CNC 90 system

User's manual

version 3.1

tpa Tecnologie Prodotti per l' Automazione SpA

20099 Sesto S. Giovanni (Mi) via Carducci, 221

tel. 02/26223139 fax 02/2486201

5. DESK OPERATIVITIES (D.O.) MODE

6. AUTOMATIC MODE

7. HEADS PARAMETERS

8. TOOL DESIGN

9. TECHNOLOGICAL PARAMETERS

10. PROGRAMS EDITOR

11. PROGRAMS COMPILATION

12. LIST EDITOR

13. FILES MANAGEMENT

14.MANUAL MODE

1. INTRODUCTION

GENERALITIES

CNC90 is a numerical control system based on 2 basic elements:
- a Man-Machine interfacing-programming unit, constituted by a Personal Computer (PC)
- a Central Unit, constituting the electronic part of the control, named **module.**

The module can contain one or more cards named Stations, which can be of the following types:

PTP200N	for control of 2 axes with DC motors
PLC200	with PLC functions

The PTP200N card supports an expansion card, named ESPAS, for control of 3 more DC axes.

Each Station can be provided with Input/Output expansions (I/O) by means of cards INOUTR and IOMOD.

The PC is connected to the Central Unit by means of a standard RS232 serial communication line connected in Ring topology (See Interface RS232).
The module can be equipped with up to 8 stations, numbered from 0 to 7.
Each module controls one machine.

A **multimodular structure** can be arranged as follows:

$\boxed{1}$ - each module controls a machine and is connected in serial line with a PC

$\boxed{2}$ - Maximum configuration on each module is the above mentioned

$\boxed{3}$ - Maximum number of modules is 4

$\boxed{4}$ - The PCs are linked in a network

$\boxed{5}$ - One PC works as master and the others as slaves. Master/slave functionality is only related to logical operation mode of the production plant constituted by single machines.

Directly contact TPA for choosing and installing network. It has to be noticed that network operation may require a server unit, that should be added to the multimodule structure described hereafter. A server unit is a PC committed to network control and resources management.

The possible configurations are shown below.

First picture considers a single module configuration: the two links between PC and module represent signals on serial line RS232. In particular, TxD = transmission signal and RxD = Received signal.

Refer to section "RS232 Interface" for more details.

A multimodular configuration is then considered, by showing the case of 3 modules.

The first PC is assigned as master, while the other two will work as slaves.
Refer to proper appendix for more details on network operation.

MODULE CONFIGURATION

As already mentioned, a Module controls a machine. An example could be a pantograph processing marble or wood workpieces.

For workpiece we mean a unitary product worked on the machine. It can designate, for instance, a wood panel (in case of wood working) or a slab of marble, and so on.

A workpiece is usually defined with three dimensions:
- length (X dimension)
- heigth (Y dimension)
- thickness (Z dimension)

and with five processing faces:
- one frontal
- four laterals

Each machine is able to run up to ten processing heads, called **groups**.
A group can be defined by:

$\boxed{1}$ two controlled axes defined as X and Z

$\boxed{2}$ one controlled Y axis and pneumatic Z

$\boxed{3}$ both X and Y pneumatic; in this case the group is named **dummy**.

On a module 10 groups can be defined, of which:

$\boxed{1}$ 8 controlled (cases 1 or 2 above)

$\boxed{2}$ 2 dummy (case 3 above)

The maximum number of controlled axes on groups is therefore 16.
Groups are numbered from 1 to 10. Groups 9 and 10 are alloted to dummy groups, if existing.

On a module it is also possible to assign:

1 X axis, as piece longitudinal axis

2 auxiliary axes

Auxiliary axes may be for instance piece locking, piece rotation or translation, etc.

Therefore, a maximum number of 19 axes is available on each module.

Up to 8 cards may be installed on each module: specifically, 5 PTP200 cards and 3 PLC200 cards.

INPUT /OUTPUT NUMBERING

All Input/Output connections on module are set in groups of 8, each one named **Port**.

Each port contains 8 Inputs or 8 outputs, each one of one bit.

Bit numbers go from 0 to 7, and any number refers to the position occupied by an Input or Output in the Port; in facts, when programming, Inputs and Outputs should be indicated as Bit+Port.

Ports are numbered from 000 to 255, and they are determined as follows:

station 0	16 Input ports	000 - 015
	16 Output ports	016 - 031
station 1	16 Input ports	032 - 047
	16 Output ports	048 - 063
station 2	16 Input ports	064 - 079
	16 Output ports	080 - 095

and so on, with all stations configured on the module.

The following pages show I/O configuration structure.

Ports unused by expansion cards can be used as internal programs flags.

bit

INPUT

7	6	5	4	3	2	1	0	port	card
									PTP200N
									PLC200

									ESPAS

									INOUTR
									IOMOD

									INOUTR
									IOMOD

									INOUTR
									IOMOD

									INOUTR
									IOMOD

Station ______________________	**Module** ________________________

bit

OUTPUT

7	6	5	4	3	2	1	0	port	card
									PTP200N
									PLC200
									INOUTR
									IOMOD
									INOUTR
									IOMOD
									INOUTR
									IOMOD
									INOUTR
									IOMOD
									FLAGS

Station ________________________	**Module** __________________________

2. SYSTEM HARDWARE

PROGRAMMING UNIT

The programming and man-machine interfacing unit is composed by a Personal Computer, which can be an Olivetti(*), IBM (*), IBM compatible or an industrial PC (IC 286) manufactured by TPA; all working in ambient MS-DOS (*) and with Controller 286.

However, the PC should have at least the following configuration:

- 1Mbyte ram memory
- VGA monochromatic graphic screen.
- 20Mbyte hard disk and 1.44 Mb 3.5" floppy-disk drive.
- Serial interface RS232 for asynchronous communication with central unit PTP1000.

Optionally, the unit can be equipped with:

- Ram memory expansions.
- More capable hard disk for higher performances.
- VGA graphic colour screen.
- Centronics interface to run the printer

Starting from PC supplied configuration. it is possible to plan, according to the different requirements, the choice of programming unit and optional equipment.
Main cases and conditions are shown below:

Modules Number
1

more than 1

1. network installation is required;
2. PC choice may be imposed by network used:
3. Necessary ram memory must be evaluated according to used network.

---

1. Olivetti is a registered trademark by Ing. C. Olivetti & C., SpA.
2. IBM is a registered trademark by International Business Machines Corp.
3. MS-DOS is a registered trademark by Microsoft Corp.

2.3

Network installation must be provided also when data sharing is required between machine PC and other programming units. Examples are management programming or CAD-CAM designing : these programs generally run on dedicated programming units, installed in an office ambient, and information transmission with machine PC is realised by a network hardware and software.

Multitask management
not required

required

1. computer >= 386
2. DESQview1 386 program is required;
3. ram memory >=4 Mbytes;

For multitask running see Chapter 3 and dedicated appendix.

Installation on CAD-TPA
not required

required

1. colour screen;
2. the installation of a mathematics coprocessor or of a computer 486 is suggested;
3. consider if the use of mouse, graphic tablet or tecnigraph is required;
4. CAD-TPA software must be bought separately:

A Personal running on CAD-TPA program is generally installed in office :
connection machine PC is realised by network or floppy disks.

Printing usage
not required

2.4

required

1. A Centronics interface must be available for the PC to communicate with printer;

---

1. DESQview is a registered trademark of Quarterdeck Office Systems.

2.5

Serial connection for dedicated usage
not required

required

1. PC must be implemented with a second serial interface for asynchronies connection towards other peripheral:

Use of printed labels on bar code
not required

required

1. a bar code driver must be supplied compatible with ALFA 39 code and working as a keyboard;

2.6

CENTRAL UNIT

The central unit is constituted by a 6U 19" modular structure, based on DIN standards cards for machine interfacing, suited for location in armoire.
It is composed by the following hardware modules:

- cpu PTP200N
- ESPAS card for axes expansion.
- HSINT card for quick interpolation.
- PLC200 card with plc functions
- INOUTR card for I/O expansions.
- IOMOD card for modular I/O expansion.
- Power feeder.

PTP200N

CPU card based on microprocessor 8085AH1 at 10MHz with:

- 32K RAM memory
- 64K ROM memory
- Interfacing circuits with 2 CC motors.
- 24 opto-coupled inputs
- 24 opto-coupled outputs with transistors 24Vcc/1A.
- Serial interface RS232 for asynchronous communication with the PC.
- Parallel interface for quick communication between cards PTP200 and PLC200.
- Interface for plug-in connection with axes expansion card.
- Local interface for I/O expansion cards connection (INOUTR and IOMOD).

2.7

ESPAS

Axes expansion card with plug-in connection for PTP200N, with:

- Interfacing circuits with 3 CC motors.
- Auxiliary analogic Output ca. 10V
- 4 opto-coupled inputs.

HSINT

Interpolation card based on microprocessor 80C186 at 12.5 MHz with:
- 64K RAM memory
- 64K ROM memory
- Interface for plug-in connection with card PTP200N.

PLC200

CPU card based on microprocessor 8085AH1 at 10 MHz, with:

- 32K RAM memory
- 64K ROM memory
-Serial interface RS232 for asynchronous communication with the PC.
- Parallel interface for quick communication between cards PLC200 and PTP200N.
- 6 interfaces for plug-in connection with I/O modular cards.
- Serial interface RS232/RS485 for asynchronous communication with I/O digital devices.
- Local interface for I/O expansions cards (like INOUTR and IOMOD) connection.

INOUTR

I/O expansion card with:

- 24 opto-coupled inputs
- 24 output relay 24Vcc/2A, 220Vac/10A.

IOMOD

I/O expansion card with:

- 6 interfaces for plug-in connections with I/O modular cards.

MODINP

Module for plug-in connection with cards PLC200 and IOMOD, with:

- 8 opto-coupled inputs.

MODOUTR

Module for plug-in connection with cards PLC200 and IOMOD, with:

- 8 output relay 24Vcc/2A, 110Vac/10A.

POWER FEEDER

Drawer version insertable in a 19" rack.

- +5V 10A
- +- 12V 2A

Version for control panel rear assembly.

- +5V 20A
- +-12V 2A

CE1004

Rack complete with power feeder with 4 positions available.

Size: L. 332; H. 265; D. 275.

CE1010

Rack complete with power feeder with 10 positions available.

Size: L. 483; H. 265; D. 275

The rack, if supplied with power feeder for control panel rear assembly, can have up to 16 free positions.

2.9

HARDWARE MODULES

Module configuration requires 8 PTP200N or PLC200 stations.

The module highest number of I/O (1024 Inputs and 1024 Outputs) is equally divided among the cards in blocks of 128 inputs and 128 outputs.

Interfacing circuit with CC motors.

This circuit is constituted by the following elements:

- Analogic output +- 10V with offset and gain regulation trimmer realised by means of a 12 bit D/A converter.
- Opto-coupled inputs for encoder phases for frequencies up to 50KHz per phase (200KHz working frequency with multiplication *4) suited for encoder with output "push pull" or "open collector" at square wave fed with 5 or 12 Vcc; encoder feeding power supplied by the card: it can be selected by means of 5 or 12 Vcc/100mA strappings.
- Position counter with direction probe (sampling frequency : 5Mhz).

Card PTP200 provides interfacing circuits for 2 CC motors. The control characteristics are:

- working frequency max 150 KHz (equivalent to 90 mt/1' with resolution 0.01 mm).
- Acceleration time selectable in milliseconds.
- Continuous velocity and position control.

On every PTP200N card can be added an axes expansion card with plug-in connection with circuits for 3 more DC motors (ESPAS).

RACK CONFIGURATION

As already mentioned, the rack as supplied is a standard case size "double europe" with 16 connectors to insert the same number of cards, or 10 cards plus drawer size power feeder.

The signs on the basic logic circuit are divided into:

- feedings
- parallel interface for communication between PTP200N and PLC200 (ISB)
- local interface for connection between PTP200N and INOUTR or IOMOD and between PLC200 and INOUTR or IOMOD (ILB)
- RS232 serial interface

Feedings

All feeding pulses, constituted by the filtered voltages +10V +- 10%; +-20v +-10%, are multiplexed along the basic logic card on the connector (8+8): therefore it is enough to connect the feeding circuits to the connector located on basic card (this connection is not necessary when using a drawer type power feeder.

Parallel interface (ISB)

Used for quick data transfer between cards PTP200n and PLC200.

Local interface (ILB)

Used to interface a PTP200N or a PLC200 card with I/O expansion cards (INOUTR and IOMOD).

Serial interface RS232

Used for serial communication between the PC and the central unit. The link is ring-type, i.e. the PC transmission pulse is linked (through the connector on basic card) to the reception pulse of the first card on the left (MASTER); the pulse from MASTER is linked to the reception pulse of the second PTP200N card, and so on.
The pulse is finally sent to the PC receiver by the last PTP200N card in the rack.

3. INSTALLING SYSTEM CNC90

SUPPLY OF SYSTEM CNC90

System CNC90 is supplied with a set of 1.4 Mb, 3.5" floppy disks and it can be used with operating system MS-DOS 3.3 and subsequent versions.

The disks are 3 and are named CNC90 INSTALLATION DISKS.

Each installation disk is assigned to a "label" (label means a programmed word on the same disk).
The first two disks (label: "DISK-1" and "DISK-2") contain installing programs and basic CNC90 programs; they are the same for any installation. The third disk (label: "CUSTOM") contains configuration programs of the specific machine.

Disks can't be used for MS-DOS operative system installation and therefore it must be installed separately. Usually, if the PC is supplied by TPA, the operative system is already installed on hard-disk, in a dedicated directory.

PTP1000 SYSTEM installation disk is supplied together with CNC90 installation disks. It is a 1,4 Mb/3.5" disk named OS1000 SYSTEM INSTALLATION DISK.

CNC90-INSTALLATION DISK 1 DISK LABEL : "DISK1"

This disk contains the following files:

INSTALL.EXE	system installation
DISINST.EXE	system update
LINGUE.MNU	selection national idioms for installing program
MESSINST.ITA	messages for installing procedure (in Italian)
MESSINST.ENG	messages for installing procedure (in English)
MESSINST.DEU	messages for installing procedure (in German)

and directories:

..\DOS

SHARE.EX@	file MS-DOS version 5.0
HIMEM.SY@	file MS-DOS version 5.0
RAMDRIVE.SY@	file MS-DOS version 5.0
SMARTDRV.SY@	file MS-DOS version 5.0
SHARE.3X@	file MS-DOS version 3.3
HIMEM.3Y@	file MS-DOS version 3.3
RAMDRIVE.3Y@	file MS-DOS version 3.3
SMARTDRV.3Y@	file MS-DOS version 3.3
SHARE.4X@	file MS-DOS version 4.0
HIMEM.4Y@	file MS-DOS version 4.0
RAMDRIVE4SY@	file MS-DOS version 4.0
SMARTDRV.4Y@	file MS-DOS version 4.0

..\CNC90

*.DR@	icons for auxiliary graphic symbols
FONT1.@	character 1 fonts
FONT3.@	character 3 fonts
WORK.FO@	graphic fonts for CNC90 first page
IMP.BA@	auxiliary file batch
SALVAINA.BA@	batch file for updating parameters on installation disk
SALVAINC.BA@	batch file for updating parameters on hard disk
WORK.EXE	initial paging of CNC90
CNC90CAD.EXE	CAM90-TPA start up
ESTERNO.EXE	auxiliary program for custom procedures
ESTERNO2.COM	auxiliary program for custom procedures
DISCO12.EXE	auxiliary program for disassembling compiled programs
MESSAGES.EXE	auxiliary program for messages file
PACKTIF.EXE	elaboration of files in "TIF" and "BMP" format
SPY.EXE	diagnostic program of serial line
COMUNICA.EXE	comunication by means of serial line
IMPLODE.EXE	files compression
EXPLODE.EXE	files de-compression
LEGGI.EXE	read shuttle file
VENTOSE.EXE	clamping management
EDIGRAF.EX@	programs editor
GRAFSET.EXE	editor configuration
EDISTMP.EXE	graphic print
DEBMSG.CO@	debug of messages
SPY.HL@	help for spy.exe

..\CNC90\LINGUE

Number of files within this directory depends on the language installed.

..\PTP1000

TSIMB.(Ing) messages file, in national idioms, for the Menu of system auxiliary program

..\DV

contains configuration files for DESQview 386 program, for multi-task and multi-module operation. Specifically :

directory DV\DV_1M	configuration files for single-module machines
directory DV\DV_2M	configuration files for two-modules machines
directory DV\DV_3M	configuration files for three-modules machines
directory DV\DV_4M	configuration files for four-modules machines.

..\STARTUP

contains customised system files, if existing (AUTOEXEC.BAT, CONFIG.SYS).

CNC90-INSTALLATION DISK 2 DISK LABEL : "DISK2"

This diskette contains the following directories:

..\CNC90

PRINTDIR.EXE	printout of programs directory
GO-CNC90.EXE	CNC90 bootstrap
PLANCIA.EXE	machine control
EDILIST.EXE	editor list
COMPILA.EXE	compiler
EDIPAR.EXE	heads parameters control
EDIPART.EXE	technological parameters and tooling control
COPIA.EXE	file management
MANDIAG.EXE	manual operating control

..\MATCNC90

DIGISET.EXE	configuration of peripheral units
POINTING.BAT	launch of mouse driver
DIGIVGA.DE@	driver for VGA graphics
DIGIEPSN.DE@	printer configuration
DIGIHEW.DE@	printer configuration

CNC90-INSTALLATION DISK 3 LABEL DISK : "CUSTOM"

This diskette contains the following directories:

..\CNC90 :

*.EXE	configured executables

..\CNC90\MOD1\USER

*.PAR	configuration and data files for CNC90
HEADLNK.@	graphic help for CNC90 Editor
SUBLNK	graphic help for CNC90 Editor

..\CNC90\MOD1\MASK

MASKLNK.*@	graphic help for CNC90 Editor
EDILNK.@	graphic help for CNC90 Editor

..\CNC90\LINGUE

ERRSHELP.*@	System error files in national language

..\PTP1000

PTP1000 environment files

..\PTPSYS
..\PTP0

directories of PTP1000 environment

For what concerns files description refer to WORK DISK Chapter.

CNC90 SYSTEM INSTALLATION AND SET-UP

The installing procedure may be used for two purposes:

1) To install system on hard-disk.

2) To update the version already installed on hard disk.

In order to install the full program CNC90, perform in sequence the following operations:

1) install first PTP1000 using the corresponding installation diskette

2) then install CNC90

PTP1000 installation is synthetically described in the final part of this Chapter; for a detailed description, see PTP1000 system operating manual.

CNC90 is installed by running INSTALL program. This program will ask the user to introduce all information required.

Notice Use of the installing procedure is compulsory, as all files on the diskettes are in compressed format, and are converted into executable files during the installing procedure.

HOW TO EXECUTE INSTALL PROGRAM

To install system CNC90 follow instructions below.

1) Select drive A:

2) Insert "DISK 1" in drive A:

3) Digit: A:INSTALL and push ENTER

4) Select installing language, as asked in the first dialogue screen.

The following message is displayed:

Installation or Up-date processing ? (I/U) : I

5) Select I to install CNC90 or U to update the system .

When indicated, You may press key ESC to quit the installation.

6) In case of Up-date:

a new screen appears, proposing choice between System Updating (first two installation diskettes) and Custom Updating (third installation diskette).

Then this window appears, allowing selection of path names on the hard disk:

CNC90 System on disk/directory: C:\CNC90
PTP1000 System on disk/directory: C:\PTP1000
DOS System on disk/directory: C:\DOS
CAD System on disk/directory: C:\MATCNC90

confirm these paths or specify new ones with reference to:

- CNC90 System
- PTP1000 System
- Operating System
- CAM90-TPA System (in case this optional package is not installed, confirm the path proposed)

7) In case of **installation:**

user is asked if the procedure should execute a copy of some files of operating system, necessary for CNC90 programs running :

System files copy in directory DOS (Y/N) : N

3.8

Answering NO: no copy is executed of these files, but they must be found on PC;
Answering YES: a copy is executed, as specified in a configuration file contained in the installing procedure (MESSINST.LNG).

The installing procedure will then verify the existence of PTP1000 Software on the PC; in case of non-existence, it will prompt the user with a warning message.

Presence of PTP1000 system on installation disk is verified through the existence of file ENVIRON.TPA, within the directory confirmed or assigned in the window described above.

After this, an option menu is displayed :

SET-UP DATA	
Language (ITA/ENG/FRA/TED/ESP):	ITA
Number of Modules to be installed (1/2/3/4):	1
Start up from AUTOEXEC.BAT (Y/N)	N
MULTITASK installation: (Y/N)	No
Archive on network server: (Y/N)	No

Language (ITA/......)
Select the national language preferred for CNC90 start up.

Number of Modules

Start up from AUTOEXEC.BAT
In case of installation starting from AUTOEXEC.BAT: if file is already existing, the following message is displayed :

Modify or create AUTOEXEC.BAT : **MODIFY**

User can choose if create a new AUTOEXEC.BAT or modify the old one : untrained users are suggested to opt for creation.
Program then executes modification of system files AUTOEXEC.BAT and CONFIG.SYS.

If automatic bootstrap of CNC90 from AUTOEXEC.BAT is not required, installation procedure creates files C:\CNC90.BAT and CONFIG.TPA, with the necessary settings for a correct running of CNC90. Setting configuration must be performed by user on his personal computer so that CNC90 programs may work.

MULTITASK installation

Archive on network server
This option is only offered when a number of disk units greater than 4 appears to be present on the PC. In case of archive installation on a server unit, the user will be asked to specify disk name.

During installation procedure, the operator will be asked to change diskettes in drive A.

The following graphic display will announce the end of installing procedure

SYSTEM PROGRAM INSTALLATION COMPLETED

INSTALLED DIRECTORIES :

PTP0

PTPSYS

Also the following messages will be displayed:

OPERATIVE SYSTEM SHOULD BE INSTALLED IN THE DIRECTORY : C:\DOS AND THE FOLLOWING FILES SHOULD EXIST :

SHARE.EXE HIMEM.SYS RAMDRIVE.SYS ANSI.SYS COUNTRY.SYS

THESE FILES CAN BE COPIED FROM INSTALLATION DISK1, FROM DIRECTORY A:\ROOT

Installation requires use of an **extended memory PC** (memory EMS) used to configure a virtual 200 Kbytes disk.

This justifies the requirement of file installation on operative system:
RAMDRIVE.SYS corresponds to installation drive on virtual disk.

In case the PC has no memory extension the following operations are required:

1) In file C:\CONFIG.SYS:
   delete line DEVICE=RAMDRIVE.SYS 200/E

2) IN file C:\CNC90\CNC90.BAT:
   replace line: SET VDISK=D
   with: SET VDISK=%ROOT%\MOD1\TEMPOR

CNC90 installation procedure **renames** files AUTOEXEC.BAT and CONFIG.SYS existing in C:\ to AUTOEXEC.BAK and CONFIG.BAK.

The corresponding files suitable for CNC90 environment are copied onto C:\.. If other applications are installed on the PC, check the renaming and the new files AUTOEXEC.BAT and CONFIG.SYS copied.

UPDATING INSTALLATION DISKS

File DISINST.BAT allows performing a complete copy of

a) system CNC90 from hard-disk C: to INSTALLATION DISK 1 and 2.

or:

b) CUSTOM disk updating

Notice This procedure allows updating basic CNC90 programs on the installation diskettes; it is normally used in case of preliminary installation, and can't be used for updating diskettes containing old version of CNC90.

In case of CUSTOM disk updating, the following directories are copied from hard disk into installation disks:

- CNC90
- CNC90\LINGUE
- CNC90\MOD1\USER

and also the following files:

- C:\CNC90\MOD1\PTPSYS\ERRSYS.LNG
- C:\CNC90\MOD1\PTPSYS\VIDEOGPL.LNG
- C:\PTP1000\TSIMB.LNG

This procedure allows updating machine parameters on the CUSTOM disk.

Notice Take care of the CUSTOM disk and Keep it updated; in case of damage of the hard disk, this will be the only source from where to recuperate all parameters. Making a copy of the CUSTOM disk is a suggested practice.

INSTALLATION WITH MULTITASK

CNC90 system can run in multitask mode, that is performing different operations at the same time.
In particular, programming (Editor-CNC90 operating mode) is allowed while control executes workings cycles on the machine.
This operative mode is usually defined as two-task operation; of the two tasks:
- one is reserved to Editor
- the other handles complete operation

This working mode is supported by using a trade management program on multitask processes : DESQview 386.
DESQview 386 program must be bought separately and installed on personal computer 386, in the respect of licence conditions.

If multitask mode is required, more restrictive conditions must be fulfilled when choosing personal computer.

a) computer must be 386 or more powerful
b) it must have 4 Mbytes of RAM memory

DESQview program must be installed before CNC90. The following installation sequence must be followed:

a) operating system
b) DESQview program
c) PTP1000 and then CNC90 programs

If multitask mode is required, during CNC90 installation user must specify it, when asked.
In this case an additional disk will be used, arranged for DESQview 386 program configuration.

INSTALLATION IN MULTIMODULE MODE

Multimodule installation is required when two or more machines must be controlled by the same computer.
During installation, the number of modules (up to a maximum of 4) shall be specified.
The installing procedure will then require a number of CUSTOM disks equal to the number of modules; correspondence between disks and modules is recorded on the disk labels as follows:

- "CUSTOM" for module 1
- "CUSTOM2" for module 2

- "CUSTOM3" for module 3
- "CUSTOM4" for module 4

INSTALLING SYSTEM PTP1000

This installing procedure has two purposes:

1 - install system on the hard disk
2 - update the existing version

Installation is accomplished by following the sequence described below.

1) Select drive A:

2) Insert INSTALLATION disk in drive A:

3) Type on keyboard: A:INSTALL and press ENTER

4) Select installation language, as asked on the display.

The following message is displayed on the screen:

"System on disk/directory : C:\PTP1000

Define disk and directory on which system PTP1000 should be installed, or updated. Normally, confirm with ENTER on C:\PTP1000.

5) Confirm all default selections proposed afterwards

6) User is then asked whether the installation should be carried on or stopped; by answering NO, the program ends without installing PTP1000.

7) Otherwise, by answering YES to the previous point, PTP1000 system installation starts on PC hard disk.

STRUCTURE OF FILE ENVIRON.TPA

File ENVIRON.TPA is automatically generated on the hard disk during the first execution of the installing procedure.

This file contains data concerning the installation of system PTP1000 and the environment in which TPA programs work. Environment data can then be modified by using INSTALL, except the ones concerning colours for which direct editing on the file (ENVIRON.TPA) is necessary; this operation may be performed with any Editor.

File ENVIRON.TPA is a sequential ASCII file and contains:

data on WORKING DISK:

SYS = name of system directory
0 = name of module working directory

data for batch file PTP1000.BAT:

LINGUA: (= extension name for message file MESSPTP), indicating the language used in the file.
Languages actually installed are:

ITA Italian
ENG English
FRA French
DEU German
ESP Spanish

installation data:

AUTOEXEC= displays whether file AUTOEXEC.BAT shall load automatically the system when starting the PC (AUTOEX=1) or not (AUTOEX=0). In case of system PTP1000 installed in environment CNC90 the value is 0.

OS1000 operative system data

VIDEO=type of screen (Monochromatic [BW]/Colour [CO]).

COLOUR= [n. data], [background colour], [foreground colour]

where:

1) n. data is the data number to which colours are referred

2) background colour is the background colour expressed in hexadecimal.
3) foreground colour is the foreground colour expressed in hexadecimal.

Here below the basic colours are listed:

N.info	Code	Backg. col.	Foreg. col.	Description
0	07	BLACK	WHITE	main colour
1	0F	BLACK	DEEPWHITE	alternativecolour
2	87	GREY	WHITE	keys area
3	6E	ORANGE	YELLOW	user error messages
4	8A	GREY	DEEPGREEN	keys available
5	2F	GREEN	DEEPWHITE	VIDEO instr messages
6	4E	RED	YELLOW	system error
7	6E	ORANGE	YELLOW	cycle error messages
8	13	BLUE	LIGHTBLUE	box frame
9	1E	BLUE	YELLOW	box head line
10	1A	BLUE	DEEPGREEN	box main colour
11	17	BLUE	WHITE	box alternative col.
12	3F	LIGHTBLUE	DEEPWHITE	main colour input
13	1F	BLUE	DEEPWHITE	alternative col. input
14	3F	LIGHTBLUE	DEEPWHITE	reverse
15	1C	BLUE	DEEPRED	first menu digit
16	2E	GREEN	YELLOW	help
17	CF	DEEPRED	DEEPWHITE	line 1 on screen.
18	30	LIGHTBLUE	BLACK	function keys title
19	6E	ORANGE	YELLOW	function key pressed

If, for instance, user wants to modify main colour (0 in the list) in order to display BLACK on a WHITE background, he should insert in file ENVIRON.TPA the line: COLOUR = 0,70.

Codes of available colours are:

BLACK	0
BLUE	1
GREEN	2
LIGHT BLUE	3
RED	4
MAGENTA	5
ORANGE	6
WHITE	7
GREY	8
DEEP BLUE	9
DEEP GREEN	A
DEEP BLUE	B
DEEP RED	C
DEEP MAGENTA	D
YELLOW	E
DEEP WHITE	F

4. WORKING DISK

SYSTEM CNC90 WORKING DISK ORGANIZATION

It is intended as working disk (abbreviated eventually in WD) the place where all CNC90 system files are stored, together with user's files: in practice, WD is a section of hard-disk.

On WD it is created a general CNC90 directory, named **System Directory**.
System Directory it is then divided into several branches, each corresponding to a file typology.
Directories are named according to system standards, and they are:

\CNC90	system directory
\CNC90\LINGUE	language files directory
\CNC90\MOD1\USER	parameters (technology, heads, toolings) and configuration files.
\CNC90\MOD1\TEMPOR	temporary files
\CNC90\MOD1\LISTE	listing files
\CNC90\MOD1\SORG	working programs directory file.
\CNC90\MOD1\SORG\SRG	working programs in ASCIII form
\CNC90\MOD1\SORG\TAB	working programs in binary form
\CNC90\MOD1\SORG\DGT	working programs generated from CAM90-TPA
\CNC90\MOD1\COMP\NOC	routine compiling on programs
\CNC90\MOD1\COMP\SPC	specular compiling on programs

In **System Directory** are stored all CNC90 system files with extension:

EXE data processing files
BAT batch files
DRW graphic icones file for CNC90 menu.

In directory \CNC90\LINGUE are stored all messages files in the following languages:, the extentions are:

ITA messages in Italian
ENG messages in English
DEU messages in German
FRA messages in French
ESP messages in Spanish
DAN messages in Danish
FLM messages in Dutch.

In directory \CNC90\MOD1\USER are stored files:

GESARC.PAR	configuration file for files management
GESEDI.PAR	configuration file for programs editor
GESDIMOD.PAR	configuration file programs editor
DEFTEC.PAR	definition file for technological parameters
PARTEC*.PAR	technological parameter files
DEFATT.PAR	definition file for toolings parametric cycles.
PARATT*.PAR	toolings parametric file
TESTINE.PAR	heads parameters file
EDILNK	graphic icones files on editor programs
MASKLNK.*	graphic icones files on editor programs
SUBLNK.*	graphic icones files on editor programs
HEADLNK.*	graphic icones files on editor programs

In directory **\CNC90\MOD1\TEMPOR** are stored temporary files concerning different CNC90 data processing.

In directory **\CNC90\MOD1\LISTE** are stored definition files on working lists. These are ASCII files defined by extension LST.

In directory **\CNC90\MOD1\SORG** is stored the working program directory file. The directory is a binary file named INDICE.PRG

PTP1000 SYSTEM WORKING DISK ORGANIZATION

Other directories are created corresponding to PTP1000 system manegement.

These directories are named according to standard names, and they are:

\PTP1000	system PTP1000 directory 1
\PTPSYS	system directory 2
\PTPO	module directory

In system directory 1 \PTP1000 are stored all files with extension:

EXE	PTP1000 processiong files
TPA	PTP1000 auxiliary files
CMP	PTP1000 auxiliary files
BAT	batch files
ITA	

In system directory 2 \PTPSYS are stored the following files:

CONFSYS.PAR	system configuration data
OPTSYS.PAR	system options data
OPTMOD.PAR	module options data
TAMP.DAT	picture on buffers mode; used for retransmissions.
FKEY.DAT	function keys assigned to Editor by the user.

In module directory \PTPO the following files are stored:

CONFGEN.PAR general	module configuration data
CONFSTAZ.PAR	module stations configuration data
CONFCMP.PAR	compiled configurations data; these are the parameters sent to stations.
DESCNT.PAR	descriptional data on counters; existing only when using the option "Describe counters" in configuration.
DIRSOR	directory for user-created files and for other files : Programs, I/O Definitions, Functions, etc.

NAMING FILES

The name of a file generated with system PTP1000 is composed by the section NAME + the Station number to which the file belongs; amidst the two sections there will be a dot (.).
Example: SETP.0

where SETP is the NAME section and 0 the Station to which the file belongs.

A name can be made of up to 8 alphanumerical digits.
During storage the system adds to file name also its type, and the station number is converted into hexadecimal.

If a DIR command is entered under MS-DOS, the file will be displayed as: SETP.0S (where S indicates file type Setpoint).

If the name should be: SETP.15 : it would be obtained : SETP.FS

Some files need a fixed name to be recognized by the system. They are:

SETP	Setpoint	(filetype S)
DEF	I/O Definitions	(filetype D)
ERRCYC	Cycle errors	(filetype E)

naming for different filetypes is free.

FILES TYPOLOGY

The user can create in system PTP1000 files of variuos type. When stored on disk, their names are inserted in a directory where some data are automatically added:

comment to the file
date of last modification
symbol * to indicate if compiled, in case of programs or functions.
byte number occupied in card RAM memory by the file filetype.

File types are the following:

A Automatic program; it is the processing cycle.
S Setpoint program
F Function file
D I/O definition file
Q table of values
E Cycle error file.
R Report file
T Generical text file

Filetype definitions are displayed in column T of Directory display.

The partition into different types of files allows an easy access from any mode to the desired filetypes list by means of menu:

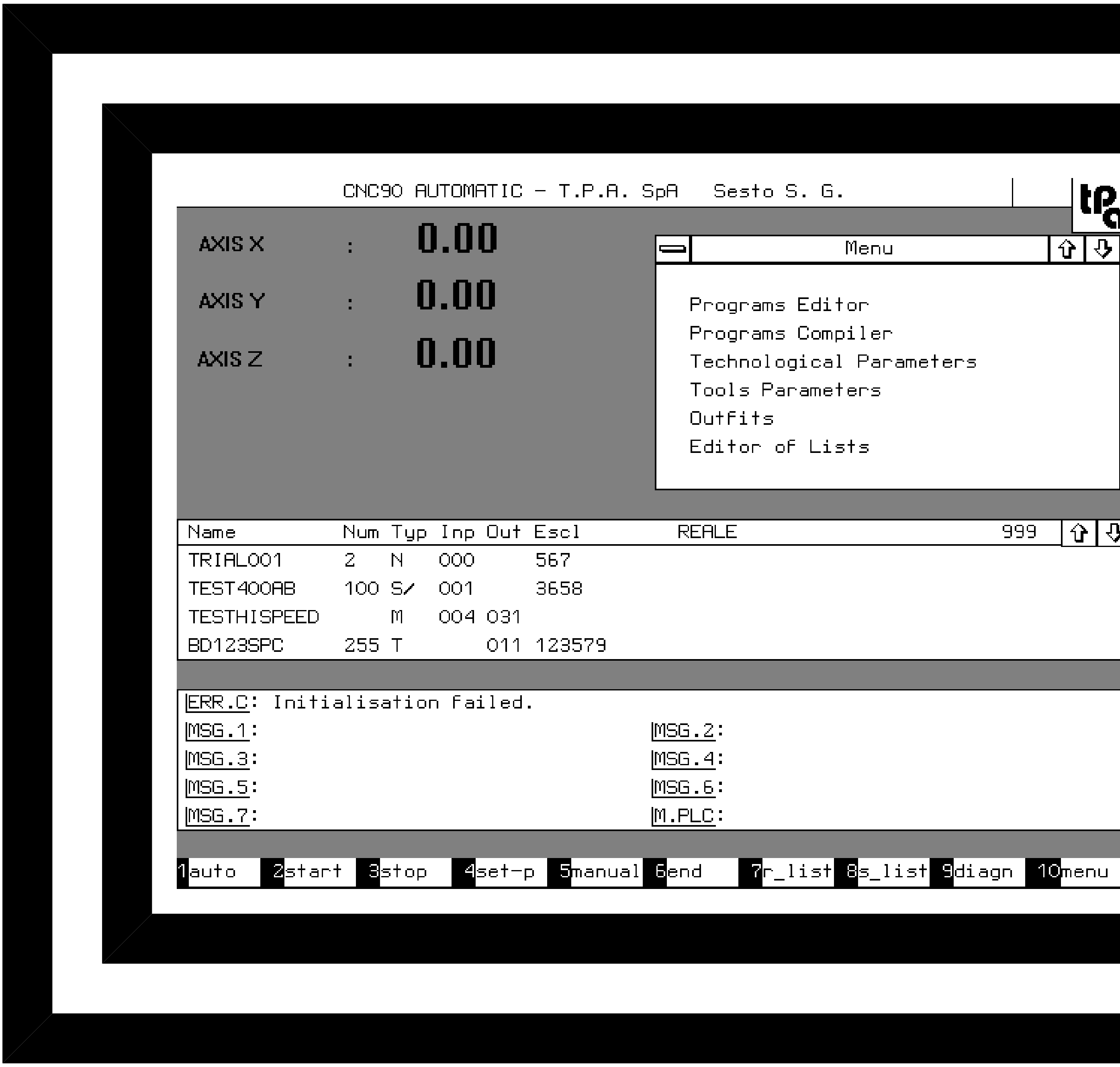
Program
Setpoint
Function
I/O definitions
Tables of values
Cycle error
Report
Text
Directory.

5. DESK OPERATIVITIES (D.O.) MODE

GENERALITIES

As already mentioned, man/machine interface is a PC and software management is performed by CNC90 system through a simple and clear graphic interface.

Here below it is shown a tipical CNC90 D.O. mode display parted in its operative sectors.



5.1 **INITIALISATION**

Once the system has been started, after introduction, it activates Desk Operativity that runs all stations contained in the module linked to PC. This operation can last from few seconds up to about two minutes in more complex systems.
However, D.O. is the mode assigned to system start-up, i.e. PC and modules linking.

At this stage, all operations determining system starting will be displayed on line marked by MSG.1. They are:

- Parallel interface test : checks the integrity of device enabling quick communication among system stations.

- Parameters transmission : transfers system general parameters such as: General Parameters, Axes Parameters, I/O expansions, Interpolation Parameters, Optional Parameters, and Emergency Tables.

- Flags transmission to PLC : transfers PLC flags defined in General parameters to all station composing the module. The three bytes representing the flags are inscribed respectively in ports 240, 241, and 242.

- Reset communication port : if the port has been instructed, it will reset the port indicated as output port for multimodular connection.

- Mask transmission to enabled axes : transmits a byte, representing the mask, on the five bit belonging to the station enabled axes, in port 243 (only the first five bit are significant and represent respectively, starting from the less significant, axis X, Y, Z, W, V). This mask can be tested in program SETPOINT, described later, to check which are the enabled axes and therefore on which set point should be run.

- Functions transmission : transmits all machine cycle functions written in System Auxiliary, to the stations. If, among the functions, the function Parallel Task is defined and enabled for definition on a integrated PLC (Programmable Logic Control) program, its execution is automatically started at the end of function transmission to the station.

-Linearity Correctors transmission : transfers linearity correctors to stations controlling the axes (an exponential value void in the correctors of one axis disables transmission, and therefore the use, of linearity correctors on that specific axis).

If initialisation fails, it will be displayed in line Cycle Errors (ERR.C) the message INITIALISATION FAILURE which, together with a second message displayed on the line immediately below (MSG.1) invites the operator to retry or abort initialisation. If the system has not been initialised the message INITIALISATION FAILURE will remain displayed.

In any case this imply a state of error in cards connections. Check serial connection between PC and PTP200N card.
Only if the initialisation stage (i.e. connection with card verified and functioning) is successfully accomplished it is possible to run processing programs, set point, diagnostic, and manual control.

Once initialisation has been performed, the system is ready to interact with commands sent by the operator by means of the PC keyboard or other peripheral linked to it.

5.2 MAIN DISPLAY

D.O. main display is, as already shown in picture 5.0, parted in different operating zones.

The screen is infact mainly divided into three large windows (pict. n. 5), each with specific functions. Two windows have the same size and are positioned in the lower part of the screen. The first of them, higher on the screen represents the Automatic window, while the lower represents the Messages window. Both windows will be fully described later.

The third window lays on the right side of the upper screen and represents the Menu window, which allows to activates the different operativities. Also this window will be described in a different Chapter.

In the first line, red in a colour screen, are displayed some general information, among which current mode name: in this case the name will be AUTOMATIC.

The last line displays the functions activated by function keys, described shortly here since they will be fully discussed in the indicated sections.

F1 - AUTO (automatic control)

Allows to enter Automatic mode, and therefore to enable the Automatic window to edit and run a working programs list (see chapter 6).

F2 - START

Allows to start basic system functions like Set-point, Diagnostic program DIAGN, or the programming sequence defined in Automatic.

F3 - STOP

Stops the execution of all programs, axis in motion stops executing a deceleration ramp. Re-starts by pressing again function key together with command START.

F4 - SET_P (setpoint)

Allows to use SETPOINT mode.

F5 - MANUAL (manual control)

Allows to access MANUAL and DIAGNOSTIC mode, used for manual axis motion and for I/O control of active stations.

F6 - END

Ends execution of all programs running on stations, either working, set point or diagnostic (DIAGN) programs.

F7 - R_LIST (load list from directory)

Allows to load a programs list from directory and run it in Automatic.

F8 - S_LIST (store list in directory)

Allows to store in lists directory a list running in Automatic mode.

F9 - DIAGN (diagnostic program)

Allows access in diagnostic programs Control.

F10 - MENU (Menu control)

Enables the Menu window and controls Main Menu mode.

Finally, in the left side of the upper section of the screen are displayed names and values of the axes in the enabled station. If the enabled station is a PLC station or it does not have axes, this section will be empty.

5.3 MESSAGE WINDOW

The message window (picture 5.1) is the section of the screen displaying the messages sent from the various stations and from the system.
Messages displayed can be of many kinds, from cycle error messages to instructions for the use of a given station, to others requiring keyboard operations.

ERR.C: Inizializtion failure	
MSG.1 :	**MSG.2 :**
MSG.3 :	**MSG.4 :**
MSG.5 :	**MSG.6 :**
MSG.7 :	**M.PLC :**

Pict. 5.1 Messages window

In the first string, headed ERR.C, cycle errors, coming either from the stations or from the PC are displayed.

To allow the stations to display a cycle error it is necessary to enter, in machine cycle functions, the calls to command ERROR of GPL1000, only when it is necessary to display the error to user.

In this string is displayed also the message INITIALISATION FAILURE when module initialisation has failed for some reason.

The other strings headed from MSG.1 to MSG.2 and from M.PLC are used to display messages (VIDEO o MESSAGE) coming from the stations. In string MSG.1 are displayed the VIDEO without control characters and sometimes also system messages, while in MSG.2 are displayed only messages coming in from the video having as first character the control character: "|".

Example: VIDEO |VIDEO DISPLAYED IN MSG.2

The M.PLC string is used to display messages coming in from PLC stations or from PLC functions; to enable this function it is necessary to activate all VIDEO messages with the control character : open bracket: "(".

Example: VIDEO (VIDEO DISPLAYED IN M.PLC

5.3.1 INTERNATIONAL MESSAGES (VIDEOGPL.Ing)

Besides standard management of VIDEO GPL, a new messages management is now available, according to the selected language in CNC90.
This new procedure utilises a file called VIDEOGPL.ING (where Ing means current language es. ITA, FRA, ENG...) allocated in the directory of TPT1000 Auxiliary System, version 4.3 (in general C:/PTP1000). Each file line represents a message that can be called abd displayed by GLP VIDEO and MESSAGE instructions. It is a test file editing by any text editor (es. MS-DOS EDIT or PC-DOS EDLIN).

Let's see now in details the structure of this file.

In each line words in inverted comas represent the displaying message, the words not included in inverted comas are used only to make clearer file reading. The largest number of messages included in the file is 300, and the maximum length must respect instruction VIDEO limits.

At this point if user would display the message Second Message it would be sufficient to enter message number 2 displaying command, in the correct way, to VIDEO or MESSAGE instructions; and according to the selected language it would be displayed.

5.3.2 MESSAGES ADDRESS

Let's see now how to address the different messages coming from cards into lines of Messages Window. Each message can be addressed in 8 different positions coming from MSG.1 to MSG.7 AND IN ADDITION m.plc.
Let's see in details how this operation is executed, before by VIDEO instruction and then by MESSAGE instruction.

To address a VIDEO message into a particular line of the window it is sufficient to put the characters #n before the message, where # is a control character and n letter can have different values from 1 to 8 (one for each position).
Therefore if user would display the message "Hallo" in the position MSG.4 he should write:

VIDEO #4Hallo

at the same way if he would display the same message in the position M.PLC he should write:

VIDEO #8Hallo

If on the contrary user would use GLP message instruction and display the message number 100 in the position MSG.4 he should write:

MESSAGE 100,4

at the same way if user would display the same message in the position M.PLC one should write:

MESSAGE 100,8

5.3.3 SPECIAL USE OF VIDEO AND MESSAGE INSTRUCTION

At this point we can see other uses of these commands.
To make clearer the use of these GPL instructions let's see some examples.

We will consider the same language file (VIDEOGPL.Ing) of chapter 5.3.1.
let's see the use of VIDEO instruction.

Let's display the message Second Message in the position MSG.7.
To address the message in the wished position, place the characters #7 before the message as explained above; on the contrary to select the message to display included in language file, replace message by the characters &nnn. Where & (trade character) is the control character, and nnn is the address of the message included in language file (value from 1 to 300).
This is the correct way to input VIDEO instruction to display the message Second Message.

VIDEO #7&002

or

VIDEO #7$2

at the same way the wished message was tree hundredth and last message in the position M.PLC write:

VIDEO #8&300

Likewise if MESSAGE was the wished instruction, the result was:

MESSAGE 2,7

to display the message Second Message in the position MGS.7:

MESSAGE 300,8

and for the message three hundredth and last message in the position M.PLC.

The difference between the two instructions is in the orders of their parameters; in VIDEO first positioning address is displayed and then message address; in MESSAGE on the contrary first message address is displayed and then positioning address.
Besides in MESSAGE control characters are not used, but only a separating character (coma) allowing to distinguish and separate the tow parameters.

5.4 SET POINT

It is intended with SET POINT the preliminary operation allowing the machine to realise in any moment the absolute position of its axes. Since this data is lost any time the machine is turned off, it is very important to perform again this operation immediately after the control stations initialisation stage.

Set point defines the machine initialisation procedure such as required by the machine itself according to its electro- mechanical configuration.

Generally speaking, a proper setting of the machine includes:

- Setting on machine output signals

- Checking machine input signals

- Resetting positions on all controlled axes by automatic search of reference point zero.

Finally, SET POINT is a necessary operation to restore machine mode after the display of any error message related to axes control (servo-error, uncorrect encoder connection, and in general any system error) or to machine emergency.

Function key F4 enables set point procedure on all enabled axes and displays function keys available in this procedure.
Afterwards, function key F2 starts set point execution, which can be suspended or stopped any time by means of function keys F3 and F6.

If the machine is specified to be multitoolings in Setting Technological Parameters (the value existing in Parameters is different than zero), at the end of the procedure it will be displayed a window (pict. 5.2) requiring to enter the tool data existing on the machine; this data will then be related to toolings selected for each program.

```
-----------------------------------------------------------------

  Enter tooling number existing on machine ... : 35

-----------------------------------------------------------------
```

SET POINT FUNCTION KEYS

F2 - START (start execution)

Starts execution of set point programs on all stations of every module.
Command START is accepted only if proper set point programs, included in serial ring, are found in all stations.

F3 - STOP (stop execution)

Stops set point execution and the shifting axes stops according to a deceleration ramp.
Re-start is performed by pressing function key together with command START.
It is accepted only if function key START is indicated.

F6 - END (end execution)

Ends set point execution and quits the procedure.

5.5 RECALL AND SAVE LIST

It is possible to store a list before or after having processed it, so that in the future the user will be in condition of running it in Automatic mode without being forced to re-write it.

Saving and recall are operations performed respectively by means of function keys F8 and F7.

Key **F7** (recall) displays a small window (pict. 5.3), headed r-list, in which are presented in alphabetic order all names of files stored in the directory. Scroll names in the lists by means of keys "arrow down" or "arrow up" or "page up" or "page down".

Once the desired list has been found, position the indicator onto its name and press ENTER to load it.

To abort the procedure, press ESC.

r_list
000-0010
AAPF000
ART100-5
DEMOART
SPF8000
TRIALART
ZULUP40

Name	**Num**	**Typ**	**Inp**	**Out**	**Escl**	**TEST-ALL**
TRIAL001	*2*	*N*	*000*		*567*	
TEST400AB	*100*	*S/*	*001*		*3658*	
TESTSPEED		*M*	*004*	*031*		
BD123SPC	*255*	*T*		*011*	*13457*	

Pict. 5.3

F8 (SAVE) saves (stores) the Automatic mode list in the list directory. Since an Automatic list does not have a name, the first thing to do is to assign a name to the working list. This operation imply to answer to the requirements of a small window (pict. 5.4) superimposed on the Automatic mode window. Type list name and press ENTER.

Insert the name of destination list ... :

pict. 5.4

If a list with the same name is already existing, or if the name is typed incorrectly, an error message warns the user about the aborted storage, explaining the reasons for it.

It is maybe useful to recall that the lists should be named according with DOS specifications about file names. For further information, please refer to DOS user's manual.
Also in this case press ESC to abort the command.

5.6 PROGRAM : "DIAGN"

Diagnostic program DIAGN is a program quite similar to set point, with the difference that it can be used to solve out particular problems connected to machine functioning. The use of this program is not bound to any limit, and it is therefore the manufacturer that should personalise this program for special purposes.

Function key F9 selects program DIAGN on all enabled stations and indicates available diagnostic programs names on function key (DIAGN1, DIAGN2, DIAGN3, DIAGN4, DIAGN5, DIAGN6, DIAGN7, DIAGN8 AND DIAGN).
Therefore pressing function key related to the wished program, program execution is enabled.
Afterwards, function key F2 starts execution, which can be interrupted at any time by pressing function key F3 or F6.

Diagnostic program names goes from Diagn1 to DIAGN8 for what concern function keys from F1 to F8 and DIAGN for function key F9 and they must be written by Auxiliary programs editor of PTP1000 System version 4.3.

PROGRAM FUNCTION KEYS

F2 - START (start execution)

Starts execution of DIAGN programs on all module stations.
Command START is accepted only if station program DIAGN, included in serial ring, is found in all station.

F3 - STOP (stop execution)

Stops execution of DIAGN and stops shifting axes according to a deceleration ramp.
Re-start is performed by pressing again function key together with command START.
However, the command is accepted only if function key START is indicated.

F6 - END (end execution)

Ends execution of DIAGN procedures and quits the mode.

5.7 MAIN MENU

```
==            MENU          ^  v

        Programs editor
        Programs compiler
        Technological parameters
        Tools parameters
        Tools design
        Lists editor
```

Pict. 5.5

From this menu it is possible to access the system's main modes.

Here below it is given a short summary of each mode. For further explanations, please refer to the chapter discussing each voice in detail.

Once function key F10 has been pressed the menu will be displayed with an indicator on the first line of the window. Use the indicator to select the proper voice.

To reach all menu's voices use keys "ARROW UP" or "ARROW DOWN" or "PAGE UP" or "PAGE DOWN". Once found proper mode set indicator onto it and press ENTER.
To abort current mode press key ESC.

Main modes are:

PROGRAMS EDITOR

Allows access to Graphic Video Editor to create or modify working programs (see chapter 10).

PROGRAMS COMPILER

Allows access to Compilation mode to compile one or more working programs (see chapter 11).

TECHNOLOGICAL PARAMETERS

Machine Parameters mode: General Parameters, Groups Parameters, Linearity Correctors, Settings and Custom Parameters.

The selection of a certain group of parameters is performed with the help of a secondary menu window superimposed on Main Menu, in which are listed all kind of parameters.
Select proper parameters by following the same instructions given for Main Menu (see chapter 9).

TOOLS PARAMETERS

Allows access to Tools Parameters mode, to specify all information related to tools configuration (see chap. 7).

TOOLINGS DESIGN

Allows access to Toolings Design configuration mode to define the 10 possible tools design on machine groups. Before accessing Toolings mode a menu window, superimposed on Main Menu, will allow to define the tooling design to configure. Select proper tool design by following the same instruction given for Main Menu (see chap. 12).

LIST EDITOR

Allows access to List Editor to create or modify program lists (see chap. 12).

FILE CONTROL

Allows access to File Control mode to perform all file control operations such as deletion, copy, open, etc. (chap.13).

SYSTEM AUXILIARY

PTP1000 System Auxiliary mode to define all data concerning system software such as: number of stations, general parameters, axes parameters, besides all functions defining machine cycles, set point and Diagnostic programs DIAGN (see PTP1000 user's manual version 4.3).

LOAD CAD

Allows to load a general CAD system, like for instance TPA's CAD system "MATISSE" to create programs with system CNC90. Optional mode : for further information please contact T.P.A. in Sesto S. Giovanni (Milan) - Italy

LINK NETWORK

Allows to link two PCs in a network. The system runs file CONNETTI.BAT containing all commands needed for the operation. File CONNETTI.BAT can be personalised for use on a specific network system.

LOAD EXTERNAL PROGRAM

Allows to run a generic program, eventually supplied by TPA or other software houses.

CHANGE STATION FOR AXES VALUE

Displays a menu, superimposed on Main Menu, to select the station to enable and whose axes coordinates are to display. Select as for Main Menu.

SHELL TO DOS

This mode allows to suspend the execution of program CNC90 and to run one or more DOS commands. However, it should be remembered that in any case it is just a shell to DOS system, and that CNC90 it is still occupying part of the available memory forbidding the execution of many DOS programs is not guaranteed.
To return to System CNC90 digit DOS command EXIT and press ENTER.
Important : NO resident program (TSR programs) should be run when Shell to DOS is loaded, to avoid any malfunction of system CNC90.

QUIT SYSTEM CNC90

After the question :

- Are you sure? (Y/N) :

to which is possible to answer Yes with key Y or No with key N, or with the space bar, the PC returns to DOS operative system. To load again system CNC90 digit C:\CNC90\CNC90 and press ENTER.

If, when confirming, key ESC is pressed, the system aborts the operation and displays again the main display.

MENU MODES PROTECTION

Each mode in the menu can be protected by a password to avoid any nuisance due to improper use of the control.

A window is displayed for key word entering (pict. 5.6)

Enter password to proceed .. :

Pict. 5.6 : password window

Password in this case is WORD, and it keeps valid for all menu's voices. Besides, once it has been entered for a voice it will be stored for all further loading till the PC is turned off.

5.8. RE-TRANSMISSION

Sometimes it can be necessary to re-transmit a second time some parameters normally sent to the stations only during initialisation. These parameters can concern, for example, linearity correctors, emergency tables or other control parameters, and machine cycle functions.
For re-transmission press at the same time keys ALT and R. When this command runs, the stations are re-initialised.

5.9. HELP

Command HELP is enabled during the execution of all main D.O. modes. Press at the same time keys ALT and H (H for help).
To scroll all pages contained in HELP, use keys ARROW UP or ARROW DOWN or PAGE UP or PAGE DOWN. To quit HELP press key ESC.

5.10. MONITOR

Monitor is enabled by pressing at the same time keys ALT and M in the main D.O. display, or in the Automatic Mode window.

For further information, please refer to PTP1000 User's Manual version 4.3.

5.11. OSCILLOSCOPE

Oscilloscope is enabled pressing ALT and O keys at the same time on the main screen display of Operating Desk.
For further information please refer to user's manual of PTP1000 system version 4.3.

5.12 ERROR DISPLAY

For various reasons, ranging from machine malfunctioning, to emergency situations, scarce maintenance, human mistake and more, there can happen some error during the execution of D.O. or Automatic lists.

Error messages are displayed in three different ways, according to the kind of error.

-Cycle Errors: As already mentioned, these errors are displayed in the Message window in D.O. Main Display.
The error concerns machine cycle and other errors about machine functioning in general.

-System Errors: These errors are automatically monitored by the cards and sent to the PC for their display. The reasons for them are many, and range from problems concerning the axes to programs reception. The messages are yellow and are displayed in a red-orange window in the centre of the screen. They give some information about the card and the error monitored.

These messages are here below examined.

By the first string of the window are displayed the hour in which the error took place, the name of the station by which the error took place and the module to which the station belongs and sometimes the name of the task where the error took place. By the second and third string are displayed the error message and suggestions to continue.

The error message is formed by a number preceding the message representing error code number and by the error message itself.

During error displaying it is possible to recall a help screen display to understand error type, and therefore the possible solution. To recall this help screen display press HALT and H keys together.

```
16:07:20 Station n. 0          module 0
Program not in directory
Press ENTER to continue
```

System Error window.

-General Errors: they are red and displayed in a black window appearing at the centre of the screen. They are of different type like System Errors, with the difference that they never concern directly the module and the machine.

```
List 000-001 already existing, storage aborted!
```

General Errors window

Please refer to chapter "SYSTEM ERRORS" of SYSTEM PTP1000 user's manual chapter 4.3 for further information on the various types of errors.

6. AUTOMATIC MODE

Once pressed function key **F1** in D.O. (Desk Operativities) main screen (see picture 5.0) the Automatic window is displayed and a different colour indicates the function key selected.
D.O. is now running according to Automatic Mode.

Automatic mode allows the user to superintend machine/s processes controlled by the system.

Some function keys in this mode are commands acting directly on machine cycles; others allow to access optional operativities.

In this mode, the PC is in constant connection with every module to acknowledge their condition, monitoring the messages received from module and dividing them into error messages or control messages for the user.

Meanwhile, the display provides to update:

-the axes value in the enabled station

- some data on the running programs like its name, overall dimensions, comments, and more.

- the messages generated by VIDEO and MESSAGE instructions entered in machine cycle functions.

- error messages, either Cycle or System errors.

6.1 AUTOMATIC MODE FUNCTION KEYS

F2 - START (start execution)

Starts the execution of a program list in Automatic on all stations of every module.
Re-start programs from the point they stopped if key STOP has been pressed during the process.
Command START is accepted only if the system has been initialized, if Set Point has been properly performed and if the data entered by the operator are correct.

F3 - STOP (stop execution)

Stops the execution of all programs, and the shifting axes stop according to a deceleration ramp.
Re-start is performed by pressing again function key F2 (START).
The command is accepted only if function key F2 START is indicated.

F6 - END (end execution)

Ends the execution of all programs running on the stations and quits Automatic Mode.

F7 - R_LIST (load list from directory)

Loads a list stored in a directory (see chapter 5.5)

F8 - S-LIST (save list in directory)

Loads option Save List (see 5.5)

F10 - DIR (programs directory)

Displays a window containing the programs directory (pict. 6.0). Unscroll the list by pressing keys PAGE UP or PAGE DOWN to see the complete directory.

Pict. 6.0 Programs directory

By means of SPACE BAR and direction keys (ARROW UP or ARROW DOWN) it is possible to select one or more programs to transferover to the AUTOMATIC list. They will be entered above cursor line.
To select a program, position the indicator onto its name in the list and press SPACE BAR. Once selected, press ENTER to end the operation or ESC to abort it.
Once selected, a program is distinguished by a particular character, a small triangle, placed before its name. Select again the program to abort its selection.

6.2. AUTOMATIC LIST EDITING

The first step to create and run an automatic list is to fill it in. These lists are a whole of programs ordered according to users needs. Each list can contain up to 200 programs and can bear a multiplying factor of up to 999.

A list is not only a whole of program names but also of process codes. In fact for every program are displayed, besides its name, five more fields representing respectively the single program **multiplying factor** (a value included between 1 and 255), the **process code** (normal, specular, ...), the **input code** and the **output code** (used in particular occasions, with values from 0 to FFF, to personalize the system) and **exclusion code** (a field of 8 figures with value from 1 to 8).

Once entered in AUTOMATIC it is possible to fill immediately the program list. The editable area is composed by 200 lines (maximum number of programs in a list) and 6 columns (name, multiplying factor, etc.). Since the lines displayed in the Automatic window are only four, use direction keys ARROW UP or ARROW DOWN or PAGE UP or PAGE DOWN to unscroll the entire list.

Each line in the window corresponds to a program contained in the list, and each column of the line represent an editable field.

The first editable field, i.e. the left column, is the field containing the name of the program to write in; the second column contain the multiplying factor of each program, a value included within 1 and 255; the third field displays the processingcode, which is a code composed of three digits to identify 28 different kind of processes (see Table of Processing Codes); the fourth end fifth columns contain respectively Input code and Output code. These codes can be used to personalize the cycles of a certain machine capable of clever handling workpieces processing. The last column contains the sxclusion code where it is possible a maximum og 8 exclusions, numbered from 1 to 8, corresponding to the homonimous field in CNC90 graphic editor.

Exclusion code allows to exclude some program lines from processing, if programmed with exclusion nymber from 1 to 8 (field "Es"on Graphic Editor) .

For instance to exclude processing lines having 2 and 7 exclusion number, in automatic list, compile exclusion code with 2 and 7 value then write 27.

DEFAULT VALUES IN A LIST

Among these six fields, the only one that should always be filled in is Program Name; the remaining four can be omitted. In this case, however, the system provides automatically to assign a value where missing. These standard or default values are here listed:

-Multiplying factor : it is assigned 1.

-Processing code : it is assigned "N" (Normal processing)

- Input code : it is assigned value 000

- Output code : it is assigned value 000.

- exclusion code : it is asigned value 00000000, no exclusion is enabled

In case program name is missing, the system considers this empty field, and automatically deletes the values inscribed in the other five fields.

LIST EDITING COMMANDS

The table below presents the commands allowing the operator to write in, modify, and delete an Automatic Mode list.

These commands are related with a sequence of keys; for example, by pressing key CTRL together with key Y the current line, the one where the cursor is positioned, will be deleted from the list. In effect, some commands relate to a sequence of keys and some to a single key.
Therefore, pay attention to the description of commands here listed: sign "+" between two keys means that they should be both pressed, following the sequence given.

KEYS	COMMANDS
INSERT	Enables INSERT mode, to insert new characters in the text. Press again INSERT to disable the function.

DELETE	Deletes the first character on the right side of the cursor
BACKSPACE	Deletes the first character on the left side of the cursor.
HOME	Position the cursor on the first edited character.
END	Position the cursor on the last edited character.
ESC	Deletes field value.
ARROW RIGHT	Moves the cursor one character right.
ARROW LEFT	Moves the cursor one character left.
PAGE UP	Skip to previuos page.
PAGE DOWN	Skip to next page.
TAB	Shift cursor to the first editable field at the right of current field. If positioned on the last field it returns to the first.
SHIFT + TAB	Shift cursor to the first editable field at the left of urrent field. If the cursor is positioned on the first field, it shift at the last.
ARROW UP	Return to previous line.
ARROW DOWN	Move to next line.
ENTER	Shift cursor to next editable field.
CTRL + D	Empty the whole Automatic list.
CTRL + Y	Delete current line.
CTRL + U	Re-insert above current line the last line deleted by CTRL + Y (UNDO of the LINE).
CTRL + N	Position the cursor by the upper right edge of Automatic window to insert the list's multiplying factor.
CTRL + ENTER	Insert an empty line above current line.
CTRL + HOME	Skip to first editable field in the first line of the list (rapid setting atop the list).

CTRL+S	find a word within a list, starting from the present postition of cursor till list end
CTRL+P	Positons the cursor on upper edge of Automatic Window to insert list prefix (see Chap. 6.9 Processing with Prefix)
F2 (function key F2)	Runs a program list (see chapter 6.7, Automatic start.
F3 (Function key F3)	Stops the execution of a program list (see chapter 6.1)
F6 (function key F6)	Quits Automatic running and stops all programs in execution by the stations.
F7 (function key F7)	Load a list from directory (see 5.5).
F8 (function key F8)	Save in directory an Automatic list.
F10 (function key F10)	Open a window displaying the programs directory with possibility of direct selection (see 6.1).
ALT + H	Load HELP.
ALT + M	Enables Stations Monitor.

In case it should not be possible to run a command a beep will warn the operator. If the beep cannot be heard, like with PC very close by the machines or for any other reason, the operator will realize the command failure only by its outcome, which can be a fully processed workpiece if the command has been run correctly, a partially processed workpiece if the command has been run partially or an unprocessed workpiece if the command has not been run at all.

6.3 DEFINITION OF WORKING AREAS AND WORKING CODES

Automatic execution modes allow control on different working areas (or fields) with the double purposes of optimize processing and execution time on specular workpieces, without having to re- write the programs.

The most complete use of the machine can be made on four different working areas unlimited in the number of possible combinations with adjacent areas. These four working areas are defined as in the following graphic scheme:

*	*	*	*
Area S	Area A	Area T	Area R

The outer frame represent working plane XY.

The four symbols * represent the reference points to be handled as workpiece fulcrums. Starting from the first ref. point on the right, they are identified as:

- Ref. point on processes in N and S

- Ref. point on processes in A

- Ref. point on processes in T

- Ref. point on processes in M and R

The meanings of these processes will be explained later in this same section.

However, to each area is assigned one fourth of working plane XY.

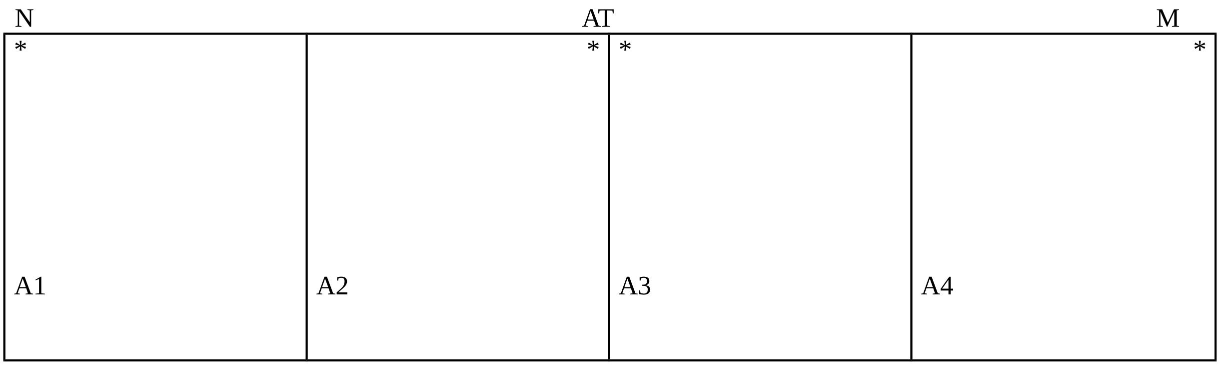
Areas S and T are for normal processing:
the execution of a program follows the program scheme, with axes originated on the left side of the machine and working values of X increasing toward the right side.

Areas R and A are for specular processing:
the execution of a program follows the program scheme speculated on axis X. The origin of the axes will be, during execution, speculated on the right side of the machine, with working values of X increasing toward the left.

6.4 EXECUTION MODES ON WORKING AREAS

The type of executions to perform are defined on the basis of the four areas previously described.

It is possible to specify up to 16 kind of executions, that is all possible combinations of four adjacent areas.
Returning to the working plane XY, this time with wording opportunely completed and modified, executions are here defined:



N long	=	A1	A2	A3	A4	=	N
N short	=	A1	A2	A3		=	N/
S long	=	A1	A2			=	S
S short	=	A1				=	S/
SM short	=	A1				=	SM/
T long	=			A3	A4	=	T
T short	=			A3		=	T/
TM short	=			A3		=	TM/
M long	=	A4	A3	A2	A1	=	M
M short	=	A4	A3	A2		=	M/
R long	=	A4	A3			=	R
R short	=	A4				=	R/
RN short	=	A4				=	RN/
A long	=			A2	A1	=	A
A short	=			A2		=	A/
AN short	=			A2		=	AN/

6.4.1 EXECUTION TYPES N AND N/

Execution type **N (N** long) it is used to process a workpiece along the entire working plane. Also blocking takes place on the whole plane XY.
No contemporary lock on the plane is possible.

Execution type **N/** (**N** short) it is used to process workpieces with lenght limited within the three areas S, A and T.
The workpiece is blocked on this three areas; therefore, it is possible a contemporary lock on area **R.**

In both executions the workpiece rests on ref. point **N** and the process is normal.

6.4.2 EXECUTION TYPES T AND T/

Execution type **T** (**T** long) it is used to process workpieces no longer than the right side half-plane (areas **T** and **R**). The workpiece is blocked on the whole right side half-plane.
It is possible a contemporary lock on the left side half-plane, on a single area or on the whole half-plane (i.e.: **S+A** = **S** long; or **A+S** = **A** long; or single area **S** or **A** defined as short areas).

Execution type **T/** (**T** short) is used to process workpieces no longer than area T.
The workpiece is blocked only on area **T,** and that leaves the possibility of a contemporary lock on the three remaining areas (**S, A,** and **R**), either on a single area or a combination of adjacent areas (i.e. **S+A** = **S** long; or **A+S** = **A** long; or single area **S, A,** or **R** defined as short areas).

On both both processing mode, workpiece rests on rif. point **T** and processing is normal.

6.4.3 EXECUTION TYPES S AND S/

Execution types **S** (**S** long) it is used to process workpieces no longer than the left side half-plane (areas **S** and **A**). The workpiece is blocked on the whole left side half plane.
It is possible a contemporary lock on the right side half plane, on a single area or on the whole right side half plane (i.e. **R+T = R** long; or **T+R = T** long; or single area **R** or **T** defined as short areas).

Execution type **S/** (**S** short) is used to process workpieces no longer than area **S**.
The workpiece is blocked only on area **S**; it is therefore possible a contemporary lock on the three remaining areas (**A, T** or **R**), either on a single area on on a possible combination of adjacent areas (i.e **R+T+A = M** short; or **R+T = R** long; or **T+R = T** long; or single area **A, T** or **R** defined as short areas).

In both execution types the workpiece rests on ref.point **N** and the process is normal.

6.4.4 EXECUTION TYPES M AND M/

Execution type **M** (**M** long) is used to process workpieces as long as the whole working plane. The workpiece is blocked on the entire plane **XY**.
No contemporary lock on the plane is possible.

Execution type **M/** (**M** short) is used to process workpieces with lenght limited within the three areas **R, T**, and **A**.
The piece is blocked on these three areas; it is therefore possible a contemporary lock on area **S.**

In both execution types the workpiece rests on ref.point **M** and the process is specular.

6.4.5 EXECUTION TYPES R AND R/

Execution type **R** (**R** long) is used to process workpieces no longer than the right side half-plane (areas **R** and **T**). The workpiece is blocked on the whole right side half-plane.
Therefore, it is possible a contemporary lock on the left side half plane, either on single area or on the whole half plane (i.e. **S+A = S** long; or single areas S or A defined as short areas).

Execution type **R/** (**R** short) is used to process workpieces no longer than area **R.**
The workpiece is blocked only on area **R**, and that leaves the possibility of a contemporary lock on the three remaining areas (**T, A,** and **S**) either on a single area or on a possible combination of adjacent areas (i.e. **S+A+T = N** short; or **S+A = S** long; or single areas **T, A** or **S** defined as single area).

In both execution types the workpiece rests on ref.point **M**, and the process is specular.

6.4.6 EXECUTION TYPES A AND A/

Execution type **A** (**A** long) is used to process workpieces no longer than the left side half plane (areas **A** and **S**). The workpiece is blocked on the whole left side half-plane.
Therefore, it is possible a contemporary lock on the right side half plane, either on single areas or on the whole half plane (i.e. **R+T = R** long; or **T+R = T** long; or single areas **T** or **R** defined as short areas).

Execution A/ (A short) is used to process workpieces no longer than area A.
The workpiece is blocked only on area A, leaving the possibility of a contemporary lock on the three remaining areas (S, T and R), either on single areas or on a possible combination of adjacent areas (i.e. R+T = R long; or T+R = T long; or single area S, T or R defined as short areas.

In both execution types the workpiece rests on ref.point A and the process is specular.

6.4.7 SPECIAL EXECUTION TYPES SM/, AN/, TM/, and RN/.

These executions are defined "special" because they concern only machines capable of performing the SET POINT on the workpiece top or bottom. This happens usually only in machines exploiting axis X to handle the workpiece.

Executions SM/ and TM/ are equivalent to executions S/ and T/, with the only difference that the process is specular instead of normal.

Likewise executions AN/ and RN/ are equivalent to executions A/ and R/, with the difference that the process is normal instead of specular.

WARNING

The possibility of exploiting the plane and the related working areas as described above correspond to an optimal use of system CNC90 working features, besides a proper use not only of programming and control mediums but also of the informations supplied by the system concerning them.
Basilar, among the most important factors, are:

- Definition modalities of the machine wiring chart (for an effective indipendent control on four different areas should be realized indipendent wirings for control pulses by each area).

- Definition of the functions handling workpiece load and unload stages.

- Working list compiling.

It is anyway possible a limited use of the machine: for example, handling only two indipendent areas (defined on two working half planes).

6.5 EXAMPLES OF AUTOMATIC LISTS

EXAMPLE 1:

P01	1	N	000	000	873
P02	2	M			36458

This list defines two different programs, with executions:

- N long, in program P01.

- M long, in program P02.

Besides, on program P02 is specified a multiplying factor of 2.

First the workpiece is blocked on area N long; for that reason:

- Workpiece rests on ref.point N.

- Locking is enabled on the whole working plane.

It is no possible any contemporary lock on the working plane.

Once the workpiece in area N long has been locked, program P01 starts in normal execution.

Once the workpiece in area N long has been unlocked starts locking in area M long; then:

- workpiece rests on ref.point M.

- Locking enabled on the whole working plane.

It is no possible any contemporary lock on the working plane.

Once the workpiece in area M long has been locked, program P02 starts in specular execution.

Once execution ends and area M long is unlocked, locking in area M long will be enabled again. Infact in program P02 the multiplying factor is specified as 2.

Once the workpiece in area M long has been unlocked after the second execution of program P02 the machine will perform one of the following:

- If the list multiplying factor is 1 the execution ends.

- If the list multiplying factor is more than 1 the execution will start again from the beginning (P01 in area N long), till ending the repetitions of the compiled list.

EXAMPLE 2 :

P01	1	S	000	000	1248
P01	1	T			17
P01		A	000	000	1234567
P01	1	R			

This list is defined on four programs with execution types:

- S long
- T long
- A long
- R long

The program to run is P01 on every line of the list; the repetition factor is 10.

Program P01 will run alternatively 10 times on each of the four specified execution types. The processing sequence will follow the sequence on the list.

For what concerns the sequence of processed areas, it is useful to remember that it is always possible, when processing a workpiece on a sector of plane, to lock the next workpiece on the remaining free sectors.

Once the programs list starts running, locking on both half- planes is enabled in execution mode S (ref.point N) and T (ref.point T).
The process starts first on program P01 execution S as soon as the workpiece has been locked.
The operator can always lock a workpiece to the right side half- plane while the machine is processing the workpiece in area S.
Once the workpiece in area N long has been unlocked the execution of program in area T can be started immediately (after locking); during processing in area T, the operator can proceeed to lock a workpiece in area A (left side half-plane, ref.point A)
And soon till endind the 10 repetition entered.

The process dynamics (area occupations in the machine) could be depicted as follow :

ref.point N

ref.point T

Panel processed in execution "S".

Panel locked and ready for execution "T".

ref.point A

ref.point T

Panel locked and ready for execution "A"

Panel processed in execution "T"

Ref.point A

Ref.point M

panel processed in execution "A".

panel locked and ready for execution "R".

Ref.point N

Ref.point M

Panel locked and ready for execution "S".

Panel processed in execution "R".

6.6 TABLE OF PROCESSING CODES

N long	=	A1	A2	A3	A4	=	N	(2F)
N short	=	A1	A2	A3		=	N/	(2E)
S long	=	A1	A2			=	S	(2C)
S short	=	A1				=	S/	(28)
SM short	=	A1				=	SM/	(18)
T long	=			A3	A4	=	T	(23)
T short	=			A3		=	T/	(22)
TM short	=			A3		=	TM/	(12)
M long	=	A4	A3	A2	A1	=	M	(1F)
M corta	=	A4	A3	A2		=	M/	(17)
R long	=	A4	A3			=	R	(13)
R short	=	A4				=	R/	(11)
RNshort	=	A4				=	RN/	(21)
A long	=			A2	A1	=	A	(1C)
A short	=			A2		=	A/	(14)
ANshort	=			A2		=	AN/	(24)

N long bis	=	A1	A2	A3	A4	=	N1	(6F)
N short bis	=	A1	A2	A3		=	N1/	(6E)
S long bis	=	A1	A2			=	S1	(6C)
S short bis	=	A1				=	S1/	(68)
T long bis	=			A3	A4	=	T1	(63)
T short bis	=			A3		=	T1/	(62)
M long bis	=	A4	A3	A2	A1	=	M1	(5F)
M short bis	=	A4	A3	A2		=	M1/	(57)
R long bis	=	A4	A3			=	R1	(53)
R short bis	=	A4				=	R1/	(51)
A long bis	=			A2	A1	=	A1	(5C)
A short bis	=			A2		=	A1/	(54)

6.7 START AUTOMATIC PROCESSING

Once entered START, the system checks that all data contained in the list are correct; in case of an uncorrect definition, the system warns the operator about the error and sets automatically on the field to modify. The possible errors in the list are described later.

After this phase, the system checks the programs compiled in the programs list. This checkings verify the parametric release, the toolings and the existence of the compiled programs. In case a program should be compiled differently, Automatic mode loads directly the Compiler, which compiles the program and returns to Automatic to run the list.

Once the list is being effectively run, the screen will display, besides the five edited fields, four more non-editable fields representing, from left to right: workpiece size; unit of measure; tooling; and comment to the program.

A red indicator bar by the first line of Automatic window indicates the program currently being run. The red indicator is always on the first line, to distinguish the program currently running from the remaining ones.

During the execution, the multiplying factors are displayed decreasing for each single program and for the whole list. This can help a continuous control on the workpieces to process and already processed. Before passing to next program, the system reset the multiplying factor and refers to the one entered by the operator.

During the execution of a program system error could occur for various reasons ranging from scarce manutention to malfunctioning of the machine, emergency situations, distraction, etc. In this case machine stops working immediately and an error message is displayed in the window System errors. Please refer to PTP1000 User's manual version 4.01, chapter System Errors.

Once the execution of the Automatic list has ended the system quits the Automatic Operativity and return to D.O. until the next command is entered.

6.8 DATA SENT TO CARDS UPON STARTUP

Any time a program is running, the system send to the stations data concerning the current program and the next ones in the list.

These data can be used to personalize machine cycles, for example by a pre-programmed control on machine load and unload or by controlling different working areas (S, A, T, R)

The data sent to stations are the fields entered by the operator during the compiling stage, and represent: multiplying factor of the program; processing code; input code; output code.

Here below it is described to which ports these data are sent.

Multiplying factors

PORT	DESCRIPTION
206	Multiplying factor of the first program in list (0 - 255)
207	Multiplying factor of the second program in list (0 - 255)
208	Multiplying factor of the third program in list (0 - 255)
209	Multiplying factor of the fourth program in list (0 - 255)

Processing codes (see also Table of processing codes)

PORT	DESCRIPTION
216	Type of process of the first program in list
217	Type of process of the second program in list .
218	Type of process of the third program in list
219	Type of process of the fourth program in list

Input codes.

BIT	PORT	DESCRIPTION
0 1 2 3	200	Less significant part or last number of first program input code.
4 5 6 7	200	
0 1 2 3	201	Most significant part or first number of first program input code.
4 5 6 7	201	Less significnat part or last number of second program input code.
0 1 2 3	202	Middle part or middle number of second program input code.
4 5 6 7	202	Most significant part or first number of second program input code.
0 1 2 3	203	Less significant part or last number of third program input code.

4 5 6 7	203	Middle part or middle number of third program input code.
0 1 2 3	204	Most significant part or first number of third program input code.
4 5 6 7	204	Less significant part or last number of fourth program input code.

0 1 2 3	205	Middle part or middle number of fourth program input code.
4 5 6 7	205	Most significant part or first number of fourth program input code

Output codes.

BIT	PORTO	DESCRIZIONE
0 1 2 3	210	Less significant part or last number of first program output code.
4 5 6 7	210	Middle part or middle number of first program output code
0 1 2 3	211	Less significant part or last number of second program output code.
4 5 6 7	211	Middle part or middle number of second program output code.
0 1	212	Middle part or middle number of second program output code

2 3		
4 5 6 7	212	Most significant part or first number of second program output code.
0 1 2 3	213	Less significant part or last number of third program output code.
4 5 6 7	213	Middle part or middle number of third program

0 1 2 3	214	Most significant part or first number of third program output code.
4 5 6 7	214	Less significant part or last number of fourth program output code.
0 1 2 3	215	Middle part or middle number of fourth program output code.
4 5 6 7	215	Most significant part or first number of fourth program output code.

Exclusion code.

BIT	PORT	DESCRIPTION
0-7	221	Exclusion code of running program.

Additional flags for general use:

BIT	PORT	DESCRIPTION
0	220	First program after starting.
1	220	First NNN program.
2	220	If current program and the next reach the same width.

3	220	
4	220	
5	220	
6	220	
7	220	

6.9. ESECUTION WITH PREFIX

Use of prefix allows working with a production list composed of programs having a commun initial part in their names, called prefix, and a final variable part, that represents programs identifiers.

For instance, let's examine this list:

Name	Num	Typ	Inp	Out	Excl	ZEBRA	**EUR**	350
00AB123	4	S	000	000	2346			
01AC666	255	T			12			
01AC65		A	000	000	568132			
06BD8345	1	R						

The list is composed by 4 programs with different names (00AB123, 01AC666, 01AC65, 06BD8345) and a commun prefix (EUR).
After START, execution will be launched of all programs whose names are composed by prefix + names, that means: **EUR00AB123**, **EUR01AC666**, **EUR01AC65** and **EUR06BD8345**.

In order to introduce a prefix in the production list, press keys CTRL and P; the cursor will appear in the upper border of the *Automatic window*, in order to allow insertion.
Maximum length allowed for prefix is 8 characters, but total length of prefix + name shall not exceed 12 characters.

6.10 ERRORS IN AUTOMATIC START

It is here intended to list a quick guide to all possible errors displayed when command START is given on a list in Automatic Mode. In the examples it is used an imaginary program called GOOFY.

TOOL DESIGN NOT COMPLYING WITH PROGRAM : GOOFY
The program has a tool design different than the one currently configured. Solutions: modify tool design in program GOOFY or assemble tools as required by program GOOFY.

PROGRAM 12 TOOLING, PROGRAM 46 TOOLING
This message is displayed after the previous one and it shows tooling number entered in Editor and current tooling on the machine.

IMPOSSIBLE TO START AUTOMATIC PROCESSING: SET POINT NOT PERFORMED
Command START has been entered without first performing the Set Point. Solutions: perform Set Point and start again.

IMPOSSIBLE TO READ FILE "INDEX.PRG"
The system could not read the programs index. Most likely the file does not exist anymore or the disk is damaged. Call the Assistance.

PROGRAM : GOOFY NOT EXISTING IN DIRECTORY
The program has been inserted in the list but not in the directory. Solutions: Change name to program GOOFY or delete the string where the program name is inscribed.

WRONG NUMERIC FIELD IN PROGRAM : GOOFY
One of the figures in the three numeric fields is wrong. Most likely in that field are inscribed also letters or the value given exceed the limits set for each field. Solution : modify the wrong figure.

PROCESS NOT ALLOWED IN PROGRAM GOOFY
The program's processing code is wrong, and it is thus necessary to modify the code and enter the correct one. At this purpose, please refer to Table of Processing Codes.

FUNCTION NOT EXISTING : PROGRAM GOOFY, Funct. 5, Station 0
In executing program GOOFY it is required a function on a station (f.e. function 5 in station 0) not existing in machine cycles functions. Contact Assistance.

NO MEMORY - PROGRAM TOO LONG
The program to run is too long to be accepted as a single program. Solution: split the program in more parts.

ERROR IN PROGRAM GOOFY COMPILED FILE
In the file compiled by program GOOFY some error has been found, due to a damaged disk or to a system error. Contact Assistance.

WRONG NUMBER OF PARAMETERS IN FUNCTION 5
The number of parameters in function 5 of machine cycles does not correspond with the number of parameters as defined by Compiler. Solution: contact Assistance.

LIST NOT EXISTING, IMPOSSIBLE TO PROCEED IN AUTOMATIC MODE
The master PC has required to the slave PC to run a list not existing in its directory. Solution: create the missing list or modify the list in execution on the master.

PROGRAM GOOFY NOT EXECUTABLE ON THE SPECIFIED AREA
Program GOOFY is entered in a list not executable on the selected area (S, A, T,N, ...). This is due probaly to mechanic limits of machine. Solution : Run program GOOFY on a different area.

SUCTION CUPS CONFIGURATION WRONG OR MISSING
Suction cups have not been properly configured in current program. Solution: If the configuration just miss, it is enough to produce it and run the program again; if the configuration is wrong it could mean that the related file is damaged, or that it does not exist in memory. However, if the file does not exist at all just configure the suction cups to create it; if the file is damaged contact Assistance.

CAD SUCTION CUPS CONFIGURATION OF GOOFY FILE NOT FOUND
Suction cups configuration of program GOOFY is missing. Solution : Create suction cup configuration with T.P.A CAD and run the program again.

TOO MANY SLEEPERS ACTIVATED FOR SECTORS IN GOOFY PROGRAM
GOOFY program activated more than half sleepers for working sector, when suction cups had been configurated by T.P.A CAD. This error take place only if machine is configured with a double working field (SA and TR). Solution : Configure again suction cups with CAD, or cofigure the machine on one only sector.

GOOFY PROGRAM SLEEPERS OUT OF LIMITS
Suction cups codification of GOOFY program does not comply with specified limits in suction cups parameters of technological parametrics (general parameters). Solution : Verify VINCOLI.VNT FILE and if the case contact Assistance.

CODIFICATION NUMBER ALLOTED TO CAD WRONG OR UNSUFFICIENT
Suction cups codification number alloted to T.p.A CAD is wrong or unsufficient. Solution : modify parameter **CAD first codification number**, on suction cups parameters of Technological Parametrics (general parameters), selecting value 56. In this way 200 codifications are keept for CAD management, in correspondence with maximum lenght of a program list; remainig 55 codifications (from 1 to 55) can be used by operator.

GOOFY PROGRAM TO BE COMPILED AGAIN, EXECUTION ENDED.
GOOFY program entered automatic list with barcode needs a new compiled file. List conpilation is ended. Sulution : compile GOOFY program and run it again.

PROGRAM NOT INCLUDED IN BARCODE LIST
Program introduced in the list with barcode, during the execution list last program, is not present in the list of BARCODE definition; it can't be executed. Solution : input such program in definition list and execute it again.

7. HEADS PARAMETERS

HEAD PARAMETERS SELECTION

Heads Parameters mode is accessed from D.O. main menu.
The module consist of a graphic-parametric editor to configure the working heads (tools). It is a tools file programmed on the machine.

Video displays the following window:

HEAD GRAPHIC	HEADS PARAMETERS
	Program n. .... Corrector X .... Corrector Y .... Corrector Z .... Max. dimensions of X+ ..... Max. dimensions of X- ...... Max. dimensions of Y+ ..... Max. dimensions of Y- ...... Max. dimensions of Z+ ...... Tool diameter .... Number of tools........ Tools interaxis........... Head type Min. velocity........... Angular head............. Parameter 17............

HEAD :	**COMMENT :**	**DATA INSERTION**
A1	head fitting x	

As displayed above, the screen is divided into five different fields:

displays the heads name; it is indicated in bold characters to remember that Head selection is the first operation to perform.

displays the comment defined on the head.

graphic display of the heads

displays heads parameters.

input field to define heads parameters
displayed only by pressing key INSERT

In **Heads selection**, the operator can select proper head by moving up and down with keys ARROW UP and ARROW DOWN, or can edit the name directly by pressing key INS to access field DATA INSERTION, where it is possible to edit the Head's name.
Heads names are assigned by following a sequence starting from (A1 - A4) to (V1 - V4) for a total amount of 88 different definitions.

Once selected the head/s, access Parameters field by pressing key TAB.
Use keys ARROW UP and ARROW DOWN to scroll the list.

To insert or modify a parameter in the table:

- Press INS key;
- Position the cursor on the selected parameter;
- edit data and confirm by ENTER;
- start again from point 2 to modufy another parameter;

If key TAB is pressed a second time, modify mode is quit and the cursor moves to **graphic display** field.

Graphic display is not executed if the entered parameters are not correct. In this case, an error message will appear showing the error found.
Graphic display is not executed also in case of angular head input: i.e., with a parameter different than 0.

The heads are graphically displayed by taking as reference point plane XY according to geometrical parameters entered by the operator. These are:

- corrector X
- corrector Y
- max. dimensions of X+ and Y-
- max. dimensions of Y+ and Y-
- N. of drills and interaxis

However, the graphic display is independent from the machine real reference system to plane XY.

In graphic display mode it is possible to access field **Comment** for any modification or insertion.
Press key INS to load the Comment window, then digit the desired string and press ENTER.

It is possible to quit Head Parameters only during upon head selection (field HEADS indicated in bold characters). To quit the mode, press key ESC. If some modifications have been introduced, a message window is displayed to confirm changes. Answer "S' to save the parameters, "N" to abort.

These operations are facilitated by a HELP menu loaded by pressing together keys ALT and H.

DESCRIPTION OF PARAMETERS

Progressive numbering
This parameter can be updated directly by the operator; otherwise, it is updated by the system after any modification occurred to the corresponding head parameters.
It is meant therefore as progressive updatation on the head.

CORRECTOR X
CORRECTOR Y
CORRECTOR Z
Assign heads correctors with reference to application point as entered by the operator for each axis.
On plane xy, head application point is the theoretical point where the head can be draught, when it is used on the working machine.

The meaning of head correctors changes according to head type.
<u>Analyse the case of a vertical working head</u>.

The correctors are significant only if related to a sign (+ or -) and they are used in algebric operation to determine processing values.

Generally : x/y head correctors have a valid value on heads linked in asymmetric way.
Figures define geometric meaning of two parameters, on x and y axes.

Corrector z can be used to show point lenght, on vertical heads : only if spindles z correctors don't reckon on this already (see "Technological parameters" for spingles correctors input") On the contrary head parameter is ientered with value 0.
The picture shows parameter geometrical meaning in case of correction of point lenght.

Consider now the case of working head on lateral faces.

The correctors x/y are significant only if related to a sign (+ or -) and they are used in algebric operation to determine processing values. It is significant the geometrical value of vertical head.

Corrector z has a particular meaning : see "parameter 17".

Max. dimension X+
Max. dimension X-
The parameters of maximum dimension are used to locate the heads on the axis involved. The Ref.Point of these values is assigned on the correspondent corrector x; the indications X+ and X- indicate the direction on axis X in which the parameter is to apply.

These parameters are significant for fitting vertical heads and for working heads on lateral faces.

Below see the following cases:
a) vertical head for fitting (in the first picture)
b) horizontal head (in the second picture)

<u>Vertical head for fitting</u>

In the picture a 9 heads head is shown, with application axis on the central point. Axis x dimensions is shown :
"x+" input the dimension in positive x direction
"x-" input the dimension in negative x direction.
Program positive values for both parameters.

In case of vertical points placed along y axis (head for fitting y) : valid value are introduced on y+- dimension. In the above picture (head for fitting x) : dimensions y+- are input void.

<u>Horizontal head :</u>

In the picture an horizontal head is shown, with application axis on head symmetry axis. The head is seen from the top, with section on xy plane.
Head has two points, for possible working on horizontal top and bottom faces : drilling axis is X.

Axis x dimensions are shown :
"x+" inputs the dimensions in x positive direction
"x-" inputs the dimension in x negative direction.
Positive direction parameter (x+) should be entered negative.

The two parameters are shown hereafter:
"y+" max. dimension Y+
"y-" max. dimension Y-
These two parameters introduce the deplacement of the points with reference to y application axis of head (central axis).
Parameters y+- are summed to tool y corrector.

MAX. DIMENSION Y+
MAX. DIMENSION Y-
Same as above, with the difference that now the parameters are applyed to axis Y.

MAX. DIMENSION Z+
On axis Z should be assigned only max. dimension toward Z+.
This parameter is significant only for heads processing the sides of the workpiece: in fact, it assign tool lower dimension calculated on the drill axis.
The picture shows the geometrical meaning of the parameter.
The head is see from the side, with section along vertical plane.

ANGULAR HEAD
Numeber 1 means theat the head is angular : they are working heads on side faces.

PARAMETER 17

Parameter named "Parameter 17" issignificant in case of head working on the side faces of the piece.

The picyure shows the geometrical meaning of parameter.

In the picture are shown :
a) axis Z;
b) tool corrector z on group (COR. Z : see chapter "Technological parameters") : COR.Z express the distance between head centre and piece bearing surface;
c) head parameter "max.dimension Z+", shown as : z+;
d) parameters "p.17" =parameter 17 and "C.Z"=corrector z of head. These two parameters impose the deplacement of drills with reference to axis z of heads application (central axis); thay are subtracted to tool z corrector.

TOOL DIAMETER
Assigns tool diameter. This parameter is used to introduce diameter value and to determine tool radius correction.

NUMBER OF TOOLS
Number of drills on a head. If a number higher than 1 is entered, the parameter become significant only for drilling tools.
The minimum value to enter is 1.

TOOLS INTERAXIS
Distance between the drills calculated on the axes of the drills themselves.
The picture shows the geometrical meaning of parameter.
In the picture an head for vertical fitting is shown, with 9 drills placed in x direction, with interaxis introduced by operator.
Head is displayed in front, with section along vertical plane.

MINIMUM VELOCITY
MAXIMUM VELOCITY
Define minimum and maximum rotational speed of the spindle. Programming is in positive integer value, in rpm (round per minute).

HEAD TYPE
Parameter to specify the typology of the head. The significant values to be entered are:

0 Non-significant head (head not defined)
1 Vertical drilling (face 5)
2 Horizontal drilling front tool (face 3)
3 Horizontal drilling back tool (face 4)
4 Side drilling lower tool (face 1)
5 Side drilling upper tool (face 2)
6 Horizontal drilling front-back tool (faces 3 and 4)
7 Side drilling upper-lower tool (faces 1 and 2)
8 Horizontal cross-drilling tool (faces 1, 2, 3, 4)

11 Vertical pantograph tool (face 5)
12 Front horizontal pantograph (face 3)
13 Back horizontal pantograph (face 4)
14 Lower side pantograph (face 1)
15 Upper side pantograph (face 2)
16 Back-front horizontal pantograph (faces 3 and 4)
17 Upper-lower side pantograph (faces 1 and 2)
18 Cross horizontal pantograph tool (faces 1, 2, 3, 4)

20 Blade in X
21 Blade in Y
22 Blade rotating of 90 (x + y)
23 Blade rotating of A

30 Base plate inserter in face 5
31 Bush inserter in face 5
32 Hinge inserter in face 5
33 Iron fitting inserter in face 5
34 Inserter for generic 1 in face 5
35 Inserter for generic 2 in face 5

36 Dowel inserter in face 5

37 Universal inserter in face 5
38 Universal inserter in faces 1, 2, 3, 4 and 5
40 Universal inserter in face 1
41 Universal inserter in face 2
42 Universal inserter in face 3
43 Universal inserter in face 4
44 Universal inserter in faces 1 and 2
45 Universal inserter in faces 3 and 4

51 Vertical tapping tool (face 5)
52 Lower side tapping tool (face 1)
53 Upper side tapping tool (face 2)
54 Front side horizontal tapping tool (face 3)
55 Back side horizontal tapping tool (face 4)

90 Measuring probe in face 5
91 Measuring probe in face 3
92 Measuring probe in face 4
93 Lower side measuring probe (face 1)
94 Upper side measuring probe (face 2)
95 Universal probe (faces 1, 2, 3, 4 and 5)

8. TOOLINGS

TOOL DESIGN PARAMETERS

Select TOOLING in D.O main menu to access tooling mode.
Tooling parameters allow selection of up to 50 different machine configurations, among the 10 usable groups.

A tooling defines the complete configuration of the machine, that is :
a) working or free tools position, on each configurable group
b) type of each working tool position

Upon selection, the Cycle Selection main menu will be displayed.
Scroll along the list by using arrow keys :
ARROW UP, or ARROW DOWN, or PAGE UP or PAGE DOWN of numeric keyboard.
To select an item from the menu, locate it with the indicator bar and confirm with ENTER.
To quit TOOLING mode press key ESC to return to D.O main menu.

Tooling main menu contains the following items:

Tooling 0
Tooling 1
Tooling 2
Tooling 3
Tooling 4
Tooling 5

The items corresponding to configurable tooling are then displayed.

After selecting one of these items a second menu opens, allowing assignment of configurations of each group, for a maximum of 10 groups:

```
Group  1
Group  2
.......
Group 10
```

In each group are defined 80 parameters such as: spindle n. (from 1 to 80) to define all usable tools in the group.

On each parameter of type "**spindle n**. " a head should be defined, as defined in Head parameters mode (A1, A2, A3, A4, B1..... V4). If a spindle is not used leave the parameter as not defined.
In any case, a correct definition for all usable tools should be entered, with typology as programmed in Head Parameters.

The following keys are available in tooling assignment mode:

Alt, H	opens a help menu on all commands. Key ESC closes the window
ESC	exits assignment mode, without saving parameters on the hard disk
END	exits assignment mode, saving parameters on the hard disk
Pg UP/PgDn	skips to previous/next page of current menu
F1	on parameter menu, repeats last value inserted.

9. TECHNOLOGICAL PARAMETERS

To access technological parameters mode select the corresponding voice in D.O. main menu.

Technological parameters configures machine data in form of numeric tables, organized in different cycles.

Desk operativity menu is displayed on Main menu for cycles selecting. Move along the scrollable list with direction keys :

ARROW UP, or ARROW DOWN to scroll single voices

PgUp, PgDn to scroll different pages

To select a certain voice of the menu position the indicator bar (bar in colour contrast) and confirm by pressing ENTER.

To quit the mode press :

END to quit the mode with parameters updating on working disk

ESC to quit the mode without parameters updating on working disk

Technological Parameters Main Menu is defined as follow:

General parameters

Parameters of group 1
Parameters of group 2
Parameters of group 3
Parameters of group 4
Parameters of group 5
Parameters of group 6
Parameters of group 7
Parameters of group 8
Parameters of group 9
Parameters of group 10
Linearity correctors
Settings
Custom parameters

In the next pages each of these voices will be discussed in detail (cycles and parameters).

GENERAL PARAMETERS

The selection on GENERAL PARAMETERS displays a second menu window to select among the following cycles:

Groups definition
Working speed
Air values
Max. working values
Drums offset
Ref.points of fields
Coord. of Discharge
Axis X Parameters
Horizontal locking
Vertical locking
PLC Flags
Network parameters

GROUPS DEFINITION

This cycle is defined on 5 parameters for each of the 10 groups managed by the control.

A group defines a working head on the machine, and can be:

- Controlled on two axes: Y and Z
- Controlled only on axis Y while Z is pneumatic
- Entirely pneumatic: in this case the group is defined dummy.

On a machine it is possible to define up to 10 groups, of which:

- 8 controlled, on two axes, (axis Y and axis Z) or on a single axis (Z pneumatic);
- 2 dummy. Such a group can be defined only as group 9 or 10.

In Groups Definition, for each group are defined the following parameters:

CARD NUMBER : enter a value from 0 to 4 (no decimals).
Definition of the PTP200 card to which the group is assigned.

TASK NUMBER : enter a value from 0 to 2 (no decimals).
This number defines the task (procedure) by which the group is controlled on the appointed card. If 0 is entered the group is considered as not defined.

ANTICOLLISION MODE : enter a value from 0 to 10 (no decimals).

It defines the group operating in anticollision mode within the group defined. If 0 is entered no group is operating in anticollision mode.

AXIS Z : enter 0 if this axis is pneumatic. Enter 1 if it is controlled.

AXIS Y : parameter is significant indipendently from unity or tens figure.

Unity figure :

enter 0 if the group is nearby zero position on axis Y; in any other case, enter 1. This parameter is significant if a group operates in anticollision mode.

Tens figure :

enter 0 if group works only on right side of machine;
enter 1 if group works only on left side of machine;
enter 2 if group works only on both sides;
In case of machine with only one side enter 0 on ten figure.

Example : entering 20 means
- group of axis Y near 0
- working group on both sides of the machine.

WORKING SPEED

This cycle is defined by 18 velocity parameters.
Enter only positive values.
For velocities defined in mt/min. : minimum programmable value is 0.01.

Interpolation speed : defines the maximum interpolation speed.
It defines also the speed related to programmed interpolation cycles if direct programming is missing.
Values are reported in mt/min.
Interpolation speed expresses a tangential velocity to required trajectory : therefore, it relates to all interpolating axes.

Cutting speed : defines the maximum velocity of blades displacement in grooving a workpiece. It defines also the speed related to programmed cycles if direct programming is missing.
Cutting speed relates to axes :
- X or Y, respectively to blade **x** and **y**.
- to diagonal trajectory in XY, with tilted blade.
Values are expressed in mt/min.

Horizontal and lateral drilling speed : defines maximum penetration speed on the workpiece to drill on one of side faces. It represent also the velocity related to the cycle if direct programming is missing. This velocity relates to axis X (holes in face 3 or 4) or Y (holes in face 1 or 2).
Values are expressed in mt/min.

Horizontal and lateral drilling rate of rise : defines the velocity of drills rise from the workpiece on one of the side faces. This velocity relates to axes X and Y and it is not programmable.
Values are expressed in mt/min.

Vertical drilling speed
Vertical drilling rate of rise
Same as the previous definitions, with the difference that drilling is now performed on face 5. These velocities relate to axis Z, and, as before, only penetration speed is programmable.
Values are expressed in mt/min.

Pantograph penetration speed
Pantograph rate of rise
Define penetration and rise velocity of workpiece during milling. Velocities relate to axis Z when milling on face 5, to axis X when milling in faces 3 or 4, or to Y when milling in faces 1 and 2. Only penetration speed is programmable.
Values are expressed in mt/min.

Blade penetration speed
Blade rate of rise
Define penetration and rise velocity of workpiece during cutting. Velocities relate to axis Z. Both velocities are not programmable.
Values are expressed in mt/min.

Inserter penetration speed
Inserter rate of rise
Same as above but related to the inserter tool. The axis to which the velocities relate is defined by the face to process. Only the penetration speed is programmable.
Values are expressed in mt/min.

Measuring probe penetration speed
Measuring probe rate of rise
Same as earlier velocities, but releated to measuring probe. The axis to which the velocities relate is defined by the face to process. None of the two velocities is programmable.
Values are expressed in mt/min.

Speed on inserted junction : defines reference interpolation speed to determine speed on inserted junctions to correct tool radius. It is the velocity assumed on a interpolation radius of 100 mm.
Values are expressed in mt/min.

% slowing down while entering
% slowing down while exiting
Define the brake percentages to apply to working speed when running a slow penetration/rise, if the value is not entered by operator.
Enter values never exceeding 100. They are no dimensional values.

Max. rpm of spindle : Enter a value not exceeding 3200 (no decimals). Values are in rpm.
This value defines spindle rotational speed corresponding to max. output +- 10 Volts on PTP200 card analogical-digital converter.

AIR VALUES

This cycle is defined on 7 parameters. Programming unit of measure: millimiters (mm).

Air..... : an air value defines the distance from the workpiece and working tool when operating in consecutive displacements in air for processing on the same face of the workpiece. The programming axis varies on the face itself, since it defines the tool penetration axis in the workpiece. Air quotes vary according to different processings:

Air values of pantograph
Air values of blades
Horizontal air values
Lateral air values
Vertical air values
Air values of inserters

Ref.points max. height : defines the distance between the plane onto which the workpiece rests and the larger ref.point. The parameter relates to axis Z, and the workpiece lays on xy.

9.6 Technological Parameters

MAX. WORKING VALUES

This cycle is defined by 5 parameters. Measuring unit are mm.

Pantograph offpiece processing : defines maximum working area for millings off the workpiece, on coordinated axes X and Y.

Blade offpiece processing : defines max. working area for blades off the workpiece, on coordinated axes X and Y.

Depth of horizontal drilling : defines max. programmable depth for drillings in faces 3 and 4.

Depth of lateral drilling : defines max. programmable depth for drillings in faces 1 or 2.

Value of rise : defines max. programmable value for tool raise from vertical workpiece (work face : 5).

DRUMS OFFSET

This cycle is defined by 5 parameters to describe workpiece locking drums on plane xy. Unit of measure are mm.

offset x drum 1
Offset x drum 2
Offset x drum 3
Offset x drum 4
define positions of offset x for a maximum of 4 drums handled by the control.

Lenght x drums : defines drums on dimension x.

REF.POINT OF FIELDS

Locates the four reference points of workpieces, on coordinates X and Y.
These ref.point are named: N, T, M and A, i.e. the same as working area on processing programs.
Parameters are expressed in programming units: millimiters.

O = mechanic references

O **O** **O**

<- Offset N (x) Offset M (x) ->

Offset A (x) -> <-..Offset T (x)

WORKPIECE UNLOADING

Defined by two values of workpiece unloading:

Front unload value
Back unload value

Parameters are expressed in mm (programming units).
These parameters are used to establish an automatic unload procedure for processed workpieces.

AXIS X PARAMETERS

Linearity multiplier : defines a corrective value (multiplication) for all displacement on axis X. This parameter is significant only for belt displacement on axis X.

HORIZONTAL LOCKING

This cycle is significant only in case of machines with controlled axis for workpiece horizontal locking. It is defined by 4 parameters:

Right hand locking : enter one among the values: 0; 0.5; 1.
Value 0 means that the locking is not active on the right side;
value 0.5 means that the locking is simmetrically active both on the right and on the left side;
value 1 means that the locking is active only on the right side.

Left hand locking : enter one among the values: 0; 0.5; 1.
value 0 means that the locking is not active on the left side;
value 0.5 means that the locking is simmetrically active both on the left and on the right side;
value 1 means that the locking is active only on the left side.

If 0 is entered on both right and left hand locking mode, the horizontal locking axis is not controlled.

Zero offset : defines the maximum displacement on the horizontal locking axis.

Processing speed : velocity of displacement on the horizontal locking axis. values are in mt/min.

VERTICAL LOCKING

This cycle is significant only on machines with controlled system for workpiece vertical locking. It is defined by 3 parameters:

Type of lock : enter 0 or 1.
Value 0 means that vertical locking is not enabled; enter 1 in any other case.

Offset of zero : defines maximum displacement on vertical locking axis.

Processing speed : velocity of displacement on vertical locking axis. values in mt/min.

PLC FLAGS

This cycle is defined on 24 parameters, selected with 0 or 1.
Every parameter correspond to a line mode of an output port in PLC card.
The ports involved are 3, each defined on 8 lines. These ports are n. 240; 241; 242.

Any setting on this cycle means to define these ports, as required by system starting.

NETWORK PARAMETERS

This cycle is defined by 5 parameters, to define network data processing sequence, master/slave configuration.

Master/slave/void (1/2/0) : enter one of the values in brackets.
Value 1 means that the PC is configured in the network as master PC;
value 2 means that the PC is configured in the network as slave PC;
Value 0 means that the PC is not configured into a network.

List or article transmission : on master PC configured into a network, enter:
- Value 0 : Automatic running on slave stations takes place according to the list (or article) transmitted by the master PC, while the whole list and each single program are stored in the lists and program file of the slave PC.
- Value 1 : Automatic running on slave stations takes place according to an explicit programs sequence (list) defined on the master PC. Only single programs are stored in the memory of the slave stations.

Number of slave stations : defines the number of slave stations and it is significant only by the master station.

Output port of slave : defines on all stations belonging to the network the address of the output port used for network data processing. The port assigned should be entirely free.

Input port of slave : defines on all stations belonging to the network the address of the input port used for network data processing.

GROUP n PARAMETERS

Selection on GROUP (n) PARAMETERS, operated in Technological Parameters Main Menu, opens a second menu window with possible definition of 6 kind of cycles. (N) defines a generical group from 1 to 10.

The selection of a certain cycle of group parameters is in any case possible only if the groups have been enabled by General Parameters, discussed in the previous section. For the groups not enabled the cycle is assigned on the label "NOT ENABLED"; their accidental selection will, therefore, be aborted.

The second menu window displayed when selecting Parameters of Group (n) is displayed here below:

Spindle x correctors
Spindle y correctors
Spindle z correctors
Offset (x, y)
Drums offset
Anticollision values

Spindle x correctors
Spindle y correctors
Spindle z correctors

A definition cycle of 80 different values is displayed for every voice. They define the correctors in x, in y, and in z, for each of the 80 tools usable in the process.

In case of unused tools, enter void corrector values.

Correctors x and y related to a tool define tool distance from air values entered (x = x front offset zero; y = front offset zero). The distance is calculated from the contact point of the tool or from the central axis.

Corrector z defines the distance, calculated along Z, from the tool drill to position z=0, calculated when the machine is in Set Point mode.

To calculate the effective working positions on a workpiece, the correctors are considered in algebric additions; values should, therefore, be defined with positive sign.

Unit of measure is millimiters (mm).

When defining correctors x and/or y the use of function keys F2 or F3 can help quick operations. Refer to HELP menu.

OFFSET (x, y)

Zero offset front x
Zero offset front y
Define offset x and y of the group related to the position defined with x=0 and y=0 in Set Point machine mode.
Values are expressed in mm.

===============

Generally "offset zero" point is entered on the first head tool (tool T1). In this case :
a) offset zero parameters of head are programmed with reference to distance of T1 tool from zero point of the machine (on xy plane)
b) x and y correctors of tool 1 are entered with value 0
c) x and y correctors of remaining tools are entered with a value equal to the distance of each tool from T1.

The picture displays the geometrical data of the problem :

9.12 Technological Parameters

On the picture below the geometrical meaning of z corrector is shown, on different types of tools. The letters written in the picture define:

P = bearing surface
V = tool for vertical working
H = tool for horizontal working
L = tool for side working

DRUMS OFFSET

It is defined on 5 parameters, to describe the locking drums related to the group. Values are expressed in mm. and the number of controlled drums is 2.

Offset x drum 1
Offset y drum 2
Define offset positions x and y on the first drum.

Offset x drum 2
Offset y drum 2
define offset positions x and y on the second drum.

Drums lenght x : defines drums dimensions on x.

ANTICOLLISION VALUES

It is defined on 2 parameters of value tipology. Values are expressed in mm.

Anticollision value : defines the minimum distance at which two groups can be run without danger of collision.

Value of escape : defines the minimum safety distance at which two groups operating not contemporarily in anticollision mode can be run.

LINEARITY CORRECTORS

The selection on Linearity Correctors opens a second menu window displaying the following voices:

Correction parameters Axis X correctors Axis Y1 correctors Axis Y2 correctors Axis Y3 correctors Axis Y4 correctors Axis Y5 correctors Axis Y6 correctors Axis Y7 correctors Axis Y8 correctors

From the picture it is possible to realize that the system is able to control 9 linearity correctors on 9 different axis: axis X and axes Y belonging to t groups.

CORRECTION PARAMETERS

On each of the a.m. 9 axes are defined the two parameters:

Number of negative corrections : enter an enteger number (no decimals), positive or negative.
It defines the number of correction intervals on the axis negative values field.
On axis X can be defined 80 intervals, divided into positive and negative; thus the value to enter should never exceed 80.
On axes Y are possible 40 intervals, divided between negative and positive; thus the value to enter should never exceed 40.

Interval exponent enter a positive enteger value (no decimals). This parameter defines the correction interval.

By indicating with:
n = value entered
he first 8 controlled groups. The number is limited to 8 because groups 9 and 10 are definable only as dummyi = correction interval
r = axis resolution, expressed in mm.

the result is:

Example:

It is defined an axis X on working fields:
negative (-500; 0) mm.
positive (0; 300) mm.

Axis resolution: r=0.05 mm.

The 80 correction intervals available are so divided:
Number of negative intervals = 12
number of positive intervals = 68

Minimum correction interval (dimension) **i** is calculated as follow:

Imposing the value of n is obtained assuring correction on the value of i just found.
In this case: n=10, which determines a correction interval i = 51.2 mm.

AXIS X CORRECTORS

80 correctors are available in correspondence with controlled intervals on axis X. Values are expressed in mm.

AXIS Yi CORRECTORS

40 correctors are available in correspondence with controlled intervals on axis Yi. Values are expressed in mm.

SETTINGS

The selection on SETTINGs opens a second menu window, displaying the following voices:

Version Flags Colours Enable

VERSION

It is defined on 4 parameters, displayed as:

SW version
Date SW version
FW version
Date FW version

SW means Software and refers to CNC90 Computer Control.
FW means Firmware and refers to PTP200 Card Control Program.
Enter the correspondent dates and initials as documentation for the programs installed and running.

FLAGS

Unit 0=[mm]/1=[inch] : enter 0 : all measures are preset on mm.

Reference system : enter an entire number (no decimals) between 0 and 3, according to the xy reference system known to the machine. The correspondence on the values is shoewn below :

Machine code : enter the code as discussed when defining machine specifications.

COLOURS

Colours are defined on 5 parameters to assign the same number of basic colours, controlled by the menus of CNC90.

Normal text : assigns colours to labels as preset in CNC90 menus.

Normal background : assigns background colour as preset in CNC90 Menus

Reversed text : allows the operator to select label colours in CNC90 menus.

Reversed background : allows the operator to select background colours in CNC90 menus.

Help text : assigns colour to names of Help menu.

The list here below proposes the numeric correspondence of possible colours:

Black	=	00
Blue	=	01
Green	=	02
Light blue	=	03
Red	=	04
Magenta	=	05
Yellow	=	06
White	=	07
Gray	=	08
Bright blue	=	09
Bright green	=	10
Bright light blue	=	11
Bright red	=	12
Bright magenta	=	13
Bright yellow	=	14
Bright white	=	15

9.18 Technological Parameters

ENABLE

Language : enter the option corresponding to the national selected language.

Ita = Italian
Ing = English
Fra = French
Ted = German
Esp = Spanish
Dan = Danish
Flm = Dutch

Debug : enter 0 or 1.
Value 0 cuts out debug procedures connected to execution control on the PC.
Value 1 enables debug procedures.

Error file name.ext
Production file name.ext
Enter error file and production report file full name (i.e. name + extension).

Letter fonts : enter an entire number (no decimals) among 1, 2, and 3 to select letter fonts for CNC90 Menus.

CUSTOM PARAMETERS

Number of tools : enter 0 if the machine has a single tooling.
Enter a different value if the machine has more toolings.

HELP

In Technological Parameters mode it is available a HELP menu, loaded by pressing together keys **ALT** and **H**.

ESC : by pressing key ESC, the system quits current mode without updating any parameter on working disk.

END : key END allows to quit current mode with parameters updatation on working disk.

PG UP : moves cursor to previous page of current menu.

PG DN : moves cursor to next page of current menu.

F1 : in Parameters menu repeats last value entered.

F2 : in Parameters menu adds 32 to actual parameter. This function is used mainly when setting tool correctors.

F3 : in Parameters menu substract 32 from actual parameter. This function is used mainly when setting tool correctors.

10. PROGRAMS EDITOR

Programs Editor mode is described in the following sections.

Editor module allows to:

a) use general Control modes on working programs.

These functions manage the following procedures:

- display available programs, that is programs loaded into the memory unit (Programs Directory)
- rename an existing program (Rename)
- remove a program from directory (Delete)
- copy a program from a directory to another (Copy)
- load program compiling modes
- print a program ISO text

b) Also the following operation are available:

- create a new program
- modify a program in directory
- store a new or a modified program with possibility of immediate compiling.

MAIN MENU

Once selected EDITOR mode in Machine Menu, Main Menu will be displayed, as shown here below:

0 Open/New
1 Save
2 Save on name
3 Delete
4 Copy
5 Rename
6 Print
7 Graphic Print
8 Settings
9 Compile
A Quit

To select among the modes proposed in Main Menu:

a) Use keys ARROW UP and ARROW DOWN on numeric keyboard to shift to selected mode.

b) Position on desired mode by digitizing its number (0, 1..);

c) Press key ENTER to confirm and start processing

Description appearing in the menu can vary according to language used; on the contrary identification numbers do not change. For example the number 7 always corresponds to selection of Directory Display mode, independently from description appearing on menu by number 7.

0 Open/New
Opens a new program or loads one from memory.

1 Save
Stores a program into memory (enabled only when editing).

2 Save on name
Stores a program assigning a name (enabled only when editing)

3 Delete
Deletes a program from memory.

4 Copy
Copies a program already in directory on a new program. No change occurs to the original program.

5 Rename
Renames a program already in directory assigning it a new name. However, the new name is not yet displayed.

6 Print
Prints ISO text once the program in edit mode has been saved.

7 Graphic Print
Graphic Print of the program under editing layout.

8 Running
Allows starting and editing current program.

9 Compile
Compiling of the program in Edit mode.

A Quit
Quits Editor mode and returns to D.O. Main Menu.

Modes **Save, Save on name, Delete, Copy, Rename, Print, Directory** can be accessed any time, also when editing, without being forced to quit current program.

In the following sections the above listed modes will be discussed in detail.
See further in this Chapter for more information about programs Editor mode.

SAVE

This mode can operate only when editing the program.

If the program is a new one, that is a program not yet stored in directory, Save mode will:

Record the file in the directory and place it according with its name.
Record on disk the program text as defined by the operator during the edit.

In case of program already stored in directory:

The directory is updated with the modified data (i.e. date of recording, dimensions of workpiece and/or tooling number, comment to the program, mask to working heads). In particular: the program results at the moment not compiled.

The program text as eventually modified is recorded.

In case it should not be possible to run the program just saved, once the recording ends an error message will be displayed by means of an icon representing an exclamation mark (!). For further information se the paragraph "Indefinite Geometric Interpolations".

Once the text has been properly stored (i.e. performable), a confirmation is required for programs compiling:

Press Y to confirm
Press N to end storage without compiling.

Upon confirmation, programs compiling is performed and the system returns to Editor: the program compiled is automatically assigned to edit.

Note: In all confirmation patterns, confirmation answer Y (YES) is preset : therefore, it is enough to press key <--' (ENTER) to start compiling; press instead N (NOT) and then <--' to quit without compiling. In this Chapter, key ENTER will be indicated by the symbol <--'.

> **Note**: Confirmation pattern will be always "S" and "N", for Italian language: they can be changed according to the language used on the control.

> **Note**: key ESC is always available to cancel a selection made on the menus available in Editor mode.

SAVE ON NAME

Program storing is realized assigning memorization name. The name where to execute storing is required (destination name).

a) digit the name and confirm by enter to start registration. Destination name must not be present in directory, on the contrary an error message is displayed : "Program xxx in directory". It is possible to reconfirm the name already assigned to the program.

b) ESC cancels the selection

Recording stage executes processing in the same way as "SAVE" command, just described.

DELETE

Delete mode is always available.

When Delete is selected, the programs directory is displayed to select the program to delete. Press key ESC to quit.

Select the program to delete (by digitizing its name on the keyboard or by selecting it from directory page) and confirm with ENTER.

A confirmation of erasure is then required :

Press Y	to confirm.
Press N or ESC	to cancel the selection.

If the name digitized on the keyboard does not correspond to any name contained in the directory, two interrogative marks (??) will appear near the digitized name to indicate a wrong selection.

Program selection mode from directory will be discussed later.

COPY

This mode is always available

After selecting this command, the directory will be displayed to select the program to copy (origin program). Press key ESC to quit; otherwise:

Select origin program (by digitizing its name on the keyboard or by selecting it on directory page) and confirm with ENTER. As Delete mode, the program must be present in directory;

then program name, on which to perform the copy, is required (Destination program) :

a) Digit the name and confirm by ENTER to start copy. Destination file must not exist in the directory: on the contrary an error message will be displayed : "Program xxx in directory";

b) Press ESC to cancel selection.

Copy program executes :

source program copy on destination program

copy of compiling source program

new program is compiled as source program and it is initialized with the date of copy execution.

RENAME

RENAME mode is always available.

After having selected Rename mode, directory will be displayed to select the program to rename (source program). Press key ESC to quit; otherwise:
select Source program (by digitizing it on keyboard or by selecting it from directory page), and confirm with ENTER. Source program must be present in the directory

The programs name to rename is then required (Destination program):

a) digit the name and confirm with ENTER to perform renaming. Destination name should not be found in the directory, otherwise an error message will be displayed.

b) Press ESC to cancel the selection.

If present program compiling are renamed: the new program results therefore compiled as source program. When rename is executed, source program can't be found in the directory.

PRINT

PRINT mode is enabled only with program loaded and allows program printing of ASCII-ISO text, as tested in Editor mode.
The text is printed on pages of 55 lines with progressive numeric heading on the program blocks. The first two Heading Blocks (see further) are numbered as Blocks "0".

Every printed page is headed with:

a) CNC90 Editor Mode
b) Program name
c) Page number
d) Print date and hour.

Each program line is marked by:

a) line progressive number
b) program side (right, left)
c) block on ISO text

In case of editor working on bilateral configuration, printing is executed in sequence on both programmable sides.

When printing command is selected, user is asked if also subprograms texts are required : if the answer is yes, also the text of selected subprogram is printed, in correspondence with of subprogram recall blocks, on backward printing lines.

DIRECTORY

This mode is always available. The directory is listed with programs names in alphabetic order. Various data are associated to each program in directory : here below a basic display is shown :

		DIRECTORY				
NAME:_____			**N.o prg**: 10:45		**Pag**: 1:05	
Name	**Comment**	**U**	**L**	**H**	**S**	**Date**
PRG1	program 1	M	1000	700	30	12/07/91
PRG2	Example n.2	M	500	450	30	13/09/91
PRG3		M	500	450	30	13/09/91
PRG4		M	800	300	19	13/09/91
PRG5		M	1200	700	19	15/10/91
PRG6		M	3000	1200	19	17/10/91
QQQ		M	1000	450	19	17/10/91
RR	shelf n 28		800	500	19	22/05/91
RR4	shoulder	M	2000	600	19	20/05/91
TABLE 12	table top	M	1250	1250	35	20/06/91
<Alt,H=Help>						

The directory is proposed on scrolling pages and displays at most ten programs per page.
The directory can store a maximum of 30.000 programs.

The following commands are available:

(on numeric keyboard) : switch to previous page; without effect on the first page.

(on numeric keyboard) : switch to next page; without effect on the last page.

(numeric keyboard) : return to front page.

(numeric keyboard) : moves to last page.

(right arrow, on alphanumeric keyboard) displays other fields of the directory

to quit Directory mode.

Next to label "N. of Pr." are displayed two numbers (example : 10 and 45)

The first number indicates the position in directory corresponding to the last program in displayed page; the second number indicates the total number of programs in directory.

Next to word "PAGE", two numbers are displayed (in the example 1 and 5):

The first number indicates current page; the second number indicates the total number of pages in the directory.

Every program is displayed in a line, with indication of:

Programs name (example: PRG1). The name of the program can be assigned on a maximum of 12 alphanumeric characters (using letters from A to Z and figures from 0 to 9);
Comment (example: drills) Program comment can be assigned on a maximum of 12 characters
Unit of measure, indicated in field U (M for metrical and I for Inches).
Panel dimensions (L=length, H=heigth, S=thickness).
Date of last program updating (written in the following order : day/month/year)

Information related to each program in the directory are more numerous than those displayed on screen page. Key selection ---> (right arrow) allows to "scroll" directory in horizontal way , on all data fields concerning any single program.

In particular it is possible to store also information concerning:

Tooling, indicated in field **a**;

Progressive number of tooling and parameters related to last compiling programs (field : **#a** and **#p**);

list of specified fields working on program (field : **Mask**) The information is displayed on a mask defined by 10 figures, written with 1 or 0 : 1 means working head, 0 not working head . The first figure at the right corresponds to head 1, till to head 10, on the first figure from left. For instance a program executed on the heads 1 and 3 has working heads mask in this way "000000101".
Two fields of masks are displayed on working heads : one for the right side and one for the left. See after for allocation on one side or two sides machine.

Compiling flag, indicated in field **F**, where:

(space), that is an empty field, indicates that the program is not compiled;
N indicates that the program has been compiled with normal typology;
M indicates that the program has been compiled with specular typology (mirroring);
* indicates that the program has been compiled with Normal and Specular typology;
n indicates that the program cannot be compiled.

In case of empty directory (no program contained in it), the following message will appear on the first line of Menu window: "????????".

The directory is always displayed as shown in the example, any time an operating mode requires directory program displaying. Additional commands can be used to select a program.

A first command allows the operator to select a program by digitising its name on the keyboard (available in the following modes: Open/New, Delete, Copy, Rename).
The flashing cursor in field **NAME** (on the first line of directory) indicates that this mode is enabled :

Digit the name and confirm by ENTER to select the program on the keyboard.

Select TAB to access the directory with direct selection on the graphic page.

Available commands are:

: to quit without operating any selection.

: to switch from field NAME (with possibility of calling a program by keyboard) to Graphic display and direct selection, and vice versa

The following commands are available on graphic directory:

- - - - : to select on directory page (as already mentioned).

- (on numeric keyboard) : to shift selection to on preceding or following program. Indicator bar is displayed in a different colour and covers the whole line.

= (on numeric keyboard) : shift indicator in horizontal, on the field allocated in the directory:

: Press ENTER to confirm selection.

: pressing an alphanumeric key (letter or number) directory pages on the first page containing programs beginning by the same letter of pressed key. Select for instance letter T : directory pages on the first page showing programs with initial letter T; if names with specified letter is not found, a name with the selected letter should be displayed.
The search is recursive over a key sequence and it is reinitialised by change page commands. "S" field indicates the active search string.

Its always possible to recall an help window, describing commands that can be used on graphic directory. Help window can be selected by keys (ALT+H) and quitted by ESC key.

RUN

Run mode is available only with loaded program and allows :

1. save
2. compile
3. program processing

Command can be activated only if enabled in Editor-CNC90 program configuration

COMPILE

Compile mode is enabled only when a program is loaded.

Compiling request is associated to the program loaded in Editor, and it requires to quit Edit mode.

Before starting compiling, the program is recorded with consequent updating of Programs directory (see SAVE mode).

Once compiling has ended, the control returns to D.O. Main Menu.

QUIT

Quit Selection executes the immediate exit from Editor mode and the return to Main Menu.
During editor mode execution, the program will NOT be saved.

OPEN/NEW

OPEN/NEW selection allows to open a program already in directory or a new program.

The directory is displayed to select the program to be opened or created.
Select ESC to cancel the operation mode.
Select from Directory page or digit its name on the keyboard to select a program.

It is also possible read a program from a floppy disk and/or any directory : for instance from a diskette placed in drive A of PC.
This operation can be executed only if it is enabled.

For this reason:
a) put cursor on field NAME;
b) Digit (ALT+E) : a window is opened on video line under the flied NAME;
c) digit the complete path name on program and confirm by ENTER

If an error is found (program not found, bad searching path) a beep is heard and the cursor is placed at the beginning of editing zone.

If Editor processing has been already activated, before loading the selected program a confirmation is required:

ESC or N	to cancel the operation
Y	to confirm the selection.

Upon confirmation, the following Menu is displayed:

OPEN/NEW

Name	PRG1
Comment	Test1
Unit	M
Length	1000.00
Height	450.00
Thickness	20.00
Tooling	0
Side	D

allowing modifications or confirmations of:

comment (max. 25 characters)

Measure unit (M. for[mm], I for [Inches])

Dimensions (length, height, thickness)
The dimensions of the workpiece are expressed in mm (millimetres) or inches. Max. sizes available are: 5.3 if [mm] and 4.4 if [inches].

In other words, it is possible to enter:
- 5 entire numbers and 3 decimals in the first case;
- 4 entire numbers and 4 decimals in the second case.
Do not enter signs "+" or "-".

Tooling used (values from 0 to 49)

Side where program is displayed, when loaded. This item appears only if Editor is configured in bilateral processing. It is possible selecting "D" (right side) or "S" (left side). If working on one side only, right side displaying is assumed by default.

> NOTE right or left characters selection will be expressed by the letters 'D' and 'S', if Italian language is entered, they can be changed in relation to the used language on the control.

It is no possible to modify programs names.

A further voice can be proposed to define a personalised parameter according to machine typology assigned in Technological Parameters.

Commands enabled on the proposed menu are:

to move cursor to the following or preceding field;

to quit selection in Open/New mode;

to confirm entered data.

to select pages of particular data input. Only tow pages can be managed, according to selected configuration for CNC-90 Editor module.

A first page is introduced by 8 characters , named:

a, b, c, d, e, f, g, i.

These 8 characters can be entered in numeric or parametric form and they can be used in the program when geometric data of programmable workings are introduced in parametric form.
For a complete definition of parametric programming syntax see the following paragraph.

If introduced in numeric form, they can utilise:

5 integer figures and 3 decimals (if unity = mm)
4 integer figures and 4 decimals (if unity = inches)

Sign + or - can be introduced.

If entered in parametric form, data can be parametrized by :

piece dimensions (coordinates : l, h, s)
preceding coordinates. In this way:

- coordinate "b" can be entered also with reference to a"
- coordinate "c" can be entered also with reference to "a" and "b" and so on.

A second page can be introduced by 8 characters, named from "Flag 0" to "Flag 7".
They can be entered with value 0 or 1. The meaning of these values can change according to the machine configuration and to the installed program version.

Once data has been confirmed by ENTER:

In case of data checked and correct, Open/New loads required program.

In case of wrong data, an error message appears on each wrong field, with consequent return to Open/New.

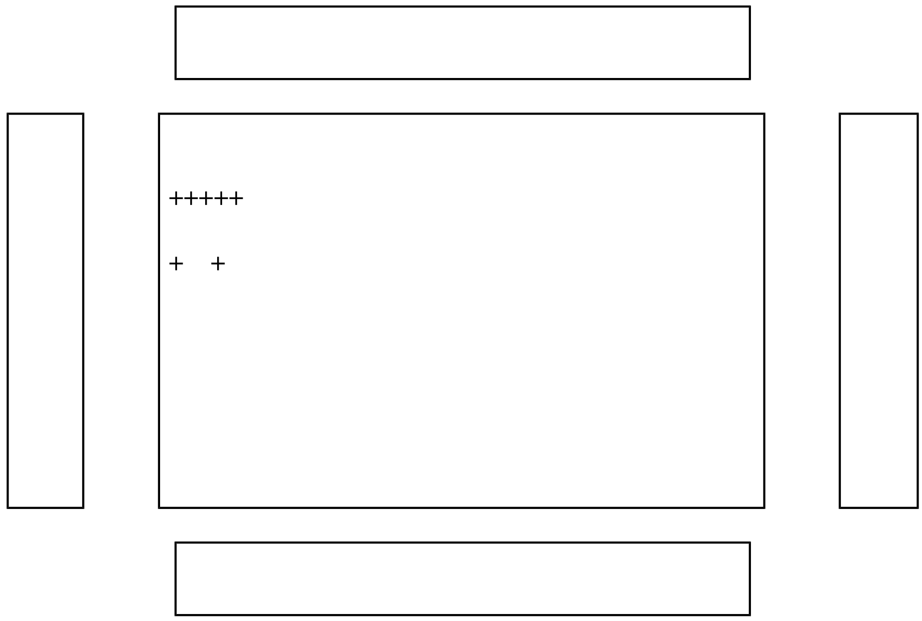
An error message will be displayed when:

a) Measure unit not specified or different from Meters or Inches
b) One or more workpiece dimensions not specified or incorrect.
c) Tooling not specified or incorrect. A tooling is valid only if configured in Parameters mode.
d) "a, b, c" coordinates incorrect, both in parametric or in numeric form.

The program is loaded if it is present in the directory or it is initialised if it is new.
Program reading is displayed by a watch, on the lower part of the screen.
Watch erasure means that Editor can accept a new command by keyboard. Press ESC to reset this operation : in this case the selected program is initialised void.

Once the program is entered, the screen will display the following:

EDITOR CNC90	**PRG1** o			**XYZ**(100;100;5)



<- LxHxS : 1000;450;20 0001:0020

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

N :**1** **DRILL (x,y,z)**
O :**0** Ex:_ M1 :_ a/r:**a**
T :**1 1,2_** S :**200**
F :**2** M2 :__
X :**100**
Ri:**2** Y :**200**
Ro:**0.0** Zp:**5**

<secondary menu> <operating field>

The first string displays:

heading of Editor operation mode (left part of screen)
Program name in memory (in the example: PRG1)
If the program is new, a small **n** will be displayed next to the name; on the contrary, a small **o** next to the name indicates a program already in directory. In this particular case, letter **o** means "Open", while **n** is for "New".
On the right side of the string are displayed the working coordinates and eventual special parameters concerning a selected line of the program (in the example: line 1).

A graphic representation of the panel and selected workings are displayed at the centre of the screen.
The panel is represented on 5 faces: a central one and 4 side faces; their numbering is discussed later.

Two **red cursors**, placed at the edges of any of these faces, display the working corresponding to the selected program line.

The lower part of the screen displays:
1) On the left the **Secondary machine Menu** with its possible selections. Secondary menu is devised in 8 areas each one marked by a number from 0 to 7 and by a name. Name can change according to language translation, numbering does not change.

2) On the right the parameters defining programmed process on a selected line **(Mode Field)**. Program line displayed in working field is normally displayed as current line.

On the right side of Secondary Menu an arrow is displayed indicating the "activated command field".

1. An arrow to the right indicates Mode Field.

2. An arrow to the left (as in the example) indicates Secondary Menu.

To switch from left to right and vice-versa, press key TAB.

In each of the two fields a HELP Menu is available, it is displayed selecting ALT and H keys. Press then ESC to quit HELP.

Note: When two different keys are in parenthesis it means that they should be pressed together (example above : press **ALT** and **H**)

If a Secondary Menu is selected, positioning on keys is realised by:

the four direction keys on numeric keyboard shift cursor in horizontal/vertical
for direct positioning on FILES
for quick positioning on last menu voice
shift to the following page of the secondary menu. Voices on the second page are marked by the characters from 8 to F.
confirms the selection made on enabled position (string in a different colour).

Selection may be executed digitising a number or a letter corresponding to the desired field : numbers or letters are displayed on the menu, with values from 0 to F. Characters from 0 to F can be considered as hexadecimal figures.

switches to operating mode.

Selection of operation mode allows to scroll the program and to look over some specific operation, generally displayed as graphic help window.

shifts to preceding/following line of program.

the cursor shifts on the first line.

shifts to the last menu line (the one marked with "END" operation mode). Closing line is introduced automatically at the end of any edited text. In case of new program the text is initialised with one line dimension, that is on the closing line.

shifts the cursor 10 blocks up or down.

switches current program to the secondary menu

opens a graphic window displaying the selected working on the current line

opens a graphic window displaying ISO text of the program, with possibility of scrolling the text

opens a graphic window displaying the working head/s of the machine, corresponding to the tooling selected by the program

opens a window showing the introduced data on "a, b, c" parameters, when starting a program

See paragraphs on auxiliary help for a specific information.

When Operation mode Field is enabled, secondary menu of machine can be managed without switching to key TAB : digitising the number corresponding to selected menu field.

To modify a block of the program displayed in operation mode field, digit number "3" and directly enter block modify mode.
On the contrary machine Main menu is opened digitising number "0" .

When Operation mode field is enabled, the selection of any key, not read as a command (for example ENTER), causes the flashing of the working corresponding to the actual line (in the central part of the screen, showing a graphic display of the program)

COMPLETE DISPLAY

Here below a complete screen display is shown when the program is loaded.
Some part of the screen will not be considered later, when explaining graphic displaying of different functions or workings, for this reason the figure below can be used as reference point also for the following paragraphs.

On the first line of the screen are displayed:
- heading: CNC90 EDITOR
- program name in editor operating mode : PRG1
- "o" letter means program OPEN already present in directory
- coordinates values on program current block : XYZ (...)

The graphic description of selected piece is displayed in the central part of the screen , with superimposed the window of Main menu.
The two cursors for piece positioning, corresponding to current block, are displayed on the figure.
The cursor for menu selection is displayed in a square frame on the window of Main menu (here it is positioned in "o" field). Selection voices are displayed near the selection numbers, while a sequence of dots takes the place of names.

On the lower part of screen there is:
- Secondary menu (at the left)
- Operating mode field (at the right)

The selection of one of the two fields is marked by coloured arrow, placed between the two fields (here towards left).

Secondary menu is displayed with selection numbers in the first page (selection from 0 to 7).

Operating mode field is left indefinite, for what concern fields of working parameters input. On the contrary you can see in details:
piece dimension : LxHxS
information about current block and full blocks... In the example : the word "0001:0020" means that current line is the number 1 and program is inserted on 20 lines.
information about current block is displayed also on area N:...
block working information is displayed in the field here marked by the general name : HOLE ...

Two particular fields are displayed in the operating mode area at the right:
a first one marked by an arrow graphic symbol oriented towards the right: it is displayed only in case of working loaded on two pages and shows the selection for switching among following pages

a second field is marked by the word "OK" : it appears on each working, when introducing or modifying a program block and displays selection for input data confirming

A piece sketch is displayed, on operating mode field, showing working faces, if the following selections are performed:
face (in colour contrast)
xy reference system
programming origin
See later for further information chapter concerning HOLE workings.

SECONDARY MENU

Selections allowed in Secondary Menu constitutes the permanent connection between Main Menu and effective Programs Editor. Secondary menu is displayed on two pages : selection is executed by **PgUp** and **PgDn** keys.
Some Menu items can be configured and for this reason are not displayed.
Here below are listed available selections. and a brief description of function related to the selection : for a detailed description see specific paragraphs.

FILES

Files selection enables Main Menu; to return to Secondary menu:

1) press key ESC, when operating in Main Menu

2) automatically, when one of the operation available in Main menu is ended:

for example, after deleting a program.

INSERT UP
INSERT DOWN

These selections manage the insertion of one or more new lines in the program. Insertion takes place above (UP) or below (DOWN) current line as required.

insert up adds lines before the current one
insert down adds lines under the current one

A paragraph is dedicated to this function.

MODIFY

Modifies one or more workings already programmed.
It allows to modify geometric and technological values of a working already selected, but not the working type.
For instance : an hole programmed in Cartesian coordinates can't be modified in polar coordinates by this modify; but coordinates, speed, tools values can be modified on the same hole.
A paragraph is dedicated to this function.

DELETE

Deletes one or more lines, starting from current line.
An incorrect inquiry of deletion is filtered : for example with end program operating mode.

Erasure of a program line (the current one) can be executed directly pressing (Ctrl, Y) keys, without selecting DELETE command.

A paragraph is dedicated to this function.

LINE

Allows positioning on a specified line.
Digit line number on displayed window :

Line.. : **23**

and confirm by ENTER. ESC key cancel the command.

DIM

DIM command displays the menu for programs general values selections : measure unity, piece dimension, tooling, "a, b, c.." parameters, "Flag0,..". Selection of these values takes place in a similar way as already explained about command "Open/New"(on main menu).

Confirm selections by ENTER : reading and graphic displaying of selected workings are executed again, performing the selected modifications when necessary.
To reset command DIM, press ESC.

MILL RADIUS

Updates Program Graphic Display with programmed mill radius corrections.
Command is enabled only if mill radius correction program is configured in Editor.
A paragraph is dedicated to this function.

SIDE

Switches to sides of piece programming.
Command is enabled only if bilateral working is configured in Editor.

EXCLUDE

This command allows to list voices to exclude from program graphics. When opening a program (new or already present in the files) no exclusion is enabled, at graphic level.
To Select the program, digit the numbers of graphic exclusions on displayed window:

Esclude.... : **2357**

and confirm by ENTER. ESC key cancel the command.
In the example are selected the exclusions : 2, 3, 5 and 7.
The numbers must be listed one after the other, from 1 to 8, without separating characters, up to a maximum of 8 exclusions.

When Editor is working on bilateral machine, the selection of graphic exclusion is the same for both the sides.
Command EXCLUDE is enabled only if configured on field, for example (exclusions field) activated..

MERGE
Allows insertion of a program of part of a program existing in the archive.

COPY
Allows copying one or more program lines, starting from current line.

RETRIEVE
Allows inserting program lines "captured" with the last Delete or Copy command

ZOOM - ZOOM OFF
Magnifies graphic representation

CONFIGURATION OF EDITOR MODULE

Before discussing the procedures for programs creating or modifying it is necessary to examine the Editor Module configuration.

Editor mode is configurable on most of performances related to program management: selection of programmable processes groups: (drillings, milling, insertions, etc.); selections of the single processes available in each group; selection of parameters related to any single process (processing speed, mandrel velocity and rotation direction, etc.); particular elaborations on the original program (mill radius corrections, etc.).

Configuration on Editor mode is a task performed by the system supplier according to the specifications defined for machine functioning.

Main advantage of this configuration mode is that end user can "see" only the configurations concerning his machine, while not required selections remain "invisible", and therefore not selectable.

Some configuration parameters allow to specialise Editor program on applications very different one from the other geometrically and for this reason reducible to one graphic methodology with difficulty.

The complete configuration of Editor is stored in a file (system program) different on each installed control.

Access to configuration operating mode is allowed by external software program, its use is usually reserved to installation and assistance staff.

PERFORMANCES OF PROGRAMS EDITOR

The functionality of Graphic Editor allows programming a processing cycle, however defined, by means of guided procedures on interactive menus. In any case, no Line or VIDEO (screen) Editor programming is managed.

The operator practically "assembles" its own program made of drillings, millings, insertions, etc. each defined by the geometrical and technological parameters necessary to recognise and perform the processes (working coordinates, velocities, tools, etc.).

The program is displayed by means of Graphic Display, which shows the processable faces continuously updating the representation of the processed workpiece.

However, the operator is not obliged to design origin text processing cycle (program made of instructions, where each instruction is defined by a sequence of codes and operating commands). Main advantage of this configuration is that no programming language is needed to program the machine.

In any case, Editor uses a programming language to define the program "assembled" by the operator on simple selections and graphic settings.

In fact, two recorded texts correspond to each program in directory :

1. An ASCII text in pseudo-ISO format .
2. A binary format (numeric)

Program codification on ASCII text is performed by means of a representation of single processes defined pseudo-ISO: this codification respects, as much as possible, the codification prescribed in ISO Standard Rules for numeric control machines. The control of processes not codified by ISO Standards has implied the introduction of additional codes, significant only for CNC90 Editor environment.
If necessary, the operator can print the ISO codified program by operating the proper selection in Main Menu.

Program codification should be acknowledged in case of programs prepared externally, for instance concerning a specific application, defined and developed directly by the end user : in such an hypotheses, CNC90 Editor should read the program from an external environment, test it, and finally input it in CNC90 working environment.
Such an interfacing takes place on program ASCII text: the external software generates the program in ISO format and delivers it to CNC90 Editor mode.

For this reason, in this user manual each process will be described also with reference to ISO format defining it. For what concerns direct interfacing methods with CNC90-Editor program see specific appendix.

The parameters identifying a program are:

assignment to a module, where **module** means a machine.
name of the program (defined on max. 8 alphanumeric characters).
recording on index file (programs directory).

In this chapter module meaning is not specified : see specific appendix concerning CNC90-Editor working in multimodular applications.

A program is defined upon :

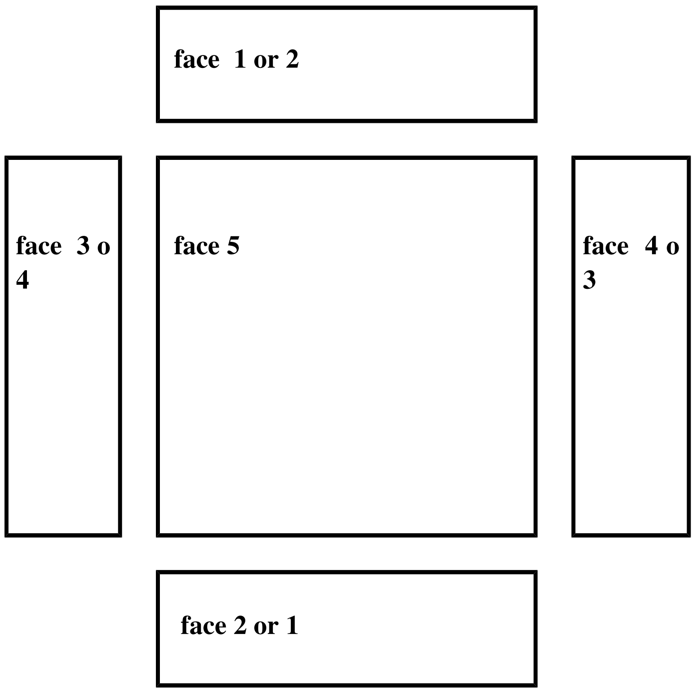
a comment field (defined on max. 30 digits)
programmed Unit of Measure, selectable between [mm] and [inch].
Workpiece dimensions : Length (axis X), Height (axis Y) and Thickness (axis Z).
Working tooling, selected among the ones configured in Technologic Parameters mode.
List of workings to perform on workpiece.

REFERENCE SYSTEMS CONTROLLED

The workpiece is represented graphically by displaying its five processable faces : front face and side faces.

Their numbering depends upon the kind of Reference System defined (see Technological Parameters):

Front face is always defined Face 5 by default.
Side faces along the horizontal axis have numbers 1 (lower face) or 2 (upper face).
Side faces along the vertical axis have numbers 3 (top face) and 4 (bottom face).



Face 5 is defined on plane xy with:

- X as horizontal axis.
- Y as vertical axis.

Each side face has a dimension Z replacing dimension X (faces 3 and 4) or dimension Y (faces 1 and 2). The dimensions not replaced have the same points of Origin and the same orientation as front face, while dimension Z is always originated by the side corner with front face and follows an increasing orientation toward the outer side of the geometrical figure.

Coordinated axes can be assigned on any of the four systems shown in the next pages.

Reference system type 0 :

Reference system type 1 :

Reference system type 2

reference system type 3

In the picture below, the four Reference Systems are reported to a single graphic display, with definition of the number assigned (from [0] to [3]) to the orientation on plane XY of the two axes X and Y.

In Graphic Editor faces selection is always related to a graphic draft made on the 5 faces, so to disengage memory notion from effective location of the faces.

Programming is performed taking the workpiece as reference system, developed on the three Cartesian dimensions: length, height, thickness .

EDITOR ON ONE OR TWO SIDES

Editor program can be configured so that it can work on one or two sides.

EDITOR ON ONE SIDE

Working on one side requires vertical workings management only on top face of piece (face 5). Program is assigned to right side by default.

EDITOR ON TWO SIDES

Bilateral working requires the management of vertical workings also on bottom face. In such configuration, program is assigned in two different parts:
- a first one for right side
- a second one for left side

When working on two sides, each program is formed by two indipendent programs, both for programming and for graphic displaying, but allotted to one workpiece only. As already seen, function 8 allows to switch from one side to the other (Secondary menu "8 SIDE").
Program entering, in bilateral configuration, requires selected programs reading both on right side and on left side. This occurs also for program recording .

The two programming sides are marked as left and right side : right side programs vertical workings on top face, left side programs vertical working on bottom face of piece. Identification characters on right and left side are always marked with "D" and "S" according to Italian language use : however the two characters can be freely assigned, according to the national language used.

Faces numbering does not change, when programming on left side. In particular:
face in vertical workings is marked face 5
face 5 is displayed in transparency, to maintain unchanged reference system representation on coordinated axis
lateral faces position does not change as well as coordinated axes direction on the four lateral faces. In particular : axes Z origin is always related to top face

AUXILIARY HELPS

As already said, auxiliary help windows can be recalled during CNC90-Editor operating phases, in order to supply particular helps when using the different commands.

See below the detailed list of each help, with indication of calling, and using modalities.

ALT + H

Selection of auxiliary help "HALT + H" allows to display a menu window explaining available commands. Displayed menu changes with reference to current operating functions. In particular four different menu of general help are available, each one can be enabled during different operating phases:

a first menu is available when using program directory window
a second menu is available for secondary menu management
a third menu is available for operating area management
a four menu is available for insertion or modification management of program line

keys are available with "ALT+H" window to scroll menu lines.
Press ESC to reset "ALT+H" selection.

ALT + G

"ALT+G" auxiliary help allows to display a graphic window explaining selected working in operating area.

Selection is enabled during:
program text displaying (management in operating area or secondary menu)
modification or insertion on program block

Each working is in general related to a different graphic window. It is besides possible assign a graphic window to a group of similar workings.

A set base of graphic descriptions for programming working is supplied during CNC90 program installation. It is moreover possible to personalise the windows and to comply with any specific request of personalisation.

An example of a circular interpolation working Type C1 is displayed here below.

ALT + L

"ALT+L" of auxiliary help allows to display a window on ASCII text of the program.

The cursor is now positioned within the window, with possibility of scrolling the text : operating area is updated on text block pointed by the cursor.

The following keys are available on the window "ALT+L" :

to scroll the text on programmed lines: cursor is shifted one block up and down

: to scroll the text on programmed lines, turning one page up or down

: to scroll the text in horizontal, in case of lines not completely displayed in the window opened on the screen

: to quit "ALT + L" COMMAND : screen is restored on workpiece graphic and commands on operating area.

If the cursor is positioned on a line corresponding to a subprogram working, selection of ENTER key superimposes a second window, on ISO text related to recalled program. This second window scrolls with **arrow up / arrow dn** and **PG UP / PG DN keys**. Use ESC or ENTER to close the window.

"ALT+L" command is enabled with operating mode area or with secondary menu.

ALT + T

"ALT+T" auxiliary help allows to display a graphic window explaining the used tooling in the selected program.

Selection is enabled during program text displaying (operating area or secondary menu management).

Each tooling is related to a different graphic window, corresponding to tools and enabled groups position.

A graphic descriptions set for utilizable toolings can be supplied by request during CNC90 installation, on the base of a tooling documentation. It is moreover possible to personalise the windows and to comply with any specific request of personalisation.

See specific appendix for a detailed exam of personalisation modalities on graphic windows managed in CNC90 Editor operating mode.

Below a graphic window is displayed, arranged on a head tooling.

It is moreover possible to personalise the windows and to comply with any specific request of personalisation.

An example of graphic window arranged for head tooling is displayed here below.

Press ESC to reset "ALT +T" selection.

CTRL + T

"CTRL+T" auxiliary help , in phase of insertion or modification of a line, allows to display a visualisation window on the used tools of a group or on a particular tool of a group.

In particular:

cursor positioned in area T for group data entering : the window displays the tooling of selected group
cursor positioned in area T for toolings data entering : window displays heads parametric data of selected tool

In both case, data are arranged on following pages, and they can be selected by arrow up /arrow dn and PG UP / PG DN .
Press ESC to quit the window and to return to operativity menu.

Case 1 : group selection

The window is named with Ctrl T - nn, where nn means the selected group. On the following pages see the configured toolings of the group (numbered from 1 to 80). Near each tooling number are displayed :

- configured head abbreviation (Ex. : A1)
- head type selection (parameter : head type, stored in head configuration)
- head diameter (parameter: tool diameter, stored in head configuration)

Tools displayed in colour contrast can be selected if head typology is consistent with selected working.

Below a possible visualisation on the first page of help menu is displayed :

Ctrl + T (1)	
T1 =A1 , 1, O 08	T2 =A1 , 1, O 08
T3 =A1 , 1, O 08	T4 =A1 , 1, O 08
T5 =A1 , 1, O 08	T6 =A1 , 1, O 08
T7 =A1 , 1, O 08	T8 =A1 , 1, O 08
T9 =J1 , 11, O 15	**T10=J2 , 11, O 15**

Assume the case of a milling set-up working on face 5 as suggested in the example : colour contrast displayed on T9 and T10 tools, configured on typology 11, means that these tools can be selected for working.

Case 2 : tool selection

If the cursor is in area T for tools selection, configuration menu in heads parameters is displayed with reference to the tool pointed by the cursor. In particular it is possible to see:

a) head type (ex : J1)
b) head controls (on x, y, z axes) in mm
c) head dimensions (x+, x-; y+, y-; z+) in mm
d) points diameter in mm
e) points number
f) interaxe points in mm
g) rotation speed (minimum and maximum) in round/min
h) angular head parameter
i) tool control on belonging group

If visualisation on the window is in colour contrast, tool is allocated with an head typology consistent with selected working.

The Auxiliary Help fonction on tool parameters is active also in Operating field, with current program statement pointed to the group and selected tool/s. In the case of many tools selected, the first on the list has been considered.
In this case, both the group stored data and tool parameters are displayed: the exchange of the set up windows is made by TAB key.

ALT + D

"ALT+D" auxiliary help selection displays an option window on programmed inputs for parameters "a, b, c, ".

Selection is activated during program text visualisation (management of operating mode area or of secondary menu) or during insertion/modification of a program line.
Below a possible help page is displayed:

a=	l/2 +100........	=600
b=	h/2+50..........	=300
c=	s-5.............	=25
d=	30..............	=30
e=	....	=
f=	....	=
g=	....	=
i=	....	=

In the window there are 8 lines, corresponding to program selecting parameters.
Each line is headed with a letter identifying a parameter (for instance : a).
Near to the letter, parameter input is displayed, in numeric or parametric form. At the right of the line parameter real value is displayed, after possible development of parametric expressions.
Unselected parameters assume zero value.

Press ESC to reset selection on the window "ALT+D".

ALT + S

"ALT+S" Auxiliary help selection displays a graphic window showing a subprogram.

Selection is activated during program text visualisation (management of operating mode area or of secondary menu) or during insertion/modification of a program line, but only on current block corresponding to subprogram working.

Each program is related to a dedicated graphic window.

A set base of graphic representations for a group of subprograms is supplied during CNC90 program installation.
Subprogram personalisation is allowed, in order to satisfy every specific required personalisation.

See specific appendix, to examine personalisation modes on CNC90-Editor graphic windows in detail.

Below a graphic window is displayed showing an example of subprogram 900 : selected working (in this case : a closed milling cycle) and parameters meaning that define the figure from geometrical point of view.

Press ESC to reset selection on the window "ALT+S".

ALT + P

"ALT+P" auxiliary help selection displays a program directory window allowing direct selection of required subprogram.

Selection of window "Alt+P" is activated only during insertion/modification of a program line, corresponding to subprogram working.

On directory pages, programs with valid names for the subprogram (from "000" to "999") are marked in a particular way : a graphic symbol is placed at the left of the name.

Only the selection of a subprogram name found in directory is allowed.

Press ESC to reset selection on the window "ALT+P".

ASCII TEXT DEFINITION

This section will discuss the composition rules of ISO text related to a general program.

As already said most information here given is not necessary to modes definition of CNC90 Editor programming.
An operator using only CNC90 Editor programming can examine the current chapter briefly, paying attention only to some necessary explanation.
The text structure of an ISO program is explained in detail, to cover with this manual, also interfacing needs towards an external environment by CNC90 Editor program.

The program is divided into data blocks, and each block is divided into words.
A word is composed by an address (a letter) plus a numerical part: the address defines the kind of instruction. The numeric value connected to an address can express a number without dimensions or can have its own measuring function. This depends on the specified address.

Each program block has a variable size, i.e. it can contain a variable number of words. Each block ends with character LF (Line Feed).

However, a block can contain at most 200 characters, which express all necessary instructions for the execution of a certain operation.

A block containing a program includes generally the following words:

- Preparatory functions (G);
- Coordinates (X, Y, Z);
- Processing speed (F);
- Rotational speed of the spindle (S);
- Tool functions (T);
- Auxiliary functions (M).

A preparatory function defines the command system (for example: kind of interpolation, plane of interpolation, etc.)

Auxiliary functions are functions to define all displacements (for example: spindle rotation).

In the interpretation of an ASCII text are not influential:

Sign + on positive coordinates;
Not significant zeroes before and after the comma (decimals)

Example of equivalent writings:

G01 G90 X+100 Y 20.300 ZO
G1 G90 X100 Y 20.3 ZO

The first function G of the block should define the required mode (function G of operative code).

Upon registration of an ASCII block from Graphic Editor module:

The functions of the block are not divided from spaces (except some particular case), and this to optimise recorded programs dimensions. When reading ASCII text the spaces among words are filtered automatically.
Positive numeric values are always considered without sign +.
All unnecessary zeroes (0) are filtered.
No function contains contradictory or double definitions.
Each program block is allocated on all functions necessary to define it completely.

In each program both initial blocks assume a significant meaning:

1st block = contains assignments on Unit of Measure, workpiece dimensions and tooling.
2nd block = contains assignments on comment field and on parameters "a, b, c,..".

These first two blocks are exclusively used to acquire an ASCII text from an external environment : it is not necessary a relation with a directory file from which to acquire such necessary information.
In CNC90 Editor mode the insertion on the two blocks takes place automatically.

Both blocks **should be defined in the described format as below specified.**

Structure of the 1st block

**G70/G71X(length)Y(height)Z(thickness)T(tooling)
N(n. of ASCII blocks)L(n. of binary blocks)**
Example: G70 X1000 Y500 Z20.5 T5 N10 L17

G70	= programming in mm.
X1000	= workpiece length
Y500	= workpiece height
Z20.5	= workpiece thickness
T5	= Tooling
N10	= Total number of blocks memorised on ASCII text, including the heading and the end blocks.
L17	= Total number of numeric blocks used by CNC90 Editor to control the program.

Block is headed with the assignmrnt preparatory function on measure unity (G70 or G71).
The last two words mentioned (addressed as N and L) are obtained from Editor module according to the structure of the program, and they are used for particular modes such as Load Subprogram. As N and L addresses also the addresses n and l can be present , to introduce text length on left side.
When an ASCII text is introduced from an external environment, these words should not be present. Preceding block should be generate on a simplified format :

G71 X1000 Y500 Z200.5 T5

Structure of the 2nd block

G150 (comment) A.. B.. C.. D.. E.. F.. G.. I..
Example: G150 (head decoration) A=1/2+100 B=h/2+50 C=s 2 D30F50

G150	=	function of video message
(	=	message opened
decor	=	comment to the program
)	=	message closed
A..	=	a value introduction
B..	=	b value introduction
C..	=	c value introduction
D..	=	d value introduction
F..	=	F value introduction

The block is headed with the preparatory function of video message (G150). Parameters list follows comment message, closed between round brackets. In the comment no other round brackets can be displayed.

In the example of the block above displayed:

5 values are introduced : for the remaining (values : e, g, i) is assumed value 0
a, b, c values are input in parametric mode : space character is here compulsory, at the end of data entering.
Parametric mode input is executed by "="character found at the right of the character entering the value : for example A= Maximum length with parametric programming is ten characters.

d, f values are input in numeric mode : value entering character is directly followed by numeric value. No void space is required at the end of numeric field

In case of incorrect message, comment field should be found, in "()" format (round brackets open or round brackets closed). In this case, the block above displayed has the following format:

G150 () A=1/2+100 B=h/2+50 C=s 2 D30F50

AVAILABLE FUNCTIONS

Definition of preparatory functions (G)

CODE	Cancel with	FUNCTION
G00		Rapid positioning
G01		Linear interpolation
G02		Clockwise circular interpolation
G03		Counterclokwise circular interpolation
G11		Linear interpolation in polar coordinates
G12		Clockwise circular interpolation in polar coordinates
G13		C.clockwise circular interpolation in polar coordinates
G37		Clockwise helical interpolation
G38		C.clockwise helical interpolation
G04		Stopping time span in seconds
G17	G18/G19	Selection of plane XY
G18	G17/G19	Selection of plane XZ
G19	G17/G18	Selection of plane YZ
G40	G41/G42	Cancel tool radius correction
G41	G40/G42	Left hand correction on tool radius
G42	G40/G41	right hand correction on tool radius
G54	G55/G56/G57	Selection on Origin O
G55	G54/G56/G57	Selection on Origin 1
G56	G54/G55/G57	Selection on Origin 2
G57	G54/G55/G56	Selection on Origin 3
G63	G62/G64	Automatic contouring
G62	G63/G64	Contouring off (change block with a slow down)
G64	G62/G63	Contouring On (change block without slow down)

G70		System of measure (inches) (*)
G71		System of measure (mm.) (*)
G76		Exit of milling tool (*)
G81		Drilling cycle
G83		Drilling cycle with final unload
G84		Tapping cycle
G85		Saw cycle along axis X
G86		Saw cycle along axis Y
G87		Saw cycle on A
G88		Set up of milling tool
G90	G91	Definition with absolute coordinates
G91	G90	Definition with incremental coordinates
G92		Offset programming
G110		Linear interlope. on polar coordinates, with pole on milling
G112		CW circular interlope., with arc assigned on three point
G113		CCW circular interpolation with arc assigned on three points
G114		CW circular interpolation (Arc 1 - Arc 2)
G115		CCW circular interpolation (Arc 1 - Arc 2)
G150		Message displaying
G183		Drilling cycle on Fitting along X axis
G184		Drilling cycle on Fitting along Y axis
G185		Drilling cycle on Repeat along X axis
G186		Drilling cycle on Repeat along Y axis
G187		Drilling cycle on Repeat XY
G190		Base plate insertion cycle
G191		Bush insertion cycle
G192		Hinge insertion cycle

G193		Shelf bearing insertion cycle
G194		Generic 1 insertion cycle
G195		Generic 2 insertion cycle
G196		Pin insertion cycle
G197		Measuring cycle
G200		Dima for in circle holes

(*) Code G76 will not appear on the ASCII text because it is reserved for mill raise. Other reserved codes are: G188, G250, G251 and G255

.

Codes G70/G71 are found (only one of the two) in 1st program block to assign programming units.

Assignments on auxiliary functions

	Value field	Function
N/n	1..1000	Number of ASCII blocks (only on first heading block)
X Y Z W V	± 0.001.....±99999.999 ±0.0001 ±3999.9999	Path inf. on coordinated axes; the first value field is in [mm]; the second is in units of [inches]
X	0.001.. 99999.999	Stopping time code G04, unit [sec]
R	± 0.001. .±99999.999, ±0.0001 ±3999.9999	* for drillings, fitting, repeat, saws: brake value in entry. * disch. drills: value of wood chips discharge * Tapping: rapid approach value * On milling set up or subprog. call: mill radius value (positive values)
Q	± 0.001..±99999.999 ±0.0001...±3999.9999	* Drillings fitting, repeat : brake value in exit * blades: final value on coordinated axis
A	0.. 359.99999	angle degrees for: polar coordinates and rotation on loaded subprogram
A	0.001..99999.999 0.0001..3999.9999	in a oval : minor radius
A	1.. 4	in a oval: number of quadrants in execution or selection of a full oval.
C V	0.001..359.999	angle (in degrees) for Alfa and Beta parameters in Milling tool set-up or Tapping cycle
A B C D E F G	± 0.001...± 99999.999 ±0.0001...±3999.9999	on 2 heading block : input "a, b, c" parameters, that can be used in parametric programming on the program
U	0.001..99999.999 0.0001...3999.9999	* polar coordinates: vector * circular interpolation : radius * oval : semiaxes of an oval * grooving : grooving value * junctions : corner junction radius
I J K	± 0.001..±99999.999 ±0.0001..±3999.9999	parameters related to coordinated axes X,Y,Z to assign: - centres of circular interpolation - final value on repeat xy (I and J)

I	0.001... 99999.999 0.0001... 3999.9999	* discharge drills: increment of discharge value * fitting and repeat: X axis final quote
J	0.001... 99999.999 0.0001... 3999.9999	* discharge drills: minimum increment of discharge value * fitting and repeat : Y axis final quote
K	0.001... .99999.999 0.0001... 3999.9999	* tapping : thread pitch * repeat: reference.points pitch.
K	0.01 .. 1.00	* discharge.drill: reduction coefficient
F	0.001 ..999.999 0.01.. 3900.00	working speed: in [mt/min] when in first value field; in [inches/min] when in second value field.
B	0.001 ..999.999 0.01 ... 3900.00	*working speed in blunting or junction area. *entered connection speed in mill radius correction (programmed on mill set up)
S	1..32000	Rotation. speed of the spindle in [rpm] .
T	0 .. 49	* first heading block: tooling number
T	00.. 05	* rapid xyzwv : stations number
T	ggnnmm..	group and working tools: group gg=01 .. 10 tool nn=01 .. 80
H	0 - 255 1000 - 1255 2000 - 2255 3000 - 3255 4000 - 4255	Auxiliary Functions board 0 Auxiliary Functions board 1 Auxiliary Functions board 2 Auxiliary Functions board 3 Auxiliary Functions board 4
L	1000 - 1255	Subroutine number
L/l	0 - 999	Number of numeric blocs (only on the first head bloc)
E	1 - 99	Exclusions number
P	1 - 5	Working faces
M	02	End of program (last bloc)
M	03 04 05	CW rotation of the spindle CCW rotation of the spindle Spindle rotation Stop
M	50 - 99	Direct set of a Flag on PLC card
	( )	Start of message End of message

	/	* After the G54-G57 functions , to indicate secondary selection on the origin
	.	Decimal point
	+ - / *	* '-' caracter: for negatives quotes * arithmetical operators in parametric programming
	( )	parenthesis for parametric programming
	=	start of assign in parametric programming
	l h s	in parametric programming: panel dimension
	c s	in parametric programming: codes for Cosine and Sine trigonometric functions
	a b c d e f g i	locals parameters in parametric programming
	n	Iteration parameter for parametric programming
	Rn = (value)	set up over working parameter (50 to 99)
	\n	end of bloc

MEASURE UNITS

In the table above displayed some definitions assume a dimensional meaning while others do not relate to any dimension (significant as pure numbers).

The listed dimensions are examined here below:

. [mm] or [inches] for coordinates parameters. Programmable fields are:

0.001 .. 99999.999 to program in [mm]
0.0001 .. 3999.9999 to program in [inches]

The indication on the decimals (figures at the right of the point) means the maximum number of programmable figures. Coordinates parameters can signify on signs + and (example : working coordinates), or on positive value (example: interpolation radius, thread pitch).

In this second case no sign should be programmed.
Where necessary: program for any value only sign " ", as sign "+" is assigned automatically.

. [mt/min] or [inches/min] for axes drive parameters.
programmable fields are:
0.01 .. 99 = program in [mm]
0.1 .. 3900.0000 = program in [inches]
No sign should be programmed.

. [RPM] to define the rotation speed of spindle.
Program an entire value (no decimals), in field **1.. 32000.**
No sign should be programmed.

. [sec] to program execution times.
programmable field is : **0.001 .. 99999.999.**
Minimum time unity is the millisecond.

. [°] to program on angle.
The programmable field is : **001 .. 359.999.**
Programming also on negative values or bigger than absolute value of 365° is managed : control processes and introduces programmed values automatically into the selecting field here allocated.

PARAMETRIC PROGRAMMING

Parametric planning allows to input coordinates values, angles, replication factors on algebraic expressions and directly numeric values.

A parametric programming can also input variables of piece dimensions and/or of parameters "a, b, c .. " entered by the program.

An example of parametric planning is displayed below:

X=1/2+5*1/10

Letters are entered to represent piece dimensions:

l	length	(dimension on X axe)
h	height	(dimension on Y axe)
s	thickness	(dimension on Z axe)

Letters **a, b, c, d, e, f, g, i** can be used to represent homonymous parameters.

Letter **n** is used as execution iteration : its use is available for subprogram working, with subprogram iteration programming. For a detailed exam of iteration parameter **n** see pages referring to such working.

Arithmetical operations are displayed by the characters:

+	algebraic sum
-	subtraction
*	multiplication
/	division

It is allowed the use of round brackets, not headed.

Trigonometric functions of sine and cosine can be programmed too. Pay attention to the following examples:
s30 means sine on 30 degree angle
c60 means cosine on 30 degree angle
Cartesian characters are used to represent angles with values growing from 0 to 360 on counter clockwise rotation.
The argument related to trigonometric function of sine and cosine can't be expressed in parametric mode : it can be introduced only on a positive numeric value.

To compute coordinate value use main rules of operations and brackets valid in mathematical algebra.

So:
 multiplications have precedence over sums and subtractions,
 operations with the same precedence degree : preceding operation (from left) is processed first
 brackets have precedence on all the operations

Not valid programming are for instance:

(1 (100) inserted brackets
l2 100 * sign is compulsory between letter 'l' and the multiplying 2

Notice the following programming:

X=50+(1/10+20)

compute value of X coordinate, with length dimension l=2000

 substitute value to variable l
 resolve the introduced multiplications and divisions : X=50+(200+20)
resolve brackets : X=50+220
 resolve remaining sum : X=270

As second example, notice the following example

X=1/10*3+800/2

compute value of X coordinate, with the following dimensions l=2000 and h=800

 substitute values to the variables:
 X=2000/10*3+800/2
 compute multiplication and division
 X=200*3+400
 X=600+400
 compute remaining sum : X=1000

As third example notice a programming of trigonometric functions:
X=1/10*3+s30*h/2

compute value of X coordinate, with the following dimensions
l=2000 and h=800

substitute values to the variables:
X=2000/10*3+800/2
calculate trigonometric expressions :
X=2000/10*3+0.5*800/2
compute multiplication and division
X=200*3+400/2
X=600+200
compute remaining sum : X=800

Parametric programming generally allows to link selected points to piece edges or centre, or to plan working along any piece dimension.

A parametric programming should not be programmed only on a piece dimension : also numeric programming are correct. See the following examples :

X=700+100
X=1000 (500/4)
X=100*c30

On such purpose, an interesting application concerns inch programming, with expressions of inch fractions. For instance :

X=5+7/8 means 5 inch+7/8 of inch.

A specific use of parametric programming is the execution of the same working cycle on pieces of different dimensions, calling the program as subprogram.

Parametric programming syntax with ISO text.

There are two different cases:

<u>input with only literal auxiliary function</u>

programming X function, used format is displayed above:

X=(expression) (space)

For instance : G81 ... X1/2+100 Y200 ...
That is :

1. character '=' is placed at the right of X function
2. after character '=' the expression is repeated as programmed
3. a space character is placed after parametric expression. Space is not necessary at the end of ASCII block. Space character can be substituted by character ';'.

In case of non parametric programming with auxiliary function X, syntax should be :
X100 ..
without character '=' at the right and without spacing after numeric field.

<u>input with auxiliary function type Rn</u> :
format used is displayed above

Rn=(expression) (space)

For instance : G81 ... R60=1/2+100 R61

That is:
1. character '=' is placed at the right of function Rn. In the example: n=60
2. after character '=' the expression is repeated as programmed
3. a space character is placed after parametric expression. Space is not necessary at the end of block ASCII. Space character can be substituted by character ';'.

In case of non parametric programming with auxiliary function X, syntax should be :
R60=X100 ..

Character '=' always appears at the right of characters 'R60', but space character does not appear after a numeric field.

A parametric programming can be introduced with a maximum number of characters, that is 10 or 20 characters can be introduced. In general :

a) working coordinate programming allows parametric programming with an expression 20 characters long

b) auxiliary field programming (vector, angle, ..) allows parametric programming with an expression 10 characters long

During Editor, programming field length is clearly displayed in colour contrast on the video area kept for programming.

GRAPHIC PRINT

The Graphic Print operating mode is active when program loaded in Editor for allowing the graphic print of the panel working layout. PC must be connected with on line graphic printer.

When selected, this command opens the following window

GRAPHIC PRINT	
remake graphic (Y/N)	**Y**
all screen (Y/N)	**Y**
colours (Y/N)	**N**

to insert graphic print modalities, like:

(Yes, for default) To obtain a better reproduction, it is recommended to use a monochromatic printer when a graphic print is required.

(Yes, for default). With this option, a better display resolution on programmed workings is available. Typing 'N' the graphic area is reduced to usual graphic field on the Editor page: this selection is compulsory if the printing is requested over a limited area by a Zoom command.

(No, for default). To select (by Yes) a colour graphic print, not recommended with B/W printer.

When confirmed, this window is proposed:

SAVE	
Confirm (Y/N)	**Y**

Require alwais the file store, when new or modified, because return in Editor mode causes that alls new programmed informations would be lost. With 'Y' the program is saved .

If required, the graphic representation is displayed on the screen, with all programmed workings, and the printing started.

The printout is headed with informations about:

a) Head on CNC90 Editor mode
b) Program name
c) Dimension of the panel (LxHxS)
d) Identifier of System of reference (XY), following the Technological Parametrics setup.

Printer Setup

Printer must be connected to the parallel port of the computer.
At the installing, the printer setup and the printing modes must be verified or modified relating to the peripheral typology.
The file DIGISET.EXE allows the operations of verify or modify: this file is installed in a different environ from CNC90 (directory \MATCN90, usual environ for CAD/CAM programs of TPA).

DIGISET provides two programming pages:
a) for selecting the printer driver (EPSON MX-80 or HP LASERJET)
b) for selecting the printing mode.

During installation, a file for printer setup is copied in \MATCN90, to allow the use of the maximum number of graphic printer availables.

PROGRAMMABLE WORKINGS

In the following sections all possible programmable processes will be examined

Each process will be examined in its specific aspects.

a) Process definition
defines the data characterising the process and some aspects of general definitions.

b) Selection modes
Defines required modes for process selection. Selection modes refers to insertion in a process.

c) Operativity area displaying.
displays the menu proposed in Operating mode area.

d) Fields description
examines the fields proposed on Operating mode Menu to define:
- meaning
- programming format

e) Notes on the process
Programmed process is examined considering programmable parameters and its definition features, subdivided into:
- Geometrical features
- Technological features

f) Errors displayed during process
Examines all error situations found when :
- reading a program
- entering or modifying a process.

g) Syntax of ASCII block
examines process codification in ISO format (ASCII text format).

DRILLING IN CARTESIAN COORDINATES

Process definitions

Drilling in Cartesian coordinates can be selected on all five faces.

a) In face 5: the selection corresponds to Vertical Drilling with penetration axis along selected workpiece Z coordinate.

b) In faces 1 or 2: the selection corresponds to lower or upper Horizontal Drilling with penetration axis along of assigned workpiece Y coordinate.

c) In faces 3 or 4: the selection corresponds to top or bottom Horizontal Drilling with penetration axis along assigned workpiece X coordinate.

Selections Modes

1. Access from Operativity Menu, first page (upon command INSERT or INSERT D) :

FACE
RAPID (xyx H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on menu DRILL:

DRILL (z, y, z)
DRILL (x, y, u, a)
FITTING X
FITTING Y
REPEAT XY

3. Selection on process DRILL (x, y, z).

Display of Mode definition Area :

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE
1	INSERT U	**5**	LINE
2	INSERT DOWN	**6**	DIM
3	MODIFY	**7**	R.MILL

N :**3** **DRILL(x,y,z)**
O :**0** Es:_ M1 :_ a/r:**a**
o :________ S :**200**
T :**1** **1,2**
F :**2** M2 :__
X :**100**
Ri:**2** Y :**200**
Ro:**0.0** Zp:**5**

{}** On the right side of Operating mode Area a graphic representation of the workpiece subdivided in its five working faces is displayed.

Description of Mode Area:

Some fields do not change with any process:

1. Workpiece dimensions are displayed at the right of **LxHxS** writing, in the following sequence: Length, Height, Thickness: **'mm'** writing or **''''** means measure unit.

2. At the right of piece dimensions are displayed two numbers, in the following sequence: **nnnn:tttt**. In the example this number is 0003:0020. The first number represents the current program block; the second number indicates the total number of program blocks. In the example: current line is number 3, the program is composed of 20 lines, which includes also end program line, introduced in automatic.
In case of running on both sides : at the right of "nnnn:tttt" area the letter "D" is displayed, if right side is activated, "S' if left side is activated.

3 **Field N** : defines the number of the block currently under examination (in the example number 3). Field N is automatically compiled, and is therefore not programmable.

4. The area at the right of field N displays the description of programmed process.
Example : DRILL (x, y, z) to express a drilling in Cartesian coordinates.
This writing can be translated into the different national languages and changes therefore according to the selected language.

5. The graphic display, in reduced format, of the 5 panel faces indicates:

face currently processed (in a different colour)
the Origin of appointed Reference System (circle in white colour)
faces number
programmed Origin (coloured circle)

= Working origin
= system XY origin

Areas to assign Drilling in Cartesian Coordinates

1 **O-field : programming origin**

Inputs 4 pre-defined programming origins, each displayed on the four corners of piece face number 5. Default origin is assumed O = 0 (coincident to the origin of the Cartesian axes in face 5), positioned on the panel corner nearby zero machine point. In case of disabled programming on field O (origins), Origin 0 will be automatically assumed.

Indicating with L = workpiece Length, and with H = workpiece Height, the four Origins result positioned as here below specified, related to zero machine point.

O0 { X=0 ; Y=0}
O1 { X=0 ; Y=H}
O2 { X=L ; Y=0}
O3 { X=L ; Y=H}

Coordinate axes orientation is related to each origin, so to carry positive values however within the workpiece. The picture represents the case of Reference System on Cartesian axes xy type 3.

Programming of any of the four Origins will have effect only on coordinates in X and Y.

Next to **O-field** appears a graphic symbol representing a stylized face:
This symbol means that value selection on the concerned field takes places in a guided way. Infact, it is enough to press the space bar on the keyboard to switch data input on all selecting values.

In case of Origin field : values range from 0 to 3. Only the last selected value is displayed.

2. Es-field : exclusion number

Enter an integer number (no decimals) selected between 1 and 99. If not programmed, this field will result empty. Field ES is displayed only if enabled in configuration.

3. **A/R-field : selection of absolute or relative programming**

Also in this case it is defined a guided Editor field. Entering values are:

a = Absolute
r = Relative

Selection on Absolute or Relative is applied to non-deep axis. Otherwise, the axis results however defined with absolute programming.
An Absolute programming is besides input with selection in O field (Origin field):
- different than previous line, or:
- on the first program line

A value programmed as absolute refers directly to selected Origin. A value programmed as Relative refers to previous position.

Example
Define a drilling on face 5 in the following positions (X=100, Y=300, Z=20):

- Axis of depth is Z: therefore, the value expressed is absolute.

The drilling position in coordinated axes X and Y is introduced according to Absolute/Relative selection, and according to selected Origin; consider the following cases:

Absolute, Origin 0 : drilling position defined in (X=100; Y=300);

Absolute, Origin 1 : drilling position defined in (X=100; Y=H-300), with H = workpiece Height.

Absolute, Origin 3: drilling position defined in (X=L-100); Y=H-300, with L = workpiece Length;

Relative, Origin 1 : drilling position defined in : (X=absolute position of preceding X +100 ;
Y=absolute position of Y preceding -300).

4. M1-field : Spindle start and rotation sense
S-field : Spindle rotation speed

In M1 field :

Edit is guided.
Available values are:
3 = right hand rotation
4 = left hand rotation
(Void = empty field) rotation not defined.

In S-field :

Programs are in rpm (round per minute)
values field is 0 - 3200. In any case, max. rotation speed of working tool will be tested (see Heads Parameters).

In case of a non-void value entered in Speed field and void value in M1-field, right hand rotation will be introduced by default. The same principle is valid also for field M1 disabled in configuration and field S enabled in configuration.
If the value in field S is missing, rotation speed entered in Parameters mode for working head will be used.

5. T-field : selection of a group of tools

Working tools selection corresponds to T-field. The fields to be defined are the following two:

1) Group number, assigned with value from 1 to 10. remember that a group defines a working head
2) Working tools on selected group.

A working group can be defined upon:

a) Two controlled axes (X and Z);
b) Single axis Y controlled and Z pneumatic
c) No controlled axis (Y and Z pneumatic = dummy group).

Cases of assignments on groups with a pneumatic axis of Depth are:
a) Drilling in face 5 (Z pneumatic)
b) Drilling in faces 1 or 2 (Y pneumatic).

In this case all information related to drilling axis are forcedly considered non valid; they are:
Drilling velocity F
Slowing coordinates Ri and Ro.

In any case, the group and tools should be specified. except if tool is programmed according to diameter. Diameter programming is examined in the following point.

Working tools should be displayed with their correspondent number (valid values from 1 to 80). Here below are listed valid definitions:

a) **1; 25; 5.**
Correspond to a selection of 3 tools, identified by means of the three numbers : 1, 25, 5. The first tool (in the example: 1) performs the drilling at the programmed position. Tool selection numbers can be divided by one of the following signs:
; (semicolon)
. (point)
: (colon)
, (comma).
Therefore, the maximum number of selected tools is :
10 for planned drilling in face 5
5 for planned drilling on other faces

Example: wrong programs are:

1;2;3;4;5;6;7;8;9;10;11	: selection of 11 tools.
1-2-3	: No valid space between the numbers.

In case of single tool selection: enter only identification number without separation character.

b) **2/20**
corresponds to implicit tools selection from 2 till 20, ends included.
Separation character should be /. This format allows the selection of up to 80 tools, max. number of selectable tools on a group.
The first selected tool (in the example: 2) performs the drilling at programmed position.

The first tool number can be higher or lower than the second tool number.
2/20 assumes as reference tool number 2
20/2 assumes as reference tool 20.

c) **1,2,5/12,20/25,7**
corresponds to an explicit programming of tools numbered 1, 2, 7 togher with implicit programming of tools introduced on (5-12) and (20-25) range. The first selected tool (in the example : 1) performs drilling at programmed position.
Syntax limitations are similar to the preceding case a)
Therefore, the maximum number of selected tools is :
10 for planned drilling in face 5
5 for planned drilling on other faces

In the above example 9 explicit programming are performed : 7 on tools and 2 on character '/'

A wrong selection is for instance :
1;2;3;4;5;6;7;8;9;10;11 : selection of 11 explicit programming.

All selected tools should be configured in Parameters on particular Head parameters value (see Heads Parameters).

The following definitions are considered valid for drillings:

1 Vertical drilling tool (face 5)
2 Top horizontal drilling tool (face 3)
3 Bottom horizontal drilling tool (face 4)
4 Lower side drilling tool (face 1)
5 Upper side drilling tool (face 2)
6 Top-and-bottom horizontal drilling tool (faces 3 and 4)
7 Upper-lower side drilling tool (faces 1 and 2)
8 Cross horizontal drilling tool (faces 1, 2, 3, 4)

6. Ø field: tool diameter:

Tool diameter programming is alternative to group and working tool direct input (field T).
In case of programming both in T field and in diameter field, input in T field prevails.
If tool diameter is assigned not void, selection of working tool is assigned by a optimisation program, according to machine tooling.
Principles used by optimisation program are listed on a specific appendix.

Tool diameter is introduced in units of [mm] or [inch], significant positive.
Parametric programming is allowed.

7. F-field : definition of processing speed

Programming is in units of [mt/min] or [inches/min].
Velocity F represents Tool Penetration Velocity in the workpiece: therefore, it relates to drilling axis.
Missing any definition, the velocity defined in Technological Parameters will be used as reference for Penetration Velocity in vertical drilling (face 5) and on horizontal and side drillings in all other faces.
If the penetration axis is pneumatic: any programming in F-field is considered disabled, and it is imposed a void value.

8. M2-field
It is used for direct input of a flag in PLC. Enabled only at the end of the block.
M2 area is displayed only if enabled in configuration.

9. Ri-field: brake in entry
Ro-field: brake in exit (or final brake)

Both these values refer to drilling axis. Here below it is displayed a representation of values related to drilling axis, in case of drilling performed in face 5 (drilling axis: Z).

Ri = Entry (initial) brake. It defines the coordinate at which the drill brakes with respect to previously programmed penetration velocity (programmed or introduced from Parametric mode). The reduction of applied velocity is defined in Technological Parameters, and it should result as:

Ri <=Zp, with symbol <= meaning "minor or equal".

Zp = final drill depth.

Ro = Exit (final) brake. It defines the final coordinate at which the drill brakes until reaching programmed depth. Brake coordinate should however be defined so to comply with the following conditions:

Ro > Ri where > means "major" if Ri is defined.
Ro <= Zp , where <= means "minor or equal".

Slowing down coordinates are defined in units of [mm] or [inches]. Values Ri and Ro can be programmed without any sign. Parametric programming is allowed.

Initial and final brakes are performed only if programmed. In case of execution on a non-controlled drilling axis the values programmed for both of them are not taken into consideration and considered void (0.0 = brakes not performed).

10. X, Y, Zp field : Coordinates of centre drill and final depth.

Meaning of programmed coordinates in X, Y and Z is defined according to the face processed:
- face 5 : position (X, Y) depth Z
- faces 1 and 2 : position (X, Z) depth Y
- faces 3 and 4 : position (Y, Z) depth X

Values in X, Y, Z are defined in units of [mm] or [inches]; they are significant with sign (program - if they are negative). Parametric programming is allowed.

Letter P, placed next to the letter identifying the axis, displays depth value of drilling axis. Example: with drilling in face 5 the writing on axis Z appears as "Zp".
Remember that an absolute value is assigned to depth coordinates.

Selection of working face takes place directly on Workings Menu by selecting item FACE and defining the number.

FACE 5 RAPID (xyx H) RAPID (xyzwv) DRILL BLADE X

On the right side of voice FACE appears the number of selected face.

Notes on process

Execution modes of any process are defined by the operator: it is possible to configure either the sequence of performable operations (displacements, logic operations on machine I/O) and the machine Parameters (example: brakes, working coordinates, processing speed, technological parameters) to define the operation to perform.

In any case, a drilling cycle on face 5 is usually performed this way:

1. rapid positioning of axes X and Y (axis of the drill) at programmed coordinates (corrected by the first working tool corrector).
2. rapid descent to the plane defined on air value (see Technological Parameters)
3. Descent to plane programmed on Ri, at brake speed.
4. Penetration at processing speed (velocity F) till plane programmed in Ro.
5. Descent to programmed plane Zp at brake speed.
6. Return to plane defined on air value.

Errors displayed when processing

Error 4: Value in S-field not valid
This error is displayed:

a) In case of programmed value higher than 32000 or than the value defined in Technological Parameters in "spindle rotational speed corresponding to a 10V analog voltage".

b) In case of programmed value exceeding the minimum and maximum rotational values as defined in Heads Parametrics. In case of definition made upon more than a single tool, S speed control is performed on the first tool.

Error 4 : Value in F-field not valid
This error is displayed:

a) Velocity F exceeding programmable values : 999 (in programs using [mt/min.]) or 3900 (in programs using [inch/min]); or in case of a number of decimals exceeding 2 [mt/min] or 4 [inch/min].

b) Programmed value exceeding Parametrics velocity corresponding to programmed drilling: entry velocity for Vertical drilling in face 5, and entry velocity for horizontal/lateral drilling in different faces.

Error 4 : value in M2 field not valid
Valid programming values are : 5, and values within 50 and 99 included.

Error 4 : value in T-field not valid
A number external to the interval 1 .. 10 has been programmed.

Error 5 : group not defined
Displayed if the group has not been enabled in Technological Parameters.

Error 6 : wrong syntax in Tool field
Error 7 : tool not valid
Displayed in case of errors in defining Tool field:

a) Tool identification number minor than 1 or higher than 80
b) Separating character between tool numbers not valid or character '/' not admitted.
c) Max. number of definable tools in explicit format exceeding.
d) Tool configured by a non-valid typology or tool configured by no typology
e) Group not valid for working on selected side (see group selection in technologic Parameters).

Error 8 : # field not defined
Character # means a field into which a definition is due. Involved fields are:

a) Field : T (group and tools) : in case of tool diameter not programmed
b) Field : X
c) Field : Y (only if working group processes axis Y controlled, and not dummy)
d) Field : Z (only if working group processes controlled axis Z).

Error 4 : Value in X-field (Xp) not valid
Error 4 : Value in Y-field (Yp) not valid
Error 4 : Value in Z-field (Zp) not valid
Error 4 : Value in Ri-field not valid
Error 4 : Value in Ro-field not valid
Displayed in case of values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and 3999.9999 for programs in [inches]. The number of programmable decimals is, respectively, 3 and 4.

Besides:
a) On Ri-field: if different than 0.0 this value should result at most equal to the final brake value (if different than 0.0) and in any case it should never exceed the value of total drilling depth.
b) In Ro-field: if different than 0.0 this value should never exceed total drilling depth.

Error 3 : wrong syntax in # field
A message error is displayed in case of wrong parametric programming.
Example : wrong syntax in X field

Error 11: Code not valid
This error is displayed in case of program reading, when executing command OPEN/NEW. Message means that the required mode has not been enabled in CNC-90 Editor configuration.

Error 13 : enabled face
This error is displayed in case of program reading, when executing command OPEN/NEW. The message means that the required face has not been enabled in CNC-90 Editor configuration.

These last three error messages are not reported in the sections concerning the other programmable selections, but they are common to everyone of them.

Syntax of ASCII block

```
G81 G90 G54  X.. Y.. Z.. T..   P.. S..        MO3 R.. Q.. F.. E.. M..
    G91 ..               R54=..               M04
        G57
```

----
compulsory fields

Compulsory fields :

G81 is the operative code on drilling function and it should head the block

G90/G91 Absolute or Relative programming.

G54/G55/G56/G57 selection of Programming Origin. The selection are valid, in sequence, on Origins from 0 to 3.

X..Y..Z.. values to define the centre of the drill and the final depth.
The indication of compulsory fields in definitions X.. and Y.. is limited to the possibility of selecting a dummy group or a group with a controlled axis Z.

T.. specification of groups and working tools.

R54=.. tool diameter compulsory programming, in case of function Tl not selected.

Non compulsory fields

S..	Rotation speed of spindle
F..	Penetration velocity.
M03/M04	Rotation direction of spindle
R.. Q..	initial and final brake value
E..	Exclusion Number
M..	Auxiliary M2 function

Here below see the ASCII block corresponding to the proposed values programmed when introduced to mode field:

G81 G90 G54 P5 X100 Y200 Z5 T010102 R2 F2 S200

DRILLING IN POLAR COORDINATES

Process definitions

Drilling in polar coordinates can be performed only in face 5. If a different face is selected Mode Menu will not display the correspondent selection.
If compared to Drilling in Cartesian coordinates, Drilling in polar coordinates is different as to drilling positioning inputs, under a geometrical point of view. A polar reference system is infact adopted in this latter case.

Selection modes

1. from Mode Menu, first page (upon command INSERT U or INSERT D)

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on menu DRILL

DRILL (x, y, z)
DRILL (x,y; u,a)
FITTING X
FITTING Y
REPEAT XY

3. Selection on process DRILL (x,y; u,a)

Display of Mode definition Area

				<-	LxHxS : 1000;450;20	0003:0020
0	FILES	4	DELETE	N :3 O :0	RAPID (xyz) Es:_ M1 :_	 a/r:a
1	INSERT U	5	LINE	 T :1 1	 M2:__	S :200
2	INSERT DOWN	6	DIM	F :2		M2 : - X :100
3	MODIFY	7	R. MILLING	Ri:2 Ro:4	A :30 U :100	Y :200 Z :5

{**} A representation of the workpiece in its five faces is displayed on the right side of Mode Window.

Description of Mode Area

The parameters to define drilling position change their meaning as regards to drilling in Cartesian coordinates.

The working group should have axis Y controlled (non-dummy group)

1. Px/ Py fields : pole (centre of polar system) coordinates.
2. U-field : vector radius.
3. A-field : corner

The coordinates of the pole and the radius are defined in units of [mm] or [inches]. Parametric programming is allowed.

Pole coordinates can be defined absolute or incremental on a previous point. Vector U should be defined positive and not void.

Corner A is programmed in degrees and decimals of degrees on positive values lower than 360. Up to 3 decimals can be programmed. Parametric programming is allowed.

To assign a corner please refer to the system here below depicted (c.clockwise rotation for positive corners).

Negative values programming corresponds to clockwise rotation selection and it is displayed in an automatic way in the interval of 0 -360 of positive value.

Also programming of values exceeding 360 is automatically replaced in the interval of 0 - 360.

See the following examples :

1- programming an angle equal to - 50 : angle value put in a positive field is 360+50=310

2 - programming an angle equal to 420 : angle value put in field 0 is 420-360=60

The following picture defines a point Q assigned on polar coordinates:
P Pole, coordinates Px and Py
U vector radius
A Angle

Errors displayed when processing

All error messages previously described in Cartesian coordinates drilling should be considered valid besides the ones below listed:

Error 4 : Value in T-field not valid
It can display the selection of a dummy group, not allowed in this mode.

Error 4 : Value in U-field not valid
Vector radius lower than the value reckoned as:
10 * axis resolution in [mm].
With axis X resolution of 0.05 mm, the smallest programmable vector radius is 0.5 mm.

Error 4 : Value in U-field not valid
Same as reported for field X (see DRILL xyz).

Error 4 : Value in A-field not valid
Programmed value, with a maximum number of programmable decimals superior to 3.

Error 8 : field not defined
It involves also the following fields:
a) Field A (angle)
b) Field U (vector radius)

Syntax of ASCII blocs

G81 G90 G10 G54 X. Y. Z. A. U. T.. R. Q. F. S.. M03 E.. M..
G91 . G57 R54=..
----
compulsory fields

Only particular significant definitions are examined, in comparison to drilling in Cartesian coordinates:

G10 Polar coordinate programming function**.**

X..Y.. Pole coordinates (centre of polar system)
A.. U.. Angle and vector radius.
Z.. Final drilling depth.

See here below the ASCII block corresponding to proposed values programmed in mode area:

G81 G10 G54 X100 Y200 Z5 A30 U100 T0101 R2 Q4 F2 S200.

FITTING X

Process definitions

Process Fitting X can be performed on faces 5, 1 and 2. If a different face is selected Mode Menu will not display the correspondent selection. Fitting performs multiple drilling along axis X.

Fitting is performed:

1. With a single working head configured with multiple drills at the same distance along fitting axis, or:

2. With more drill heads with a single drill assembled at the same distance on working group so to keep a constant distance between the drills.

Selections Modes

1. Inquiry on Mode Menu, first page (upon command INSERT U or INS D)

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on menu DRILL

DRILL (x, y, z)
DRILL (x,y; u,a)
FITTING X
FITTING Y
REPEAT XY

3. Selection on process FITTING X.

Mode area description

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** O :**0**	**FITTING X** Es:_ M1 :_	 a/r:**a**
1 INSERT U	**5** LINE	Of:**2** T :**1 1,2,3**		S :**200**
2 INSERT DOWN	**6** DIM	F :**2** Ri:**2**	 Ro:-	M2 : -
3 MODIFY	**7** R. MILLING	X :**100** Xf:**50**		Y :1**00** Zp :**5**

Areas descriptions

1. **field O: programming origin of start fitting**
   field Of: programming origin of end fitting

   On both fields reference origins are programmed with regards to positions of start and end fitting.

2. **field a/r : absolute or relative programming**

   Absolute programming is defined with no possibility of variation.

3. **field T: on group and tool**

   Group and working tool selection.
   Tools are valid for typologies and selection modes already mentioned as regards single drilling.
   Working group can be dummy or with axis Z configured pneumatic.

4. **Field F : on work speed**
   Programming is in unity of [mt/mm] or [inch/min] and runs only with depth axis not pneumatic.

5. **Field M2**
Usable for direct writing of a flag in PLC. It is activated at block end.

6. **Field Ri: entrance slowing down**
Field Ro: end slowing down
(the same as DRILL xyz)

7. **Field X/Y/Zp: values of initial fitting and of end depth on drills**

Meaning of programmed coordinates X, Y, Z is defined as regards to working face:
- face 5 : position (X, Y) depth Z
- face 1 and 2 : position (X, Z) depth .

Position programmed in X is the position of the first fitting drill.
X, Y and Z coordinates are programmed in unity of [mm] or [inch]; they are valid if positive (program if negative) and allow parametric programming.
Coordinate defined on drilling axis shows depth value on the letter "P", displayed near axis identification letter.
For example : with drilling on face 5, writing on axis Z is "Zp".

8. **Field Xf : end fitting X coordinate**
End fitting of X coordinate corresponds to drilling position only if it is far from initial point of (n * drilling interaxis) with an integer number.
On the contrary the last drill executed is on the nearest coordinate approximate in defect.

Notes on process

Usable format for tool programming are those examined in case of DRILL xy:

1. explicit note (n1,n2,n3,..), up to 10 tools can be selected;
2. implicit note (n1/n2), up to 80 tools can be selected;
3. note both for implicit and explicit, up to 10 tools can be selected;

Two cases can be examined :
 a) programming with only one tool
 b) programming with more than one tool

In case a) :

- drills number, arranged on heads in x direction, should be higher than 1 (see : heads parametric)
- defined interaxis among drills should be not void (see : heads parametrics) and it is used as drilling pitch, that is : distance between two following holes.

In case b) :

- drills number is equal to selected tools number. Example: 2,3,4,5 correspond to 4 drills; 2/20 correspond to 19 drills;

- drilling pitch is computed on x correctors value of the two programmed end tools and on selected tools number.
  Example : 2,3,4,5 end tools are 2 and 5; computed pitch :

  significant without sign. Also in this case defined pitch should not be void;.

- no control is executed on pitch continuity concerning two tools defined in sequence

- tools numbering is free

- any selected tool can be defined by a variable number of drills and placed on x and y axes : each tool is considered in fact as a single drill.

In both cases:

1. number of performed holes is computed on positive integer value, approximated by defect, of the expression :

2. strokes number when running a fitting is computed on positive integer value, approximated by defect, of expression:

Strokes number should be at least equal to 1.

3. last stroke can repeat some holes executed by last but one : only if total number of holes is not an integer multiple of drills number;

4. last hole is executed at the final coordinate or at the drilling position nearest to programmed coordinate, approximated to the nearest whole number below, if final coordinate does not correspond to drilling position.

Fitting x graphic : each drilling position corresponds to a point displaying.

In case of an incremental operation following a fitting, the defined increment is relative to end fitting programmed point (defined as the absolute value according to programming selected in field Of, on fitting).

Errors displayed when processing

Error 4 : field S value not valid
Error 4 : field F value not valid
Error 4 : field M2 value not valid
Error 4 : field T value not valid
Error 5 : group disabled
Error 6 : wrong syntax in tolls field
Error 7 : tool not valid
(see hole xyz)

Error 8 : Field # not defined
Character # means a field into which a definition is due. Involved fields are:

a) Field : T (group and tools)
b) Field : X (start fitting coordinate)
c) field Xf (end fitting coordinate)
d) Field : Y (only if working group processes axis Y controlled, and not dummy)
e) Field : Z (only if working group processes controlled axis Z).

Error 4 : Value in X-field not valid
Error 4 : Value in Xf field not valid
Error 4 : Value in Y-field (Yp) not valid
Error 4 : Value in Z-field (Zp) not valid
Error 4 : Value in Ri-field not valid
Error 4 : Value in Ro-field not valid
Displayed in case of values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and 3999.9999 for programs in [inches]. The number of decimals programmable is, respectively, 3 and 4.

For notes on fields Ri and Ro: see HOLE xyz.

Error 9 : pitch = 0 or drills number = 1
displays one or both situations not correct:

a) void pitch (computed or defined from heads parametric);
b) drills number not higher than one

Error 10 : strokes number = 0
Displays strokes number if void : working drills number is loaded in a field higher than the difference between programmed coordinates of start and end fitting.

Syntax of ASCII blocks

```
G183  G54  X.. Y.. Z.. I.. T..  P.. R.. Q..   G54/   F.. S.. M3  E.. M..
      .                                          M4
      G57                                     G57/
```

----
compulsory fields

Compulsory fields :

G183 is the operative code on FITTING X function;

G54/G55/G56/G57 selection of Programming Origin. Selections are valid, in sequence, with Origin from 0 to 3.

X..Y.. start fitting coordinates
I end fitting X coordinate
Z holes total depth

T.. group and tools.

Non compulsory fields

P1..5 working face; selection is available on faces: 5, 1 and 2. Lacking any specification, face 5 is assumed.
R.. Q.. initial and final slowing down values
G54/ ..G57/ programmed origin for end fitting coordinate. The selections are valid, in sequence, with origin from 0 to 3, origin programming. In case of origin on end fitting point equal to start fitting point, this word is not entered on ASCII block.
Bar after numeric value of G function differentiates programming of end origin position from start fitting origin position.
S. Rotational spindle speed
F.. Penetration velocity.
M03/M04 Rotational direction of spindle
E.. Exclusion Number
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed when introduced to mode field (working face 5):

G183 G54 P5 X100 Y100 Z5 I50 T01010203 G56/ R2 F2 S200

FITTING Y

Process definition

Fitting y working can be executed on faces 5, 3 and 4. Working on a different face, in mode menu corresponding selection is not available.
Fitting executed multiple drillings along y axis.
All notes already mentioned for x fitting are valid for y fitting mode, replacing coordinate y list by coordinate x list

Selection modes

1. Inquiry on Mode Menu, first page (upon command INSERT U or INS D)

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on menu DRILL

DRILL (x, y, z)
DRILL (x,y; u,a)
FITTING X
FITTING Y
REPEAT XY

3. Selection on process FITTING Y.

Mode area description

		<-	LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**FITTING Y** Es:_ M1 :_		 a/r:**a**
1 INSERT U	**5** LINE	Of:**2** T :**1 1,2,3**			S :**200**
2 INSERT DOWN	**6** DIM	F :**2** Ri:**2**	 Ro:-		M2 : -
3 MODIFY	**7** R.	X :**100**			Y :1**00**

	MILLING

Xf:**50** Zp :**5**

Process notes

Axis Y should be enabled on working group.

Errors displayed when processing

It is valid what mentioned for fitting x, adding :

Error 7 : **tool non valid**
displayed when entering a dummy group (axis Y pneumatic)

Syntax in ASCII blocks

```
G184  G54  X.. Y.. Z.. J.. T..  P.. R.. Q..  G54/   F.. S.. M3  E.. M..
      .                                             M4
      G57                                       G57/
```

----
compulsory fields

Compulsory fields :

G184 is the operative code on FITTING Y function;

X..Y.. start fitting coordinates
J end fitting Y coordinate

All the remaining words have the same meaning defined with fitting x working.

REPEAT X and REPEAT Y

Process definitions

Repeat X can be performed on faces 5, 1, and 2 , same as Fitting X.
Repeat Y can be performed on faces 5, 3, and 4, same as Fitting Y.
If a different face is selected, Mode menu will not display the corresponding item.
Repeat on an axis (X or Y) performs multiple drillings with a single tool along the axis specified by direct pitch definition.

Slection mode

1. Select on Mode Menu, page 1 (upon command INSERT U or INSERT D):

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on menu DRILL

DRILL (x, y, z)
DRILL (x,y; u,a)
FITTING X
FITTING Y
REPEAT X

3. Select process REPEAT X (or REPEAT Y on the second page of DRILL menu).

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE
1	INSERT U	**5**	LINE
2	INSERT DOWN	**6**	DIM
3	MODIFY	**7**	R. MILLING

N :**3**	**REPEAT X**	
O :**0**	Es:_ M1 :_	a/r:**a**
Of:**2**	o : -	S :**200**
T :**1 1**		
F :**2**	K :32	M2 : -
Ri:**2**	Ro:-	
X :**100**		Y :1**00**
Xf:**50**		Zp :**5**

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE
1	INSERT U	**5**	LINE
2	INSERT DOWN	**6**	DIM
3	MODIFY	**7**	R. MILLING

N :**3**	**REPEAT Y**	
O :**0**	Es:_ M1 :_	a/r:**a**
Of:2	O : -	S :**200**
T :**1 1**		
F :**2**	K :32	M2 : -
Ri:**2**	Ro:-	
Y :**100**		X :1**00**
Yf:**50**		Zp :**5**

Description of Mode Area

1.O-field: define Start Repeat programmed origin
Of-field: define End Repeat programmed origin

Please refer to Fitting X.

2. A/R field: select Absolute or Relative programming
It imposes Absolute programming without possibility of intervention.

3. T-field: selection of group and tools
Selection of the group and of the tools performing the process.
The tools are valid with the typologies and the selection modes already discussed for Single Drilling. Selection syntax, already explained for drilling working, can be used.
In case of Repeat Y: the working group cannot be dummy.

4. O field: tool diameter

Tool diameter programming is alternative to direct input of working group and tool (T-field).
In case of programming both in T-field and in diameter field, T-field input prevails.
If tool diameter is entered not void, selection of working tool is defined by a obtimizer program, according to machine tooling. Principles used by obtimizer program are explained in a specific appendix.
Tool diameter is defined in units of [mm] or [inches]; it is are significant with sign. Parametric programming is allowed.

5. F-field: working speed
6. M2-field :
7. Ri-field : slowing down in entry
8. Ro-field : slowing down in exit

Same as DRILL xyz.

9. K-field : drilling pitch

The programmed value imposes the pitch on consecutive drillings. It does not require sign, therefore avoid programming + or -.
Parametric programming in K-field is allowed.

10. X/Y/Zp fields: Start Repeat and final depth of drills values.

The meaning of the programmed values in X, Y and Z is assigned according to the face to be processed:

- face 5 :	position (X, Y)	Depth : Z
- faces 1 and 2 :	position (X, Z)	Depth : Y, only for Repeat X
- faces 3 and 4 :	position (X, Z)	Depth : X, only for Repeat Y

The position programmed in (X, Y) should be intended as first drilling position, but not necessarily it will be the position of the first drill performed by Repeat.
values in X, Y, and Z are defined in [mm] or [inches] and they are significant with sign (digit sign "-" if negative). Parametric programming is allowed.

The value assigned to drilling axis includes the information concerning its depth by letter "p", displayed next to the character identifying the axis. Example: if drilling in face 5, axis Z will be defined as "Zp".

11. X f field : Coordinates of End Repeat X
Y f field : Coordinates of End Repeat Y

Programmed End Repeat coordinates correspond to drilling position only if they is far from entry point of (n * pitch), where n = integer number. Otherwise, the last drilling performed is at the closest coordinate approx. below next number.
Parametric programming is allowed with Xf and Yf.

Notes on processing

When programming, it is possible to specify just a single tool.

Drilling Pitch is defined in programming and it should result not inferior than the value defined on (10 * axis X resolution). Example: with resolution = 0.05 mm, it is possible to program a minimum pitch of 0.5 mm.

The number of performed drillings is computed according to the positive integer number, approximated to the nearest whole number below, of the expression:

In case the difference between Start Repeat and End Repeat coordinates should be minor than programmed Pitch, only one drilling will be performed, at Start Repeat programmed initial position.

Start Repeat programmed value is not necessarily first drilling position: positioning by the coordinates takes place as for Single Drilling, without correcting the order of heads drills.

Repeat X or repeat Y working Graphic :

- working on face 5, head typology graphic is completely displayed
- working on face 1 or 2, graphic of drills placed along axis X is completely displayed.
- working on face 3 or 4, graphic of drills placed along axis Y is completely displayed.

When an incremental operation follows Repeat, its increment is applied to the End Repeat coordinate (corrected to absolute value according to "Of" field programming in Repeat).

Errors displayed when processing

Besides what discussed by Fitting X and Y :

Error 4 : Value in K-field not valid
(see DRILL xyz on similar error message displayed for a different value field)

Error 8 : K-field not defined
K-field (pitch) has not been programmed.

Error 9 : pitch=0 or drills=1
A smaller pitch than the minimum one defined has been entered.

Syntax in ASCII blocks

```
G185 G54 P.. X.. Y.. Z. I.. K.. T..    G54/  P..R.Q. F. S. M3  E. M.
G186.                                                M4
     G57                             G57/
-----------------------------------
```

compulsory fields

G185 and G186 are the operative codes of functions Repeat X and Y, and as such they should head the block;

G54/G55/G56/G57 on main programming origin. Selection are valid, in sequence, on origin from 0 to 3.

X.. Y.. Start Repeat coordinates
I End X Repeat coordinates
J End Y Repeat coordinates
Z Drills depth
K.. Pitch of next drillings.

T.. Group and tool
R54=.. tool diameter, compulsory in case of void programming in T-field

Non compulsory fields

P1..5 working face. Lacking any definition face 5 is loaded.

R.. start slowing down coordinate
Q.. end slowing down coordinate

G54 ..G57/ programmed origin for end repeat coordinate. Selection are valid, sequence, on origin from 0 to 3, as for main origin programming. In case of origin on end repeat coordinate equal to origin on start repeat coordinate, this word is not defined on ASCII block. Bar placed after G function numeric value differentiates origin programming on end position as regards to start repeat origin position.

F Penetration speed
S. Rotational speed of spindle
M03/M04 Rotational direction of spindle
E.. Exclusion Number
M.. Auxiliary M2 function

REPEAT XY

Process definition

Repeat XY working can be performed only in face 5.
If a different face is selected, Mode Menu will not display the corresponding item.

Repeat on both axes X and Y performs multiple drillings with a single tools, along the diagonal imposed by programmed positions for Start and End processing. Drilling Pitch is defined by programming.

Repeat Xy can be geometrically programmed as follows:

a) On Cartesian coordinates (Start and End processing assigned as absolute coordinate values).

b) On polar coordinates (Start process assigned on polar system with its centre on the point of Start Repeat).

Selection modes

1. Selection on Mode Menu, first page (upon command INSERT U or INSERT D).

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
BLADE X

2. Selection on drill menu

REPEAT X
REPEAT XY
REPEAT XY (u,a)

3. Selection on the second Menu page (press key PG DN by numeric keyboard) :

4. Select **REPEAT XY** or **REPEAT XY (u,a)**.

Display of Mode Area

		<-	LxHxS : 1000;450;20	0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**REPEAT XY** Es:_ M1 :_	 a/r:**a**
1 INSERT U	**5** LINE	Of:**3** T :**1 1**	o : -	S :**200**
2 INSERT DOWN	**6** DIM	F :**2** Ri:**2**	K :**32** Ro:-	M2 : - Xf:**100**
3 MODIFY	**7** R. MILLING	X :**100** Xf:**50**		Y :**100** Zp :**5**

		<-	LxHxS : 1000;450;20	0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**REPEAT XY (u,a)** Es:_ M1 :_	 a/r:**a**
1 INSERT U	**5** LINE	 T :**1 1**	O : -	S :**200**
2 INSERT DOWN	**6** DIM	F :**2** Ri:**2**	K :**32** Ro:-	M2 : - A :**110**
3 MODIFY	**7** R. MILLING	Y :**100** Yf:**50**		X :**300** Zp :**5**

Description of Mode Area

1. O-field : define Start Repeat Programming Origin
2. Of-field : define End Repeat Programming Origin

Of-field is displayed only in Repeat XY in Cartesian coordinates.

3. A/R-field : Select Absolute or Relative programming
It imposes Absolute programming without any possibility of intervention.

4. T-field : select group and tools
Selection of current working tool and group.
Tools are valid for drilling typologies. Working group cannot be dummy.

5. O-field: tool diameter

Tool diameter programming is alternative to direct input of working group and tool (T-field).
In case of programming both in T-field and in diameter field, T-field input prevails.
If tool diameter is entered not void, selection of working tool is defined by a obtimizer program, according to machine tooling. Principles used by obtimizer program are explained in a specific appendix.
Tool diameter is defined in units of [mm] or [inches]; it is are significant with sign. Parametric programming is allowed.

6. F-field : define processing speed
7. M2-field :
8. Ri-field : slowing down in entry
9. Ro-field: slowing down in exit

(same as DRILL xyz)

10. K-field : Drilling pitch
Programmed value imposes the pitch on consecutive drillings. It is significant without sign "+" or "-", parametric programming is allowed.

11.. X/ Y/ Zp fields : Start repeat and final depth of the drills
Programmed position in (X, Y) defines first drilling position, but it does not necessarily defines first drill of Repeat.
Values in X, Y, Z are programmed in [mm] or [inches] and they are significant with sign (program sign "-" if negatives). Parametric programming is allowed.

12. X f -field : Coordinates of End Repeat X
Y f -field : Coordinates of End Repeat Y
The value programmed as End Repeat corresponds to last drilling only if equivalent to (n * pitch) where n is an integer number. Otherwise, last drilling will be at the closest coordinate approx. below next number. Xf/ Yf fields allow parametric programming.

13. A-field : Angle
14. U-field : Vector radius
In polar coordinates the final point is calculated at distance U from the point of Start Repeat, with A degrees c.clocwise rotation.
A and U allow parametric programming.

Notes on process

It is possible to program just a single tool.

Drilling pitch is defined when programming and it should be not inferior than the value assigned to (10 * resolution axis X). Example: with resolution of).05 mm, it is possible to program a minimum pitch of 0.5 mm.

The number of performed drillings is now calculated on the distance between Start and End process, as programmed:

Number of drillings means Number of drills, and it is reckoned on the obtained integer positive value approx. below next number.

In case the difference between Start Repeat and End Repeat coordinates should be lower than programmed pitch it will be performed just a single drilling, at the position programmed as Start Repeat.

The value assigned to Start Repeat does not necessarily correspond to the position of the first drill: the positioning at the assigned value take place as discussed in Single Drilling, without correcting the position of drills on the Head.

Repeat XY Graphic Display reports a drill for each drilling, independently from the configuration assigned to working head.

In an incremental operation following a Repeat, the increment is applied relative to the point programmed as End Repeat (corrected to absolute value according to the definition specified in field Of, for Repeat in Cartesian coordinates).

Errors displayed when processing

Same as Fitting and Repeat on a single axis.
In case of Repeat XY (u,a) : in fields U and A are performed the checking already discussed for HOLE (x,y; u,a)

Syntax on ASCII blocks

REPEAT XY :

```
G187 G54 X.. Y.. Z.. I.. J.. K.. T.. R. Q. G54/  F. S. M03 E.. M..
     G57                    R54=..       G57/      M4  -----------------------------------
compulsory fields
```

REPEAT XY (u,a) :

```
G187 G10 G54 X.. Y.. Z..  A..U.. K..T..   R..Q. F. S. M3 E.. M..
G57                    R54=..       G57/      M4
     ----------------------------------------
compulsory fields
```

Compulsory fields

G187 Repeat XY function operative code; it should head the block,

G10 programming in polar coordinates (second case).

G54/G55/G67/G57 programmed origin for end repeat coordinate. Selection are valid, in sequence, on origin from 0 to 3

X.. Y.. Start Repeat values.
Z.. Depth of the drills.
K.. Pitch on further drillings.

I.. J.. End Repeat values (X or Y) in Cartesian coordinates.
A.. U.. Angle and vector radius in polar coordinates.

T.. Group and tool
R54+.. Tool diameter, compulsory if T-function is programmed void

Non compulsory fields

R.	start slowing down coordinate
Q..	**e**nd slowing down coordinate
G54 ..G57/	programmed origin for end repeat coordinate (only on xy repeat in Cartesian coordinates).
F	Penetration speed
S..	Rotational speed of spindle
M03/M04	Rotational direction of spindle
E..	Exclusion Number
M..	Auxiliary M2 function

DRILLS ON CIRCLE

Process definition

It is a drilling template on circle. Working can be performed only on face 5.
Working on a different face, in mode menu corresponding selection is not displayed.

Any number of drills can be programmed on a circle, defined in a polar system.
Programming geometrical parameters are the same as "drills in polar coordinates" programming :
- pole (centre) of polar system
- vector
-rotation angle

Selection mode

1. Inquiry on Mode Menu, first page (upon command INSERT U or INS D)

FACE RAPID (xyz H) RAPID (xyzwv) **DRILL** SAW X

2. Selection on menu DRILLS
3. Selection on the second page of menu (press PG DN key of numeric keyboard)

REPEAT Y REPEAT XY REPEAT XY (u,a) DRILLS ON CIRCLE

4. Selection of process DRILLS ON CIRCLE.

Display of Mode Area

Mode area menu is divided into two pages, that can be switched selecting the sector displayed with an heavy type arrow, on the right side of mode area.

First page :

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** O :**0**	**DRILLS CIRCLE** Es:_ M1 :_	**ON** a/r:**a**
1 INSERT U	**5** LINE	T :-	0 : 10	S :**200**
2 INSERT DOWN	**6** DIM	F :**2**		M2 : - Px:**100**
3 MODIFY	**7** R. MILLING	Ri:**2** R0:**0**	A :**30** U :**100**	Py:1**00** Zp :**5**

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3**	**DRILLS CIRCLE**	**ON**
1 INSERT u	**5** LINE	Nr:**10** An:20		
2 INSERT DOWN	**6** DIM			
3 MODIFY	**7** R. MILLING			

Description of mode area

1. O-field : Programming Origin

O-field defines origin for pole Px, Py coordinates.

2. A/R-field : Select Absolute or Relative programming
It imposes Absolute or relative on Px and Py pole coordinates.

3. T-field : group and tools
Selects working tool and group : programming of a single tool is available.
Tools are valid on drilling typologies.
Working group cannot be dummy.

4. O-field: tool diameter

Tool diameter programming is alternative to direct input of working group and tool (T-field).
In case of programming both T-field and diameter field, T-field input prevails.
If tool diameter is entered not void, selection of working tool is defined by an obtimizer program, according to machine tooling. Principles used by obtimizer program are explained in a specific appendix.
Tool diameter is defined in units of [mm] or [inches]; it is significant with sign. Parametric programming is allowed.

5. F-field : processing speed
6. M2-field :
7. Ri-field : brake in entry
8. Ro-field: brake in exit

(same as DRILL xyz)

9. Ao-field : start angle
10. U-field : vector radios of polar coordinate

Define vector and angle on the first drill, same as drill programming in polar coordinates.

11. Px, Py fields : pole coordinates

(same as DRILL in polar coordinate)

12. Nr fields : Drills total number
12. An : Increment on angle

Nr is drills number : programmable values ranges from 1 to 99. If Nr-field value is void or equal to 0, it is recorded as Nr=1 : working is reduced to only one drill.

An is the angle increment on drills following the first : programmable values are positive, from 0 to 360. If An-field value is void or equal to 0, an automatic assignment of drills is performed on the whole circumference with U radius and centre in (Px, Py).

Notes on processing

Working performs a drills number equal to the value programmed in Nr. Drills are placed along a circular direction, defined by geometrical parameters :

1. circle centre in Px, Py
2. circle radios in U

The first drill is performed at the programmed position on angle in A, as polar coordinates drill.
The remaining drills (nr-1) are executed with counter-clockwise rotation, according to angle increments equal to An.

Following figure displays geometrical parameters meaning.
Drilling positions are numbered from 1 to Nr (in this case Nr=5). Each drilling position is placed on the circumference with centre in (px,Py) and with radios U and it is defined by polar coordinates.
First drilling position is performed on polar coordinates : radios U, angle AO.
Second drilling position is performed on polar coordinates : radios U, angle (AO=An).
And so on till last drilling position, performed on polar coordinates : radios U, angle (AO+ An *(Nr-1).

Besides what explained in the example, a particular case is managed, An-field programming void or equal to 0.
This performs an automatic assignment of programmed drills on the whole circumference, so that they are placed at the same distance.

Consider the case of programming on:
Nr=8 A0=45 An=0

The 8 drillingS are placed as in the following figure

First drill is performed on angle A0; pole distance (Px, Py) is defined by programmed value U.
Increment angle An is computed imposing drills equidistant positioning on the circumference.
Computing formula is : An=360/Nr. in the example is An=45.

Errors displayed when processing

Same as polar coordinates drill, adding what listed below.

Error 4 : Nr-field not valid value
Nr programming value higher than 99.

Error 4 : An-field not valid value
decimals number higher than 3.

Syntax in ASCII blocs

G200 G54 X. Y. Z . A. U. T. R.. Q.. R.52=An R51=Nr F. S. M3 E. M.
G57 R54=.. M4
-------------- ----------------------
compulsory fields

Compulsory Field:

G200 operating code and it should head the block

G54/G55/G56/G57 on main programming origin. Selection are valid, in sequence, on origin from 0 to 3.

X.. Y.. Pole coordinates
Z Drills depth
A.. U.. A0 angle and vector radios.

T.. Group and tool
R54=.. tool diameter, compulsory in case of void programming in T-field

Non compulsory fields

R.. start brake coordinate
Q.. end brake coordinate

R52=.. incremental angle parameter. If not displayed, automatic computation is performed on An;

R51=.. drills total number. If not displayed, only one drill is executed

F Penetration speed
S. Rotational speed of spindle
M03/M04 Rotational direction of spindle
E.. Exclusion Number
M.. Auxiliary M2 function

See below ASCII block corresponding to the programmed values displayed in mode area :

G200 G54 X100 Y100 Z5 A30 U100 R54=10 R=2 R52=20 R51=10 F2 S200

RAPID POSITIONING

Process definition

Rapid modes are selectable on every face.

Rapid positioning performs a point-to-point displacement on the programmed axes.

It is possible to define this mode on the physical axes of a control station: in this case, the displacement can be programmed for up to 5 axes, indicated by their naming on cards (from X to V).

A second possibility is to program group axes and a tool of the same group. In this case the displacement is limited to axes X, Y, and Z to control the group specified.

Displacement speed related to each axis is the maximum velocity defined for air displacements.

Selection modes

1. Select on Mode Menu page 1, upon command INSERT U or INSERT D:

FACE **RAPID (xyz H)** RAPID (xyzwv) DRILL SAW X

2. Select process **RAPID (xyz H)** or RAPID (xyzwv)

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELET E	N :**3** O :**0**	**RAPID** Es:_ M1 :_	 a/r:**a**

1 INSERT U	**5** LINE	T : 1 1	M2:-	S :**200** H : -
2 INSERT DOWN	**6** DIM			X:**100**
3 MODIFY	**7** R. MILLING			Y: - Z :**5**

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** O :**0**	**RAPID (xyzwv)** Es:_	a/r:**a**
1 INSERT U	**5** LINE	T :**1**	M2:-	H : -
2 INSERT DOWN	**6** DIM			X:**100**
3 MODIFY	**7** R. MILLING	W **:300** V **:-**		Y:**200** Z : -

Description of Mode Area

1. O-field: define Programming Origin

In RAPID (xyz) it is possible to define one of the four available Origins.
In RAPID (xyzwv) only Origin 0 will be loaded, without possibility of intervention.

2.A/R field : Absolute or Relative programming

In RAPID (xyz) it is possible to program Absolute/Relative programming.
In RAPID (xyzwv) it is applied Absolute programming, without possibility of intervention.

3. M1-field : Activate and define spindle rotation
S-field : Rotational speed of spindle.

Displayed only upon selection of RAPID (xyz).

4. T-field: select group and tools

In RAPID (xyz) program a group and a tool. Tool typology can be however selected, if defined.
Axes X, Y, Z are positioned by applying Tool Correctors.

In RAPID (xyzwv) program station number. Valid values range from 0 to 4.

5. M2-field :

Available for direct Flag writing into PLC. Enabled at the end of the block.

6. H-field :

Used to load a User's Function during execution. Valid definitions are:

0..	255	Auxiliary functions in card 0.
1000..	1255	Auxiliary functions in card 1.
2000..	2255	Auxiliary functions in card 2.
3000..	3255	Auxiliary functions in card 3.
4000..	4255	Auxiliary functions in card 4.

7. X/ Y/ Z/ fields : positioning on group axes (RAPID (xyz))

Values in X, Y, and Z are defined in units of [mm] or [inches]. They are significant with sign (program sign "-" if negative). Parametric programming is allowed.

Absolute/Incremental selection is introduced to all axes in motion. Programming is not compulsory: when a field is empty the related axis is not considered under motion; otherwise, assigned programming is input.

8. X/ Y/ Z/ W/ V/ fields: displacement to station physical axes (RAPID (xyzwv)

Values in X, Y, Z, W, V are defined in units of [mm] and [inches] and they are significant with sign (program "-" if negative). Parametric programming is allowed. It is applied Absolute programming and Origin 0 on all programmed coordinates.
Programming is not compulsory: when a field is empty (no value programmed) the corresponding axis is not considered under motion; otherwise, on assigned program.
Axes names should be related to selected board physical axes and not to the group. For instance, a movement in RAPID (xyzwv) on X axis can correspond to vertical axis (Y) deplacement of a group.

Errors displayed when processing

Error 4 : value in S-field not valid
(see DRILL xyz)

Error 4 : Value in M2-field not valid
valid programming values are: 5 and the values included within 50..99 included.

Error 4 : value in T-field not valid
a) displayed upon definition of a number not included in the interval 1.. 10 (RAPID xyz).

b) displayed upon definition of a station number not included within the interval 0.. 4 (RAPID xyzwv).

Error 4 : value in H-field not valid
Displayed when a non-valid value has been defined.

Error 5 : group not defined (only for RAPID xyz)
The group has not been enabled in Technological Parameters.

Error 6 : Wrong syntax in Tools field (only for RAPID xyz)
Indicates some error in defining the working tool:

a) Tool identification number less than 1 or more than 80.

Error 7 : Wrong tool
Indicates some error in defining the working tool.

a) Tool configured on no typology (field Heads in Heads Parameters).

Error 8 : T-field not defined
programming in this field is compulsory.

Error 8 : XYZWV-field not defined
message displays neither axes deplacement nor auxiliary function have been selected. No working is executed.

Error 4 : value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Z-field not valid
Error 4 : value in W-field not valid
Error 4 : value in V-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm] and 3999.99999 for programs in [inches]. The number of programmable decimals is, respectively, of 3 and 4.

Syntax of ASCII blocs

RAPID (xyz)
G0 G90 G54 T.. X.. Y.. Z..H.. P.. S.. M3 E.. M..
G91 .G57 M04

----
compulsory fields

<u>Compulsory fields :</u>

G0 is the function operative code of rapid xyz and should head the block.

G90/G91 Absolute or relative programming

G54/G55/G56/G57 select programming origin. Selections are valid, in sequence, on origin from 0 to 3.

T.. Reference group and tool

<u>Non compulsory fields</u>

X ..Y..Z.. Displacement values
H Auxiliary function; at least one of coordinates or auxiliary H function fields should be defined
P1..5 Working face. Selection is possible on any of 5 faces. If function is not selected, face 5 is
entered by default.
S. Rotational speed of spindle
M03/M04 Rotational direction of spindle
E.. Exclusion Number
M.. Auxiliary M2 function

RAPID (x, y, z, w, v):

GO T.. X.. Y.. Z.. W.. V.. P.. E.. M.. H..
----
compulsory fields

Compulsory fields

G0 is the function operative code of rapid xyz and should head the block.

T.. card number

Non compulsory fields

X ..Y..Z.. Deplacement values
H Auxiliary function; at least one of coordinates or auxiliary H function fields should be defined
P1..5 Working face. Selection is possible on any of 5 faces. If function is not selected, face 5 is
entered by default.
E.. Exclusion Number
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the values defined when programming Mode Area:

GO G90 G54 X100 Z5 T0101 S200 (rapid xyz)
G0 X100 Y200 W300 T01 H1100 (rapid xyzwv)

SAW X and SAW Y

Process definition

Saw processing is selectable only in face 5.
Process takes place by displacing only one of axes X or Y and it manages a point-to-point control of the axis performing a groove.

Selection modes

1. Select on Mode Menu, first page, upon command INSERT U or INSERT D:

FACE
RAPID (xyz H)
RAPID (xyzwv)
DRILL
SAW X

2. Select process **SAW X** or SAW Y

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE	N :**3** O :**0**	**SAW X** Es: -M1 :3	a/r:**a**
1	INSERT U	**5**	LINE	Of**:0** T :**1 1**	M2:-	S **:200**
2	INSERT DOWN	**6**	DIM	F :-		M2 :- Y:**100**
3	MODIFY	**7**	R. MILLING	X **:100** Xf**:900**		Zp:**5** Z2:**10**

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** O :**0**	**SAW Y** Es: -M1 :-	a/r:**a**
1 INSERT U	**5** LINE	Of**:0** T :**1 1**	 M2:-	S **:200**
2 INSERT DOWN	**6** DIM	F :-		M2 :- X:**100**
3 MODIFY	**7** R. MILLING	Y **:100** Yf**:900**		Zp:**5** Z2:**5-**

Description of Mode Area

1. O-field: define Start Grooving Programming Origin
Of-field : define End Grooving Programming Origin

Reference Origins for Start and End Grooving are programmed in this two fields.

2. A/R field : Absolute or Relative programming

It is imposed Absolute programming, without possibility of intervention.

3. T-field : select group and tools

Select working group and a single working tool.
In SAW Y, the working group cannot be dummy.

Tools are acknowledged according to the following typologies:
20 : saw in X
21 : saw in y
22 : 90 degrees rotating saw (x+y).

It is also possible to perform a grooving with a milling tool in face 5 (typology 11), if enabled in configuration.

4. F-field : define processing speed

Programming is in units of [mm] or [inches].
F-field is to program Saw Progression velocity. In case of definition missing, the system will apply the corresponding value entered in Technological Parameters. In any case, Saw Entry velocity will be always assumed by the value entered in Technological Parameters.

5. M2-field
Available for direct Flag writing into PLC. Enabled at the end of the block.

6. X/ Xf fields (saw x)
Y/ Yf fields (saw y)
Start and End Grooving positions, on the saw processing coordinate. Parametric programming is allowed.

7. Y-field (saw x)
X-field (saw y)
Groove position on the saw non-working coordinate. Parametric programming is allowed.

8. Zp-field
Depth of the groove. Parametric programming is allowed.

9. Z2-field
Dept of the groove on the second pass. Parametric programming is allowed.
The second pass is executed only if Z2 is greater then Zp: in this case the tool rise up only at the end of the second pass.

Errors displayed when processing

Error 4 : value in S-field not valid
(see DRILL xyz)

Error 4 : value in F-field not valid
Displayed when:

a) velocity F has been defined above 999, if programming in [mt/min], and 3900 if programming in [inches/min], or on a number of decimals exceeding 2 decimals if mt/min., and 4 decimals if in inches/min.

b) Programmed value is higher than the corresponding value entered in Parameters for saw grooving.

Error 4 : value in M2-field not valid
Error 4 : value in T-field not valid
Error 5 : group not defined

(see DRILL xyz)

Error 6 : Wrong syntax in Tool field
Reveals some error in tool definition (it is allowed just a single tool):

a) Tool identification number higher than 80 or lower than 1.

Error 7 : wrong tool

Reports some error in tool definition:

a) Tool configured on a non-valid typology;
b) Tool not configured at all (field Heads in Heads Parameters).

Error 8 : # field not defined
where # means any field in which programming is compulsory. Fields involved are the following:

a) Field : T (define group and tool).
b) Field : X.
c) Field :Y, with saw in X (if working group has axis Y controlled).
d) Field : Xf (saw in x) or Yf (saw in y).
e) Field : Z (only if working group runs axis Z controlled).

Error 4 ; value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Zp-field not valid
Error 4 : value in Xf-field not valid
Error 4 : value in Yf-field not valid
Error 4 : value in Z2-field not valid

With values exceeding controlled ranges: up to 99999.999 when programming in [mm] and 3999.9999 when programming in [inches]. The number of programmable decimals is, respectively, 3 and 4.

Syntax of ASCII blocs

```
G85 G54 X.. Y.. Z.. Q.. T..       G54/  F.. S.. M3  E..   M..
G86 .  G57                          G57/          M4
   G57                         G57/
-----------------------------
compulsory fields
```

Compulsory fields

G85 Saw X function operative code
G86 Saw Y function operative code.
G85/G86 should head ASCII block

G54/G55/G56/G57 Selection on programming origin. Selections are valid in sequence from 0 to 3.

X.. Y.. position of Start working
Z.. value of Depth.
Q.. Position (X or Y) of End working

G54....G57 Origin of Start working position.

T.. working group and tool

Non compulsory fields

G54/..G57/ Origin of end working position.

F.. Saw progression velocity

R.. Second pass depth

S.. Spindle rotation velocity
M03/M04 Spindle rotation sense

E.. Exclusion number
M.. M2 auxiliary function

See here below the ASCII block corresponding to proposed values programmed upon display of SAW X Mode Area:

G85 G54 X100 Y100 Z5 Q900 T0101 S200 M3

SAW ON A

Process definition

It is a particular saw processing, selectable only in face 5.
Process takes place by displacing both X or Y axes at the same time, on a linear tilted trajectory.

Selection modes

1. Select on Mode Menu, upon command INSERT U or INSERT D:

SAW X
SAW Y
MILLING
SPECIALS
INSERTIONS

2. Select process **SPECIAL** workings

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE	N :**1** O :**0**	**SAW A** Es: -M1	a/r:**a**
1	INSERT U	**5**	LINE	Of:**0** T :**1 1**		S :**200**
2	INSERT DOWN	**6**	DIM	F :-		M2 :-
3	MODIF	**7**	R.	A :**50** U :**100**	X : **100** Y : **300**	Zp :**5** Z2:**5**

Y	MILLIN G

Description of Mode Area

1. **O-field: define Start Grooving Programming Origin**

   In this field reference Origin for Start is programmed.

2. A/R field : Absolute or Relative programming

It imposes Absolute programming, without possibility of intervention.

3. T-field : select group and tools

Selects group and working tool. Working group cannot be dummy.

Tools are valid according to the following typologies:
23 : rotating saw in A

It is also possible to perform a grooving with a milling tool in face 5 (typology 11), if enabled in configuration.

4. F-field : define processing speed

Programming is in units of [mm] or [inches].
F-field is to program Saw Progression velocity. In case of missing definition, the system will apply the corresponding value entered in Technological Parameters.
Saw Entry velocity will be assumed by the value entered in Technological Parameters.

5. M2-field
Available for direct Flag writing into PLC. Enabled at the end of the block.

6. X/ Y
Start and End Grooving positions, on the two saw processing coordinates.
Parametric programming is allowed.

7. Y-field (saw x)
U-field
Grooving length. It is valid without sign. Parametric programming is allowed.

8. Zp-field
Depth of the groove.

9. Z2- field
Depth of the groove on the second pass. Parametric programming is allowed.
The second pass is executed only if Z2 is greater then Zp. The tool rise up at the end of the second pass.

10. A-field : angle
11. U-field : vector radios

End point is computed at U distance from start point, with a rotation of A degrees.
A and U allow parametric programming.

Notes on process

Saw grooving working starts at the defined position in (X,Y) and continues along a trajectory of U length with A inclination. The working depth is Zp.
If Z2 is programmed (# Zp), the tool go down at Z2 quote and come back to the initial position (X,Y), the rise up.

Errors displayed when processing

Error 4 : value in S-field not valid
(see DRILL xyz)

Error 4 : value in F-field not valid
Displayed when:

a) velocity F has been defined above 999, if programming in [mt/min], and 3900 if programming in [inches/min], or on a number of decimals exceeding 3 decimals if mt/min., and 2 decimals if in inches/min.

b) Programmed value is higher than the corresponding value entered in Parameters for saw grooving.

Error 4 : value in M2-field not valid
Error 4 : value in T-field not valid
Error 5 : group not defined
(see DRILL xyz)

Error 6 : Wrong syntax in Tool field
Reveals errors in tool definition (it is allowed just a single tool):

a) Tool identification number higher than 80 or lower than 1.

Error 7 : wrong tool
Reports some error in tool definition:

a) Tool configured on a non-valid typology;
b) Tool not configured at all (field Heads in Heads Parameters).

Error 8 : # field not defined
where # means any field in which programming is compulsory. Fields involved are:

a) Field : T (define group and tool).
b) Field : X, Y.
c) Field : A, U.
d) Field : Z (only if working group runs axis Z controlled).

Error 4 ; value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value inU-field not valid
With values exceeding controlled ranges: up to 99999.999 when programming in [mm] and 3999.9999 when programming in [inches]. The number of programmable decimals is, respectively, 3 and 4.

Error 4 : A field value not valid
with value exceeding the programmable number of 3 decimal figures.

Syntax of ASCII blocs

G87 G54 X.. Y.. Z.. A.. U.. T.. F.. S.. M03 E.. M..
. G57

----
compulsory fields

Compulsory fields

G87 Function operative code and it should head the block;

G54/G55/G56/G57 Selection on programming origin. Selections are valid in sequence from 0 to 3.

X.. Y.. position of Start working
Z.. depth value.
U.. grooving length
A.. inclination angle

T.. working group and tool

Non compulsory fields

F.. Saw progression velocity

S.. Spindle rotation velocity
M03/M04 Spindle rotation sense

E.. Exclusion number
M.. M2 auxiliary function

R.. Second pass Depth

Here below see the ASCII block corresponding to proposed values programmed upon Mode Area displaying:

G85 G54 X100 Y100 Z5 Q900 T0101 S200 M3

DELAY

Process definition

Delay programming means to perform a working stop for a certain time

Selection modes

1. Select in second page of Mode Menu:

SAW Y MILLING **SPECIALS** INSERTIONS SUBPROGRAM

2. Select process **SPECIAL** workings

DELAY MESSAGE SIZE L SIZE H SIZE S

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM

N :3 **DELAY**

Es:- M2:-

time : 5

3 MODIFY	**7** R. MILLING

Description of Mode Area

1. **Time field :**

delay programming in seconds or second decimals. Minimum programmable delay is 1 m/sec.

Errors displayed when processing

Error 4 : value in M2-field not valid
(see DRILL xyz)

Error 4 : value in TIME field not valid
valid programming values are included within the interval 0.001....99999.999 with a max. of 3 decimals.

Syntax of ASCII blocs

G4 X.. E.. M..
----
compulsory fields

G4 DELAY function operative code.

X.. time in units of seconds.

Here below see the ASCII block corresponding to proposed values programmed upon display of Mode Area:

G4 X5

MESSAGE

Process definition

Message program corresponds to the display of the message itself during program execution.

Selection mode

1. Select Mode Menu and select the second Menu page :

SAW Y MILLING **SPECIALS** INSERTIONS SUBPROGRAM

2. Select process **SPECIAL** workings

DELAY MESSAGE SIZE L SIZE H SIZE S

3. Select voice MESSAGE

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE
1 INSERTU	**5** LINE

N :3

MESSAGE
Es:- M2:-

time : 5

2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

Description of Mode Area

1. **Msg-field**

Message defined on a max. of 10 alphanumerical digits.
'(' and ')' characters are not allowed : if selected they are substituted by { } parenthesis.

Errors displayed when processing

Error 4 : Value in M2-field not valid
(see HOLE xyz)

Syntax of ASCII blocs

G150 (.....) E.. M..

G150 is the operative code of function :

(Message heading character.
) Message closing character.
..... message.

Here below see the ASCII block corresponding to proposed values programmed upon display of Mode Area:

G150 (....) E10.

MEASURE FUNCTIONS

Process definition

To program a measuring function means to execute an automatic learning procedure of workpiece dimensions.

MEASURE S function can be selected for processing face 5: workpiece thickness learning.

MEASURE H can be selected for processing faces 1 and 2 : workpiece height learning..

MEASURE L can be selected for processing faces 3 and 4 : workpiece length learning.

Selection modes

1. Select on second page of Mode Menu:

SAW Y MILLING **SPECIALS** INSERTIONS SUBPROGRAM

2. Select process **SPECIAL** workings

DELAY MESSAGE SIZE S

3. Select voice **SIZE S** (or **SIZE H, or SIZE L**).

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :3 o :0	**SIZE S** Es: 10	a/r:a
1 INSERT U	**5** LINE	T ;1 1		
2 INSERT DOWN	**6** DIM			M2:-
3 MODIFY	**7** R. MILLING	X **:100** Y **:300**		

Description of Mode Area

1. **Axes field**

According to the selected measuring function, the two fields programming of working position result compiled as follows:

SIZE L : fields Y and Z
SIZE H : fields X and Z
SIZE S : fields X and Y

No programming corresponds to measuring axis.
Absolute or relative programming values are in units of [mm] or [inches] and only parametric format is allowed.

2. **T-field**

In Group Selection field : measuring axis should be controlled.

Only one measuring tool can be programmed. Accepted typologies are:

90 measuring probe in face 5 (thickness)
91 top measuring probe (length)
92 bottom measuring probe (length)
93 lower side measuring probe (height)
94 upper side measuring probe (height)
95 universal probe

Notes on processing

A measuring function correspond to a learning procedure on a workpiece dimension. Learning can take place upon:

a) A measuring probe tool : in this case the program on group and tool correspond to a real tool;

b) A measuring system not directly identified as a tool. An example can be the case of a workpiece in motion and length is learned by means of a photocell. In this case the program on group and tool corresponds to a dummy tool, however configured by a typology valid for a measuring tool.

Errors displayed when processing

Error 4 : value in M2-field not valid
(see DRILL xyz)

Error 4 : value in T-field not valid
It can report:

a) Dummy group or with Z pneumatic when measuring thickness.
b) Dummy group when measuring Height.

Error 5 : group not defined
(see HOLE xyz)

Error 6 : wrong syntax's in Tool field
Indicates some error in current Tool field:

a) Tool identification number minor than 1 or higher than 80.

Error 7 : wrong tool
Indicates some error in current Tool field:

a) Tool configured on a non-valid typology.
b) Tool configured on no typology at all

Error 8 : # field not defined

writing displayed at the place of character # means any field in which programming is compulsory.

The fields involved are:

a) Field : T (group and tools)
b) Field : X (for height and thickness)
c) Field :Y (for height and thickness and only if the selected working group manages controlled Y axis : that is not doomy group).
d) Field :Z (for length and height and only if the selected working group manages controlled Y axis : that is not doomy group).

Error 4 : value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Z-field not valid

For values exceeding controlled ranges: up to 99999.999 for definitions in [mm], and 3999.9999 for definitions in [inches]. The number of decimals programmable is, therefore, respectively : 3 and 4.

Syntax of ASCII blocs

```
G197  G90  G54        X..  Y..  Z..  T..  P..  E..   M..
        G91 .
              G57
-----------------------------------
compulsory fields
```

compulsory fields

G197 is function operative code and it should head the block.

G90/G91 absolute or relative programming

G54/G55/G56/G57 programming origin selection. Selections are valid in sequence from 0 to 3.

X.. Y.. Z.. Size values. Only functions without sizes are displayed.
Compulsory fields display is limited by the possibility of selecting not doomy working group
or with not controlled Z axis.

T. size group and tool

Not compulsory field:

P.. measured face. Face selecting means to indicate the kind of measuring required:
P5 size S, programmed axes : X and Y
P1 or P2 size H, programmed axes : X and Z
P3 or P4 size L, programmed axes : Y and Z

See here below the ASCII block corresponding to proposed values programmed upon display of Mode Area:

G197 G90 P5 G54 X200 Y300 T0101 E10

OFFSETS

Process definition

Programming of an Offset function can be selected only in face 5.
Offset functions use should be considered strictly related to machine specification.

Selection mode:

1. Select on Mode Menu second page:

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBPROGRAM

2. Select second page of menu : SPECIAL

OFFSET

3. Select OFFSET.

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE
1	INSERT U	**5**	LINE
2	INSERT DOWN	**6**	DIM

N :**3**
o :**0**

SIZE S
Es: **10**

M2: -

xyz: on- off off

X :**200**

3	MODIFY	**7**	R. MILLING	Y **:300** Z **:10**

Description of Mode Area

1. O-field : Origin definition

Reference Origin for programmed coordinates.

2. A/R field: select Absolute or Relative programming

This field is not displayed: programmed positions are elaborated as Absolute.

3. xyz-field :

Controls three different value-fields, one for each axis: the axes are defined in sequence as indicated by the wording: x, y, z.

In Mode Area displaying, fields selections: On, OFF and OFF are shown.
This indicates that the function has been defined only in field X, while it is disabled for the values defined in fields Y and Z.

Definition of the three values corresponding to xyz-field is guided (selection performed by pressing SPACE BAR on keyboard). Select between On and Off.

4. X/ Y/ Z/ fields

Define offset values on enabled axes.
Programming is in units of [mm] or [inches]. Values are significant with sign and allow parametric programming.

Notes on process

Offset programming does not correspond to any process performed on the workpiece, but it determines, during program execution, the setting and the imposition of an offset on one or more axes from actual program position till a differently imposed position.

When editing the program, it is considered as a transparent function for what concerns program elaboration and geometrical construction:
- It has no effect on subsequent functions programming
- It has no correspondence with graphic program displaying .

An incremental program following an offset program is applied incrementally on absolute values determined before offset programming.

Errors displayed when processing

Error 4 : value in M2-field not valid
Valid programming values are: 5 and the values included within the interval 50..99 included.

Error 4 : Value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Z-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm] and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Syntax of ASCII blocs

```
G92  G54          X.. Y.. Z..   E.. M..
   .
   G57
------------------
compulsory fields
```

G92 is the function operative code.

Here below it is proposed the ASCII block corresponding to the proposed values programmed upon display of Mode Area:

G92 G54 X200 E10

INSERTIONS

Process definition

Insertion Mode can be selected for any faces composing the panel. The insertions are divided into 7 typologies: 5 defined on specific items and two generic.

Selections modes

1. Select on Mode Menu, second page:

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBPROGRAM

2. Select second page of menu : INSERTIONS

BASE
BUSHING
HINGES
SHELVE
GENERIC 1

3. Select 7 are the item that can be selected on two pages.

Display of Mode Area

		<-	LxHxS : 1000;450;20	0003:0020
0 FILES	**4** DELETE	N :**3** o :**0**	Es: **10**	a/r:a
1 INSERT U	**5** LINE	T :1 1		
2 INSERT DOWN	**6** DIM	F :-		M2:- X :**200**
3 MODIFY	**7** R. MILLING	**Ri:-**		Y :**300** Zp:**10**

Description of Mode Area

Operating mode is similar to drilling mode, except:

M1 and S fields can't be selected (spindle rotation is not controlled in insertion mode) and the same for diameter programming;
Ro-field can't be selected (output brake)

1. **T-field : group and tool**

Working group can be defined by:

a) two controlled axes (Y and Z)
b) axis Y controlled and Z pneumatic
c) no one controlled axes (both Y and Z pneumatic)

Definition cases of group with pneumatic depth axis:

a) insertion in face 5 (Z pneumatic)
b) insertion in face 1 or 2 (Y pneumatic)

In these cases all information related to insertion axis are forcedly void; that is:
velocity F;
Ri brake value.

Two selected tools can be configured on particular values (see : heads parameters) :

30	Base in face 5
31	Bushing in face 5
32	Hinges in face 5
33	Shelves bearing in face 5
34	Generic 1 in face 5
35	Generic 2 in face 5
36	Pins in face 5
37	Universal insertion head in face 5
38	Universal insertion head in face 1, 2, 3, 4, 5
40	Universal insertion head in face 1
41	Universal insertion head in face 2
42	Universal insertion head in face 3
43	Universal insertion head in face 4
44	Universal insertion head in face 1, 2
45	Universal insertion head in face 3, 4

2. **X/ Y/ Zp fields : insertion position and deepness value**
Same as drilling (xyz).
A single remark : in case of insertion, generally insertion axis is pneumatic and correspondent coordinate not significant.

Notes on processing

Each typology is related to a symbol, used in graphic display:

base		
bushing	O	(circle with central point)
hinge	. o .	(full circle with two lateral points)
shelf bearing	--l	
generic 1	x	
generic 2	+	
pin	o	(full circle).

Errors displayed when processing

Please refer to DRILL xyz.

Syntax of ASCII blocs

G190 G90 G54 X.. Y.. Z.. T.. P.. F.. R.. E.. M..
.. G91 .
G191 G57
----
compulsory field

Compulsory fields

The following operative code define each insertion :

G190 base insertion
G191 bushing insertion
G192 hinge insertion
G193 shelf bearing insertion
G194 generic 1 insertion
G195 generic 2 insertion
G196 pin insertion

G54/G55/G56/G57 programming origin selection. Selections are valid in sequence from 0 to 3.

X.. Y.. start working position
Z.. depth value
Definition of compulsory fields on Y and Z position is limited by the possibility of selecting a dummy group or with Z axis not controlled.

T.. working tool and group;

Non compulsory fields:

P.. working face
F.. feed rate on insertion axis
R.. input brake
E.. exclusion number
M.. M2 auxiliary function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area:

G109 G90 G54 P5 X200 Y300 Z10 TO1O1 E10

DRILLING WITH DISCHARGE

Process definition

This working is selectable on all the faces.
Process is used for deep drillings, where several drillings with chips discharge are required.

Selections modes

1. Select on Mode Menu, last page, upon command INSERT U or INSERT D:

SPECIAL DRILLS

2. Select Menu SPECIAL DRILLS

DRILL WITH DISCHARGE TAPPING

3. Select working DRILL WITH DISCHARGE

Display of Mode Area

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

<- LxHxS :	**1000;450;20**	**0003:0020**
N :**3**	DRILL WITH DISCHARGE	
O :**0**	Es: -M1	a/r:**a**
		S **:200**
T :**1 1**		
F :**2-**		M2 :-
		X :**100**
R **:-2**	J :**0.5**	Y:**200**
I **:3**	K :**0.8**	Zp:**15**

Fields description

1. **T-field : group and tool**
Selects a group with controlled depth axis (non pneumatic). One single tool can be selected, if configured in drilling typology (see Drill (xyz)).

2. **F-field : Working speed**
Programming is defined in units of [mt/min] and [inch/min].
F is penetration speed in the piece : it is related to drilling axis.
If not differently programmed, speed defined in technological parameters is applied as start speed to vertical drills (for drills in face 5) and to horizontal/side drills, for the other faces.

3. **R-field : chips discharge value**
R value is start drill coordinate. Raise value for chips discharge on drill axis : it is generally programmed with negative sign.
It should result : R <=Zp ("<=" read : "smaller or equal"), where Zp means drill end depth drill.
R value is defined in units of [mm] and [inch] and allows parametric programming.

4. **I-field**
I value is increment coordinate after each extraction cycle for chips discharge. Increment on depth axis is in units of [mm] and [inch], without sign, and it allows parametric programming.

5. **J-field**
J value is the smallest increment of extraction cycle after which constant increment are performed.
J is defined in units of [mm] and [inch], without sign, it allows parametric programming.

6. **K-field :**

The value in K-field represents the reduction coefficient of Parameter 1 till value J is reached. Value K should be adimensional and it is defined with values ranging from 0 to 1 and a max. of 2 decimals.

7. **X/ Y/ Zp fields: values of centre drill and final depth**

Value meaning defined in X, Y and Z is assigned according to the face to be processed:
- face 5 : position (X and Y) depth Z
- face 1 and 2 : position (X and Z) depth Y
- face 3 and 4 : position (Y and Z) depth X

Values in X, Y and Z are defined in units of [mm] or [inches], and they are significant with sign (program "-" if negative) and allow parametric programming.

The value assigned to drilling axis displays depth by letter "p", displayed next to the letter identifying the axis. Example: if drilling in face 5 axis Z will be displayed as "Zp".

Finally, it is useful to remember that the value of depth (letter "p" next to axis name) is assigned as Absolute programming.

Notes on the process

Machine cycle related to the execution of DRILLING WITH DISCHARGE performs the following motions.

Example : drilling in face 5.

1) Case in which K and J have been programmed non-void

Rapid positioning of axes X and Y at programmed coordinates (corrected by working tool correctors)
Rapid descent to the plane defined in Air Values (See Technological Parameters).
Descent to the plane programmed in R, at brake speed.
Drive at processing speed F with a displacement equal to I.
Rapid raise to position of shaving discharge (R position).
Rapid at D mm. from working coordinate previously reached (point "d") with D as defined in Machine Cycle function.
Acknowledgement of new value of I :

I = I * K if I * K >= J
I = J if I * K < J.

Phases (e, f, and g) keep on repeating themselves till reaching programmed value of depth.
J not programmed, or J=0.0, is equivalent to not applying the reduction of parameter I

2) Case in which K and J have been programmed void (not programmed or with void values: 0.0)

Rapid motion till drilling position
Air Value management and rapid positioning to the coordinates corresponding to brake value in Ri.
drive at speed F with a displacement equal to I.
Stop for a time span defined in Machine Cycles.
Further drive with a displacement equal to I.

Phases (d, and e) keep on repeating themselves till reaching programmed depth.

Errors displayed when processing

Error 4 : value in S-field not valid
Error 4 : value in F-field not valid
Error 4 : value in M2-field not valid
(see DRILL xyz)

Error 4 : value in T-field not valid
Displayed upon:
a) programming a group identification number external to the interval 1.. 10.
b) defining a group with axis of depth not controlled.

Error 5 : group not defined
The group has not been defined in Technological Parameters.

Error 6 : wrong syntax in Tool field
Reports some error in defining Working Tool field.
a) Tool identification number lower than 1 or higher than 80.

Error 7 : wrong tool
Reports some error in defining Working Tool field.

a) Tool configured on a typology not accepted.
b) Tool configured on no typology at all (field "Head Type" in Heads parameters.)

Error 8 : # field not defined
where # means any of the field in which programming is compulsory. They are:

a) Field : T (group and tools)
b) Field : X
c) Field : Y (Only if the appointed working group knows axis Y as controlled, i.e. non-dummy group.)
d) Field : Z (only if appointed working group knows axis Z as controlled)
e) Field : I (increment value)

Error 4 : value in X-field (Xp) not valid
Error 4 : value in Y-field (Yp) not valid
Error 4 : value in Z-field (Zp) not valid
Error 4 : value in R-field not valid
Error 4 : value in I-field not valid
Error 4 : value in J-field not valid
For values exceeding programmed ranges : up to 99999.999 if programming is in [mm]; up to 3999.9999 if programming is in [inches]. The number of programmable decimals is respectively of 3 and 4.

Besides:
a) The value in field R, if different than 0.0, should result at most equal to the value of total drill depth.

Error 4 : value in K-field not valid
valid values are included within the interval 0.00 - 1.00.
Maximum number of programmable decimals is 2.

Syntax of ASCII blocs

G83 G90 G54 X.. Y.. Z.. I.. T.. P..R.. J. K.. F. S. M03 E.. M..
G91 ..
G57
----
compulsory fields

Compulsory fields

G83 is the operative code of Drilling with Discharge function and as such it should head the block.

G90/ G91 Absolute and Relative programming

G54/G55/G56/G57 Programming Origin selection. The selections are valid, in sequence, on Origins ranging from 0 to 3.

P1..5 defines the face to process. Selection is possible on any of the five faces.

X.. Y.. Tool descent position
Z.. End depth value
I.. Increment value.
T.. Working tool and group

Non compulsory fields

R.. Chips discharge coordinates.
J.. Minimum coordinates increment.
K.. Coefficient of coordinates reduction.

F.. Entry (penetration) velocity.
S.. Spindle rotational speed .
M03/M04 Spindle rotational direction .
E.. Exclusion number
M.. Auxiliary M2 function.

See here below the ASCII block corresponding to the value proposed programmed upon display of Mode Area (face processed : n. 5):

G83 G90 G54 P5 X100 Y200 Z15 T0101 R-2 I3 J0.5 K0.8 F2 S200

TAPPING

Process definition

Tapping can be performed on all five faces, and corresponds to drilling with a threaded tool.

Selection mode

1. Select on Mode Menu, last page, upon command INSERT U or INSERT D):

SPECIAL DRILLS

2. Select Menu SPECIAL DRILLS

DRILL WITH DISCHARGE
TAPPING

3. Select working TAPPING

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**TAPPING** Es: - M1 : **3-**	a/r:**a**
1 INSERT U	**5** LINE	T :**1 1**		S **:200**
2 INSERT DOWN	**6** DIM			M2 :- X :**100**
3 MODIFY	**7** R. MILLING	R **:2** K **2**	alfa : **0** beta : **0**	Y:**200** Zp:**15**

Description of Mode Area

1. **T-field : select group and tool**
This group can have depth axis not controlled (pneumatic).
Use of a single tool is allowed, configured on the following typologies:

51 Vertical tapping tool
52 Lower side lateral tapping tool (face 1)
53 Upper side lateral tapping tool (face 2)
54 Top horizontal tapping tool (face 3)
55 Bottom horizontal tapping tool (face 4).
56 Universal Tapping tool (all faces)

2. **R-field : value of end brake in entry**
Value R represents the value of approach in Rapid mode: if so, it is usually programmed negative.
In any case, it should result : R <= Zp, where symbol <= means "major or equal" and Zp represents Final Depth.

R-field should be however programmed when Editing. The values are defined in units of [mm] or [inches] and allows parametric programming.

3**. K-field**
K-field defines thread pitch in [mm] or [inches], with no sign needed.
Minimum programmable value for thread pitch is : 10 * axis X resolution.

4. **X/ Y/ ZP fields: centre drill and final depth value**

Meaning to programmed values (X, Y, and Z) is assigned according to working faces :

- face 5 : position (X, Y) depth : Z
- faces 1 and 2 : position (X, Z) depth : Y
- faces 3 and 4 : position (Y, Z) depth : X

values in X, Y, Z are defined in units of [mm] or [inches], and they are significant with sign (program "-" if negative) and allow parametric programming.

The co-ordinate assigned to the drilling axis shows the depth by letter "p", displayed next to the axis identification letter. Example: if drilling in face 5, axis Z will be displayed as "Zp".

It is useful to remember that Absolute programming is assigned to Depth value.

5. M1 and S fields :
It is compulsory to program spindle speed and rotational sense.
Spindle rotation sense programming is compulsory only if M1-field is activated in configuration; otherwise clockwise rotation is entered by default.

6.- Alfa / Beta fields : angles of tool rotation
Angles for directional posictionnig of the tool, like:
a) alfa = rotation angle on working plane
b) beta = training angle referring to axis of the tool.

Notes on process

When performing Tapping, the machine follows the following cycle (taping on face 5) :

rapid positioning of axes X and Y to programmed coordinates (corrected by working tool correctors)

rapid descent to the plane defined on Air value (See : Technological Parameters).

Descent to programmed plane on R, at brake speed.

Drive at F-speed till reaching programmed plane Zp.
Drive speed is obtained from the following:

F = S * p.

where: S = programmed rotational speed of spindle
p = Thread pitch, defined in K.

Spindle rotation reverse once reached the bottom of the drill.

Stop for a time span defined in Machine Cycles.

Return to the plane defined in R.

Errors displayed when processing

Error 4 : value in S-field not valid
Error 4 : value in M2-field not valid
(see HOLE xyz)

Error 4 : value in T-field not valid
Displayed upon:
a) definition of a number external to the interval 1..10.
b) selection of a group with axis of depth not controlled.

Error 5 : group not defined
The group has not been enabled in Technological Parameters.

Error 6 : wrong syntax in Tool field
indicates some error in defining the working tool:
a) Tool identification number lower than 1 or higher than 80.

Error 7 : wrong tool
Indicates some error in defining the working tool:
a) Tool configured on a non-valid typology.
b) Tool configured on no typology at all (field "Head Type" in Heads Parameters)

Error 8 : field # not defined
where the symbol # means any of the fields in which programming is compulsory. They are:

a) Field : T (group and tool)
b) Field : X
c) Field : Y (only if appointed group knows axis Y as controlled, i.e. not dummy)
d) Field : Z (only if appointed working group manages axis Z as controlled).
e) Field : K (thread pitch)
f) Fields : M1 and S (rotational speed and direction of spindle).

Error 4 : value in X-field (Xp) not valid
Error 4 : value in Y-field (Yp) not valid
Error 4 : value in Z-field (Zp) not valid
Error 4 : value in R-field not valid
Error 4 : value in K-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. the number of programmable decimals is respectively 3 and 4.

Besides:
a) In R-field : if the value is different than 0.0, it should result at most equal to the value defined for Total Hole Depth.

Error 4 : value in K-field not valid
The value defined for thread pitch is below minimum value.

Syntax of ASCII blocs

```
G84 G90 G54    X.. Y.. Z.. K.. T.. S..    M03     C..  V..   R.. P.. E.. M..
    G91               .                   M04
    G57
--------------------------------------------
compulsory fields
```

Compulsory fields

G84 Tapping function operative code. It should head the block.

G90/G91 Selection between Absolute or Relative programming.

G54/G55/G56/G57 To select programming origins. Selection are valid, in sequence, on values ranging from 0 to 3.

X..Y..Z Centre drill and Final depth values.
The indication of compulsory fields for Y.. and Z.. is limited to the possibility of selecting the group as dummy group, or with axis Z not controlled.

T.. Specifies working tool and group.

K.. Thread pitch

S.. Rotational speed of the spindle
M03/M04 Rotational direction of the spindle

Non compulsory fields

P1..5 To assign any of the five faces. Missing a definition face 5 is entered.

R.. Value of end brake upon entry.

E.. Exclusion number
M.. Auxiliary M2 function.

C.. Alfa angle.
V.. Beta angle

Here below it is proposed the ASCII block corresponding to the proposed values programmed upon Mode Area display (face 5):

G84 G90 G54 P5 X100 Y200 Z15 T0101 R2 K2 S200 M03

MILL SET-UP IN CARTESIAN COORDINATES

Process definition

Mill set-up starts the sections describing interpolating motion functions on axes.
Mill set-up opens a milling cycle, and performs all operations preceding a milling.

A milling cycle is defined by the operations performed between a mill set-up and the subsequent mill raise. If necessary, it can also be defined just by mill set-up.

A milling cycle is assigned :

On the processing face of the corresponding set-up.
On group and tool/s selected on corresponding set-up.
It is no necessary to program a "End Milling Cycle" mode: end cycle is automatically acknowledged.

A milling cycle can be defined as follows:

Linear interpolation
Circular interpolation
Helical interpolation

The interpolating motions above listed are all programmed on a cycle defined in face 5. On all other faces, it is possible to program only Linear Interpolation.

In the following section a complete picture of Interpolation programming modes and typologies will be explained.

Mill set-up can be programmed according to:
Cartesian coordinates (selectable on all working faces)
Polar coordinates (selectable only in face 5).

Mill set-up in Cartesian coordinates will be examined first.

Selection mode

1. Select in Mode Menu, second page, upon command INSERT U or INSERT D.

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBPROGRAM

2. Select menu MILLINGS:

SET-UP MILL (x,y,z)
SET-UP MILL (x,y; u,a)
L1 (x,y,z)
L2 (x,y; u,a)
L3 (u,a)

3. Select working SET-UP (x,y,z)

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

N :**3** o :**0**	**SET-UP MILLING** **(x,y,z)** Es:-__ M1:__	a/r: **a**
T :1 2	Rf:D- 10- 0 :___	S :**200**
F :2 Fr:0.5		M2:___ X :**200**
alfa : **0** beta : **0**		Y :1**00** Zp:**7**

Description of Mode Area

1. **O-field : define programming origin**
(see HOLE xyz)

2. **Es-field : Exclusion number**

An integer number can be programmed (no decimals) included within the interval of 1..99. If not defined, this field results empty. Es- field is displayed only if enabled in configuration.

A programmed exclusion in Mill set-up is applied automatically to the whole milling cycle.

3. **A/R field : Select Absolute or Relative programming**
The selection of absolute or relative programming is applied to non- depth axes. However absolute programming is applied to depth axis in working face.

4. **M1-field : Enable and define spindle rotation.**
S-field : Spindle rotational speed
Rotational speed defined in Parameters mode for working head will be applied if a definition in S-field is missing. Spindle rotation is defined on clockwise or c. clockwise rotation, on speed and on mill set-up, and it is valid for the whole milling cycle.

5. **T-field : select group and tools**
Working group can be defined:
a) non-dummy (with axis Y controlled) in face 5.
b) with axis Z controlled, for set-up in different faces.

Coordinated axes on working face should be therefore both controlled.
The axis perpendicular to working face can be pneumatic: in this case it cannot be defined as interpolating axis.

Max. number of selectable tools is 2. However only a single tool is displayed in set-up and further milling graphic visualisation.

Any selected tool should be configured with the following typology:
11 Pantograph in face 5
12 Top pantograph
13 Bottom pantograph
14 Lower side pantograph
15 Upper lateral pantograph
16 Top-and-bottom pantograph
17 Upper-lower side pantograph
18 Cross pantograph
19 Universal pantograph

6. **O field: tool diameter**
Tool diameter programming is alternative to group or working tool (field T) direct input. In case of programming both in T-field and in diameter area, T-field input prevails.
If tool diameter is entered not void, selection of working tool is defined by an obtimizer program, according to machine tooling. Principles used by obtimizer program are explained in a specific appendix.
Tool diameter is defined in units of [mm] or [inches]; it is significant with sign. Parametric programming is allowed.

7. **F-field : define processing speed**
Program in units of [mt/min] or [inches/sec].
F represents Mill Penetration Speed. If this parameter is missing, the one defined in Technological Parameters will be applied.

8. Fr-field : mill speed on inserted connections
Programming is defined in units of [mt/min] and [inch/min]. Fr is the interpolation speed selected on inserted connections to correct mill radius. Missing a programming, corresponding parameter value defined in technological Parameters will be applied (see : Reference speed on inserted connections).
For further details see paragraph concerning Mill radius correction.
Inserted value can't exceed max. speed of parametric interpolation.

9. **M2-field**
Used for direct flag writing in PLC. Enabled at the end of the block.

10. **Rf-field : tool radius correction**
It is managed only for processes in face 5. Possible selections are:
1. selections of correction type (right, left or void)
2. input of correction value

For correction type two selections can be performed (guided editor) :

D: Correction performed at the right side of the trajectory
S: Correction performed at the left side of the trajectory

Characters here named 'D' and 'S' are variables of national language.

A positive number can be applied to correction value, in units of [mt/min] or [inch/min], with the same format of coordinates programming.
If enabled, correction value is applied on each correction required on current milling cycle.
If milling radius value is not enabled, in case of corrections milling radius value is applied as assigned in Heads Parameters.

If milling radius correction is activated, working tool should be configured with tool radius not void.

Tool Radius correction proceeds in normal direction with respect to the programmed milling profile, made by segments of straight lines and circumference arcs.
For what concerns the application of tool radius correction, please refer to specific section.

11. **X/ Y/ Z/ fields: set-up coordinates.**
The meaning of values defined in X, Y, and Z is assigned according to working face:

- face 5 : position on plane (X, Y) approach on Z
- face 1 and 2 : position on plane (X, Z) approach on Y
- face 3 and 4 : position on plane (Y, Z) approach on X.

Values in X, Y, Z, are defined in units of [mm] or [inches], and they are significant with sign (program - if negative). Parametric programming is allowed.

It is useful to remember that Plane Approach coordinate is assigned as Absolute programming.

12. Alfa/Beta fields: rotation angles of the tool

Angles of directional positionnig of the tool, in terms of:
a) alfa = rotation angle on the working plane
b) beta = training angle, relative to the tool axis

These angles are programmables in degrees, in numerical or parametric mode.

Notes on process

A typical execution of a Mill set-up cycle takes place as follows (in face 5) :

milling tool is lifted to air value, on the selected plane xy.
It is then performed a displacement to programmed coordinate, on axis of depth (Z).

In case the programmed position should be at a lower coordinate than air value, the execution of point a) will not take place.

If the value defined in Z is positive, Mill Set-up will process the workpiece immediately; but if the value defined in Z is negative, Mill Set-up will perform the process only after a further interpolation command.

Errors displayed when processing

Error 4 : Value in S-field not valid
Error 4 : value in M2-field not valid
Error 5 : group not defined
Error 6 : wrong syntax in Tool field
Error 7 : wrong tool
(see DRILL xyz)

Error 4 : value in T-field not valid
It could be displayed by a selection on a group not enabled for axes control. Both coordinated axes related to working face should be controlled.

Error 4 : value in F-field not valid
Displayed upon:
a) definition of F-speed higher than 99 (if mt/min.) or 3900 (if inches/min), or definition of a number of decimals higher than 3 (if mt/min.) or 2 (inches/min.).

b) Programmed value is higher than the one entered in Parameters.

Error 8 : # field not defined
where the symbol # means any of the fields in which programming is compulsory. They are:

a) Field : T (group and tools), if tool diameter is not defined
b) Field : X
c) Field : Y (only if appointed working group knows axis Y as controlled , that is not doomy group).
d) Field : Z (only if appointed working group knows axis Z as controlled).

Error 8 : Rf-field not defined
Displayed upon inquiry of Tool Radius correction (left or right) when Mill Radius is not defined. Mill Radius parameter is read in Heads Parameters, by the first milling tool under process.
Visualisation is displayed only in case of tool directly programmed in T-field.

Error 4 : value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Z-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results, respectively, of 3 and 4.

Syntax of ASCII blocs

```
G88 G90 G54  X.. Y.. Z.. T.. F.  B.  P. C..  V..   S.. M03 G40 R.. E..  M..
G91                      R54=                      M04 G41
         G57
                                                       G42
 ----------------------------------
compulsory fields
```

<u>Compulsory fields</u>

G88 Mill Set-up operative code

G90/G91 selection of Absolute or Relative programming

G54/G55/G56/G57 selection of programming origin. Selections are, in sequence, on origins from 0 to 3.

X..Y..Z.. Set-up and Approach coordinate values.
Compulsory fields display is limited to the possibilities of selecting a dummy group or with axis Z not controlled.

T.. definition of working tools and group.
R 54=.. tool diameter, compulsory in case of programming in T field

<u>Not compulsory fields</u>

P1..5 : definition of working face. It is possible to select any of the 5 faces. P5 function is assigned by default, missing a definition by word P.

F.. Pantograph entry velocity.
B.. Inserted connection velocity

G40/G41/G42 control of Mill Radius correction.
G40 defines a correction not required.
G41 defines a left side correction (S)
G42 defines a right side correction (D)

R.. mill radius value correction

S.. Spindle rotational velocity
E.. Exclusion number
M.. Auxiliary M2 function.

C.. Alfa angle
V.. Beta angle

Here below it is introduced the ASCII block corresponding to proposed values programmed upon display of Mode Area (working face : n. 5) :

G83 G90 G54 P5 X200 Y100 Z7 T0102 F2 f0.5 S200 G42 R10

MILL SET-UP IN POLAR COORDINATES

Mill set-up in polar coordinates is now examined.
This process is possible only in face 5.

Selection mode :

1. Select in Mode Menu, second page, (upon commands INSERT U or INSERT D):

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

SET-UP MILL (x,y,z)
SET-UP MILL (x,y; u,a)
L1 (x,y,z)
L2 (x,y; u,a)
L3 (u,a)

3. Select working **SET-UP (x,y,z)**

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** o :**0**	**SET-UP MILLING (x,y; u,a)** Es: ____M1: __	a/r: **a**
1 INSERT U	**5** LINE	T :**1 2**	Rf:___ _____ 0 :____	S :**200**
2 INSERT DOWN	**6** DIM	F : **2** Fr: **0**		M2:- X :**200**
3 MODIFY	**7** R. MILLING	A : **30** U : **100**	alfa : **0** beta : **0**	Y :1**00** Zp:**7**

Description of Mode Area

The two fields characterising a program on polar coordinates geometry should be examined too:

1. A-field : angle
2. U-field : vector radius
Their meaning is the same as reported for DRILL in polar coordinates

3. Px/Py-field: Polar coordinates
Values in X, Y, and Z are defined in units of [mm] or [inches], and they are significant with sign (program - if negative). They allow parametric programming.
Mill set-up point is assigned at a U distance from the pole in (Px, Py) with A degrees C.clockwise rotation.

Notes on process

The related execution is valid as defined for Mill set-up in Cartesian coordinates: the geometrical definition of working position changes, but mode execution remains the same.

Errors displayed when processing

Same as the cases previously discussed for milling in Cartesian coordinates, with the addition of a few messages belonging to polar coordinates geometry, already discussed in DRILL (x,y; a, u).

Syntax of ASCII blocs

```
G88 G90 G10 G54 X.. Y.. Z.. U.. A. T.  F. B. P. C.. V..S.. M03 G40  R.  E. M.
       G91.                            R54                M04 G41
              G57                                             G42
----------------------------------------------------------------
compulsory fields
```

Compulsory field

G88 Mill Set-up operative code

G90/91 Absolute or relative programming

G10 Polar system selection

G54/G55/G56/G57 select programming Origin: Possible origin are, in sequence, on origins ranging from 0 to 3.

X..Y.. pole position
Z.. approach to working face
The indication of compulsory field in Y.. and Z.. is limited to the possibilities of selecting a dummy group, or with axis Z not controlled.

A..U.. angle and vector radius.

T.. working group and tools.

R54=.. Tool diameter, compulsory if T-field is not defined

Non compulsory fields

P1..5 : definition of working face. It is possible to select any of the 5 faces. P5 function is assigned by default, missing a definition by word P.

F.. Pantograph entry velocity.
B.. Inserted connection velocity

G40/G41/G42 control of Mill Radius correction.
G40 defines a correction not required.
G41 defines a left side correction (S)
G42 defines a right side correction (D)

R.. mill radius value correction

S.. Spindle rotational velocity
E.. Exclusion number
M.. Auxiliary M2 function.

C.. Alfa angle
V.. Beta angle.

LINEAR MILLING L1

Process definitions

A linear milling performs a motion on a straight line (linear stroke), contemporary and coordinated on all axes enabled in the programs block, with programmed processing speed and stopping modes at the end of the block.
A Linear milling can be performed in different ways, according to the geometry closer to its definition.

A list of the available modes to describe geometrically a linear milling is displayed here below.

L1 (x, y, z):	allows the motion on three Cartesian axes (x, y, and z) with direct programming of linear stroke end point.
L2 and L3 :	linear stroke is defined on plane xy, on a polar reference system.
L4 :	Linear stroke is defined on plane xy, tangent to direction of arrival on Start Milling point. Linear stroke length is assigned.

The milling indicated as L1 (x, y, z) represents the simplest way of defining a linear milling.
Milling in L1 can be selected for any of the 5 faces.

Selection mode

1. Select in Mode Menu, second page (upon command INSERT UP or INSERT DOWN)

SAW Y
MILLINGS
SPECIALS
INSERTIONS
SUBROUTINES

2. Select menu MILLINGS:

SET-UP MILL (x,y,z)
SET-UP MILL (x,y; u,a)
L1 (x,y,z)
L2 (x,y; u,a)
L3 (u,a)

3. Select working **L1 (x,y,z)**

Display of Mode Area

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

<- LxHxS : 1000;450;20 0003:0020

N :**3**	**L1 (x,y,z)**	
O :**0**	Es: ____M1:__	a/r:**a**
CT :**A**	Rf:___	F :**3**
M2 :___		
	xyz: **on_off off_**	
X :**200**		
Y : **200**		
Z : **7**		

A graphic representation of the geometry necessary to define interpolation stroke is displayed on the right side of operating mode. This happens with all Milling Modes.

Description of Mode Area

1. O-field : define programming origin
(see DRILL x,y)

2. Es-field : exclusion number
Displays the value programmed on the corresponding Mill set-up, without possibility of intervention.

3. A/R field : Select Absolute or Relative programming
The selection between Absolute or Relative programming is applied to all programmed axes.

4. F-field: define processing speed
Defined in units of {mt/min.] or [inches/min].
F is the tangential feed velocity : it is therefore related to the geometrical interpolation trajectory, divided in its components referring to all interpolating axes.

If this field is not defined, the value entered in Technological Parameters for Interpolation Speed will be applied.

5. Ct-field : contouring
This is a field with guided editor. Possible selection are:
A : Speed automatic control at the end of the block.
0 : Start new block with brake
1 : Start new block without brake.

Also the word "Contouring" is used to define this parameters. Therefore, the above mentioned selections will become:

A : Automatic contouring control
0 : Start new block with contouring disabled at end block.
1 : Start new block with contouring enabled at end block.

Please refer to proper section for a more detailed description of Contouring.

6. Rf-field : Tool radius correction
(see also MILL SET-UP)
Insert tool radius selection typology : disabled, left or right correction.
It is not allowed to switch from D-correction to S-correction (or vice versa) except when programming a disabled correction. Otherwise: the new program is ignored by the system, which will apply the value corresponding to the previous milling block.

7. xyz-field
Controls 3 value fields, one for each axis. The axes correspondence respects the sequence above described: x, y, z.
In Mode Area visualisation fields definitions are displayed: ON, OFF and OFF. This means that the function is enabled in X-field while it is disabled on the values in Y and Z-fields.

The definition of the three values corresponding to xyz-field is guided. Select proper values by means of SPACE BAR on keyboard.

8. X/ Y/ Z/ fields
Defines coordinate values (Absolute or Relative) on the corresponding axis.
If a value is programmed on a axis not defined (word OFF in corresponding enabling field) the definition is ignored.
Values in X, Y, Z are defined in units of [mm] or [inches], and they are significant with sign (program - if negative).
Parametric programming is allowed.
End Milling position is reckoned by programmed values with the application of Absolute/relative and of programmed Origin.

Notes on process

L1 working corresponds to an interpolation on a linear stroke.
Axes defined in motion can be however selected: just a single moving axis, or two, or three.

The only limitation is represented by the working group typology: an axis (Y or Z) not controlled cannot be programmed for interpolation displacements.

The following notes should be regarded as general notes for all milling functions (Set-up included).

- In a milling cycle, the data concerning working tool is displayed only on the set-up: it is not possible to change tool when milling cycle is running.

- Some milling parameters are acquired by propagation (spreading). They are:

- Origin
- F-interpolation speed
- Rf-value : Mill radius correction
- Ct-value: Contouring

Default conditions assigned to these parameters are, respectively:

- Origin as defined in Mill set-up.
- Speed parameter F corresponding to F=0
- Mill radius correction not enabled.
- Automatic contouring.

On a milling cycle (beginning with a Mill set-up function and ending on the last milling mode) the parameters are:

1. Initialised on programmed values or on default values if not programmed.
2. spread on the following blocks till differently specified.

This allows, for example, to change parameter "Rf" from disabled correction to correction "D" by simply changing its definition on the first block where required.
At the same way it is possible to change from milling radius right correction to left correction changing its definition on the first block where defined.
The same criterion is applied to velocity "F", to contouring and to Origin programming on the panel.

Errors displayed when processing

Error 4 : value in M2-field not valid
(see DRILL x,y)

Error 4 : value in F-field not valid
(see Mill set-up)

Error 8 : Rf-field not defined
Displayed upon request of mill radius correction (D or S) with mill radius void. The value of Mill Radius is read from Head Parameters, by the first milling tool performing the process.

Error 4 : value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in Z-field not valid

For values exceeding controlled ranges : up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Syntax of ASCII blocs

```
G01 G90 G54         X.. Y.. Z.. G40 G62  F.. M..
      G91 .                     G41 G63
            G57                 G42 G64
-----------
compulsory fields
```

Compulsory fields

G01 Linear milling function operative code.

G90/G91 select Absolute or Relative programming

G54/G55/G56/G57 select programming Origins. The possible selections are, in sequence, Origins from 0 to 3.

Non compulsory fields

X..Y..Z.. Final milling values. The axes indicated on display are the ones considered in motion.

G40/G41/G42 definitions to control Mill Radius correction
G40 correction not required
G41 Correction on the left side (S)
G42 correction on the right side (D)

G62/G63/G64 definitions to control contouring:
G63 Automatic contouring
G62 Start new block with contouring disabled at end block
G64 Start new block with contouring enabled at end block.

F.. Interpolation speed
M. **A**uxiliary M2 function

REMARK
Functions that can be defined are:
- origin (G54/..G57)
- contouring (G62/G63/G64)
- milling radius correction (G40/G41/G42)
- interpolation speed (F)

on interpolating blocks they are spread till different definition.
For this reason they should be codified on ASCII blocks only when modified, without duplicating programming already valid on preceding block.

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G01 G91 G54 X200 Z7 F2 G63

LINEAR MILLINGS L2 and L3

Process definition

The functions named L2 and L3 allow to program a linear stroke in polar coordinates; they can be selected only in face 5.
Linear interpolation with geometry assigned in L2 or L3 involves the coordinated axes X and Y, while the position in Z is the same.

Selection Mode

1 Select on Mode Menu, second page (upon command INSERT U or INSERT D)

SAW Y
MILLINGS
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

SET-UP MILL (x,y,z)
SET-UP MILL (x,y; u,a)
L1 (x,y,z)
L2 (x,y; u,a)
L3 (u,a)

3. Select working **L2 (x,y; u,a)** or **L3 (u,a)**

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** o :**0**	**L2 (x,y; u,a)** Es:____	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:___	Rf:___ Ax: **xy**	F:**3**
2 INSERT DOWN	**6** DIM	A :**200**	U :**100**	
3 MODIFY	**7** R. MILLING	Px:**100** Py:**700**		

Description of Mode Area

Mode Area propose again selection fields generally valid for polar geometry. They are:

1. A/R field: define Absolute or Relative programming
Programming is performed on L2 interpolation.
Absolute/relative programming is not displayed on L3, because it is not significant.

2. A-field : angle
3. U-field : vector radius
Vector radius and angle for final point location.

4. Ax-field: working plane
Defines linear interpolation plane. Programming is imposed on plane xy, without possibility of modification.

5. Px-Py fields
Centre of polar system.
If L2 is selected: coordinates Px and Py are programmable.
If L3 is selected: values defined for Px and Py are fixed and perform again the (absolute) coordinates of Start Linear Milling point as defined in L3.

Examples of programming in polar coordinates

DEFINITIONS IN L2 POLAR GEOMETRY:

P = pole (centre of polar sister)
U = vector radius
A = angle
Q = working point as reckoned.
Pi= start point of linear segment (programmed on previous block)

DEFINITIONS IN L3 POLAR GEOMETRY:

U = vector radius
A = angle
Q = working point as reckoned (linear stroke point).
Pi= start point of linear segment (programmed on previous block) and system pole

PROGRAMMING EXAMPLE:

1.	G88 G90 X0 Y0 Z5	; set-up mill {point (1)}
2.	G11 X0 Y0 A15 U40	; L2 {point (2)}
3.	G110 A60 U30	; L3 {point (3)}
4.	G110 A30 U35	; L3 {point (4)}

Errors displayed when processing

Error 4 : value in M2-field not valid
(see DRILL x, y, z)

Error 4: value in F-field not valid
(see L1)

Error 8 : # field not defined
The fields involved are:

a) Fields : Px and Py (only for L2)
b) Field : A (corner)
c) Field : U (vector radius)

Error 4 : value in X-field not valid
Error 4 : value in Y-field not valid
Error 4 : value in U-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 4 : value in U-field not valid
Displayed when the value programmed as Vector Radius is inferior than the following computed value:

10 * axis resolution in [mm]. If axis resolution is 0.05, minimum

programmable vector radius would be 0.5 mm.

Error 4 : value in A-field not valid
With a max. number of programmable decimals equal to 3.

Syntax of ASCII blocs

For L2:

```
G11 G54  X..  Y..  U..  A..          G40 G62  F.. M..
                                      G41 G63
     G57                              G42 G64
--------------------------------
compulsory fields
```

Compulsory fields

G11 L2 function operative code.

G90/G91 select Absolute or Relative programming

G54/G55/G56/G57 select programming Origins. The possible selections are, in sequence, Origins from 0 to 3.

X.. Y.. pole position

U vector
A angle

Non compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for contouring control:

F.. Interpolation speed
M.. Auxiliary M2 function

For L3:

```
G110 U..  A..        G40 G62  F.. M..
                      G41 G63
                      G42 G64
---------------------
compulsory fields
```

G110 : functions L2 and L3 operative code.

U vector
A angle

Non compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for contouring control:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G11 G54 X100 Y700 A200 U100 F3 G63

CIRCULAR MILLING (C1, C2, C3)

Process definition

A circular milling defines a circular clockwise or c.clockwise motion, contemporary and coordinated on two axes, with programmed working speed and stopping modes at the end of instruction block.
The execution of an interpolation on a circumference arc is possible in milling cycle on all the faces.

Here below a list describing available modes to define geometrically a circular milling is displayed:

Circular interpolation programming on one of the three planes xy, xz and yz. The arc is defined by : centre coordinates, final point and rotational direction. It is the only allowed programming on all the faces.

Circular interpolation programming only on plane xy with arc defined by: final point coordinates, radius and rotational sense.

Circular interpolation programming only on plane xy, with arc defined by polar geometry: centre coordinates, angle on final point and rotational sense.

The arc is defined on plane xy with assigned: arc final point, arc starting point tangent and rotational sense.

The arc is defined on plane xy with assigned: arc final point and a third point on the circumference.

In the present session only selection on C1 is examined.

Selection mode

1. Select on Mode Menu, second page (upon command INSERT U or INSERT D) :

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

L4 (tg, u)
C1 (x1, x2; c, rot)
C2 (x, y; u, rot)
C3 (c, u,a: rot)
C4 (tg; x,y; rot)

3. Select working **C1 (x1, x2; c, rot)**

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** o :**0**	**C1 (x1,y1;c,rot)** Es:____	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:-	Rf: - Ax: **xy**	F:**3** **Cw:2**
2 INSERT DOWN	**6** DIM	c1:**100** c2:**100**		
3 MODIFY	**7** R. MILLING	x1:**100** x2:**700**		

Description of Mode Area

1. **O-field : define programming Origin**
(see DRILL x,y,z)

2. **Es-field : exclusion number**
Displays programmed value on corresponding Mill set-up, with no possibility of intervention.

3. **A/R field : select Absolute or Relative programming**
Absolute or Relative selection is applied to fields x1 and x2 programming (End arc point coordinates).

4. F-field : Interpolation speed
5. Ct-field : Contouring
6. Rf-field : Tool radius correction
(see Linear milling L1)

7. **Ax-field: milling plane**
The definition of this field is guided and it is free only programming face 5. Side faces are related to selected face:

xz on faces 1 and 2
yz on faces 3 and 4

On face 5 selectable values are:

xy to select plane xy
xz to select plane xz
yz to select plane yz

Plane selection imposes the coordinates defined for fields c1, c2, x1, x2. In practice:

plane xy:	c1 and x1 define the coordinates of axis X.
	c2 and x2 define the coordinates of axis Y.
plane xz:	c1 and x1 define the coordinates of axis X
	c2 and x2 define the coordinates of axis Z
plane yz:	c1 and x1 define the coordinates of axis Y
	c2 and x2 define the coordinates of axis Z

The selected plane is valid only if both coordinated axes defined on it are controlled.

8. Cw-field : rotational direction in arc execution
The definition of this field is guided. Selectable values are:
2 clockwise rotation
3 c. clockwise rotation

Of course the definitions: clockwise and counter clockwise (c.clockwise) rotation are always related to watch pointers.

9. c1/c2 fields
Arc centre coordinates, **incremental** on the arc starting point.
Values in c1 and c2 are defined in units of [mm] and [inches], and they are significant with sign (program - if negatives). They allow parametric programming.
Arc centre position is determined by applying increments c1 and c2 to the absolute coordinates of the arc starting point, with sign imposed by the Origin programmed in field O.

10. x1/x2 fields
Arc end coordinates on selected plane.
Values in x1 and x2 are defined in [mm] or [inches] and they are significant with sign (program - if negative), absolute or relative as imposed. They allow parametric programming.

Notes on processing

Arc is defined by the following parameters:

working plane in Ax
coordinates x1,x2 of end arc point
centre coordinates, incremental on start arc point
rotation sense

Start arc point is defined on end point of preceding geometric line.

The picture displays programming with geometrical data:

Pi = start arc point
Pf (x1,x2) = end arc point
c = arc centre
c1,c2 = centre coordinates, related to start arc point

selection of rotation sense (for instance clock-wise)allows to define an arc on a circumference.

Errors displayed when processing

Error 4 : value in M2-field not valid
(see DRILL x, y)

Error 4 : value in F-field not valid
(See L1)

Error 8 : Rf-field not defined
Displayed upon an inquiry of mill radius correction (D or S) with a void mill radius value. Mill radius value is read in Heads Parameters, on the first working milling tool.

Error 8 : # field not defined
Fields involved are:

a) Fields : c1, c2;
b) field : x1, x2.

Error 4 : value in c1-field not valid
Error 4 : value in c2-field not valid
Error 4 : value in x1-field not valid
Error 4 : value in x2-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals is, respectively, 3 and 4.

Error 4 : value in Ax-field not valid
It reports a selection made on plane xz or yz with axis z not controlled.

Error 12 : wrong interpolation : R=0
It reports a radius value equal to 0 (void). Void means: with value below the resolution programmed on axis X (see: Technological Parameters).
This value is named: "Relation Y between values".

Error 12 : Ri # Rf interpolation error
It reports that the radius reckoned on arc start point and the one on end point arc are different (difference always computed on minimum value of coordinates comparison)

Syntax of ASCII bloc

```
G02 G90 G17 G54   X..   Y.. Z.. I..  J..  K.. F..   G40 G62  . M..
G03 G91 G18    .                                     G41 G63
            G19 G57                                  G42 G64
-----------------------------------------------------
 compulsory fields
```

Compulsory fields

G02/G03 operating code C1 function, on clockwise and c. clockwise rotation respectively.

G90/G91 Absolute or Relative selection

G54/G55/G56/G57 programming Origin. The possible selections are, in sequence, Origins from 0 to 3.

G17/G18/G19 interpolation plane selection
G17 corresponds to plane xy
G18 corresponds to plane xz
G19 corresponds to plane yz;

X.. Y..Z.. end point coordinates. The two words related to programmed plane are displayed
I..J..K.. centre coordinate. The two words related to programmed plane are displayed

plane xy :	centre coordinate x is on I
	centre coordinate y is on J
plane xz	centre coordinate x is on I
	centre coordinate z is on K
plane yz	centre coordinate y is on J
	centre coordinate z is on K

Non compulsory fields

G40/G41/G42 control definitions for Mill Radius correction
G40 correction not required
G41 correction S (left)
G42 correction D (right)

G62/G63/G64 control definitions for contouring:
G63 contouring automatic control
G62 stroke with disabled contouring at end block
G64 stroke with enabled contouring at end block

F.. Interpolation speed
M.. Auxiliary M2 function

Here below see the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G2 G90..G54 G17..X100 Y700 I100 J100 F3 G63

ARC PROGRAMMING IN C2

This process can be selected only on plane xy and exclusively for milling cycle in face 5.

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0	FILES	**4**	DELETE
1	INSERT U	**5**	LINE
2	INSERT DOWN	**6**	DIM
3	MODIFY	**7**	R. MILLING

N :**3** **C2 (x,y;c,rot)**
o :**0** Es:____ a/r:**a**

Ct:**A** Rf: - F:**3**
M2:- Ax: **xy** **Cw:2**

U :**100**

X :**100**
Y :**700**

Description of the fields

1. O-filed : programming origin
{se DRILL x,y}

2. Es-field : exclusion number
Displays the value programmed on set-up without any possibility of intervention

3. A/R-field : absolute/relative programming
Absolute or Relative selection is applied to X and Y fields definitions(end arc point coordinates)

4. Es-field : define processing speed
5. Ct-filed : contouring
6. Rf-field : tool radios correction
{see linear MILLING L1}

7. Ax-field : milling plane
Field definition is xy, without any possibility of modification.

8. Cw-field : rotational direction in arc execution

The definition of this field is guided. Selectable values:

2 clockwise rotation
3 c. clockwise rotation

9. U-field

Defines arc radius. Value is defined in inits of [m] or [inches], significant with sign. Parametric programming is allowed.

10. X/Y-fields

Define the coordinates of arc end on plane xy.
Value in X and in Y are defined in units of [mm] or [inches] and they are significant with sign (program - negative), absolute or relative as specified, and they allow parametric programming.

Notes on processing

An arc programmed in C2 is defined by the following parameters:

X, Y coordinates of end arc
radius
rotation sense

Start arc point is defined by end point of preceding geometrical stroke.

The picture displays the problem with its geometrical terms:

(x1,y1) = start arc point
(x2,y2) = end arc point
value of radius U allows to locate the two circles:
circle with centre in c1
circle with centre in c2

Rotational direction data (example : clockwise) allow to locate the definable arcs by (1) and (2). Execution takes place by :
---> minor arc (1), if programmed radius U is positive
---> major arc (2), if programmed radius U is negative

Value of radius U should comply the condition:

Where :

(x1,y1) - (x2,y2)	=	distance between start and end arc points
>	=	"greater or equal".

In case of an arc with start and end arc point coinciding, a complete circle is executed, defined by the following parameters :

a) coordinate x of centre = coordinates x of end point,
b) coordinate y of centre = coordinate y of end point + radius U
c) radius = U.

The picture displays the complete circle process, according to the four executable reference systems, with positive radius.

Errors displayed when processing

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
It reports an inquiry of mill radius correction (D or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined

a) Field : U
b) Field : X
c) Field : Y

Error 4 : Value in U-field not valid
Error 4 : Value in X-field not valid
Error 4 : Value in Y-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 12 : R=0 interpolation error
It reports that a radius value has been found void, where void means : of a value inferior to programmed resolution on axis X (see Technological Parameters). This value is named "Relation Y between value"

Error 4 : value in U-field not valid
Programmed radius is inferior than the half-distance between start and end arc points.

Syntax of ASCII bloc

```
G02  G90  G54   X..    Y.. U..                G40 G62  F.. M..
G03  G91                   U-..               G41 G63
          G57                                 G42 G64
------------------------------------
 compulsory fields
```

Compulsory fields

G02/G03 C2 function operating code, on clockwise and c. clockwise rotation respectively.

G90/G91 Absolute or Relative selection

G54/G55/G56/G57 programming Origin. The possible selections are, in sequence, Origins from 0 to 3.

X.. Y.. end point coordinates.

U.. arc radius, defined positive or with sign -

Non compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 control definitions for contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G2 G90..G54 X100 Y700 U100 F3 G63

ARC PROGRAMMING IN C3

This process can be selected only on xy plane and exclusively for milling cycle in face 5.

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** o :**0**	**C3 (c;u,a,rot)** Es:___	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:-	Rf: - Ax: **xy**	F:**3** Cw:**2**
2 INSERT DOWN	**6** DIM	A :**100**		
3 MODIFY	**7** R. MILLING	CX :**100** CY :**700**		

Description of fields

1. O-filed : programming origin
{see DRILL x,y}

2. Es-field : exclusion number
Displays the value programmed on set-up without any possibility of intervention

3. A/R-field : absolute/relative programming
Absolute or Relative selection is applied to definitions in Cx and Cy fields (arc centre coordinates)

4. F-field : processing speed
5. Kt-filed : contouring
6. RFC-field : tool radios correction
{see linear MILLING L1}

7. Ax-field : milling plane
Field definition is xy, without any possibility of modification.

8. Cw-field : rotational direction in arc execution

The definition of this field is guided. Selectable values:

2 clockwise rotation
3 c. clockwise rotation

9. U-field

Defines end arc point angle. Value is defined in inits of degree.
Parametric programming is allowed.

10. Cx/Cy-field

Define x and y arc centre coordinates.
Value are defined in units of [mm] or [inches] and they are significant with sign (program - negative), absolute or relative as specified, and they allow parametric programming.

Notes on processing

An arc programmed in C3 is defined by the following parameters:

a) Cx, Cy coordinates of arc centre
b) radius
c) rotation sense

Start arc point is defined on end point of preceding geometrical stroke.

The picture displays the problem with its geometrical terms:

(x1,y1) =	start arc point
(x2,y2) =	end arc point
A	end point angle
U	radius

Radius U value is computed from the distance between end arc point and programmed centre.
End point coordinates are computed on programmed geometrical data.

Errors displayed when processing

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
It reports an inquiry of mill radius correction (D or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined
Fields involved are:
a) Field : A (angle)
c) Field : Cx, Cy

Error 4 : Value in Cx-field not valid
Error 4 : Value in Cy-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 4 : Value in A-field not valid
Value in format not valid

Error 4 : Value in Cxy-field not valid
Arc centre is programmed coinciding with start arc point

Error 12 : R=0 interpolation error
It reports that a radius value has been found "void".

Syntax of ASCII bloc

```
G12  G90  G54   X..    Y.. A..                    G40 G62   M..
G13  G91  .                                       G41 G63
          G57                                     G42 G64
-------------------------------------
 compulsory fields
```

Compulsory fields

G12/G13 C3 function operating code, on clockwise and c. clockwise rotation respectively.

G90/G91 Absolute or Relative programming

G54/G55/G56/G57 Programming Origins. The possible selections are, in sequence, Origins from 0 to 3.

X.. Y.. end point coordinates.

A.. end point angle

Non compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 control definitions for contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G12 G90..G54 X100 Y700 A100 F3 G63

HELICAL MILLINGS

Process definition

A milling helical interpolation defines a trajectory to be performed with contemporary motion of the two axes belonging to the circular interpolation plane and of the axis orthogonal to the linear interpolation plane.

The circular interpolation plane is defined by xy; the orthogonal axis is by Z.
The component of the circular interpolation can be programmed in clockwise or c. clockwise rotation.
An helical interpolation can be programmed by three different modes:

HELIX C1
HELIX C2
HELIX C3

These three programming modes are similar to the correspondent cases of circular interpolation programming (C1, C2, C3) examined in the pervious sections.
The circular interpolation plane (field : Ax) is imposed to xy, without possibilities of intervention.
Besides the control of a position parameter is added on axis Z:

1. Zp field
Defines axis Z final depth, reckoned on the basis of absolute or relative selection.

Errors displayed when processing

In addition to error messages examined in the correspondent cases of circular interpolation programming, the following error messages can be displayed:

Error 8 : Ap-field not defined

Error 4 : value in Zp-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Syntax of ASCII bloc

Same as described in correspondent cases of circular interpolation programming, with the following variations:

1) operating code variant

G37 = clockwise helical milling
G38 = c. clockwise helical milling

2) information added on Z coordinate

C5 CIRCULAR MILLING

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** o :**0**	**C5 (xi,yi;x,y** Es:	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:-	Rf: - Ax: **xy**	F:**3** Cw**:2**
2 INSERT DOWN	**6** DIM	Xi:**100** Yi:**100**		
3 MODIFY	**7** R. MILLING	X :**100** Y :**700**		

Description of fields

1. O-filed : programming origin
{se DRILL x,y}

2. Es-field : exclusion number
Displays the value programmed on set-up without any possibility of intervention

3. A/R-field : absolute/relative programming
Absolute or Relative selection is applied to fields definitions

a) Xi,Yi (coordinates of circumference third point)
b) X, Y (coordinates of arc final point)

4. F-field : processing speed
5. Ct-filed : countering
6. Rf-field : tool radios correction
{see linear MILLING L1}

7. **Ax-field : milling plane**
Field definition is xy, without any possibility of modification.

8. Cw-field : rotational direction in arc execution
The definition of this field is guided. Selectable values:

2 clockwise rotation
3 c. clockwise rotation

9. Xi/Yi-fields
Define x and y coordinates of the third point of the circumference. Values is defined in inits of [m] or [inches], significant with sign (program - if negative). Parametric programming is allowed.

10. X/Y-fields
Define the x and y coordinates of arc end point.
Value in X and in Y are defined in units of [mm] or [inches] and they are significant with sign (program - if negative), absolute or relative as specified, and they allow parametric programming.

Notes on processing

An arc programmed in C5 is defined by the following parameters:

a) X and Y coordinates of end point
b) Xi and Yi coordinates of a third passage point in the circumference

Start arc point is defined by end point of preceding geometrical stroke.

The picture displays the problems with its geometrical data:

(x1,y1) = start arc point
(x2,y2) = end arc point
(xi, yi) = third point of passage in the circumference

The definition "circumference" and not "arc" is used in relation to points Xi, Yi, because the passage of the programmed arc by points Xi, Yi is not necessarily executed.

In the picture: only clockwise programming determines the passage by the third programmed point, which in this case can be indicated as middle point.

The three points defining the circumference (start, middle and end) should be distinct and not aligned.

Errors displayed when processing

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
It reports an inquiry of mill radius correction (d or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined
Fields involved are:
a) Field : X and Y
c) Field : Xi, Yi

Error 4 : Value in X-field not valid
Error 4 : Value in Y-field not valid
Error 4 : Value in Xi-field not valid
Error 4 : Value in Yi-field not valid

For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 12 : R=0 interpolation error
It reports that a radius value has been found "void".

Error 12 : Interpolation error ==
It reports that the three points are not distinct or that they are aligned (2 or 3)

Syntax of ASCII bloc

```
G112  G90  G54   X..   Y..    I..  J..            F..   G40 G62  M..
G13    G91  .                                        G41 G63
             G57                                   G42 G64
--------------------------------------------------
 compulsory fields
```

Compulsory fields

G112/G113 C5 function operative code, on clockwise and c. clockwise rotation respectively.

G90/G91 Absolute or Relative programming

G54/G55/G56/G57 Programming Origin. The possible selections are, in sequence, Origins from 0 to 3.

X.. Y.. coordinates on arc end point.

I..J.. coordinates on the third point of the circumference

Not compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for control contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G112 G90.. G54 X100 Y700 I100 J100 F3 G63

TANGENT LINE (L4)

Process definition

Tangent line programming is here below examined, related in particular to a group of special interpolation. They are:

L4 linear interpolation assigned to starting point tangent

C4 circular interpolation assigned to starting point tangent

OVAL circular interpolation assigned to an full oval or to a part of it

GROOVE groove automatic insertion (straight line) between two blocks defined by linear interpolation

CIRC. CONNECTION automatic insertion of circular connection between two blocks defined by linear interpolation

All programming can be selected on milling cycle in face 5. This section will discuss function L4.

Selection mode :

1. Select on Mode Menu, second page (upon command INSERT U or INSERT D) :

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

L4 (tg, u)
C1 (x1, x2; c, rot)
C2 (x, y; u, rot)
C3 (c, u,a: rot)
C4 (tg; x,y; rot)

3. Select working **L4 (tg, u)**

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** DIM
3 MODIFY	**7** R. MILLING

N :**3** **L4 (tg; u)**
o :**0** Es: a/r:**a**

Ct:**A** Rf: - F:**3**
M2:-

U : **200**

Description of Mode Area :

1. **O-field : defines programming Origin**
   A/R: define Absolute or Relative programming

These fields will not be displayed, because they are ininfluential on the geometrical solutions proposed by this mode

2. Es-field : exclusion number
Displays programmed value on Milling set-up, without possibility of intervention.

3. F-field: working speed
Values is defined in inits of [m] or [inches].
F is displacement tangential speed on programmed geometric line.

4. Ct field : Contouring
5. Rf field: tool radius correction
{see preceding milling}

6. U-field
Linear line length. Values is defined in inits of [m] or [inches], significant without sign. Parametric programming is allowed.

Notes on processing

Process L4 corresponds to linear interpolation on plane xy.

The linear interpolation stroke length is programmed in U.

Linear interpolation final point is reckoned in order to follow tangent direction of starting point as defined by the preceding programs block.
The main use concerns the case of preceding block defined by circular interpolation.

DEFINITIONS

x1,y1	= linear stroke starting point
tg	= tangent direction on preceding block
U	= length on linear stroke
x2,y2	= final point on L4, reckoned

The stroke on L4 linear interpolation is performed according to the following points:

[1]. if preceding block assigns a Mill set-up, the programmed value in U is ignored and the machine performs again the set-up position

[2]. if preceding block assigns a linear interpolation : a projection of preceding stroke is reckoned on plane xy and the stroke in L4 performs an elongation on projected stroke of U-length. If preceding linear stroke has a void length value on plane xy : same as case 1.

[3]. if preceding block assigns a circular interpolation on plane xz or xy: the arc is projected on plane xy in a linear stroke assigned between start and end arc points. The same principles as in point 2 will be applied to the new segment

[4]. if preceding block assigns a circular interpolation on plane xy: stroke L4 elongates on arc tangent from end point of arc itself

[5]. if preceding block assigns an helical interpolation the same criteria as in point 4 will be applied, ignoring the variation in z

[6]. a particular case can be L4 programmed after a groove or circ. connection function: in this case groove or circ. connection will not be performed (see groove or circ. connection sections)

Errors displayed when processing:

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
Displayed upon inquiry of mill radius correction (d or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 4 : Value in U-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 12 : interpolation error ==
If preceding stroke is on mill set-up or on a different interpolation than arc on plane xy with void variation in X and Y

Syntax of ASCII bloc :

```
G01   U..          F..   G40 G62  M..
                         G41 G63
                         G42 G64
------------------------
 compulsory fields
```

Compulsory fields

G01 operative code for linear milling function

U.. linear line length

Not compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for control contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G1 U200 F3 G63

TANGENT ARC (C4)

Process definition :

The C4 function defines a circular interpolation in XY plane with an tangent assigned over the initial point.

Selection mode :

1. Select on Mode Menu, second page (upon command INSERT UP or INSERT DOWN).

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

L4 (tg, u)
C1 (x1, x2; c, rot)
C2 (x, y; u, rot)
C3 (c, u,a: rot)
C4 (tg; x,y; rot)

3. Select working **C4 (tg, u)**

Display of Mode Area :

		<-	LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**C4 (tg,x,y;rot)** Es:		 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:___	Rf:___ Ax : **xy**		F:**3** Cw : **2**
2 INSERT DOWN	**6** DIM				
3 MODIFY	**7** R. MILLING	X : **200** Y : **300**			

Description of fields

1. O-filed : programming origin
{see DRILL x,y}

2. Es-field : exclusion number
Displays the value programmed on set-up without any possibility of intervention

3. F-field : processing speed
Programming is in units of [mt/min] or [inch/min].
F is tangential deplacement speed on programmed stroke.

4. Ct-field : contouring
5. Rf-field : tool radius correction
{see preceding millings}

6. Axy-field : interpolation plane
Assigns selection on plane xy, without possibility of intervention.

7. Cw-field : rotation sense
Clockwise (2) or c. clockwise (3) programming.

8. X/Y-field
Define coordinates values (absolute or relative) on plane xy.
Values in X and in Y are defined in units of [mm] or [inches] and they are significant with sign (program - if negative), absolute or relative as specified, and they allow parametric programming.

Notes on processing

Process C4 corresponds to circular interpolation on plane xy.
Arc final point is programmed in (X,Y).

Arc parameters (centre and radius) are reckoned in order to follow tangent direction of starting point as defined by the preceding programs block.

Start point can be defined by circular or linear interpolation point.

GEOMETRIC DEFINITIONS

x1,y1 = arc start point
tg = tangent direction on preceding point
x2,y2 = programmed final point

The stroke on L4 linear interpolation is performed according to the following points:

[1]. if preceding block has assigned a Mill set-up, the programmed value in U is ignored and the machine performs again the set-up position

[2]. if preceding block assigns a linear interpolation : a projection of preceding stroke is reckoned on plane xy and the stroke in C4 performs an arc of U- radius, tangent to linear stroke of start point. If preceding linear stroke has a void length value on plane xy : same as case 1.

[3]. if preceding block assigns a circular interpolation on plane xz or xy: the arc is projected on plane xy in a linear stroke assigned between the points of start and end arc. The same principles as in point 2 will be applied to the new segment

[4]. if preceding block assigns a circular interpolation on plane xy : C4 circular stroke elongates on arc tangent from end point of arc itself

[5]. if preceding block assigns an helical interpolation the same criteria as in point 4 will be applied, ignoring the variation in z

[6]. a particular case can be C4 programmed after a groove or circ. connection function: in this case groove or circ. connection will not be performed (see groove or circ. connection sections)

Errors displayed when processing

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
Displayed upon inquiry of mill radius correction (d or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined
On fields X and Y (arc final point coordinates)

Error 4 : Value in U-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 12 : interpolation error R?
It can display

a) arc start and end points coinciding on C4.
b) preceding stroke is not defined on arc in xy plane and preceding stroke start and end points are aligned with arc end point in C4.

Error 12 : interpolation error tg?
Displayed if preceding stroke is not an arc on xy plane and variations in X and Y are void (not defined tangent)

Syntax of ASCII bloc

```
G02  G90  X..  Y..          F..   G40 G62  M..
G03  G91                          G41 G63
                                  G42 G64
------------------------
 compulsory fields
```

Compulsory fields

G02/G03 operative code for circular milling function (clockwise or c. clockwise)

G90/G91 absolute or relative

X.. Y.. x and y coordinates of end arc point

Not compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for control contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below it is proposed the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G1 X200 Y300 F2 G63

OVAL

Process definition

Programming function OVAL defines consecutive circular processes, designing the geometrical profile of an oval or part of it.

OVAL programming is limited to plane xy.

It is possible to perform 1/4, half, 3/4 and full oval.

Initial and final points are coincident with the passage points of a quadrant, and axes defining the oval are parallel to coordinated axes X and Y.

Selection mode :

1. Select on Mode Menu, second page (upon command INSERT UP or INSERT DW):

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

Helic C1
Helic C2
Helic C3
OVAL
BLUNTING

3. Select working **OVAL**

Display of Mode Area :

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** o :**0**	**OVAL** Es:___	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:___	Rf: ___ Cw:**2**	F:**3**
2 INSERT DOWN	**6** DIM	Ux: **200** Ur: **70**	Uy:**300** Nq:**3**	
3 MODIFY	**7** R. MILLING	sx:**1** sy:**1**		

Description of fields

1. O-field : define programming Origin
A/R: define Absolute or Relative programming
These fields will not be displayed, because they are ininfluential on the geometrical solutions proposed by this mode : origin 0 is assigned to geometric solution of this field.

2. Es-field : exclusion number
Displays programmed value on Milling set-up, without possibility of intervention.

3. F-field: working speed
Values is defined in inits of [m] or [inches].
F is deplacement tangential speed on programmed geometric line.

4. Ct field : Contouring
5. Rf field: tool radius correction
{see preceding milling}

6. Cw-field : rotational direction in arc execution
Selectable values are clockwise (2) and c. clockwise (3) rotation

7. Ux-Uy fields : semiaxes
Assign the values of the two semiaxes of the oval, respectively on axis X and Y.
Values in Ux and Uy is defined in inits of [mm] or [inches], they do not require sign and allow parametric programming.

8. Ur field : minor radius
Assign the values of minor semiaxes. It is assigned in units of [mm] or [inches] they do not require sign and allow parametric programming.

9. Nq-field : number of quadrant or full oval
The value Nq defines:

a) quadrants number required in execution in case of parted oval
b) performed oval selection in case of full oval

In both cases it is defined with integer values ranging from 1 to 4.

10. X/Y-field
Value in sx and in sy define the increments to determine the final point of programmed profile.

Notes on processing

An oval is defined by four circumference arcs, each two equivalent with continuity of tangency on the radius variation points.

The following example represent:

a) a full oval with major semiaxis along axis X
b) a full oval with major semiaxis along axis Y

Possible start or end execution points are: A, B, C, D.

Circumference arcs performing the profile on the full oval are:

(a) arc through points (P4, A, P1), with P4 and P1 as extreme points
(b) arc through points (P1, B, P2), with P1 and P2 as extreme points
(c) arc through points (P2, C, P3),with P2 and P3 as extreme points
(d) arc through points (P3, D, P4),with P3 and P4 as extreme points

Arcs (a) and (c) have both radius r1 and centres positioned on the line passing through the points A and C.
Arcs (b) and (d) have both radius r2 and centres positioned on the line passing through the points B and D.

On the first picture minor radius is r1; on the second picture minor radius is r2.
In any case : minor radius defines arcs with centres along the line located on the major semiaxis.
Major radius is reckoned so to realise tangent continuity when changing the radius.

Oval programming implies the execution of a variable number of circular interpolations (on plane xy):

1. two interpolations when programming 1/4 of an oval;
2. three interpolations when programming 3/4 of an oval;
3. four interpolations when programming 3/4 of an oval;
4. five interpolations when programming a full oval;

The final point of programmed profile is determined as follows:
coordinate x= coordinate x on oval initial point + semiaxis x * sx
coordinate y= coordinate y on oval initial point + semiaxis y * sy.

For example :
when programming sx=1 and sy=-1 , final point is located by :
coordinate x= coordinate x of oval initial point + semiaxis x ;
coordinate y= coordinate y of oval initial point - semiaxis y ;

when programming sx=2 and sy=0 , final point is located by :
coordinate x= coordinate x of oval initial point + 2*semiaxis x ;
coordinate y= coordinate y of oval initial point.

Correct programming of values in sx, sy, Nq are listed on the following tables :

Nq	**sx sy**	**allowed combinations sx and sy**
1	$\pm$1	+1+1; +1-1; -1-1; -1+1
2	$\pm$2,0	$\pm$2 0; 0 $\pm$2
3	$\pm$1	+1+1; +1-1; -1-1; -1+1
1.. 4	0,0	

In particular two situations must be distinguished: :

1. esecution of an not completed oval :

quadrants numbers is defined in Nq, valid value range from 1 to 3;
values in sx and sy should respect the above mentioned combinations .

2. execution of a full oval :

values on sx and sy are both defined as 0;
a value between 1 and 4 should be defined in Nq to identify the oval to perform among 4 possible ovals.
More precisely :

Nq=1 -> X-coord. of oval centre = X-coord. of Initial point
Y-coord. of oval centre = Y-coord. of I. P. + semiaxis y;
Nq=2 -> X-coord. of oval centre = X-coord. of Initial p. - semiaxis x
Y-coord. of oval centre = Y-coord. of initial point
cNq=3 --> X-coord. of oval centre = X-coord. of Initial point
Y-coord. of oval centre = Y-coord. of I. P. - semiaxis y
Nq=4 -> X-coord. of oval centre = X-coord. of Initial p. + semiaxis x
Y-coord. of oval centre = Y-coord. of Initial Point.

Contouring variation or mill radius correction on oval execution, with change from enabled value (on preceding block) to disabled value for the oval, defines a disabled value only on the last oval arc performed.

Execution of full oval programming (system xy=0):

Execution of full oval programming (system xy=1):

Execution of full oval programming (system xy=2):

Execution of full oval programming (system xy=3):

Errors displayed when processing

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 8: Rf field not defined
Displayed upon inquiry of mill radius correction (d or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined

a) Ux and Uy-fields (semiaxis)
b) Ur-field (minor radius)
c) Nq-field (quadrants number)
d) sx and sy-fields (increment from initial point)

Error 4 : Value in Ux-field not valid
Error 4 : Value in Uy-field not valid
Error 4 : Value in Ur-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 4 : Value in Nq-field not valid
Programmed value not included between 1 and 4.

Error 12 : interpolation error R=0
Displayed when minor radius is void

Error 12 : R>Ux,Uy interpolation error
Minor radius has programmed value major or equal to one or both semiaxes

Error 12 : Ux=Uy interpolation error
Two semiaxis have equal value

Error 12 : R,Ux,Uy<0 interpolation error
One of the parameters has been programmed upon void value

Error 12 : sx,sy interpolation error
Programmed values on sx and sy are not correct (see preceding table for correct allocations)

Syntax of ASCII bloc

```
G02  X..  Y..  U.. U..  A..  A..        F..   G40 G62  M..
G03                                           G41 G63
                                              G42 G64
---------------------------------------------------
 compulsory fields
```

Compulsory fields

G02/G03 operative code for oval (clockwise and c.clockwise)

X.. Y.. values on sx and sy

U..U.. first address U defines x semiaxis, the second y semiaxis

A.. A.. first address A define minor radius, the second address A defines the value on Nq

Not compulsory fields

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for control contouring:

F.. Interpolation speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G2 X1 Y1 U200 U300 A70 A3 F2 G63

GROOVE AND CIRC. CONNECTION

Selection mode :

1. Select on Mode Menu, second page (upon command INSERT UP or INSERT DW):

SAW Y **MILLING** SPECIALS INSERTIONS SUBROUTINE

2. Select menu **MILLINGS:**

Helix C1 Helix C2 Helix C3 OVAL **GROOVE**

3. Select working **GROOVE** or **CONNECTION**

Display of Mode Area

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** o :**0**	**GROOVE** Es:____	 a/r:**a**
1 INSERT U	**5** LINE	Ct:**A** M2:___	Rf:____ Ax :**xy**	F:**3** Fu:**1**
2 INSERT DOWN	**6** DIM	U: **70**		
3 MODIFY	**7** R. MILLING	e1:**100** e2:**700**		

Description of fields

1. O-field : define programming Origin
A/R: define Absolute or Relative programming
Origin and absolute/relative selection when programming x1 and x2 fields

2. Es-field : exclusion number
Displays programmed value on Milling set-up, without possibility ofintervention.

3. F-field: working speed
Values is defined in units of [m] or [inches].
F is displacement tangential speed on programmed geometric line.

4. Fu-field: working speed in grooving and angular junction
Values is defined in units of [mm] or [inches].
Fu is displacement tangential speed on grooving linear stroke or on inserted junction arc.

4. **Ct field : Contouring**
5. **Rf field: tool radius correction**
{see preceding milling}

7. Ax-field
xy selection on plane xy
xz selection on plane xz
yz selection on plane yz

The plane selection imposes the coordinates assigned to fields: x1, x2. That is:

a) plane xy :	x1 assigns the coordinate on axis X
	x2 assigns the coordinate on axis Y
b) plane xz :	x1 assigns the coordinate on axis X
	x2 assigns the coordinate on axis Z
c)plane yz :	x1 assigns the coordinate on axis Y
	x2 assigns the coordinate on axis Z

8. **U-field:**
To define:

a) length of two linear strokes near programmed edge in (x1,x2) on grooving functions

b) radius of angular junction, upon junction function.

Values U are defined in units of [mm] or [inches], they are significant without sign and allow parametric programming .

9. e**1, e2 fields : edge coordinates**
To define edge coordinates.
Values in x1 and x2 are defined in units of [mm] or [inches], they are significant with sign.

Notes on processing

Grooving and Junction programming are defined between two geometrical strokes, each defined on a plane (xy, xz, yz).

When grooving: each stroke can define a displacement of one or two plane axes with no further limitations.

For example, program the first stroke with variation of X and Z coordinates,and the second stroke with variation of X and Y. The linear grooving stroke will be usually defined on a linear segment with variation of all 3 axes.

In case of junction, junction execution arc can be defined only on one plane.
For what concerns the two strokes only the coordinates of the two axes can therefore vary : for instance, X and Z for the first stroke and only X for the second stroke.
Rotation sense of junction arc is reckoned automatically on minor arc.

These programming execute:
a) two linear strokes in case of grooving
b) a linear stroke and a circular one, in case of junction

Pictures display geometrical data definitions for grooving and junction functions.
Where:

(1) is last programmed working point, preceding grooving block or junction
(2) is the edge, as programmed on grooving and junction the block
(3) is the point programmed on the block following grooving and junction

Points indicated as (?, ?) represent the reckoned points by computer:
a) extreme points of grooving and junction stroke
b) centre of angular junction

If defined, speed in Fu-field specifies interpolation speed on grooving and junction stroke. If no value is programmed in Fu, F speed is assigned also to added strokes.

GROOVING GEOMETRY

CONNECTION GEOMETRY

On pictures, execution profile is displayed by arrows.

The block programmed next to junction or grooving should define a linear interpolation stroke according one of the following modes:

L1 (x,y,z) on one or two axes in motion;
L2 (x,y ;u,a);
L3 (u;a) with pole assumed on the point programmed in grooving or junction block (grooving or junction edge);
grooving;
Circ. connection.

If next block is not usable to reckon grooving or junction geometry, only one linear displacement from the programmed edge position will be performed. In these cases, junction or grooving functions will not be performed and the program geometric data will be not defined.

Contouring variation or Mill Radius correction, with change from the enabled value of preceding interpolation to a disabled value, will apply disabled value only to end grooving or junction stroke.

Errors displayed when processing:

Error 4 : value in M2-field not valid
{see DRILL (x,y) }

Error 4 : value in F-field not valid
{see L1}

Error 4 : value in F-field not valid
same as for speed in F-field

Error 8 : Rf field not defined
Displayed upon inquiry of mill radius correction (d or S) with mill radius defined void. Mill radius value is read from Heads parameters, by the first working milling tool.

Error 8 : # field not defined

a) U-field
b) x1 and x2-fields (coordinates on edge point)

Error 4 : Value in U-field not valid
Error 4 : Value in e1-field not valid
Error 4 : Value in e2-field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 4 : Value in Ax-field not valid
Displayed when selection on plane xz or yz with axis z is not controlled.

Error 4 : Value in U-field not valid
Reports a void programming on U-field

Messages displayed only on grooving

Error 12 : U>12 interpolation error
Reports that grooving stroke initial point (reckoned point) lays external to first linear segment.

Error 12 : U>23 interpolation error
Reports that the initial point of grooving stroke (reckoned point) lays external to second segment.

Error 12 : interpolation error ?
Displayed when the block next to grooving is defined on:

a) function L1, with three axes in motion
b) function L4
c) Mill set-up
d) arc (circular, helical, oval)

Messages displayed only while performing a junction

Error 12 : U>12 interpolation error
Displayed in the following situations:

a) start junction point lays external to first segment
b) block preceding the junction is defined on a point
c) points (1), (2), (3) are coincident

Error 12 : U>23 interpolation error
Displayed in the following situations:

a) start junction point lays external to second segment
b) points (1), (2), (3) are aligned

Error 12 : interpolation error ?
Dispels one of the following cases:

a) block next to the junction is defined on a arc
b) block next to the junction is defined on L1, with all the three axes in motion (axis ON on three axes).
c) block next to the junction is defined on L4
d) block next to the junction is defined on Mill set-up.

Error 12 : interpolation error ?
Reports that on two linear strokes 1 and 3 axes result in motion.

Syntax of ASCII bloc

```
G01  G90  G54  G17  X..  Y..  Z..  U..     F..  B..    G40 G62  M..
     G91       G18                                      G41 G63
          G57  G19                                      G42 G64
-----------------------------------------------
 compulsory fields
```

<u>Compulsory fields</u>

G01 function operative code and it should head the block

G90/G91 absolute or relative

G17/G18/G19 allocation plane selection

X.. Y.. Z.. edge coordinates (program only the two coordinates valid on defined plane)

U.. grooving stroke (if negative) or junction radius (if positive)

<u>Not compulsory fields</u>

G40/G41/G42 control definitions for Mill Radius correction

G62/G63/G64 definitions for control contouring:

F. Interpolation speed
B.. grooving and junction stroke speed
M.. Auxiliary M2 function

See here below the ASCII block corresponding to the proposed values programmed upon display of Mode Area.

G1 G17 G90 X100 Y700 U70 F3 F1 G63

At the end the section describing milling, a complete view of all graphic icons is displayed. They can be used in mode definition area (Mode Area) as graphic support in programming:

SUBROUTINE

Process definition

Subroutine selection allows to include the execution of a loop defined in another program already stored in system files.

Selection mode

1. Select on Mode Menu, last page (upon command INSERT U or INSERT D) :

SAW Y MILLING SPECIALS INSERTIONS **SUBROUTINE**

2. Select **SUBROUTINE** function:

Display of Mode Area

		<-	LxHxS : 1000;450;20	0003:0020
0 FILES	**4** DELETE	N :**3** O :**0**	**SUBROUTINE** Es: ____M1:___	a/r:**a** s:**200**
1 INSERT U	**5** LINE	T:-**1 1-**	Rf: ___ 0:___	
2 INSERT DOWN	**6** DIM		L:**1-**	tl : l <-> : **off**
3 MODIFY	**7** R. MILLING	A:**0** mir:**x**		X:**100** Y:**200**

On the right side of mode area a graphic representation of the workpiece is displayed, in its five working faces.

If some functions are enabled, operating mode can be switched also to a second page, assigned as follows:

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE	N :**3** Nr:____	**SUBROUTINE** An:_____	tl : l
1 INSERT U	**5** LINE	dX:_______ dY:_______		
2 INSERT DOWN	**6** DIM	a :_______ b :_______	e :________ f :________	
3 MODIFY	**7** R. FRESA	c :_______ d :_______	g :________ i :________	

Parameters first page enables subroutine operating mode for base functions, while second page enables some particular modes, such as:
a) multiple loop call
b) parameters local assignment

Display of Mode Area :

1. O-field : define programming origin
Selects origin in X and Y fields (subroutine translation coordinates) and in dX and dY fields (translation increment) for subroutine loop

2. Es-field : Exclusion number
An integer number can be programmed (no decimals) included within the interval 1..99. If defined significant, it forces an exclusion number on all the selected subroutine.
If exclusion is not programmed all the workings correspondent to subroutine call are assigned without exclusion number.

3. A/R field : Select Absolute or Relative programming
Absolute programming is applied, no changes are allowed.

4. M1-field : Enable and define spindle rotation.
S-field : Spindle rotational speed
They are significant only if programming in T field (group + milling tool) has been defined. In this case they assign rotation sense and speed to selected tool. They are not significant separately. Only programming on both fields defines rotation sense and speed.

5. Rf field : tool radius correction
Field is defined on two value-fields.

First value-field corresponds to selection of correction type.
If programmed on D or S (right or left correction) it will be applied to all milling cycles existing in the subroutine, from set-up to milling tool raise. If field is programmed with disabled correction, milling cycle will be performed with the corrections defined in the subroutine

Second value-field forces correction value to subroutine: programming of this field is forced to each current programming in subroutine.

6. T-field : select group and tools
A single milling tool can be selected in face 5.
If programmed, the specified tool is imposed as working tool for all millings programmed in the subroutine. The group cannot be a dummy group.

7. O field: tool diameter
Tool diameter programming is alternative to group or working tool (field T) direct input, with the same application instructions valid in case of allocation in T field.
In case of programming both in T-field and in diameter area, T-fields allocation prevails.
If tool diameter is entered not void, selection of working tool is defined by an obtimizer program, according to machine tooling.
Principles used by obtimizer program are explained in a specific appendix.

Tool diameter is defined in units of [mm] or [inches]; it is significant with sign. Parametric programming is allowed.

8. L-field : subroutine number
Inserts subroutine number : valid values range from 0 to 999

The first program point will be applied in position X.. Y.. (corrected by origin application). Therefore, X and Y become the subroutine transition values.

Values in X and in Y are defined in [mm] or [inches], they are significant with sign (program - if negative) and allow parametric programming.

Programming on X.. Y.. translation values is not compulsory; only one coordinate can be programmed.

If translation position is not defined (for one or both coordinates) first working position is assumed as valid, as programmed in the subroutine. Program examples listed below show the possible cases.

10. A-field : rotation angle
Defines the angle on which subroutine workings rotate in relation to original programming.
Programming is in degree units, up to 3 decimals are allowed.
Parametric programming is allowed.
Programming with a value equal to 0 corresponds to rotation exclusion.
An-angle can be programmed with value not void only in case of subroutine loaded in face 5.

11. Mir field : specular parameter
Defines subroutine mirroring instructions. It is guided editor field, with the following selections:

-	(void field) no mirroring
x	mirroring in x
y	mirroring in y

Calling a subroutine in face 1 or 2: mirroring y selection is not available.
Calling a subroutine in face 3 or 4 : mirroring x selection is not available.

12. T field : junction on millings
It is a guided edit field, on two possible selections:

--->	(arrow towards right)	milling junction is required
!	(arrow down)	milling junction is not required

This option allows to introduce milling programmed in different subroutines, automatically eliminating set-up operations.

13. <-> field : inversion of subroutine
It is a guided edit field, with two possible selections:

off	inversion of subroutine is not required
on	inversion of subroutine is required

This option allows inversion of execution sequence of subroutine.
Selection is allowed on all panel faces.
With option ON, repeat is not allowed (see field **Nr** hereafter).

Second page contains fields assigning repetition of subroutine execution: these fields are only treated when working on face 5.

15. Nr field : loop number
Nr is loop number to apply to subroutine execution : programmable values range from 1 to 99.
In case of programming void or with value equal to 0 no repetition is introduced : subroutine is executed only once, with the same instructions assigned to the first page of menu.
Nr value allows parametric programming.

16. An field : rotation increment on repetitions
An is angle increment on each subroutine loop.
Programming is in degree units, with values valid up to 3 decimals.
Parametric programming is allowed.

17. T field : junction on milling or specific on translation
It is a guided edit field, with three possible selections:

--->	(arrow towards right)	milling junction is required
!	(arrow down)	milling junction is not required and applies translation on preceding end application point
!	(arrow up/down)	milling junction is not required and applies translation on preceding end application point

This option allows :

a) to introduce milling programmed in different subroutines, automatically eliminating set-up operations.

b) to apply following loops translation, assuming as reference point start or end point of preceding loop

18. dX/dY fields : application increments on subroutine repetition
If dX. and dY. values are incremented, application point for each subroutine loop is reckoned : calculated increments recur at each loop and they can be introduced at start or end application point of preceding loop.

dX and dY values are in units of [mm] or [inches]; they are significant with sign (program - if negative) and they allow parametric programming,

On the second page fields allotted to parameters "a, b, c," are displayed, used with subroutine parameters :

19. a, b, c, d, e, f, g, i fields
Parameters are significant with sign (program - if negative) and they allow parametric programming.

Notes on processing :

A subroutine is, at all effects, a normal program which has been written, stored, compiled and run like any other program.
When the following conditions are fulfilled, it can be used also as subroutine:

Name defined with three numbers (000 - 999) for a max. of 1000 possible subroutines.

Program entirely defined in only one face.

It can't be defined on the following processes:
- offset
- subroutine call

It should be defined on at least one execution block

It should be compiled (that is geometrically defined)

When all the conditions above listed are fulfilled, further conditions should be considered for particular limitations assignment in case of subroutine call. In particular:

selection of rotation angle A = 0 and/or mirroring not void is not allowed during fitting, repeat or grooving

selection of rotation angle A = 0 is not allowed during circular milling on xz and/or yz plane

When loading a subroutine, check the following conditions in current program:

subroutine should be defined on the same Tooling and Unit of measure as current program

A program cannot load itself

Regarding a programmed face:

if written for face 5, the subroutine may be called only in face 5
if written for face 1 or 2, may be called either in face 1 then in face 2
if written for face 3 or 4, may be called either in face 3 then in face 4

New program (current program + subroutine) should be of a size that Editor Mode can manage.

On the basis of above said conditions, subroutine 7 loads the program named 007. Programs named 7 or 07 cannot be called.

Subroutine loading is equivalent to an automatic insertion, defined by geometrical parameters (translation, rotation, mirroring, millings locking and repetition instructions) and technological (milling too, mill rotational direction and speed, tool radius correction) of practical use.

The execution of a subroutine implies the following steps:

First the system checks the existence of the required subroutine in Program directory

Then subroutine structural conditions are tested to see if usable as subroutine and if tooling and unit of measure correspond to what specified in calling program

Then also subroutine structure is tested with programmed rotation parameters (parameter : A) and mirroring (parameter : mir)

Subroutine dimension is verified : the difference between calling program and called program should not exceed total dimension of current program (in ASCII format and in elaborated format)

Once the preceding conditions are verified the subroutine is loaded in current program and its execution started

In particular:

f1) Geometrical modifications concerning translation (on programmed position X and Y), rotation (on A angle) and mirroring (on parameter : mir) are executed in sequence. In a following section application stages of geometrical transformations are examined in detail.

f2) if subroutine has assigned tool or diameter values for milling working : all current millings in subroutine are imposed as selected tool/diameter

f3) if subroutine has assigned tool or diameter values for milling working, spindle speedy and rotation sense (S and M1 fields) : all current millings in subroutine are imposed as selected rotation speed and rotation sense

f4) if subroutine has assigned tool radius D or S correction (first value in Rf field) : all current millings in subroutine are imposed as programmed correction

f5) if subroutine has assigned milling radius (second value in Rf field) : all current millings in subroutine are imposed as programmed milling radius

f6) if subroutine has assigned milling connection, first its availability is tested. If working can't be processed, selection is reset in automatic way, without warning messages.

f7) if parametric programming is assigned to a subroutine on parameters "a, b, c, ..." : the same values as defined in corresponding fields are applied to them, see menu second page

f8) subroutine repeat is then performed, if programming is activated

In subroutine processing, subroutine dimensions and comment are ignored : origins and parametric expressions programming are applied to calling piece.
When Editor mode is working on both sides : subroutine calling automatically relates :
- right side call to subroutine right side
- left side call to subroutine left side

Geometrical considerations

When a subroutine is loaded, it is possible to apply a geometrical transformation on plane xy as : translations, rotations, mirroring.
Besides milling junctions can be applied during locking between millings and subroutines.
These aspects are examined in detail here below and explained with programming examples.

Repeat development will examined later.

TRANSLATION

Programming case on X and Y translation coordinates is now examined without milling locking.

Following values are defined :

OFSX,OFSY = absolute (x,y) position assigned to initial point of subroutine;

Xp1,Yp1 = absolute (x,y) original position on initial point of subroutine;

Xpn,Ypn = absolute (x,y) original position on generic point n of subroutine;

Position Xn,Yn of n point, according to assigned translation, results so defined:

$Xn = Xpn - Xp1 + OFSX$
$Yn = Ypn - Yp1 + OFSY$.

Consider for instance to define program named **001** as below :

N :**1** **DRILL X,Y,Z**		
O :**0**	M1 :__	a/r :**a**
		S :__
T : **01 01**		
F :__		
		X : **100**
Ri:__		Y : **200**
Ro:__		Zp:**5**

N :**2** **DRILL X,Y,Z**		
O :**0**	M1 :__	a/r :**r**
		S :__
T : **01 01**		
F :__		
		X : **50**
Ri:__		Y : **50**
Ro:__		Zp:**5**

Program is defined by two holes :
- the first executed on coordinates X=100, Y=200
- the second executed on coordinates X=100+50=150, Y=200+50=250.

Program 001 is used as subroutine. Consider here below different ways of loading, as already explained. PROOF is calling program name and the following dimensions of PROOF are programmed : L=1200, H=500, S=20.

N :**1** **SUBROUTINE**		
O :**0**	M1 :__	a/r :**a**
Rf: __ ______		S :__
T : __ ___	L :**1**	tl : l
A :__		X : **500**
mir:__	Y : **300**	

they are :

OFSX=500	OFSY=300	
Xp1=100	Yp1=200	
Xpn=150	Ypn=250	where n=2

When executing the program, the two holes programmed in **001** are processed in the following positions :

X=OFSX=500	Y=OFSY=300	first hole
X=150-100+500=550	Y=250-200+300=350	second hole.

In the following call, origin 02 was changed :

N :**1**	**SUBROUTINE**		
O :**2**		M1 :__	a/r :**a**
	Rf: __ _______		S :__
T : __ ___		L :**1**	tl : l
A :__________________________________			X : **500**
mir:__		Y : **300**	

To position origins consider programmed piece dimensions on program PROOF
They are :

OFSX=L-500=700	OFSY=300	
Xp1=100	Yp1=200	
Xpn=150	Ypn=250	where n=2

When executing the program, the two holes programmed in **001** are processed in the following positions :

X=OFSX=700	Y=OFSY=300	first hole
X=150-100+700=750	Y=250-200+300=350	second hole.

Consider now the case of void programming on translation coordinates.

PROOF program is defined on :

N :**1**	**SUBROUTINE**		
O :**0**		M1 :__	a/r :**a**
	Rf: __ _______		S :__
T : __ ___		L :**1**	tl : l
A :__________________________________			X : __
mir:__		Y :__	

where no programming is defined on fields X and Y.
In this case 001 subroutine is located as it were programmed on original text. That is:

OFSX=Xp1=100	OFSY=Yp1=200	
Xpn=150	Ypn=250	where n=2

When executing the program, the tow holes programmed in **001** are processed in the following positions :

X=100	Y=OFSY=200	first hole
X=150	Y=250	second hole.

Consider now to change origin in O3 :

N :**1**	**SUBROUTINE**		
O :**3**		M1 :__	a/r :**a**
	Rf: __ _______		S :__
T : __ ___		L :**1**	tl : l
A :__			X : __
mir:__		Y :__	

always with not programmed X and Y fields.
In this case programmed coordinates of first hole in 001 are applied on origin 3. That is :

OFSX=L-100=1100	OFSY=H-200=300
Xp1=100	Yp1=200
Xpn=150	Ypn=250, where n=2

When executing the subroutine, the two holes programmed in **001** are processed in the positions computed according to point translation formula:

X=OFSX=1100	Y=OFSY=300	first hole
X=1150	Y=350	second hole.

The following figure shows this last case of subroutine calling, with a system of type 0 XY coordinates axes.
The first rectangle displays a programmed piece on program 001.
The second rectangle displays a programmed piece on PROOF program.

Also only one translation coordinate can be programmed, while the other is defined by subroutine : in this case all the explained situations should be considered.

As already said X and Y coordinates defined according to OFSX, OFSY values assign working position on the first subroutine working.

In the particular case of subroutine starting with the work " Holes on the circle ", the positionning to the programmed quotes is referred to the POLE, not to the first effective hole.

Others particular cases, like:

a) drill in polar coordinates
b) set-up in polar coordinates

Programmed coordinates positioning is executed either on programmed pole then on drilling point or milling set-up, in function of selected configuration of the Editor program.

Consider for instance the following subroutine 001:

N :**1**	**DRILL (X,Y ; u,a)**			
O :**0**		M1 :__		a/r :**a**
				S :__
T : **01 01**				
F :__				
				Px : **100**
Ri:__		A : **30**	Py : **200**	
Ro:__		U : **100**		Zp:**5**

Loading 001 subroutine with the following parameters :

N :**1**	**SUBROUTINE**			
O :**0**		M1 :__		a/r :**a**
	Rf: __ _______			S :__
T : __ ___		L :**1**		tl : l
A :**0**_			X : **400**	
mir:__			Y : **200**	

In position (X=400, Y=200) the pole (programmed on original text at X=100, Y=200) is shifted and not the hole.

ROTATION

Consider the following figure where point (2) rotation should be applied, around point (1), on angle A. Rotation has c. clockwise sense.
(2') is the resulting point when the rotation is executed.

The following relation between angles is realised : C2 = (B2 + A)

Point(1) is the reference point for (2) and is positioned in (X1, Y1).

Angle B2 defines point (2) start position, with respect to reference point.

Point (2) is positioned in (X2,Y2).

Point (2') is positioned in (X2',Y2'), that should be defined.

If $\overline{12}$ is the distance computed between the points (1) and (2).

$X2 = X1 + \overline{12} * \cos(B2)$ $\quad$ $Y2 = Y1 + \overline{12} * \sin(B2)$

$X2' = X1 + \overline{12} * \cos(C2)$ $\quad$ $Y2' = Y1 + \overline{12} * \sin(C2)$

Consider for instance to define program named **001** as follows :

N :**1** **MILLING SET-UP**
O :**0** M1 :__ a/r :**a**
Rf : __ ______ S :__
T : _**1** _**1**
F :__
Fr:__ X : **100**
Y : **100**
____Zp: **5**

N :**2** **L1**
O :**0** a/r :**r**
Ct: **A** Rf: __ F : __
xyz: **OFF ON_ OFF**

X : **100**
Y : **100**
Z : **5**

N :**3** **L1**
O :**0** a/r :**r**
Ct: **A** Rf: __ F : __
xyz: **ON_ OFF OFF**

X : **50**
Y : **100**
Z : **5**

Program is therefore defined by a cycle of two linear millings :
- set-up is on coordinates X=100, Y=100
- first linear stroke ends on coordinates X=100, Y=200
- second linear stroke ends on coordinates X=150, Y=200.

Program 001 is therefore used as a subroutine.
PROOF is the name of calling program and dimensions of PROOF : L=1200, H=500, S=20 should be programmed.

N :**1** **SUBROUTINE**
O :**0** M1 :__ a/r :**a**
Rf: __ _______ S :__
T : __ ___ L :**1** tl : l

A :**90**_____________________________X : **500**
mir:__ Y : **150**

they are:

OFSX=500	OFSY=150
Xp1=100	Yp1=100
Xp2=100	Yp2=200 , end point on first linear stroke
Xp3=150	Yp2=200 , end point on second linear stroke.

A rotation on 90 degree is required.

Executing the program, programmed positions in **001** are placed again on points (system xy=0):

X1=OFSX=500	Y1=OFSY=150	milling set-up
X2=400	Y2=150	end point of 1° linear stroke
X3=400	Y3=200	end point of 2° linear stroke.

The following figure shows this last case of subroutine loading, with a system of 0 type XY coordinates axes.
The first rectangle displays a programmed piece on program 001.
The second rectangle displays a programmed piece on PROOF program.

A rotation can't be programmed when loading a subroutine containing operations of : fitting, repeat, lame, circular interpolations on xz and yz plane.
Besides subroutine rotation can be programmed only working in face 5.

MIRRORING

Mirroring programming can be executed in x or in y.
Mirroring in x executes a rotation of the positions around a theoretic axis parallel to coordinate Y axis, crossing subroutine positioning point.
Mirroring in y executes a rotation of the positions around a theoretic axis parallel to coordinate X axis, crossing subroutine positioning point.

Figure shows the two geometric conditions concerning mirroring operations.
Displaying is realised by a general sequence of five graphic symbols.

Hereafter see the picture showing:

- a subroutine defined by a general milling cycle, on the first rectangle;
- a subroutine defined by three subroutine callings, on the second rectangle;

A call without mirroring, one with mirroring in x, one with mirroring in y are shown. Positioning points on three calls are displayed in (1), (1x), (1y)

Here after consider the programming corresponding to the cases explained in figure, with subroutine : (**001)** and calling program : (**PROOF**).

Subroutine **001** :

```
N :1        MILLING SET-UP
O :0                           M1 :__            a/r :a
            Rf : __  ______                      S :__
T : _1  _1
F :__
Fr:__                                            X : 50
                                                 Y : 50
_________________________________________________Zp: 5
```

```
N :2        L1
O :0                                    a/r :r
Ct: A  Rf: __ F : __
                           xyz: OFF  ON_  OFF

X : 50
Y : 100
Z : 5
```

```
N :3        C1
O :0                                    a/r :r
Ct: A       Rf: __          F   : __
            Ax: xy     Cw:  2
c1 : 0
c2 : -50
x1 : 50
x2 : -50
```

```
N :4    C1
O :0                                    a/r :r
Ct: A       Rf: __          F   : __
            Ax: xy     Cw:  3
c1 : 50
c2 : 0
x1 : 100
x2 : 0
```

```
N :5    C1
O :0                                    a/r :r
Ct: A       Rf: __          F   : __
            Ax: xy     Cw:  2
c1 : 50
c2 : 0
x1 : 50
x2 : 50
```

Program is therefore defined by a linear milling and three circular millings:
- set-up is on coordinates X=100, Y=100
- first linear stroke ends on coordinates X=100, Y=200
- first circular stroke (clockwise) ends on coordinates X=100, Y=100, with centre in X=50 and Y=100
- second circular stroke (c.clockwise) ends on coordinates X=200, Y=100, with centre in X=150 and Y=100;
- third circular stroke (clockwise) ends on coordinates X=250, Y=150, with centre in X=250 and Y=100

Program 001 is therefore used as a subroutine.
PROOF is the name of calling program and on PROOF dimensions : L=1200, H=500, S=20 should be programmed.

```
N :1        SUBROUTINE
O :0                        M1 :__          a/r :a
            Rf: __   _______                S :__
T : __ ___                  L :1            tl : l

A   :0__________________________________X : 100
mir:__                                 Y : 50
```

```
N :2        SUBROUTINE
O :1                         M1 :__            a/r :a
            Rf: __   _______                   S :__
T : __  ___                  L :1              tl : l

A   :0___________________________________X : 70
mir:y_                                Y : 70
```

```
N :3        SUBROGRAM
O :2                         M1 :__            a/r :a
            Rf: __   _______                   S :__
T : __  ___                  L :1              tl : l

A   :0___________________________________X : 200
mir:x_                                Y : 50
```

First block of PROOF executes a call on 001 without mirroring : origin is assigned in 00.
Second block executes a call on 001 with mirroring in y : origin is assigned in 01.
Third block executes a call on 001 with mirroring in x : origin is assigned in 02.

Mirroring programming can be combined with a rotation (parameter A not void). In this case a rotation is executed followed by a mirroring.
Application order should be defined, because they are two not commutative geometric transformations : changing execution orders end result changes.
The figure shows what explained. An original drawing is displayed showing a letter P in capital blocks. Point marked by (1) shows start point of trajectory.

At the right consider the transformation composed by a rotation of 90 degrees followed by a mirroring in x: letter P displayed in darker lines shows end result.

In the lower part consider the transformation composed by a mirroring in x followed by a rotation of 90 degree : letter P displayed in darker lines shows end result.

Obviously the two results are different.

Mirroring is not allowed when calling a subroutine containing operations of : fitting, repeat, grooving.

Working in face 1 and 2 it is possible to execute only mirroring in x.
Working in face 3 and 4 it is possible to execute only mirroring in y.

_

Parameter concerning locking call on millings is significant when executing millings.
Consider the case displayed hereafter:

In the first rectangle an original milling cycle is displayed : subroutine 001.
In the second rectangle (PROOF) program is displayed, on this one subroutine 001 is called 4 times, with millings locking : mill executes a set-up on point marked as (1) and performs a raise on point marked as (9).

Programming on PROOF is displayed hereafter :

```
N :1        SUBROUTINE
O :0                          M1 :__              a/r :a
            Rf: __   _______                      S :__
T : __ ___                    L :1                tl : l

A   :0________________________________________X : 300
mir:0_                                  Y : 100
```

```
N :2        SUBROUTINE
O :0                          M1 :__              a/r :a
            Rf: __   _______                      S :__
T : __ ___                    L :1                tl : ->

A   :270______________________________________X :
mir:__                                  Y :
```

```
N :3        SUBROUTINE
O :0                          M1 :__              a/r :a
            Rf: __   _______                      S :__
T : __ ___                    L :1                tl : ->

A   :180______________________________________X :
mir:x_                                  Y :
```

```
N :4   SUBROUTINE
O :0                          M1 :__              a/r :a
            Rf: __   _______                      S :__
T : __ ___                    L :1                tl : ->

A   :270______________________________________X :
mir:y_                                  Y :
```

On block N=1 are programmed :

- milling locking not required
- translation coordinates X=300, Y=100 : correspond to execution on point (1);
- A=0 and mir = 0.

Execution is performed with :

- set-up in X=300,Y100
- interpolation on strokes (1) ->(2), e (2) -> (3)
- in (3) milling raise is **not** executed.

Milling locking is programmed on following blocks. On X and Y fields programming is not displayed : execution on continuos milling requires start milling positioning on end point of preceding milling.

For instance : execution on block N=2 starts in (3), directly on milling cycle.

Programming displayed on parameters A (rotation) and mir (mirroring) define the execution of milling cycle from (1) to (9) as shown in figure.

Milling locking requires the following conditions to be fulfilled:

1) preceding block should check the following conditions:
   a. to be assigned on the same face where subroutine is defined
   b. block of mill set-up or milling or subroutine
   c. in case of milling block : should be geometrically defined
   d. in case of subroutine block : should end executing mill set-up or milling
   e. in case of subroutine block : should not require repetition on subroutine execution

2) subroutine called on current block should verify the following conditions:
   a. it should start on milling cycle, defined at least by an interpolation (mill set-up + at least a milling)

If all the conditions are fulfilled, locking between milling ending on preceding block and milling programmed on current subroutine is performed. When executing a junction :

1. subroutine positioning is on preceding end point, on X, Y and Z coordinates.
2. subroutine is executed with milling tool already activated
3. subroutine executes milling radius correction, as programmed on preceding block

The parameter for inversion request on the called subroutine is operative over every faces and for all types of texts.

Consider the case displayed hereafter (for a XY system typo 0):

In the first figure, is displayed a milling cycle store in subroutine 001.

In the second is displayed a complete program (named PROVA) where the subruotine 001 is called four times, following this programming sequence:

```
N :1      SOTTOPROGRAMMA
O :0                        M1 :__           a/r :a
          Rf: __   ______             S :__
T : __  ___         L :1              t|    : |
                                             <-> :off
A   :0                                X : 300
mir:0_                                      Y : 100
```

```
N :2      SOTTOPROGRAMMA
O :0                      M1 :__          a/r :a
          Rf: __   _______          S :__
T : __  ___        L :1             t| : ->
                                          <-> :on
A   :0                              X :
mir:x__                                   Y :
```

```
N :3      SOTTOPROGRAMMA
O :0                      M1 :__          a/r :a
          Rf: __   _______          S :__
T : __  ___        L :1             t| : |
                                          <-> :off
A   :180                                  X : l/2
mir:x_                                    Y : 100
```

```
N :4   SOTTOPROGRAMMA
O :0                      M1 :__          a/r :a
          Rf: __   _______          S :__
T : __  ___        L :1             t| : ->
                                          <-> :on
A   :90                                   X :
mir:0_                                    Y :
```

On the N=1 are programmed:

- milling junction not required
- translation quotes X=300, Y=100, corresponding to point (1).
- A=0 and mir=0

causing the following execution:

- set up of milling tool on X=300,Y=100
- interpolation from the point (1) until (5)
- in (5) the milling tool dont rise up

In block N=2 are called:

- Junction required
- inversion of the subroutine 001
- A=0 and mir=x

causing:

- milling continue from point (5)
- interpolation from (5) until (9) following the previous profile inverted along X (mirrored).

Execution of block N=3 begins a new milling cycle, with set up in (10). N=4 joins the subroutine with an inversion in the sequence and a 90 degrees rotation.

With inversion command activated, repeat of subroutines on the same call. is not available.

A specific aspect and an important use is local allocation, on subprogram call, of parameters "a, b, c, .." used for parametric programming of the same program.

The following examples and the other in the following sections show the different applications.

Define subprogram **001** :

a) dimensions : L=500 H=500 S=20
b) parameters : a=100 b=70 c=40 d=30

c) text :

```
N :1      MILLING SET-UP
O :0                     M1 :__          a/r :a
          Rf : __  ______                S :__
T : _1 _1
F :__
Fr:__                                    X : 20
                                         Y : 20
_________________________________________Zp: 5
```

```
N :2      L1
O :0                                a/r :r
Ct: A  Rf: __            F : __
                         xyz: ON  OFF_  OFF
X :a
Y :
Z :
```

```
N :3      L1
O :0                                a/r :r
Ct: A     Rf: __         F : __
                         xyz: OFF  ON_  OFF
X :
Y :b
Z :
```

```
N :4      L1
O :0                                a/r :r
Ct: A     Rf: __         F : __
                         xyz: ON  OFF_  OFF
X :-c
Y :
Z :
```

N :**5**	**L1**		
O :**0**			a/r :**r**
Ct: **A**	Rf: __	F : __	
		xyz: **OFF ON_ OFF**	
X :			
Y :**-d**			
Z :			

N :**6**	**L1**		
O :**0**			a/r :**r**
Ct: **A**	Rf: __	F : __	
		xyz: **ON OFF_ OFF**	
X :**-(a-c)**			
Y :			
Z :			

N :7	**L1**		
O :**0**			a/r :**r**
Ct: **A**	Rf: __	F : __	
		xyz: **OFF ON_ OFF**	
X :			
Y :**-(b-d)**			
Z :			

Program ASCII text is :

```
G71X500Y500Z20T00
G150()A100B70C40D30
G88G90G54P5T0101X20Y20Z5
G1G91X=-a
G1G91Y=b
G1G91X=-c
G1G91Y=-d
G1G91X=-(a-c)
G1G91Y=-(b-d)
M2
```

With these assignments, program is defined on milling cycle :

1) set-up	X=20	Y=20
2) linear L1 end point in	X=20+100=120	Y=20
3) linear L1 end point in	X=120	Y=20+70=90
4) linear L1 end point in	X=120-40=80	Y=90
5) linear L1 end point in	X=80	Y=90-30=60
6) linear L1 end point in	X=80-60=20	Y=60
7) linear L1 end point in	X=20	Y=60-40=20

Program 001 is now used as subprogram.
PROOF is the name of calling program and the following values are defined on PROOF :
a) dimensions : L=1000, H=1000, S=20;
b) parameters :a=l/2=500 b=h/2=500;

c) text :

1° menu page 2° menu page

N :**1** **SUBPROGRAM**	
O :**0** M1 :__ a/r :**a**	Nr:____ An:___ tl- : l
Rf: __ _____ S :__	dX:
	dY:
T : __ ___ L :**1** tl : l	
A :__________X :	a: **100** e: **0**
mir:__ Y :	b: **70** f: **0**
	c: **40** g: **0**
	d: **30** i: **0**

1° menu page 2° menu page

N :**2** **SUBPROGRAM**	
O :**3** M1 :__ a/r :**a**	Nr:____ An:___ tl- : l
Rf: __ _____ S :__	dX:
	dY:
T : __ ___ L :**1** tl : l	
A :__________X :**250**	a: **200** e: **0**
mir:__ Y :**200**	b: **90** f: **0**
	c: **100** g: **0**
	d: **20** i: **0**

1° menu page 2° menu page

N :**3** **SUBPROGRAM**	
O :**0** M1 :__ a/r :**a**	Nr:____ An:___ tl- : l
Rf: __ _____ S :__	dX:
	dY:
T : __ ___ L :**1** tl : l	
A :__________X :**a**	a: **l/4** e: **0**
mir:__ Y :**b**	b: **h/4** f: **0**
	c: **l/4-20** g: **0**
	d: **h/4-20** i: **0**

ASCII Text is :

```
G71X1000Y1000Z20T00
G150()A=l/2 B=h/2
G0G54P5L1
R60=100R61=70R62=40R63=30R64=0R65=0R66=0R67=0
G0G57P5L1X250Y200
R60=200R61=90R62=100R63=20R64=0R65=0R66=0R67=0
G0G54P5L1X=a  Y=b
R60=l/4  R61=h/4  R62=l/4-20  R63=h/4-20  R64=0R65=0R66=0R67=0
```

Hereafter consider working cycles programmed on 001 and on PROOF. Reference system used is 2 (origin 0 on XY axes in the lower right side).

First figure displays subprogram 001 : two arrows placed at the right, within piece delimitation frame, show tool lowering and raise end point.
Trapezoidal figure is executed with clockwise movement, starting from horizontal trait of length equal to **a**.

Second figure displays PROOF program : all the three executions programmed in 001 are shown in sequence, with number from (1) to (3).
Notice variations that trapezoidal figure can assume varying parameter values : a, b, c, d according which it is parametrized. Application points on three executions are defined by different modes :

a) on the first execution application point is not defined : in this case application point is the same as defined in 001 : X=20, Y=20.

b) on the second execution application point is defined by both X and Y coordinates, on origin 03. According to piece dimension in PROOF, application absolute positioning is:

X=1000-250=750
Y=1000-200=800;

c) on the third execution application point is defined in parametric form :
X=a=l/2=1000/2=500
Y=b=h/2=1000/2=500.
In this case current parameters "**a**" and "**b**" are the same as locally programmed in PROOF program.

Subprogram programming implies:

1) at least an execution of called subprogram : milling, placing, rotation, mirroring and locking modes are defined in the previous pages;

2) following executions of called program, for a number of times programmed in **Nr** field. Modes concerning geometric working applied to execution repeat are examined afterwards. Repeat can be programmed only in face 5.

Consider a programming on subprogram "200" with Nr=5 :

in correspondence to the selected program block, subprogram "200" is executed 1+5=6 times. At the first execution (main execution) repeat number n=0 is assigned; then a repeat number from 1 to 5 is assigned to the following executions, incremented at each performing.

Repetition number n is not an abstract mode but a very useful one : as explained hereafter, parameter **n** here defined can be used to parametrize programs in order to link subprogram repeat to current repeat.

SETTING UP ON REPEAT

Parameters **dX** and **dY** assign translation increments to apply cyclically to each setting up of subprogram following the first:

a) Dx and Dy are not significant in case of milling locking execution also during repeat
b) dX and dY can be applied to start or end point of previous setting up on subprogram

ROTATION ON REPEAT

A rotation on angle (A + n * dA) is applied to each repeat where :

A = rotation angle on first execution
n = repeat number
dA = rotation increment assigned on repeat parameters

MIRRORING ON REPEAT

Mirroring programmed on first execution is applied to each repeat.
Mirroring is always applied after first rotation.

MILLING LOCKING ON REPEAT

Milling locking modes available on repeat are indipendent from first execution modes .
The following conditions should be fulfilled:
a) subprogram should start with a milling set-up function
b) subprogram should end with a milling function

If these conditions are not fulfilled, milling locking is not executed, without displaying error message : in this case a translation is applied to each repeat, according to programmed values in dX and dY, with reference point on preceding point of end execution.

Following examples define different allocation modes on subprogram repeat.

Example n. 1

Define subprogram **001** :

a) dimensions : L=100 H=100 S=20
b) parameters : a=40 b=20 c=5 i=20

c) text :

```
N :1        MILLING SET-UP
O :0                    M1 :__          a/r :a
           Rf : __  ______              S :__
T : _1  _1
F :__
Fr:__                                   X : l/2
                                        Y : h/2
________________________________________Zp: c
```

```
N :2        L1
O :0                                    a/r :r
Ct: A       Rf: __      F : __
                        xyz: ON  OFF_  OFF
X :-(a+i*n)
Y :
Z :
```

```
N :3        L1
O :0                                a/r :r
Ct: A       Rf: __      F : __
                        xyz: OFF  ON_  OFF
X :
Y :b+2*i*n
Z :
```

```
N :4        L1
O :0                                a/r :r
Ct: A       Rf: __      F : __
                        xyz: ON  OFF_  OFF
X :2*(a+i*n)
Y :
Z :
```

```
N :5        L1
O :0                                a/r :r
Ct: A       Rf: __      F : __
                        xyz: OFF  ON_  OFF
X :
Y :-(b+i+2*i*n)
Z :
```

N :**6**	**L1**		
O :**0**			a/r :**r**
Ct: **A**	Rf: __	F : __	
		xyz: **ON OFF_ OFF**	
X :**-(a+i*n)**			
Y :			
Z :			

ASCII Text is as follows :

```
G71X100Y100Z20T00
G150()A40B20I20
G88G90G54P5T0101X=l/2  Y=h/2  Z=c
G1G91X=-(a+i*n)
G1G91Y=b+2*i*n
G1G91X=2*(a+i*n)
G1G91Y=-(b+i+2*i*n)
G1G91X=-(a+i*n)
M2
```

The selected program is defined on milling cycle by :

1) set-up	X=50	Y=50
2) linear L1 with end point in	X=50-40=10	Y=50
3) linear L1 with end point in	X=10	Y=50+20=70
4) linear L1 with end point in	X=10+80=90	Y=70
5) linear L1 with end point in	X=90	Y=70-40=30
6) linear L1 with end point in	X=90-40=50	Y=30

In particular : value = 0 is substituted to parameter **n (execution repetition)** , being not activated execution on subprogram repeater.

Program 001 is now used as subprogram.
PROOF is the name of calling program and the value on PROOF are :
a) dimensions : L=2000, H=900, S=20;
b) parameters :a=l/2=1000 b=h/2=450;

c) text :

1° menu page			2° menu page	
N :**1**	**SUBPROGRAM**			
O :**0**	M1 :__ a/r :**a**		Nr:**2**___	An:___ tl- :**->**
	Rf: __ _____ S :__		dX:	
			dY:	
T : __ ___	L :**1**	tl : l		e:
A :__________	X :**l/2**		a: **50**	f:
mir:__	Y :**h/2**		b: **20**	g:
			c: **5**	i: **40**
			d:	

Figure displays subprogram 001 : two arrows placed at extreme points of programmed profile, show tool lowering and raise end point.
The open figure is executed with clockwise movement, starting from horizontal trait of length equal to **a**.

Used reference system is of type 2 (origin on XY axes at the lower right side). Values displayed on the figure sides are the assigned parameters, defined in parametric mode on parameters : a, b, and i. Reference to parameter n is not displayed, because programming does not correspond to subprogram repeat: parameter n is equal to 0.

Following figure displays PROOF program : execution programmed on 001 is shown :

a) tool lowering point is marked by an arrow turned toward bottom, at the centre of the figure;
b) tool raise point is marked by an arrow turned towards top, placed at the final point of the figure.

Execution of one milling profile corresponds to repeat programming in milling locking.
Repeat number is Nr=2, for a total of three executions of defined profile in 001.
Example here displayed on program 001 can correspond to the execution of a simple drill of a rectangular emptying.
Parameter **i** forces shifting among following repeat working.

Execution depth on rectangular emptying is here defined by parameter c, programmed with value 5.

Example n. 2

Program 001 in the example n. 1 is used as subprogram in program **PROOF**, with :

a) dimensions : L=2000, H=900, S=20;

b) test :

1° menu page	2° menu page
N :**1** **SUBPROGRAM** O :**0** M1 :__ a/r :**a** Rf: __ _____ S :__ T : __ ___ L :**1** tl : l A :**45**_______________X :**1200** mir:__ Y :**500**	Nr:**1**___ An:___ tl- :**->** dX: dY: e: a: **50** f: b: **20** g: c: **5** i: **40** d:

Where :
a) application point is defined in
X=1200
Y=500;

b) locking with preceding milling is not required;
c) rotation on A=45 is required;
d) repeat on Nr=1 is programmed, with milling locking;
e) parameters a, b,c and i are defined as in the preceding example .

Following figure displays development corresponding to program on PROOF
Used reference system is of type 2 (origin on XY axes at the lower right side).

In particular : emptying is now blended of 45 degrees, as assigned is A field.

Example n. 3

Program 001 in the example n. 1 is used as subprogram in program **PROOF**, with :

a) dimensions : L=2000, H=900, S=20;

b) text :

1° menu page	2° menu page
N :**1** **SUBPROGRAM** O :**0** M1 :__ a/r :**a** Rf: __ _____ S :__ T : __ ___ L :**1** tl : l A :________________X :**900** mir:__ Y :**300**	Nr:**1**___ An:**20**_ tl- :**->** dX: dY: e: a: **50** f: b: **20** g: c: **5** i: **40** d:

Where :
a) application point is defined in
X=900
Y=300;

b) locking with preceding milling is not required;
c) repeat on Nr=2 is programmed, with milling locking;
d) Value dA=20 is programmed
e) parameters a, b,c and i are defined as in the preceding example .

Following figure displays development corresponding to program on PROOF
Reference system used is of type 2 (origin on XY axes at the lower right side).

In particular : repeat working number 1 is blended of 45 degrees, as assigned is dA field.

Example n. 4

Program 001 in the example n. 1 is used as subprogram in program **PROOF**, with :

a) dimensions : L=2000, H=900, S=20;

b) text :

1° menu page	2° menu page
N :**1** **SUBPROGRAM**	
O :**3** M1 :__ a/r :**a**	Nr:**2**___ An:_ tl- :l
Rf: __ _____ S :__	dX:**250**
	dY:**50**
T : __ ___ L :**1** tl : l	e:
A :________________X :**200**	a: **50** f:
mir:__ Y :**100**	b: **20** g:
	c: **5** i: **40**
	d:

Where :
a) application point is defined on origin 3, as follows:
X=l-200=1800
Y=h-100=800;

b) locking with preceding milling is not required;
c) repeat on Nr=2 is programmed, without milling locking;
d) increment of repeat setting up are programmed equal to:
dX=250
dY=50
that should be applied to preceding end execution point.
e) parameters a, b,c and i are defined as in the preceding example .

Following figure displays development corresponding to program on PROOF
Used reference system is of type 2 (origin on XY axes at the lower right side).

Start milling set-up and end milling raise are performed on each execution:

a)first execution performs a milling set-up in (5) and a milling raise in (2)
b) second execution performs a milling set-up in (3) and a milling raise in (4)
c) third execution performs a milling set-up in (5) and a milling raise in (6).

Between a milling raise and following lowering point such distance is kept :
-250 in x
-50 in Y
Negative sign is compulsory when programming on origin 3.
Each execution in 001 is developed on parameters a, b, on the programmed parameters of subprograms and on the correspondent value joined to repeat parameter n : this causes a progressive development of major linear traits, at following executions.

Error 4 : value in S-field not valid
a) if programmed value is higher than 32000 or than defined value in technological parameters at voice "spindle rotation turns number correspondent to analogical tension of 10 volt"

b) if programmed value in S-field (spindle rotation speed) is higher than min and max rotation speed values, as defined in heads parameters. Displaying in case b) is available only if group + tool in T-field are defined.

Error 4 : T-field not valid
programming of a group number external to interval 1 .. 10;

Error 5 : group disabled
the group is not enabled in Technological Parameters.

Error 6 : wrong syntax in Tool field
displayed upon some errors in defying working tool :

a) tool identification number below 1 or above 80.

Error 7 : tool not valid
displayed upon some errors in defying tool field :

a) tool configured in non-valid typology (different than milling tool on current face);

b) tool configured on no typology at all (field Heads type, in Heads Parameters : void).

Error 8 : # field not defined
the character # means any field for which programming is compulsory. Fields involved are :

a) field T : programming is not compulsory, but the possibility of specifying a single group or tool is excluded;
b) field A (rotation);
c) field L (subprogram number).

Error 8 : Rf field not defined
Displayed if tool radius has been defined for programmed tool (Heads Parameters). This error can be displayed only if a R-or L-- correction (right/left) has been programmed in Rf-field, with specification also on group+tool.

Error 4 : X -field not valid
Error 4 : Y -field not valid
Error 4 : Rf -field not valid
For values exceeding controlled ranges: up to 99999.999 for programs in [mm]; and up to 3999.9999 for programs in [inches]. The number of programmable decimals results respectively 3 and 4.

Error 4 :A-field not valid
displayed when a wrong angle has been programmed.
Message may indicate that subprogram does not allow rotation not void

Error 4 : mir-field not valid
Message may indicate that subprogram does not allow mirroring selection already programmed.

Error 4 : L value not valid
Displayed when:
a) programmed subprogram number is not valid;
b) subprogram does not exist in Programs Directory;
c) subprogram is defined on a different Tooling and/or Unit of measure.
d) subprogram can't be loaded as such (impossible to compile or defined on not compatible operations, ..);
e) subprogram is coincident with current program.

Syntax on ASCII

First block :

G0 G54 L. A.P.X.Y.T. S. M3 G41 R. E. R50=. R53=. I. J. R51=. R52=.
..G57 R54=. M4 G42 R55=

compulsory fields

Second block :

R60=.. R61=.. R62=.. R63=.. R64=.. R65=.. R66=.. R67=..

Division in two blocks is compulsory if total length is above the 200 characters allowed. Otherwise it is allowed to define subprogram call function on one program only.
If two blocks are required it is compulsory to respect the division above displayed, defining second block on parameters R60 - R67.

Compulsory fields :

G0 is operating code of subprogram function and should head the block;

G54/G55/G56/G57 select programming Origins. Possible selections are, in sequence, Origins from 0 to 3.

L.. subprogram number;
A.. rotation angle;

Non compulsory fields :N
P1..5 working face. Selection is allowed on all the 5 faces. If function is not programmed, face 5 is defined by default.

X..Y.. subprogram application coordinates.

E.. exclusion number;

T.. group and milling tool definition
R54=.. tool diameter, alternative to programming on T function;

S.. spindle rotational speed;
M03/M04 spindle rotational sense;

G41/G42 milling radius correction;
R.. milling radius correction value;

R53=.. mirror x or y. In particular :
R53=1 mirror x
R53=2 mirror y .
If not present : not programmed mirroring.

R50=.. selection of milling locking in first execution and of milling locking or translation increments applications on current repeating. In particular :

R50=nm two figures are displayed after n and m:

m =0	milling locking not programmed on 1° execution;
m =1	milling locking programmed on 1° execution;
n =0	milling locking not programmed on repetitions execution. Displacement of each repetition is related to final set up position of preceding execution.
n=1	milling locking programmed in repeat executions;
n =2	milling locking not programmed in repetitions execution. Displacement of each repetition is related to final set up position of preceding execution

I..J.. increment coordinates of a subprogram, when executing repeat

R52=.. increment angle of subprogram increment, when executing repeat

R51=.. repeat numbers.

R60=. R67=. selection of subprogram "a, b, c.. " parameters.

Here below see the ASCII block corresponding to the proposed values programmed upon display of Mode Area (working face 5).

G0 G54 X100 Y200 A0 L1 T0101 S200 R53=1

INDEFINITE GEOMETRIC INTERPOLATION

Program graphic display, continuously updated during programming, provides for the case of situations geometrically not valid or not completely defined.

They are always situations connected to milling profiles : in fact, a milling block is not completely defined if taken by itself .

The initial position of a milling trait is however assigned on the preceding x-process trait.

The modification of a milling trait can imply modifications in the start and end working positions of a milling process

c) The position of end milling can be defined only on the process corresponding to next block. This concerns cases of Junction and Grooving.

As example can be considered the insertion of a grooving function as final process of a contour. In this case, grooving will be defined by two points:

a) starting point (defined on preceding block)
b) Edge point (programmed in grooving)

Such geometric data cannot perform a correct grooving as the third final point is missing. Grooving programming is therefore performed as follows:

when a grooving is introduced, wrong diagnostic situation is not displayed

geometric visualisation is displayed with a single linear trait : from starting point to programmed edge point

geometric displaying of linear trait is marked by a dedicated colour, as assigned in editor configuration, to show line particularity

If a milling cycle ends with a grooving function, when displayed :

an acoustic beep is heard
an error message "???" is displayed, in colour contrast, near to the field showing piece dimensions
during recording, when the program is loaded, an exclamation mark is displayed : this means that the program cannot be compiled because assigned with geometric data not completely defined.

On program directory if a program cannot be compiled character "!" is visualised in the field displaying programmed sides.

Geometrically wrong or not defined situations are listed here below:

circular interpolation (helix) with radius on arc start point and radius on end arc point computed of different value. Geometric displaying turns an arc into a linear trait crossing start and end arc programmed points

L4 function (linear segment assigned per tangent) preceded by linear or circular interpolation on xz or yz plane, with variations void on X and Y axes. Geometric displaying turns linear segment into a point, on starting point.

C4 function (arc assigned per tangent) preceded by :

- linear or circular interpolation on xz or yz plane, with variations void on X and Y axes.

- linear or circular interpolation on xz or yz plane, with start or end points ranged into end arc point in C4.

- coincident start and end arc point

Geometric displaying turns arc into a linear trait crossing start and end points of arc programmed.

grooving or junction with :

- following block assigned on operation different than milling

- following block assigned on operation of type : L4, arc (circular, helicoidal, oval)

- extreme points computed on grooving or junction trait (one or both) external to preceding or following geometric lines

In any case, grooving or junction are performed with a single geometric line, of linear typology, ending on programmed edge point.

TOOL RADIUS CORRECTION

Tool radius correction is usually applied to the profile, in the plane related to the programmed face:

plane XY	for the face 5
plane XZ	for the faces 1 and 2
plane YZ	for the faces 3 and 4

Default correction value is equal to tool radius as defined in Heads Parameters.
Another correction value can be assigned, when programming a milling set-up and when calling a subprogram.

Upon demand of tool radius correction, the control starts performing a set of trajectories internal or external to programmed profile, also with non-tangent traits to programmed profile, and with the possible insertion of junctions on circle arcs with radius equal to tool radius.

Programming takes place in Rf field, examined in sections dedicated to interpolation blocks, with possibility of:

a) disabled correction
b) left side correction
c) right side correction

Right or left sides are determined by following orientation sense of programmed profile.

Correction orientation change (left-right and vice versa) requires that correction should be first cancelled.

Start correction

The first block on which correction (left and right) should operate will be corrected:

a) at the arrival point according to normal trajectory defined by next block, if correction is operating on a linear milling block,. therefore : no set-up or circular interpolation

b) directly on staring point in case of set-up or circular milling block

End correction

In the block of correction cancellation, correction is applied:

a) with gradual ending linearity in case of linear interpolation block

b) with ending at arrival point in total correction, in case of circular milling block.

If a milling cycle is ending with an enabled mill radius correction the correction will be performed till the milling ending point.

START LINE CORRECTION WITH GRADUAL LINEAR ENTRY

G88 Xo Yo Zo..
G01 G41 Xa Ya ..
G01 Xb Yb ..

Points indicated by letters **o**, **a** and **b** correspond to programmed positions :

1) Point **o** is milling set-up position, with void tool radius correction;

2) **a** is final point on linear trait, with left side tool radius correction;

3) **b** is final point on linear trait with unchanged tool radius correction value.

Corrected profile starts from point **o** (programmed set-up point) and ends with gradual (linear) correction on normal (perpendicular line) to the programmed linear trait programmed from point **a** to point **b**, at a distance from **a** equal to tool radius. . The corrected profile continues then parallel to linear trait (**a**)-(**b**).
Circle in figure represents mill dimension.

START ARC CORRECTION WITH GRADUAL LINEAR ENTRY

G88 Xo Yo Zo ..
G01 G41 Xa Ya .....
G02 Xb Yb .....

Points indicated by letters **o**, **a** and **b** correspond to programmed positions :

1) Point **o** is milling set-up position, with void tool radius correction;

2) **a** is final point on linear trait, with left side tool radius correction;

3) **b** is final point on linear trait with unchanged tool radius correction value.

Corrected profile starts from point **o** (programmed set-up point) and ends with gradual (linear) correction on normal (perpendicular line) to the programmed circular trait programmed from point **a** to point **b**, at a distance from point **a** equal to tool radius. . The corrected profile continues then parallel to an arc concentric to programmed arc, assigned with radius=origin radius + tool radius value.

END LINE CORRECTION WITH GRADUAL LINEAR EXIT

G41 Xa Ya...
G01 G41 Xb Yb ..
G01 G40 Xc Yc ..

Points indicated by letters **a, b** and **c** correspond to programmed positions:

1) Point **a** is arrival position anyway defined, on milling profile with left side tool radius correction;

2) **b** is final point on linear trait, with left side tool radius correction;

3) **b** is final point on linear trait with void tool radius correction.

Corrected profile check starts from corrected point **a** and continues parallel to linear programmed trait (from **a** to **b**), at a distance from programmed trait equal to tool radius value.

Profile continues with linear trait till programmed point c, gradually reducing tool radius correction.

END ARC CORRECTION WITH GRADUAL LINEAR EXIT

G42 Xa Ya ..
G03 G42 Xb Yb ..
G01 G40 Xc Yc ..

Points indicated by letters **a, b** and **c** correspond to programmed positions:

1) Point **a** is arrival position anyway defined, on milling profile with right side tool radius correction;

2) **b** is final point on circular trait, with tool radius unchanged correction;

3) **c** is final point on linear trait with void tool radius correction.

Corrected profile check starts from corrected point **a** and continues on an arc concentric to programmed arc with radius = programmed radius + tool radius. End point of corrected arc lays at a distance = useful radius from **b** point, on the normal to right line tangent to arc in **b**.

Profile continues with linear trait till programmed point c, gradually reducing tool radius correction.

START LINE CORRECTION WITH IMMEDIATE CORRECTION

G88 G41 Xo Yo Zo .
G01 Xa Ya.....
G01 Xb Yb.....

Points indicated bY letters **o**, **a** and **b** correspond to programmed positions :

1) Point **o** is milling set-up position, with left side tool radius correction;

2) **a** is end point on linear trait, with tool radius unchanged correction;

3) **b** is end point on linear trait with unchanged tool radius correction value.

Corrected profile starts at a distance= useful radius from point **o** (programmed set-up point) on the normal (perpendicular line) to the programmed linear trait from **o** to **a** points, continues parallel to line **(o)-(a)**, till intersection point i on the corrected trait parallel to segment from **a** to **b.**

The corrected profile continues then parallel to the linear trait **(a)-(b)** till the projection corresponding to point **b**.

START ARC CORRECTION WITH IMMEDIATE CORRECTION

G88 G41 Xo Yo Zo
G02 G41 Xa Ya
G01 Xb Yb

Points indicated by letters **o**, **a** and **b** correspond to programmed positions :

1) Point **o** is milling set-up position, with left side tool radius correction;

2) **a** is arc end point, with tool radius unchanged correction;

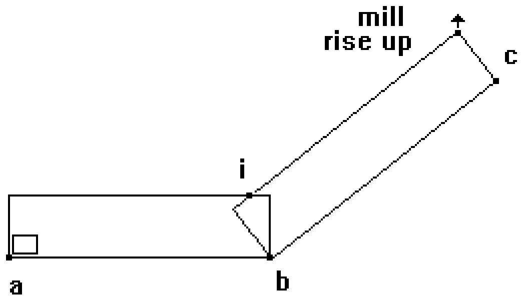
3) **b** is the end point of linear trait with unchanged tool radius correction

Corrected profile starts on point **o1** at a distance= tool radius from point **o** (programmed set-up point) on the normal (perpendicular line) in **o** to programmed arc; it continues on a concentric arc to programmed arc, with radius = programmed radius + tool radius, till reaching intersection point **i** on corrected trait parallel to segment from **a** to **b.**

The corrected profile continues then parallel to the linear trait **(a)-(b)** till the projection corresponding to point **(b)** and indicated as **(b1)**.

END LINE CORRECTION WITHOUT GRADUAL LINEAR EXIT

G41 Xa Ya....
G01 G41 Xb Yb ..
G01 G41 Xc Yc ..
{ mill raise}



Points indicated by letters **a, b** and **c** correspond to programmed positions:

1) Point **a** is arrival position anyway defined, on milling profile with left side tool radius correction;

2) **b** is final point on linear trait, with left side tool radius correction;

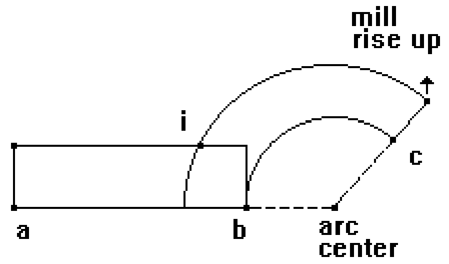
3) **c** is the final point of linear trait with void tool radius correction.

Corrected profile check starts from corrected point **a** and continues parallel to linear trait programmed by (a) and (b), at a distance from programmed line equal to tool radius. The trait continues till crossing intersection point (i) on the second linear programmed segment.

On the second corrected trait : profile continues at a distance = programmed trait tool radius, till the point projected on (c); in this letter point is defined Mill Raise.

END ARC CORRECTION WITHOUT GRADUAL LINEAR EXIT

G41 Xa Ya .
G03 G41 Xb Yb .
{mill raise}



Points indicated by letters **a**, **b** and **c** correspond to programmed positions:

1) Point **a** is arrival position anyway defined, on milling profile with left side tool radius correction;

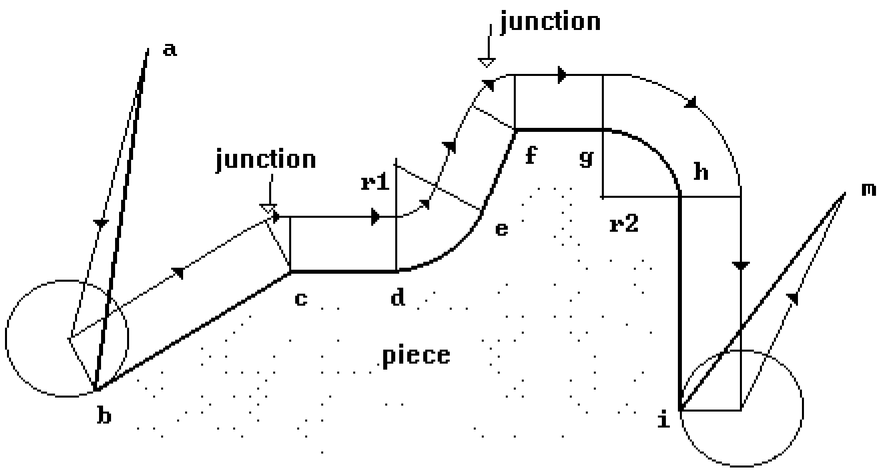
2) **b** is the end point of linear trait, with unchanged tool radius correction;

3) **c** is the end point of circular trait with void tool radius correction.

Corrected profile check starts from corrected point **a** and continues on a linear trait parallel to segment **(a) - (b),** till intersection point **(i)** with corrected arc. The arc corrected between (**b**) and **(c)** is concentric to programmed arc, with radius = programmed radius + tool radius.

The end point of corrected arc lays at a distance = tool radius from point **(b)**, on the perpendicular to the straight line tangent to the arc in **(c)**.
Tool raise is performed on arc end point.

EXAMPLE OF TOOL RADIUS CORRECTION ON A OPEN PROFILE

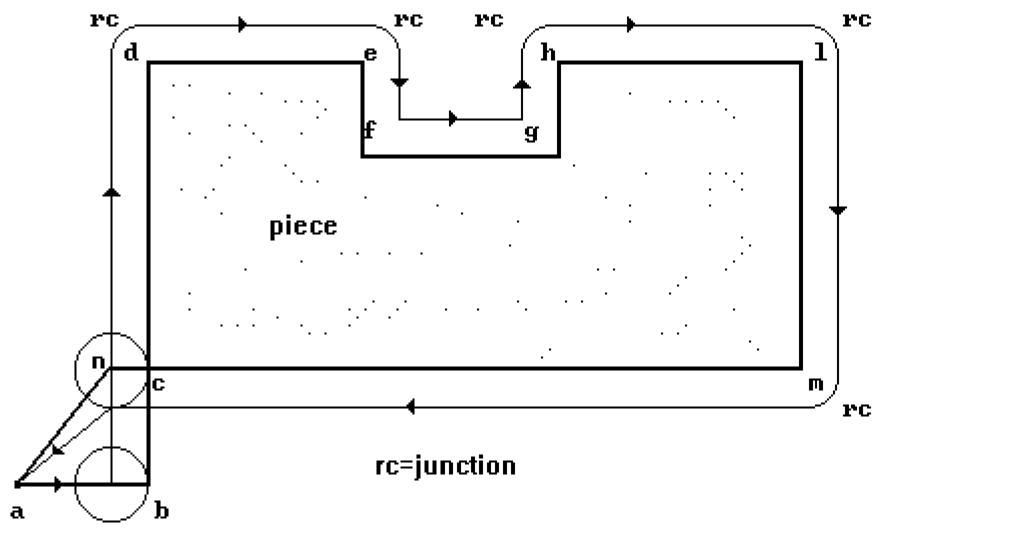


Mill set-up is at point (**a**), without mill radius correction.
(**b**) is the first programmed milling point : linear trait with left correction (S-correction).
Milling cycle ends in (**m**), with correction cancelling as programmed.
Entry and exit are performed with gradual correction.

Junction insertion is displayed at corrections points (**c**) and (**f**).

The programmed radius r1 and r2 are marked on the defined arcs in (**e**) and (**h**).
In the first case (arc on radius r1) the corrected arc is reckoned on radius = r1 - mill radius..
In the second case (arc on radius r2) corrected arc is reckoned on radius = r2 + mill radius.

EXAMPLE OF CORRECTION IN A CLOSED PROFILE JUNCTION



Mill set-up (point **a**) is programmed without mill radius connection.
The first programmed milling point (**b**) is on linear trait, with left hand correction.
The milling cycle ends on the same set-up point with correction cancellation as programmed.
Entry and exit are performed with gradual correction.

A junction will be inserted by the correction in points (**d**), (**e**), (**h**), (**l**), (**m**).
Programming allows to obtain the full workpiece as useful piece.

Milling cycle is defined by points : **abcdefghlmna**.

Any geometrical profile is made by the sequence of elementary geometrical elements, defined by the succession of two geometrical traits:

a) linear trait - linear trait
b) linear trait - circumference arc (or helix)
c) circumference arc (or helix) - linear trait
d) arc - arc.

Tool arc correction is performed on plane xy.

Profile traits displaying coordinate variations on axis Z, during tool radius correction, are considered in the projections corresponding the plane xy.

The following figure represents some correction example of the four elementary cases above listed.

Original profiles are the ones marked with arrows. Each profile is defined with initial point in (1), junction point between the two traits in (2) and the final pint in (3).
The corrected traits are marked on the correspondent original geometrical traits, indicating the geometrical construction used to define corrected traits.

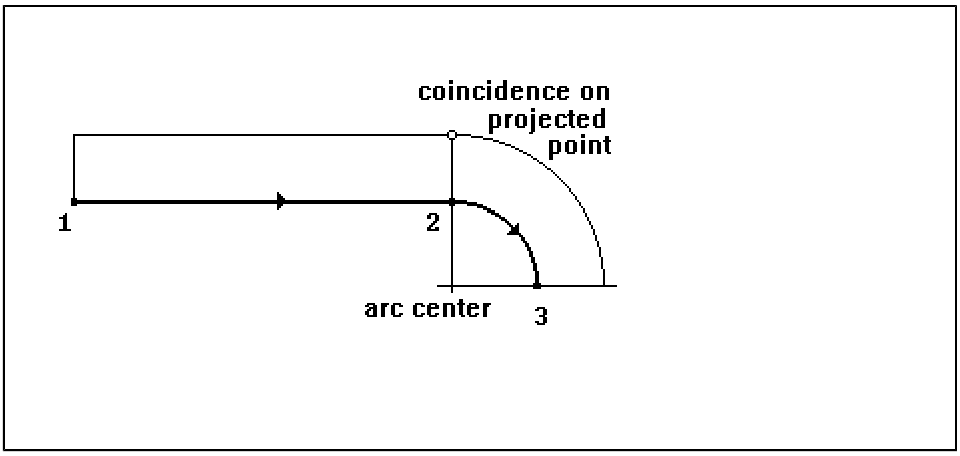
The real purpose is to define the possible cases in the junction point between the two geometrical traits. That's why the examples have been divided in two groups:

a) On the left side the corrected geometrical traits result without intersection point: these are the cases in which correction mode inserts an **angular junction** between the two corrected traits. With left hand correction, the junction will have a clockwise rotation; with right hand correction, the rotation will be c.clockwise;

b) On the right side geometrical traits intersect one another.
The corrected profile is defined by two geometrical traits, and middle point of corrected profile will be assigned on intersection point of two corrected traits.

There is a third geometrical case besides the already mentioned cases of **junction** and **intersection.**
It is the situation of **coincidence** of the projections related to the junction point of two geometrical traits. The case of coincidence corresponds to the geometrical condition of tangential continuity between the trait arriving on the junction point and the trait departing from the same point.

An example of linear trait + arc is displayed here below.



Interpolation speed F' of junction insertion is computed by parametric value "Reference speed", or it is assigned as programmed (Fr-field on mill set-up).

In the first case :

Where :

Fp = parametric reference speed
Rf = mill radius value.

F' speed is defined only if inferior than interpolation speed programmed on preceding trait, on the contrary preceding trait speed is confirmed.

Another geometric condition may occur, when defining programmed F' speed: if the distance between start and end junction points (arc span) is less than reference value defined in parameters mode, tangential speed defined on preceding geometric trait is confirmed on junction.

GEOMETRIC ERROR CONDITIONS

Particular geometric conditions are considered wrong and in these cases an error message is displayed. Error message is displayed as follows :

Error 16 : MILL RADIUS c1c2 (L=ll)

where :

c1 and c2 are two characters, showing trait type involved in control.
Possible cases:

c1c2= a	arc type trait,
c1c2= r	straight line type trait,
c1c2= ra	two straight line-arc type traits,
c1c2= ar	two arc-straight line type traits,
c1c2= rr	two right line-right line traits,
c1c2= aa	two arc-arc type traits.

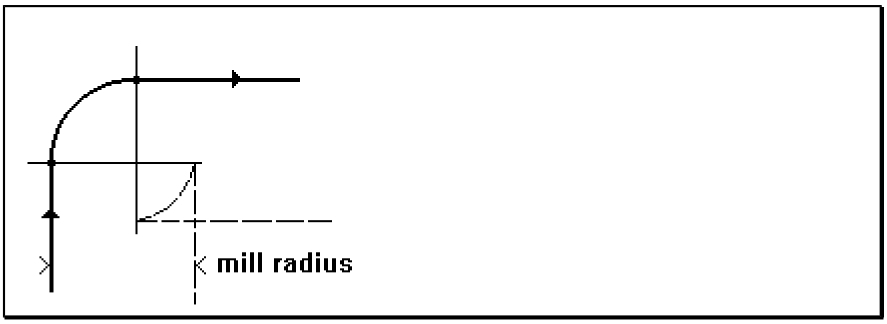
ll shows program block corresponding to error displaying.

Unvalid geometric conditions are defined in the following points:

1) correction of circular clockwise milling, with right hand correction and mill radius value major than junction radius; or correction of circular c.clockwise milling with left correction and mill radius value major than junction radius. The first case is shown in the figure.

Error is displayed as follows :

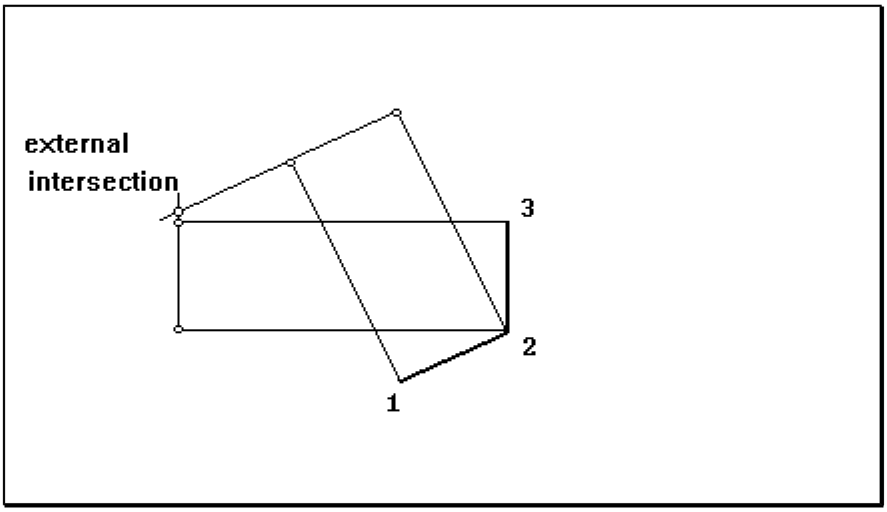
Error 16 : MILL RADIUS a (L=ll)



2) correction of two consecutive linear geometric traits, with linear trait length inferior than mill dimension. The case is shown in the figure.

Error is displayed as follows :

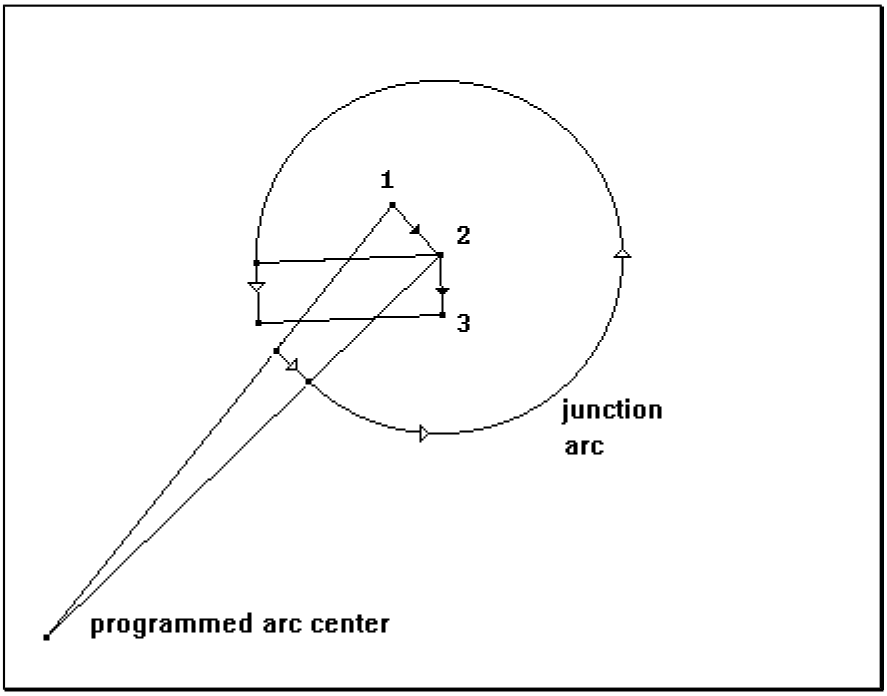
Error 16 : MILL RADIUS rr (L=ll)



3) correction between linear geometric and consecutive circular traits (or vice versa), junction can be corrected if computed junction arc does not reproduce original profile. The case is shown in the figure.

Error is displayed as follows :

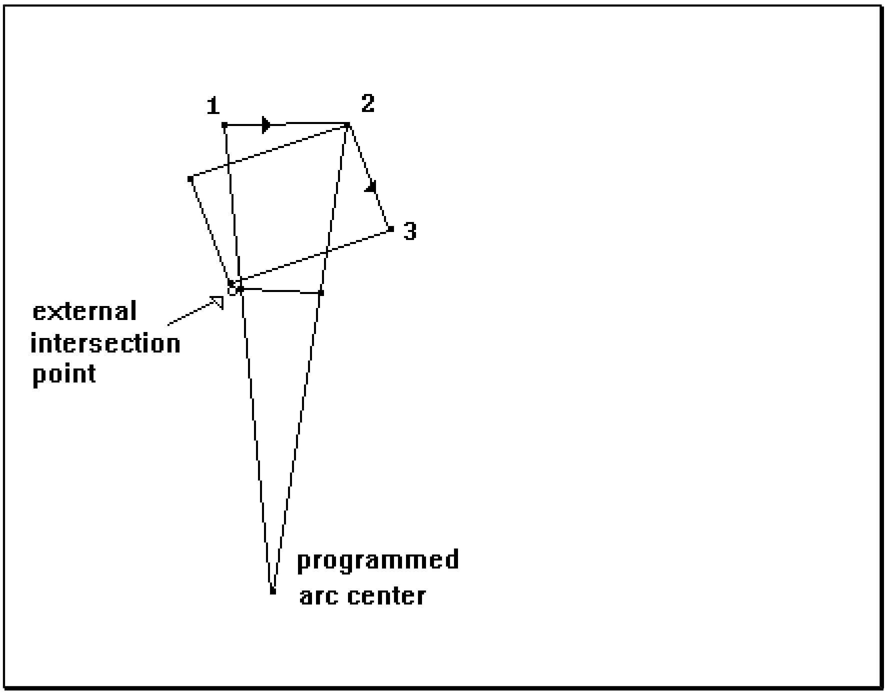
Error 16 : MILL RADIUS ra (L=ll) if right line-arc;
Error 16 : MILL RADIUS ar (L=ll) if arc-right line;



4) correction between geometric linear and consecutive circular traits (or vice versa), intersection can be corrected, if intersection point is external to corrected geometric traits. The case is shown in the figure.

Error is displayed as follows :

Error 16 : MILL RADIUS ra (L=ll) if right line-arc;
Error 16 : MILL RADIUS ar (L=ll) if arc-right line;



CONTOURING

Contouring defines a characteristic of milling **execution**, related to programmed profile geometry and profile programming modes.

A sequence of linear and circular interpolation instructions executed without:

- acceleration and deceleration ramp
- stop on connection points

defines a contouring working.

Contouring can be controlled at each connection point between two interpolation strokes.
As already seen it is possible to choose among three different options :

a) contouring automatic control,
b) forcing when contouring is activated,
c) forcing when contouring is de-activated.

CONTOURING AUTOMATIC CONTROL

Control is enabled to automatic management of contouring, when executing a mill profile anyway assigned. The following principles are valid in this case:

a) interpolation speed can be changed at each program block;

b) movement continuity (contouring) is kept between a milling and the following one, with reference to interpolation speed defined for each stroke, if **geometric continuity** conditions among the two concerned millings are valid.

Geometric continuity conditions are valid if connection point tangents are distinct inside an angle of 10 degree (anyway positioned in the space).

When the control does not execute a contouring, speed values defined on each single program line are complied, but it executes geometric stroke stopping on connection points.

Possible geometric situations, defining a connection point on an interpolation profile, are listed here below:

CASE 1 : right line - arc (on plane)
CASE 2 : arc (on plane or segment of helix) - right line
CASE 3 : arc (on plane or segment of helix) - arc (on plane)
CASE 4 : right line - right line

In these cases "geometric continuity" is tested : such condition is considered valid if two right lines tangent on connection point are distinct inside angle of 10 degree (anyway positioned in the space).

Two tangent straight lines are reckoned on connection point :

a) first one, on arrival trajectory,
b) the second one, on starting trajectory.

In case of straight line, the tangent is coincident with straight line itself , both in case it defines arrival trajectory on connection point, and in case it defines starting trajectory from connection point. Straight line orientation coincides with tangent to arc in connection point

In case of arc, tangent right line coincides with tangent to arc, in connection point. Straight line orientation follows execution direction of circular milling.

CASE 5 : right line - arc (on segment of helix)
CASE 6 : arc (on plane or segment of helix) -arc (segment of helix)

They are the cases with exit connection point on a helix segment.

Geometric continuity condition is tested in a different way with reference to what above said. On two tangent straight lines in fact components along Z axis are not computed but are considered as projected on plane XY. "Geometric continuity" conditions is therefore tested if:

1. two tangent right lines, so defined in connection point, are distinct inside an angle of 10 degrees

2. programmed deplacement on Z axis, of the two defined milling, should not cause deplacement on opposite directions.

z1 is the deplacement defined by input milling in the connection point and z2 is the deplacement defined by exit milling in connection point (z1 and z2 are coordinates increments, with sign +/-) :

a) cases : (z1 * z2) => 0 [read : product of z1 and z2, is positive or equal to 0] the condition is verified;

b) case of product minor than zero, the condition is not verified.

Geometric control performed in cases 5 and 6 is less restrictive, with reference to the control performed on preceding cases : such control does not consider the deplacement value required on Z axis.

CONTOURING PROGRAMMED CONTROL

During program editing, contouring can be enabled or disabled on each connection point of milling profile.

a) <u>forcing on activated contouring</u>

enables contouring execution, starting from programmed point. Selection is enabled till different specification on current profile.

b) <u>forcing on disabled contouring</u>

disables contouring execution starting from the programmed point. Selection is enabled till different specification on current profile.

INSERTION MODES

Insertion selections for program blocks are available on **Secondary menu**, at the items:

INSERT UP
INSERT DOWN

INSERT UP allows insertion above current program block (read : placed in mode operativity).

INSERT DOWN allows insertion below current program block .

Block insertion is subject to the following limitations :

only INSERT UP can be selected on end program block;

insertion should respect max performable number of blocks, both on ASCII text and on compiled text;

milling blocks (linear, circular interpolations, exc.) can be inserted only if a milling has already been defined at least in set-up mode.

When INSERT UP/DOWN is selected, working menu enabled in configuration is displayed, and it can be scrolled on a window of 5 lines.
Menu selection can be performed by keys:

to scroll available items;

confirms selection of item in color contrast;

cancels last performed selection.

Operating Menu is displayed only for selectable items, according to :

configuration on editor module,
current selected face,
constraints defined by particular selections.

Here below full Menu of all available modes is shown.

The first window framed by a double line, displays menu main structure and the items on the following pages can be selected scrolling main page :

```
FACE       5
RAPID (xyz H)
RAPID (xyzwv)
HOLES
SAW X
```

```
SAW Y
MILLINGS
SPECIALS
INSERTIONS
SUBROUTINE
```

```
SPECIAL HOLES
```

The first item of menu allows the selection on current face (number from 1 to 5 near word **FACE**) : it does not correspond to working selection, but to a setting preliminary to the real working.
FACE selection item does not appear when only in face 5 is active according to Editor configuration.

To change face selection :

1. select item FACE
2. define number corresponding to selected face
3. confirm by ENTER.

On the right part of the screen (in operating mode) a panel draft is displayed on programmable faces : as usual system origin and selected face are shown.

Operating mode menu is modified according to selected face, in order to display only significant items.
For instance : selecting a face different than 5, on menu selections the words SAW X, SAW Y do not appear.

After selection :

1. if no subprogram is available in the selected operation , working parameters should be defined;
2. otherwise a second menu is displayed, with the same selection modes.

In the first case these selections are available :

RAPID (xyz H)
RAPID (xyzwv H)
SAW X
SAW Y
SUBROUTINE

In the second case :

a) DRILLS : displayed sub menu

DRILL (x,y,z)
DRILL (x,y; u,a)
FITTING X
FITTING Y
REPEAT X

REPEAT Y
REPEAT XY
REPEAT XY (u a)
DRILL ON CIRCLE

b) MILLINGS

MIL SET (xyz)
MILL SET (xy;u,a)
L1 (x,y,z)
L2 (x,y; u,a)
L3 (u,a)

L4 (tg; u)
C1 (x1,x2; c; rot)
C2 (x,y; u;rot)
C3 (c; u,a; rot)
C4 (tg; x,y; rot)

HELIX C1
HELIX C2
HELIX C3
OVAL
GROOVING

CONNECTION
C5 (xi,yi; x,y)

c) SPECIALS

DELAY
MESSAGE
MEASURE
OFFSET
SAW A'

d) INSERTIONS.

PLATE
BUSHING
HINGE
SHELF BEARING
GENERIC 1

GENERIC 2
PIN

e) SPECIAL DRILLS

DRILL WITH DISCHARGE
TAPPING

Above displayed menu can be activated only is configured in Editor for any programmable working. Item corresponding to disabled workings are automatically excluded from menu pointed out.

Remember the possibility of using a graphic Help (window : ALT,G), during selection of working menu.

After selection, operating mode is enabled as examined in the sections dedicated to the different workings.

Consider for instance a DRILL (x,y,z) :

<- LxHxS : 1000;450;20 0003:0020

0 FILES	**4** DELETE
1 INSERT U	**5** LINE
2 INSERT DOWN	**6** SIDE
3 MODIFY	**7** R. MILL

N :**3** **DRILL (x,y,z)**
O :**0** Es:_ M1:_ a/r:**a**
S :200
T :**1 1,2**
F :__ M2: __
X :**100**
Ri:**2** Y :**200**
Ro:**0** Zp:**5** OK :**S**

In operating mode field a cursor is managed, moving on the programmable fields and enabled on the first geometric field : in the example on Ri (starting slowing)

Fields in a frame show geometric parameters assignments : these parameters are generally introduced different for each working.

Available commands in operating mode field are :

or
shifts the cursor to the following field (for instance : from field O to field Es:);

or
shifts the cursor to the preceding field;

shifts the cursor to confirmation field OK;

if cursor is not on OK field : shifts the cursor to the confirmation field OK;
if cursor is on OK field : confirms selected data;

shifts the cursor to the first programmable field (in the example : field O)

cancels working selection and returns to working menu.

The same criteria of parameters propagation from preceding block are applied to operations insertion in field mode compiling.

Help menu available for working input are :

help for available commands;
graphic help for selected workings;
graphic help for tooling;
with cursor in T field, for tooling assignment or tooling parameters;
graphic help for subprogram input;

To confirm defined parameters in operating mode field **OK** : select S and confirm by <--' (Enter).

After confirmation :

a) control checks if selected geometric and technologic data are correct

b) with positive check :
- graphic is performed on input working;
- working graphic modified by input block is corrected, canceling wrong working and performing the new graphic;
- insertion procedure of following block is started.

c) with negative check :
- an error message is displayed with error number;
- message error is deleted by ENTER and parameters input procedure is started again, to modify or correct wrong data.

For what concerns error conditions refer to exam of single workings.

When introducing a milling (linear, circular...), a draft of milling geometric data is displayed at the place of draft of piece faces.

Panel draft showing working face, xy system origin and selected origin, is displayed if the cursor is placed on O-field (origin assignment)

A new operation introduced in a program block (see also : modify or block erasure) can cause an error on the assignments of the following block .

An example is the case of an arc defined with initial radius different from final radius. Such condition is marked by:

graphic of not executable working in color contrast
In particular, an arc with radius not equal is displayed as a linear stroke;

a change on a block not significant implies an error message on related checks;

program storing with not executable blocks:
a) ends with an error message (icon of exclamation mark);
b) ends without compiling demand;
c) marks the program as not executable on program directory.

MODIFY MODE

Modify selection mode is available on **Secondary Menu**, at the item : MODIFY (selection number : 3).

Modify mode allows to change assignment parameters on a program block, but not to change mode typology Mode modify (for example : milling L3 instead of milling L2) can be executed by:

block erasure
block assignment on different mode.

Modify mode is executed as an input : a moving cursor allows to shift to all the parameters, with possibility of modifying.

Besides the following keys are activated :

to shift to preceding and following block, without changing current block

Field OK selection and following confirmation (S <-'), end modify mode with confirmation.

The same error conditions as block input are managed for block modification, with the possibility of error message display and of modify demand.

Working modification implies :

a) program graphic check as already examined for block input;
b) the possible introduction of indefinite geometric conditions

DELETE MODE

Program block deletion is available in the following two modes:

a) in **secondary menu**, select DELETE (direct selection number : 4), to delete one or more program block;

b)directly in **mode field**, select **keys**, to delete current program block

Delete selection is not executed with :

a) end program block;
b) milling set-up followed by milling block and preceded by a working different than milling and milling set-up or defined on a different working face.

Deletion with (CTRL,Y) key performs a direct deletion of current program block without any confirmation.

When selecting deletion command in secondary menu, a graphic window is opened, such as :

DELETE	
from	10
thru	10

current block number (in the example 10) is displayed on the first line of the window : shows the first number to delete.
On the second line operator must impose end erasure block.
Suppose to impose end block = 15 : when confirming erasure, the blocks from 10 to 15, ends included are deleted.
End block should be defined major than start block : it is not correct to impose an erasure from block 10 to 5.

Press ENTER to confirm command.
Press ESC to cancel selection.

Working deletion implies :

a) a graphic re-elaboration of the program as already examined for block input;
b) the possibility of crating geometric undefined conditions

MERGE MODE

The Merge command is available on **Secondary Menu**, at the item : MERGE (direct selection letter: B)
On command run on the screen is presented the Program Directory: to select a program confirm with ENTER key or abort with ESC.

When the command is confirmed, the following window is opened:

MERGE : nameprg	
from	**1**
thru	**999**
Up/Down	**U**
Side	**D**

to define what blocs of the selected program must be inserted (merge) in a current program in editing.

In the head line is indicated the name (nameprg) of the selected source program.

In the other four following items user can specify:

the initial line (from ..) of the bloc of instructions to be merged (default = 1)
the final line (thru ..) of the bloc (default = 999)
the modality of the insertion of the bloc, before (Up) or after (Down) the current line of the destination program. If the current is the last line (statement with M02) the insertion will be made compulsory in Up mode.
the side (Left or Right) of the program-panel where the insertion is required (valid only for double side machine). Then it is possible to insert in the right side of program PROVA, for example, some lines programmed on the left side of another program PRG1.

For example, with these selections:

MERGE : 001	
from	5
thru	20
Ud/Down	U
Side	R

We suppose that the program in editing, named PROVA, has a total of 7 lines, and the line current is the n=3.

With the Merge command of the example:

an automatic test for compatibility is made: the right side of 001 may be included without overcome the maximum limit (700 lines in ISO code) otherwise a message " Text completed " is displayed.

the block of the lines, from 5 thru 20 comprised, of the program 001 are read and inserted in the text of program PROVA **before** of the line N=3.

follows a total re-elaboration of the program PROVA, with a new graphic presentation.

If errors occurs, the relative message is presented, to permit:

- program correction for single process
- delete of some lines block (Ctrl+Y)
- abort of the command (ESC), deleting all blocs of the program.

For other information, see a paragraph " Particular situations in Open/New ".

MILL RADIUS MODE

Mill radius selection corresponds to the request of program graphic display with applied the corrections programmed on milling tool radius.
In case of normal program display, graphic display does not perform mill radius corrections : milling profiles are displayed as programmed.

Mill radius selection implies the execution of:

Error. First the program is checked, so to determine if it is geometrically defined and if applications on mill radius are effectively requested. If the text is not geometrically defined, or if it does not require the application of a correction, a message will be displayed and Mill radius mode will be quitted.

Error. If modifications are correct : the program is re-elaborated so to determine its structure on the basis of the corrected milling profiles. In this stage some error can be found concerning correction procedures : an error message will be displayed and mill radius mode will be quitted. Please refer to sections concerning Mill radius for details on error messages.

Error. In case of correct execution of b) point : the resulting program is displayed graphically on the profiles corresponding to machine execution.

Graphic display upon Mill radius selection allows the simultaneous display of programmed profile and of corrected profile, or eventually only of corrected profile: select proper display mode in Editor configuration.

Once graphic updating is ended, the word Rf displayed next to workpiece dimensions shows that Mill radius correction is enabled.

To return to normal Editor mode:
a) select again MILL RADIUS (secondary menu);
b) select block input, modify or delete command;
c) select program recording command.

Mill radius correction is applied to program only during compiling mode.

COPIE MODE

Copie selection mode is available:

a) on **Secondary Menu** at the item COPIE (direct command letter : C), to copy one or more blocs of program;

b) directly on **Operation field** , using key, to copy the current bloc of the program.

This command is denied on the last bloc of the program.

The copie by [CTRL+INS] copies the current bloc, without confirm request.

When the command is called by Secondary Menu, this window is opened:

COPIE	
from	**5**
thru	**20**

where:

"from" defines the initial line number of the bloc to be copied (default = current line)

"thru" defines the final line number (default = maximum number bloc)

The command is activated by ENTER key, aborted by ESC.

When the command is confirmed, the selected lines (i.e. from 5 thru 20) are saved in a temporary buffer, to be recalled by a RETRIEVE command (see later).

Copie command does not modify current program in editing.

Remember that similar save of a block of lines may be made by the command DELETE (or [CTRL+Y]), where the deleted lines are temporarily saved, to be recovered by Retrieve command.

RETRIEVE MODE

With the RETRIEVE command it is possible to recover one or more saved lines (by Copie or Delete commands) inserting them into a current program in editing.

The command may be recalled:

a) on **Secondary Menu**, at the item RETRIEVE (direct selection letter : D), to insert one or more of the saved lines;

b) directly by **Operation field**, using keys, to insert ALL saved lines <u>before</u> the current line or using keys, to insert them <u>after</u> the current line.

The command is denied if no lines have been saved in a temporary buffer.

When the Retrieve command is called by Secondary Menu, the following window is opened:

RETRIEVE	
from	**3**
thru	**10**
Up/Down	**U**

where on the first two lines may be programmed the initial and the final line number of the block of instructions saved. On the third line is defined the inserting modality (Up=before / Down=after, the current line) into the program in editing.

The command is activated by ENTER key.

For example, we suppose to be displaced 3 programmed lines, numbered from 5 to 7, on the head of the program:

a) first, we must we place the current line at the bloc n=5;
b) then the DELETE command is called, with " from 5... thru 7 " selection
c) after, the current line is displaced on the n=1 bloc
d) with [CTRL+U] command, all the saved lines are inserted at the head of the program.

If we need to recover again the last two saved lines, to be inserted after the line n=8:

a) the current line is displaced on the bloc n=8;
b) the RETRIEVE command is recalled, assigning:

RETRIEVE	
from	2
thru	3
Up/Down	D

The retrieve command does not delete the saved lines, allowing multiple recovers until necessary.
The saved lines are also stored during Exit and Reentry in Editor mode, to maintain this utilities always available.

ZOOM and ZOOM OFF MODE

The ZOOM and ZOOM OFF commands are available on Operation Field, at the corresponding item, with direct selection letters E and F.

The ZOOM command allows to display a particular area of the piece, with enlargement.
When the command is called, on the screen is placed a special cursor, in color reversed: by Arrows keys the cursor may be displaced over the first point, the upper left corner, of the rectangular zone to be enlarged.

When confirmed by ENTER, the second point (lower right corner) must be selected by arrows and confirmed by ENTER. If required, the first point may be deleted by ESC and reprogrammed.

The selected area is then enlarged, using for displaying all graphic area.

Recursive Zoom command are admitted, to enlarge particular until needed.

The ZOOM OFF command may be used to return at the original scale of representation.
Graphic enlargements are deleted in case of change of side or program.

PARTICULAR SITUATIONS IN OPEN/NEW

Upon command OPEN/NEW (select 0 in Main menu) the execution of a program already in directory can cause particular error situations, related to:

Tool design and/or heads and/or technological parameters modified when recording the program and in contrast with the program itself.

program loaded from an external menu (CAD-TPA or other custom program)

program stored with indefinite geometrical situations

program stored with a different configuration than Editor module

In case one of the above listed situations would take place, the system would:

propose the program block found wrong in mode area. Consider the case of drilling with tool defined by a wrong typology;

an error message is displayed in mode area; in the example : wrong tool

by pressing ENTER operator can modify program block: in facts the mobile cursor will appear on first programmable field of mode area.

operator has now three possibilities :

modify program block with final data confirmation (see also Confirmation)

cancel the block from program text by pressing (**Ctrl,Y**) keys;
delete all program blocks by pressing **ESC key**. In this way the program will be initialized only on the End program block.

Selection of or points is compulsory if the following error is displayed:

"Wrong Code" : in this case the block is not accepted directly by configuration file and the situation can be resolved only by changing Editor configuration.

ARC 1 _ ARC 2

Selection mode :

1. Select on Mode Menu, second page (upon command INSERT UP or INSERT DW):

SAW Y
MILLING
SPECIALS
INSERTIONS
SUBROUTINE

2. Select menu **MILLINGS:**

Helix C3
OVAL
GROOVE
JUNCTION
ARC 1 - ARC 2

3. Select working **ARC 1 - ARC 2**

Display of Mode Area

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3**	**Arc1-Arc2**	
		o :**0**	Es:____	a/r:**a**
1 INSERT U	**5** LINE	Ct:**A**	Rf:____	F:**3**
		M2:___	ACw**: 2**_ c1-c2:	**C1-C2**
2 INSERT DOWN	**6** DIM	C1x: **100**		
		C1y: **0**		
3 MODIFY	**7** R. MILLING	X :**300**		
		Y :**200**		

The Mode Area is commutable also over the second page:

		<- LxHxS : 1000;450;20		0003:0020
0 FILES	**4** DELETE	N :**3**	**Arc1-Arc2**	
1 INSERT U	**5** LINE	U1:____	c1-c2:	**C1-C2**
		U2:____	Cw2:	**2**
2 INSERT DOWN	**6** DIM	C2x:: **0**		

3 MODIFY	**7** R. MILLING	C2y:**30**

Description of Mode Area

1.- **O field: Origin of programming**

to select the origin for quotes programming:
X and Y
C1x and C1y
C2x and C2y.

2.- **a/r field: absolute/relative**

select absolute or relative mode in programming for quotes X and Y.

3.- **Es field: Exclusion number**

Present the programmed value (in set-up milling tool), without modificability.

4.- **F field: Working speed**

The programming is in unit [mt/min] or [inch/min].
F represents the tangential speed over the programmed arcs.

5.- **Ct field: contourning**
6.- **Rf field: Radius tool correction**

(see previous milling works)

7.- **Cw field : sense of rotation**

Program for clockwise rotation (2) or counterclockwise (3), for the arc 1.

8.- **c1-c2 field: selection on geometry**

Assigns the geometrical rules for the process, with a aided programming:

Arc 1	Arc 2	Description
C1	C2	Arc 1 is defined by Center coordinates (C1x,C1y) Arc 2 is defined by Center coordinates (C2x,C2y)
C1	U2	Arc 1 is defined by Center coordinates (C1x,C1y) Arc2 is defined by radius U2
C1	tg H	Arc 1 is defined by Center coordinates (C1x,C1y) Arc 2 is defined with horizontal tangent on final point
C1	tg V	Arc 1 is defined by Center coordinates (C1x,C1y) arc 2 is defined with vertical tangent on final point
U1	C2	Arc 1 is defined by radius U1 Arc 2 is defined by Center coordinates (C2x,C2y)
tg H	C2	Arc 1 is defined with horizontal tangent on initial point Arc 2 is defined by Center coordinates (C2x,C2y)
tg V	C2	Arc 1 is defined with vertical tangent on initial point Arc 2 is defined by Center coordinates (C2x,C2y)

The final point of the arc is alwais assigned on the plane XY. The selection of the geometrical mode defines what types of parameters are necessaries.

C1x-C1y field

coordinates of the center of arc 1, programmed incrementals from the initial point of the arc. C1x and C1y are set in units [mm] or [inch]; the values are signed and the parametric programming is allowed.

The C1x,C1y field is significant only if arc 1 is programmed by Center coordinates (C1).

X/Y field: quotes of final point on arc 2

Assigne the values of quotes (absolutes or relatives) of the corresponding axes, programmed in units [mm] or [inch], of the final point of teh arc 2.

U1 field
U2 field

Assigne the radius U1 of the arc1 and U2 of the arc2. The values must be programmed in [mm] or [inch]. These fields are significants only the geometry is programmed on U1 and U2 (see c1 and c2 field).

Cw2 field: sense of rotation

Sense of rotation for arc2:
2 = clockwise
3 = counterclockwise
(empty field) = sense reverse to arc 1.

C2x/C2y field

coordinates of the Center of arc 2, **incrementals** from the Final point of the arc.
The quotes in C2x,C2y are to be programmed with signed values in units [mm] or [inch], also in parametric mode.

C2x/C2y field is significant only if C2 is selected on the c1,c2 field geometry (see before).

Notes on processing

The profiles selectables by " Arc 1 - Arc 2 " may be two circular arcs on the XY plane, like showed in the following figures, where:

[1] is the last programmed point, in the previous bloc,

[2] is the final point of arc 1 (= to initial point of arc 2),

[3] is the final point of the arc 2, programmed in the current bloc,

C1 is the center of the arc 1 : (c1x,c1y) are the coordinates of C1, programmed in incremental mode relative to point [1],

C2 is the center of the arc 2 : (c2x,c2y) are the coordinates of C2, programmed in incremental mode relative to point [3],

Cw1 is the sense of rotation programmed on the arc 1,

Cw2 is the sense of rotation programmed on the arc 2,

U1 is the radius of the arc 1,

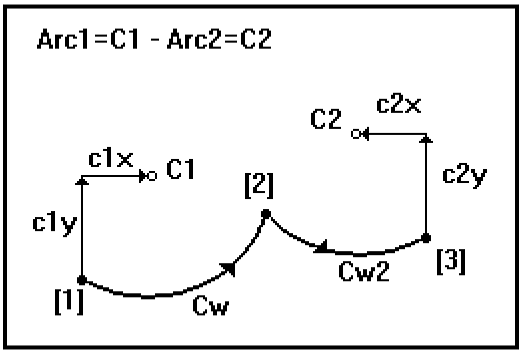
U2 is the radius of the arc 2,

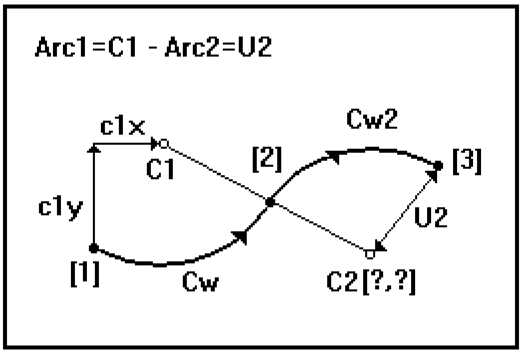
tg H represents the straight line with horizontal tangent,

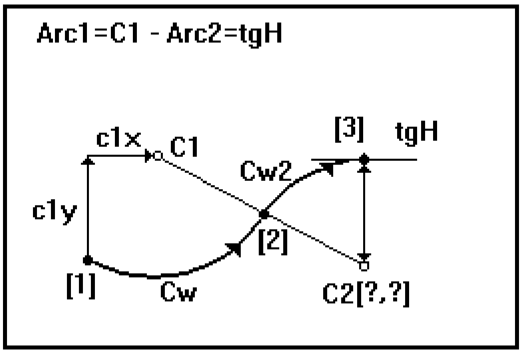
tg V represents the straight line with vertical tangent.

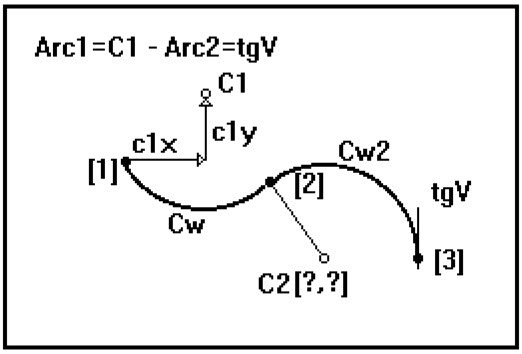
All the points marked by (?,?) are calculated by the numerical control.

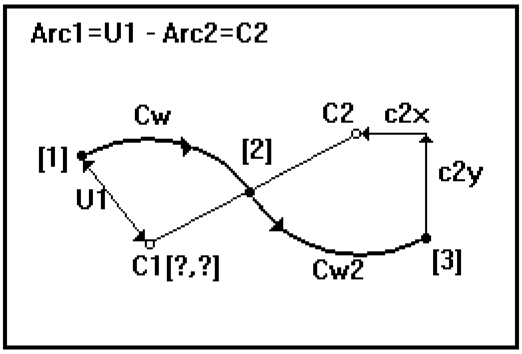
The executing profile is marked by arrows.

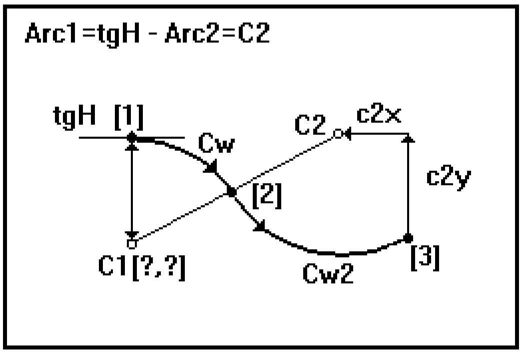

Arc1=C1 - Arc2=C2
C2
c2x
c1x
C1
[2]
c2y
c1y
[3]
Cw2
[1]
Cw

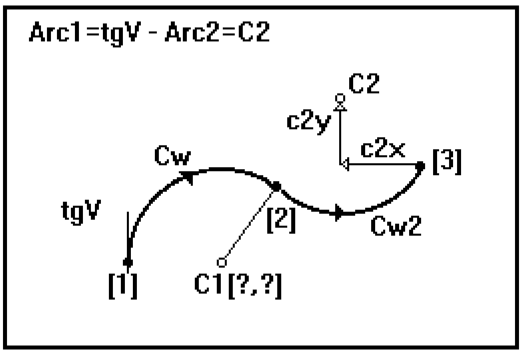

Arc1=C1 - Arc2=U2
c1x
Cw2
[2]
[3]
C1
c1y
U2
[1]
Cw
C2[?,?]


Arc1=C1 - Arc2=tgH
[3]
tgH
c1x
C1
Cw2
c1y
[2]
[1]
Cw
C2[?,?]


Arc1=C1 - Arc2=tgV
C1
c1y
c1x
Cw2
[1]
[2]
tgV
Cw
[3]
C2[?,?]


Arc1=U1 - Arc2=C2
Cw
C2
c2x
[1]
[2]
c2y
U1
C1[?,?]
Cw2
[3]


Arc1=tgH - Arc2=C2
tgH
[1]
C2
c2x
Cw
c2y
[2]
C1[?,?]
[3]
Cw2


Arc1=tgV - Arc2=C2
C2
c2y
c2x
Cw
[3]
tgV
[2]
Cw2
[1]
C1[?,?]

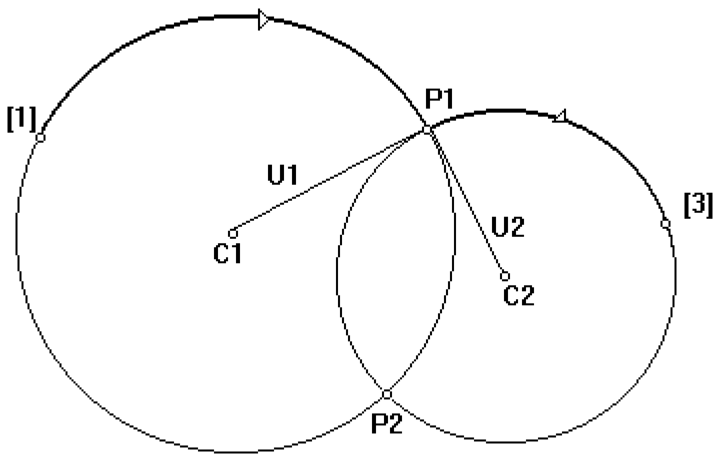
Arc 1 = C1 Arc 2 = C2

Both the arcs are defined by the coordinates of the center, with:

a) the coordinates of the center of the arc 1 are programmed incrementals relatively to the point [1];
b) the coordinates of the center of the arc 2 are programmed incrementals relatively to the point [3];

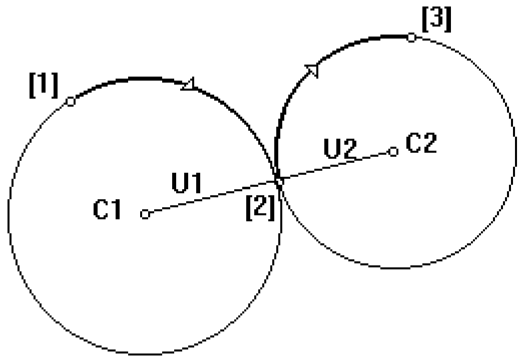
The two radius U1 and U2 are automaticaly computed.
The point [2] is the intersection point enter the circumferences: there are three situations:

Error. there are presents two differents point of intersection, named P1 and P2



In the figure: the senses of rotations are showed by the arrows on the arcs.
In this case for the point [2] is assumed the first intersection point found going along the arc 1 starting from [1] with the programmed CW sense, then P1.

Error. the distance enter the centers is equal to the sum of the radius, then only one intersection point to assign to [2].



Error. if there are no intersection points enter the circumferences, a diagnostic message is displayed.

Arc 1=C1 - Arc 2=U2

The arc 1 is defined by the center coordinates and the arc 2 by the radius U2.
The second circumference is defined so as in the point of intersection [2] the two arcs have the same tangent.

Arc 1=C1 - Arc 2=tg H

The arc 1 is defined by the center coordinates and the arc 2 by the horizontal tangent on the final point.
The second arc is defined as so in the point [2] has the same tangent as the first arc.

Arc 1=C1 - Arc 2=tg V

The arc 1 is defined by the center coordinates and the arc 2 by the vertical tangent on the final point.
The second is defined as so in the point [2] has the same tangent as the first arc.

Arc 1=U1 - Arc 2=C2

The arc 1 is assigned by radius U1 and the arc 2 by the center position.
The first arc is defined as so in the point [2] has the same tangent as the second.

Arc 1=tg H - Arc 2=C2

The arc 1 is defined by the tangent horizontal on the initial point and the arc 2 by the center position. The first arc is defined as so in the point [2] has the same tangent as the second.

Arc 1= Tg V - Arc 2=C2

The arc 1 is defined by the tangent vertical on the initial point and the arc 2 by the center position. The first arc is also defined as so in the point [2] has the same tangent as the second.

Change of contourning or Radius mill correction, from active value to inactive, is applied only on the second arc.

Errors displayed on processing

Error 4 : M2 field value not valid
{ see DRILL (x,y) }

Error 4 : F field value not valid
{ see L1 }

Error 8 : Rf field not defined
when a Radius mill correction is required with Radius =0.

Error 8 : # field not defined
programming missing in the field

Error 4 : X field value not valid
Error 4 : Y field value not valid
Error 4 : C1x field value not valid
Error 4 : C1y field value not valid
Error 4 : C2x field value not valid
Error 4 : C2y field value not valid
Error 4 : U1 field value not valid
Error 4 : U2 field value not valid

if the numeric value are out of the limits: 99999.999 in [mm] and 3999.9999 in [inch].

Error 12 : error interpolation C1=C2

The centers C1 and C2 are the sames.

Error 12 : error interpolation ?

The programmed interpolation has no solution.

Syntax of ASCII bloc

```
G114 G90 G54     X.. Y.. I.. J.. R60=.. R61=.. R56=.. R57=.. R50=.. F.. G40 G62
G115 G91 ..              K..    U..                                   G41 G63
           G57                                                        G42 G64
----------------------------------------------------
     compulsory fields
```

Compulsory fields

G114/G115 Op. code of the function, at the beginning of the bloc, for CW/CCW rotation

G90/G91 absolute or relative

G54..G57 origin of programming

X.. Y.. Final point (of arc 2) coordinates

I.. J.. Arc 1 center coordinates (compulsories for selection type Arc 1=C1)

K.. Arc 1 radius (compulsory if Arc 1=U1 programmed)

R60=. R61=. Arc 2 center coordinates (compulsories for selection type Arc 2=C2)

U.. Arc 2 radius (compulsory if Arc 2=U2 programmed)

Not compulsorys fields :

R50=.. rotation on arc 2:
R50=2 CW rotation
R50=3 CCW rotation
( ) if not present: reverse to sense of arc 1

R56=.. selected typology for the arc 1:
R56=0 programs the center
R56=1 programs the radius
R56=2 programs hor. tangent on the initial point
R56=3 programs vert. tangent on the initial point
( ) if not present: like R56=0.

R57=.. selected typology on the arc 2:
R57=0 programs the center
R57=1 programs the radius
R57=2 programs horiz. tangent on the final point
R57=3 programs vert. tangent on the final point
( ) if not present: like R57=0.

G40/G41/G42 Mill radius Correction functional codes

G62/G63/G64 Contourning control codes

F.. Working speed in interpolation mode

M.. Auxiliary function M2.

Exemple of ASCII bloc relative to values inserted in the Mode Area table:

G114 G90 X300 Y200 R60=0 R61=30 F3 G63

11. PROGRAMS COMPILATION

Select one of the following voices to call programs compilation mode:

Programs Compilation in main menu;
Program Editor mode, when storing a program after compilation confirm;
Program Editor mode, when compiling a program.

Only in case options in compilation menu can be selected.
In the other two cases on the contrary compilation in editor program is started in automatic mode, with automatic selection of compilation for any program.

OPERATING MODE MENU

Video is paged as displayed here below :

Program :	**MENU**
	SINGLE PROGRAM TOTAL ARCHIVE INITIAL PROGRAM
Messages :	PER GROUP PER TOOLING ERRORS PRINT

Menu selections are displayed on the right side of the screen.
On the left side of the screen two windows are displayed respectively for program name and messages.

Select ESC key to quit the program.

SINGLE PROGRAM

Allows the compilation of a program selected on program directory.
Program selection is realised directly on directory page or digitising thename on keyboard. Press ENTER to confirm selection; press ESC to reset selection.

TOTAL ARCHIVE

Manages the compilation of the whole programs archive.
Eventual errors are displayed at the end of all compilations.
This procedure is suggested if machine parameters should be modified.

INITIAL PROGRAM

Manages the compilation of all programs having established character - number or letter as initial letter. This procedure is suggested if programs archive should be structured using an initial letter to differentiate programs groups.
Eventual compilation errors are displayed in a single block when compilation is completed.

PER GROUP

All programs using a defined group are compiled again.
This mode is useful when a parameters change is realised for a defined Group : in this case user may compile only programs using group of which parameters have been modified.
Eventual compilation errors are displayed when compilation is completed.

PER TOOLING

All programs using a defined tooling are compiled again.
Eventual compilation errors are displayed when compilation is completed.

PRINT ERRORS

Allows to see all the errors found during last compilation session.

Window PROGRAM

It is used to display the name of programs involved in compilation. It assumes a particular meaning in multiple compilations.

Window MESSAGES

It is used to display programs errors immediately after diagnosis, allowing user to interrupt multiple compilations when checking errors with high frequency.

PROGRAM COMPILATION

Compiler translates program, stored in Editor mode, into a language interpretable by control.
In particular : each programmed working is translated into a specific machine function, which is joined to numerical parameters necessary to the correct execution of the same function.

Each programmable CNC90 working is joined, in an established way, to the execution of a defined instructions cycle of the control : all instructions cycle should be loaded as a function in GPL1000 language.

The available environment to edit these functions is defined by **Auxiliary System** , selectable in console Main menu.

A function edited in GPL1000 language is a mini-program, and can be recalled by a main program.
Functions are loaded in the control when he system is initialised andthey remain resident and available on the cards.
Each function is marked by a number, going from **0** to **255**.
Refer to PTP1000 System user's manual for a more detailed description of GPL1000 functions.

Notice here the points listed below :

the execution of a GPL1000 function is joined to each programmable working;
this association is defined by :
function number corresponding to each working,
number and typology of parameters that define the function
function text is not defined in an established way. In particular :
function parameters can be used only partly, as forced by machine features,
function text must be congruent with parametric structure which is defined a priori in an invariable way.

Data that compiler reads when compiling are :
program recorded in phase of CNC90-Editor
machine parameters.

Same particular feature of **measure** and **speed** parameters are examined here below.

<u>Measures compilation:</u>

During programming all **measures** are joined to working piece: when compiling an initial transposition of measures from piece zero to machine zero is realised, forcing offset and correctors to groups and tooling.

During measure transposition compute checks on limit measures of x, y and z axes are executed, according to machine parameters setting.

Finally, measures values are converted into pulses of machine computing (resolution pulses) : conversion parameter is read on axis resolution.

For instance, picture displays geometrical data used for computing working X coordinate programmed for a drilling working in face 5. Only measures of X axis are displayed to facilitate reading.

In the figure a head (a group) is displayed. It is drafted with two significant tools :

tool T1 : taken as reference for tools correctors setting (group zero x/y offset are defined on T1). In particular : T1 has x and y correctors empty;

general working tool, named Tn.

Zero point on XY plane is marked in the point : (0,0).

The following parameters are displayed :
OFFx_T = x offset of T stroke : automatic working is executed with T reference;

OFFx_0 = zero x offset of group (imposed on T1);
Cx_Tn = x corrector of Tn tool.

A drilling point is marked on working piece, at programmed measure = Qx.
Suppose Tn working too is defined with head type D1.
An empty corrector is imposed on head D1 : head reference is on the central point. Mark head x corrector as : Cx_D1.

Working on programmed drill at piece Qx measure, Z axis operates in Xl position computed as follows :

Xl = Qx + OFFx_0 + Cx_Tn + Cx_D1 + OFFx_T.

During compilation, Xl is computed without last term : stroke offset is in fact executed directly by control.

Speed Compilation:

Speed is programmed in m/min units.
During compilation a conversion is executed as for coordinates.
Space measure unit is machine resolution, and machine clock (real time) is used as time measure unit.
In the parameters table generated by compiler, speed is expressed in: pulses/real-time.

Mathematical formula used to obtain conversion is the following :

====

Treatment of subjects in the following paragraphs may result sometimes not complete, but these arguments are not of main interest for final user, while it is an essential documentation for design and control installation staff.

MACHINE FUNCTIONS

As already said a machine function is defined by a sequence of elementary instructions, that are able to run the selected execution of any programmable working.

Function instructions are read by card program in real time, during automatic execution.
The whole machine functions, characterising machine working, are loaded on numeric control memory during initialisation (that is when starting the system) and they are memorised till the system is switched off.

Machine functions structure is defined by tool machine builder: this available programming flexibility allows to manage mechanic in a very optimised way, both for what concern security and productive performances.

All the necessary functions to realise a specific working can be foreseen when writing machine functions. The following parameters on the contrary can't be defined in an established way:

speed
working measures
selected tools.

These parameters are variable, according to program text and parametric impositions: for this reason they are joined to functions only during working execution in relation to selected values.

<u>Example :</u>
a drilling working in face 5 always implies the execution of function **105**, but with different executions parameters can change :

XY plane working measures
deep axis measure (air measure, slowing, deep measure, final depth)
penetration and slowing speed
group and tools
spindle rotation speed.

Here below a detailed description of working and parameters related to different functions is expound.
Each function is defined by :

joined working
function number.

Parameters are described for each function, they are named with **Pnn**,
nn=number of two figures; for instance : P01.
Function parameters list complies order of parameters input.

Personalised parameters can be introduced: in these cases involved parameter is displayed only by corresponding number, without description of its use. These parameters can be used only with particular applications, or with different meaning on different machines.

For all functions, the first three parameters are common, and named "**dummy bytes**". A detailed description of these dummy bytes is given hereafter:

P01 Dummy byte 1	Bit 0=1 if next stroke has same tools
	Bit 1=1 if preceding stroke has same tools
	Bit 2=1 if next stroke is on same working
	Bit 3=1 if preceding stroke is on same working
	Bit 4=
	Bit 5=1 if contouring start
	Bit 6=1 if pneumatic group (without axis Z)
	Bit 7=1 if dummy group (without Y/Z)
P02 Dummy byte 2	Bit 0=1 if insertions screwed; 0 if under pressure
	Bit 1=1 if reference origin 1 or 3, 0 if 0 or 2
	Bit 2=1 if reference origin 2 or 3, 0 if 1 or 0
	Bit 3=1 if contouring end
	Bit 4=1 if Group follows
	Bit 5=1 if Group 1 high spindle speed
	Bit 6=1 if Group 2 high spindle speed
P03 Dummy byte 3	Bit 0=1 if exclusion 1 enabled; 0 on the contrary
	Bit 1=1 if exclusion 2 enabled; 0 on the contrary
	Bit 7=1 if exclusion 8 enabled; 0 on the contrary

FUNCTION	WORKING CYCLE	FACE
101	drilling	1
102		2
103		3
104		4
105		5
106	fitting	1
107		2
108		3
109		4
110		5
111	insertion	1
112		2
113		3
114		4
115		5
119	rotating saw	5
120	X or Y saw	5
121	beginning of routing	1
122		2
123		3
124		4
125		5
126	end of routing	1
127		2
128		3
129		4
130		5
131	linear routing	1
132		2
133		3
134		4
135		5
136	helicoidal routing	1
137		2
138		3
139		4
140		5
145	circular routing	5
146	tapping	1
147		2
148		3
149		4
150		5
155	xyz rapid	5
160	measure of thickness	5

DRILL IN FACE 1 Fun101

DRILL IN FACE 2 Fun102

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx working	Tool deplacement X coordinate
P06 Qy air	Air Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Qz working	Working Z coordinate
P19 Vy entr. slowing	Entry slowing y speed
P20 Qy ent. slowing	Entry slowing Y coordinate
P21 Vy working	Penetration Y speed
P22 Qy exit slowing	Exit slowing Y coordinate
P23 Vy exit slowing	Exit slowing Y speed
P24 Qy working	Maximum penetration Y coordinate
P25 Qz air ex.	Z lift coordinate (air or tool exchange coordinate)
P26 Byte tool	Byte of tool after 10 working cycles

DRILL IN FACE 3 Fun103

DRILL IN FACE 4 Fun104

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx air	Air X coordinate
P06 Qy working	Working Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Qz working	Working Z coordinate
P19 Vx entr. slowing	Entry slowing X speed
P20 Qx ent. slowiing	Entry slowing X coordinate
P21 Vx working	Penetration X speed
P22 Qx exit slowing	Exit slowing X coordinate
P23 Vx exit slowing	Exit slowing X speed
P24 Qx working	Maximum penetration X coordinate
P25 Qz air ex.	Z lift coordinate (air or tool exchange coordinate)
P26 Byte tool	Byte of tool after 10 working cycles

DRILL IN FACE 5 Fun105

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx Working	Working X coordinate
P06 Qy working	Working Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Vz ent. slow	Entry slowing Z speed
P19 Qz ent. slow	Entry slowing Z coordinate
P20 Vx working	Penetration Z speed
P21 Qz exit slow	Exit slowing Z coordinate
P22 Vz exit slow	Exit slowing Z speed
P23 Qz working	Maximum penetration Z coordinate
P24 Qz air ex.	Z lift coordinate (air or tool exchange coordinate)
P25 Byte tool	Byte of tool after 10 working cycles

FITTING IN FACE 1 Fun106

FITTING IN FACE 2 Fun107

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 stroke number	Drilling stroke number
P05 V Spindle	Spindle rotation speed (Max 10 Volt)
P06 Qx working	working X coordinate
P07 Qy air	Air Y coordinate
P08 Tool mask 0	Tool mask enabled from 01 to 08
P09 Tool mask 1	Tool mask enabled from 09 to 16
P10 Tool mask 2	Tool mask enabled from 17 to 24
P11 Tool mask 3	Tool mask enabled from 25 to 32
P12 Tool mask 4	Tool mask enabled from 33 to 40
P13 Tool mask 5	Tool mask enabled from 41 to 48
P14 Tool mask 6	Tool mask enabled from 49 to 56
P15 Tool mask 7	Tool mask enabled from 57 to 64
P16 Tool mask 8	Tool mask enabled from 65 to 72
P17 Tool mask 9	Tool mask enabled from 73 to 80
P18 Qz air	Air Z coordinate
P19 Vy ent. slow	Entry slowing Y speed
P20 Qy ent. slow	Entry slowing Y coordinate
P21 Vy working	Penetration Y speed
P22 Qy working	Maximum penetration Y coordinate
P23 X pitch incr.	X fitting pitch
P24 Y pitch incr.	Y fitting pitch
P25 Qx end.	Last stroke X coordinate
P26 Qy end	Last stroke Y coordinate
P27 Qz working	Working Z coordinate
P28 Qz exchange.	Exchange z coordinate

P29 Byte tool	Byte of tool after 10 working cycles

FITTING IN FACE 3 Fun108

FITTING IN FACE 4 Fun109

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Stroke number	Drilling stroke number
P05 V Spindle	Spindle rotation speed (Max 10 Volt)
P06 Qx air	Air X coordinate
P07 Qy working	Working Y coordinate
P08 Tool mask 0	Tool mask enabled from 01 to 08
P09 Tool mask 1	Tool mask enabled from 09 to 16
P10 Tool mask 2	Tool mask enabled from 17 to 24
P11 Tool mask 3	Tool mask enabled from 25 to 32
P12 Tool mask 4	Tool mask enabled from 33 to 40
P13 Tool mask 5	Tool mask enabled from 41 to 48
P14 Tool mask 6	Tool mask enabled from 49 to 56
P15 Tool mask 7	Tool mask enabled from 57 to 64
P16 Tool mask 8	Tool mask enabled from 65 to 72
P17 Tool mask 9	Tool mask enabled from 73 to 80
P18 Qz air	Air Z coordinate
P19 Vx ent. slow	Entry slowing X speed
P20 Qx ent. slow	Entry slowing X coordinate
P21 Vx working	Penetration X speed
P22 Qx working	Maximum penetration X coordinate
P23 incr. X pitch	Fitting X pitch
P24 Y pitch incr.	Y fitting pitch
P25 Qx end.	Last stroke X coordinate
P26 Qy end	Last stroke Y coordinate
P27 Qz working	Working Z coordinate
P28 Qz exchange.	Exchange z coordinate
P29 Byte tool	Byte of tool after 10 working cycles

FITTING IN FACE 5 Fun110

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Stroke number	Drilling stroke number
P05 V Spindle	Spindle rotation speed (Max 10 Volt)
P06 Qx air	Working X coordinate
P07 Qy working	Working Y coordinate
P08 Tool mask 0	Tool mask enabled from 01 to 08
P09 Tool mask 1	Tool mask enabled from 09 to 16
P10 Tool mask 2	Tool mask enabled from 17 to 24
P11 Tool mask 3	Tool mask enabled from 25 to 32
P12 Tool mask 4	Tool mask enabled from 33 to 40
P13 Tool mask 5	Tool mask enabled from 41 to 48
P14 Tool mask 6	Tool mask enabled from 49 to 56
P15 Tool mask 7	Tool mask enabled from 57 to 64
P16 Tool mask 8	Tool mask enabled from 65 to 72
P17 Tool mask 9	Tool mask enabled from 73 to 80
P18 Qz air	Air Z coordinate
P19 Vz ent. slow	Entry slowing Z speed
P20 Qz ent. slow.	Entry slowing Z coordinate
P21 Vz working	Penetration Z speed
P22 Qz working	Maximum penetration Z coordinate
P23 incr. X pitch	Fitting X pitch
P24 incr. Y pitch.	Fitting Y pitch
P25 last X pitch	Fitting last stroke X pitch
P26 last Y pitch	Fitting last stroke Y pitch
P27 Qz air lift	Lift coordinate Z (air or tool exchange coordinate)
P28 Vz exit slow	Exit slowing Z speed
P29 Qz exit slow	Exit slowing Z coordinate

P30 Byte tool Byte of tool after 10 working cycles

INSERTION FACE 1 Fun111

INSERTION FACE 2 Fun112

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Tool mask 0	Tool mask enabled from 01 to 08
P05 Tool mask 1	Tool mask enabled from 09 to 16
P06 Tool mask 2	Tool mask enabled from 17 to 24
P07 Tool mask 3	Tool mask enabled from 25 to 32
P08 Tool mask 4	Tool mask enabled from 33 to 40
P09 Tool mask 5	Tool mask enabled from 41 to 48
P10 Tool mask 6	Tool mask enabled from 49 to 56
P11 Tool mask 7	Tool mask enabled from 57 to 64
P12 Tool mask 8	Tool mask enabled from 65 to 72
P13 Tool mask 9	Tool mask enabled from 73 to 80

P14

P15

P16

P17

INSERTION FACE 3 Fun113

INSERTION FACE 4 Fun114

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Tool mask 0	Tool mask enabled from 01 to 08
P05 Tool mask 1	Tool mask enabled from 09 to 16
P06 Tool mask 2	Tool mask enabled from 17 to 24
P07 Tool mask 3	Tool mask enabled from 25 to 32
P08 Tool mask 4	Tool mask enabled from 33 to 40
P09 Tool mask 5	Tool mask enabled from 41 to 48
P10 Tool mask 6	Tool mask enabled from 49 to 56
P11 Tool mask 7	Tool mask enabled from 57 to 64
P12 Tool mask 8	Tool mask enabled from 65 to 72
P13 Tool mask 9	Tool mask enabled from 73 to 80

P14

P15

P16

P17

INSERTION FACE 5 Fun115

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Tool mask 0	Tool mask enabled from 01 to 08
P05 Tool mask 1	Tool mask enabled from 09 to 16
P06 Tool mask 2	Tool mask enabled from 17 to 24
P07 Tool mask 3	Tool mask enabled from 25 to 32
P08 Tool mask 4	Tool mask enabled from 33 to 40
P09 Tool mask 5	Tool mask enabled from 41 to 48
P10 Tool mask 6	Tool mask enabled from 49 to 56
P11 Tool mask 7	Tool mask enabled from 57 to 64
P12 Tool mask 8	Tool mask enabled from 65 to 72
P13 Tool mask 9	Tool mask enabled from 73 to 80
P14 Qx working	Working X coordinate
P15 Qy working	Working Y coordinate
P16 Vz working	Penetration Z speed
P17 Qz working	Maximum penetration Z coordinate
P18 Qz lift air	Lift Z coordinate (air or tool exchange lift)

SAW IN FACE 5 Fun120

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx saw ent.	Saw entry X coordinate
P06 Qy saw ent.	Saw entry Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Vz working	Penetration Z speed
P19 Qz working	Maximum penetration Z coordinate
P20 Vx working	Saw X speed
P21 Vy working	Saw Yspeed
P22 Qx working	Working X coordinate
P23 Qy working	Working Y coordinate
P24 Qz air ex.	Lift Z coordinate (air or tool exchange coordinate)
P25 Qrot	Rotation of aggregate
P26 Byte tool	Byte of tool after 10 working cycles

SAW ON A IN FACE 5 Fun119

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx saw ent.	Saw entry X coordinate
P06 Qy saw ent.	Saw entry Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Vz working	Penetration Z speed
P19 Qz working	Maximum penetration Z coordinate
P20	Axes mask
P21	Axes direction (1 = '-'; 0 = '+')
P22	Rateic
P23	Resol (step)
P24	Axis 1 coordinate (in Resol units)
P25	Axis 2 coordinate (in Resol units)
P26	Axis 3 coordinate (in Resol units)
P27	diagonal
P28 Qz air ex.	Lift Z coordinate (air or tool exchange coordinate)
P29 Qrot	Rotation of aggregate
P30 Byte tool	Byte of tool after 10 working cycles

ROUTER PENETRATION IN FACE 1 Fun121

ROUTER PENETRATION IN FACE 2 Fun122

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx work	Tool entry X coordinate
P06 Qy air	Air Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 a 08
P08 Tool mask 1	Tool mask enabled from 09 a 16
P09 Tool mask 2	Tool mask enabled from 17 a 24
P10 Tool mask 3	Tool mask enabled from 25 a 32
P11 Tool mask 4	Tool mask enabled from 33 a 40
P12 Tool mask 5	Tool mask enabled from 41 a 48
P13 Tool mask 6	Tool mask enabled from 49 a 56
P14 Tool mask 7	Tool mask enabled from 57 a 64
P15 Tool mask 8	Tool mask enabled from 65 a 72
P16 Tool mask 9	Tool mask enabled from 73 a 80
P17 Qz air	Air Z coordinate
P18 Qz working	W working Z coordinate
P19 Vy working	Penetration Y speed
P20 Qy working	Maximum penetration Y coordinate

ROUTER PENETRATION IN FACE 3 Fun123

ROUTER PENETRATION IN FACE 4 Fun124

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx air	Air X coordinate
P06 Qy working	Working Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Qz working	W working Z coordinate
P19 Vx working	Penetration X speed
P20 Qx working	Maximum penetration X coordinate

DRILL PENETRATION IN FACE 5 Fun125

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 V Spindle	Spindle rotation speed (Max 10 Volt)
P05 Qx working	Working X coordinate
P06 Qy working	Working Y coordinate
P07 Tool mask 0	Tool mask enabled from 01 to 08
P08 Tool mask 1	Tool mask enabled from 09 to 16
P09 Tool mask 2	Tool mask enabled from 17 to 24
P10 Tool mask 3	Tool mask enabled from 25 to 32
P11 Tool mask 4	Tool mask enabled from 33 to 40
P12 Tool mask 5	Tool mask enabled from 41 to 48
P13 Tool mask 6	Tool mask enabled from 49 to 56
P14 Tool mask 7	Tool mask enabled from 57 to 64
P15 Tool mask 8	Tool mask enabled from 65 to 72
P16 Tool mask 9	Tool mask enabled from 73 to 80
P17 Qz air	Air Z coordinate
P18 Vz working	Penetration Z speed
P19 Qz working	Maximum penetration Z coordinate
P20 Qx air 2	X air coordinate for tool change
P21 Qy air 2	Y air coordinate for tool change
P22 Qz air 2	Z air coordinate for tool change
P23 Qrot	Rotation of aggregate

ROUTER LIFT IN FACE 1 Fun126

ROUTER LIFT IN FACE 2 Fun127

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Qy air	Air Y coordinate
P05 Qz air	Lift Z coordinate (air or 0)
P06 Byte tool	Byte of tool after 10 working cycles

ROUTER LIFT IN FACE 3 Fun128

ROUTER LIFT IN FACE 4 Fun129

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Qx air	Air X coordinate
P05 Qz air	Lift Z coordinate (air or 0)
P06 Byte tool	Byte of tool after 10 working cycles

ROUTER LIFT IN FACE5 Fun130

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04 Qz air	Lift Z coordinate (air or 0)
P05 Byte tool	Byte of tool after 10 working cycles

LINEAR ROUTING IN FACE 1 Fun131

LINEAR ROUTING IN FACE 2 Fun132

LINEAR ROUTING IN FACE 3 Fun133

LINEAR ROUTING IN FACE 4 Fun134

LINEAR ROUTING IN FACE 5 Fun135

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04	Axes mask
P05	Axes direction (1='-' ; 0='+')
P06	Rateic
P07	Resol (step)
P08	Axis measure 1 (in Resol units)
P09	Axis measure 2 (in Resol units)
P10	Axis measure 3 (in Resol units)
P11	Diagonal

CIRCULAR ROUTING IN FACE 5 Fun145

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04	Axes mask
P05	Rotation sense
P06	Rateic
P07	Resol (step)
P08	Arc (in Resol units)
P09	Axis 1centre measure (in Resol units)
P10	Axis 2 centre measure (in Resol units)
P11	Axis 1 final measure (in Resol units)
P12	Axis 2 final measure (in Resol units)
P13	Radius (in Resol units)
P14	umber of turns

HELICOIDAL ROUTING IN FACE 1 Fun136

HELICOIDAL ROUTING IN FACE 2 Fun137

HELICOIDAL ROUTING IN FACE 3 Fun138

HELICOIDAL ROUTING IN FACE 4 Fun139

HELICOIDAL ROUTING IN FACE 5 Fun140

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04	Circular axes mask
P05	Linear axes mask
P06	Rotation sense
P07	Rateic
P08	Resol (step)
P09	Axes 3 measure (in Resol units)
P10	Arc (in Resol units)
P11	Diagonal ((in Resol units)
P12	Axis 1centre measure (in Resol units)
P13	Axis 2 centre measure (in Resol units)
P14	Axis 1 final measure (in Resol units)
P15	Axis 2 final measure (in Resol units)
P16	Radius (in Resol units)
P17	Turns number

RAPID IN FACE 5 Fun155

P01 Dummy byte 1
P02 Dummy byte 2
P03 Dummy byte 3

P04	Axis X coordinate
P05	Axis Y coordinate
P06	Axis Z coordinate
P07	User function.

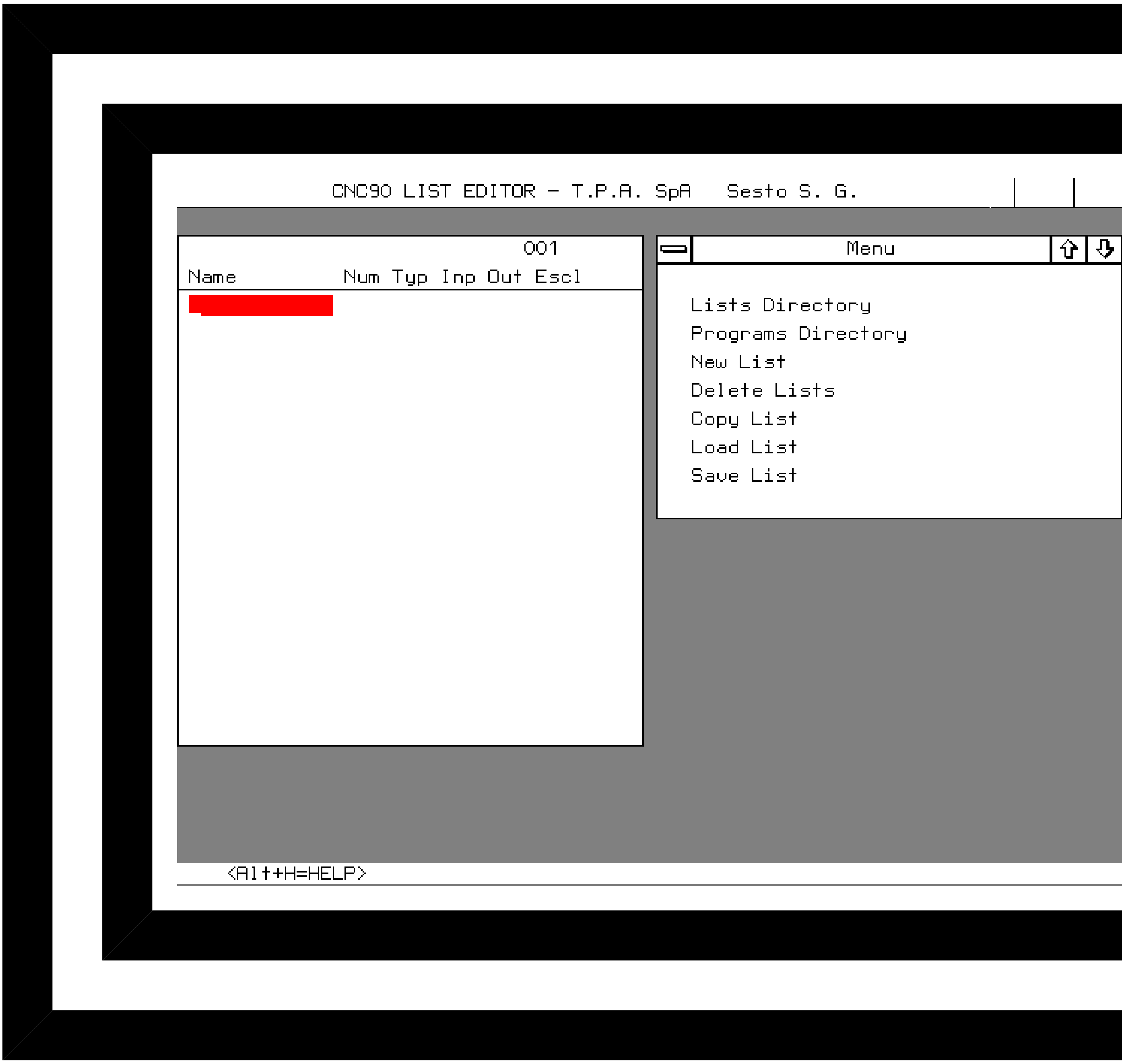
MEASURE OF THICKNESS (IN FACE 5) Fun160

P01 Qx working	Working X coordinate
P02 Qy working	Working Y coordinate
P03 Q1 corr. Z	coordinate 1 of Z corrector
P04 Q2 corr. Z	coordinate 2 of Z corrector
P05 Byte tool	Byte of tool after 10 working cycles

12. LIST EDITOR

After having selected voice *List Editor* in main menu, ambient List Editor is accessed for creating and modifying program lists to be executed later in *Automatic* mode.

Here below it is depicted LIST EDITOR Main Display with its operative fields.



Picture 12.0

12.1 MAIN DISPLAY

As shown in picture 12.0, List Editor Main Display is made by two program windows, one placed on the right side of the screen (Main Menu) and the other one on the left (Editing window).

This two windows perform different tasks. Infact, the activation of one forbids the activation of the other. To enable one window and disable the other press key **F1**, and the control will switch from a window to the other.
When a window is enabled, an indicator bar is displayed allowing to insert data in the list or to select menu voices.

Editing window is composed by 15 lines for 6 columns and represent the area where the programs list is edited, while the Menu is used to load List Editor Accessory Modes.
The Menu will be discussed later, in a different section; here, it will be examined how to operate with Editing.

The lists are a whole of programs ordered according to user's needs. Every list can contain up to 200 programs and can have a list multiplying factor of up to 999.

A list is not only a whole of program names, but also of processing codes.
In facts, besides Name field, to each programs are related four more editable fields representing respectively: single program multiplying factor (a value included within 1 and 255); processing code (normal, specular, ...), input code; output code (two codes with special purposes and values included within 0 and 999, used for personalisation) and exclusion code (defined exclusions).

Once the window has been activated it is possible to start immediately with editing the programs list. The editable area is composed by 200 lines (max. number of programs in a list), and 6 columns (program's name, multiplying factor, etc.); since the lines displayed by the window are only 15, it will be necessary to scroll the list by means of direction keys ARROW UP, ARROW DOWN, PG UP, and PG DN.

Each line of the window corresponds to a program contained in the list and each column corresponds to an editable field.

The first editable field, thus the outermost left column, represents the field containing the name of the program to insert in the list. The second contains the List Multiplying Factor (a value among 1 and 255). The third contains the Processing Code, made by three digits which identify 28 types of different processes (see Table of Working Codes). The last two fields contains respectively the program Input and Output Codes to personalise machine cycles for managing workpieces load and unload.

Last column represents the exclusion code which identifies 8 exclusions, numbered from 1 to 8, in correspondence with programmable field in CNC90 graphic editor.

Exclusion code allows to exclude from program execution some working lines, if defined by an exclusion number from 1 to 8 ("Ex." field in CNC90 Graphic Editor).

If for instance working lines with exclusion number equal to 2 and 7 should be excluded, in automatic exclusion code should be compiled with values 2 and 7 that is write 27.

COMMANDS IN LIST EDITOR

In this section the commands allowing insertions, modifications and deletion of programs lists will be examined.

Some command runs only when a sequence of keys is pressed: for example, by pressing CTRL key and Y at the same time current line, the line onto which the cursor is set, will be deleted from programs list.
However, not all commands relate to a sequence of keys: in this table, all commands related to more than 1 key interpose sign + between each key.

KEY	**FUNCTION**
INSERT	Enables Insert mode, to insert new characters in the text. Press again key INSERT to disable Insert mode.
DELETE	Delete first digit at the right side of the cursor.
BACK SPACE	Delete first digit at the left side of the cursor.
HOME	Shift cursor on the first edited digit.
END	Shift cursor on the last edited digit.
ESC	Delete the edited field.
ARROW RIGHT	Shift cursor one digit right.

ARROW LEFT	Shift cursor one digit left.
PAGE UP	Display previous page.
PAGE DOWN	Display next page
TAB	Shift cursor to first editable field on the right side of current one. When the cursor is by the last field, it shift to the first one
SHIFT + TAB	Shift cursor to first editable field on the left of current one. When the cursor is by the first field, it shifts to the last one.
ARROW UP	Move cursor to pervious line.
ARROW DOWN	Move cursor to next line.
ENTER	Move cursor to next editable field.
CTRL + D	Empty Programs List.
CTRL + Y	Delete current line.
CTRL + U	Insert last line deleted by command CTRL + Y above current line (UNDO line).
CTRL + N	Sets cursor on upper right edge of Automatic row to edit the list multiplying factor.
CTRL + ENTER	Insert an empty line above current line.
CTRL + HOME	Shift cursor atop the list (quick positioning a top the list).
CTRL + S	Locate a word within a list, starting from cursor current position up to the end of the list
F1	Exit from editing mode and enter Main menu mode
ALT + H	Loads HELP.

In case a command can't be executed, a beep will warn the operator. If the beep cannot be heard, for instance when the PC is very close the machines or for any other reason, the operator will realise command failure only by its outcome, which can be a fully processed workpiece if the command has been run correctly, a partially processed workpiece if the command has been run partially or a unprocessed workpiece if the command has not been run at all.

12.2 MAIN MENU

As previously mentioned, from this menu it is possible to access List Editor Accessory Modes.

Once the Menu window has been enabled, an indicator bar will appear on the first voice of the menu. The bar is used to select the proper voice among the ones proposed.

Use direction keys ARROW UP, ARROW DOWN, PAGE Up and PAGE DOWN to move along the list. To select a given voice position onto it the indicator bar and press ENTER.

To quit the menu, enabling therefore the Editing window, press key **F1**

In the next pages will be discussed all modes displayed in the Menu window.

LISTS DIRECTORY

A small window is displayed, below Main Menu, containing lists directory.

Menu
000-0010
AAPF000
ART100-5
DEMOART
SPF8000
TRIALART
ZULUP40

Fig. 12.1 Lists directory.

Since only 7 list at a time can be displayed, it will be necessary to move along the scrollable list with direction keys to display the full directory.

Press key ESC to return to Main Menu.

PROGRAMS DIRECTORY

A window containing the programs directory (Fig. 12.2) is displayed; the list is scrollable: use direction keys to move along and display the full directory.

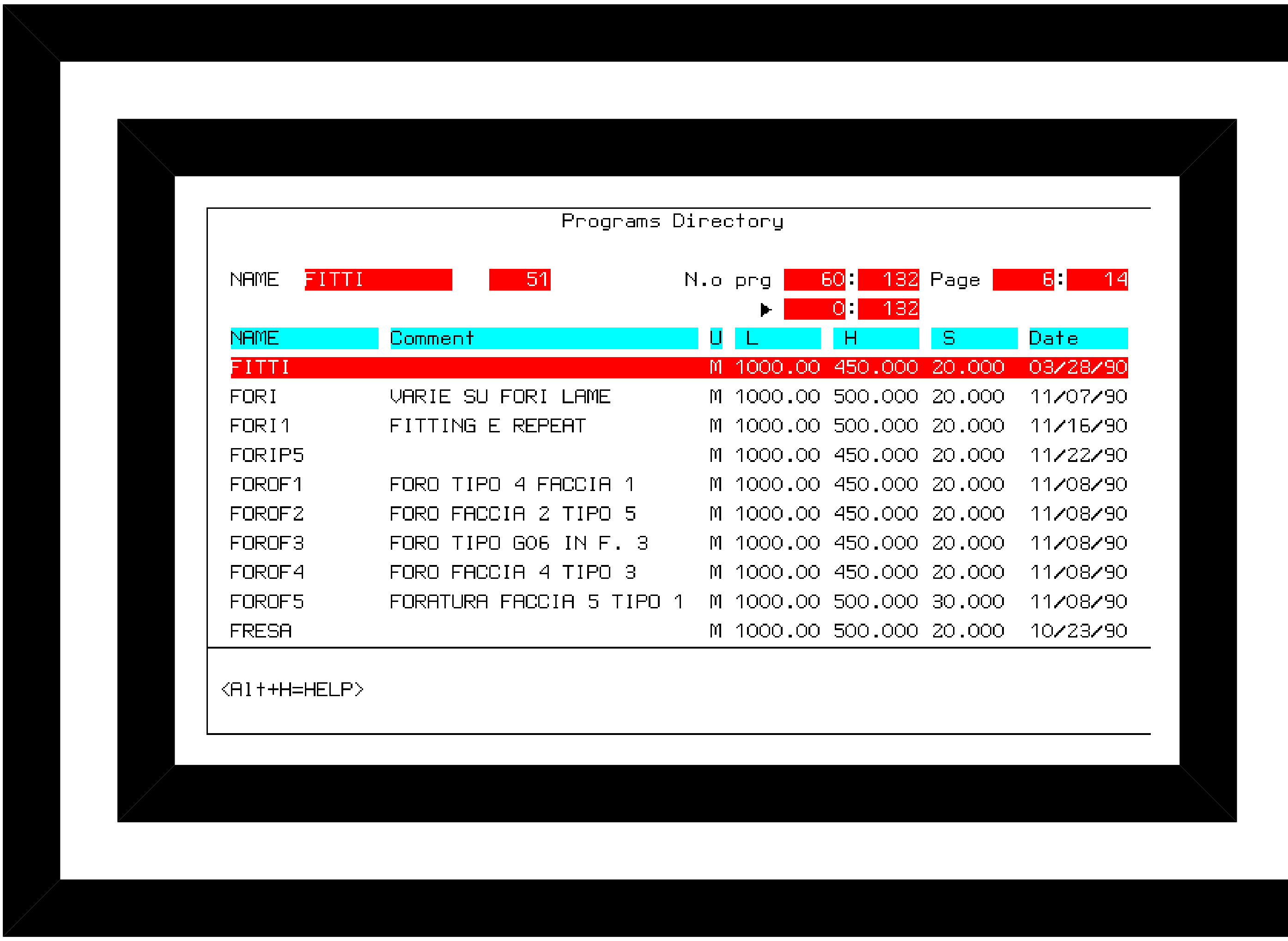


Fig. 12.2 Programs directory.

Use SPACE BAR and direction keys ARROW UP and ARROW DOWN to select one or more programs to input in Automatic Mode list. They shall be automatically inserted in the line immediately above the one in which the cursor is positioned.

To select a program set indicator bar onto it and press SPACE BAR. Once a program has been selected, press again key ENTER to confirm or key ESC to abort selection (A selected program is marked by a small triangle preceding its name).

Select a program a second time to disables it. The small triangle will disappear, and the program returns to normal mode.

NEW LIST

It allows creation of a new list. After selection, a question string will appear (fig. 12.3)

Are you sure of selected requirement (S/N) ? N

Fig. 12.3

The answer will be logically Yes if the creation of a new list was the meaning, and No otherwise. Also SPACE BAR can be used.
Once answered Yes to this question string, a second string requiring the insertion of the new lists name (Fig. 12.4) is displayed. The name should comply with DOS specifications.

Insert new list name ... :

Fig. 12.4

Important: If a list is present in the Editing window, save it before selecting this command otherwise it will be deleted.
By pressing key ESC the command is aborted.

DELETE LIST

Lists directory is displayed to select the lists to delete. Move along the list as explained for the Directory windows (by means of direction keys ARROW UP, ARROW DOWN, PAGE UP). To select the list to delete position indicator bar onto it and press SPACE BAR (Fig. 12.5).

A small triangle preceding the list name indicates selected list. Disable a selected list by selecting it a second time; the small triangle will disappear.

Menu

000-0010
AAPF000

DEMOART
SPF8000
TRIALART
ZULUP40

Fig. 12.5

Once selection has been performed, press ENTER and a question string will appear (see fig. 12.3) to require confirmation. Answer by pressing Y (Yes) or N (No), or press SPACE BAR.
Press ESC to abort the command.

COPY LIST

This command allows to copy the list displayed in the Editing window into a list with a different name. Immediately after selection, a string will appear (Fig. 12.6) and ask the operator to enter the name of destination list; once this name has been entered, press ENTER to confirm or ESC to abort the operation.

Enter the name of destination list ... :

Fig. 12.6

LOAD LIST

Loads and enables a list from List Directory to Editing window.
Again the lists directory is displayed with its indicator bar (Fig. 12.7)

Menu
000-0010
AAPF000
ART100-5
SPF8000
TRIALART
ZULUP40

Fig. 12.7

Move along the scrollable list with direction keys. Once the required list has been found, position indicator bar onto it and confirm with ENTER, otherwise press ESC to abort the command.
If a list is present in the Editing window save it before running this command, otherwise it will be deleted.

SAVE LIST

Allows to save in Lists Directory the list present in the Editing window. If the list to save has not been named, a string will appear asking the operator to assign a name to the list (Fig. 12.8); after having performed this operation, press ENTER to confirm or ESC to abort storage.

Save list name ... :

Fig. 12.8

IMPORT LIST FROM AUTOMATIC

Introduces last list edited in Automatic Mode allowing thus to display and to activate it for some modifications, etc.

Before proceeding to inscription a string will appear requiring a confirmation of the command just given (Fig. 12.3). Answer by using Y (Yes), N (No) by SPACE BAR. Once performed this operation a second string will appear requiring a name to assign to the list (pct. 12.4). Warning : if a list is present in the Editing window, save it before entering this command or it shall be deleted.
Key ESC allows to abort the command.

PRINT LISTS

Lists directory is displayed to select the lists to print. Move along the list as explained for the Directory windows (by means of direction keys ARROW UP, ARROW DOWN, PAGE UP). To select the list to print position indicator bar onto it and press SPACE BAR (fig. 12.5).

A small triangle preceding the list name indicates selected list. Select the list a second time, to disable it; the small triangle will disappear.

Once selection has been performed, press ENTER to print selected lists. Press ESC to abort the command.

RENAME LISTS

As for printing lists directory is displayed to select the lists to rename.
Use direction keys ARROW UP, ARROW DOWN, PAGE UP to scroll the Directory. To select the list to rename, position indicator bar onto it and press SPACE BAR (fig. 12.5).

A small triangle preceding the list name indicates selected list. Select the list a second time, to disable it; the small triangle will disappear.

Once selection has been performed, press ENTER to print selected lists.
Press ESC to abort the command.

Rename list name ... :

Insert new list name and press ENTER to confirm.

Press ESC key to abort the command.

QUIT LIST EDITOR

After confirmation (fig. 12.9) :

Quit Editor : are you sure ? (Y/N) N

Fig. 12.9

to which the answer can be given with Y (Yes), N (No) or with SPACE BAR, the PC returns to D.O main Menu.
To quit List Editor simply press key ESC in Main Menu.
Press ESC key to abort the command and return to Main Menu.

12.3 ERRORS IN LIST CREATION

Errors are displayed within a black window with red labels and frame, displayed at the centre of the screen.

List 000-001 already existing, storage aborted !

List TEST-ALL already existing, storage aborted !

General Errors Window

Let's now examine all the errors that can be displayed .

List 000-001 already existing, storage aborted !
If the name of a list already existing is assigned to a list to save or copy,the system abort the storage.
Solution: change name to the list to save or delete the list with the same name.

Impossible to save list 000-001 storage aborted !
For some internal reason, which can range from a damaged disk to a wrong list (example, if the list is empty), the list cannot be saved. Try again, and if it is really impossible to perform the operation contact Assistance.

List 000-001 already existing in directory !
The same name of a list already existing in Directory has been given to a list; rename the list or delete the list already existing in directory.

Impossible to save two lists with the same name !
When copying the list, the system has realised that destination list name is the same as origin list. Change the name of destination list.

List name incorrect, impossible to use !
The list name as defined by the operator does not comply with DOS specifications. Therefore, the system cannot use this name to identify the list. Please refer to DOS User's manual for more detailed information concerning the compilation of DOS names.

Impossible to read list 000-001!
List 000-001 recorded on disk is damaged, impossible to read.
Remove disk errors and try again. (Verify disk integrity by DOS CHKDSK command, check disk.

Impossible to rename list 000-001!
List 000-001 new name is not correct, impossible to rename the list; or another list with the name is already existing in Lists directory. Change name and try again.

13. FILES MANAGEMENT

FILES MANAGEMENT mode is accessed from D.O. Main Menu by selecting the same voice.

It allows handling all files defined on CNC90 control. In particular the following:

- Programs files
- Lists files
- Parameters files

These files allow general operations of copy and/or delete. Main uses relate to:

1) Creation of backup copies on non-working disks;
2) File copy from different disks to working disk;
3) File deletion from working disk or others.

Video is paginated as follows :

SOURCE O. DRIVE byte occ. **DESTINATION** D. DRIVE byte free **MESSAGES**	**MENU** COPY PROGRAMS DELETE PROGRAMS COPY LISTS DELETE LISTS COPY ALL PROGRAMS DELETE ALL PROGRAMS COPY PARAMETERS ORIGIN DRIVE DESTINATION DRIVE

On the right side of the screen it is displayed the menu with all selectable commands.

On the left side of the screen are displayed three windows, headed as:

- **SOURCE** to assign the Source drive. It also reports drive occupation.

- **DESTINATION** to assign the Destination drive. It also reports drive occupation.

- **MESSAGES** to display messages.

Drive windows are initialized on default disk units, as indicated in Files Management module configuration program (GESARC.PAR)

Messages window is initialized with indication of the first operation to be performed.

To select among the menu voices use indicator bar (word in reverse), moved by the direction key on numeric keyboard, and confirm with ENTER.

Allowed modes are listed here below:

SOURCE DRIVE

Used to select disk unit from which Copy or Delete operations are performed.
On the lower part of the screen the available selections are displayed.

DESTINATION DRIVE

Used to select disk unit on which Copy or Delete operations are performed.

Drives may be selected as follows:

1 - a symbolic name associated to the drive
(for instance: HARD DISK C)
2 - a disk unit (for instance hard disk addressed as C:)
3 - a path for source programs
(for instance: C.\CNC90\MOD1\SORG)
4 - a path for compiled programs
(for instance: C.\CNC90\MOD1\COMP)
5 - a path for lists
(for instance: C.\CNC90\MOD1\SORG)
6 - a path for machine parameters
(for instance: C.\CNC90\MOD1\SORG)

On the lower part of the screen the available selections are displayed.

PROGRAMS COPY

Used to copy programs from a SOURCE file to a DESTINATION file.

- First, the system checks if the two selected files are defined as distinct. In case the files should be found equivalent, the operation is not performed and the following message will appear in the message box: "SOURCE AND DESTINATION DRIVE ARE EQUIVALENT". Press ENTER to reset the error situation.

- If Programs Directory assigned on Destination Drive does not exist, the following message will be displayed:
"DESTINATION DIRECTORY NOT EXISTING: CREATE A NEW ONE? Y/N"

Answer Y (Yes) to confirm and keep in the mode.
Answer N (No) to quit the mode.

- After that, Programs Directory recorded in Source Drive will be displayed to select the programs to be copyed.
Use direction keys ARROW UP and ARROW DOWN to scroll the list and select correct programs.
When desired program is found press SPACE BAR: a graphic symbol will appear on the left side of program name, indicating its selection.
To display previous/next pages use keys PG UP and PG DN.
Finally, confirm selections by pressing ENTER.

Once the selections have been confirmed:

- The module checks if selected programs exist in Destination drive. If so, a list will propose the programs already existing in Destination archive: select on the list the programs to be copyed, or quit by pressing ENTER;

- If memory availability on Destination drive is unsufficient, the following message will be displayed:
"NOT ENOUGH SPACE ON DESTINATION. DRIVE"
Press ENTER to return to Main Menu.

During copy, the system automatically displays the program being copied.

Once the operation has been succesfully accomplished, a message will appear:
"PROGRAMS COPY HAS ENDED CORRECTLY "

Otherwise, in case the operation should have ended uncorrectly, various diagnostical messages will report the specific error.
Messages are displayed in the message box and may be scrolled: use therefore keys PG UP and PG DW to read the full message. Press ESC to quit.

The Error messages displayed are:

- "SOURCE FILE NOT FOUND"
  There is no Source file in Source drive.

- "COMPILED FILE NOT FOUND"
  There is no compiled file in Source drive

- "COPY ABORTED"
  Copy has been interrupted by pressing key ESC.

Window for error messages may be recalled by pressing function key F1.

PROGRAMS DELETION

It allows deletion of programs in Source archive.

- First, a confirmation of Source drive is required.

- Then, Programs Directory on Source drive is displayed, for selection of programs to be deleted.

Select the programs as for Copy.

During deletion, the system automatically displays the program being deleted.

The only Error message displayed in this operation is the following:

"DELETION ABORTED"

in case deletion has been interrupted by pressing key ESC.

COPY LISTS

It allows copying Programs Lists from a Source to a Destination archive.

The procedure is the same as described for the copy of programs.

If the operation has been correctly performed, the following message will appear:
"LISTS COPY HAS ENDED CORRECTLY"
Press ENTER to continue.

In case the operation should not have ended correctly, diagnostic messages will warn about the specific error.

Error messages displayed are:

- "COPY ABORTED"
  When Copy has been interrupted with key ESC.

- "ERROR: LISTS FILE NOT FOUND"
  When Lists directory appears to be empty.

LISTS DELETION

It allows deletion of lists in Source file.
The procedure is the same as previuosly described for Programs deletion.

COPY ALL LISTS

Allows complete copy of an entire archive of programs, without specific selection within programs directory.
In case of copy into a diskette, a new disk is automatically asked when necessary; each disk has a specific programs directory, and can be used separately for program transfers.

DELETE ALL LISTS

This operation mode allows complete deletion of the archive on the source drive; it requires confirm, because of its destructive effects.

COPY PARAMETERS

It allows copying given parameters from Source drive to Destination drive; specifically:

a) technologic parameters
b) heads parameters
c) toolings assignment
d) special configurations (for CNC90 Editor, backup program, etc.)
e) parameters and functions for System Auxiliary Menu

The system verifyes whether the two archives selected are nor coincident. In case of coincident archives, the operation is aborted and the message :

"SOURCE AND DESTINATION DRIVES ARE EQUIVALENT"

will report the Error situation. Press ENTER to continue.

If space available in Destination drive is unsufficient, the following message will appear:

"NOT ENOUGH SPACE ON DESTINATION DRIVE"

Press ENTER to return to Main Menu.

After these checks, the system starts copying the selected parameters.

During this operation, the system automatically displays the programs being currently copied.

Once the operation has eneded correctly, the following message will appear:

"PARAMETERS COPY HAS ENDED CORRECTLY"

Press ENTER to continue.

If Copy is interrupted by pressing key ESC, the following message will warn the operator:

"PARAMETERS COPY ABORTED."

DESCRIPTION OF FILE : GESARC.PAR

File GESARC.PAR contains all information needed to define disk units (drives) available on the system, and the ones active by default.
This file is necessary for correct handling of File Management mode. If missing, the system displays an error message:

ERROR: DRIVER DATA FILE NOT FOUND

File GESARC.PAR should be created under directory USER, and should contain what follow:

"NN"	Total n. of disk units (drive)
"1"	Default n. of Source drive.
"0"	Default n. of Destination drive
"1ST DRIVE SYMBOLIC NAME"	Max. 12 digits - symbolic name
"1ST DRIVE SOURCES PATH"	Max. 32 digits
"1ST DRIVE FILES PATH"	Max. 32 digits
"1ST DRIVE LIST PATH"	Max. 32 digits
"1ST DRIVE PARAMETERS PATH"	Max. 32 digits
~ ~	
"N. DRIVE SYMBOLIC NAME"	Max. 12 digits - symbolic name
"N. DRIVE SOURCES PATH"	Max. 32 digits
"N. DRIVE FILES PATH"	Max. 32 digits
"N. DRIVE LISTS PATH"	Max. 32 digits
"N. DRIVE PARAMETERS PATH"	Max. 32 digits

NOTICE: Every string in the file should be enclosed in apexes ("). Between a string and the next one there should be no empty lines.

Up to 10 drivers may be selected.

14. MANUAL MODE

Manual mode allows manual control of axes motion and of Input/Output rate.

Manual mode is accessed with at least one linked card, by pressing function key F5 in D.O. Main Menu.

Two main windows will be displayed on screen: one reports Axes Motion and the other reports Digital Input/Output.

AXES MOVEMENT			
Axis		coordinate(mm)	State
X	**Axis X**	0.000	step
Y	Axis Y	0.000	step
Step	:	1 mm	
Speedy	:	2 m/min	

INPUTS/OUTPUTS			
	000(i)	001(i)	002(i)
0.>>	STOPX	_______	_______
1.	STOPY	_______	______
2.	_______	_______	_______
3.	_______	_______	_______
4.	_______	_______	_______
5.	_______	_______	_______
6.	_______	EMERG	_______
7.	_______	_______	_______

In this mode, all data displayed on screen are continuously updated. To quit Manual mode press function key **F10 - exit**.

AXES MOTION WINDOW

The window representing axes motion displays data related to the axes originated on the card. For each axis it is reported:

- Name (X/Y/Z/W/V)
- Operator description
- Coordinates
- Current mode.

One of the existing axis will be different than the others to indicate its selection. Reported data on Pitch and Velocity refer to selected axis.
To select a different axis it is enough to name it by pressing the correspondent key: all data on Pitch and Velocity shall immediately switch to the new axis selected.

To displace the selected axis it is enough to press key <+> or key <->, and the axis shall shift according to the parameters entered by the operator: Mode, Pitch and Velocity.

INPUT/OUTPUT WINDOW

This window displays control card digital Input/Output.
Data are ordered in strings and columns:
- Each column represent a port on the card, that is a group made of 8 lines (bit).
- Each line represent a string, for a total amount of   8 lines numbered from 0 to 7.

In the windows are represented three ports, for a total of 24 I/O (Input/Output) bits.

The address of a bit is located by combining bit number and port number.

Example: the first upper left bit has address 000, because it is bit 0 of port 0; the second bit in first port has address 1000; the third bit of first port
2001.
For each I/O bit are available 8 description characters on a string, as assigned in Editor mode. In case of definition not found, the string will be displayed empty.
Each bit mode (enabled/disabled or high/low) is represented marking the related area, in correspondence with an eventual description.

FUNCTION KEYS

Function keys related to axes motion are listed here below:

F1 - JOG/STEP (type of axis motion)

Press this key to switch axis current mode between JOG and STEP.
In JOG mode axis motion is started by pressing motion keys (or), and it keeps on untill the key is released.
In STEP mode, the axis performs a shifting of defined length as an incremental coordinates pitch in direction + or - from its current position.
The displacement is started by pressing keys or , and it can be interrupted by pressing key SPACE (SPACE BAR).

F2 - FREE

Press this key to switch the axis current mode from JOG or STEP to FREE.
In FREE mode the axis is not controlled by manual motion, while mechanical axis can be physically moved (manually, when possible) without the operation being recorded by the digital numeric control.
In facts the position loop is released. However, axis position is not lost and current coordinates are constantly being updated on screen.

F3 - AXPAR (Axis parameters)

Used for defining pitch and velocity of selected axis.
If current axis is in FREE mode, these parameters cannot be modified.
Data for each axis are then stored into memory and displayed again after each new selection.

F4 - AXIS (Axis selection)

Used for selecting one of the axes represented in the window. The result is the same as with selection operated by naming the axes with correspondent keys.
This mode enables also F1-JOG/STEP and F2-FREE modes, to modify the condition of any axis.
When this mode is active, only the data in AXIS MOTION window are updated, while the data in I/O window are disabled. This allows faster updatating of axes coordinates.

The remaining function keys available are:

F6 - IN/OUT (I/O selection and testing)

Press this key to update only the data contained in I/O window, excluding Axes Coordinates, to obtain a faster updatating of I/O data.
Key In/OUT allows to modify the ports represented in I/O window to check current condition of Inputs, Outputs and Virtual pulses.
Virtual Memory Areas are not physically connected with any I/O pulses and are usually named Flag Areas.

Besides the condition of any displayed bit can be modified, i.e. to enable or disable physical outputs or virtual outputs (flags).

Keys available in I/O window are:

: to move the cursor along the lines.

: move the cursor through the columns.

: change page.

- Shift cursor to first page.

- Shift cursor to last page.

: Enables or disables the line in which the cursor is positioned.

When this mode is aborted, only the last three current I/O ports are displayed, except when operating in Change Module and Stations mode (see CHANGE).

Next to each port address one of the following two letters is displayed, enclosed into brackets:

: to indicate that it is a port in Input area (physical or virtual);
: to indicate that it is a port in Output area (physical or virtual).

F7 - FUN

This key is used for immediate start of a function.
Upon selection, a list of all resident functions will be displayed, listed according to their number. Here below it is depicted a window displaying a functions list.

FUNCTION CONTROL		
057	SETPOI	Set-point function
092	CART2	Load task 2
093	CART3	Load task 3
198	FEMERG	Emergencies
.		
.		

Next to each function name appears the name of origin file and its
description, if defined.

Keys available in Functions window are the following:

: shift cursor on the lines.

: change page.

: Starts execution of selected function.

During the execution of selected function, all data in the different windows are automatically updated. Press SPACE BAR to abort the execution.
Only non-parametrical functions can be run: the proposed list displays only possible functions.

F8 VOUT

Allows to impose a voltage value on one of analogic output. Valid voltage values are included in the interval of : -10 Volt / + 10 Volt.

F9 - CHANGE

This key is used for modifying the module and the card onto which all operation of manual control and I/O control are performed. The change of module and/or card can be executed only if there exist more than one configured in the system.

During Card Selection a window containing all existing cards is displayed on the right side of the screen, supplying a comment on the cards, if defined.

Once the card has been selected, the Axes Motion window is reset so to comply with card configuration, allowing direct acknowledgement of the coordinates of all axes in the module and their condition.
Once ended card selection, the I/O window will be updated with the descriptions of the new card, if it has been changed. In this case, in I/O window will be displayed the first physical port of the new card.
Finally, key AXIS is automatically enabled to select the new axis.

MONITOR MODE

To access Monitor Mode, press together keys ALT and M.
Function is expounded in user manual of PTO1000system.

ERROR AND SERVICE MESSAGES

Error messages are displayed on the screen immediately below the two main square windows.

APPENDIX A. SYSTEM ERRORS

FOREWORD

System Errors are automatically checked by the cards and are sent to PC for their display.

These errors are of different natures: they can refer to problems with the axes, or problems in programs reception, etc.

In any case, System Errors ends the execution of any program on the module where the error originated.

In this section all kind of possible errors are expounded; where necessary, a short explanation is also supplied about the causes originating them.

Each error is identified by a message number, that can be translated into national language.

ERRORS CONCERNING EMERGENCY INPUTS

As indicated in Emergency Tables, this emergency normally indicates the activation of emergency procedures for any case in which program and axis motion must stop immediately.

(12) Auxiliary emergency
(13) Auxiliary emergency
(14) Auxiliary emergency

As indicated in Emergency Table, it is used to enable the execution of Emergency Control function.

ERRORS CONCERNING AXES CONTROL

Any error condition related to axes control stops immediately axes motion and resets to zero, if not differently defined, speed reference signal, disabling position control for a second to allow their automatic stop without oscillations, with the maximum available torque.

(2) Axis X - emergency
(3) Axis Y- emergency
(4) Axis Z - emergency
(5) AxisW- emergency
(5) Axis V- emergency

As indicated in Emergency Tables, Emerghency input is used for an immediate stop of the axes after activation of Emergency Limit switch.

(7) <u>Axis X - zero limit switch</u>
(8) <u>Axis Y- zero limit switch</u>
(9) <u>Axis Z - zero limit switch</u>
(10) <u>Axis W- zero limit switch</u>
(11) <u>Axis V - zero limit switch</u>

As indicated in Emergency Tables, it is used for an immediate stop of the axes after activation of a Limit switch used either as an emergency or as zero Limit switch. Infact, this emergency is conditioned by SPEX FLAG: if SPEX = 0 the emergency is not enabled, and this to allow correct execution of zero resetting.

(21) Axis X - wrong encoder connection
(22) Axis Y - wrong encoder connection
(23) Axis Z - wrong encoder connection
(24) AxisW - wrong encoder connection
(25) Axis V - wrong encoder connection

This error is displayed when, after stopping axes, a difference between main axis theoretical and real coordinates of more than 256 Encoder steps takes place. Usually, this happens when the Encoder phases are reversed. After this display, the reference pulses are reset to zero, and the axis is no more controlled.

(26) Axis X - not enabled
(27) Axis Y - not enabled
(28) Axis Z - not enabled
(29) AxisW - not enabled
(30) Axis V - not enabled

This error is displayed when entering a point-to-point motion data for a given axis, and the axis is not enabled for the required motion because it is currently under interpolation, coordinated motion, etc.

A message of axis not enabled is besides displayed after an error of
" WRONG ENCODER CONNECTION"

(31) Axis x - motion not ended
(32) Axis Y - motion not ended
(33) Axis z - motion not ended
(34) AxisW - motion not ended
(35) Axis V - motion not ended

This error is displayed at the end of a displacement if after 5 seconds from the end of the theoretical displacement the difference between axis theoretical and real coordinates exceeds the value indicated in Station Configuration window (Axes Parameters mode). This is usually due to a wrong setting of the Reference or Actuation Analogic Output offset. It can also be caused by some mechanical clearance on the axis or from an excessively low gain of the axis position ring.

(36) Axis X- servo error
(37) Axis Y - servo error
(38) Axis Z - servo error
(39) AxisW - servo error
(40) Axis V - servo error

This error is displayed during any kind of motion when the difference between axis theoretical and real coordinates exceeds 2047 Encoder steps in positive direction, and 2048 in negative direction. Normally, it is due to a wrong regulation of axis position ring gain or of bottom value of Actuation Speed Scale, or to an excessive axis inertia.

(41) Axis X - beyond positive limit
(42) Axis Y - beyond positive limit
(43) Axis Z - beyond positive limit
(44) AxisW - beyond positive limit
(45) Axis V - beyond positive limit

This error is displayed when the axis theoretical coordinates reaches or exceeds the maximum positive coordinates defined in Station Configuration (Axes Parameters mode).

(46) Axis X - beyond negative limit
(47) Axis Y - beyond negative limit
(48) Axis Z - beyond negative limit
(49) AxisW - beyond negative limit
(50) Axis V - beyond negative limit

This error is displayed when the axis theoretical coordinates reaches or exceeds the maximum negative coordinates defined in Station Configuration (Axes Parameters mode).

ERRORS CONCERNING MEMORY AREAS

(51) Function memory full
(52) Immediate programs memory full
(53) Parameters tables memory full

These errors are displayed when a.m. memories do not have enough space available to store all data received from PC. To solve out this disturbance, modify the number of assigned bits to desired memory in Station Configuration mode.

ERRORS CONCERNING CONFIGURATIONS

(54) Axes expansion card not found

This error is displayed when, during parameters transmission to station, at least one among Z, W and V axes has been configured without proper ESPAS card (axes expansion card).

(55) Interpolation card not found

This error is displayed when, during parameters transmission to station, HSINT card has not been equipped in Configuration.

(56) I/O serial module not found

This error is displayed when, during parameters transmission to station, remote flag has not been equipped in Configuration.

ERRORS CONCERNING INITIALISATION

(57) I/O serial module wrong communication

This error is displayed when, communication between PTP200N or PlC200 and I/O serial module is interrupted, or a breakage is found on remote reception card.
Cheque connection optic fibre and remote reception card feeder.

(81) card 1 parallel interface error
(82) card 2 parallel interface error
(83) card 3 parallel interface error
(84) card 4 parallel interface error
(85) card 5 parallel interface error

This error is displayed a wrong configuration of cards dressing jumpers or a breakage of the same parallel interface.

ERRORS CONCERNING PROGRAMS EXECUTION

(61) Function not found

This error is displayed when instruction FCALL is performed by loading a function not previously created.

(62) Function already running

This error takes place when instruction FCALL is performed in a function by loading another function at a lower insertion level. For example: main program is running a FCALL 10, which is running a FCALL 20. If the latter runs again a FCALL 10, the error is displayed.

(63) Too many functions inserted

This error is displayed when the number of function insertions exceeds 4.

(64) FRET instruction not loaded by FCALL

This error is displayed when a FRET instruction has been entered without first performing a FCALL instruction.

(65) Function parameters wrong
(66) Table of function parameters not found

These errors are displayed when some incongruency has been found between number of parameters bytes required by a given function and number of parameters bytes assigned to main program or to Table of Parameters; or also if a FCALL instruction of a Parameters function is performed by some other Function.

(67) Pointer of coordinates table not initialised
(68) Parameters values table wrong

This error is originated by a wrong use of Coordinates Tables. Usually they are displayed after selection of instructions based on tables not previously entered.

(69) Index of coordinate tables wrong

This error is displayed when using instructions imposing to a Coordinates Table pointer a higher value than the one defined by Tables Edit Mode.

(70) Too many subprograms

This error is displayed when the number of subprograms insertions exceeds 4.

(71) Instruction RET not loaded by CALL

This error is displayed when an instruction RET is entered without performing instruction CALL first.

(72) Operative code wrong

This error is displayed when performing an instruction whose operative code is not included in the ones known in GPL1000.TPA file (which is part of Operative System OS1000).

(73) Axis control wrong

This error is displayed when an error in axis control occurs, for instance if running a FREE instruction while the axis is in interpolation or in CHAIN mode.

(74) Axis corrector index wrong

This error is displayed when running an instruction referring to an axis corrector out of the range allowed.

(75) Synchro parameters wrong

This error is displayed when an instruction SYNC or WSYNC is performed by referring it to a station not configured.

(76) Too many repetitions entered

This error is displayed when the number of repetitions inserted (instruction REPEAT) exceed 4.

(77) Instruction ENDREP not loaded by REPEAT

This error is displayed when an instruction ENDREP is performed without performing the correspondent instruction REPEAT.

(79) Program not in directory

This error is displayed if a program has been called without being previously stored in the card.

APPENDIX B. PROCESS AND ERROR REPORTS

Two consulting files can be written by the system in the MS-DOS path indicated by environment variable DIRTMP. In any case this files will be compiled only if they have received a valid DOS name in Technological Parameters Setting. Their size will keep on increasing during machine functioning, and they must be periodically erased by the operator. This appendix is intended to describe their formats and contents.

Process file : File in which all data are enclosed between symbols < >, divided by a space. The first data, always existing, indicates date and hour of the beginning of a certain process, according with a fixed pattern:

<dd/mm/yy> <hh:mm:ss>

followed by data concerning the current process, data usually of two different type : command (**<START>, <STOP> or <END>**) and panel processing, which has the following format:

<dd/mm/yy> <hh:mm:ss> <AAA....> <PPPPPPPP CCC>

meaning, from left to right: Date (day/month/year) and Hour (hour:minutes:seconds) of process beginning; List name (size variable according to the name, max. 12 digits); Program name (fixed size: 12 digits justified on the left); Type of Process (3 digits justified on the left). If Processing Code has not been entered in its field, in that field shall appear three empty spaces.
If the list running did not have a name, in field List Names the name PLANCIA.LST will appear.
At the end of last program execution the following command is executed :

<gg/mm/aa> <oo:mm:ss>
<--->

Error File : file in which all data are enclosed between symbols < >, divided by a space. A first data, always displayed, indicates date and time in which took place machine errors stopping the machine waiting for operator instruction. The found error follows, on the next line, i.e. a message of different sizes enclosed between symbols < >. It is the exact copy of the error message displayed by the system when the error was found.

<dd/mm/yy> <hh:mm:ss>
<mmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmmm>

APPENDIX C. EXTERNAL PROGRAM

Select the item " External Program " on Main Menu to access to this operation mode, allowing the execution of an external program, in EXE or COM mode, running directly in MS-DOS environ.

An external program, called directly by CNC90, may be a custom procedure for generic utilities.

The external program is supplyed in CNC90 installation, on the files :

ESTERNO.EXE and ESTERNO2.COM
installed in the environ stated by the variable ROOT (normally on : root=C:\CNC90);

files ESTERNO.(lng) installed in the environ stated by the variable DIRLING (normally on : dirling=C:\CNC90\LINGUE).
These are the message files writed in national language: the extensions available are:

ITA	italian
ENG	english
FRA	french
DEU	deutch, etc.

When the program starts, a variable menu is presented, with the different selections availables.

For the selections on the menu, the availables keys are:

to place a cursor on the voice;
to confirme.

When confirmed, the selected program starts.
At the end, the control returns to the external main program, where it is possible :
restart another external program of the menu;
return to the CNC90 Console main program, selecting ESC key.

The configuration of External Program Menu is defined by the final part of the Messages file:

the first 16 messages of the file are reserved to the external program during execution (head messages, errors,..);
the message from position 17 are reserved for the menu configuration. The maximum number of messages is 100.

For exemple:

60"@%#&IMP.BAT%#&@Import of programs"
60"@%#$FORMAT A: /V%#$@Format Floppy Disk 1.44 Mbyte"
60"@%#$FORMAT A: /N:9 /T:80 /V%#$@Format Floppy Disk 720 Kbyte"
60"@%#$SALVAINA.BAT%#$@Update disk CUSTOM in A"
60"@%#$SALVAINC.BAT%#$@Copy disk CUSTOM in C"
60"@%#$PRINTDIR.EXE%#$@Print of the program Directory"
60"@%#%MAPMEM%#%@Mapmem"

where, on every line :

the head number at the head (60) states the maximum lenght of the characters string enter ("), excluding the characters @ : do not modify.

in this string:
"@%#$PRINTDIR.EXE%#$@Print of program Directory"

the main fields, in sequence, are :
defines the beginning of a substring non modifiables during translation in nationals languages. The @ character is optional.

substring of specials characters.
The first two are alwais %#; the third may be :

when the execution of the program PRINTDIR.EXE is completed, the control returns directly to the external program;

at execution completed, waits the confirme by key, before to return to the external program, as so to see the output corresponding to the execution of the program.

disables the corresponding voice in the menu: in this case the PRINTDIR.EXE program would not be available for execution.

is the command to run the program, as to be writed after the prompt of DOS.

repeats the special characters string.

defines the end of the substring not translable. The character @ is optional.

comment of the program PRINTDIR.EXE: it is the message displayed on the corresponding voice of the main menu. It is the only part of the string to be translable.

With the file of the exemple, the main menu will be presented so:

```
Format Floppy Disk 1.44 Mbyte
Format Floppy Disk 720 Kbyte
Updates disk CUSTOM in A
Copy disk CUSTOM in C
Print of the program Directory
Mapmem
```

The comment of the program "Import of programs" is not replied in the menu, because the special **&** character disables them.

In the exemple, selecting the line , the PRINTDIR.EXE runs in execution.

Selecting "Mapmem" :
program MAPMEM starts;
at the end, the message "Type any key to return to external program": in this mode is possible to see the output of this program before to return to external program menu.

All the other programs return directly to the main menu.

A generic program actived by external program runs as activec by operating system in the working directory of CNC90 (normally =C:\CNC90\).
If the executable lies in another directory, not comprised in the command PATH of the AUTOEXEC.BAT system file, all the command path must be specified in the command.

For exemple, in the case of:
"@%#$A:\STAMPA.EXE%#$@Auxiliary Print"

the STAMPA.EXE program is started in the drive A.

APPENDIX D. OPTIMIZER

In the case of machines with a single tool-head, a module for part program optimization is available.

This module works in background, at level of part program compiler, not in interactive mode.

The optimization procedure may be enabled or disabled by setting one of the "Flag n", in the Editor mode. It works in two successive phases:

the first phase matches the drilling operations, programmed by diameters, with a multiple tools selections, to reduce the total number of singles drilling phases. Drilling process programmed directly by tool selections are not modified.

the second phase schedules the sequences of movements of the tool head, ordering the drill operations as so minimize the total displacements for the program.

No optimization is made over the programmed measure process.

Tools selection and grouping

The process may be programmed by diameter are:
1) singles drills
2) repeat of drills
3) milling.

The maximum number of tool-spindles (at single or multiple tool) is selected to match the programmed holes layout, as so to minimize the operation pass and relatives head movements, with the following criteria:
a) if possible, the spindle with the maximum number of useful tools is selected, for obtain the maximum number of holes.
b) in any case, the prevalent criteria is to avoid or to reduce at the minimum the number of the drills worked two times, when the spindle has many tools.

D.2 Ottimizzatore

<u>Example :</u>
we consider to program 11 single drills, spaced 32 mm in X direction, with 8 mm of diameter.
Are available, with this diameter:
-> a spindle at 4 tools, at step 32mm
-> a single tool spindle.

The consequent execution will be:
-> the first 8 holes made by two pass with the 4 drill point;
-> the last three made by three pass with the single tool.

In the case of more possibility, the tool that minimize the displacement of the head is selected.

Repeat of drills are untied in single drills, to take back to the previous case.

The first milling tool found, in the order of number, with the programmed diameter, is selected.

In the case of programming of process by diameters, the selection of the tool considers always all technological parameters of the machine, to verify the relative compatibility with the programmed values:

1) first, the programmed diameter;
2) the number and the interaxis of the drill points;
3) the min and max rotation speed of the spindle;
4) the useful working area of the tools, in function of the programmed positions of the drills, over all the axes.

In the previous example, the optimization procedure checks:

1) the compatibility of the rotation speed on the drills;
2) the tools arrive to the programmed positions, either on the XY plane or on depth axis.

Other conditions are checked on grouping similar process (i.e. drillings):

1) exclusion field;
2) required depth;
3) slowing down in input and output (if programmed);
4) Spindle rotational speed and sense.

In the example, we need for the grouping that:

1) all the drills must have the same depth;
2) all the drills must be programmed with the same exclusion code or without;
3) all the drills must have the same slowing down parameters;
4) for all, the same speed and sense of rotation.

Grouping of many drilling process may cause :

1) the programmed process are executed by a single spindle (Example : 4 drills with interaxis 32 mm and 8 mm of diameter may be processed by a single spindle with 4 drill points, with the same interaxis and diameter);

2) the programmed process are executed in a single pass, with many tool-spindles

During the grouping, the quotes are not discriminated for differences less than the encoder resolution.
Example :
with resolution on the axis X = 0.05 mm,
the points P1(X=100; Y=200) and P2(X=100.04; Y=200) are considered coincident.

Process Ordering

When the drill and mill groupings are stated, the optimization procedure checks all head positions to found the sequence of the displacements minimizing the total travel of the head.

The ordering phase works only over the drill process (doesn't matter if programmed by diameter or direct tool): all other process (millings, saws,..) are executed by last in order of programming.

APPENDIX E. PROGRAMS EDITOR CONFIGURATION

PROGRAM : PACKTIF

All graphic help windows performed by Editor-CNC90 program can be personalised. A set base of graphic representations is supplied during control installation and possible modification or/and new implementations can be agreed with TPA or executed directly by CNC90 system user.

Graphic windows that can be modified or implemented are the ones concerning the following commands:

Alt,G description windows about programmable workings
Alt,T description windows about tool groups and machine tooling
Alt,S description windows about notable subprograms

Each graphic window that can be loaded corresponds to a source file in **TIF** or **BMP** format.
They are two very common formats of recording image file.
In particular : BMP format is a standard in Windows 1 environment.
Numerous graphic programs are available on sale for the two formats processing. These programs allow to create drawings by using mouse eventually combined with keyboard or alternative to it.

Drawings should fulfil the following image characters :

1) drawing in black and white
2) max dimensions correspond to 590 pixel x and 245 pixel y window
3) recording without applied compression
4) drawings names should be assigned in an established way (see later)

If requested, T.P.A supplies source drawings corresponding to graphic windows in phase of installation : drawings are available in BMP format.

As already said source drawings should have notable names. In particular:

a) drawings describing programmable workings should have **MASKnnmm** name , with:
nn = two figures number ranging from "00" to "29"
mm = two figures number ranging from "00" to "29"

Here below correspondence between workings and names is displayed:

Drawing name	Working
MASK0100	Rapid (x,y,z) H
MASK0200	Rapid (x,y,z,w,v) H
MASK0300	Drill in Cartesian coordinates
MASK0301	Drill in Polar coordinates
MASK0302	Fitting x
MASK0303	Fitting y
MASK0304	Drill repeat in x
MASK0305	Drill repeat in y
MASK0306	Drill repeat in xy, cartesian coordinates
MASK0307	Drill repeat in xy, polar coordinates
MASK0308	Drill on circle
MASK0100	Blade x
MASK0100	Blade y
MASK0100	Milling set-up in Cartesian coordinates
MASK0100	Milling set-up in polar coordinates
MASK0100	Milling L1
MASK0100	Milling L2
MASK0100	Milling L 3
MASK0100	Milling L 4
MASK0100	Milling C2
MASK0100	Milling C2
MASK0100	Milling C3
MASK0100	Milling C4
MASK0100	Milling Elic -C1
MASK0100	Milling Elic-C2
MASK0100	Milling Elic-C3
MASK0100	Oval
MASK0100	Grooving
MASK0100	Connection
MASK0100	Milling C5
MASK0100	Measure L
MASK0100	Measure H
MASK0100	Measure S
MASK0100	Offset
MASK0100	Blade on A

MASK800	Plate insertion
MASK801	Bushing insertion
MASK80	Hinge insertion
MASK80	Shelf bearing insertion
MASK80	Generic 1 insertion
MASK80	Generic 2 insertion
MASK80	Pin insertion
MASK80	Subprogram
MASK80	Drill with discharge
MASK80	Tapping

b) Description drawings of tooling groups, on different machine tooling should have HEADnnnn mane, with :
nnnn = four figures number ranging from "0000" to "0049"

.

Correspondence between tooling and names is :

HEAD0000 Head drawings on tooling 0
HEAD0001 Head drawings on tooling 1
.
.
HEAD0049 Head drawings on tooling 49.

c) Description drawings of subprograms should have SUBnnnn mane, with :
nnnn = four figures number ranging from "0000" to "0999"

Correspondence between subprograms and names is :C

SUB0000 Subprogram drawing "000"
SUB0001 Subprogram drawing "001"
.
.
SUB0999 Subprogram drawing "999"

PACKTIF program executes the following functions :

1) it can read files in TIF or BMP format and translate them into one single format, defined by T.P.A. detailed list, named ZIF format.
Files in ZIF format have smaller dimensions than original format, either TIF or BMP. Besides : files in original format, have all the same dimension, corresponding files in ZIF format have on the contrary variable dimensions.

2) they can create a single file for each typology of graphic windows, executing a "linker" operation of single programs in ZIF format.
These files, obtained by the "linker" operation, are used by Editor-CNC90 program and therefore they should be configured in CNC90 working environment.

Name

MASKLNK	description file of programmable workings
HEADLNK	description file of machine toolings
SUBLNK	description file of subprograms

Extension

it is an extension of national language, as for what concerns messages; that is :

ITA	Italian language
ENG	English language
FRA	French language
DEU	German language
FLM	Flemish language

with the possibility of extension of any other language user will introduce in the control.
It may be necessary to define the national language extension only in the case in which graphic windows display words that can be translated. This generally occurs with MASKLNK file, but it may occur also with tooling or subprograms files : in this case it is better not to define any extension, in order to have one single file valid for all selectable languages, instead of one file for each language.

Let's examine PACKTIF program working in detail.
Program can be started either by operating system or in shell by MS-DOS system.
Menu displays two items :

Compression on ZIF file Linker of ZIF files

Selecting one of the two items the system will create a file or will execute a linker of files in ZIF format.

Compression on ZIF file

This menu is displayed :

Compression on ZIF file	
Source path ....:	**C:\BMPCNC90**
Source extension .:	**BMP**
File/files name ......:	**MASK***
Destination path ...:	**C:\ZIFCNC90**

Source path : defines files research path in original format (TIF or BMP)

Source extension : defines extension in original format (TIF or BMP)

File/files name : defines original file or files name. If the name is assigned full, compression is executed on one single file. If the name is assigned with jolly character "*", compression is executed on all the files with the name assigned on the defined matrix. In the example all the files named with "MASK" are involved in the operation

Destination path : defines the path for files writing in ZIF format.

Press ENTER to confirm.

In case of selections displayed in the menu :

in C:\BMPCNC90 all files with "BMP" extension and with the initial part of the name corresponding to "MASK" are found and in C:\ZIFCNC90 corresponding files in ZIF format are recorded. Files translated into ZIF format keep their original name while extension is now defined on "ZIF".

Linker of ZIF files

Selection menu on linker files is displayed above :

Linker
MASK
SUB
HEAD
New linker

Menu here displayed corresponds to the setting read on a configuration file of PACKTIF.EXE program : it is PACKTIF.PAR file, installed in the machine parameters directory (directory set on environment variable: USER).
Items correspond to already described linker : main working parameters are already defined for these files and they will be proposed on following menu
At last "New linker" selection is proposed : it can be used for future development configuration.

After selecting one of menu items, the following menu is the displayed :

ZIF files linker	
Source path ....:	**C:\ZIFCNC90**
File/files name ......:	**MASK***
Destination path ...:	**C:\CNC90\MOD1\USER**
Linker file name .:	**MASKLNK**
Language............:	**ITA**
Line number ....:	**30**
Column number ..:	**30**
Update (Y/N) ? .....:	**N**

Source path : define research path of files in ZIF format;

File/files name : define original file or files name. If the name is assigned full, compression is executed on one single file. If the name is assigned with jolly character "*", compression is executed on all the files with the name assigned on the defined matrix. In the example all the files named with "MASK" are involved in the operation.
Selecting MASKLNK linker, field is initialised on: MASK*.

Destination path : define the path for linker file writing. Field is initialised on environment variable : USER.

Linker file name : select linker file name

Language : select national language abbreviation, or leave the field void, in case of extension not necessary. Field is initialised void;

Lines number

Column number	these values are defined :
lines = 30, column = 30	for MASKLNK linker file;
lines = 50, column = 1	for HEADLNK linker file;
lines = 1000, column =1	for SUBLNK linker file.

Update (Y/N) ? : select “Y” if linker file must be updated, only on defined ZIF files; select "N" if linker file must be created on defined ZIF files. Field is initialised on "N".

Press ENTER to confirm selections.

In case of selections displayed in the menu :

in C:\ZIFCNC90 all files with "ZIF" extension and with the initial part of the name corresponding to "MASK" are found and in C:\CNC90\MOD1\USER the file MASKLNK is recorded. MASKLNK file is recorded with "ITA" extension and it is copletely created.

If a destination path is selected different than parameter directory, linker file must be copied in CNC90 environment, in the directory specified by USER environment variable : generally it is : USER=C:\CNC90\MOD1\USER.

PACKTIF.EXE program installation

PACKTIF.EXE program uses messages file, that can be translated into national language, it is named PACKTIF.LNG.
packtif program installation is executed when installing CNC90 programs :

a) PACKTIF.EXE program is installed in C:\CNC90 (main directory of CNC90 programs, set on environment variable :ROOT)

b) language files are installed in C:\CNC90\LANGUAGES (set directory on environment variable : DIRLING) : PACKTIF.LNG is initialised on Italian language messages

c) configuration file installed in C:\CNC90\MOD1\USER (parameters directory, set on environment variable: USER) : file name is PACKTIF.PAR.

APPENDIX F. IMPORT PROGRAM OF ISO TEXT

Inport program of ISO text is a special tool, which is supplied apart from CNC90 Installation disks.

Import program installation is realised on the following files :

IMPORT.EXE software file
IMPORT.LNG messages file in national language.

Software file IMPORT.EXE is generally installed in CNC90 working directory, defined by environment ROOT variable.
Message file IMPORT.LNG should be installed in CNC90 message directory, defined by environment DIRLING variable (generally is : DIRLING=C:\CNC90\LINGUE).

Software file is generally started :

by "External Program" available in CNC90 menu (see appendix C). Batch file IMP.BAT is available for such process, installed in CNC90 system directory.

directly by operating system.

IMPORT.EXE program **can't** be started in shell to DOS.

IMPORT.EXE program can be started with or without argument. In the first case when the process is ended control returns to operating system.
In the second case, argument should input return modes, on a complete path name. Assume following starting example :
IMPORT E :\UTILI\TOOL.EXE <--'

Exit mode is executed as follows :
a) eventual disk change, if different from current one (in the example : E:)
b) eventual directory change, if different from current one (in the example : E:\UTILI)
c) TOOL.EXE procedure starting Import program can execute two types of operations :
files import in ISO format, available on any disk unity or
directory, in CNC90 files;
programs erasure from CNC90 files.

Import program working modes are assigned to a commands file : IMPORT.CMD.
Commands file should be copied in the directory where IMPORT.EXE software file is installed.

IMPORT.EXE program is able to work if the following conditions are fulfilled :

IMPORT.CMD command file existing;
IMPORT.LNG message file existing (found on environ DIRLING);
full setting of CNC90 environment variables;
NAVETTA.CMD CNC90 command file existing, (found on environ ROOT).

COMMAND FILE STRUCTURE : IMPORT.CMD

IMPORT.CMD file is in ASCII format, with established fields length.
Each file record :
- is headed by '<' character
- is ended by characters sequence : '>' + CR + LF.

First 6 records head the file : define working general modes, without assigning executing files.

Records numbered from 7° till file end are definition records of operating command : either import or erasure program.

1° RECORD

<78 CHAR>

Program import directory of ISO source.
If all spaces are defined : current directory is considered valid.
If defined, directory must exist.
Setting example :

```
<A:\EXAMPLE                                                              >
```

2° RECORD

<78 CHAR>

Writing directory of IMPORT.ERR error file.
If all spaces are defined : current directory is considered valid.
If defined, directory must exist.
Setting example :

```
<A:\EXAMPLE\ERRORS                                                       >
```

3° RECORD

<4 CHAR> (<RRCC>)

RR = messages display line
CC = messages display column
RR and CC should be defined by two characters each.
RR valid values : from "01" to "25";
CC valid values : from "01" to "80".

If RR=" " (spaces) : IMPORT.EXE program does not execute displays.

Setting examples :
<1004> define : RR=10, CC=4
< 00> define : display not present.

4° RECORD

<4 CHAR> (<C1C2>)

C1 and C2 are significant only if 3° record defines display line and column.
C1 = blackground colour
C2 = text colour.

C1 and C2 should be defined by two figures each, with values valid from "00" to "15".
In case of C1 and C2 value equal or not correct, default value are assumed : C1="00" (black) and C2="07' (white), with white writings on black background display.

Setting examples :
<1201> define : C1=12 (colour : light red), C2=1 (colour : blue).
<1212> define C1=C2=12 : in this case default values are set C1="00" e C2="07".

5° RECORD

<20 CHAR> :

1° CHAR	Defines video mode set when calling import program : 0 = video text mode on 80 columns 1 = graphic mode VGA.
2° CHAR	Defines if mother programs should be found in an external directory or in CNC90 files : 0 = external mother program 1 = mother program in CNC90 file In case of program from external directory: record 1 imposes research directory. In case of program from CNC90 file : research takes place according to environment variable DIRSOR set.
3° CHAR	Defines working mode of Editor-CNC90 program : 0 = Editor-CNC90 non interactive 1 = Editor-CNC90 interactive Non interactive editor setting allows to start automatic setting procedure. Interactive editor setting is available for import procedures tests.
4° CHAR	Characters font selection for VGA video mode : 0 = default Font selection 1 = Font 1-TPA 3 = Font 3-TPA
5°- 20° CHAR	Reserved for future developments. Characters from 5° to 20° must be defined by space character, or figures from "0" to "9".

6° RECORD

<3 CHAR>

extension of all mother programs. Extension is significant only in case of mother programs loading from external directory.

Setting examples :

<SRG> all mother programs should have extension "SRG".

< > mother programs have no extension

from 7° to'N RECORD

<NNN PrgSONS PrgMATHER DimL DimH DimS Comment Command/answer>

field	description	format
NNN	progressive number of command record : "001" on first command record "002" on second command record and so on.	3 CHAR
PrgSON	program name from : a) enter CNC90 file (if import command) b) delete from CNC90 file (if erasure command)	12 CHAR
PrgMOTHER	ISO program name to be defined	12 CHAR
DimL	significant only on import command : defines son program length dimension. Related Unity measure : [mm] or [inch], as programmed on mother program. Non significant if defined by spaces.	8 CHAR 4.3 or 5.2
DimH	significant only on import command : defines eight dimension of son program. Related Unity of measure : [mm] or [inch], as programmed in mother program. Non significant if defined by spaces.	8 CHAR 4.3 or 5.2
DimS	significant only on import command : defines thickness dimension of son program. Related Unit of measure : [mm] or [inch], as programmed on mother program. Non significant if defined by spaces.	8 CHAR 4.3 or 5.2
Comment	significant only on import command : defines son program comment. Non significant if defined by spaces.	25 CHAR
Command **Answer**	character of command definition or response of command execution: C = import command G = import command + compiling K = erasure command c = correct setting g = correct setting + compiling k = correct erasure	1 CHAR

IMPORT.CMD file EXAMPLE

```
<A:\EXAMPLE                                                              >
<C:\CNC90\MOD1\TEMPOR                                                    >
<1010>
<0015>
<1000     >
<SRG>
<001PROOF                                                               K>
<002                NOME       1000.0001000.000 30.000   IMPORTATION n.1  C>
<003PR1             NOME       1000.000800.000           IMPORTATION n.2  G>
```

IMPORT.EXE PROGRAM PROCESSING

IMPORT.CMD file should be in the same starting directory of import program.
Suppose IMPORT.EXE is installed in C:\CNC90 directory: command program too should be in C:\CNC90.
Errors conditions found in this first stage cause :

a) video setting in text way on 80 columns;
b) warning message display;
c) import program exit.

Possible error conditions are :

(1) ERROR : environ ... not found
environment variables used by CNC90 program have not been set.

(2) ERROR : TASK 2 IMPORT procedure can't be started
procedure loaded on CNC90 secondary task, when a procedure is already running on main task.

(3) TASK 1 IMPORT procedure can't be started
Procedure loaded on CNC90 main task, when a procedure is already running on secondary task.

(4) IMPORT.LNG file not found
IMPORT.LNG message file can't be read. Remember that messages file is in the directory of DIRLING environment variable
(normally : DIRLING=C:\CNC90\LINGUE).

(5) NAVETTA file backup not allowed
(7) NAVETTA file backup not allowed
(6) NAVETTA file not found
CNC90 navetta file can't be read and/or written. Navetta file is recorded on working disk when starting CNC90 and deleted when quitting operating system.

(9) IMPORT.CMD file can't be loaded
IMPORT.CMD command file can't be read. Remember that command file is recorded in the directory where IMPORT.EXE software program is installed: generally it is in CNC90 working directory.

first 6 heading records are read by command file : in case of non significant data processing is aborted and a warning message is displayed.
Error conditions found in this first stage cause :
a) video setting in text way on 80 columns;
b) warning message display;
c) import program exit.

Possible error conditions are :

(21) Syntax ERROR record n° ...
Format error of displayed record. Message can correspond to :
a) bad record length;
b) bad heading ("<") or record exit (">") character;
c) bad format in record 1 or 2 field , respectively : mother program and errors file recording directories.

(23) ERROR : directory of reading out programs not found
Selected directory does not exist. Error refers to check of record number 1.

(24) ERROR : deposit directory for errors files not found
Selected directory does not exist. Error refers to check of record number 2.

(25)line and/or column ERROR : wrong visualisation
Bad value in defined line and/or column for displaying. Error refers to check of record number 3.

(26) Colour ERROR : bad background and/or text colour
Bad value in selected colour for displaying. Error refers to check of record number 4.

(27) Not significant value/s ERROR record n° ...
Error refers to check of displayed record.
On record number 5, message may mean :
a) 1° character defined different from "0" and "1"
b) 2° character defined different from "0" and "1"
c) 3° character defined different from "0" and "1"
d) 4° character defined different from "0" and "1" and "3".

(29) ERROR : video mode can't be set different than starting one
Error refers to record number 5 checking and means a discordance between video mode setting and setting defined on the 1° character of the field introduced in the record.

(28) ERROR : IMPORT.CMD file not full
IMPORT.CMD file is defined by a records number smaller than 6.

reading of first command record (record number 7) is started:
Command is interpretated and the corresponding procedure is started. Commands can be three :

command C
requires mother program import, with eventual reintroduction of :
name (field data : PrgSUN)
dimension/s (fields data : DimL, DimH, DimS)
comment (field data : Comment).

Each reintroduction can be input in a independent way..
PrgMOTHER program import in PrgSUN is done calling Editor-CNC90 program, with interactive or non interactive mode as defined in record 5.

Import procedure requires PrgSUN introduction in CNC90 program file, updating programs directory file. In particular : PrgSUN program is not compiled. Used CNC90 environment should be defined on set environment variables.
Import procedure may cause error situations; in general, the following errors can be found: ISO text format, ISO text incongruency with machine parameters, working disk entering.

command G
requires mother program import on PrgSON, with the same modes examined for command C : PrgSON is also compiled.

command K
requires erasure of PrgSUN program found in CNC90 file..

when the command is ended as explained in preceding point 3, command character written in current record :

a) is substituted with correspondent character of small letter, if import/erasure has not found error situation;
b) on the contrary it is not changed.

Procedure checks following record, and starts again processing.

Procedure goes on in this way untill defined commands on IMPORT.CMD file are fulfilled.

When reading and processing commands listed in IMPORT.CMD file current operating mode and warning messages are displayed only if IMPORT.CMD file allocates a video position messages management (record number 3).
On the contrary video messages are not displayed.

Warning messages display lasts 2 seconds and a beep is heard : during visualisation, current processing can be interrupted; press ESC to abort IMPORT.EXE program processing.

In case of imports/erasure programs procedures wrong, such conditions are stored on an error file. The file is IMPORT.ERR and it is recorded on the second record of command file directory. In the above example : C:\CNC90\MOD1\TEMPOR.
Error file format is explained in the following paragraph.

Assume again the preceding example on IMPORT.CMD command file.

```
<A:\EXAMPLE                                                            >
<C:\CNC90\MOD1\TEMPOR                                                  >
<1010>
<0015>
<1000     >
<SRG>
<001PROOF                                                             K>
<002              NOME       1000.0001000.000 30.000   IMPORTATION n.1 C>
<003PR1           NOME       1000.000800.000           IMPORTAZION n.2 G>
```

Suppose to start IMPORT.EXE program from CNC90 working directory, located on environ **ROOT=C:\CNC90**.

IMPORT.CMD command file should be available in the directory : **C:\CNC90**.

IMPORT.CMD file is defined by 9 record : the first 6 for heading and the following 3 for commands.
Directory for IMPORT.ERR error file recording is : **C:\CNC90\TEMPOR**.
Mother program allocation are :
a) read out from external tracks;
b) reading directory : **A:\EXAMPLE**;
c) common extension : **SRG**.

When IMPORT.EXE program is started video is set in VGA graphic mode.
Display are addressed to line 10 and column 10, with colour setting : black background (colour : 0) and light white writings (colour : 15).

Editor-CNC90 program is called in not interactive mode.

Assume now 3 commands record, each displayed here below.

```
<001PROOF                                                             K>
```

it is an erasure command on PROOF command, found in CNC90 files.

```
<002              NOME       1000.0001000.000 30.000   IMPORT n.1      C>
```

it is an import command NAME mother program. Mother program is read as follows :
A:\EXAMPLE\NAME.SRG

Son program is loaded with the same name : NAME.
Dimensions defined for son program are :
  length =1000 mm
  height =1000 mm
  thickness =30 mm.
Comment assigned to son program is : "IMPORT n. 1".

<003PR1 NAME 1000000800.000 IMPORT n.2 G>

it is an import and compiling command on NAME mother program. Mother program is read :
A:\EXAMPLE\NAME.SRG

Son program is imported with name : PR1.
Dimensions defined for son program are :
  length =1000 mm
  height =1000 mm
  thickness =same as previous one
Comment assigned to son program is : "IMPORT n. 2".

DIAGNOSTIC MESSAGES WHEN READING AND EXECUTING A COMMAND

Error situations that can be found, when reading or executing a command, are here examined, as defined on IMPORT.CMD file.
Errors display is listed here after : indicating error number and message.

(50) Syntax error
Bad format in command record read by IMPORT.CMD file. Possible errors are: unvalid record heading ("<") or losing (">") character.

(51)Bad numeric expressions
Possible errors :
b) field NNN non numeric;
c) piece dimension fields (DimL, DimH, DimS) non numeric and non defined with spaces.

(52) Bad record length
Record is defined by a bad characters number.

(53) Bad command field
Bad character in command field.

(54) Processing cannot be performed
An error is found in import or erasure processing : it is an error checked in Editor or CNC90 compiler mode.

When calling Editor-CNC90 module in non interactive mode, diagnostic error is recorded on CNC90.ERR error file, with queuing mode of following diagnostic messages.
CNC90.ERR file is recorded in directory defined by DIRTMP environment variable.

Diagnostic situations may correspond to :
a) bad working disk access;
b) bad ISO text format;
c) bad geometric or technological expressions, in ISO format programming;
d) bad program compiling..

(55)NAVETTA file not found or not modifiable
Displays an error situation in CNC90 navetta file :
a) file not found, or
b) disk error when reading and/or writing.

(56) Communication file cannot be opened
An error is checked when recording an auxiliary file on working disk.
Current file is named AUX2 and it is recorded in case of import command.

(57 IMPORT.CMD file not found or not modifiable
Displays an error situation in IMPORT.CMD command file :
a) file not found, or
b) disk error when reading and/or writing.

(58) Program name not specified
Displays that mother program name has not been defined in import command.
Displays that son program name has not been defined in erasure command.

(59) Mother program not found
Displays mother program not found with import command.

(60) Bad IMPORT.CMD file structure
Command record read by IMPORT.CMD file is defined by an empty string.

(61) Non progressive record number
Record NNN field is not progressive with reference to current number of command record.

(62) Records number exceeding 200
IMPORT.CMD file is defined by a number of commands record greater than 200. Import program processing is quitted by this message.

ERROR FILE STRUCTURE : IMPORT.ERR

IMPORT.ERR file is in ASCII format, with established length fields.
Each file record ends with the following sequence of characters : CR + LF, and has the following format :

<NNN ERROR description of COMMAND code>

field	description	format
NNN	record progressive number (corresponds to NNN field in IMPORT.CMD commands file record)	3 CHAR
ERROR description	message explaining error : displays error number and a part of the description	50 CHAR
COMMAND code	command cede, as defined in IMPORT.CMD command file	1 CHAR

Current records number is defined by errors number found.

APPENDIX G. BARCODE MANAGEMENT

BARCODE MINIMUM REQUIREMENTS

BARCODE must work emulating keyboard
must find ALFA 39 code or 3 of 39
should not have any character of preface and notes.
labels must be printed in ALFA 39 cede or 3 of 9.

LABEL SET POINT PROCEDURE.

To execute SET POINT procedure read label with BARCODE

$SETP$

when reading is ended machine will start processing SET POINT.

SIMULATION LABEL OF FUNCTION KEY F6 (END).

-

When reading this label SET POINT or program list processing may be quitted. Label is accomplished by only one character (minus sign) to assure a speed and sure reading.

LABELS FOR AUTOMATIC LISTS WITH ONLY ONE BOARD (STANDARD MANAGEMENT).

Label format:

%NAME.NUM REPETITIONS.WORKING TYPE.COD INPUT.COD OUTPUT$

Character **%** (per cent) means data editing start from BARCODE ; character **.** (point) separates the different fields (see name, working type, input and output codes, ecc..) of the list; at the end character **$** (dollar) means data editing end from BARCODE, that is the same as

to activate START from the list containing only the board just entered.

Es. 1
Create label for PIPPO board repeated twice in working S.

%PIPPO.2.S..$

that is the same as

%PIPPO.2.S$

Es. 2
Create label for PLUTO board repeated 59 times in working M with input code 123 and output code 456.

%PLUTO.59.M.123.456$

REMARK:
This mode allows to queue new programs to processing list. This is possible only if the machine is executing last program of last repetition of the list.

LABELS FOR AUTOMATIC LISTS WITH SEVERAL BOARDS (STANDARD MANAGEMENT).

Labels format :

%NAME1.NUM REPETITIONS.WORKING TYPE.COD INP.COD OUT+

NAME2.NUM REPETITIONS.WORKING TYPE.COD INP.COD OUT+

" "
" "
" "

NAMEN1.NUM REPETITIONS.WORKING TYPE.COD INP.COD OUT+

NAMEN.NUM REPETITIONS.WORKING TYPE.COD INP.COD OUT$

As already said characters **%** (per cent) and **$** (dollar) represent start and end editing data from BARCODE; character **.** (point) is used to separate different fields of each labels; in these labels a new character **+** (plus sign) is introduced, that means current label end and following label start.

Es. 3
Create labels to execute list :

Name	Num	Typ	Inp	Out
PIPPO	1	N	001	003
PLUTO	10	S/	05	00
LINO	100	M		004

First board label

%PIPPO.1.N.001.003+

second board label

PLUTO.10.S/.05.00+

third board label

LINO.100.M..004$

If on the contrary two labels are created, one containing character **%** and the other character **$**, that will be read in case of start and end data editing, three preceding labels can be create in this way :

first board panel

PIPPO.1.N.001.003+

second board panel

PLUTO.10.S/.05.00+

third board panel

LINO.100.M..004

or

LINO.100.M..004+

Let's see now how to create the label in the example 1

%PIPPO.2.S..+$

or

%PIPPO.2.S+$

or reading characters **%** and **$** from an external environment.

PIPPO.2.S

Remark:

All levels can be separated in several pieces without adding characters (end, preface) that can cause a bad label reading.

L A B E L S F O R A U T O M A T I C L I S T S

Advanced management of BARCODE allows to use a support file to define those fields, redundant or expected during working planning (label print), found in automatic list.

This means that Advanced management labels will compute only main information to identify the program, that is only the name; all the other information, defined a priori and that are necessary for programs executing are read by control in a file written by the user.

Notice that information about working field or type can depend on a free area on the machine, and therefore not presumable a priori.

Let's see now the format of this particular label :

+NAME PROGRAM$

SUPPORT FILE (BARCODE)
This file is a simple list file (therefore editable by CNC90 LIST EDITOR) named BARCODE.

All necessary information for program executing are contained in the file; therefore the name (for identification), repetitions number, working type and input and output codes.

Let's see all the operations that user should perform to work with this king of BARCODE management.

First BARCODE list must be fully compiled by user (this may be done also once).
Afterwards BARCODE will read a label containing program name to be executed with the same format above described; at this point control will read BARCODE list and will look, in mane field, for program name written in the label; once found it will read following fields and load on automatic list to be executed.
At this point program will start with all information read by BARCODE support file.

As for Standard management of a single panel, also in this case several panels can be put in line to execution list, only if last panel and last repetition have been executed.

This management is correct only if working type can be defined a priori. Very often working code can't be presumed a priori, and therefore it can't be written in an established way in BARCODE support file. In these cases user working on the machine and that knows on which field or area a certain piece can work (availability or breakdown), must insert this information in the control.

BARCODE list should be compiled in a particular way so that machine user can insert working code, that is to define which working codes of one or more programs are not established but should be input by user upon demand during processing.
This can be obtained assigning value **?** (question mark) to **Typ** field of concerned programs in BARCODE list, only on those programs requiring manual insertion of working code.