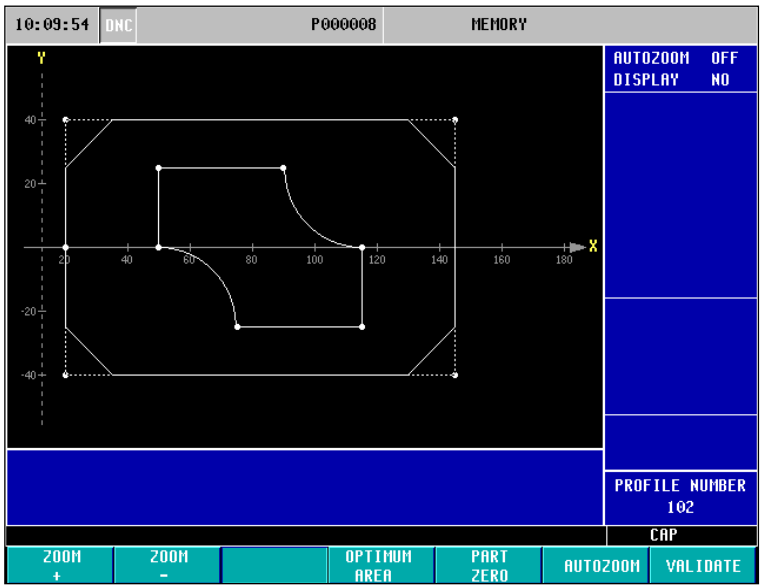
RECALL

Enter the profile editor

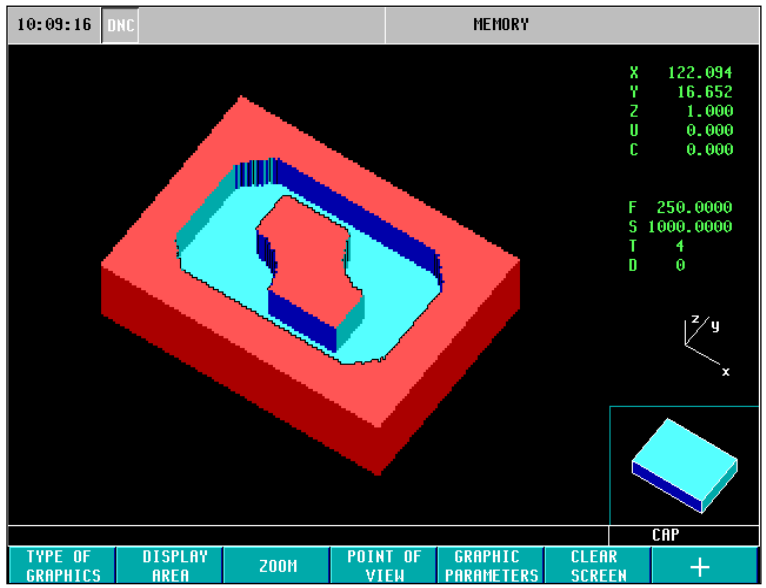
Entry point

Profile 2D pocket with islands

Edit screen

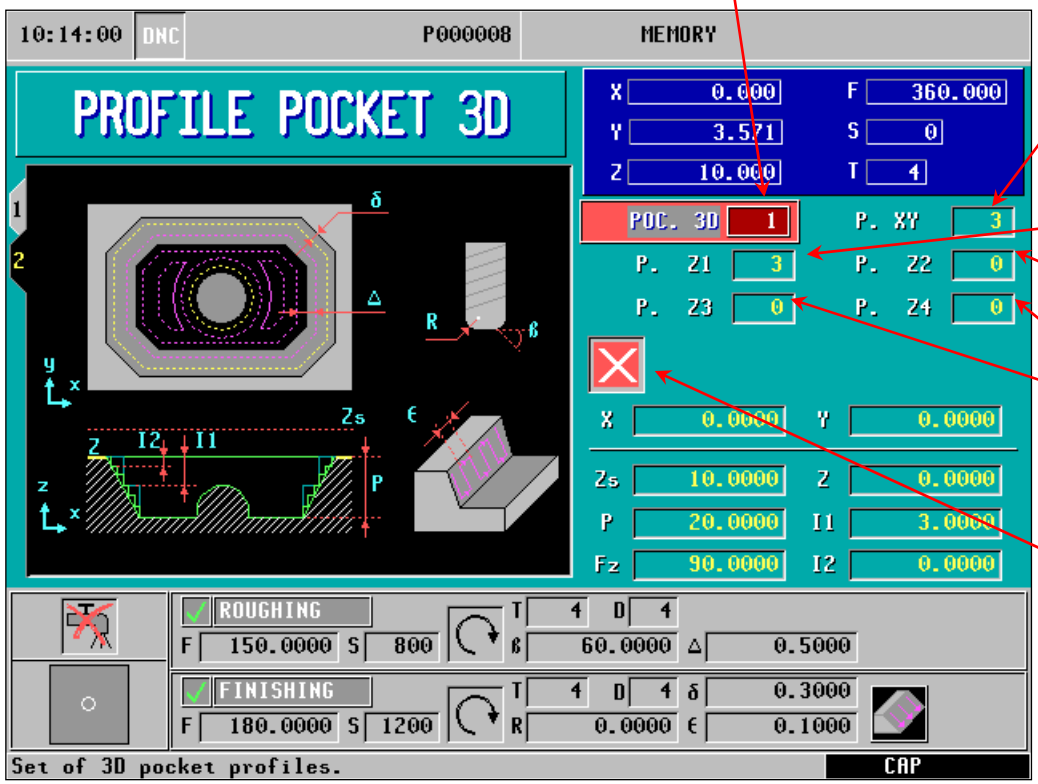


3D simulation screen



Profile 3D pocket

F5



Number for the new part program (0 -999)
This will create a program 995xxx in the CNC memory

Number of the XY profile
Creates program 998xxx in the CNC memory

Number of the Z profile for the XY contour
Creates program :996xxx in the CNC memory

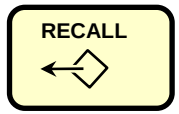
Number of the Z profile for the first island
Creates 9996xxx in the CNC memory

Z profile for more islands

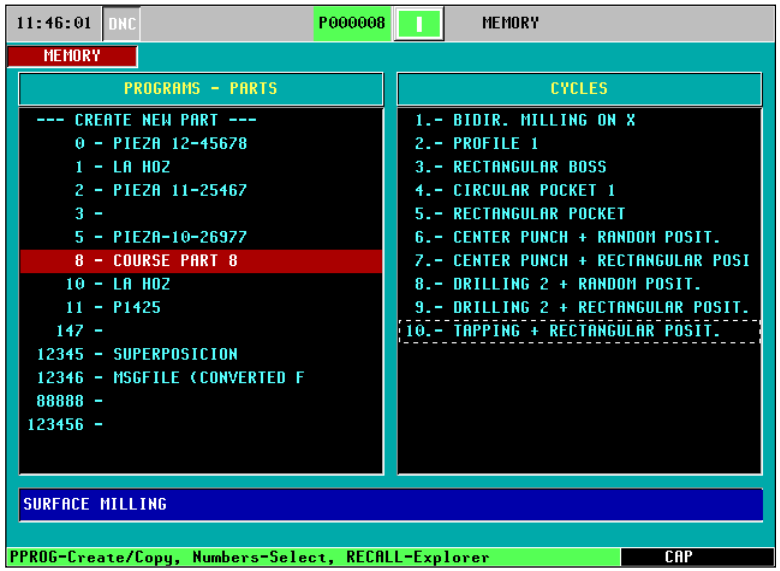
Creates the new program

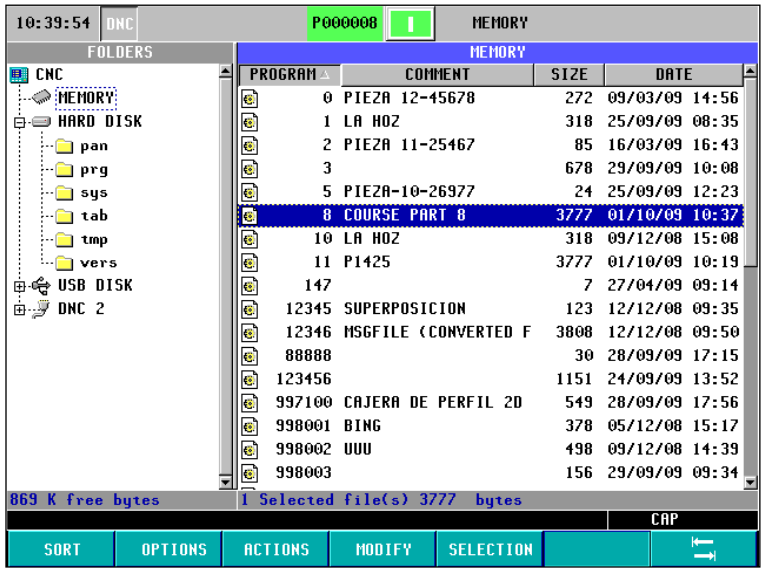
Expanded program manager - explorer

Select PROG screen



Press RECALL to open the explorer





Expanded program manager - explorer

Main programs: **0 – 999 999**

0 – 899 999	user area
900 000 – 999 999	Fagor- and OEM-area

Saved user cycles:

995 xxx	3D pocket programs
996 xxx	Z profiles of 3D pockets
997 xxx	???
998 xxx	2D pockets with islands

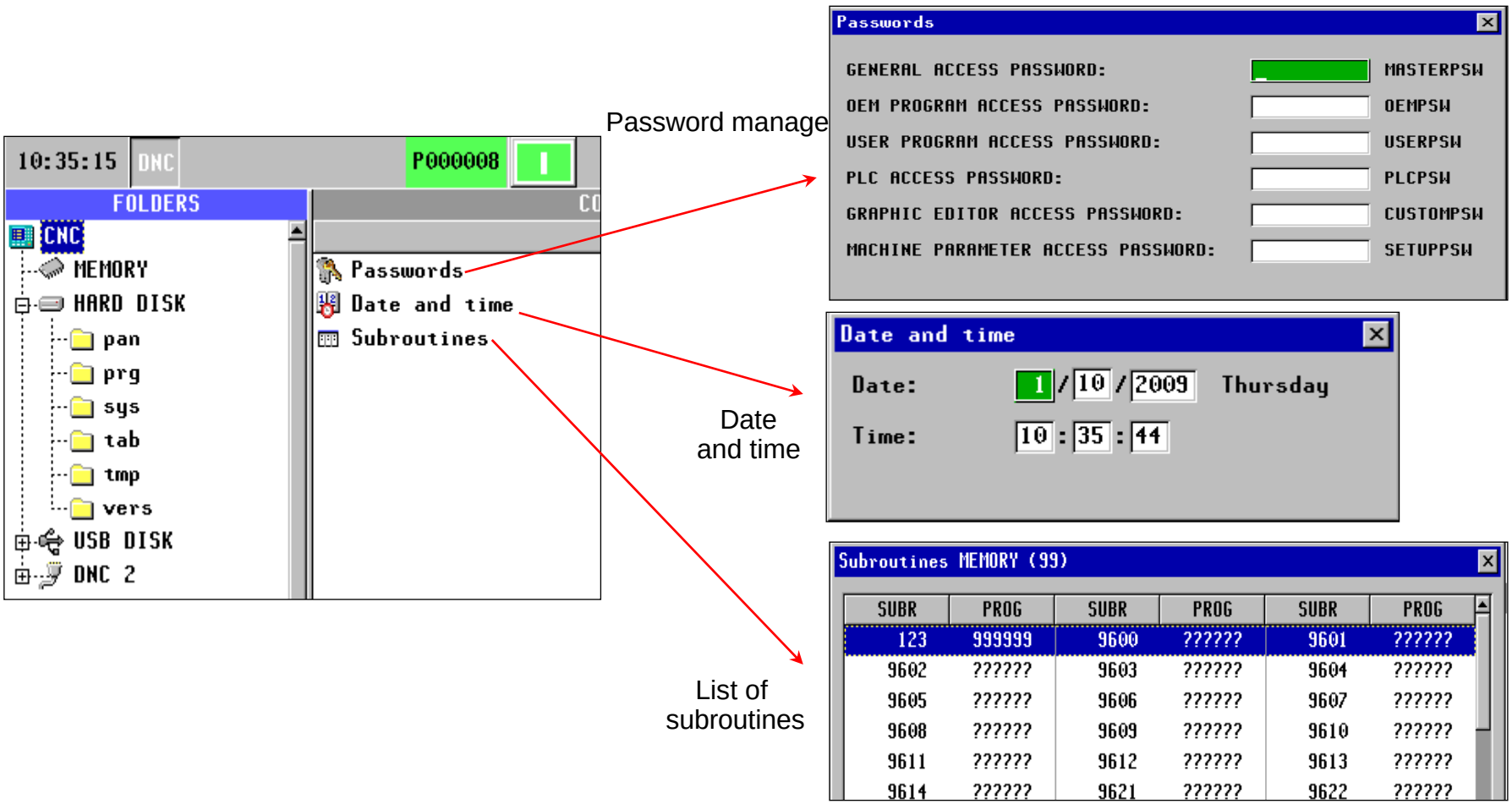
These programs are created through user cycles.
They must be saved separately

Subroutines: SUB0 – SUB9999

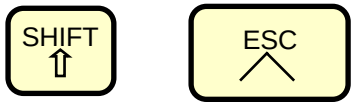
0 – 8999	User area
9000 – 9999	Fagor- and OEM-area

User defined subroutine in reserved main programs
See also “list of subroutines” at the explorer

Expanded program manager - explorer

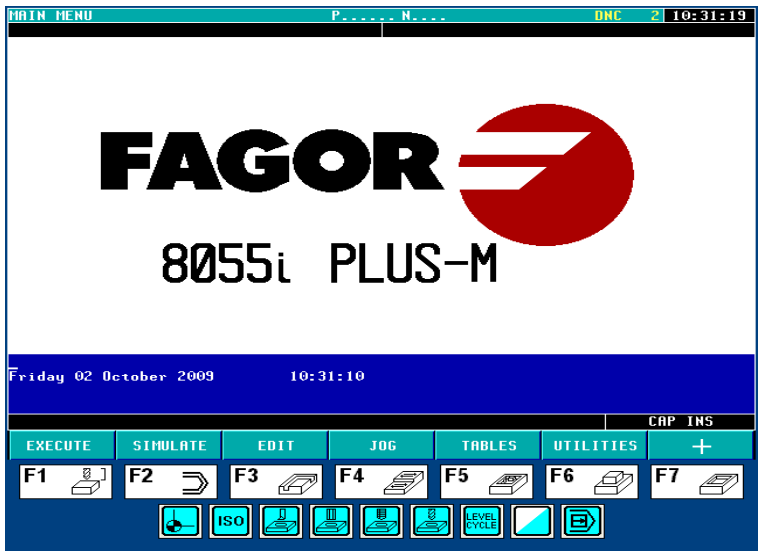


Operation in M mode

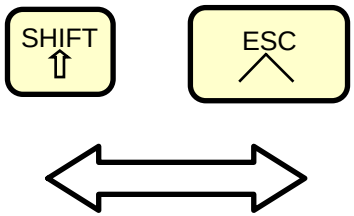
Change the HMI's with 

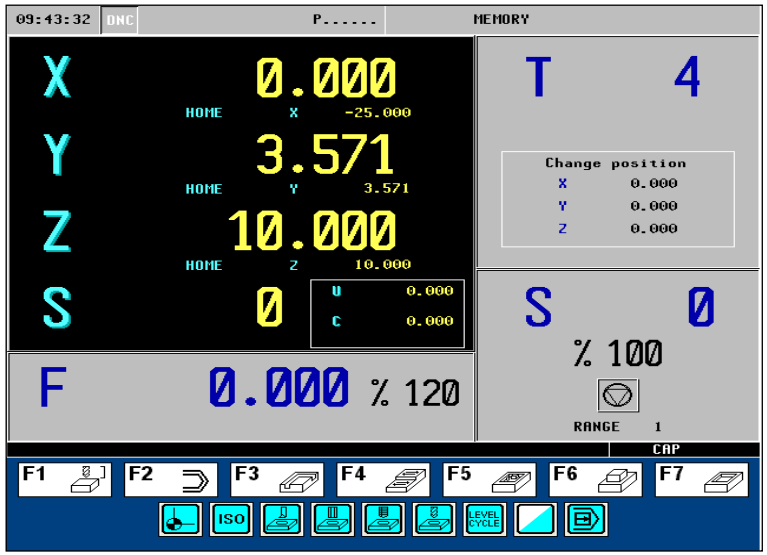
MC, conversational HMI

M basic HMI

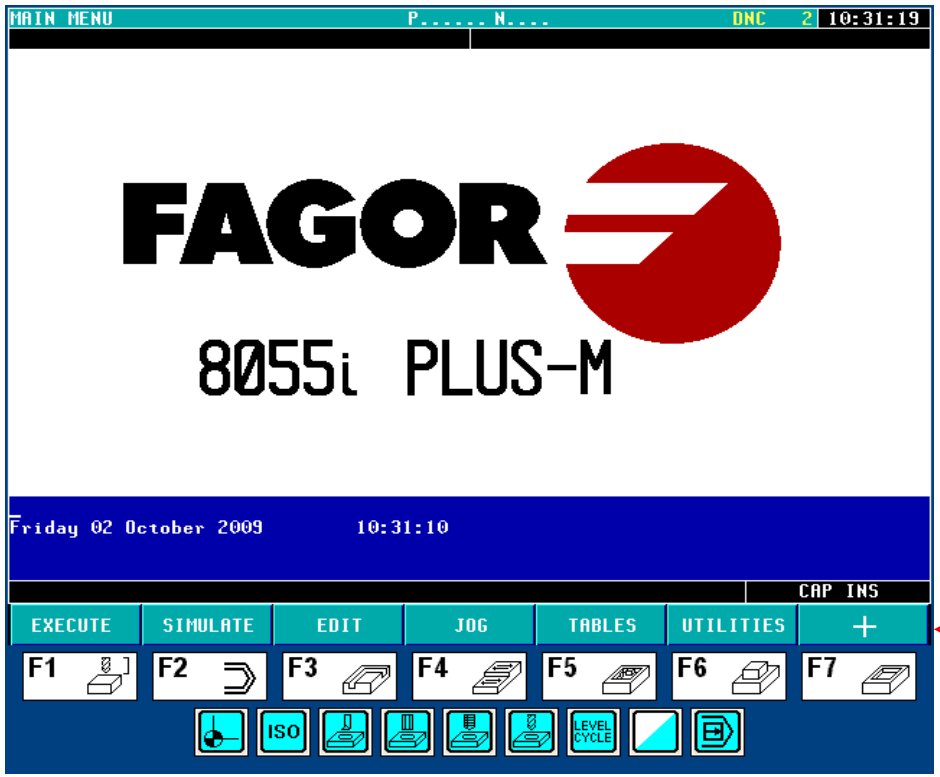


toggle





Operation in M mode

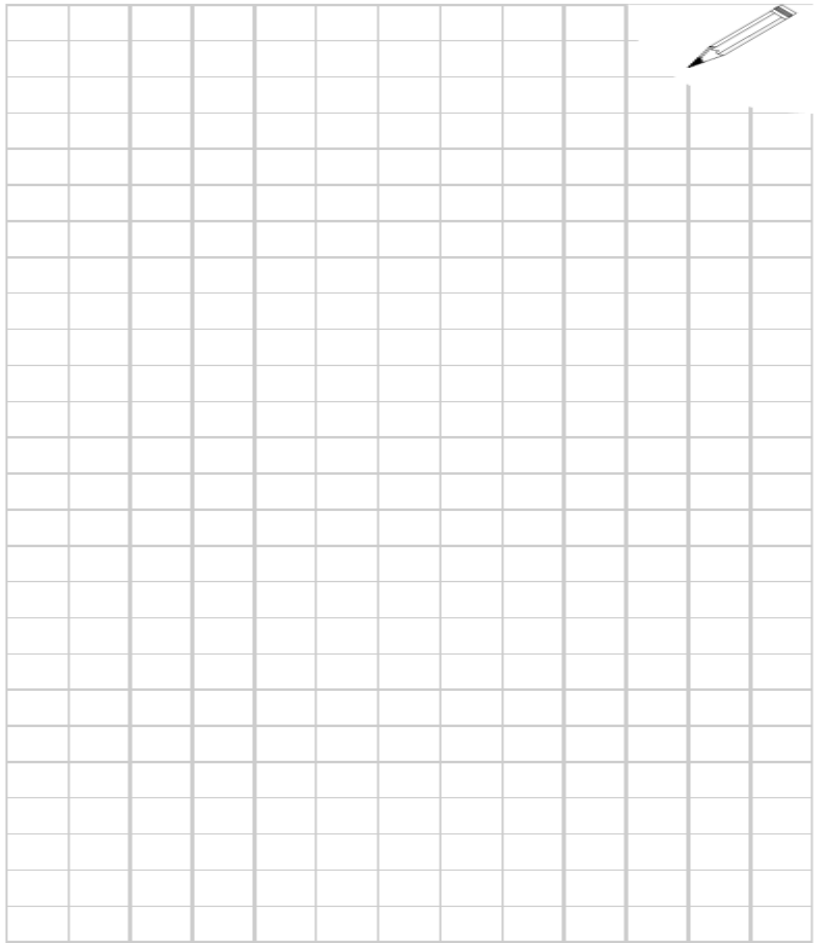


To work with the T HMI, you have to use the functions of the softkeys.

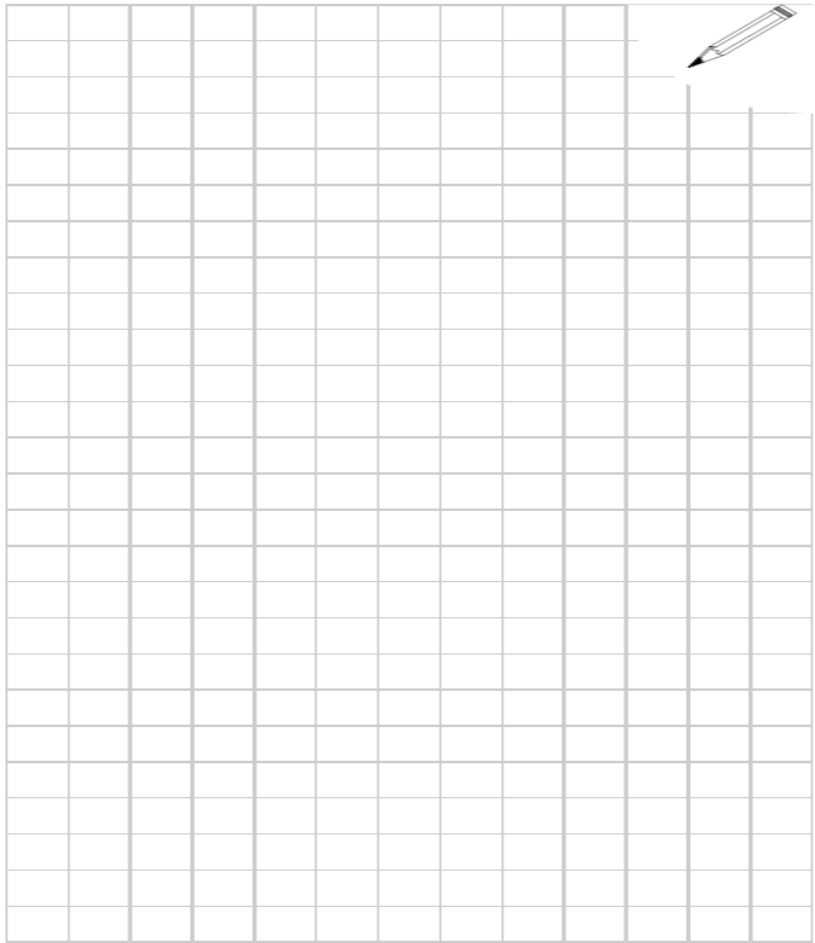
You can control all functions inside the CNC system like edit CNC programs, simulation, tool table, parameters, PLC diagnosis etc.

Softkeys to use and control the M HMI

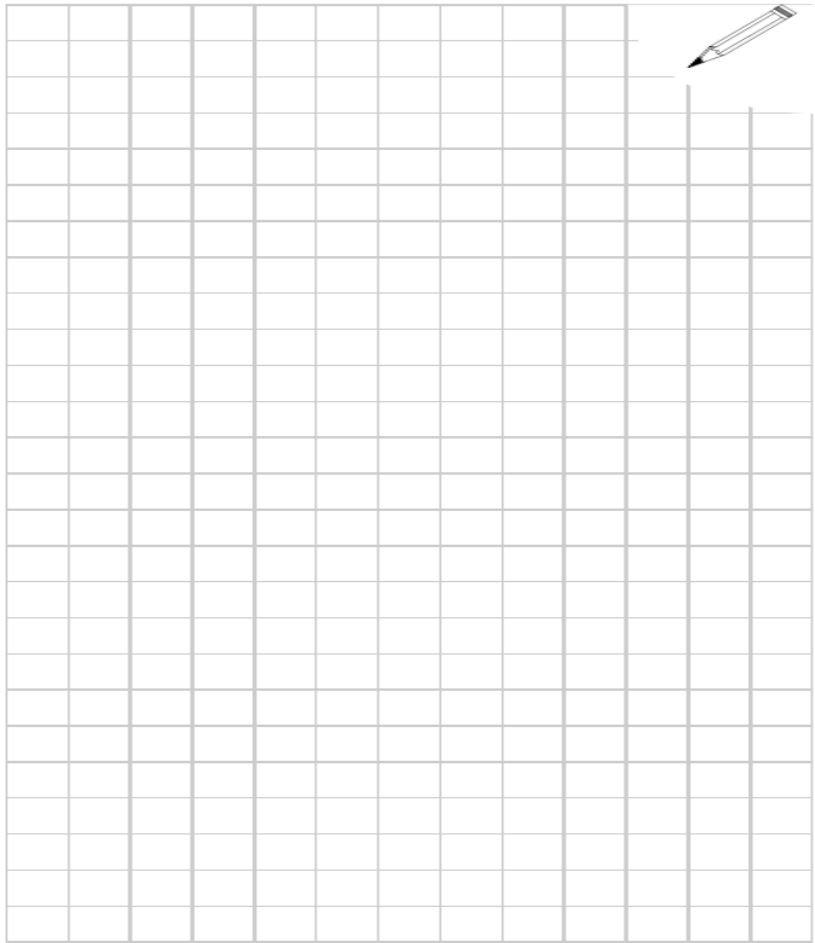
Notes



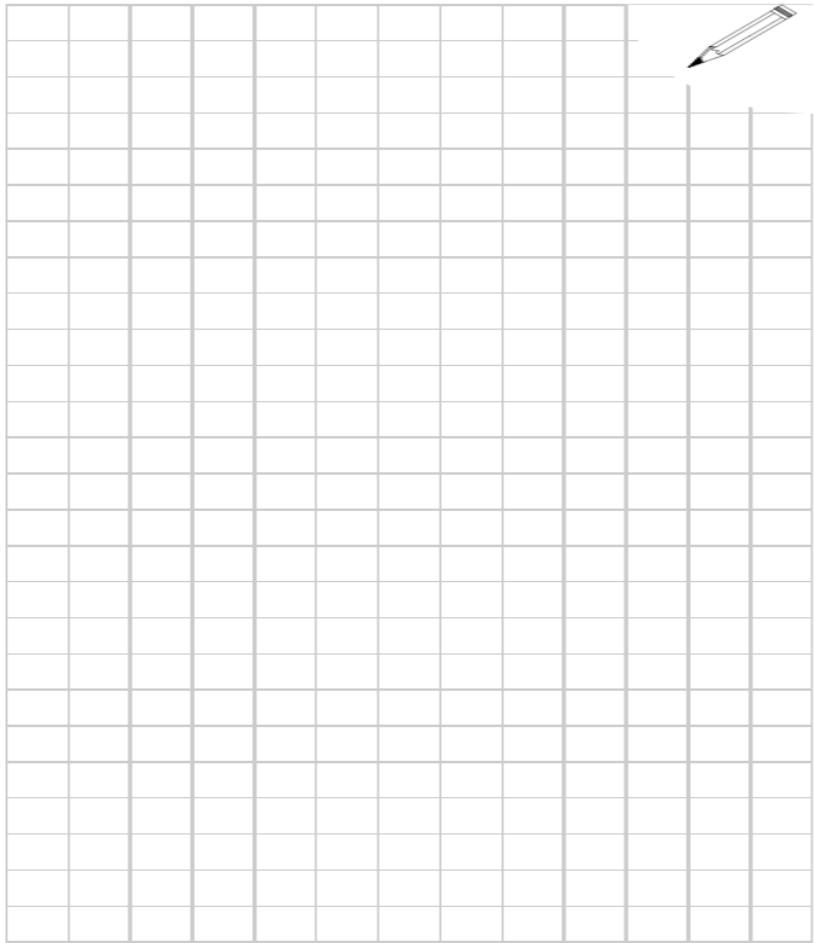
Notes



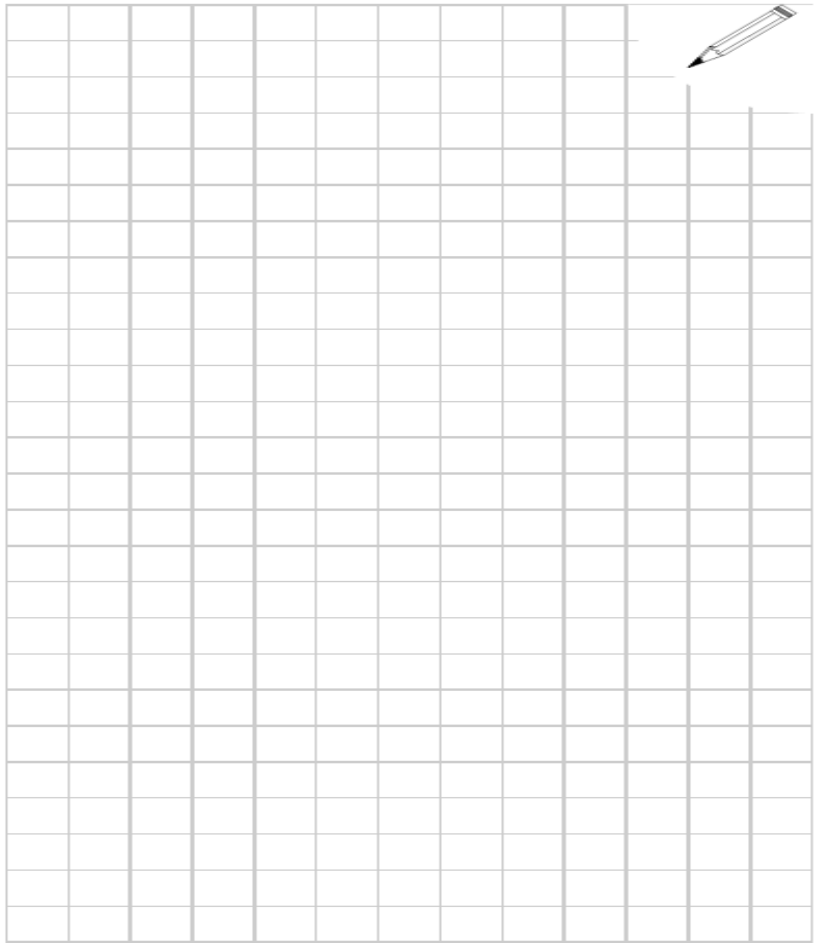
Notes



Notes



Notes



Notes