NUM 760 T

Presentation Manual

En No. 938560/A

ED. 04-83

www.num.com

CNC HighEnd Applications

NOTE TO THE USER

This manual is written according to the standards wich are referred to in it and within the framework of the design requirements imposed by the NUM-SA systems.

However, NUM-SA reserves the right to modify all or part of this manual as they see fit in the light of current standards and of technical developments.

CONTENTS

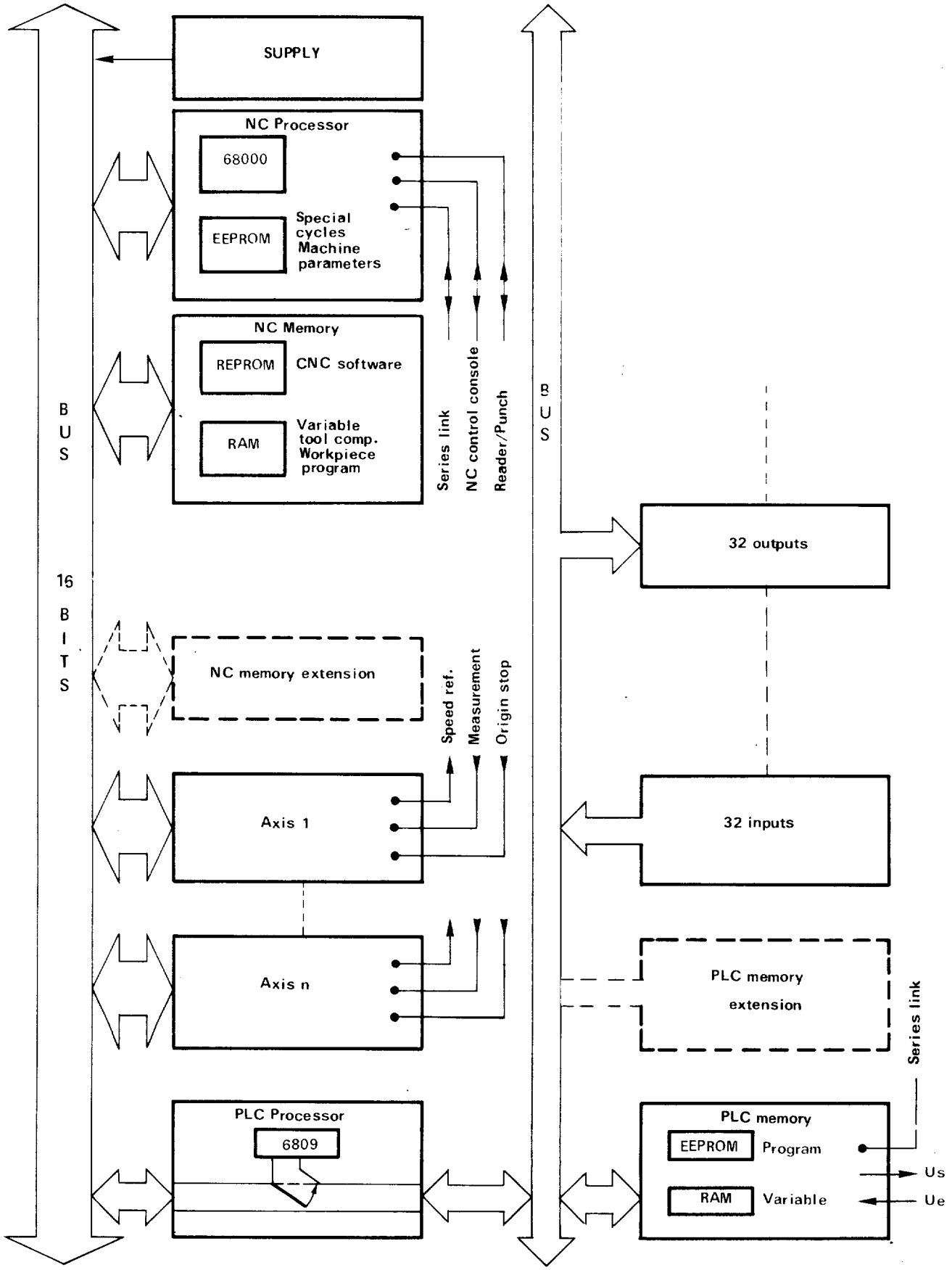
1 - INTRODUCTION

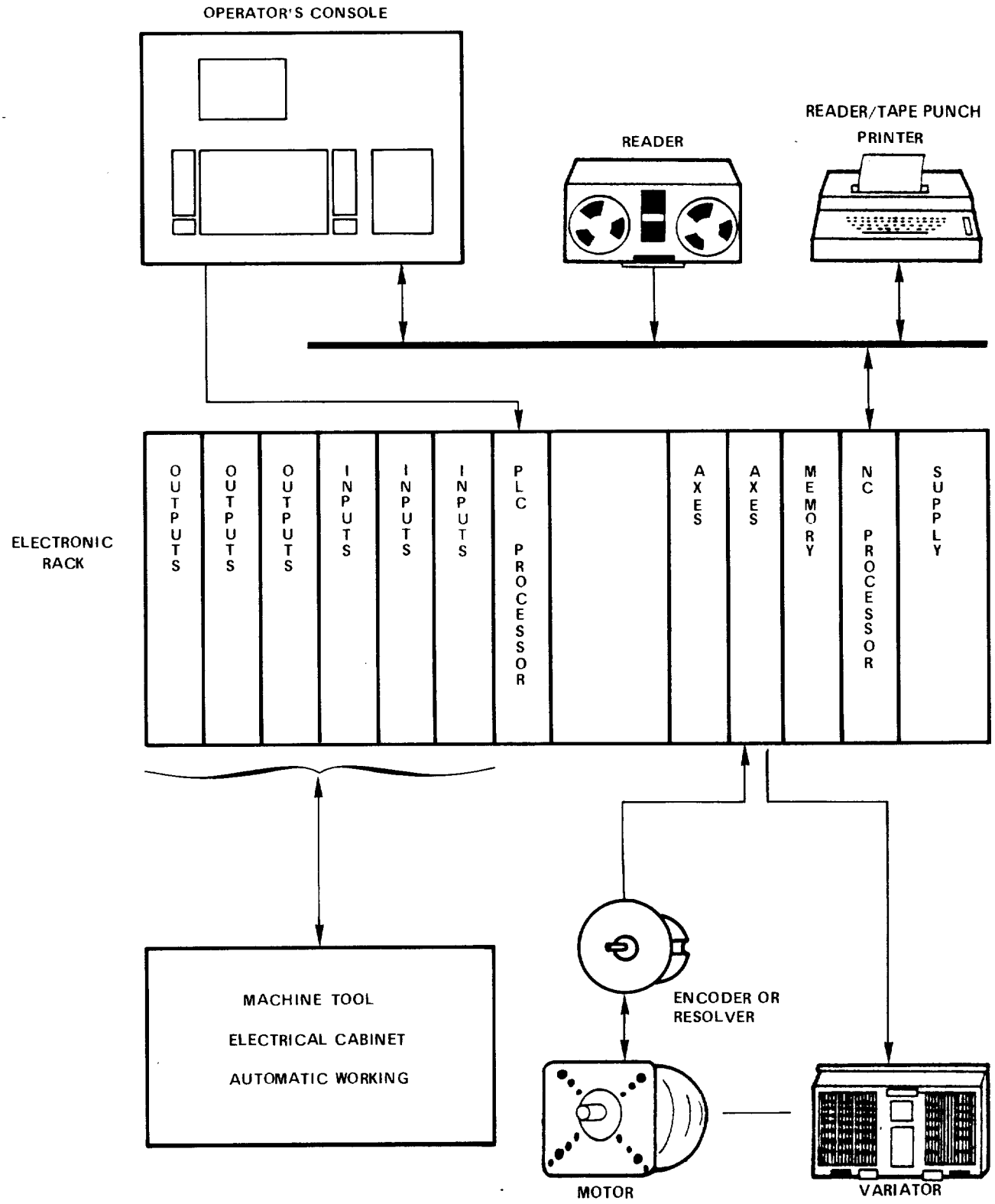
The NUM 760 T is a numerical control designed for use with the greater part of the lathe market, form tool room lathes, due to its simple programming language, to machining centres or flexible machine shops due to its capacity for data exchange with an external source.

The design of this CNC, which makes use of the most advanced LSI components and 8 and 16/32 bit microprocessors, is modular with regard to both the hardware and the software. It is designed to be able to receive those developments which the market requires.

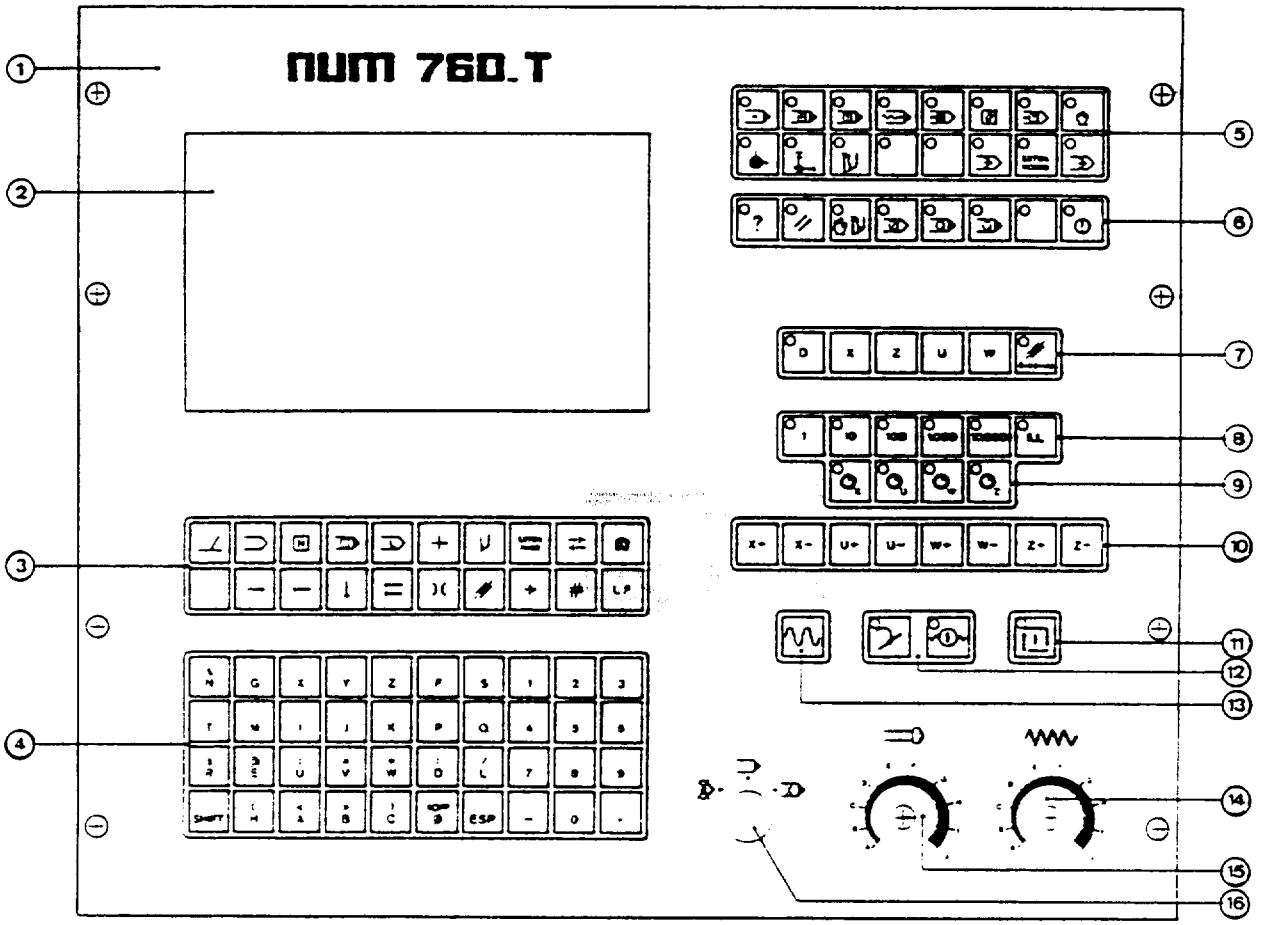
An incorporated high performance PLC makes possible the adaptation of the numerical control to the functions specific to the machine without altering the basic system functions. A parameterized programming system and those of the numerical control which make possible the automization of functions such as tool compensation, origin offsets and dimension acquisition thus rendering the system capable of being integrated into an automated production process.

2 - GENERAL ORGANISATION




OPERATOR'S CONSOLE
READER
READER/TAPE PUNCH
PRINTER
ELECTRONIC
RACK
OUTPUTS
OUTPUTS
OUTPUTS
INPUTS
INPUTS
INPUTS
PLC PROCESSOR
AXES
AXES
MEMORY
NC PROCESSOR
SUPPLY
MACHINE TOOL
ELECTRICAL CABINET
AUTOMATIC WORKING
ENCODER OR
RESOLVER
MOTOR
VARIATOR

3 - CONTROL CONSOLE



1 Console
2 12 line, 40 character display
3 Choice of page and cursor control keyboard
4 Alphanumerical keyboard
5 Mode selection keyboard
6 Various functions keyboard and signal lamps
7 Dynamic compensation for tool wear
8 Selection of increment for manual movement
9 Selection of axis controlled by electronic handwheel
10 Control keys, axis on manual
11 Cycle start key
12 Stop machining and axis recall key
13 Rapid key, in manual
14 Feed rate potentiometer 0-120 %
15 Spindle potentiometer 50-100 %
16 Console lock key

The operator makes use of the NUM 760 CNC by means of the control and information displayed on the numerical control console and the controls on the machine console.

The NC console with its sensitive keys has a buzzer indicating the activation of one of the keys.

3.1 - DISPLAY SCREEN

Two types of display unit are available :

- a 9 inch cathode ray tube screen
- a plasma display screen

Display of 12 lines of 40 characters.

Depending upon the page selected the following data can be displayed :

- the workpiece program in the memory, the whole program being obtained by repeated action on the "liste" (list) key,
- the workpiece program in progress, the cursor indicates the block in progress,
- the modal functions ant the complete composition of the block in progress as well as the modulations of spindle speed and feed rate,
- the values of the machining parameters,
- the current point in relation to the workpiece or machine origin, the distance remaining to be covered, the following error and the value of the workpiece origin offset,
- the tool gauge values and the wear compensation values,
- Inputs/outputs and internal P L C variables,
- Machine and system faults,
- Messages to the operator,
- Messages necessary for maintenance or integration,
- Entries through the alphanumeric keyboard.

3.2 - PAGE SELECTION AND CURSOR CONTROL KEYBOARD

This keyboard comprises 2 rows of 10 keys. The upper keys are for selection of the page to be displayed, the lower ones, in the modification or program writing mode, make possible the control of the cursor, the cancellation or insertion of a character in a block, the deletion of the dialogue line, the last key corresponds to the end of the block (LF).

3.3 - ALPHANUMERICAL KEYBOARD

Comprises the complete alphabet, figures from 0 to 9 and special characters.

3.4 - MODE SELECTION KEYBOARD

This keyboard comprises tow rows of 8 keys with integral signal lamps ; the modes are mutually exclusive.

- Automatic continuous mode (CONT). This type of operation is only possible with a workpiece program in the memory.

- Sequential mode (SEQ), running sequence by sequence of the program in the memory.

- MDI mode. Entry of data at alphanumerical keyboard of one block with immediate execution (not kept in the memory).

- Rapid (RAP) mode, execution of program in memory with rapid movement of axes at high speed.

- Sequence number search mode (RNS). Search for a sequence of a programm stored in the memory, storage and output of modal auxiliary functions for resumption of machining.

- Modification mode (MODIF). Makes possible the cancellation, replacement or insertion of characters in a block. The insertion of a complete block in a program stored in the memory or the cancellation of such a program.

- Test mode. Complete test of a workpiece program without any action on the machine or movements of machine slides. Complete analysis of preparatory data for interpolated linear or circular movements ... Generation of error messages allowing program corrections to be made.

- Manual mode : allows use of axis controls with limited or unlimited movements of the axes by means of the handwhell. Three feedratio are available, very slow, slow or fast. Information is output which unlocks the other controls ; spindle, turret ...

- Machine origin setting mode (POM). Makes possible the initialisation, on stops or on a master workpiece, of the machine origins. When the unit is switched this mode must be selected before the selection of the continuous, sequential, RNS or rapid modes.

- Entry of workpiece (PREF). Allows either direct entry through the keyboard of the numerical values for each axis or the storage of dimensions when the machine slide is at the program origin point. The workpiece origin values are displayed in the current point page.

- Semi-automatic entry of tool gauges (REG OUT).
  From a master workpiece the compensation values to be applied to a tool are determined on the machine and introduced into the table of compensations (see tool processing chapter).

- Data entry (CHARG). Storage of workpiece programs from the keyboard or a tape reader or of a supplementary line. Storage of tool compensation tables from a tape or supplementary line. The choice is made on the keyboard according to a displayed menu.

- Unloading of memory (DECHARG). The information contained in the memory can be sent to a tape punch, a cassette or to a supplementary line. The choice of the type of information to be sent (workpiece programm or tool table) is made on the keyboard according to a displayed menu.

3.5 - TOOL WEAR COMPENSATION TABLE

A special keyboard makes possible the entry during machining of the tool gauges and of the compensations for wear into the allocated tables.

3.6 - MANUAL CONTROL KEYBOARD

This keyboard has on it the keys controlling the movement of the axes and the selection of the movement value (1-10-100-1000-10000 or unlimited microns) and the enabling of movements by the electronic handwheel.

Rapid key in manual mode.

3.7 - VARIOUS FUNCTIONS KEYBOARD, COMPRISING

- the block skip keys with signal lamps, stop option, zeroing of active memories.
- fault signal lamps (internal to NC or programming error), operator, stop (feed stop by interface).

3.8 - CONTROL KEYS

Cycle start, machining stop and axis recall.

3.9 - POTENTIOMETERS

Feed rate override from 0 to 120 % potentiometer
Speed override from 50 to 100 % potentiometer.

These potentiometers can be placed on a machine panel.

The selector switch for mode modification or mode change.
This selector is lockable by means of a key.

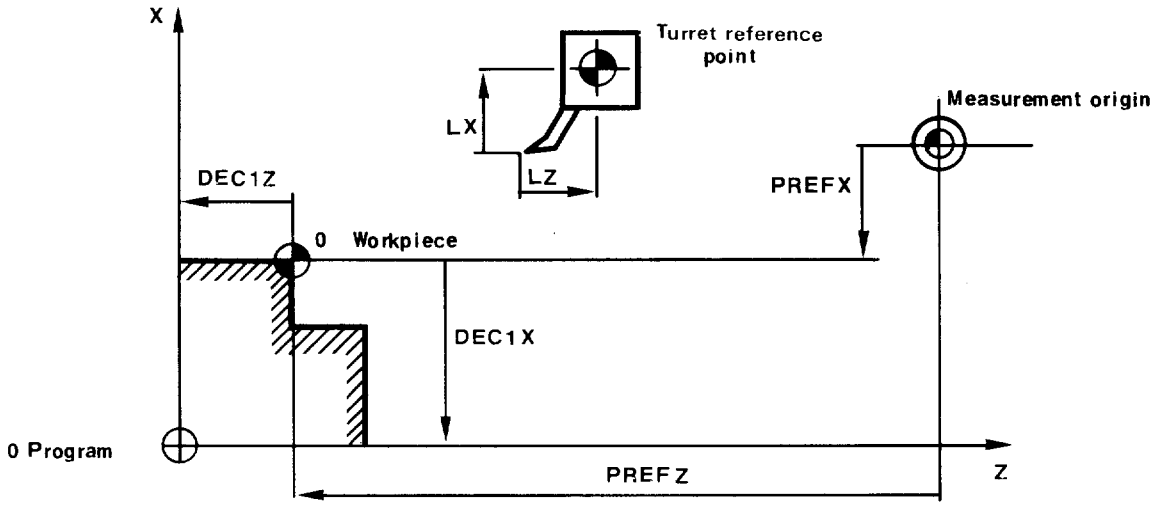
4 - GENERAL FUNCTIONS OF NUMERICAL CONTROL

4.1 - ORIGIN AND OFFSET

To reach the program starting point fixed by the programmer it is necessary, each time the system is switched on, to perform a measurement origin setting on the stop ⊕

Workpiece origin (◕ the point sensed) and the programm origin (⊕ dimensioning origin) are entered at the keyboard as offsets PREF and DEC1. These values are retained after switching off.

Furthermore the tool gauges must be entered according to the process described below.



PREFX and PREFZ = Offset between measurement origin and workpiece zero.

DEC1X and DEC1Z = Offset between workpiece zero and program zero.

LX and LZ = Tool gauge in relation to turret reference point.

The origins, offset and tool gauge in X are defined for radius.

The dimensions for the current point displayed on the screen are either the coordinates for the slide reference point in relation to the measurement origin or the coordinates of the tool point in relation to the program origin.

4.2 - LOADING AND UNLOADING OF WORKPIECE PROGRAM

The workpiece program data is stored in CMOS RAM memories with back up batteries which give a safeguarded time of 400 hrs.

Storage capacity is 16000 characters or approximately 40 metres of tape which can be supplemented by a 32000 characters module i.e. 80 extra metres of tape.

The system allows for the storage of different programs within the memory capacity limits, the programs may be numbered up to 9999 and their selection is either by the keyboard or by the P L C.

Links with the various peripherals is by means of a V24 series link.

4.2.1 - Loading of workpiece program

Can be from the numerical control keyboard, from a tape (paper or magnetic) or from a supplementary line.

The program format is

```
%  N° of program (from 0 to 9999) *
N1 - - - - - - - -  *
'
'
'
Nnn - - -  M02    *  "XOFF"  *
```

4.2.2 - Unloading of workpiece program

This can be to a tape punch or cassette, to a printer or a supplementary line.

The program format is identical to the previous paragraph.

4.2.3 - Program modification

Any program in the memory can be modified at the keyboard.

The following is possible in this mode :

- search of a block by its number or by a string of characters,
- suppression, addition or modification of a block,
- suppression, addition or modification of characters in a block by control of the cursor.

The workpiece program is directly modified in its memory content ; the volume of correction is not limited by the need for the availability of a corresponding volume of workpiece memory.

4.3 - CONTROL OF AXES

Two slaved closed loop axes X and Z, with variators controlled by a voltage of ± 10 V and measurement to within 1 micron.

Semi-absolute measurement can be made by :

- linear or rotary pulse sensors,
- resolver,
- inductosyn rules.

The maximum permitted feed is 15 m/mn, the minimum feed 0.01 mm/mn. These feeds can be varied from 0 to 120 %.

The maximum machine travels are ± 99.999 metres.

Backlash on axis reversal can be compensated to ± 0.1 mm.

Axis calibration makes possible the memorising of differences between measurement and the true position of the slide within ± 1 mm. The maximum number of points compensated is 120 for all of the axes.

Axis displacement is obtained either by programming or manually (slow or fast) or by rotation of the electronic handwheel.

4.4 - SPINDLE CONTROL

The spindle speed is programmed in revolutions per minute under the address S4. The system takes into account the range programmed (M40 to M45) or automatically defines the choice of range to output the motor reference : analogue output ± 10 V at 5 mA.

The spindle speed can be varied from 50 to 100 % by the potentiometer.

At constant cutting speed (G96) expressed in r.p.m. the spindle speed can be limited to a value expressed in r.p.m. by means of the address S and the code G92.

4.4.1 - Spindle with pulse sensor or resolver necessary for thread cutting operations

The system makes possible the indexation of the spindle over 360 programmable positions by the address C.

A supplementary C axis can be controlled, which is considered as a rotary axis having 360° at a resolution of. 0.001 degrees.
This axis can be used as a precise positionning head, mechanically locked , or as an axis linearly interpolated with the other X and Z axes .
To achieve these objectivies a closed loop drive system and incremental positional encoder are necessary.

The resolution on Axis C in 1/1000 of a degree.

4.4.2 - Auxiliary spindle : drilling head, milling

Control can be by means of the P L C which produces a binary or analogue output.

4.5 - PROCESSING OF DIMENSIONS AND TOOL COMPENSATIONS

This data is stored in a RAM backed up by battery with an independant operation time of 400 hrs.

Tool compensations are called up by the address D and are independent of the tools called up by the address T.

The maximum storage capacity is of 99 compensators, each compensator comprises the tool gauge in X and Z, the tool nose radius, the orientation code for the centre of the tool nose and the wear compensation in X and Z.

The maximum permitted values are :

LX or LZ $\pm$ 999.999 mm for ganger

ΔX or ΔZ $\pm$ 999.999 mm for tool compensation

R $\pm$ 999.999 mm

C : orientation of tool nose centre from 0 to 8

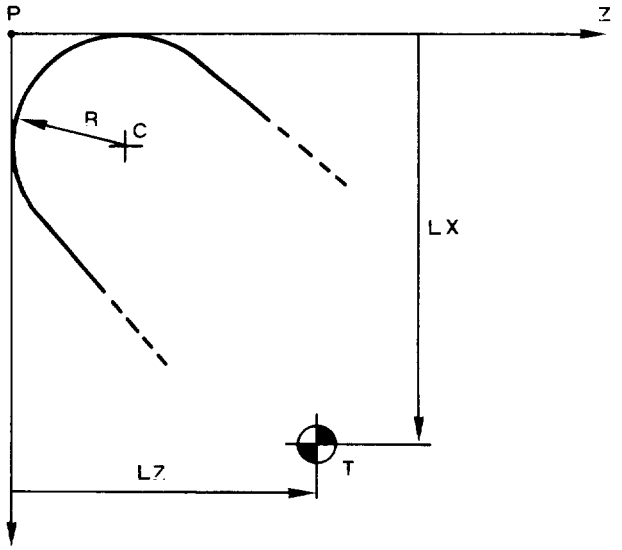
The dimension and tool wear tables are displayed on the tool page. Repeated pushing of the tool page key gives alternating access to one then the other.

The tool gauge table can be transferred to a tape or other peripheral using the unload key.

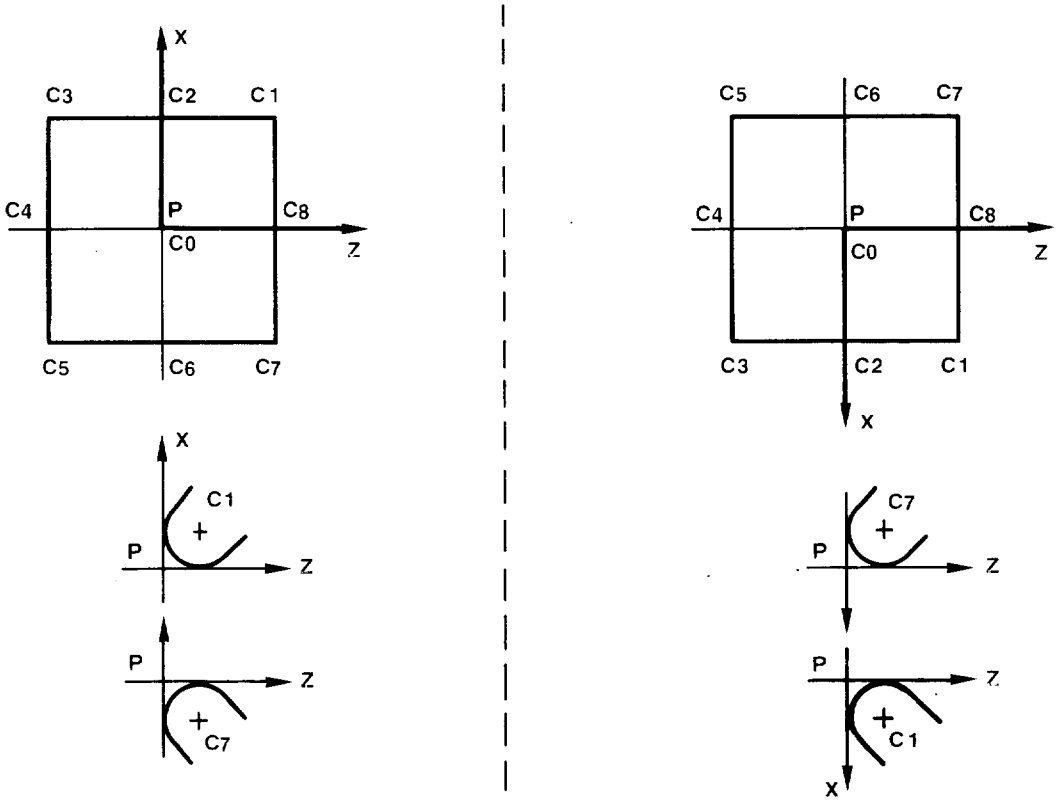
4.5.1 - Definition of the tool nose centre orientation (C)

The tool gauges are measured on the cutting faces of the tools parallel to the machine axes. If there is a slope or other trajectory the tool radius must be known for correction perpendicular to the profile.

The theoretical cutting point is P. When correction of the tool radius (G41 or G42) is called up the system translates the vector $\overrightarrow{PC}$ and then applies the correcting vector of the radius R perpendicular to the profile. The vector $\overrightarrow{PC}$ is defined by a code from C0 to C8 entered at the keyboard for each tool.



Examples :



4.5.2 - Tool gauges

There are four possible ways of entering the tool gauges into the memory.

- Manual entry through the keyboard (OUTIL - TOOL)

  Format : D2 X± .... Z± .... R± .... C0 to 8.
  D2 indicating the compensator number.

- Semi-automatic entry : REG OUT mode

  The values are calculated directly by the CNC after the sensor has scanned a master workpiece whose dimensions hase previously been entered in the memory.

  The data entered is :

  D2 X R ± 3.3 C1 * } X or Z without data
  or D2 Z R ± 3.3 C1 * }

- Entry by tape or supplementary line

  This mode is exclusive of the others and is activated by pushing the loading button (chargement).

  The data format is :

  % D2 X ± 4.3 Z ± 4.3 R ± 3.3 C1 *
  '
  '
  '
  Dn ---------- X off

- Entry from the PLC

  Parameterized programming gives access to numerical values present at the interface and at the dimension tables (parameters of E5 and E4 type).

  The workpiece program then provides transfer of values from the interface to the tool gauge table.

4.5.3 - Tool wear compensation

The tool gauges can be modified by compensation for wear of a limited amplitude. For each compensator the wear values are added together and may be compared to a threshold using parameterized programming.

The maximum value witch can be entered for each operation is ± 0.999 mm.

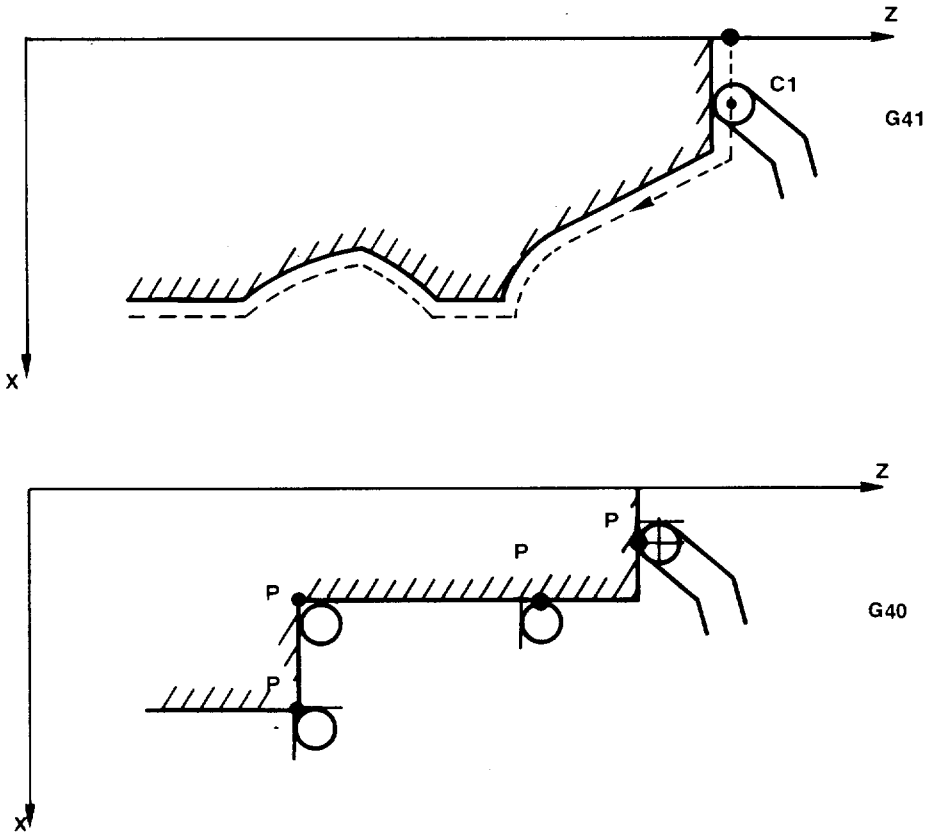
The compensations are expressed to the diameter in X.
The added total for each compensator can be zeroed by the operator, by the interface or by entry of a new tool dimension.

Entry of compensation values and of the compensator number is performed :

- on request by the operator,
- by the P L C connected to a workpiece monitor.
- by the P L C on request from the workpiece program using the paramaterized programming (E5 and E4 type parameters).

4.5.4 - Tool radius compensation

The system calculates the offset perpendicular to the profile and calculates the intersections of trajectories parallel to the programmed elements (straight lines, circles) and applies the offset according to the direction of the travel (G41 or G42). The cancellation of the radius compensation (G40) for turning or dressing affects the cutting point at the point P. (§ 4.5.1).



5 - PROGRAMMING

5.1 - GENERAL DATA FORMAT AND DEFINITION OF ADDRESSES (ISO/DIS 6983/1 Annexe C)

```
% 04 / DS N04 G0E H04 X+053 Z+053 I+053
K+053 A+033 B+053 P053 Q053 R053 F052
M02 S04 T03 D02 L02 E113 *
```

Programming of dimensions and machining dimensions is expressed with an explicit decimal point : X+36 equivalent to X=36 mm
X-0.3 equivalent to X=-0.3 mm

% 04	Program number
N04	Sequence number
G02	Preparatory functions
H04	Machining sub-program in call sequence comprising G77
X + 053	Movement in X programmed to diameter or radius
Z + 053	Movement in Z
I + 053 K + 053	In G2 or G3 absolute or incremental coordinates of centre of circle In G33 or G38, K thread pitch projected on X or Z according to cone angle In G64 or G65 Roughing over-dimemsion according to X or Z
EA + 033	In G1 angle in degrees of the straight line in relation to axis Z In G33 angle to cone to be threaded In G65 angle of penetration line for bore roughing In G66 angle at bottom of bore
EB + 053	EB + in G1, G2 or G3, fillet between any two elements EB - in G1 chamfer between to straight lines EB033 in G33 expresses as an absolute value the angle of penetration
C033	Spindel axis used as circular axis
P053	In G33, total thread depth In G64 or G65 penetration at each pass on X In G66 pitch value on X In G63 or G87, value of the first penetration
Q053	In G33 depth of final pass In G83 or G87 value of final penetration
R053	In G3 or G3 radus of circle In G33 projection on X of clearance cone In G64 or G65 penetration in Z at each pass In G66 pitch value according to Z
F052	In G94 feed rate expressed in mm/mn . Max. 15000 mm/mn In G95, F023 : feed rate in mm/r. Max 32 mm/r. In G33, F01 : number of threads In G04, F022 : value of dwell in seconds : Max. 99.99 secs.
M02	Auxiliary functions : 32 decoded, 68 coded
S04	In G97 spindle rotational speed in r.p.m. In G96 cutting speed in m/mn In G92 maximum spindle speed in r.p.m. In G33 : number of passes In G77 : number of repitions of a sub-program
T03	Tool number from 0 to 999
D02	Compensator number from 0 to 99
L02	Program variables from 0 to 19
E113	External parameters
C1	Tool nose position

5.2 - G FUNCTIONS

CODE	COUNTERMAND	DESIGNATION
G00	01-02-03 33.....	Rapid linear interpolation
G01	00-02-03 33.....	Linear interpolation at programmed feed
G02	00-01-03 33.....	Circular interpolation at tangential programmed feed : clockwise
G03	00-01-02 33.....	Identical to G02 but anti-clockwise
G04		Programmable dwel with address F
G09		Precise stop at and of block before next block
G10		End of machining on acquis. of "End of block stop" and start of next block
G16	End of block	Specification of tool axis by P, Q, R
G33	00-01-02-03	Constant speed thread cutting
G38	00-01-02-03	Threading continued on cone
G40	41-42	Cancellation of tool comp. on radius
G41	40-42	Comp. of tool raduis to left of profile
G42	40-41	Comp. of tool radius to right of profile
G52	End of block	Abs. programming of dimensions to meas. origin
G53	G54	Suspension of program zero to mach. zero
G54	G53	Enabling of programmed zero to mach. zero
G59	End of block	Offset of programmed origin. Added to offset enabled by G54
G64	G80	Paraxial roughing cycle
G65	End of block	Bore roughing cycle
G66	End of block	Deep boring cycle
G70	G71	Entry of data in inches
G71	G70	Entry of data in metric
G75	End of block	Enable emerg. tool lift off sub-program
G77	End of block	Unconditional call up of sub-program or of cont. of sequences with return
G79	End of block	Cond. or Uncond. skip to seq. without return
G80	64-83-87	Cancellation of machining cycle
G83	G80-G64-G65 G66-G87	Deep hole drilling with withdrawal
G87	G80-G64-G65 G66-G83	Deep hole drilling with chip break
G90	91	Absolute programming in rel. to prog. origin
G91	90	Incremental programming in rel. to block start point
G92		Limitation of spindle speed in CCS
G94*	93-95	Feed rate in mm/mn
G95	93-94	Feed rate in r.p.m.
G96	G97	Constant cutting speed
G97	G96	Spindle speed in r.p.m. (autom. choice of range)
G98		Definition of starting radius or diameter when using interpolation chuch with X and Z or alone for calculation of rotational speed.

NOTA : Indicates the functions initialized on switching on or after reset to zero

5.3 - M FUNCTIONS

CODE	COUNTERMAND	DESIGNATION
M00	Action on DCY	Program stop
M01	Action on DCY	Optional stop
M02	% EOR	End of workpiece program
M03	M4-M5-M0-M19	Spindle rotation, clockwise
M04	M3-M5-M0-M19	Spindle rotation anti-clockwise
M05	M3-M4	Spindle stop
M06	Report	Tool change
M07	M9-M0	Coolant on n° 2
M08	M9-M0	Coolant on n° 1
M09	M7-M8	Coolant off
M10	M11	Axis clamped
M11	M10	Axis unclamped
M19	M3-M4	Spindle indexation
M40 to M45		6 spindle ranges
M48	M49	Enabling of spindle and feed potentiometers
M49	M48	Inhibition of spindle and feed potentiometers

5.4 - SUB-PROGRAMS AND SKIPS

5.4.1 - Call up of sub-programs with return (G77 function)

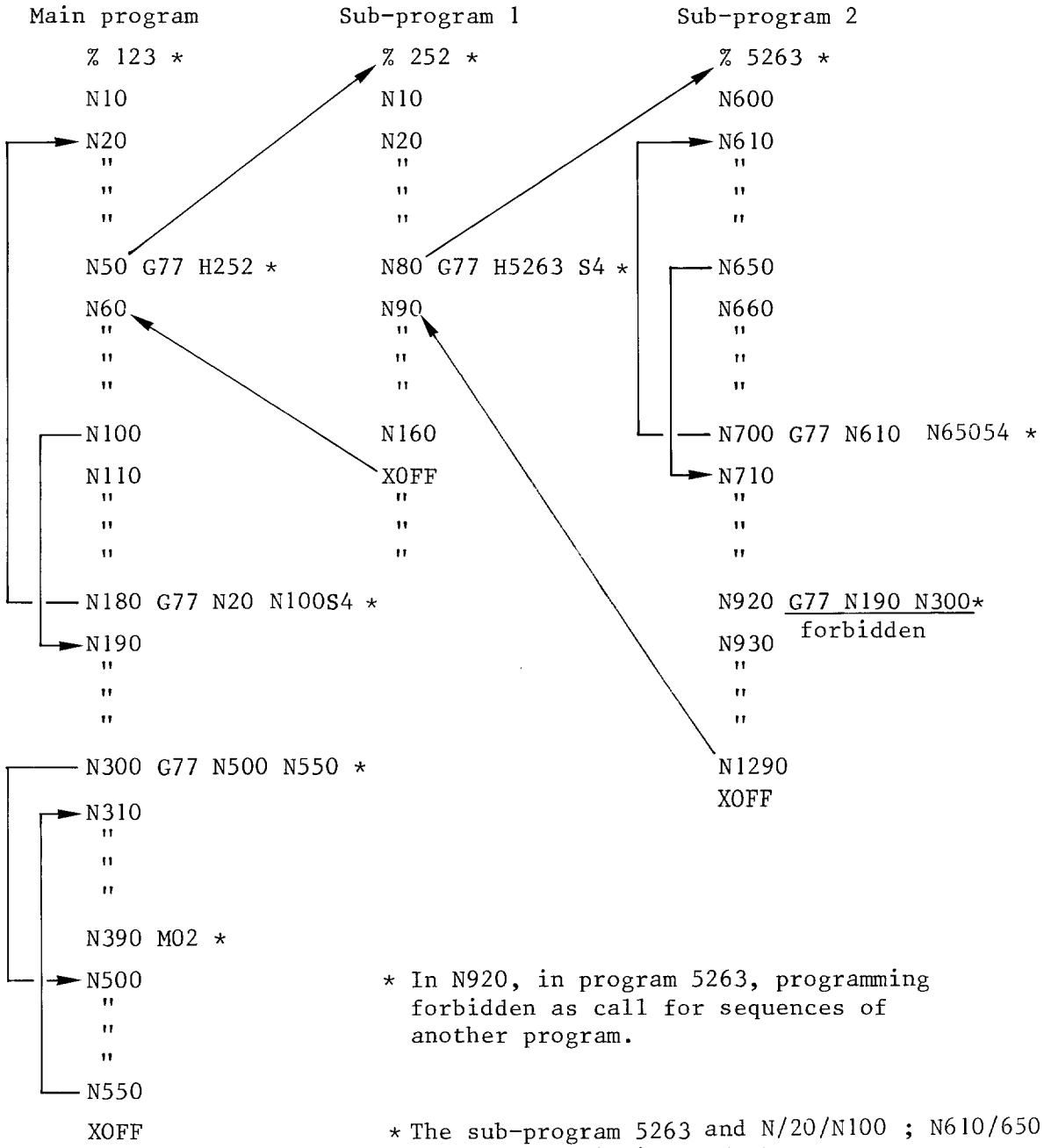
The call is unconditional.

The sub-programs (or machining sub-routines) may be defined in the main program . Each sub-program can be repeated 9999 times (S4) and 4 interleaving of sub-programs are possible.

A sub-program in the main program is defined by the starting and finishing sequences ; it can be situated between the % and M02 or beyond M02. It is not possible to call the continuation of an external sub-program from the main program.

An external sub-program is called by the word H followed by the sub-program number.

Example :



* In N920, in program 5263, programming forbidden as call for sequences of another program.

* The sub-program 5263 and N/20/N100 ; N610/650 are executed 4 times (S4).

5.4.2 - Skip of sequences without return

The sequence skip is either conditional or unconditional and allows direct passage to the sequence called which must be contained in the same program.

- Unconditional skip

  G79 N280 : skip to sequence 280 which can be situated before or after the call sequence.

- Conditional skip

  Use of a program variable or of an external parameter in the skip block and of a condition > or = or <.

  G79 L2> = 3 N280. If L2 is greater than or equal to 3, skip to the sequence 280, if not continuation to following block.

5.5 - GEOMETRIC PROFILE PROGRAMMING

With the NUM 760 T it is possible to write the program directly using drawing dimensions. The system performs the calculations of connecting points, contacts and intersections which are not defined between two elements of travel on the workpiece (straight line to straight line, straight line to circle, circle to circle).

The standard method of programming is still valid and may be used in conjunction with geometric programming.

5.5.1 - General

Programming is by blocks each block containing a geometrical element (segment of a straight line, arc of a circle).

A geometric element may be wholly defined in a block (end points for straight line, extremities of arc of circle and centre coordinates) or its definition in the block may be incomplete.

If the definition is incomplete the remainder of the information is contained in the following block or if necessary in the following two blocks.

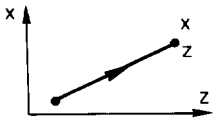
It must be indicated whether the following element is tangent or secant (ET or ES) ; where several solutions are possible (circle of radius R passing through a point and tangent to a straight line) the choice is made by the discriminator E+ or E-.

The processing excludes stationary points.

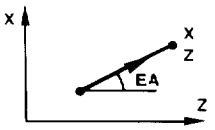
The discriminator may be omitted in which case the system chooses the circle whose travel is less than 180° as is generally the case for workpieces on a lathe.

5.5.2 - Programming of a single geometric element

Straight line :

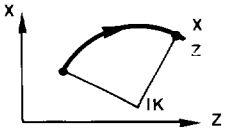


G1 XZ

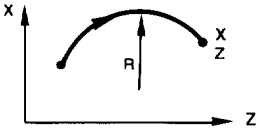


G1 EAX or EA2

Circle :



G2 XZ IK

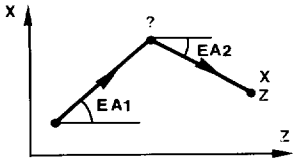


G2 XZ R

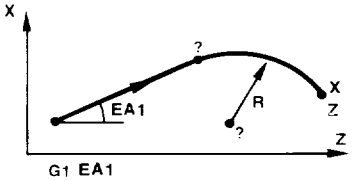
5.5.3 - Associations of geometric elements

Some of the combinations which the system is capable of :

- programming of two secant elements with an unknown intersection point,
- programming of two tangent elements with an unknown point of contact :

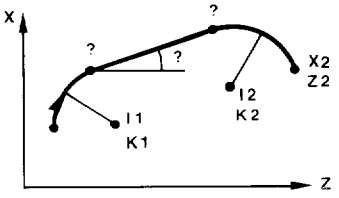


G1 EA1 ES
G1 EA2 XZ

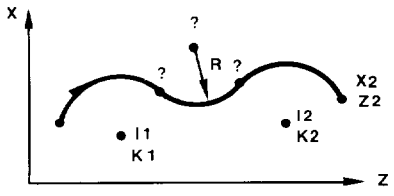


G1 EA1
G2 XZ R

- programming of three tangent elements whose two points of contact are unknown :

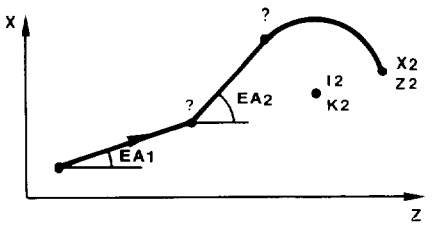


G2 I1 K1
G1 ET
G2 I2 K2 X2 Z2



G2 I1 K1
G3 R
G2 I2 K2 X2 Z2

- programming of three elements with an intersection and an unknown tangent connection :

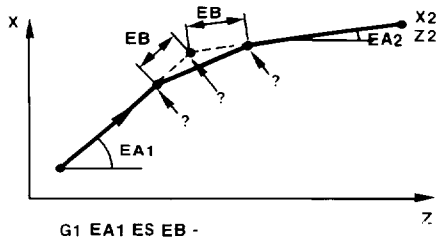


G1 EA1 ES
G1 EA2
G2 I2 K2 X2 Z2

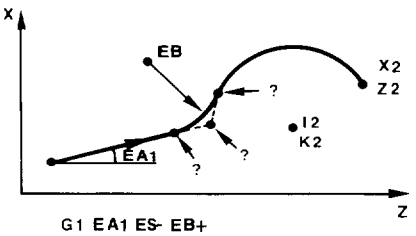
5.5.4 - Programming of chamfers and fillets

A chamfer may be inserted between two successive straight lines and a fillet between any two elements.

The chamfer is programmed by the code EB - and the value, the fillet by the code EB + and the value.

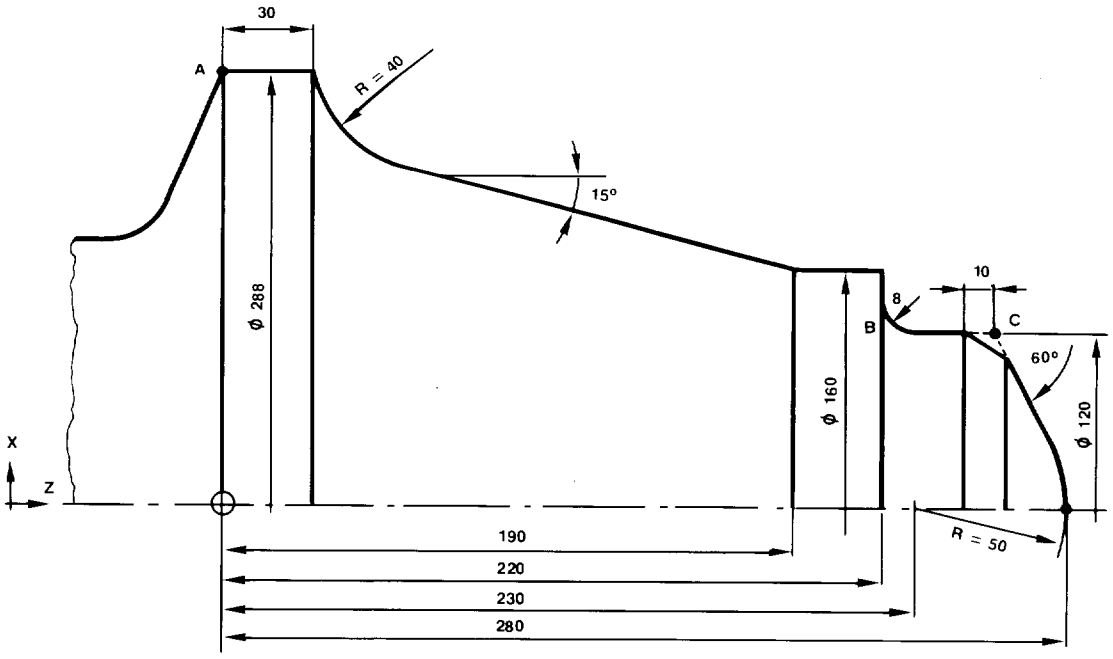


G1 EA1 ES EB -
G1 EA2 X2 Z2



G1 EA1 ES EB+
G2 I2 K2 X2 Z2

5.5.5 - Example of programming

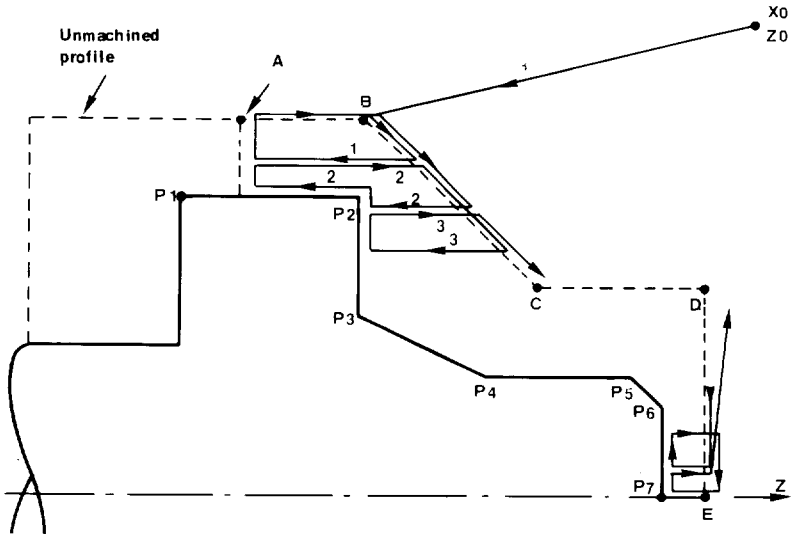


N100 X288 Z0 (point A)
N110 G1 Z30
N120 G3 R40
N130 G1 EA-15 X160 Z190
N140 Z220
N150 X120 EB8 (fillet in B)
N160 EA ES EB-10 (chamfer in C)
N170 EA-60
N180 G2 X0 Z280 I 0 K230

5.6 - MACHINING CYCLES

5.6.1 - Rough clearing paraxial cycle : Function G64

Makes possible from the definition of a finished profile and the definition of an unmachined profile to carry out rough machining of a workpiece in paraxial on axis X or axis Z.



Format : G64 N_{P1} N_{P7} XA ZA I K P or R F S

N_{P1} to N_{P7} definition of final profile : rough from NP1 towards NP7 the order of the sequences giving the direction of machining.

XA ZA : stopping coordinate for first roughing passes.

I, K : machining allowance in X and Z.

P or R : depth of pass :

- . P longitudinal rough machining
- . R transverse rough machining

F - S : feed rate and spindle speed.

P or R,F,S can be modified during rough machining.

Programming

N10 G X_0 Z_0 Initialization in X and Z for starting of rough machining.

N20 G79 N100 Skip to call sequence of rough machining cycle.

N30 G1X_{P7} Z_{P7}
'
'
'

N40EA 90EB-
N50EA 180 Z P4
'
'
'

N90 X_{P1} Z_{P1}

} Definition of final profile

N100 G64 N100 N30 XAZA I K P F S Rough machining zone

N120 X_B Z_B

N130 X_C Z_C PFS

N140 X_D Z_D

N150 X_E B_E

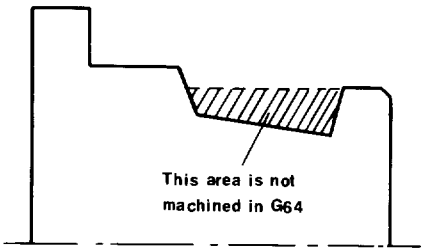
Value changing sequence for P, F and S } Definition of blank profils

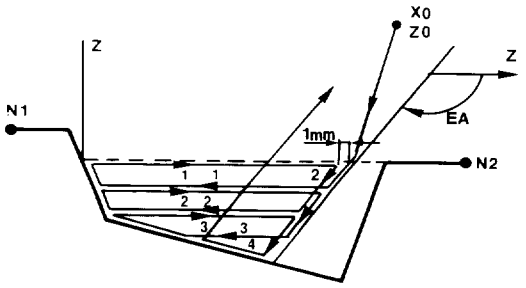
N160 G80 S Z End of rough machining program (lift off)

N170 G77 N30 N90 G41 T D F S Execution of finishing

5.6.2 - Area clearance cycles for bevelling or special groove cutting : function G65

In the definition of the finished profile the groove A is programmed but it is not machined during paraxial rough machining (G64)





Format :

G65 N1 N2EA A P or X or Z Z I K F S

N1 and N2 are the finished profile sequences of both sides of the groove.

EA is the machining penetration angle (penetration is at the programmed speed F).

P or R depth of pass on either X or Z.

X or Z stop coordinates for 1st pass.

I and K machining allowance on either X or Z.

F, S feed rate and spindle speed.

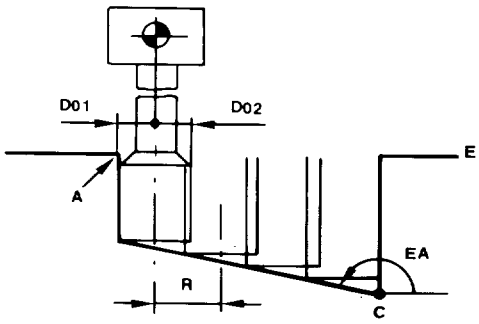
The order of the sequences N1 - N2 gives the part to be machined :

N1 - N2 : machining to the left of straight line A.

N2 - N1 : machining to the right of straight line A.

5.6.3 - Groove cutting with angled entry : function G66

This function makes possible the making of grooves by successive penetrations with a programmable dwell at the end of penetration (G4F).



G00 X_A Z_A D01

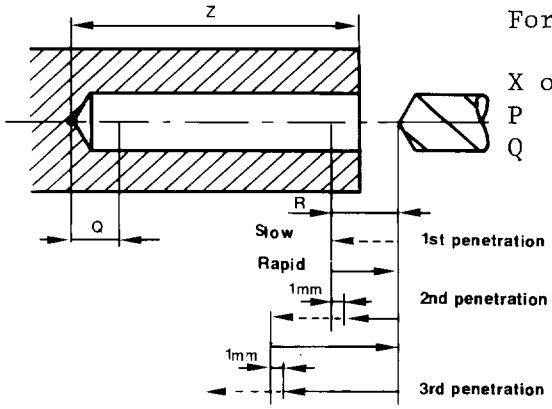
G66 X_C Z_C EA P or R D02

A : Angle at bottom of groove by omission
A = 0

PorR : Value of step on X or Z

5.6.4 - Deep hole drilling with withdrawal : function G83

This function makes possible drilling by successive passes with drill-bit withdrawal on Z axis for drilling tool mounted on the turret or on X or Z for an auxiliary head mounted on the turret.

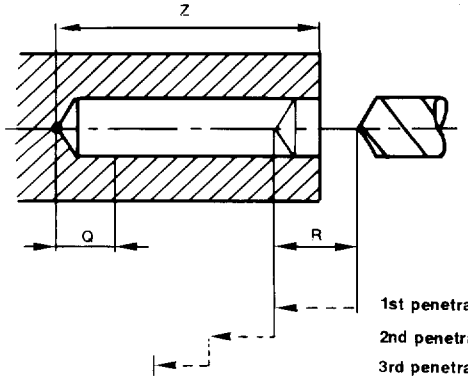


Format : G83 X or Z P Q

X or Z : depth of hole
P : first penetration
Q : last penetration (optional)

5.6.5 - Deep hole drilling with chip break : function G87

This function makes possible drilling by successive penetrations without withdrawal of the tool on the Z axis for a tool mounted directly on the turret or on the X or Z axes for an auxiliary head mounted on the turret.



Format : G87 X Z P Q

X or Z : depth of hole
P : first penetration
Q : last penetration (optional)

5.6.6 - Thread cutting cycle : G33 Function

A complete cylindrical, conical or frontal thread cutting cycle with constant pitch, decreasing machining depth and a constant turning cross-section can be defined and executed by the programming of a single block (G33)

Maximum pitch value = 250 mm.

Format of thread cutting block :

G33X Z K A B R P Q F S

X,Z : Absolute or incremental dimension of thread end point

K : Thread pitch projected on X or Z according to cone angle

EA : Angle of cone to be threaded in relation to Z axis

EB : Penetration angle of each pass

R : Projection on X or Z of lift off cone (incremental)

P : Total depth

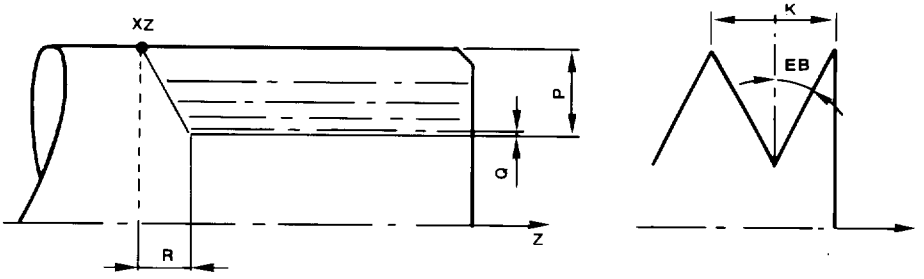
Q : Depth of last pass

F : Number of interlaced threads

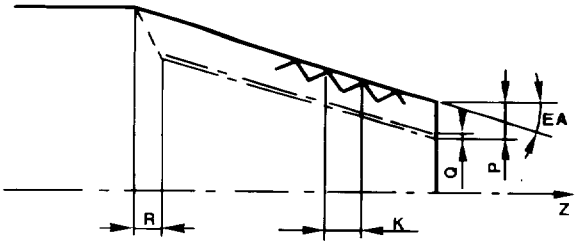
S : Number of passes

Example of thread

Cylindrical thread with lift off cone.



Conical thread with lift off cone.



5.6.7 - Linked threads : G38 Function

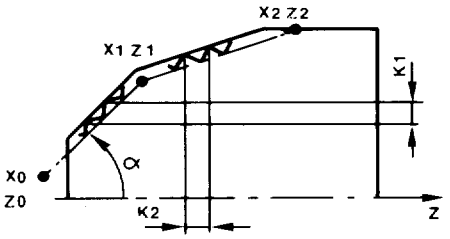
Format :

G38 X Z K

X and Z : Absolute or incremental dimension of final point of thread.

K : Projection of pitch on Z if the angle is less than 45°.

Projection of pitch on X if the angle is greater or equal to 45°.



G X_0 Z_0

G38 X_1 Z_1 K_1

X_2 Z_2 K_2

The cycles can be performed by re-looping of the sub-program.

5.7 - PARAMETERIZED PROGRAMMING

With the NUM 760 T the programmer has two types of parameter at his disposal :

-

-

5.7.1 - Program variable L

To allow for the machining of a series of workpieces or of special profiles requiring numerous repetitive sequences the programmer has 20 variables at his disposal.

These values may either be fixed or result from the following operations : addition (+), subtraction (-), multiplication (*), division (/), square root (R), sine (S), cosine (C), truncation (T).

The program variables can be allocated to all of the dimension and function programmable addresses. With the G79 function they make possible conditional skips in the program.

Example of writing :

L2 = 5
L1 = L2 + 5.3 * 3 * S30 equivalent to L1 = 15.45
XL1 Z30 : The parameter L1 is applied to the dimension X.

5.7.2 - External parameters E

The external parameters have been introduced into the programming to give a high degree of automization to the unit formed by the numerical control, the associated PLC, the machine tool and other associated units.

They make possible a dialogue between the peripherals and the numerical control via the PLC. The data exchanged (numerical or logic) is processed so as to ease the use of functions such as tool failure, workpiece count, tool wear, measurement by feeler, offset modifications, tool compensation modifications etc ....

Like the program parameters they can be allocated to all of the program addresses and participate in all operations.

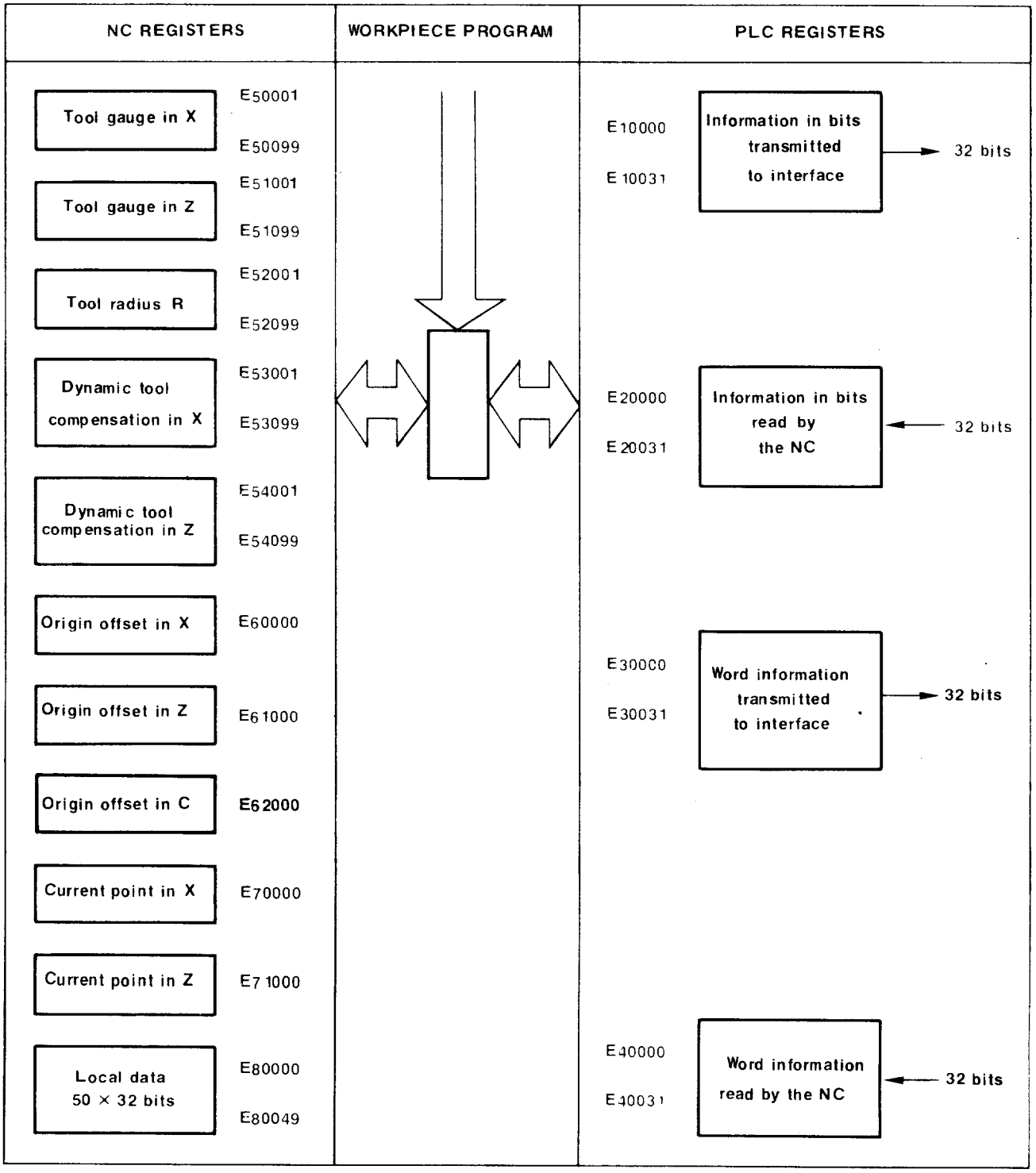
The parameter values may serve as comparative elements in blocks with conditional skip.

TYPES OF EXTERNAL PARAMETERS

They are defined by the code E followed by five figures. The left hand figure indicates the family of the parameter.

Example of writing :

E 51001 = E40000 indicates that the tool gauge number 1 in Z is loaded by the contents of the external parameter E40000.



6 - PLC

6.1 - GENERAL

The PLC is an important function of the automatic numerical control unit. Its role is the processing of the usual interface logical functions as well as the exchange of Boolean or numerical data between the numerical control and the external handling units or those units acquiring certain physical values such as dimensions, stresses, temperature.

The NUM 760 T's PLC is designed to deal with these problems. Direct access to the PLC address field by the NC, exchanges of parameters with the NC, console simulation and arithmetic operators are some of the tools which allow the numerical control to integrate naturally into an independent production cell or a flexible machine shop.

6.2 - ORGANIZATION

The PLC is incorporated into the back panel of the numerical control. It exchanges data with the NC via the NC processor 16 bit bus and with the exterior via the 8 bit input/output bus.

The 8 bit processor processes the instructions of the "client program", a monitor deals with a certain number of basic tasks such as initialization, input/output exchange synchronisation with NC bus, control of segments and basic tasks, processing of sub-routine instructions.

The "client program" is stored in the volatile EEPROM memory which can be electrically cleared. Storage capacity 8 K 8 bit bytes (approx. 2.5 K for instructions).

The data and internal variables are stored in the CMOS RAM, safeguarded for 400 hours :

256 8 bit bytes
128 bits initialized
128 bits safeguarded

Extensions to the memories allow for expansion of capacity to 40 K program 8 bit bytes, 5.5 K 8 bit bytes and 1.5 K bits for internal variables.

The NC processor has access to the whole address field of the PLC which allows for rapid exchange of external parameters (numerical or boolean data), of the states of the PLC inputs and outputs and of its internal variables.

6.3 - PROGRAMMING

The PLC program is scanned periodically every 20 ms. Complete processing of the program implies an organization on several levels of priority which provide for the execution of instructions in the time allowed for each module.

The PLC program is structured as follows :

- a sequential executed in each cycle,
- 4 segments, each one is scanned successively every five cycles,
- 16 hierarchically arranged basic tasks, armed in the sequential and whose execution may cover several cycles.

These different tasks are controlled by the PLC monitor.

A set of thirty instructions is provided to process the boolean and arithmetical values, skips and certain other conversion operations. The operands are bits, 8 bit bytes or double eight bit bytes.

- Boolean operations in bits or 8 bits bytes AND, OR, EOR, INV.
- Arithmetical operations in 8 bit bytes or double 8 bit bytes ADD, SUB, MUL, DIV, conversion BCD/binary and binary/BCD.
- Operations on 1 eight bit byte
  Splitting up of a word into bits and regrouping, offsets, incrementation.
- Skips
  Unconditional, conditional on calculation result or logic state.
- Counting : 16 base counters 32767
- Time delays : 16 time delays from 0,1 secs. to 3276 sec/55 MM

6.4 - INPUTS/OUTPUTS

- Exchange with numerical control bus

  This exchange is by means of direct access to the memory.

  In addition to the usual control functions and the M,S,T, codes it acts on those functions which facilitate the integration of the NC into an automized system :

  . simulation of control console,

  . dynamic tool compensation,

  . program numbers,

  . sequence numbers,

  . NC error numbers,

  . machine error messages to be displayed on the screen,

  . external parameters (32 bits and 32 words of 32 bits in both directions).

- External exchanges

  . 32 input module - 24 V decoupled - 1 common 0V for 8 inputs.
  24 V supplied by the NC - 8 modules maximum.

  . 32 output module - Relay - 1 common for 8 outputs.
  110V/2A/60VA voltage free contacts - 8 modules maximum.
  The states of the inputs and outputs is displayed by means of LED's.

The basic panel can accept up to 6 input/output modules (192 I/O in all). Beyond this number the modules are placed on an extension panel :

- 2 NAC 8 bits $\pm$ 10V 5 mA
- 6 ANC 8 bits
- 1 asynchronous V24 series link.

6.5 - PROGRAM DEVELOPMENT AND CHECKING OUT

Programming is performed in the "litteral" language, i.e. a list of instructions interpreted by the system. The sequences, bits, eight bit bytes, inputs and actuators can be referenced.

The programming aid is the numerical control console/screen unit. The program is loaded into the PLC EEPROM memory without any other loading aid.

Checking out of the program is done by an editor. The program is reconstituted on the screen or on a punched tape in the programming language.

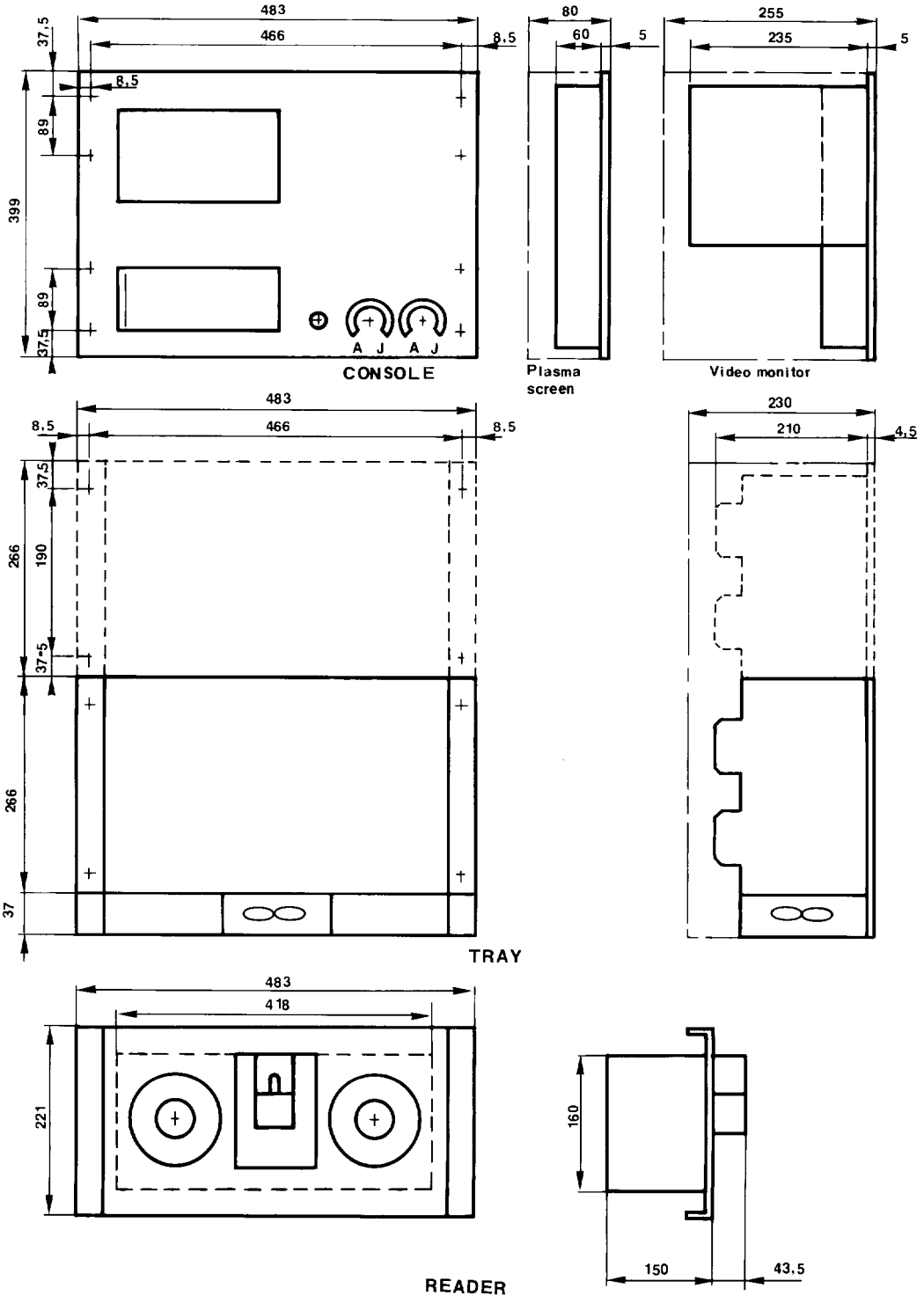
Checking out is facilitated by the display on the screen of the PLC inputs/outputs and of its internal variables. A "debugging" (checking out) procedure allows for "step by step", "cycle by cycle" operation or for the fixing of the stopping points.

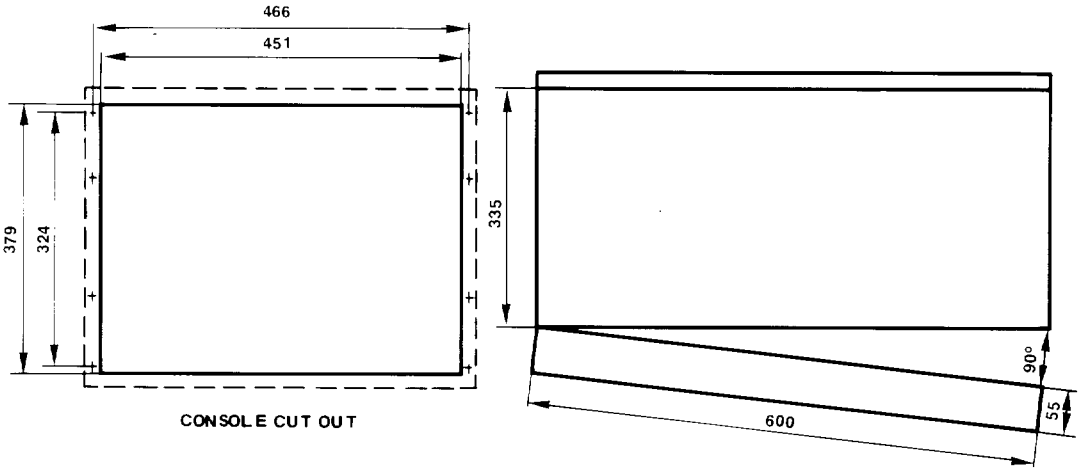
7 - CONDITIONS OF USE

7.1 - ENVIRONMENT

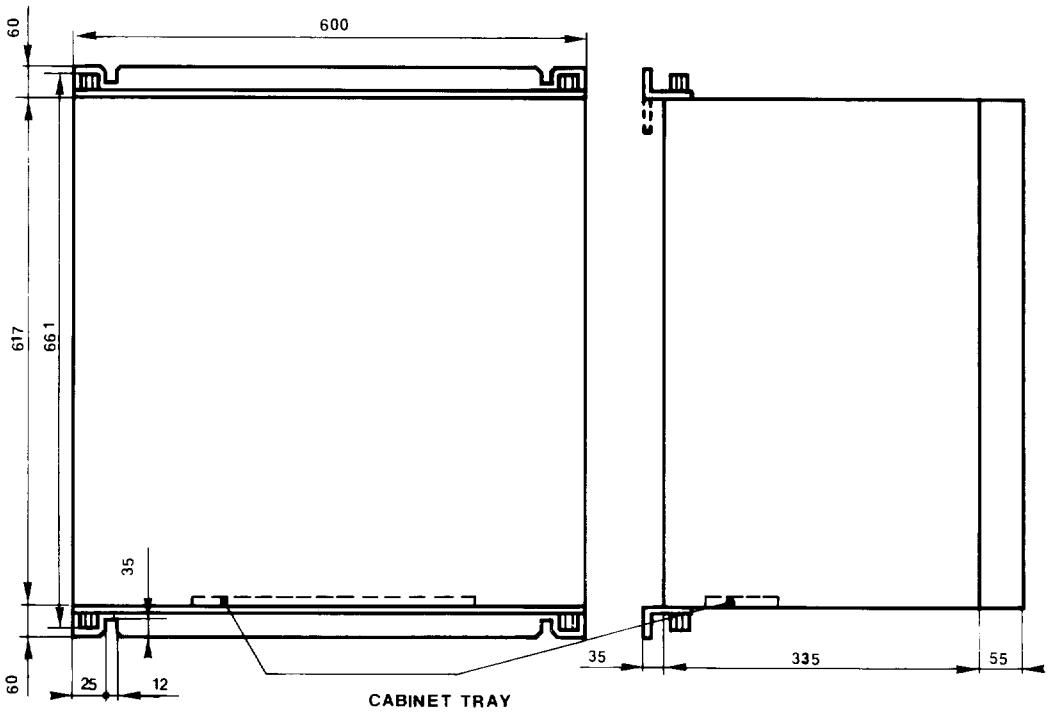
	PLASMA SCREEN CONSOLE	CRT SCREEN CONSOLE	1 RACK NC + PLC	2 RACK NC + PLC	VENTILATED SEALED CABINET	READER IN RACK	READER IN CABINET
Width	483	483	483	483	600	483	491
Height	400	400	266 +40	532 +40	615	221	237
Depth	60	250	210	210	430	240	330
Weight	8 Kg	10 Kg	12 Kg	20 Kg		8 Kg	16,5 Kg
Supply	220V	220V	220V	220V	220V	220V 140	220V 140
Power	60 VA	60 VA	100 VA	180 VA	250 VA	35	35
Permitted amblent temperature	0-50C	0-50C	0-50C	0-50C	0-45C	0-50C	0-45C
Transport temp.	- 40 - 70°C						
Protection	IP30	IP30	IP30	IP30	IP54	IP30	IP40
Front panel protection	IP54	IP54				IP54	IP54
Relative humidity	5-95%	5-95 %	5-95 %	5-95 %	5-95 %	5-95 %	5-95 %

7.2 - DIMENSIONS

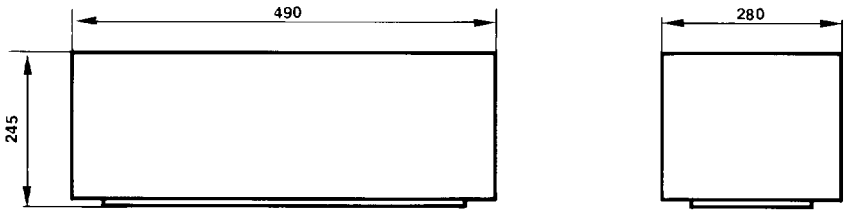




CONSOLE CUT OUT

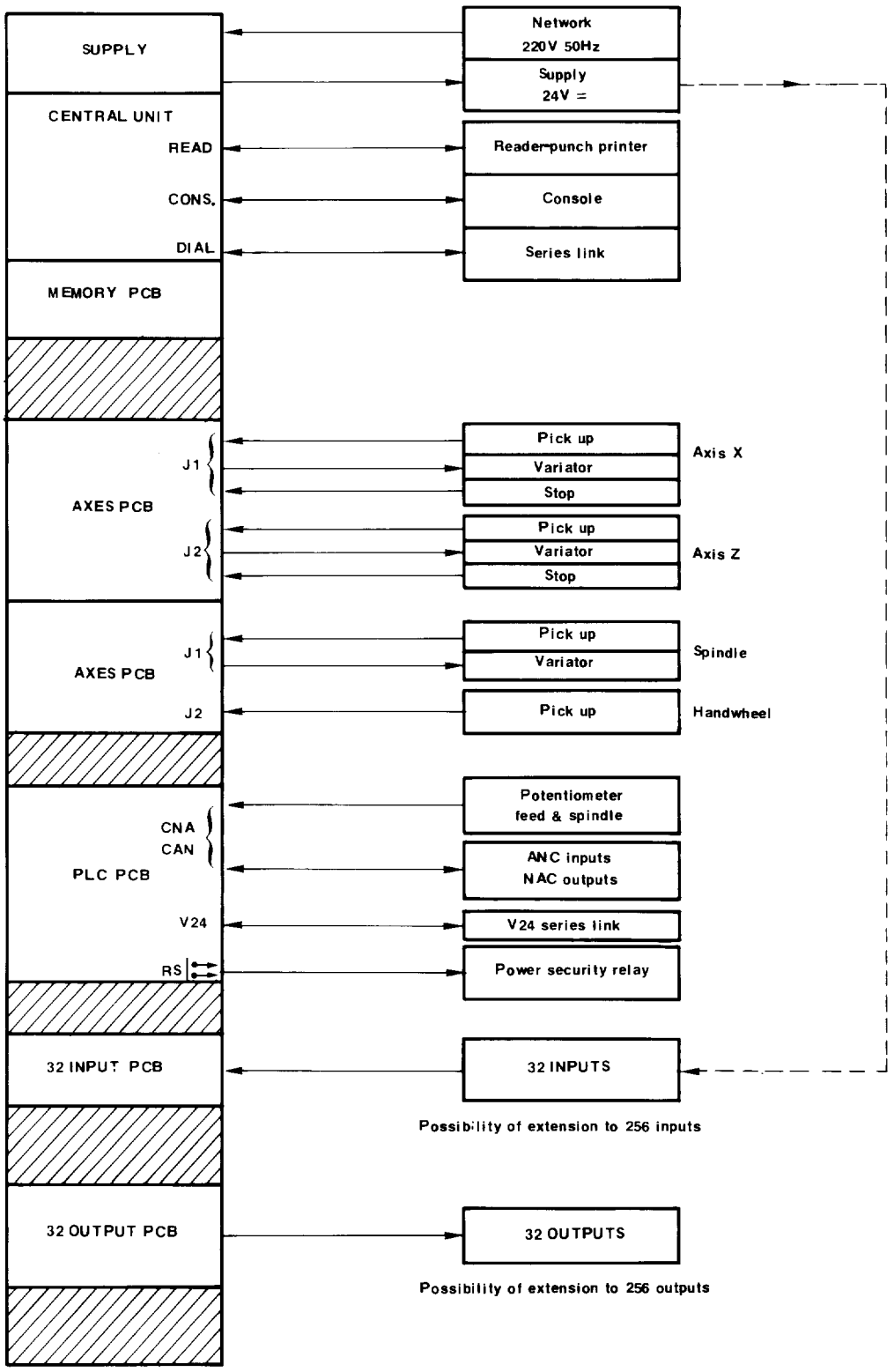


CABINET TRAY



READER CABINET

7.3 -


SUPPLY
Network
220V 50Hz
Supply
24V =
CENTRAL UNIT
READ
Reader-punch printer
CONS.
Console
DIAL
Series link
MEMORY PCB
AXES PCB
J1
Pick up
Variator
Stop
Axis X
J2
Pick up
Variator
Stop
Axis Z
AXES PCB
J1
Pick up
Variator
Spindle
J2
Pick up
Handwheel
PLC PCB
CNA
CAN
Potentiometer
feed & spindle
ANC inputs
NAC outputs
V24
V24 series link
RS
Power security relay
32 INPUT PCB
32 INPUTS
Possibility of extension to 256 inputs
32 OUTPUT PCB
32 OUTPUTS
Possibility of extension to 256 outputs