Programming of the bending machine

PROGRAMMING MANUAL

X41063C

CONTENTS

1. THE NUMERICAL CONTROL (NC)

The control system is the interface through which the compilation and launch of programs.
In the control system stored machine parameters, a database of working tools and
information about the bendable part. Modern control systems usually allow you to use several
different methods, with varying degrees of automation of calculations. Can enter the position of
the backgauge and the depth of penetration of the punch into matrix manually, or all
calculations can be performed automatically.

In manual programming, the operator enters all the axis positions directly into the bending program. No information is entered into the car neither the geometry of the product, nor the

tools used. Operator must choose the desired bending sequence.
For automatic programming, you must specify the geometry products (drawn graphically), used tools and material products. Following this, the control system automatically calculates the sequence of bending, the location of the backgauge for each bend and the depth of penetration of the punch into the die to achieve the desired angle bending. With automatic programming, the probability the occurrence of errors is significantly less than with manual programming.

1.1. INTRODUCTION

FEATURES

The number of axes displayed on the N. C. screen depends upon the machine configuration.

MEMORY: - Tools, 70 dies, 30 punches, 10 die-holders, 10 spacers.
       - The external memory can be used as a file storage facility or as a working memory.

PROGRAM: - Up to 6-digit program number.
       - Up to 13-digit work-piece reference.
       - Internal program capacity: maximum 60.
       - Maximum number of bends per program: 25.
       - Number of programs that can be saved on external memory:
       Maximum 60 programs when saved with the library.
       When saved using export function, the only limitation is the capacity of the external memory.

Y1, Y2 AXES: Management of:
       - Pressure holding time.
       - Distance of decompression.
       - Working speed.
       - Return speed.
       - Force calculation.

X1, X2 AXES: Management of:
       - Backgauge position (X for A-type backgauge, X1, X2 for B-type backgauge).
       - Backgauge retract at P.P.
       - Safety zones.
       - START delay on backgauge.

OTHER AXES:
       - "R" axis (R for A-type backgauge, R1, R2 for B-type backgauge).
       - "Z" manual axis. (Z manual for A-type backgauge, Z1, Z2 for B type backgauge).

This present programming manual applies to "CD2005" Numerical Control having a black/white monitor and the following 3 programming modes.

1. MANUAL 1 - simplified programming mode.
2. MANUAL 2 - angle programming mode or X-ALPHA.
3. MODE 3 - graphical programming mode or L-ALPHA.

DESCRIPTION OF THE THREE PROGRAMMING MODES

MANUAL 1 "Simplified programming mode".
>The operator feeds in the positioning dimensions for the axes or teaches in the positions by using the convenient keys together with the operator's footswitch for Y axis.
>A test bend can be performed for each bend sequence.

MANUAL 2 "Angle programming mode" or "X-ALPHA" programming mode.
>The N.C. calculates the depth of the tool penetration, the necessary force for bending, the backgauge position and the height of the backgauge fingers according to the programmed tool and the position of the backgauge fingers in relation to the part's length and its position.
>A test bend can be performed for each bend sequence.

MODE 3   "Graphical programming mode or L-ALPHA"
>Further to the graphical programming of the part, the N.C. specifies the bending order and calculates for each bend the necessary force for bending and the position of each machine axis.

⚠

To avoid damage to your tooling or any accident, Amada urge you to follow strictly the procedure below :

- Every time you change a program, select adjustment mode and mute stop.
- Carry out a dry cycle (without material).
- Check visually that all introduced parameters correspond to the tooling mounted on the machine and the mute point is 6 mm above sheet pinch point and the end of bend is correct.
- Then and only then choose the normal operating mode for the program in question.

1.2. CUSTOMIZING

Depending on the backgauge type fitted to your machine, several configurations are possible (see the drawing below):
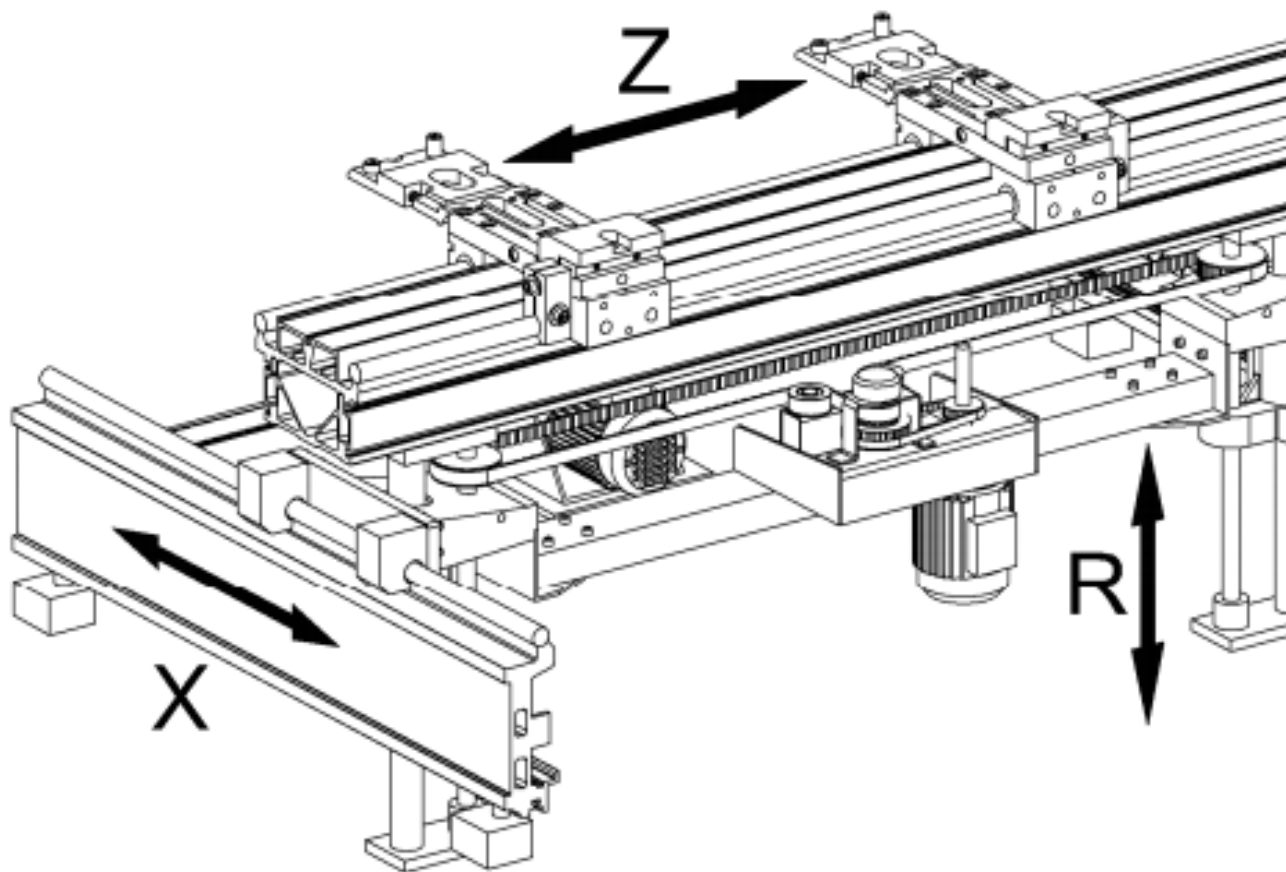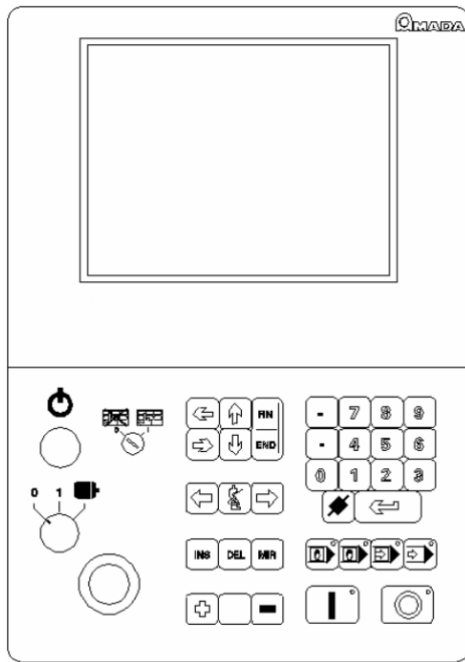
"X"-Axis: - "X" motorized (BG type "C" *, BG type "A")
"R"-Axis: - "R" motorized (BG type "C" *, BG type "A")
"Z"-Axis: - "Z" manual (BG type "C" *, BG type "A")

* Only on HFT 50-12

1.3. IDENTIFICATION OF THE CONTROL FEATURES

For ease of operation, the N.C. keypad is laid out as shown in the diagram. Different groups of keys are defined as follows:

"END/FIN" key: This is used for terminating and quitting a function such as program Edit.

These arrows are used for moving the cursor around on the displayed items.

These keys are used for "TURNING THE PAGE OVER" or move to next bend sequence.

"GRAPHICAL REPRESENTATION" key.

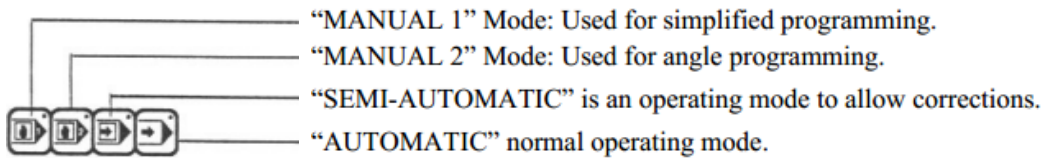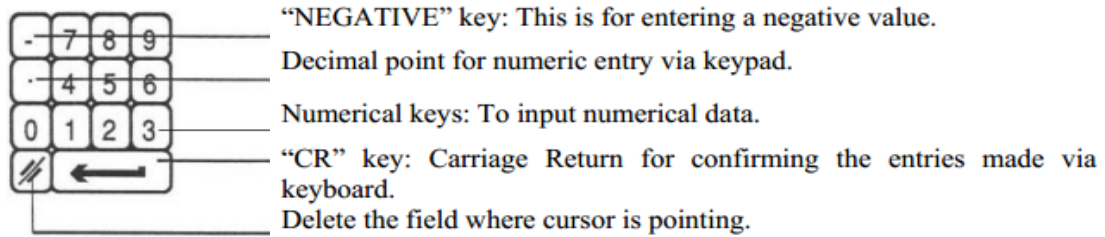"DEL" or delete key: This is used for deleting work-pieces or bend.

"MIR" or Mirroring key: used for programming symmetrical workpieces.

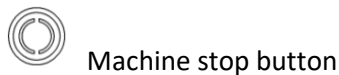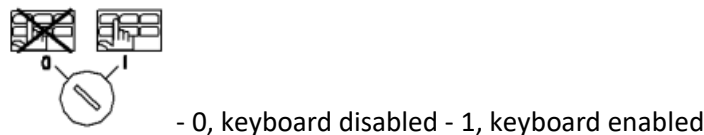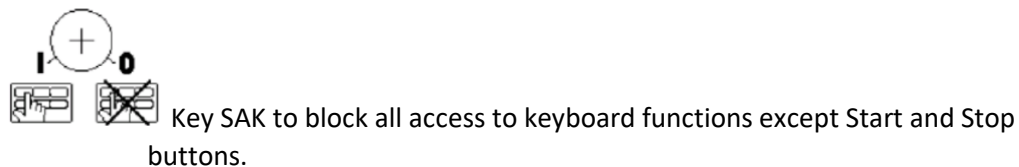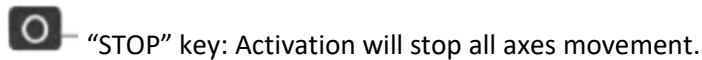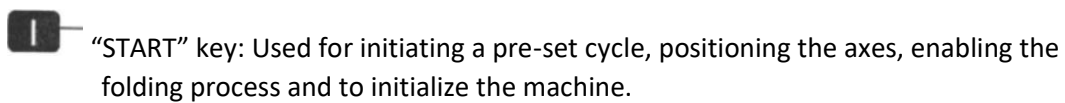"INS" or insert key: This is used for inserting a bend.

"PLUS" and "MINUS" keys: ZOOM function, i.e. enlargement or reduction of the graphic display.

"BLANK" key: This is used to activate an alphanumerical entry. This function is related to the CNC software version.

"NEGATIVE" key: This is for entering a negative value.

Decimal point for numeric entry via keypad.

Numerical keys: To input numerical data.

"CR" key: Carriage Return for confirming the entries made via keyboard.

Delete the field where cursor is pointing.

"MANUAL 1" Mode: Used for simplified programming.

"MANUAL 2" Mode: Used for angle programming.

"SEMI-AUTOMATIC" is an operating mode to allow corrections.

"AUTOMATIC" normal operating mode.

The above 4 modes are only accessible when machine is initialized.

"START" key: Used for initiating a pre-set cycle, positioning the axes, enabling the folding process and to initialize the machine.

"STOP" key: Activation will stop all axes movement.

Key SAK to block all access to keyboard functions except Start and Stop buttons.

- 0, keyboard disabled - 1, keyboard enabled

Machine stop button

START-UP

1. Turn the handle of the main circuit breaker to Position 1.

2. Turn the motor start-up key switch to the right unstable position. The motor starts running.
3. Press the green luminous reset push button.
4. The green indicator light lights up.
5. At a low ambient temperature, the oil becomes less fluid and the amber filter clogging alarm light may light up.
   The alarm light goes off as soon as the oil temperature rises. If the alarm light

remains lit, the filter is clogged.

The NC unit lights up and is ready to be initialized.

6. After "PRESS OPENING PEDAL" (Press the beam lifting pedal) - Press the pedal lifting the beam and keep it pressed until the movement will not completely stop.

7. Then the following message appears: "PRESS OPENING PEDAL" (Depress the beam down pedal): Depress the beam down pedal or the corresponding organ on the paired control panel (if selected). Keep pressed until the axes of the backgauge move. The CNC will show returning to home position of the X1 - X2 ... Z1 - Z2 axis.

8. When the general menu appears, the procedure ends.

ATTENTION: Before pressing "START" key, make sure the hazardous area is free of any obstacle. Furthermore, the loading of the worksheet should be carried out once the first positioning of the backgauge has been done.

Program selection switch in press brakes with system.
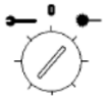
Setting mode. Working mode, the system is running.
0 – „Off". The press brake does not work.
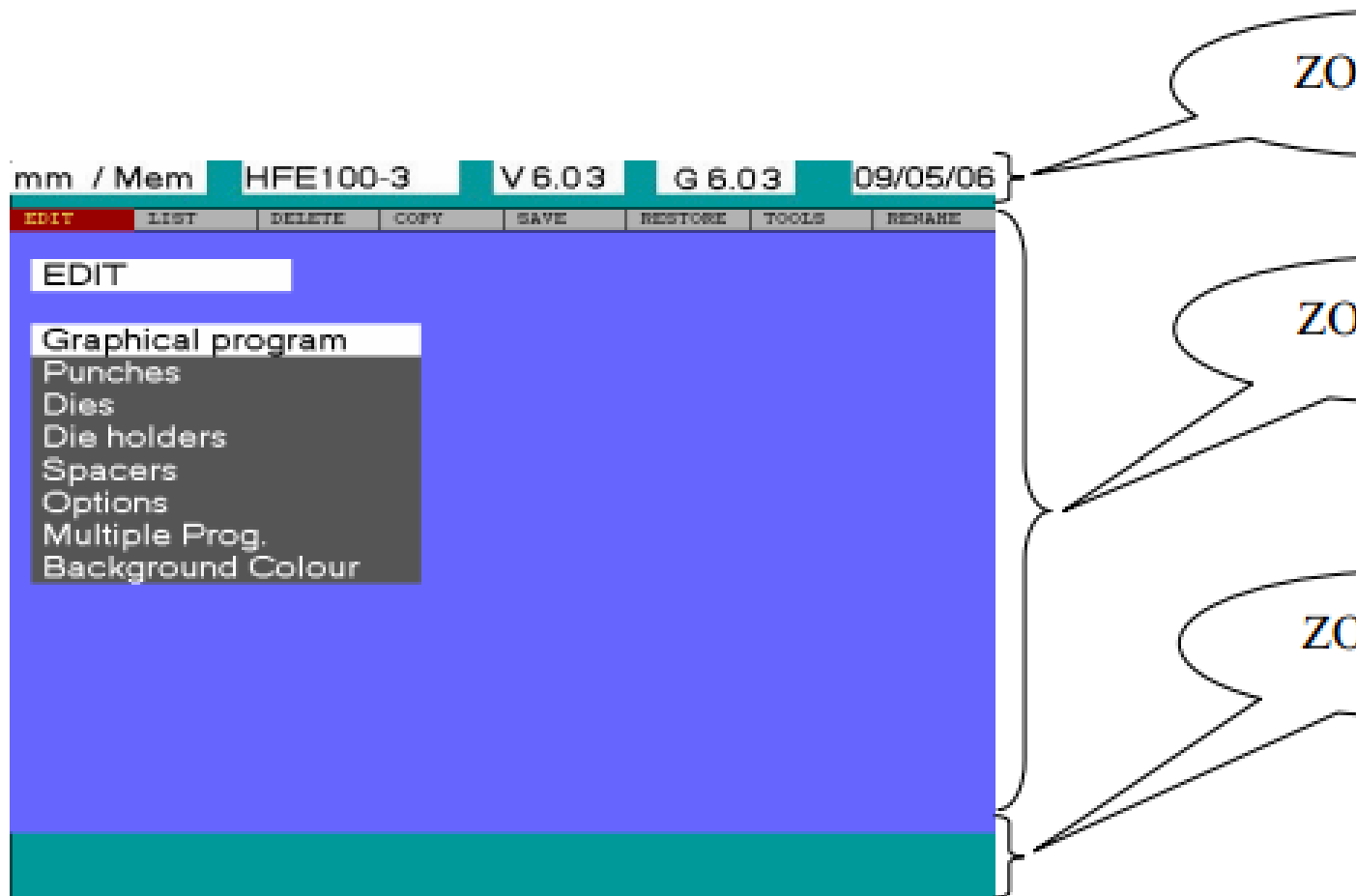
Deceleration lamp.

This lamp (blue) is used as an indicator to indicate the location of the deceleration point.
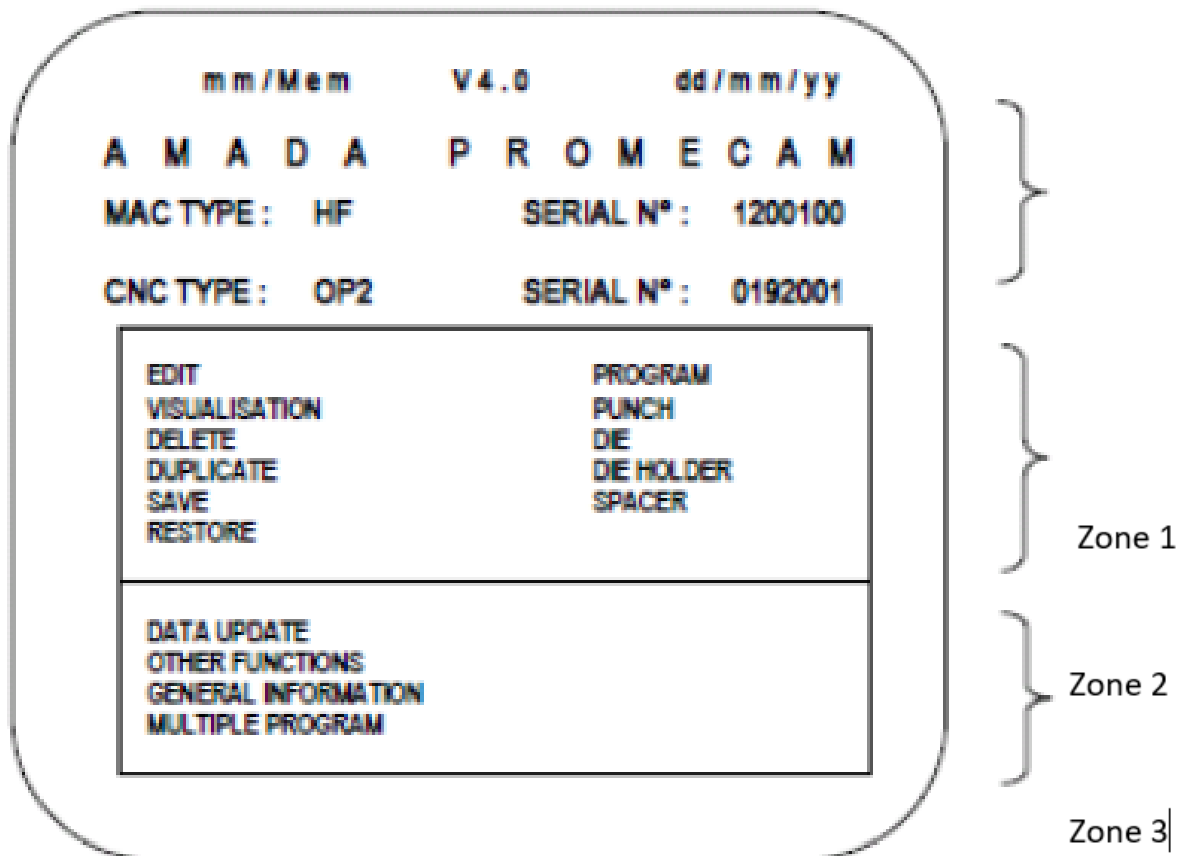
If this lamp is off and does not flash, then the laser is activated, the upper knife is above the deceleration point.

If the lamp is flashing, the bending mode is activated. Only the laser beam on the tool axis is on, the upper knife is above the deceleration point.

1.4. INTRODUCTION AND DESCRIPTION OF THE MENU PAGES

1.4.1. „GENERAL MENU" PAGE

mm / Mem	HFE100-3	V 6.03	G 6.03	09/05/06

EDIT	LIST	DELETE	COPY	SAVE	RESTORE	TOOLS	RENAME

EDIT

Graphical program
Punches
Dies
Die holders
Spacers
Options
Multiple Prog.
Background Colour

ZO

ZO

ZO

```
          mm/Mem        V4.0         dd/mm/yy

 A   M   A   D   A      P   R   O   M   E   C   A   M

 MAC TYPE :   HF              SERIAL N° :   1200100


 CNC TYPE :   OP2             SERIAL N° :   0192001
```

EDIT   PROGRAM
VISUALISATION   PUNCH
DELETE   DIE
DUPLICATE   DIE HOLDER
SAVE   SPACER
RESTORE

Zone 1

DATA UPDATE
OTHER FUNCTIONS
GENERAL INFORMATION
MULTIPLE PROGRAM

Zone 2

Zone 3

When the control panel is turned ON, the above main menu page is displayed on the NC screen, together with the different controls of the EDIT menu by default.

ZONE No. 1 Status line composed of the current measurement unit (mm or inch), current memory support. (FD: floppy disk - Mem: RAM – USB: USB key), and version of the software currently used, date, selected function and machine type.

ZONE No. 2 Working / dialog zone composed of:
- A line with 8 controls each, written in big letters, with their own scrolling menu depending upon the items concerned for each control and written in small letters.
- Whenever an item from the menu has been validated, a key in page of the different parameters is displayed over the whole zone.

ZONE No. 3 Messages and prompt zone for the operator.

Command functions

- EDIT: programming or modification of a graphical program, punches, dies, die-holders, spacers, options, multiple Prog., background colors, matrices, gaskets, matrix holder.
- VISUALIZATION - displays various libraries: programs, punches, dies, spacers or dies in memory.
- DELETE - deletes one or several programs, punches, dies, shims or matrix holders from the CNC memory.
- DUPLICATE - Copies the program stored in memory.
- SAVE - Writes programs from memory to a 3 ½ inch floppy disk either partially or completely.

- RESTORE - reads the contents of a 3 ½ inch floppy disk and transfers information to the internal memory of the CNC, either partially or completely, by choice.

  Command Function Objects

- PROGRAM - program library. Programs are arranged in numerical order by date of entry.
- PUNCH - punch library. Library of standard tools can be transferred from a 3 ”1/2 drive. Special tools can be created using the "EDIT", "PUNCH" function.
- DIE - matrix library. A library of standard tools can be transfer from drive to disk 3 ”1/2. Special tools can create using the functions "EDIT", "DIE".
- SPACER - spacer library. Library of standard tools can be transferred from a 3 ”1/2 disc. Special tools can create using the functions "EDIT", "SPACER".
- DIE HOLDER - library of matrix holders. Library of standard tools can be transferred from a 3 ”1/2 disc. Special tools can be created using the “EDIT”, “DIE HOLDER” functions.

ZONE No. 2

- DATA UPDATE - when you select this function, you access the display page with the name given above. This the page is used to compile the profile of the backgauge pin, defining the criteria for modeling the bending of the workpiece and changes the initial values of the various axes. This page allows you to select approach speed (maximum 99.9 mm / s), as well as a unit measurement (mm or inches).

```
DATA UPDATE                               mm / Mem              dd / m m / y y


   FINGER PROFIL        1              OFFSET Z / FLANGE END        50.0
      FINGER THICKNESS      5.0        UNITS mm → 1  /  inch →      1
      FINGER LENGTH        160.0       2                           100
      FINGER WIDTH          25.0       APPROACH SPEED
      SUPPORT OFFSET        50.0             AXES          PRESETS
      FINGER TOP OFFSET     10.0             Y1            451.00
      FINGER BOT. OFFSET    2.5              Y2            451.00
                                             X1            1045.00
   BEND SEQUENCE ASSIGNMENTS                 X2            1045.00
      MAX INTERMEDIATE BEND BG   12          R1            69.00
      TOLERANCE ON THE EDGE      15          R2            69.00
      TOLERANCE AT 90            15          Z1            1745.00
      FRONT EXTENDED RATIO       0.2         Z2            1754.00
      OPENING MAX = 0 MIN = 1    1
      O.D.C. INTO MUTE (1 = Y, 0 = N)  0
```

MATERIAL	BENDING COEF	UNFOLD COEF
STEEL	100 %	100 %
AG3	100 %	100 %
A5	100 %	100 %
STAINLESS	100 %	100 %

```
      MAXIIMUM VALUE OF Z1       1740
      MAXIMUM VALUE OF Z2        1740
      MUTE VALUE                 6
      DEFAULT DELTA X AUTO       1
      FD DRIVE (1 = A : 2 = B)   1
      ANGLE TOLERANCE            30 mn
      DATABASE USE (0/1/2)       2
```

- AXIS OFFSET: This offset value only applies to the Z axes with electric drive and is the same for both axes. It is positive or negative about the axis carriages (99.9 mm maximum with one single digit after decimal point) Below are various examples programming:

Carriage "Z"

offset "Z"

sheet for bending

Carriage "Z"

offset "Z"

sheet for bending
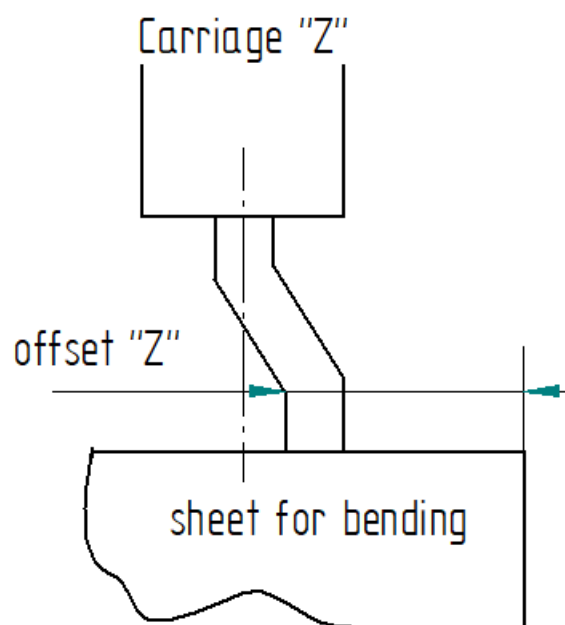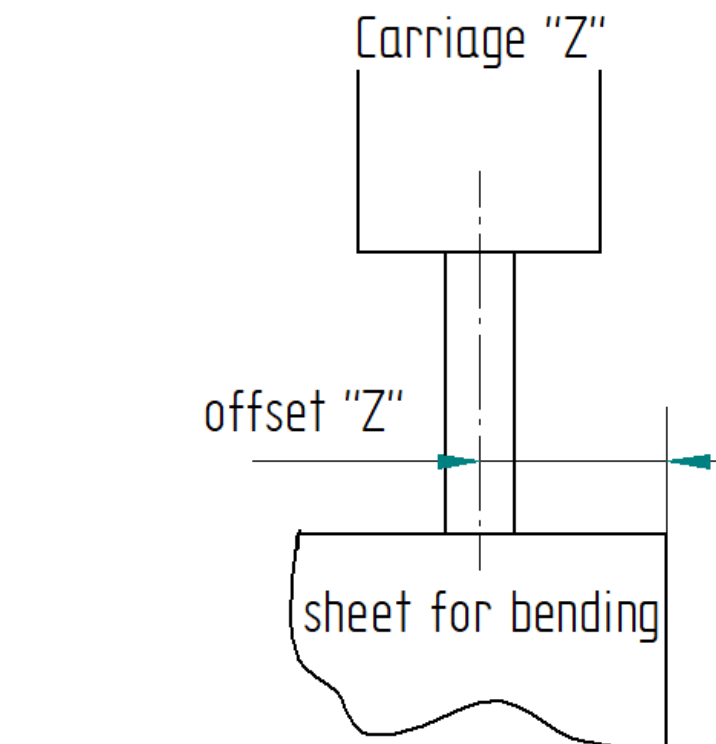
Example No. 1   Example No. 2   Example No. 3

Carriage "Z"

offset "Z"

sheet for bending
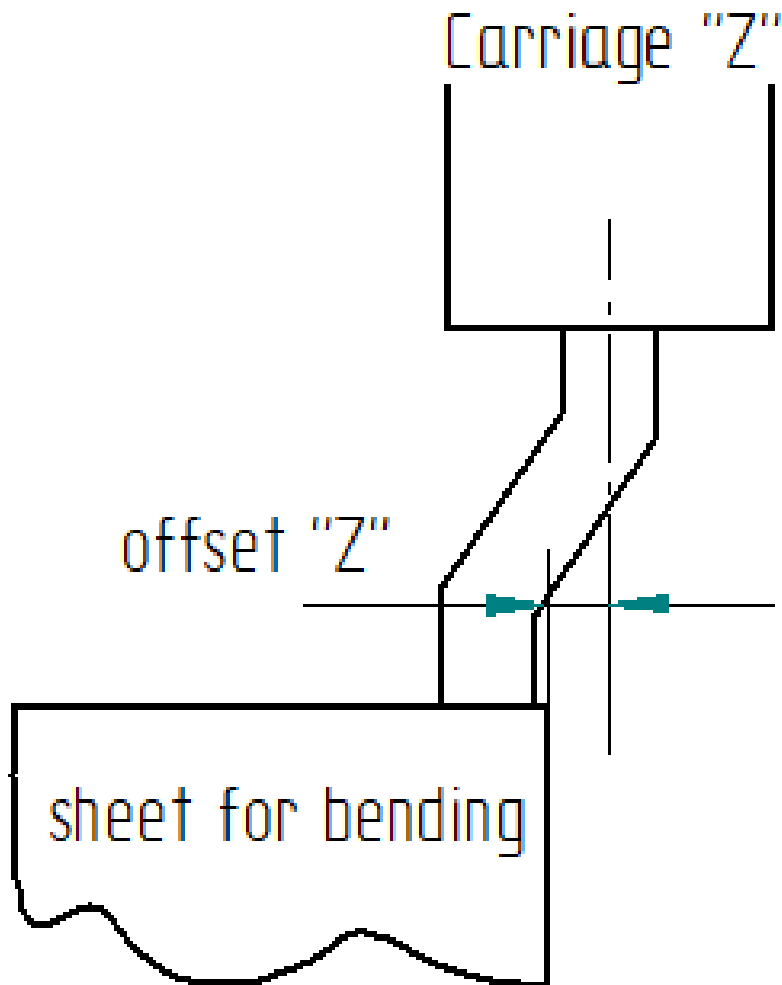
Example 1: Standard backgauge pin   = offset "Z" 50mm.
Example 2: Pin offset to RH   = offset "Z" 80mm.
Example 3: Pin offset to LH   = offset "Z" 10mm.
OTHER FUNCTIONS: This selection gives access to other functions menu page as shown below.

```
┌─────────────────────────────────────────────────────────┐
│  OTHER FUNCTIONS                    m m / M e m    d d / m m / y y │
│                                                         │
│       PROGRAMS DISK                                     │
│       PROGRAMS IN MEMORY                                │
│       SAVE PARAMETERS                                   │
│       RESTORE PARAMETERS                                │
│       LOAD PROFILES                                     │
│       CHANGE LANGUAGE                                   │
│       FLOPPY DISK FORMAT                                │
│       EXPORT PROGRAMS                                   │
│       IMPORT PROGRAMS                                   │
│       EXPLORE FD & COPY                                 │
│       LOAD PARAMETERS PROGRAM                           │
│       LOAD MAINTENANCE PROGRAM                          │
└─────────────────────────────────────────────────────────┘
```
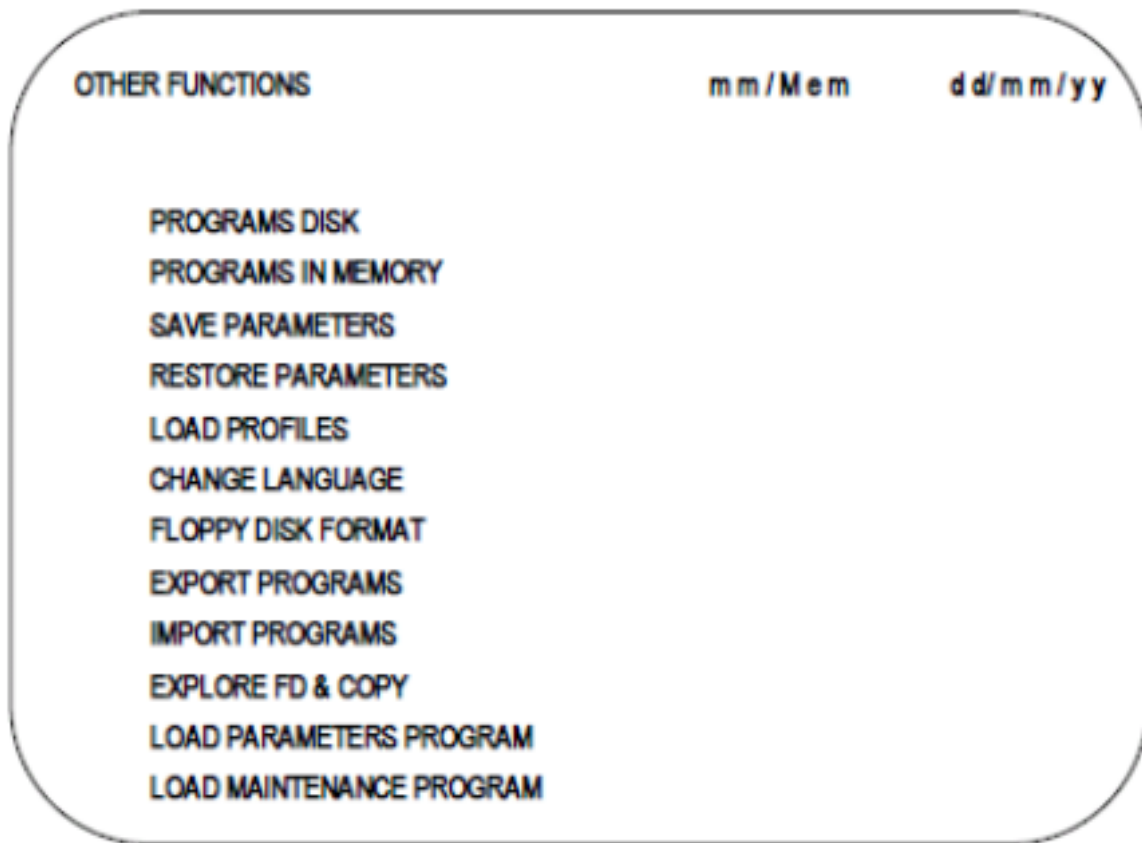
- PROGRAM DISK: This function allows you to you to carry out all your programs directly from the flexible disc 3 ½ inches.
- PROGRAMS IN MEMORY: function allows you to return to the CNC memory (after work) from a floppy disk.
- SAVE PARAMETERS: operation writing machine parameters to a floppy disk. These parameters are essential for the trouble-free operation of your machines and we advise you to keep them.
- READ PARAMETERS: the operation of reading the parameters of the machine to a floppy disk and writing from it to the CNC memory (machine parameters, profiles and language).
- LOAD PROFILES: read operation profiles from a floppy disk and recording them in the CNC memory.
- CHANGE LANGUAGE: allows you to select and load the language from the floppy disk into the CNC memory.
- FLOPPY DISK FORMATING: This operation allows you to prepare a new floppy disk for CNC. However, it is not intended exclusively for CNC and can be performed on any compatible PC.
- EXPORT PROGRAMS: this function allows you to select a directory for exporting programs and then transfer the selected programs from the CNC to the specified directory. The importing directory can accept programs up to this time as long as the capacity of the disk is full (floppy disk or main computer).
- IMPORT PROGRAMS: this function helps you choose the directory to import the program (with floppy disk or host computer) and then transfer the selected programs to the CNC import directory. The number of programs cannot exceed the NC memory size.
- LOAD PARAMETERS PROGRAM is used to access the machine parameters. A floppy disk "SYSTEME 2" is required to access the machine parameters.
- LOAD MAINTENANCE PROGRAM: maintenance and adjustment. To receive access to the service area requires a floppy disk "SYSTEME 2". This function also gives access to the summing disk, as well as functions of compensating parameters. Using a floppy disk language, it also gives access to all available languages.

GENERAL INFORMATION: gives access to the page general information, which contains information about the contents of the CNC and allows you to change the date and time of the CNC.

```
GENERAL NFORMATION                    m m / M e m        dd / m m / y y

GRILLE : GB V40007

                          EXISTING            MAXIMUM
                          QUANTITY            QUANTITY

        PROGRAM              12                  60

        DIE                  24                  70

        PUNCH                25                  30

        HOLDER                5                  10

        SPACER                6                  10

              dd / mm / yyyy              hh / mn / ss
        Date:  20 / 12 / 1993      Time:  12 / 10 / 30
```

1.4.2.   FUNCTION SELECTION

Gray ribbed line from left.

The menu horizontal allows to select the function to apply. With each function of the principal menu corresponds a vertical menu as indicated on the screen above.

| EDIT | **LIST** | DELETE | COPY | SAVE | RESTORE | TOOLS | RENAME |

LIST

> Programs
> Punches
> Dies
> D.holders
> Spacers
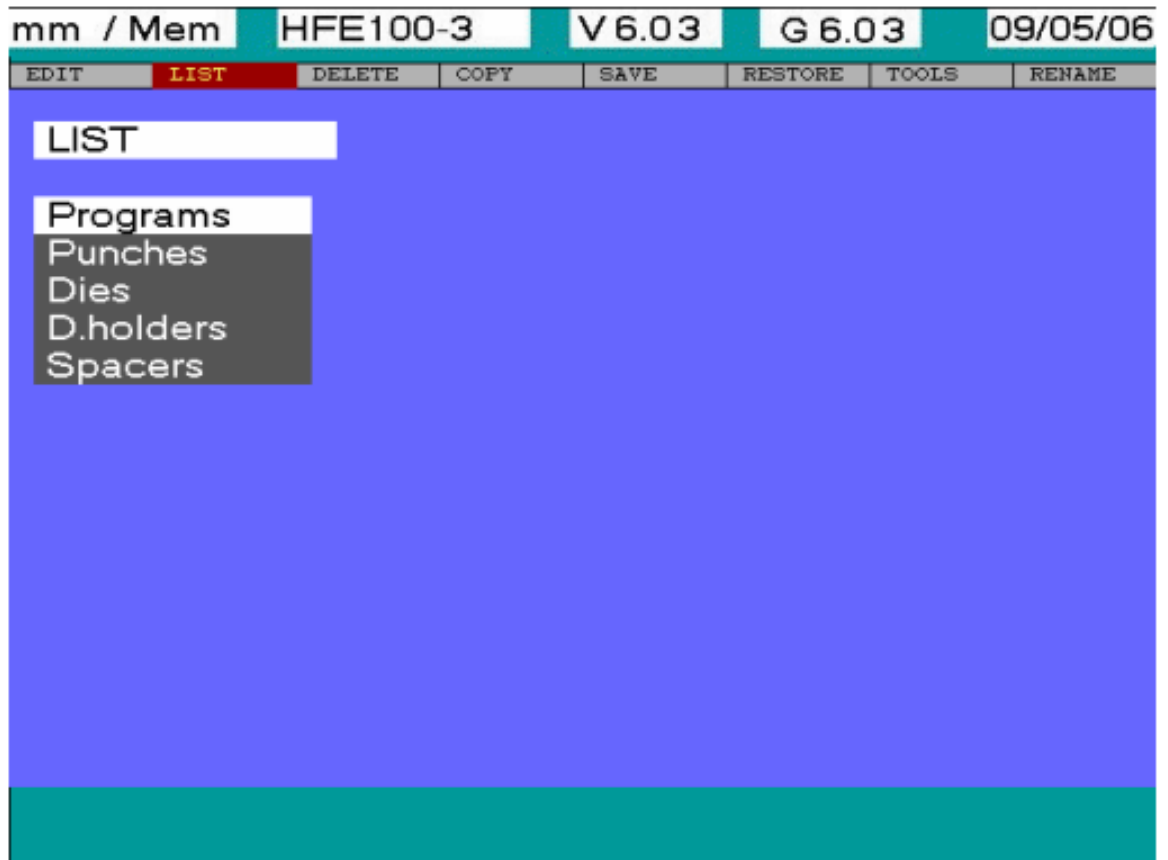
EDIT: Create or modify. The graphical program, punches, dies, die-holders, spacers, options, multiple Prog. and background color.

LIST: Display the different libraries of. The programs, punches, dies, die-holders and spacers.

DELETE: Erase one or several of the following. The programs, punches, dies, die-holders and spacers.

COPY: Copy one of the following. The programs, punches, dies, die-holders and spacers.

SAVE: Register partially or totally (depending upon the selection). The programs, punches, dies, die-holders and spacers and parameters.

RESTORE: Read either partially or totally (depending upon the selection). The programs, punches, dies, die-holders and spacers and parameters, profile and language.

To select an action it is enough to position using the arrows left and right on a function, and select the object which the function will be apply using arrows verticals. And Validate the object to execute the function.

| EDIT | LIST | DELETE | COPY | SAVE | RESTORE | TOOLS | RENAME |

TOOLS

Exporting programs
Importing programs
Formating F.D
Memory programs
F.D programs
Machine parameters
Maintenance
F.D Explorer & copy
Load ext. programs
Date and Time
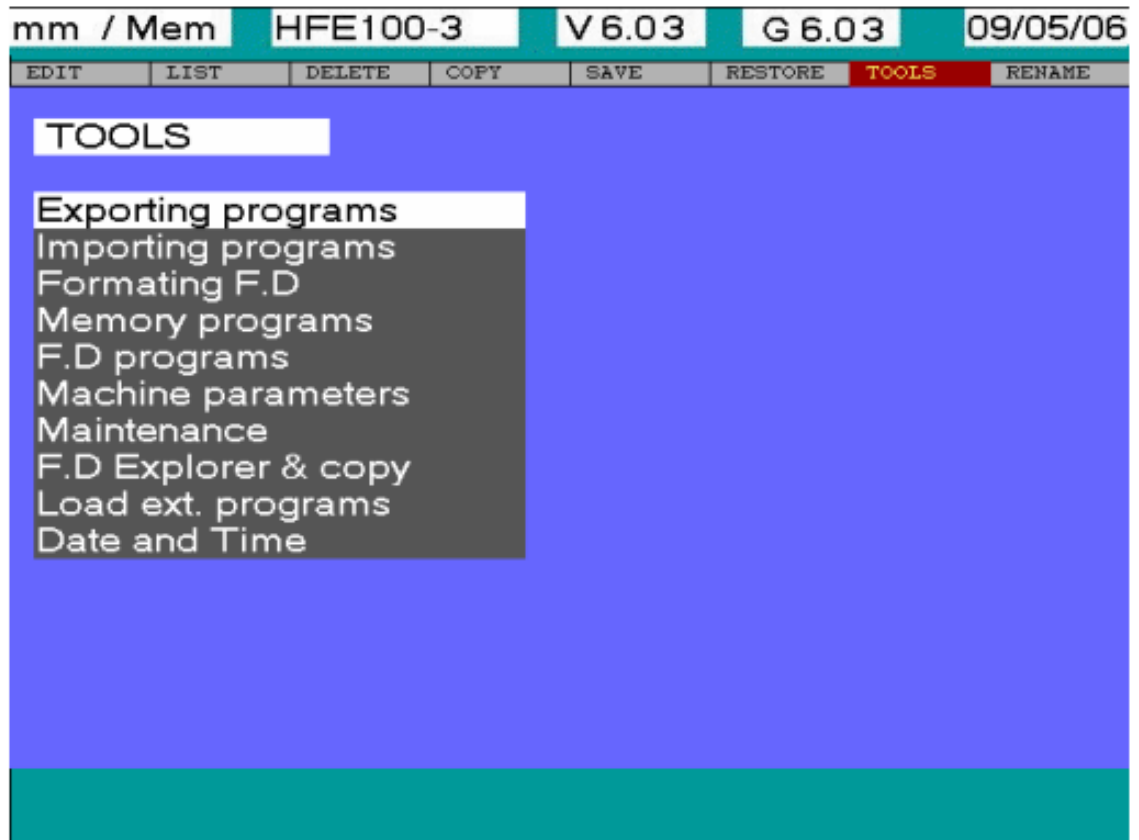
TOOLS: Several utilities functions. The exporting programs, importing programs, formatting F.D (floppy disk), memory programs, F.D (floppy disk) programs, machine parameters, maintenance, F.D explorer & copy, load ext. program and date and time.
RENAME: Rename one of the following. The programs, punches, dies, die-holders and spacers.

Objects for command functions:

PROGRAMS: Library of programs. They are classified in alphabetical order.

PUNCHES: Library of punches. A library of standard tools can be transferred from the external memory. Special tools can be created using the control's "EDIT" $\Longrightarrow$ "Punches" function.

DIES: Library of dies. A library of standard tools can be transferred from the external memory. Special tools can be created using the control's "EDIT" $\Longrightarrow$ "Dies" function.

DIE-HOLDERS: Library of die-holders. A library of standard tools can be transferred from the external memory. Special tools can be created using the control's "EDIT" $\Longrightarrow$ "Die-holders" function.

SPACERS: Library of spacers. A library of standard tools can be transferred from the external memory. Special tools can be created using the control's "EDIT" $\Longrightarrow$ "Spacers" function.

OPTIONS: Software options. Can be modified from the control. "EDIT" $\Longrightarrow$ "Options".

BACKGROUND COLOR: used to change the color on the 3 basic windows.

MULTIPLE PROG.: Programs composed themselves of several other programs. The multiple programs may be created from the "EDIT" ⟹ " Multiple prog. " function.

PARAMETERS: Machine parameters. They can be registered on the external memory with "SAVE" ⟹ "Parameters" function or restored from the external memory to the memory by using "RESTORE" ⟹ "Parameters" function.

PROFILES: Setting of the machine profiles and back gauge fingers. Use "RESTORE" ⟹ "profile" function to restore the profiles from the external memory to the NC memory. Profiles will be saved simultaneously with the parameters.

LANGUAGE: Language used in the NC. Use "RESTORE" ⟹ "Language" function to restore it from the external memory. Language will be saved simultaneously with the parameters + profiles.

EXPORTING "PROGRAMS, PUNCHES, DIES, DIE-HOLDERS, SPACERS": This function is used to register one or several items to the external memory or any other selected path by using "TOOLS" ⟹ "Export" function.

IMPORTING "PROGRAMS, PUNCHES, DIES, DIE-HOLDERS, SPACERS": This function is used to restore one or several items from the external memory or from any selected path, to the NC memory by using "TOOLS" ⟹ "Importing" function.

MEMORY PROGRAM: Function enabling to work directly with the NC internal memory by using "TOOLS" ⟹ "Memory program. " function.

DISK PROG: This function enables you to execute all your programs directly from external memory (inserted in the drive) by using "TOOLS" ⟹ "Disk Prog" function.

MACHINE PARAMETERS: Helps specifying to the NC different parameters related to the press-brake, by using "TOOLS" ⟹ "Machine parameters " function. (Valid for AMADA technicians only).

MAINTENANCE: Used to diagnose the origin of a material fault on the machine components by using "TOOLS" ⟹ "Maintenance" function. (Valid for AMADA technicians only).

DISK EXPLORER. & COPY: This operation enables you to go through and copy if necessary some items on the external memory by using "TOOLS" ⟹ " F.D. explorer & copy" function. (Valid for AMADA technicians only).

LOAD EXT. PROGRAM: Helps executing a program external to the NC, by using "TOLLS" ⟹ " Load Ext prog. ". (Valid for AMADA technicians only).

DATE AND TIME: Used to set the NC date and time by using "TOOLS" ⟹ "Date and time" function.

1.5. INSTALLATION - START-UP

Your CD2005/OP2002 numerical control comes with an external memory (USB key). The external memory can be inserted in the USB drive located in the upper part of the electrical cabinet, on the machine's sides as shown below.

NOTE: Insert the USB key before starting the machine, otherwise it will not be enabled by the NC.

DESCRIPTION OF THE USB KEY.

The USB key - as supplied with the machine as a replacement of all floppy disks previously provided with the machine – is used to:
1. INSTALL THE SOFT INTO THE NC
2. INSTALL THE NETWORK IN ACCORDANCE WITH THE USER'S NEEDS IF NECESSARY.
It also helps changing the NC language (English) by default and customizing the NC with the client's tools (Die, Punch, Holder, Spacer).

STRUCTURE OF THE SYSTEM KEY

It contains every system file necessary to the soft installation as well as files saved by the plant and specific to the client's machine.

The following folders are also included:

SYSTEM: files related to the bending soft of the machine

NETWORK: every file necessary to the network installation.

LANGUAGE: folders including languages available for the soft version.
TOOLS: every tooling folders (DIE, PUNCH, HOLDER, SPACER),
         Each file contains every tool available in Amada catalogue.

REMARKS: Change the language by:
- selecting function „TOOL, EXPLORE DISK AND COPY",
- validating „LANGUAGE" field,
- selecting the language to be transferred
- validating the transfer through „1" when requested: „CONFIRMATION
1=Y/0=N".
Customize the tools through „IMPORT TOOLS" function,
DIE for dies
PUNCH for punches
HOLDER for holders
SPACER for spacers

The soft installation contains tooling libraries by default.

1.6. MACHINE INITIALIZATION

Prior to any axis movement control, this initialization is necessary. Otherwise, message "FUNCTION PROHIBITED MACHINE RESET" will be displayed.

INITIALISATION PROCEDURE

- Turn on the main isolator switch of the machine.

- When the main menu is displayed on the NC screen:
  - Press the START key.

- If a message is shown: follow the instructions.
- INITIALISATION menu is displayed:
  - "PRESS OPENING PEDAL" (opening footswitch), press and maintain depressed the opening footswitch until all moving axes come to stand still.

  - At this stage, following message will be displayed: "PRESS CLOSING PEDAL". Press closing footswitch or optional two-hand push button control, if selected. Maintain footswitch depressed until the backgauge axis starts moving.

- Once "General Menu" appears on the screen, the initialization process is finished.

NOTE: Pressing the NC STOP key or any emergency stop interrupts the initialization process with all ongoing movements interrupted. The initialization is unfinished. Hence, it will be requested again by the NC.

1.7 PROGRAMMING

As previously stated, programming is possible in a variety of ways. The control system of the Amada HFE 80-25 bending machine allows use three programming methods: **MANUAL - 1, MANUAL - 2** and **GRAPHIC OR L-ALPHA PROGRAMMING**. Below about them in more detail.

MANUAL - 1 (simplified programming)

User enters axes positioning values or positions axle using the handwheel located near the operator's pedal. A test bend can be performed for each bend sequence.

MANUAL - 2 (angle programming) or X-ALPHA programming The CNC calculates the penetration depth of the tool, the required force for bending, backgauge position and finger height according to the programmed tool and the position of the stop fingers based on part length and position. For each of sequences can be tested.
Graphic or l-alpha programming

Used for graphical programming of a component from its drawing. The CNC gradually draws the part as the data (length and angle) is entered. It calculates the bending sequence, bending force, integration values, backgauge positioning, backgauge return, pin height back stop, the position of the stops in relation to the length of the part and their position on the machine, and MTP * (for each bend), according to a certain things are flexible.
* MTP - opening dead center.

1.7.1   PRELIMINARY NOTE CONCERNING PROGRAMMING

Irrespective of the programming mode used, "Y"-axis programming reference point is the sheet-pinch point (S.P.P.). The reference value will be:

S.P.P. = TH + SP + DH + DIE + PUNCH

where:

TH	- Thickness of the sheet to be bent, in mm.
SP	- Spacer height in mm
DH	- Die-holder height in mm
DIE	- Die height in mm
PUNCH	- Punch height in mm.

The "R"-axis zero for programming (except in MANUAL 1 mode) corresponds to the upper surface of the die with the negative values being located below and the positive ones being above the surface.
NOTE: In "MANUAL 1" or "simplified programming" mode, "R"-axis's height is programmed with respect to the press-brake table, considering that no tooling is programmed.

PUNCH

SH.THICKNESS

DIE

DIE-HOLDER

SPACER

PRESS-BRAKE
TABLE

R REFERENCE,
other modes

REFERENCE R,
MODE "MANUAL 1"
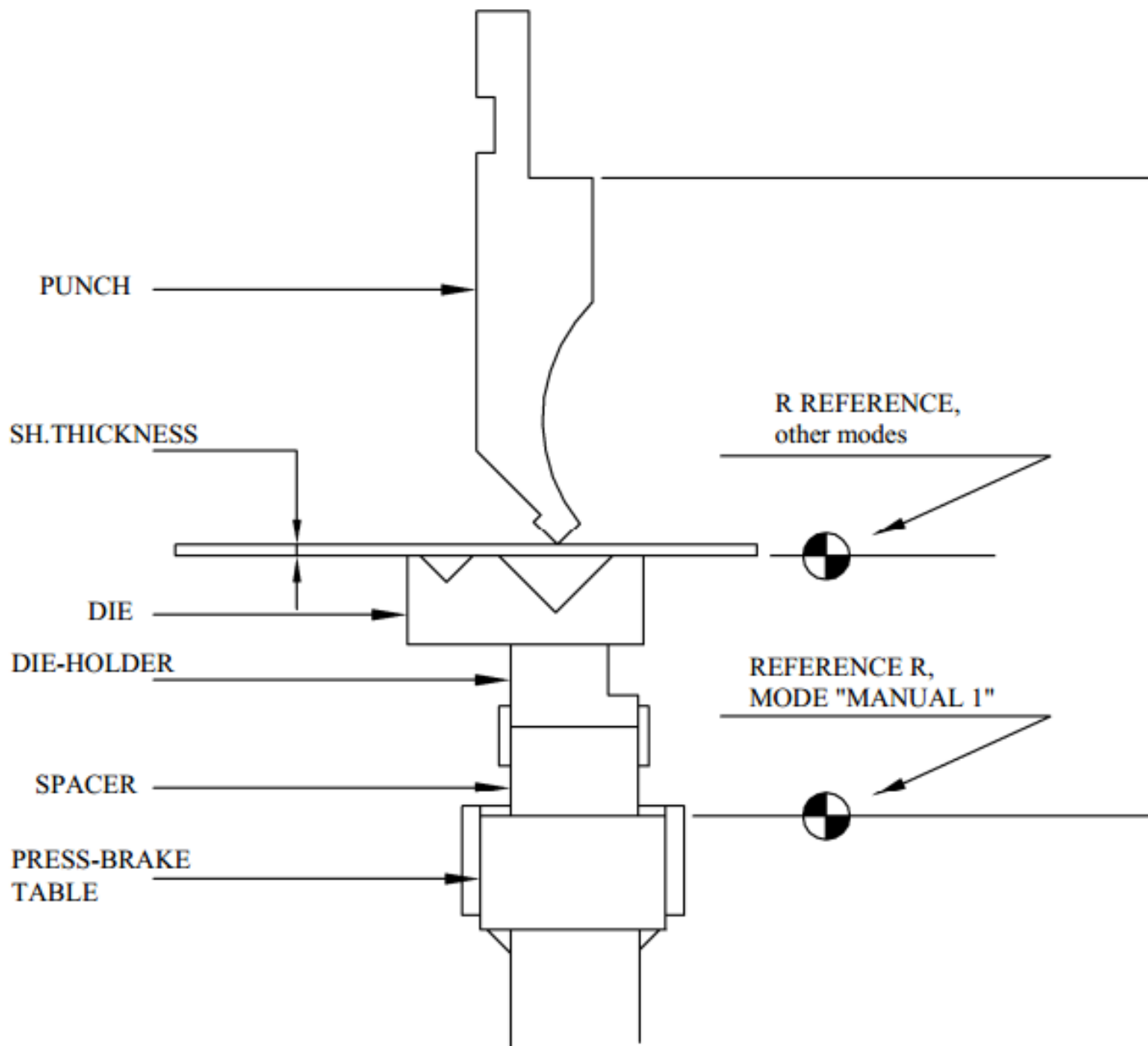
Figure

Programming the reference point for the "R" axis (except MANUAL-1) matches the following expression:

$$Zero\ R = RE + CA + DIE$$

In the "MANUAL 1" mode (simplified programming), we program the height of the R axis above the press table. In other modes, the R axis is taken as the axis, lying flush with the matrix.
The reference points of the Z axis are obtained from the diagram below (Figure below), while the zero point lies on the centerline of the press. Axis movement Z_1 to the left of the center line of the press, is taken as a positive value, and to the right of the center line of the press - a negative value. Axis movement Z_2 is viewed as moving the Z1 axis exactly the opposite.
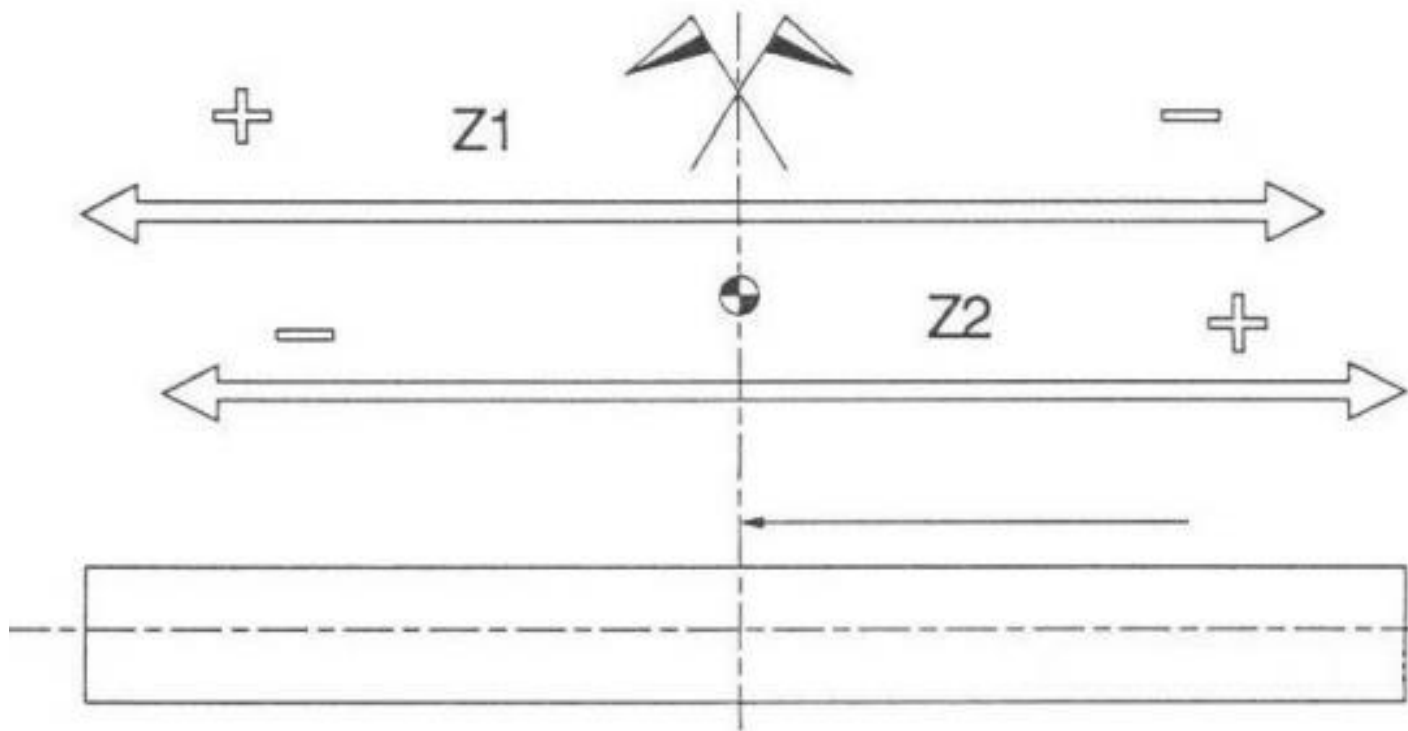
Bender center line

Bending machine table

Figure

The X and Y axes correspond to the zero point.

1.7.2   "MANUAL 1" simplified programming "PROGRAMMING MODE"

MANUAL 1 MODE (X,Y)

N° PROG   PART REFERENCE   PINCHING POINT

AXIS   MEM POS   REEL POSL

BEND   :

NB BND:

CYCLE   :
STEP   :
B.G HOLD. (s):
AUX FUNCTIONS:
COINING (0/1):

TONNAGE   (kN):
HOLD E.BEND s:
RET. DIST(mm):
W. RET(0/1/2):
WORK S.(mm/s):
RET. S (mm/s):
O.D.C   (mm):

28

```
MANUAL MODE 1        mm/Mem       dd/mm/yy


PROG N° :                 PART REF. :
PINCHING POINT :
..............................................................................
                         AXES       MEM POS        ACTUAL POS


BEND   :

BEND N :



......................................................................................
        CYCLE       :      TONNAGE (t)        :
        STEP        :      END HOLD.  (s)     :
B.B. HOLD.  (s)     :      RET. DIST. (mm)    :
AUX FUNCT.          :      WAIT RET  (0/1/2) :
  COINING (0 / 1)   :      B. SPEED (mm / s):
                           R. SPEED (mm / s):
                           O.D.C.       (mm) :
```

Zone 1

Zone 2

Zone 3

After initializing the machine with the "START" key ⌶ , activate "MANUAL 1" mode pressing ⧈ key. This will display the "MANUAL MODE 1" page as shown above.

ZONE No. 1

- Enter the program number (max. 6 digits). If the number is already used in another mode, the NC will generate an error message; otherwise the called-up program will be edited (see example below).
- Enter your work-piece reference (max 13 digits), this is an optional field.
- BEND N°: reference of the currently displayed bend.
- NB BEND: number of bends in the program.

- Enter the value of the S.P.P. according to the tools used and the sheet thickness (see paragraph "P.P." in the previous chapter) either by entering the value or by teaching the position using the 4 teaching keys.

- "Y" Value: If you don't know this value, confirm the default displayed zero then you can make your bend in teaching mode (see "TEACHING").

NOTE: The "Y"-axis values are set in respect to the S.P.P. You cannot program positive values: they correspond to an opening between the tools. Programming negative values would correspond to the tool penetration into the sheet.

- Program the desired value for the "X" and "R" axes for A-type backgauge, X1, X2, R1, R2 for B-type backgauge.

NOTE: A cursor movement either with the ⏎ key or one of the cursors moving keys will confirm the current value in that field as long as its format corresponds to the expected value. If the entered value is over the axis limit, this value will be recalculated by the NC.

- CYCLE (CY): Number of repetitions for a given cycle.
   1 = Bend cycle will be performed only once.
   n = Bend cycle will be performed "n" times (max. authorized value: 99 STEP: "X" axis distance will move in relation to its current position. This displacement can be programmed either positive or negative. The step is only active if the programmed cycle is greater than 1.

NOTE: The STEP will be executed "n" number of times. Control will check if the programmed STEP and the number "n" of CY can be performed in relation to the programmed value of the "X" axis. The value of CY will be corrected automatically if it exceeds the axis limits.

- B.G. HOLD. (s): Backgauge hold time (in seconds) before it will move to next bend position (max. 98.9 s) or if a value ≥ 99 is programmed, the backgauge will only move when operator presses the "S" key on the hand wheel control panel.

- AUX. FUNC.: There are four programmable auxiliary functions. They can be programmed in binary four digits code (Outputs F1; F2; F3 & F4).

   Examples: 000000 = None of the functions are active.
      010000 = Function F2 is active.
      110000 = Function F1 and F2 are active.
      - - - = …
      111000 = Functions F1; F2 and F3 are active.
      111100 = Functions F1; F2; F3 and F4 are active.
      000010 = Functions F5 active, press-brake opens at 10 mm/s if a robot is connected, otherwise to active mute stop.
      000001 = Functions F6 active, specific movement from the gauge i.e, R, Z then X, if a robot is connected, otherwise not active.

   ⚠ **CAUTION**: Function F4 controls a work-follower interface. If activated, the beam will stop at sheet pinch point.

- COINING (0/1): Selects coining mode for bending process; coining process requires 5 times the force used in air bending.
   0 = Working in air bending mode with coining inactive (End of bend in "Y" position reached).
   1 = Working in coining mode, folding right at the bottom of the die (end of bend with stop at the movement).

- FORCE (T): Bending force in Kilo Newton.
   In "Simplified Programming" mode, you must program the bending tonnage, this in accordance with the tools, the material, the length of the bend and the thickness of the sheet being used (see bending chart on machine's electric cabinet).

- EOB HOLD (s): Under pressure holding time (max 98.9 s).
   If default value is accepted, the press will hold part under pressure for 200 ms.
   If a value of 99 is programmed, the beam opening is only achieved at end of bend when

opening footswitch is activated.
If zero is programmed, bend accuracy problem may be encountered due to lack of under pressure holding time.

- RET. DIST (mm): The distance backgauge should retract at S.P.P. (sheet pinch point).

- (WAIT) RET. (0/1/2): Backgauge retract conditions.
  0 = If backgauge retract is programmed, backgauge will retract simultaneously with the start of bending.
  1 = If backgauge retract is programmed, beam will reach S.P.P., backgauge will retract to programmed position then bend will start.
  2 = with CY > 1
      - and backgauge retract, if RET. DIST. programmed.
      - the backgauge retract will only be executed at the end of all programmed repetitions.

- B. SPEED (mm/s): Working speed of the "Y"-axis from 1 to maximum speed between S.P.P. to the end of bend.

- R. SPEED. (mm/s): "Y"-axis return speed, from end of decompression through to S.P.P. in slow speed then 1 to maximum speed. A zero value causes fast opening right from the end of decompression.

- O.D.C. (mm): Open dead center of the "Y"-axis. Beam opening distance in mm after executing the current bend cycle. This distance is in relation to the S.P.P.

  To program another bend, simply press ➡ key. All the information on the current page will be copied to a new page. Then you can modify the desired values to build your program.

  Display your program bends with keys ➡ or ⬅.

⚠ **WARNING**: If you press this ➡ key while at the last bend of the program ⇒ you will add an additional bend following the current bend with its parameters copied.

When your program is done, press the ⊟ key to save the program.

Keys ⊡ and ⊡ are active at all times so that you can insert or delete a bend. All the keys used to quit the programming modes will save the program currently edited.

1.7.3. "MANUAL 2" MODE : "ANGLE PROGRAMMING" OR "X-ALPHA" ⊞

MANUAL 2 MODE (X,alpha)

N° PROG:_____   PART REF.:

TH   SPACER D.H.   DIE   R PUNCH   R MAT.   Õ

                ANGLE=   MEM POS   REEL POS
LG.PLI:

PART OF:

BEND   :

BEND N:

CYCLE   :   END HOLD..(s):
STEP/X   :   RET. DIST(mm):
COINING (0/1):   W. RET(0/1/2):
TONNAGE (KN) :   W.SPEED(mm/s):
HLOD B.G.(s):   R.SPEED(mm/s):
AUX. FUNCTION: O.D.C   (mm):
AUTO dX(1/0) :

- Enter the program number (max. 6 digits). If the entered number already exists in another programming mode, the N.C. will indicate an error, otherwise the called-up program will be edited (see below).

MANUAL 2 MODE (X,alpha)

N° PROG: 2   PART REF.: [   ]

TH	SPACER	D.H.	DIE	R	PUNCH	R	MAT.	Õ
1.00	1	1	9001	0	9001	0	STEEL	40

ANGLE = 90.0

LG.PLI:
119

PART OF:
0

BEND   : 1

BEND N: 2

	MEM POS	REEL POS
Y1	-2.26	169.35
Y2	-2.26	169.35
X1	488.97	712.00
X2	488.97	712.00
R1	0.07	-106.00
R2	0.07	-106.00
Z1	59.99	1230.00
Z2	60.01	1230.00

CYCLE	:1	END HOLD..(s):	0.2
STEP/X	:0	RET. DIST(mm):	0
COINING (0/1)	:0	W. RET(0/1/2):	0
TONNAGE (KN)	:142	W.SPEED(mm/s):	10
HLOD B.G.(s)	:0	R.SPEED(mm/s):	0
AUX. FUNCTION	:000000	O.D.C (mm):	169
AUTO dX(1/0)	:1		

Use keys [→] or [↓] in the blank field to display all the programs in memory and select an existing program (which has been created in "Manuel 2" mode) with [→][←][↑][↓] keys, then confirm with key [↵].

- Enter your work-piece reference (max. 13 digits), this is an optional field.
- TH: thickness of the material to be bent in mm.
- SP(ACER): Reference of the spacer. Use keys [→] or [↓] in the blank field to display all the spacers in memory and select the spacer existing on your machine with keys [↑][↓], then confirm with key [↵] on the selected spacer. Go back to "MANUAL 2" programming page.
    If "0" is programmed = no spacer.
- DH: Reference of the die holder. Follow the same procedure as for the spacer to select the die holder fitted on your tooling.
    If "0" is programmed = no die-holder.
- DIE: Enter the reference of the current die. Select the die to be used on your tooling similarly to the die-holder.
- R: Parameter indicating reverse mounting of the die.
    0 is programmed to indicate = die is mounted normally.
    1 is programmed to indicate = die is reversed.

NOTE: This only applies to two or four V dies, as they are the only ones that can be reversed. Turning the die around also influence NC X-axis safety calculations.

- PUNCH: Reference of the punch used. Follow the same procedure for the spacer to select the punch to be used on your tooling.
- R: Parameter indicating reverse mounting of the punch.
    - 0 is programmed to indicate = punch is mounted normally.
    - 1 is programmed to indicate = punch is reversed.
  NOTE: The punch-reverse option is only used as a reminder in MANUAL 2 mode.

- MAT.: type of the material to be bent. Use ↓ or ↑ key to select - among the materials stored in memory - the type of material to be bent, then confirm with key ↵ .
- Õ: standard strength displayed once the material has been selected. Should the strength be different, enter the tensile strength of material to be used in DaN/mm².
- BEND No: Indicates the bend number currently displayed on the screen.
- NB BEND: Indicates total number of bends in the program.
- BEND LGTH: Length in mm of the part to be bent.
- PART OFFSET: Part offset in respect to machine centre-line in mm (used for multi-stage bending or when bending under horns). Input zero if you are folding your job in the centre of the machine.

  NOTE: The default value is zero.

- ANGLE: Program value of the angle to be achieved.
  SPECIFIC CASE: Entering a zero (0) angle value will give access to the teaching mode for "Y" value.

- "Y" VALUE: This is calculated as soon as data for $\propto$ is fed in and confirmed. To gain access to "Y" value field, you must enter ZERO for $\propto$ (teaching mode).

  NOTE The "Y" axes values are set up in respect to the P.P. You cannot program a positive value in this field as they would correspond to an opening between the tools. Programming negative values corresponds to the penetration of the tool into the material. Identical to MANUAL 1 mode, programming of double axes (Y1, Y2…) can be carried out in two ways:
    - 1) If the values are identical, use CR key.
    - 2) If the values are different, then use the down arrow key.

- "X" VALUE: In this field, program external flange length if the sheet thickness correction for butting is adjusted in external values mode (see "dX AUTO" developed further down). Otherwise program the flange length after deducting bend allowance.
- "R" VALUE: The "R" axis zero reference is the top of the die (see paragraph PRELIMINARY NOTE ON PROGRAMMING).

- CYCLE (CY): Number of repetitions for a given cycle.
    - 1 = Bend cycle will be performed only once.
    - n = Bend cycle will be performed "n" times (max. authorized value: 99).

  NOTE Only by programming a cycle greater than 1 enables you to program the STEP.

- STEP: "X"-axis distance will move in relation to its current position. This displacement can be programmed either positive or negative. The step is only active if the programmed cycle is greater than 1.

  NOTE: The STEP will be executed "n" number of times. Control will check if the programmed STEP and the number "n" of CY can be performed, this in relation to the programmed value of the "X" axis. The value of CY "n" will be corrected automatically if it exceeds the axis limits.

- COINING (0/1): Selects coining mode for bending process; coining process requires 5 times the force used in air bending.
  - 0 = Working in air bending mode with coining inactive (end of bend in "Y" position reached).
  - 1 = Working in coining mode, folding right at the bottom of the die (bottoming with stop at movement).

- FORCE (T): Bending force in Kilo Newton. The bending tonnage will be calculated taking in to account sheet position on the beam, the programmed bend angle, the tools, the material, the length of the bend and the thickness of the sheet being used and type of bending mode (air bending, coining).

- B.G. HOLD. (s): Backgauge hold time (in seconds) before it will move to next bend position (max. 98.9 s) or if a value > 99 is programmed, the backgauge will only move when operator presses the "S" key on the footswitch control panel.

- AUX. FUNC.: There are four programmable auxiliary functions. They can be programmed in binary four digits code (Outputs F1; F2; F3 & F4).

- Examples: 000000 = None of the functions are active.
  - 010000 = Function F2 is active.
  - 110000 = Function F1 and F2 are active.
  - - - - = …
  - 111000 = Functions F1; F2 and F3 are active.
  - 111100 = Functions F1; F2; F3 and F4 are active.
  - 000010 = Functions F5 active, press-brake opens at 10 mm/s if a robot is connected, otherwise to active mute stop.
  - 000001 = Functions F6 active, specific movement from the gauge i.e, R, Z then X, if a robot is connected, otherwise not active.

⚠ **CAUTION**: Function F4 controls a work-follower interface. If activated; the beam will stop at pinch point (at contact with the sheet).

- dX AUTO (0/1): Bend allowance calculation for the worksheet thickness
  - 1 = outside dimensions
  - 0 = inside dimensions

- EOB HOLD (s): Under pressure holding time (max 98.9 s).
  If default value is accepted, the press will hold part under pressure for 200 ms.
  If a value of 99 is programmed, the beam opening is only achieved at end of bend when opening footswitch is activated.

If zero is programmed, bend accuracy problem may be encountered due to lack of under pressure holding time.

- RET. DIST (mm): The distance backgauge should retract at P.P. (sheet pinch point) before bend can start.

- (WAIT) RET. (0/1/2): Backgauge retract conditions.
  - 0 = If backgauge retract is programmed, backgauge will retract simultaneously with the start of bending.
  - 1 = If backgauge retract is programmed, beam will reach P.P., backgauge will retract to programmed position then bend will start.
  - 2 = with cycle > 1 and backgauge retract at the end of repetition:
    the backgauge retract will only be executed at the end of all programmed repetitions.

- W. SPEED (mm/s): Working speed of the "Y"-axis from 1 to maximum speed between P.P. to the end of bend.

- R. SPEED. (mm/s): "Y"-axis return speed, from end of decompression through to P.P. in slow speed then 1 to maximum speed. A zero value causes fast opening right from the end of decompression.

- O.D.C. (mm): Open dead center of the "Y"-axis. Beam opening distance in mm after executing the current bend cycle. This distance is in relation to the P.P.

After programming all the fields, you obtain a program for the bend similar to the one shown below.
The continuation of programming process is identical to that of MANUAL 1 mode:

- Program another bending sequence using ➡️ key.
- Modify the values of ANGLE,"Y", "X" or "R", as well as the parameters valid in the second programming page if necessary.

Display your program bends with keys ➡️ or ⬅️ .

⚠️ **WARNING**: If you press this ➡️ key while at the last bend of the program ⇒ you will add an additional bend following the current bend with a copy of all its parameter
,rs.

When your program is done, press the FIN/END key to save the program.

DEL and INS keys are active at all times so that you can insert or delete a bend.

1.7.4. SEMI-AUTOMATIC MODE ⟩⟩

"SEMI-AUTOMATIC" mode is accessed by pressing ⟩⟩ key.

Before entering the "SEMI-AUTOMATIC" mode, you will get an intermediate page displayed on the screen as shown below, containing the list of the programs saved in memory (NC or external) together with the name of the program previously edited and displayed as soon as it is automatically selected.

Validation with ⏎ key allows you to enter directly the semi-automatic mode on the program which you have just entered. Another possible way to execute this program directly would be to use "START" key.

This page also enables you to choose another program, which is already stored in the NC or external memory by selecting this program within the library displayed on the screen.

- Select the desired program using the → ← ↑ ↓ keys, then confirm your choice to execute the program in SEMI-AUTOMATIC mode.

  The "SEMI-AUTOMATIC" mode is an intermediate step between programming and producing the work -piece. This mode allows you to make small corrections to your program.

SEMI AUTOMATIC MODE

PROGRAM NUMB:22   PART REF.   :
BEND NUMBER :1 /5   BEND LENGTH   :
ANGLE   :90.0   DEVL. LENGTH   :1491.44
CYCLE   :1

CORRECTION	ANGLE	Y1	Y2	X1	X2	R1	R2	Z1	Z2
PROGRAM	0	0	0	0	0	0	0	0	0
BEND	0	0	0	0	0	0	0	0	0

AXIS	MEM. POS	REEL POS
Y1	-2.26	169.35
Y2	-2.26	169.35
X1	199.14	712.00
X2	199.14	712.00
R1	-2.00	-106.00
R2	-2.00	-106.00
Z1	60.00	1230.00
Z2	60.00	1230.00

Several information related to the program are displayed in the upper part of the screen. It is only possible to edit the number of the bend field in this zone. There are two methods available for you to select the bend you intend to correct.

1) Browse through your program using cursor → ← ↑ ↓ keys.

2) With the cursor at the SEQ. N° field directly input the bend number and confirm by pressing the ↵ or ↓ key, the called-up bend will be displayed on the screen.

"CORRECTION", "PROGRAM" and "BEND" are used to correct different axes. The ALPHA correction field is not accessible for programs created in the MANUAL 1 mode. The use of other correction fields remains identical whatever the programming mode selected.

In this zone, the displayed bend can be changed using page ➡ or ⬅.
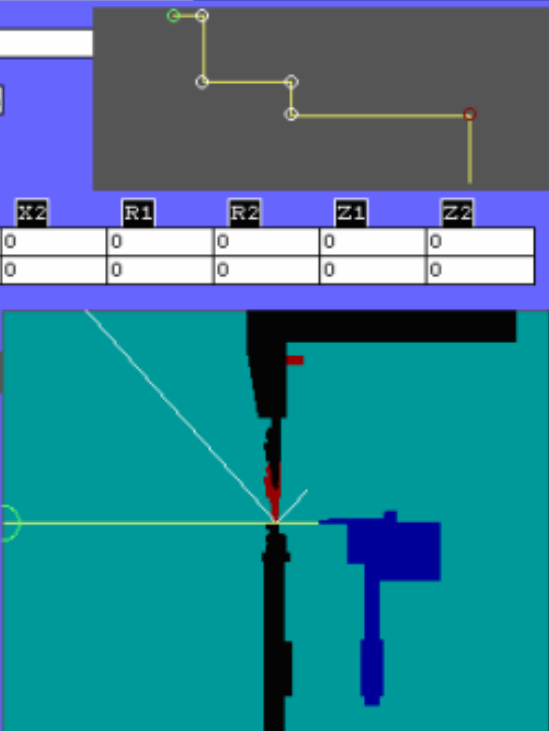
mm / Mem   HFE100-3   V 6.03   G 6.03   09/05/06

SEMI AUTOMATIC MODE

PROGRAM NUMB:22
BEND NUMBER :1 /5
ANGLE   :90.0
CYCLE   :1

PART REF.   :
BEND LENGTH   :
DEVL. LENGTH :1491.44

CORRECTION	ANGLE	Y1	Y2	X1	X2	R1	R2	Z1	Z2
PROGRAM	0	0	0	0	0	0	0	0	0
BEND	0	0	0	0	0	0	0	0	0

AXIS	MEM. POS	REEL POS
Y1	-2.26	169.35
Y2	-2.26	169.35
X1	199.14	712.00
X2	199.14	712.00
R1	-2.00	-106.00
R2	-2.00	-106.00
Z1	60.00	1230.00
Z2	60.00	1230.00

- RULE No 1: The maximum permissible value of correction, positive or negative, is 99.999-mm.

- RULE No 2: After input, as soon as the cursor is moved off the program line, the corrections applied to an axis will be accumulated (algebraic sum) and transferred to the bend line.

- RULE No 3: The corrections applied to an axis on the program line of a bend are passed on to all the bends having the same angular value within the program.

- RULE No 4: The corrections applied to an axis on the bend line cancel out and replace previously entered corrections.
  The lower part of the screen is used to visualize axes:

- "MEM. POS": this column includes all the corrections applied to a programmed position.
- "ACT. POS": this column shows the actual axes position (where the axes are at a given time).

1.7.5. AUTOMATIC MODE

The access to "AUTOMATIC" mode is done by pressing the ➡️ key.

Before entering the "AUTOMATIC" mode, you will get an intermediate page displayed on the screen as shown below, containing the list of the programs saved in memory (NC or external) together with the name of the program previously edited and displayed as soon as it is automatically selected.

Validation with ⏎ key allows you to enter directly the automatic mode on the program which you have just entered. Another possible way to execute this program directly would be to use "START" key.

AUTOMATIC MODE

PROG N° :22	REF. N° :	
S/P N° :1 /1	QTY S/P:0 / 1	
S/P NAM:22	REF NUMB:	

BEND N°:1 /5	REQUIRED QTY :9999
ANGLE :90.0	EXECUTED QTY :0
CYCLE :1	THICKNESS :1.00
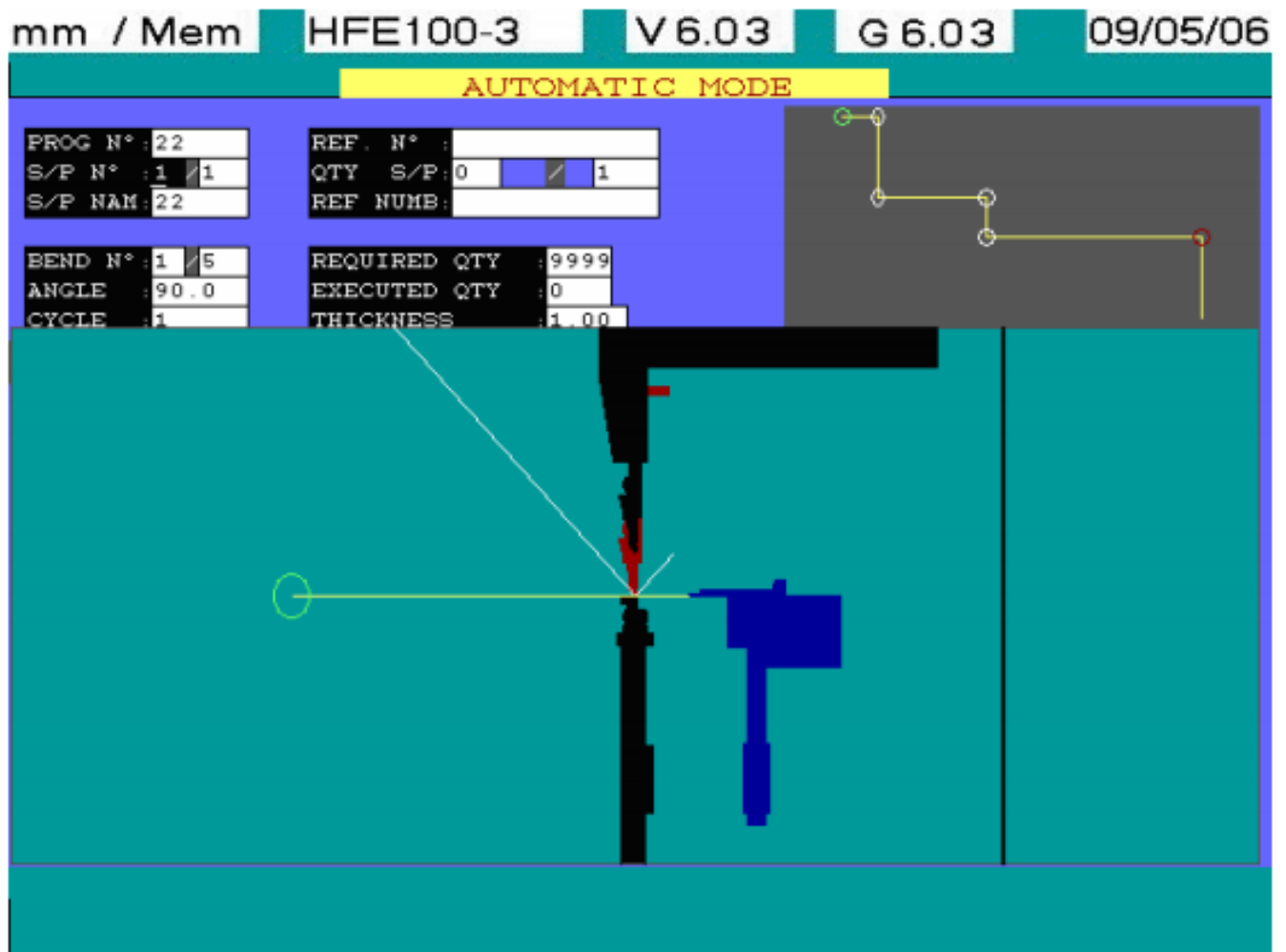
This page also enables you to choose another program, which is already stored in the memory (NC or external) by selecting this program within the library displayed on the screen.

- Select the desired program using the [→][←][↑][↓] keys, then confirm your choice to execute the program in AUTOMATIC mode.

42

The upper part of the screen displays some information related to the program. The following values may be edited:

- "N° S/P" allows selecting the current sub-program (if the edited program is a multiple one). If not, a "N° S/P" value = "1/1". is then available and cannot be modified.

- "SEQ N°" enables the selection of the current bend (which belongs to the current subprogram).

  Editing the S/P number and the bend number is accomplished in the same way as for the sequence number in the SEMI-AUTOMATIC mode.

- REQUESTED QTY: Place the cursor in this field and enter the number of work-piece required.
- EXECUTED QTY: Enter zero if none of the parts have been produced or the number of finished parts if some are already produced.

  NOTE: During the bending operation, the EXECUTED QTY counters increments automatically for each finished part. When EXECUTED QTY = REQUESTED QTY, the N.C. comes out of the AUTOMATIC mode and bending operation stops. The NC displays once again the intermediate AUTO page on the screen to allow eventual selection of another program for execution if required.

Zone for visualizing axes position:
- MEM. POS.: this column includes all the corrections applied to a programmed position.
- ACT. POS.: this column shows the actual axes position (where the axes are at a given time).
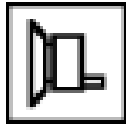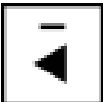
1.7.6. TEACHING MODE

Press [icon] key to have access to the TEACHING MODE. The data input is only possible if the cursor is positioned to the "PP" value or on one of the "Y1", "Y2", "X" or "R" axis value while in mode "MANUAL 1" or "MANUAL 2". When the teaching mode is activated the selected axis appears on the screen in reverse video, moving from column "MEM. POS." to "ACT. POS.", as well as the PP value.

Once the TEACHING MODE has been validated, use the following keys to modify these values:

- [icon] allows an increase of the value (axis in the positive direction) in slow speed.

- [icon] allows an increase of the value (axis in the positive direction) in high speed.

- [icon] allows a decrease of the value (axis in the negative direction) in slow speed.

- [icon] allows a decrease of the value (axis in the negative direction) in high speed.

  Depressing this [icon] key again will quit the TEACHING MODE. The edited value will be saved by copying the value of the modified axis from "ACT. POS." to "MEM. POS". Quit the teaching mode without replacing actual axis values, press and hold [icon] key, then press [icon].

  TEACHING OF THE PINCH POINT IN MANUAL 1 MODE

  1. Place the cursor at the P.P. value (PINCH POINT)

  2. Press [icon] key to change over to the "TEACHING MODE", the PP value will be shown in reverse video.

  3. Place a sample sheet, which is to be bent in between the tooling. 4. Press the closing pedal:

     the beam does not move. 5. Press one of these [icons] keys; the beams will

close.

6. Bring down the tool tip in contact with the worksheet. The work sheet should be pinched tightly in between the tools without being folded and you should not be able to move the worksheet, this means that you achieved the correct pinch point position.
7. Once you achieved satisfactory P.P., then press again the key on the pedestal control panel to save the beam position. You will automatically quit the "TEACHING MODE" and the cursor will move to the next field.
8. Release the closing pedal.

TEACHING A DOUBLE AXIS (Y1 AND Y2) WITH A SIMILAR VALUE.

Entering the "TEACHING MODE" when the cursor is located on one of the two axes of the same type (Y1; Y2; …), will teach the same value to both axes ("Y1" and "Y2") unless modified intentionally when entering different values to the keypad. The procedure for teaching "Y" value is identical to the one for pinch point. The procedure for teaching a backgauge axis is similar to the one described for teaching the P.P. with the exception that pressing the closing pedal is not necessary for moving the selected axis.

1.8. EDIT

1.8.1. L ALPHA OR GRAPHICAL PROGRAMMING
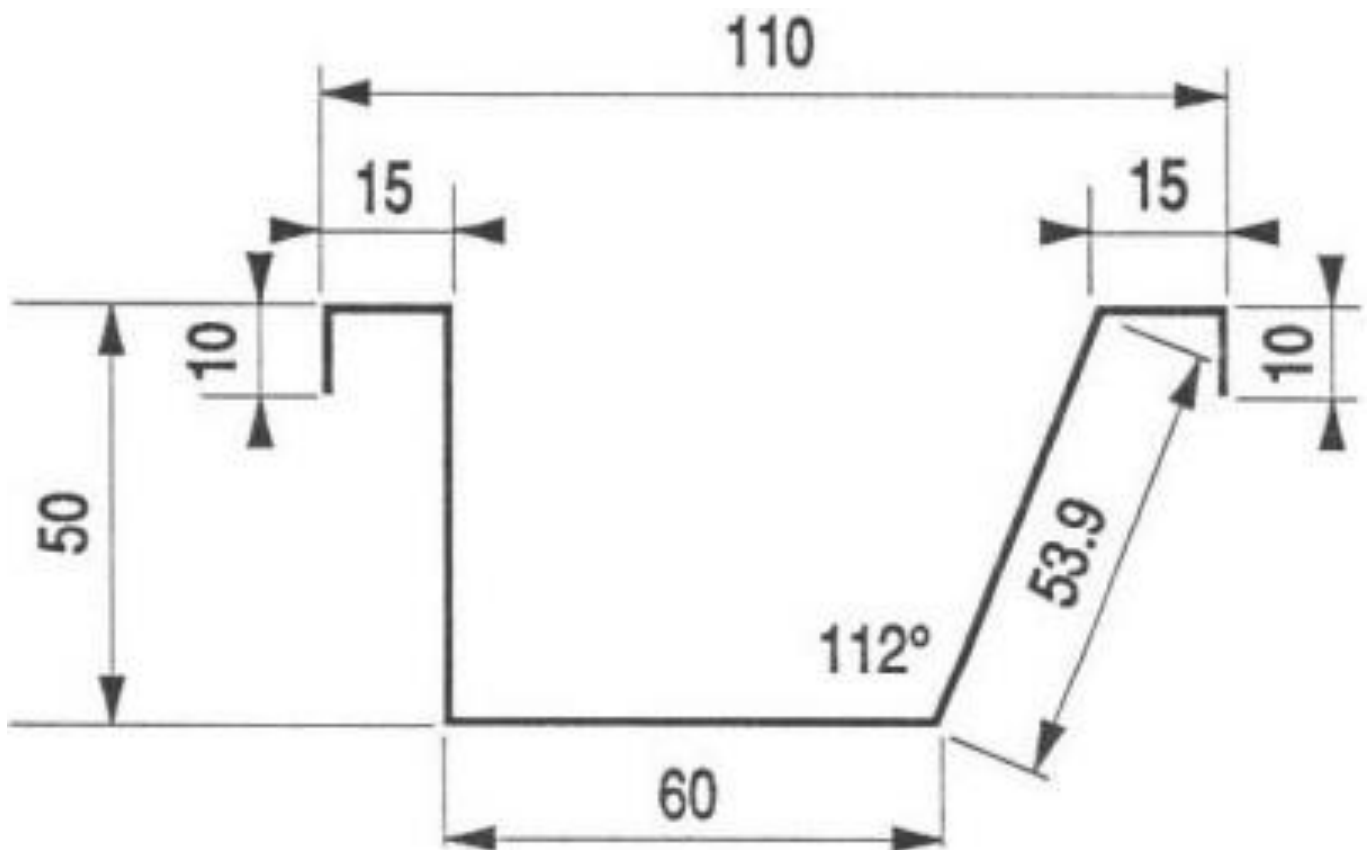
Programming in this mode is very simple. Let's program the work-piece shown above by using 1- mm thick mild steel plate.
You can access ''L-ALPHA programming mode'' by selecting "EDIT GRAPHICAL PROGRAM" from the main menu.

The N.C. first displays an intermediate page for work-piece details. It also enables you to search for an existing program from memory or enter a new number to make a new program.

PROGR: Program number (max. 6 digits) if the entered program number is already assigned to an existing program in L-ALPHA mode, the N.C. will display its characteristics with a drawing of the work-piece.
But if the number is already assigned to MANUAL 1 or MANUAL 2 program, the N.C. will display the following message "PROGRAM INCOMPATIBLE WITH THIS MODE <CR> TO CONTINUE". If the program number is not used or free then N.C. will allow you to program a new work-piece.

PART REF: Enter your part reference (max. 13 digits) if desired. This is an optional field.

THICK: Thickness of the material to be bent in mm.

MAT: Select the type of material you wish to fold using the cursor ⬇ and ⬆ keys. These keys also enable you to scroll through the most commonly used materials. When you have made your choice, confirm with the CR key ↵.

Õ: Material's tensile strength in DaN/mm². This is pre-programmed value for most common materials and will automatically be fed in when a material type is selected but you can modify this if need be.

SP: Reference of the spacer. If you don't know this reference, then press the cursor key ⏎ and NC will display all the spacers in the library. Select the spacer, which is mounted on your machine, or the one, which you intend to use, by using the cursor ↓ and ↑ then confirm your choice using the CR key ⏎ .

DH: Reference of the die holder to be used. Proceed in the same way as for the spacer to select the die holder.

DIE: Enter the reference of the die you will be using. If it is unknown, then use the ⏎ key to display the die library. The selection is made in the same way as before. If you have more than one page, then you can scroll through them using page keys.

R: DIE parameter (optional, used only for reverse mounted die).
    0 = die not reversed
    1 = die reversed

NOTE: This parameter should only be used with a two V or non-symmetrical dies. This will ensure the correct safety zone for "X" axis and also ensure correct NC calculation for the feasibility of the work-piece.

PUNCH: Enter the reference, if you know it, for punch being used. Otherwise, use the ⏎ key to display the punch library. The selection is made in the same way as for the dies.

R: PUNCH parameter (optional, used only when punch is reverse mounted).
    0 = punch not reversed
    1 = punch reversed

NOTE: This should be used then ever punch is reverse mounted so that the work-piece feasibility (production simulation) is correctly calculated.

After programming all the fields, you will obtain a page similar to the one shown below.

```
mm / Mem    HFE100-3      V 6.03    G 6.03     09/05/06

              EDIT        Prog. graphique
N°PROG Part reference TH(mm) MATER.  Õ(DaN/mm²) SPACER    D.H   DIE     RET
290                   1.00   STEEL   40         1         1     9001    0

PUNCH  RET BEND LENG DEV CREATION DATE
9001   0   -1   0        04/01/06
```

START GRAPHIC ANGLE: 0

At this stage, the cursor is in the bottom left of the screen at the field "START GRAPHIC ANGLE=". This will allow you to choose the orientation of the first segment in accordance with the trigonometric angular orientation as shown in the opposite sketch. Program the desired angle then confirmed your

choice using ⏎ key.

Every time you modify an existing program, it is possible to rotate your work-piece if you desired.
After confirming the chosen direction. The screen display will switch to the one below.
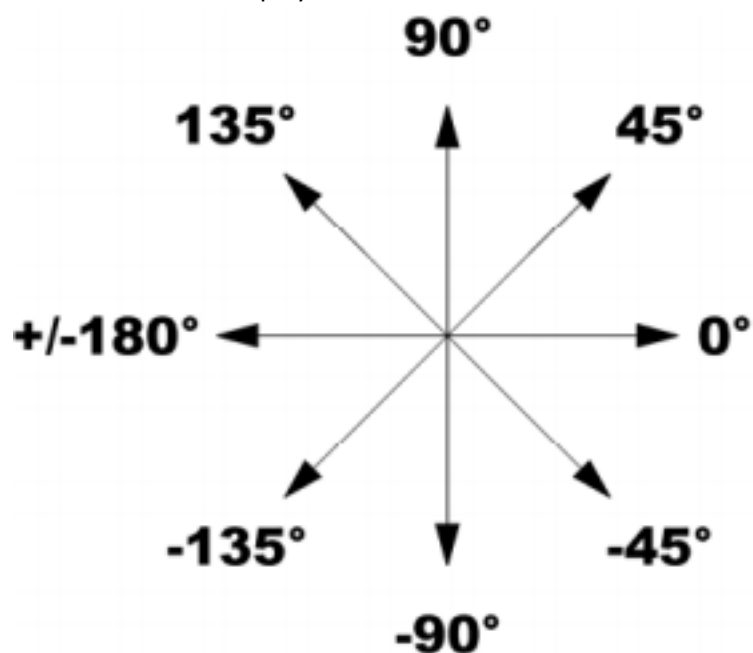
By convention, a work-piece is always drawn from left to right. The different segments are identified in the following order: 1, 2, 3, … n. The different angles will be identified in the order 1, 2, 3, … n-1.

BEND NUMBER: Line identifying the different bends.

FLANGE LENGTH mm: Length of a work-piece segment (external dimension) in mm.

ANGLE deg: Angular value of the bend in degrees.
BENDING LENGTH mm: Length of the bend in a work-piece. This value is required by the NC to calculate the minimum force required for carrying out the bend.

COINING 0/1: Permits to select coining mode for a bend (identical to the COINING in MANUAL 2 mode).

BEND OFFSET mm: Distance the part is off-centred from machine's centre line. This is used when performing bends off centre (identical to PART D in the MANUAL mode 2).

The zero value of the "BEND OFFSET" corresponds to bending in the machine axis.

TONNAGE kN: Required bending force calculated by the NC.

With the cursor located on the first line of the first column, enter the first flange length of the workpiece (as described at the beginning of this chapter) 10 mm.

The cursor will then move to the title "ANGLE" field. To determine whether the angle is positive or negative, position at the edge, as shown in the opposite sketch: enter 90, then confirm. Enter the width of the material to be bent then confirm.

The required bending force to bend this width will be automatically calculated and displayed by the NC in the field opposite to "FORCE".
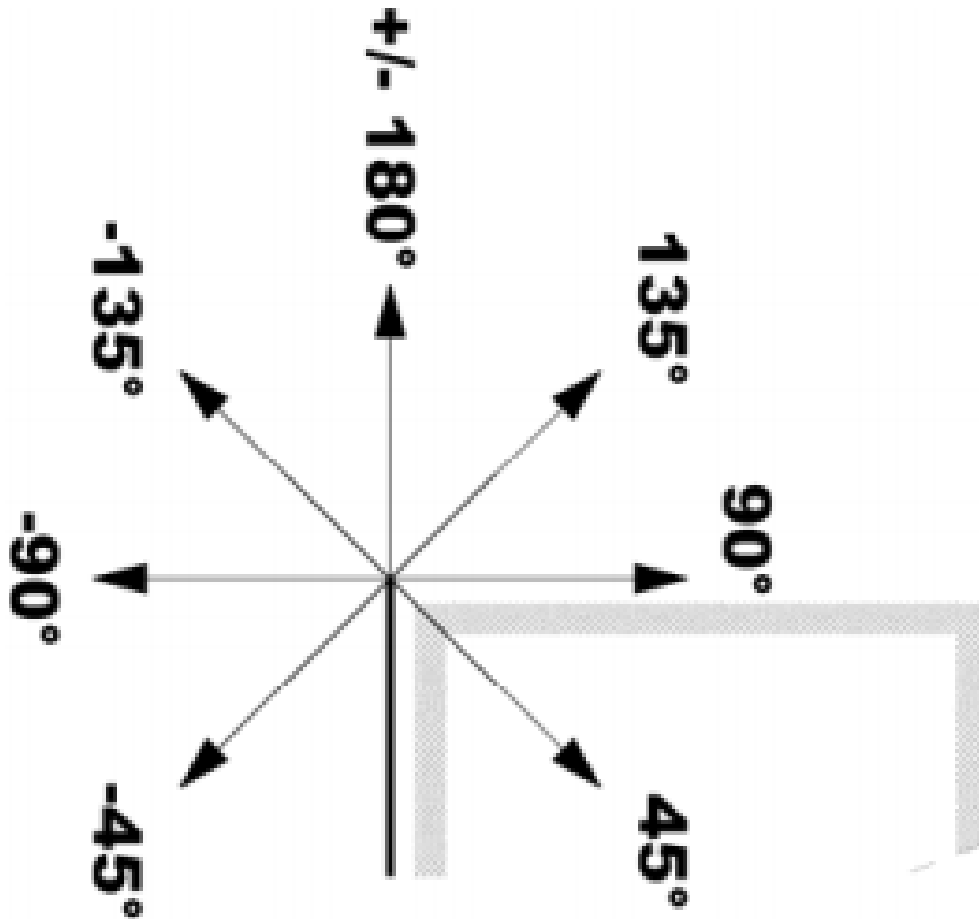
Choose the bending mode you want to use, if you want to use coining mode then enter "1" in the field opposite to "COINING", otherwise leave it to "0" which is the default selection for air bending mode. The zero value of the "BEND OFFSET" corresponds to bending in the machine axis. If you are bending off centre, then enter this distance using positive or negative signs for the value (identical to PART D in the MANUAL mode 2). The NC will re-calculate the tonnage accordingly.

Your first bend is now programmed. To program the next one, press the key, it will copy the parameters of the first bend i.e. "BENDING WIDTH", "COINING", "BEND OFFSET" and "TONNAGE". All you need to do now (in general) is to modify the flange length and the angle.

To know which direction applies to your angles (positive or negative), apply the same principle as before (see the sketch below).
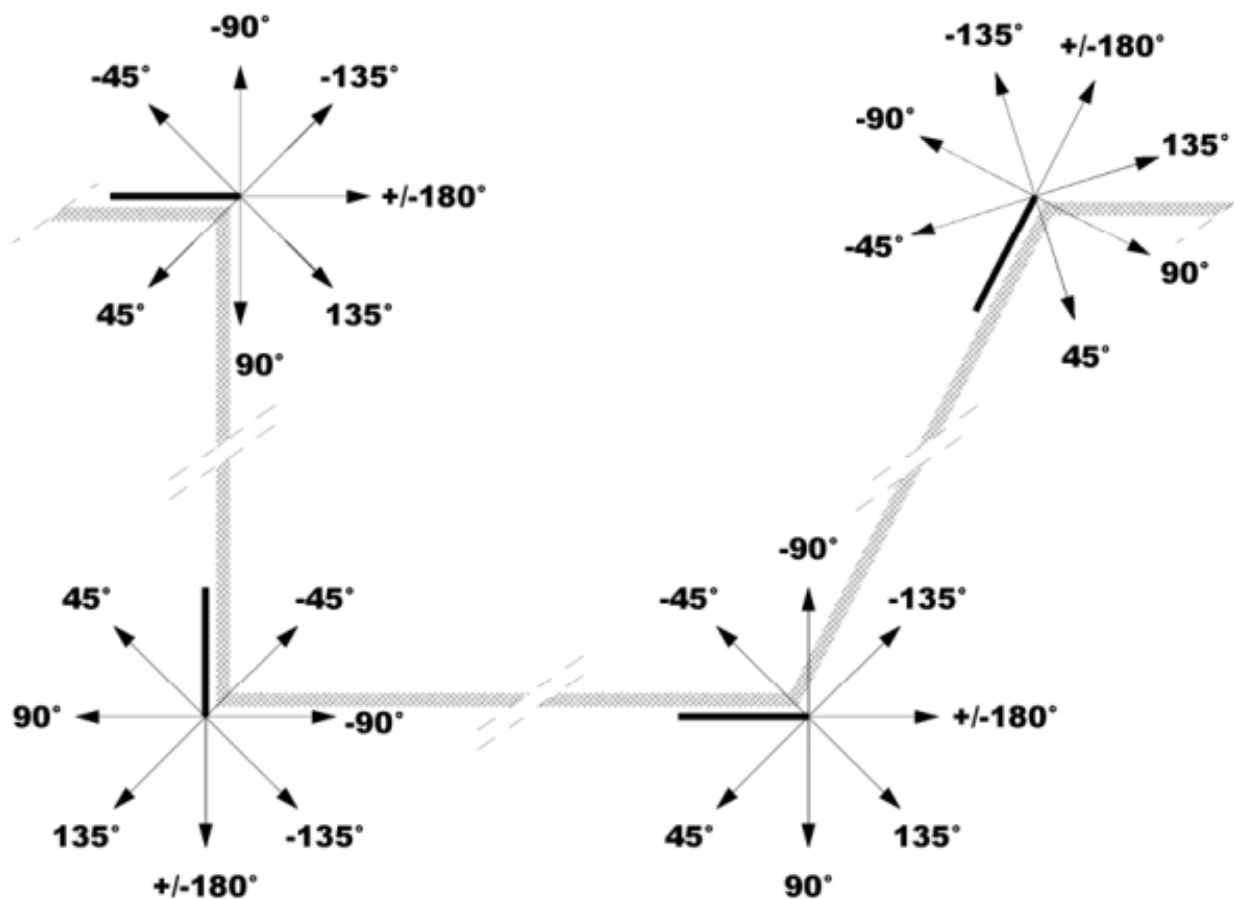
Program your work-piece through to the last segment. Remember that to go from one bend to another you must use the ![key] key. If you make a mistake on a bend, you can delete it using the

![DEL] key with the cursor placed in the column of the bend to be deleted. If you forget a bend, the

![INS] key will enable you to insert a bend at the point where the cursor is located. At the end of programming, you should have a page that looks like the one shown below.

Kasutatakse robotsüsteemi ja manipulaatori absoluutasendi täpseks seadistamiseks
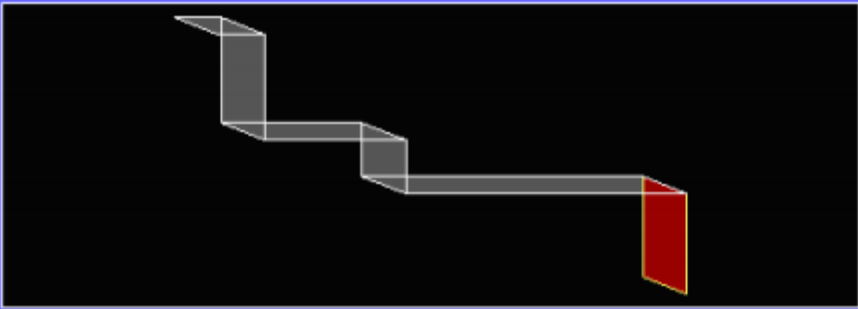
51

| mm / Mem | HFE100-3 | V 6.03 | G 6.03 | 09/05/06 |

EDIT   Prog. graphique

N°PROG	Part reference	TH(mm)	MATER.	Ŏ(DaN/mm²)	SPACER		D.H	DIE	RET
22		1.00	STEEL	40	1		1	9001	0

PUNCH	RET	BEND	LENG DEV	CREATION DATE
9001	0	5	1481.44	04/01/06

BEND NUMBER		1	2	3	4	5	6
FLANGE	mm	100.0	200.0	300.0	100.0	600.0	190.0
ANGLE	deg	90.0	-90.0	90.0	-90.0	90.0	0
BENDING LENGTH	mm	100	100	100	100	100	100
COGNING	0/1	0	0	0	0	0	0
PART OFFSET	mm	0	0	0	0	0	0
TONNAGE	kN	75	75	75	75	75	0

NOTES:

Once you have entered the flange length of the first segment, pressing the "END/FIN" key will record your program. This key enables you to stop programming mode at any time and come back to it later.

For each programmed segment, the developed length of the work-piece is recalculated and displayed in the field "DEVL LG".

In case if the part is symmetrical, then you can only draw half of the part and then use mirror key. The NC will ask you to input the angle of symmetry. Once you give this angle, the symmetry will be created according to this angle.
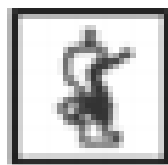
When you finished drawing your work-piece, press the   key. A new page will be displayed on the screen, showing your machine profile fitted with the selected tools, which you have chosen at the beginning of your programming.
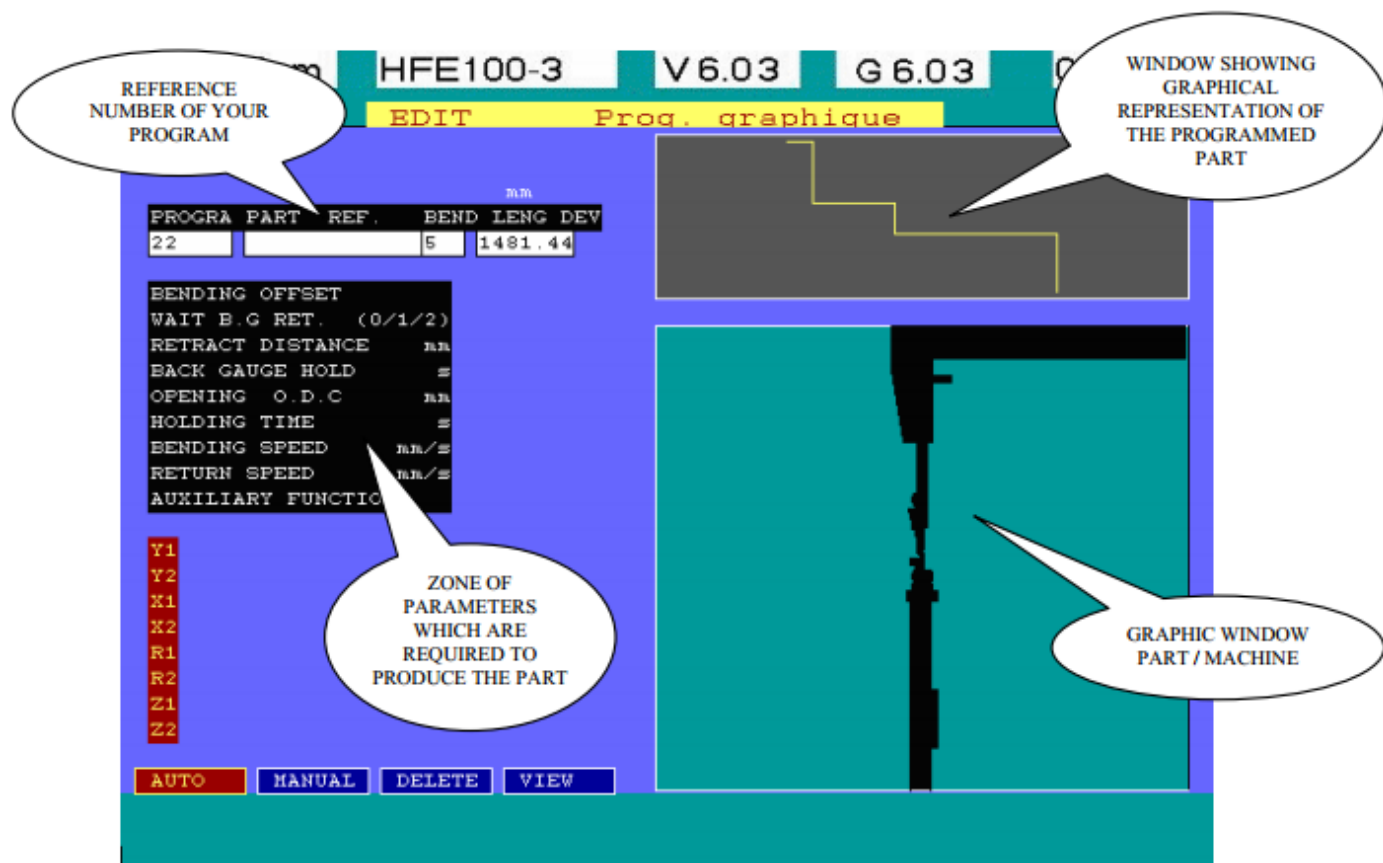
The 🔼 and 🔽 keys are active. They offer a "ZOOM" function, operable on the "GRAPHIC WINDOW PART / MACHINE" and enabling you to change the scale of the work piece display on the screen.

At the bottom, on left side of the graphic window, there are four selectable functions. You can select these using ➡ or ⬅ keys and then confirm with the ↵ key.

AUTO: Automatic calculation of the bending sequence with selected tooling.

MANUAL: Manual determination of bending sequence with selected tooling.

CANCEL: Permits to cancel a calculated bend sequence display on the screen.

VISU: Visualization of calculated bending sequences, which are determined either using "AUTO" or "MANUAL" modes.

This page also displays the characteristics of your program, e.g.; (part reference number and it's developed length) along with "graphic window part / machine", reminding you of the part drawing.

To determine automatically the bending sequence, select "AUTO" from the bottom of the page then press ↵ key. The message ODC min: 1/0: will be displayed with a default value of 1.

With the default value is 1, NC will calculate the minimum opening required to extract the produced component (fig. 1).

If the part is to be removed flat to the die (fig. 2), then program 0.

Pressing ⏎ key once again will start the calculation according to the parameters programmed on the "DATA UPDATE" page of the main menu.
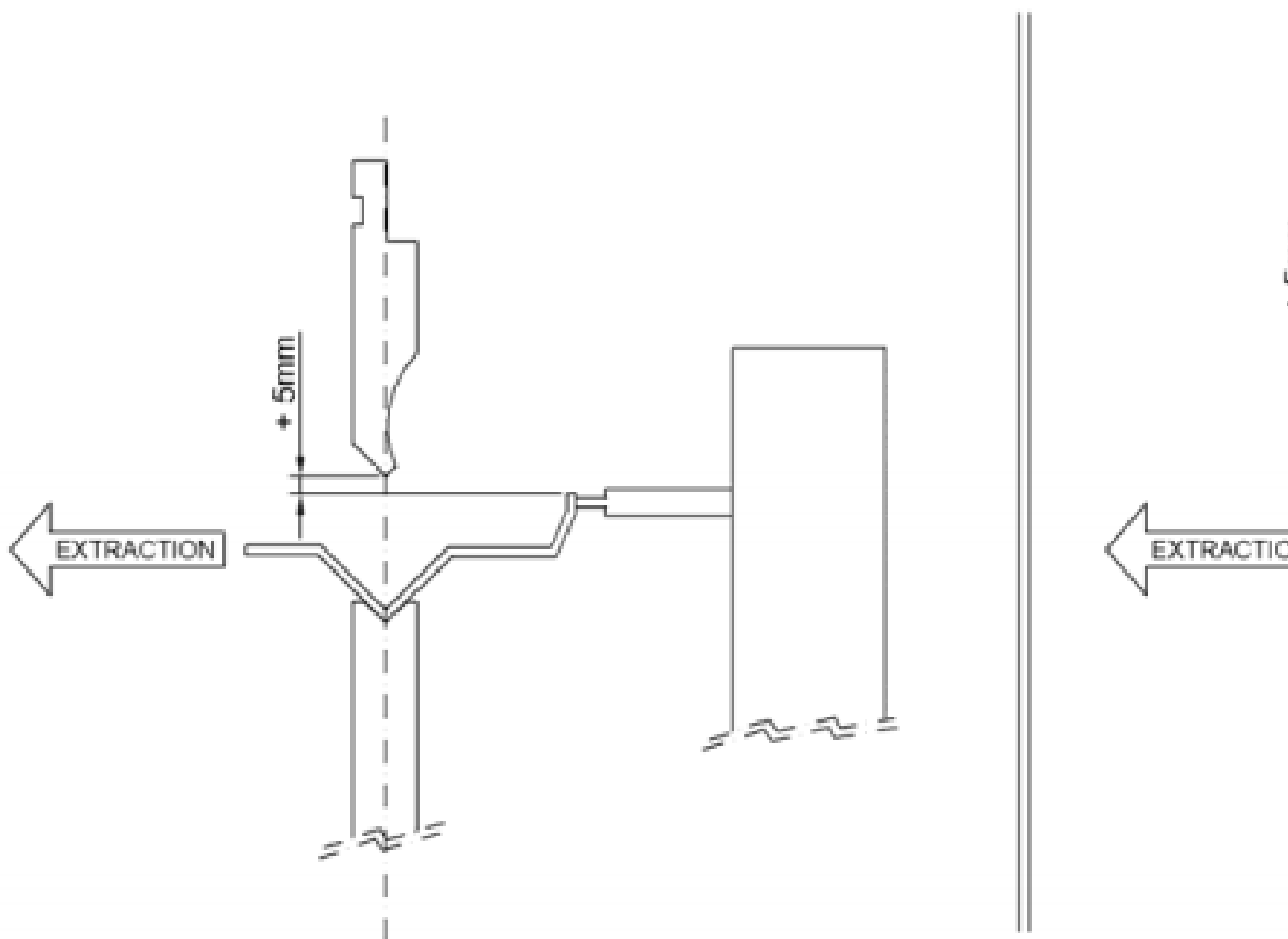
Fig: 1   Fig: 2

SIMULATION CRITERION AND AUTO

MAX INTERMEDIATE BEND BG: Maximum number of bends you authorize between the backgauge and bend to be made (99 max.).

TOLERANCE ON THE EDGE: Permissible angular tolerance on the bending flange at the edge (45° max.).

TOLERANCE AT 90: Permissible angular tolerance on the bending flange at 90° (45°max.).

FRONT EXTENDED RATIO: This is the front to back ratio from bend line to the edge of the material.

PLI MAX ENTRE PLI ET BUTEE

TOL ∝ BUTEE SUR EXTREMITE

TOL ∝ BUTEE SUR PLI A 90

RAPPORT AVANT / ARRIERE
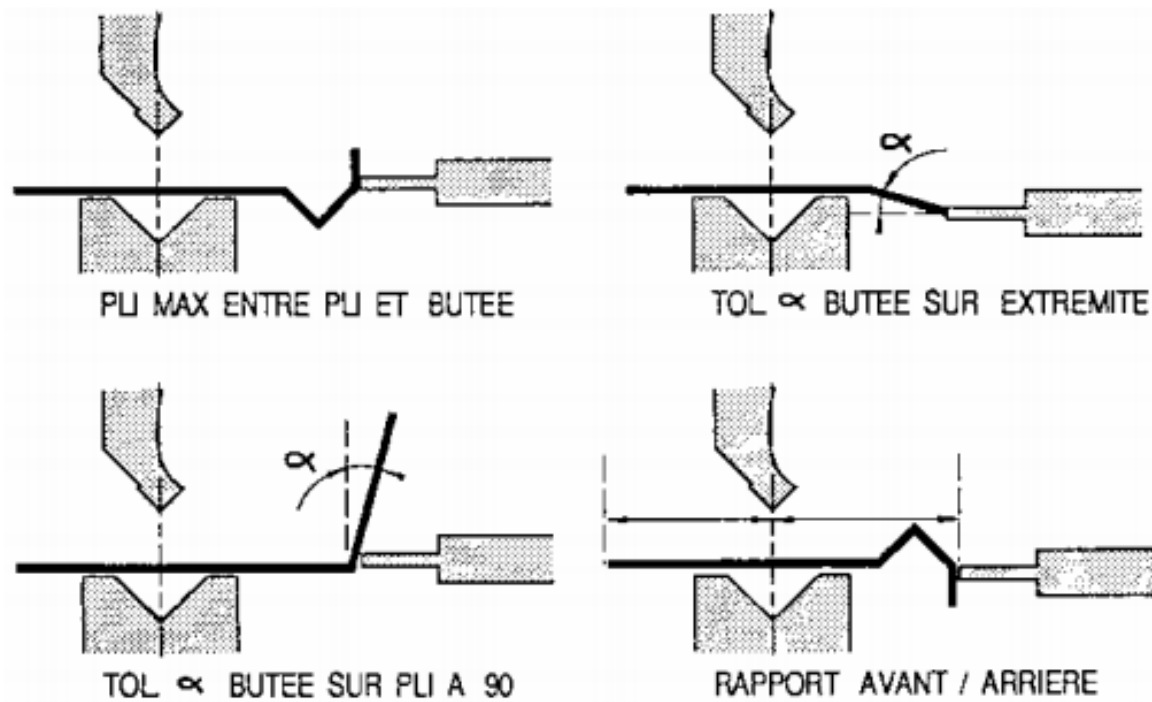
The above functions are described in details earlier, see data update.

At the end of the "AUTO" determination of the bend sequence, each angle of the work-piece profile inside the ''WINDOW SHOWING GRAPHICAL REPRESENTATION OF THE PROGRAMMED PART" will be circled. These circles indicate that a solution has been found for the programmed part and in the "GRAPHIC WINDOW PART/MACHINE", your work-piece will be positioned on the tool for the execution of the first bend. This current bend will be identified in the 'WINDOW SHOWING GRAPHICAL REPRESENTATION OF THE PROGRAMMED PART'' by a red circle. The white circles represent the bending sequences yet to be accomplished.
Selecting the "VISU" function allows you to browse through the entire calculated bending sequence

by using the [→] key. When you reach the last bend, the [←] key enables you to unbend your work-piece.

> **NOTE:** If bends are already done, they are not circled in the ''WINDOW SHOWING GRAPHICAL REPRESENTATION OF THE PROGRAMMED PART" during the bending sequence.

mm / Mem   HFE100-3   V 6.03   G 6.03   09/05/06

EDIT   Prog. graphique

PROGRA	PART	REF.	BEND	LENG DEV
22			5	1481.44

BENDING OFFSET   0
WAIT B.G RET. (0/1/2) 0
RETRACT DISTANCE   mm 0
BACK GAUGE HOLD   ≥ 0
OPENING O.D.C   mm 140
HOLDING TIME   ≥ 0.2
BENDING SPEED   mm/s 10
RETURN SPEED   mm/s 0
AUXILIARY FUNCTIONS   000000

Y1	-2.26
Y2	-2.26
X1	189.14
X2	189.14
R1	-2.00
R2	-2.00
Z1	60.00
Z2	60.00

AUTO   MANUAL   DELETE   VIEW
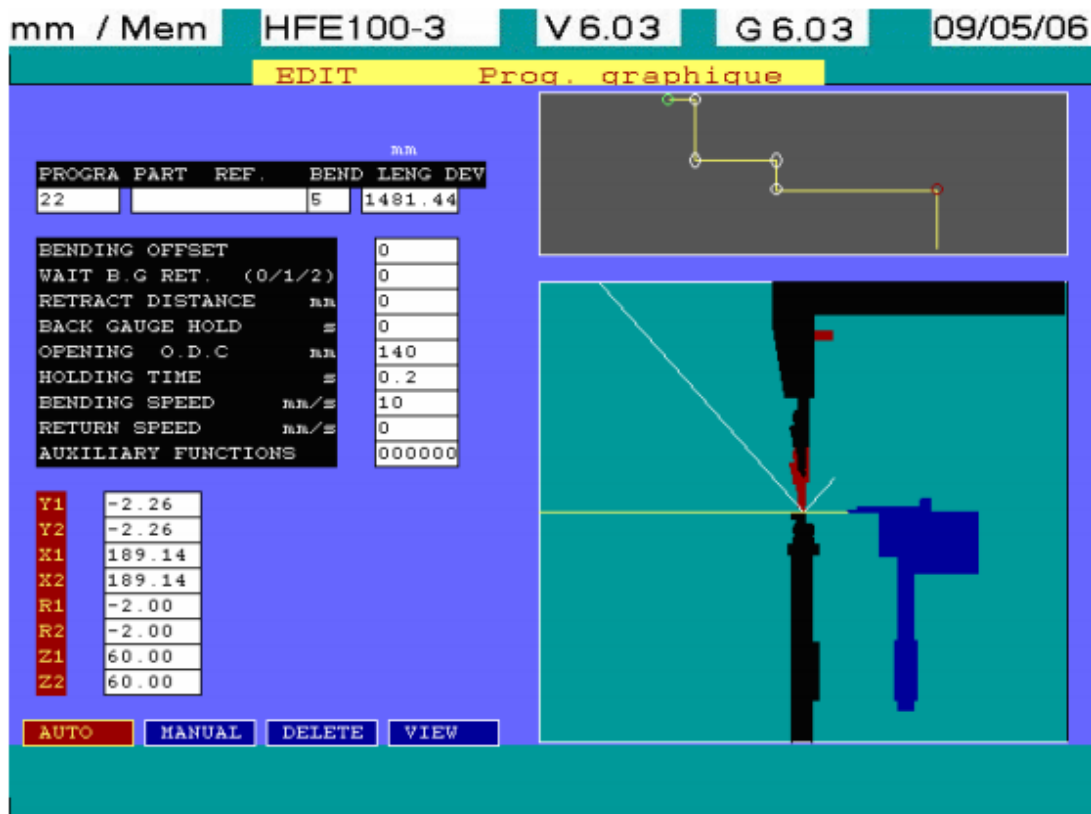
For each of the visualized bend sequence, the presence of the cursor will allow you to modify calculated values of the parameters for the displayed bend as well as the positions of axes Z1 and Z2 axes.

To exit the "VISU" window, press "END/FIN" key, which will bring you back to the preceding page where you had four selectable functions.

If, you launch "AUTO" and there are no circles on the work-piece, in the ''WINDOW SHOWING GRAPHICAL REPRESENTATION OF THE PROGRAMMED PART" and the press brake is displayed opened without any worksheet. This means that the automatic bend sequence calculation failed. (This situation can be caused by diverse reasons such as unsuitable tools selected for the job or the criterions for the bend simulation are too restricted or even contradictory).

To resolve the problem, check if the selected tools are suitable to carry out the job, they are correctly positioned and that the simulation criterions are correct.

Re-activate the "AUTO" search for bend sequence after correcting the above.

If still no solution is found, switch to "MANUAL" mode to visualize any eventual collisions. The order in which visualization can be done will always be angle n° 1, angle n° 2 etc.

Use the [←] key to locate the last angle to be produced (this search can be made by unfolding the work-piece).

The "MIRROR" key [+ ▶] will permit you to obtain a different gauging configuration by swapping your work-piece front to back.

The [→] and [←] keys allows you to choose the gauging position of the backgauge finger when more than one bend exists between the bend in question and the backgauge.

After verifying the bend parameters (BEND OFFSET, BG WAIT, RETRACT DIST. …AUX. FUNCTIONS), confirm your manually selected sequence using the [⊥] key.

Proceed in the same manner for the remaining bends. One thing you should bear in mind that to determine the bending sequence in "MANUAL", you should start from the last bend and work through to the first bend i.e. backward.

The other usage of the "MANUAL" mode consists of defining the last one or more bends the way you like to carry out then exiting the "MANUAL" function by pressing the "END/FIN" key. Then determining the remaining bend sequences using "AUTO" function.

"CANCEL" is the last available function on this page. It enables the bend sequence to be deleted.

This bend sequence cancelling function allows you to redo the calculation in "AUTO" from the cancelled bend sequence or to determine a new bend sequence in "MANUAL" to change bending order, gauge position. The bend sequence cancelling is done in the following order: bend n° 1; bend n° 2; … bend n° n.

EXAMPLE: If you wish to delete bend n° 3, you will be obliged to cancel bend n° 1 and bend n° 2 before you can cancel bend n° 3.
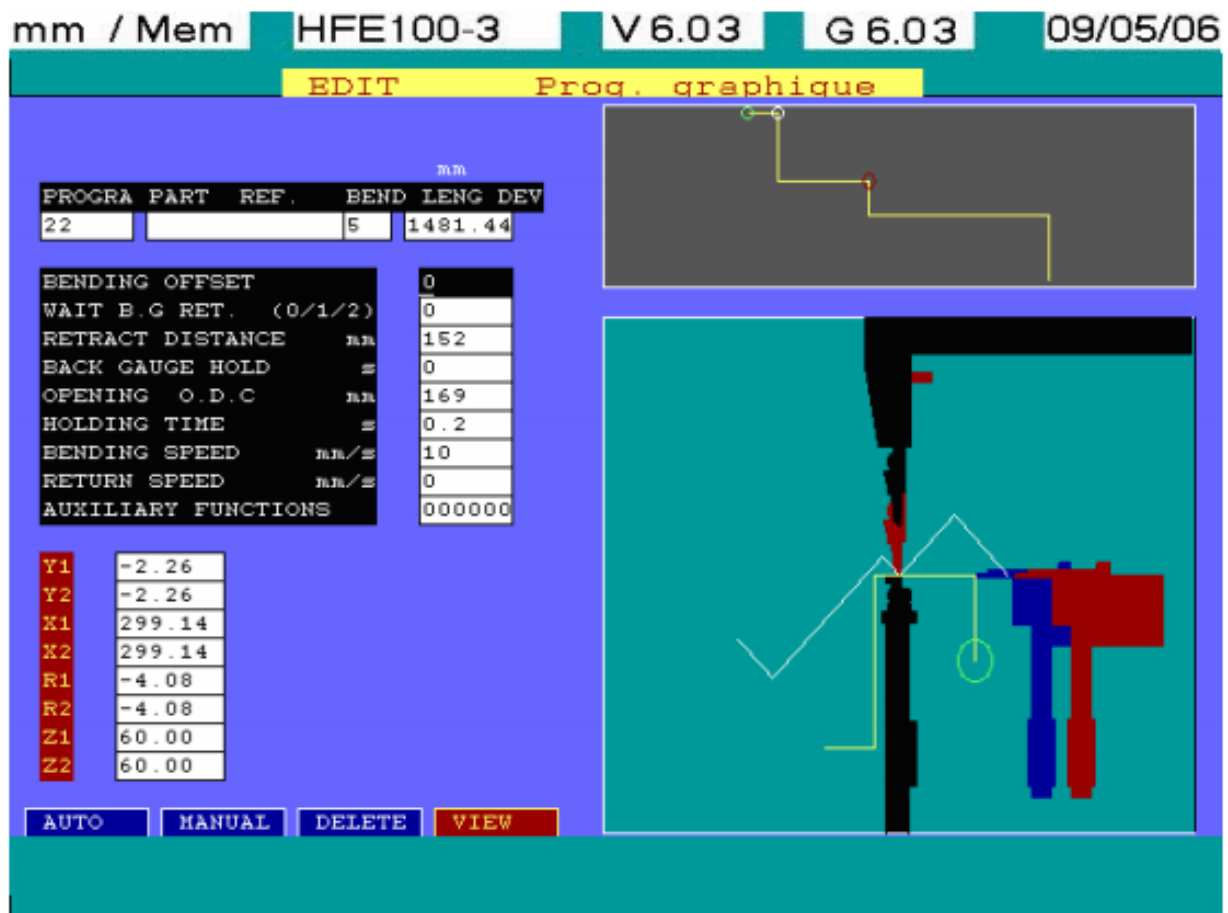
To exit "EDIT PROGRAM" mode, press ![FIN/END] key, your program will be saved automatically if your bend sequence is complete, otherwise a series of messages will be given on screen to warn you that the order is not complete with a possibility to re-use your programmed drawing.

The N.C. first displays an intermediate page for work-piece details. It also enables you to search for an existing program from memory or enter a new number to make a new program.

1.8.2. EDIT PUNCH

The "EDIT" ⟹ "Punches" selection gives access to simplified programming of special punches. This function is also used to modify the existing punches included in the standard library supplied with the N.C.

An existing punch may be modified by keying-in its reference number (the values pre-set for each punch parameter are then displayed); a new punch may be edited by keying-in a non-used reference number. Enter the following parameters:

- "PUNCH NUMBER": punch reference number.
- "ANGLE": punch angle in degrees.
- "RADIUS": punch radius in mm.
- "HEIGHT.": punch height in mm (without tail). The punch height is in between punch tip to its load bearing surface.
- "TONNAGE": Maximum permissible tonnage for the Punch in KN/m (generally marked on the punch).
- Press →, ← or ↵ key to run through the parameters.

Once the last punch parameter has been entered ("CHARG" value), press once ↵.
Once the punch profile has been sectionalized in L-ALPHA, enter them in the below-given table (LONG, ALPHA).
When it comes to the last ALPHA and LENGTH press FIN/END, the system will calculate automatically last alpha and last segment from last point to starting point on the first segment: decimal errors will therefore be avoided.

When the drawing is completed, press ⌷FIN/END⌷ to memorize the punch, a blank line will be displayed; a new punch may be entered. Press ⌷FIN/END⌷ to go back to the main menu.

1.8.3. EDIT DIES

Similar to the punches, the "EDIT" ⟹ "Dies" selection is used for programming very easily non-standard dies. Existing dies which are part of the standard library supplied with the NC may also be modified.

An existing die may be modified by keying-in its reference number (the adjustments pre-set for each die parameter are then displayed); a new die may be edited by keying-in a non-used reference number. Enter the following parameters:

The creation or modification will be carried out in accordance with the below-given diagram.

SECR SEC1
RAD.
1mm
ANG
1mm
HEIGHT
V
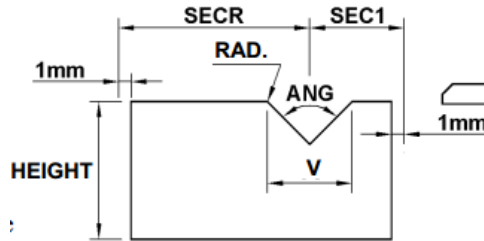
- "DIE N°": die reference number.
- "DIE": die width in mm.
- "ANG": die angle in degrees.
- "HT.": die height in mm.
- "SEC1": safety distance (sec 1) in mm.
- "SECR": safety distance (sec R) in mm.
- "RAD.": die edge radius in mm.
- "TONNAGE": maximum permissible tonnage in an kN/m.

Press [→] , [←] or [↵] key to run through the parameters.

Once the last die parameter has been entered ("TONNAGE" value), press once [↵] .

Another table containing the description of the die drawing (displayed in 2α) will be displayed. If existing, the die drawing may be modified, otherwise, draw it from scratch by entering L, subsequent α, as specified in « PUNCHES EDIT»

Press [FIN/END] to exit.

The graphical editing of a die is similar to that of a punch.

1.8.4. EDIT DIE-HOLDERS

Similar to the dies, the "EDIT" ⟹ "Die-holders" selection is used for programming nonstandard die-holders very easily. Existing die-holders which are part of the standard library supplied with the NC may be modified as well.

```
EDIT          Die holders

N°DH  HEIGTH




                                    D.H. NUMBER   8
                                    MAXIMUM D.H.  10
```

An existing die-holder may be modified by keying-in its reference number (the values pre-set for each die-holder parameter are then displayed); a new die-holder may be edited by keying-in a non-used reference number. Enter the following parameters:

- "DIE-HOLDER N°": die-holder reference number.
- "HEIGHT": die-holder height in mm.

Press [→],[←] or [↵] key to run through the parameters.

Once the last die-holder parameter has been entered ("HEIGHT" value), press once [↵].

Another table containing the description of the die-holder drawing (displayed in 2α) will be displayed. If existing, the die-holder drawing may be modified, otherwise, draw it from scratch by entering L-subsequent α, as specified in « PUNCHES EDIT»

Press [FIN/END] key to quit.

1.8.5. EDIT SPACERS

Similar to the die-holders, the "EDIT" ⟹ "Spacers" selection is used for programming very easily non-standard spacers. Existing spacers which are part of the standard library supplied with the NC may be modified as well.

```
EDIT          Spacers

SPACER NUMBER




                                    SPACER NUMBER   2
                                    Maximum SPACER  10
```

An existing spacer may be modified by keying-in its reference number (the values pre-set for each spacer parameter are then displayed); a new spacer may be edited by keying-in a non-used reference number. Enter the following parameters:

- "SPACER N°": spacer reference number.
- "HEIGHT": spacer height in mm.

Press [→], [←] or [↵] key to run through the parameters.

Once the last spacer parameter has been entered ("HEIGHT" value), press once [↵].

Another table containing the description of the spacer drawing (displayed in 2α) will be displayed. If existing, the spacer drawing may be modified, otherwise, draw it from scratch by entering L-subsequent α, as specified in « PUNCHES EDIT».

Press [FIN/END] key to quit.

1.8.6. EDIT OPTIONS

The "EDIT" ⟹ "Options" function allows you to modify and personalize certain data for your NC controller. The possibilities are briefly described below:

- "FINGER PROFILE": number of finger profile in the back gauge.

- "MUTE VALUE": This is the safety value where commutation of speed will take place. 6 mm (safety distance above sheet) is recommended by Amada and with this value you can have maximum safety and minimum cycle time. If you program higher value, machine will operate in working speed during this and cycle time will be increased.

- "DEFAULT DX AUTO": DX AUTO default value for programming and applicable for programs made in manual mode 2. This allows you to choose internal or external quotations for a given program. Any change will be valid in new programs only. The existing programs will remain as they are.

- "ANGULAR TOL.": angle tolerance in mm (valid for DIGIPRO only).

- "EDGE POS. RIGHT": valid for DIGIPRO only.

- "APPROACH SPEED": maximum approach speed, for certain applications, can be reduced. Of course, in this case, cycle time will be increased.

In the lower left-hand corner of the screen, are three columns displayed: Column 1 displays the materials treated by the software. Column 2 is for the percentage correction of the bending co-efficient related to each type of material, which is valid for all bends and programs. Column 3 "DX" is the unfold correction applied to these materials and it corrects the developed length of the program. These percentages modify the penetration and developed length value by acting directly on the calculation module of the CD 2000 bending software. Let's consider as an example a given part in steel: if the NC calculates a penetration value of 5 mm for a given bend, the operator has a choice of taking only a percentage of this value. Programming 10 in the table in front of "STEEL" will change the calculated penetration to 0.5 mm. Any percentage value inferior to 100 % will reduce the penetration and any percentage over 100 % will increase it.

- Follow the below-given procedure to modify the reference values of the different axes:
  - In the table from the lower right part of the NC screen which is composed of 2 columns ("AXES" and "REFERENCES") and 4 rows ("Y1", "Y2", "X", "R"), go to the line corresponding to the reference value to be modified, enter the new value and confirm with ⏎ key.

  NOTE The reference value should be modified only if the displayed value is different from what is obtained (in this case, the value will remain **constant**).
  Example: when X axis is displayed = 50 mm and X axis is actually positioned to 50.2 mm, a 0,2 mm error is found on X.
  Sort that out by positioning the cursor on X reference and enter once again the new value as follows:
  New X reference value = previous X reference value + 0,2 mm.
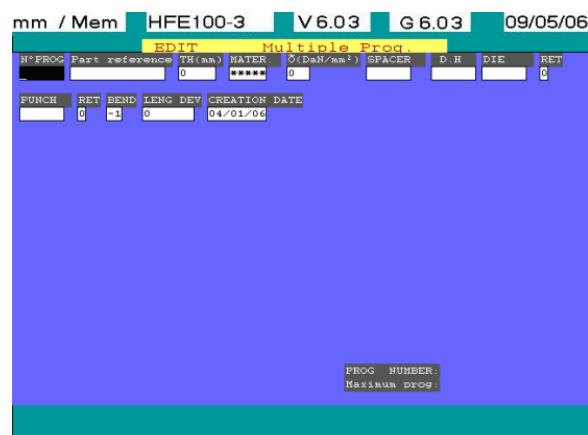
- The measurement unit system may be modified (mm or inch).
  Press 🗝 key to quit.

  The disk unit line will be used to select 1 or 2 line with floppy disk or 3 or 4 line with USB key.

1.8.7. EDIT MULTIPLE PROGRAM

"EDIT" ⟹ "Multiple Prog." displays the following screen:

A multiple part program is composed of several single programs which may be handled both in "MANUAL 1 MODE" and "MANUAL 2 MODE".

The only restriction is that a multiple program can not contain another multiple program. The maximum permissible number multiple programs can have is 15. When you execute a multiple part program, in reality, you will execute one by one (in the order you build up your multiple part program) all individual programs of which it is composed.

- Enter the multiple part program number and its reference (multiple part program reference is an optional field) and confirm by pressing either cursor ↓ , → or ↵ . This will display the list of all programs in the NC library.

- Using any of the four ← , → , ↑ , ↓ , move the cursor above the selected program. Select it with ↵ key, it is then highlighted. Select that way all the single programs you want to use in your multiple program. A screen similar to the one below is displayed:

In this specific example, the multiple program is called "12121", with "PLPLPL" as a reference number where 4 single programs have been selected with the following reference number: "11", "99", "2" et "290".

NOTE:
If the selected single program is a multiple program, the NC will display the following error message: "MULTIPLE PROGRAM <CR> TO CONTINUE" and will not allow its selection.

Press ↵ key to continue.

- Once selected, press key FIN/END .

Once the unit programs have been finally selected, they will be displayed as selected:

- Order in which programs are selected for execution (or selection for the manual mode) of each program (1st column).

- Program numbers selected from the library (2nd column).

- Quantity a single program should produce (3rd column).

The following screen is obtained:

- The quantity of execution of each single program may then be modified.
- Press [FIN/END] key so as to validate your multiple program and save it in the NC library.
- The operator is then requested by a prompt to confirm the back-up of the program. If "1" is pressed, then key [↵] or [FIN/END], the program is saved. If "0" is selected by default, press key [↵] or [FIN/END] to validate, you quit the function without saving the edited multiple program.

1.8.8. BACKGROUND COLOR EDIT

Once the color of the 3 windows has been specified, press [FIN/END] to return to the main menu and apply new colors.

1.9. LIST

1.9.1. LIST PROGRAMS

The "LIST" ⟹ "Programs" function allows to list, on one single page, all the programs existing in the NC library.

Use keys ←, →, ↑, ↓ to scroll through the page.

Position the cursor on a program, the following data will be displayed in the upper part of the page:

- The reference numbers.
- The part reference (if specified during the program creation).
- The thickness of the material used *.
- The material used *.
- The material tensile strength*.
- The spacer reference number*.
- The die-holder reference number *.
- The die reference number *.
- Die reversed or not *.
- The punch reference number *.
- Punch reversed or not *.
- Total number of bends.
- Date of creation.

    *: valid for programs created in "MANUAL MODE 2" or GRAPHICAL only.

To obtain further information on a specific program, position the cursor on it and confirm with key ↵. The type and quantity of the displayed information depend upon the type of mode in which the program has been selected. 1. The following screen will be displayed for a program selected and created in mode "MANUAL 1" or "MANUAL 2":

The following data are given:

- "PROG": Reference number of the selected program.
- "PART REF.": Reference of the part (if entered when creating the program).
- "CRE DATE": Date of creation.
- "PRE-TIME": (only parameter which may be modified in this page). This value is reset to 0 when creating the program. Once executed "PR-TIME" can be used to determine the average time for executing a part (time when the EXECUTION mode is active, START is activated, divided by the number of parts to be done). This value may be set to 0 for a new measurement of the manufacturing time. Pressing ⏎ key brings you back to the previous page.
- "BEND NB": "1" is always displayed in this field, in the case of a multiple program whatever the specified bends in each of the programs composing it.
- Column "BEND": Existing bends in the program classified in their execution order.
- Column "CY": Number of cycles per bend to be executed.
- Column "Y1", "Y2", "X", "R": Position of the axes for each bend.

The following screen will be displayed for a selected program created in "MULTIPLE PROG" mode:

The following data are given:

- "PROG": Reference number of the selected multiple program.
- "PART REF.": Reference of the part (if entered when creating the program).
- "CRE. DATE": Date of creation.
- "PR TIME": (only parameter which may be modified in this page). This value is reset to 0 when creating the program. Once executed "PRE TIME" can be used to determine the average time for executing a part (time when the EXECUTION mode is active, START is activated, divided by the number of parts to be done). This value may be set to 0 for a new measurement of the manufacturing time. Pressing ⏎ key brings you back to the previous page.
- Column "PROG": Existing single programs in the multiple program.
- Column "SP": Spacers connected to the different single programs.
- Column "D-H": Die-holders connected to the different single programs.
- Column "DIE": Dies connected to the different single programs.
- Column "PUNCH": Punches connected to the different single programs.
- Column "R": Reverse-option of the dies or punches for every single program.

Press this key ![FIN END] to quit this information page. The page containing all the programs in memory is displayed. Press ![FIN END] this key again to quit.

1.9.2. LIST PUNCHES

The "LIST" $\Longrightarrow$ "Punches" enables you to consult the punch library in the NC memory. Scroll through with 12 punches on each page (3 pages maximum) and 30 punches maximum in memory:

The following data are given in the upper part of the screen: "N°": number of the current punch (where the cursor is positioned) / total number of punches in memory.
"Maxi": maximum number of punches admissible in memory (30).

For each punch, the following data are specified in the table:

- "N°": Punch number.
- "ANGLE": Punch angle in degrees.
- "RAD": Punch radius value in mm.
- "HT.": Punch height in mm (without the tail). The height of the punch is measured between its support surface and its tip.
- "TONNAGE": tensile strength of the punch in kN/m (this value is normally marked on the punch).

  Use keys ![↑], ![↓] to scroll through a page in this table.

  If more than 12 punches are saved in the library, press key to move to the next page ![→],

  press key to go to the previous page ![←].

  Press key ![FIN END] to quit.

1.9.3. LIST DIES
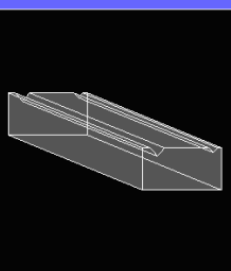
"LIST" $\Longrightarrow$ "Dies" enables you to consult the dies library in the NC memory. Scroll through with 12 dies on each page (6 pages maximum) and 70 dies maximum in memory:

The following data are given in the lower part of the screen:
"N°": number of the current die (where the cursor is positioned) / total number of dies in memory.
"Maxi": maximum number of dies admissible in memory (70).

For each die, the following data are specified in the table:

- "N° DIE": Die number.
- "V": Die width in mm.
- "ANG": Die angle in degrees.
- "HT: Die height in mm.
- SEC1": Safety distance 1 in mm.
- SECR": Safety distance R in mm.
- "RAD.": Die edge radius in mm.
- "TONNAGE": Maximum authorised tonnage in kN/m.

Use keys ↑ , ↓ to scroll through a page in this table.

If more than 12 dies are saved in the library, press key to move to the next page ➡, press key

to go to the previous page ⬅ .

Press key [FIN/END] to quit.

1.9.4. LIST DIE-HOLDERS

"LIST" ⟹ "Die-holders" enables you to consult on one page the dies library in the NC memory.
Scroll through with 10 die-holders maximum memory:
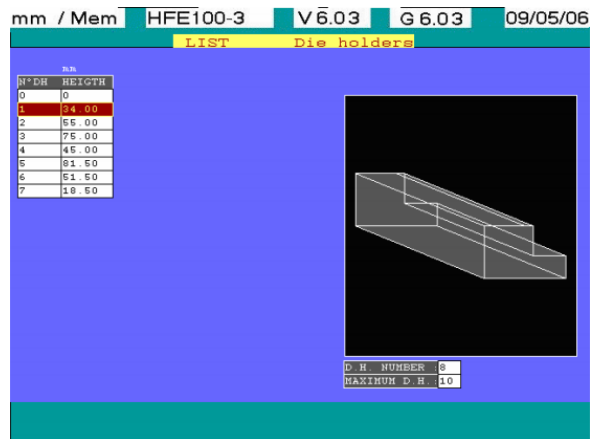
The following data are given in the upper part of the screen:

"N°": number of the current die-holder (where the cursor is positioned) / total number of die-holders in memory.

"Maxi": maximum number of die-holders admissible in memory (70).

For each die-holder, the following data are specified in the table:

- "DIE-HOLDER N°": Die-holder number.
- "HT": Die-holder height in mm.

Use keys , to scroll through the list.

Press this key to quit.

1.9.5. LIST SPACERS

"LIST" ⟹ "Spacers" enables you to consult on one page the spacers library in the NC memory. Scroll through with 10 spacers maximum in memory:
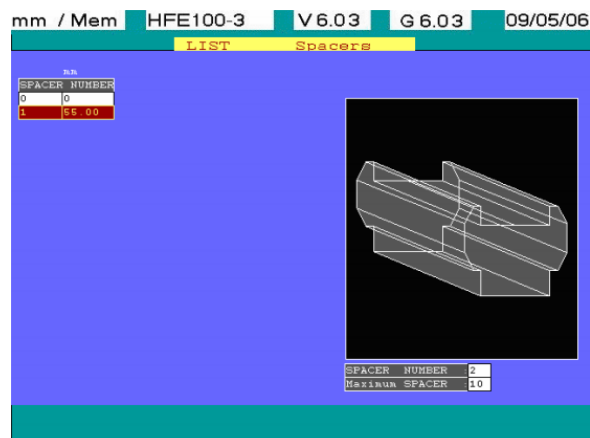
The following data are given in the upper part of the screen:

"N°": number of the current spacer (where the cursor is positioned) / total number of spacers in memory.

"Maxi": maximum number of spacers admissible in memory (10).

For each spacer, the following data are specified in the table:

- "N° SPACER": Spacer number.
- "HT": Spacer height in mm.

Use keys [↑],[↓] to scroll through the table.

Press this key [FIN/END] to quit.

1.10. DELETE

1.10.1. DELETE PROGRAMS

The "DELETE" ⟹ "Programs" function is used to delete program(s) existing in the NC library. An intermediate page will be displayed (see next page) asking for a PARTIAL ("1") or TOTAL ("2") deletion of the program(s) in memory.


```
mm / Mem   HFE100-3    V 6.03    G 6.03    09/05/06
                  DELETE      Programs




















  DELETE (1=ALL / 2=PARTIAL): 2
```

- If "2" has been selected, (TOTAL deletion), and validated with key [↵] or [FIN/END] , another page will be displayed to confirm the deletion of ALL the programs in memory:

If you confirm the total deletion with "1" (equivalent to "YES") press key [FIN/END] or [↵] to validate, ALL programs are then deleted. If "0" has been selected (default value corresponding to "NO") you quit the deletion function (programs will not be deleted).

- When selecting "1", (PARTIAL deletion), and confirm with key [↵] or [FIN/END] , a screen identical to "LIST" ⟹ "Programs" is displayed, where the programs to be deleted should be selected.

    Use key [←],[→] to scroll through the page [↑],[↓]. Position the cursor on the selected

    program, then confirm with key [↵]. The selected program is highlighted. Proceed that way

    for the rest of the programs to be deleted. Deselect it by pressing key [↵] once again with the cursor positioned above the program, the reverse video will disappear, confirming that

    the program is not selected any longer. Once finished, press key [FIN/END] to start the deletion. You are requested by the NC to confirm the deletion of the selected programs:
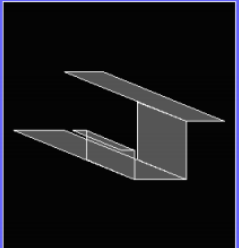
If "1" (corresponding to "YES") has been selected, then confirm with key ⧄FIN/END or ↵ , the programs will be deleted. If "0" is entered (default value corresponding to "NO"), the deletion function is quit (the selected program(s) will not be deleted).

⚠ **CAUTION**: Deleting program(s) which is/are part of a multiple part program is only possible if you first delete the multiple program which contains it. Otherwise, an error message as below will be shown.

1.10.2. DELETE PUNCHES

"DELETE" ⟹ "Punches" is used to delete program(s) existing in the NC library. The procedure is similar to the program deletion.
An intermediate page (see below) will be displayed asking for a PARTIAL ("1") or TOTAL ("2") deletion of the punch(es) in memory.

mm / Mem   HFE100-3   V 6.03   G 6.03   09/05/06
DELETE   Punches

DELETE (1=ALL / 2=PARTIAL):2

If "2" has been selected (TOTAL deletion), and confirm with key ⏎ or ⟋END, another page will be displayed to confirm the deletion of ALL the punches in memory:

- If total deletion has been selected, all the punches in the NC library will be deleted.

- If partial deletion has been selected, select the punch(es) to be deleted with keys ↓, ↑, ➡ and ⬅, if your library has numerous pages.

mm / Mem   HFE100-3   V 6.03   G 6.03   09/05/06
DELETE   Punches

N°DIE	ANGLE deg	RADI. mm	HEIT mm	TONNAG kN/m
9001	90.0	0.80	66.65	1200
9002	90.0	0.80	89.65	1200
9003	90.0	0.80	104.65	625
9004	90.0	0.60	84.18	447
9005	90.0	0.60	119.65	447
9006	90.0	0.60	99.40	269
9007	90.0	0.60	94.40	447
9008	90.0	0.20	89.92	1200
9009	90.0	0.20	66.90	1200
8801	88.0	0.80	66.65	1200
8802	88.0	3.00	65.50	1200
8803	88.0	0.80	89.65	1200
8804	88.0	3.00	88.50	1200
8805	88.0	0.80	104.65	625
8806	88.0	3.00	103.50	625
8807	88.0	0.60	119.66	447
8808	88.0	0.60	99.40	269
8809	88.0	0.60	94.40	447
6001	60.0	0.80	67.00	1200
6002	60.0	2.00	65.80	1200

PUNCH NUMBER   25
Maximum PUNCH   30

DELETE CONFIRMATION ? (1=Y/0=N)0

Follow this procedure:

- Select the punches to be deleted with key ⏎. The selected punches will be highlighted. Press key ⏎ to deselect them, the reverse video will disappear.

- Press key ⟋END when all the punches have been selected. Confirm the deletion "DELETE CONFIRMATION? (1=Y/0=N)" with "0" as a default value.

- Enter "1" to continue; press key ⏎ or ⟋END to validate. The selected punches will be deleted.

But if you decide not to carry on with deleting the punches, with a default value of 0 (no deletion), simply press ⏎ or ⟋END in which case none of the punches will be deleted.

⚠ **CAUTION**: You cannot delete a punch, which is being used in a program unless you delete the program before. If, in the selected programs, you have a punch which is used inside a part program, then following message "ILLEGAL FUNCTION, PUNCH USED <CR> TO

CONTINUE" will be displayed and no punch will be deleted when key ⏎ has been pressed.

1.10.3. DELETE DIES

The procedure for deleting a/several die(s) is exactly the same as for deleting punch(es) with the same restrictions.

1.10.4. DELETE DIE-HOLDERS

The procedure is similar to the punches deletion except that all die-holders may be selected from the same page.

1.10.5. DELETE SPACERS

The procedure is similar to the punches deletion except that all spacers may be selected from the same page.

1.11. COPY

1.11.1. COPY PROGRAMS

"COPY" ⟹ "Programs" is used to copy program(s) existing in the NC library. The procedure is similar to the program deletion. A page (identical to display "LIST" ⟹ "Programs" or "DELETE" ⟹ "Programs") will be displayed.

- Select the program to be duplicated. Use keys ←,→,↑,↓ to browse around the page. Position the cursor on the selected program; then, validate with ⏎ key. The selected programs will be highlighted. The following page is shown:

- Enter new reference number under which you like to save the duplicated program. If this number is already assigned to another program in the NC memory, N.C. the following error message is displayed:

- Press key ⏎ and re-enter the name of a program. Once keyed-in and confirmed with key ⏎, all the programs in the NC library will be displayed, you may now select another program to be duplicated. Press key ▨END to quit.

1.11.2. COPY PUNCHES

"COPY" ⟹ "Punches" is used to copy punch(es) existing in the NC library. The procedure is similar to the programs copy.

- Select the punch to be copied with keys ↓, ↑, ➡ and ⬅, if your NC library is composed of several pages.

- Once the cursor has been positioned on the selected punch, press key ⏎ and enter a number reference for the duplicated punch. If the number has already been affected to another punch from the NC library, an error message will be displayed as below:

- Press key ⏎ and re-enter the punch number. Once keyed-in and confirmed with key ⏎, the list of punches from the NC library is displayed, you may then select another punch to be copied. Press key ▨END to quit.

1.11.3. COPY DIES

The procedure for copying a die(s) is exactly the same as for copying punch(es) with the same restrictions.

1.11.4. COPY DIE-HOLDERS

The procedure is similar to the punches copy except that all die-holders may be selected from the same page.

1.11.5. COPY SPACERS

The procedure is similar to the punches copy except that all spacers may be selected from the same page.

1.12. SAVE

1.12.1. SAVE PROGRAMS

"SAVE" ⟹ "Programs" is used to save program(s) existing in the NC library to an external memory. Selecting this function displays an intermediate page (see next page) where you should choose between a TOTAL ("2") or PARTIAL ("1") back-up; in the first place, ALL the programs in memory will be saved, in the second one, only one or several programs will be saved.

- If "1" is selected, (ALL), and confirm with key ⏎ or ⏚, the back-up of the programs, punches and dies from the NC memory to the external memory can start; at the same time, you are informed by a specific message that the operation is in process.

  If you enter "2", (PARTIAL), and confirm it with key ⏎ or ⏚, a display similar to that for "LIST" ⟹ "Programs" is shown where the programs to be saved should be selected.

- Scroll through with keys ⬅,➡,⬆,⬇.

- Position the cursor on the selected program, and confirm with key ↵ to select it. The program is displayed in video reverse. Process the same way for the other programs to be saved. Deselect it by pressing once again key ↵ with the cursor positioned above the program. The video reverse disappears, confirming that the program is not selected any longer. Once selected, press key 🔑FIN/END; the back-up may start. You are then requested by the NC to confirm the back-up of the selected programs:

If a selected program has already been saved on the external memory, the program number is shown followed by a message (see below):
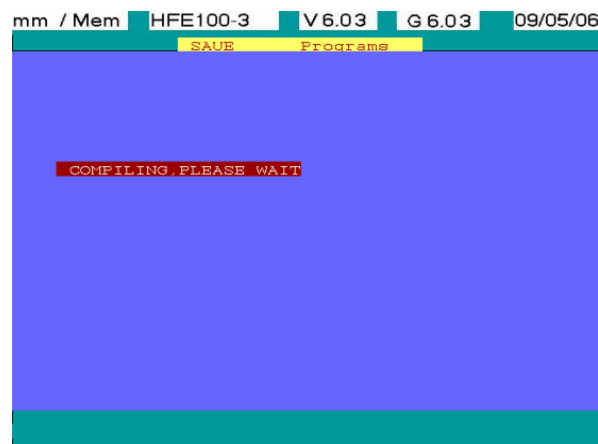
Enter "1" if you want to replace the existing program, then press key 🔑FIN/END or ↵. Otherwise, just press key 🔑FIN/END to confirm "0" (default value). In the first case, the program will be replaced; it will be ignored in the second one. Once the back-up process is finished, go back to "SAVE" menu.

- If "0" is entered (default value) the saving function is quit and no program will be saved on the floppy disk.

NOTE If the floppy disk is missing from the drive or is either damaged or is read / write protected, then the NC will display the following message.

- If you enter "0" (default value), then confirm with key ⏎ or ⊠ᴇɴᴅ , the function is deleted.
- If you enter "1", insert external memory or replace the faulty one , then confirm with key ⏎ or ⊠ᴇɴᴅ .

1.12.2. SAVE PUNCHES

"SAVE" ⟹ "Punches" is used to save punch(es) existing in the NC library to the external memory. The procedure is similar to the programs back-up. Selecting this function displays an intermediate page (see below) where you should choose between a TOTAL ("2") or PARTIAL ("1") back-up; in the first place, ALL the punches in memory will be saved, in the second one, one or several punches will be saved.

- If you enter "1" and confirm it with key ⏎ or ⊠ᴇɴᴅ , all the punches will be saved on the external memory:
- If you enter "2" and confirm it with key ⏎ or ⊠ᴇɴᴅ , select the punch(es) to be saved with keys ↓ , ↑ , ➡ and ⬅ , if your library is composed of several pages. Proceed as follows:

  - Select the punch(es) to be saved with key ⏎ . The selected punches will be displayed in video reverse. Press key ⏎ , to deselect it; the video reverse will disappear.

  - Press key ⊠ᴇɴᴅ when all punches have been selected. You are requested by the NC to

confirm it as shown on the display below:
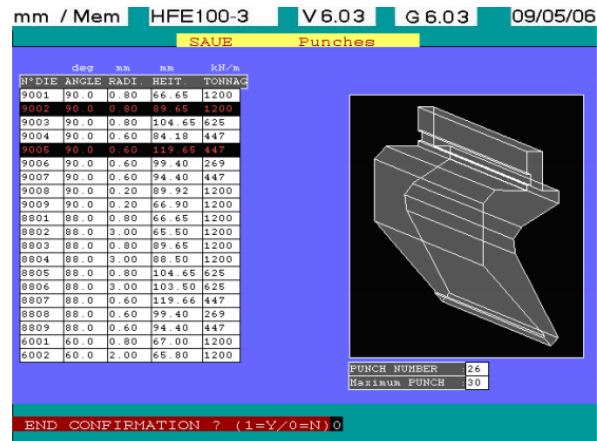
- Enter "1" to continue; press key ⏎ or ⧸END to confirm. The selected punches will be deleted from the memory.
- Press key ⏎ or ⧸END if you want to quit; in this case, the function will be deleted and no punch saved.

1.12.3. SAVE DIES

The procedure for saving a die(s) is exactly the same as for saving punch(es) with the same restrictions.

1.12.4. SAVE DIE-HOLDERS

The procedure is similar to the punches back-up except that all die-holders may be selected from the same page.

1.12.5. SAVE SPACERS

The procedure is similar to the punches back-up except that all spacers may be selected from the same page.

1.12.6. SAVE PARAMETERS

"SAVE" ⟹ "Parameters" is used to save the machine parameters to the external memory. Once executed, the following screen is displayed:

mm / Mem　HFE100-3　V 6.03　G 6.03　09/05/06

SAUE　　　Parameters

ERASE DATA->FD CONFIRM(1=Y/0=N) 0

- Enter "1" and confirm with key ⏎ or ⊠FIN/END, all parameters on the external memory will be cleared while allowing parameters from your NC memory to be saved to the external memory.

- Enter "0" (default value), then confirm with key ⏎ or ⊠FIN/END, the parameters on the external memory will not be cleared; the back-up function will be quit.

Since these parameters are essential to the press-brake operation, it is highly recommended to save them (machine parameters, fingers, language).

1.13. RESTORE

1.13.1. RESTORE PROGRAMS

"RESTORE" ⟹ "Programs" is used to restore a program(s) from the external memory to the NC library. Once selected, an intermediate page (see below) is displayed asking whether a PARTIAL ("1") or TOTAL ("2") restoration should be carried out; in the first place, only one or several programs will be restored; in the second one, all of them will be restored from the external memory to your NC memory.

mm / Mem　HFE100-3　V 6.03　G 6.03　09/05/06

RESTORE　　Programs

TRANSFER(1=ALL / 2=PARTIAL): 0

The procedure is similar to the programs copy apart from the fact that you are requested by the NC to confirm the deletion of all the programs in the NC memory before restoring those contained on the external memory as shown below:

- Enter "1", then confirm with key ⏎ or FIN/END, the programs contained in the NC memory will be deleted while being able to continue restoring the selected program(s), from the external memory to the NC memory.

- Enter "0" (default value), then confirm with key ⏎ or FIN/END, the programs contained in the NC memory will not be deleted; the restoring function will be quit and no program restored.

1.13.2. RESTORE PUNCHES

"RESTORE" ⟹ "Punches" is used to restore a punch(es) from the external memory to the NC library. Once selected, an intermediate page (see below) is displayed asking whether a PARTIAL ("1") or TOTAL ("2") restoration should be carried out; in the first place, only one or several punches will be restored; in the second place, all of them will be restored from the external memory to your NC memory.

The procedure is similar to the punches back-up apart from the fact that you are requested by the NC to confirm the deletion of all the punches contained in the NC memory before restoring those contained on the external memory as shown below:

RESTORE   Punches

INSERT FD(1=CONTINUE/0=QUIT)0

- Enter "1" then confirm with key ⏎ or ⊠FIN/END, the punches contained in the NC memory will be deleted while being able to continue restoring the selected punch(es), from the external memory to the NC memory.

- Enter "0" (default value) then confirm with key ⏎ or ⊠FIN/END, the punches contained in the NC memory will not be deleted; the restoring function will be quit and no punch restored.

1.13.3. RESTORE DIES

"RESTORE" ⟹ "Dies" is used to restore a die(s) from the external memory to the NC library. Once selected, an intermediate page (see below) is displayed asking whether a PARTIAL ("1") or TOTAL ("2") restoration should be carried out; in the first place, only one or several dies will be restored; in the second place, all of them will be restored from the external memory to your NC memory.

RESTORE   Dies

TRANSFER(1=ALL / 2=PARTIAL):0

The procedure is similar to the dies back-up except that you are requested by the NC to confirm the deletion of all the dies contained in the NC memory before restoring those contained on the external memory disk as shown below:

```
mm / Mem   HFE100-3    V 6.03    G 6.03     09/05/06
                    RESTORE    Dies




 INSERT FD(1=CONTINUE/0=QUIT) 0
```

- Enter "1" then confirm with key ⏎ or ◿ FIN END, the dies contained in the NC memory will be deleted while being able to continue restoring the selected die(s), from the external memory to the NC memory.

- Enter "0" (default value) then confirm with key ⏎ or ◿ FIN END, the dies contained in the NC memory will not be deleted; the restoring function will be quit and no die restored.

1.13.4. RESTORE DIE-HOLDERS

"RESTORE" ⟹ "Die-holders" is used to restore a die-holder(s) from the external memory to the NC library.
Once selected, an intermediate page (see below) is displayed asking whether a PARTIAL ("1") or TOTAL ("2") restoration should be carried out; in the first place, only one or several die-holders will be restored; in the second place, all of them will be restored from the external memory to your NC memory.


```
mm / Mem   HFE100-3    V 6.03    G 6.03     09/05/06
                    RESTORE    die Holders




 TRANSFER(1=ALL / 2=PARTIAL): 0
```

The procedure is similar to the die-holders back-up except that you are requested by the NC to confirm the deletion of all the die-holders contained in the NC memory before restoring those contained on the external memory as shown below:

- Enter "1" then confirm with key ⏎ or ◱ END, the die-holders contained in the NC memory will be deleted while being able to continue restoring the selected die-holder(s), from the external memory to the NC memory.

- Enter "0" (default value) then confirm with key ⏎ or ◱ END, the die-holders contained in the NC memory will not be deleted; the restoring function will be quit and no die-holder restored.

## 1.13.5. RESTORE SPACERS

"RESTORE" ⟹ "Spacers" is used to restore a spacer(s) from the external memory to the NC library. Once selected, an intermediate page (see below) is displayed asking whether a PARTIAL ("1") or TOTAL ("2") restoration should be carried out; in the first place, only one or several spacers will be restored; in the second place, all of them will be restored from the external memory to your NC memory.

The procedure is similar to the spacers back-up except that you are requested by the NC to confirm the deletion of all the spacers contained in the NC memory before restoring those contained on the external memory as shown below:

- Enter "1" then confirm with key ⏎ or ⧄END, the spacers contained in the NC memory will be deleted while being able to continue restoring the selected spacer(s), from the external memory to the NC memory.

- Enter "0" (default value) then confirm with key ⏎ or ⧄END, the spacers contained in the NC memory will not be deleted; the restoring function will be quit and no spacer restored.

1.13.6. RESTORE PARAMETERS

"RESTORE" ⟹ "Parameters" is used to restore machine parameter(s) from the external memory to the NC library.
The procedure is similar to the parameters back-up except that you are requested by the NC to confirm the deletion of all the parameters contained in the NC memory before restoring those contained on the external memory as shown below:


- Enter "1", then confirm with key ⏎ or ⧄END, the parameters contained in the NC memory will be deleted while being able to continue restoring the selected parameter(s), from the external memory to the NC memory.

- Enter "0" (default value) then confirm with key ⏎ or ⧄END, the parameters contained in

85

the NC memory will not be deleted; the restoring function will be quit and no machine
parameter replaced.

1.13.7. RESTORE MACHINE PROFILE

"RESTORE" ⟹ "PROFILE" is used to restore the backgauge fingers and profile of the machine from
the external memory to the NC library.

1.13.8. RESTORE LANGUAGE

"RESTORE" ⟹ "Language" is used to restore the NC language from the external memory to the NC
memory.

1.14. TOOLS

1.14.1. EXPORTING

PROGRAMS

"TOOLS" ⟹ "Exporting programs" is used to export one or several program(s) existing in the NC
library to a drive and a directory as requested (exporting path).

- Use key [←] or [→] to select a specific drive for the exportation of one or several program(s). If the machine is not connected on line, only "[A]" and "[B]" appear; only the external memory drive will be available. The NC will position directly on the previously specified path. Once the cursor has been positioned on the selected drive, confirm your choice with key [↵]. As shown below, the directories contained in this drive will be displayed:
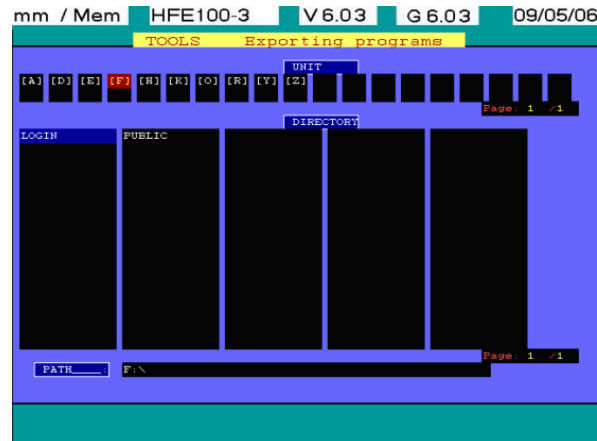

```
mm / Mem   HFE100-3    V 6.03    G 6.03    09/05/06
            TOOLS     Exporting programs
                              UNIT
[A] [D] [E] [F] [H] [K] [O] [R] [Y] [Z]
                                              Page: 1 /1
                            DIRECTORY
LOGIN       PUBLIC




                                              Page: 1 /1
   PATH     F:\
```

The directories contained in the selected drive will be displayed 3 in a row (corresponding to the 3 windows in the lower part of the page) and throughout several pages (21 directories on each page). Press key [➡] to go to the next page and key [⬅] for the previous one. Use key [←],[→],[↑],[↓] to select the transfer directory for the exported programs. Position the cursor on the selected directory then confirm with key [↵]. Your NC screen will then display the existing sub-directories which are potentially contained in the previously selected directory. If no directory is present, ". ." is displayed in the extreme left window.

Validating both dots by pressing key [↵] helps moving upward into the tree structure of the current path. You may create a directory from the current path by pressing key [◀◀]. Enter the name (in that respect follow the recommendations given under MS Dos environment). A new directory will be created enabling the exportation of program(s). The current path is displayed in the lower part of the screen enabling the selected importing path to be quickly checked.

Once the path for the importing programs has been specified, press key [FIN/END]. You are then requested by the NC to confirm it as shown below:
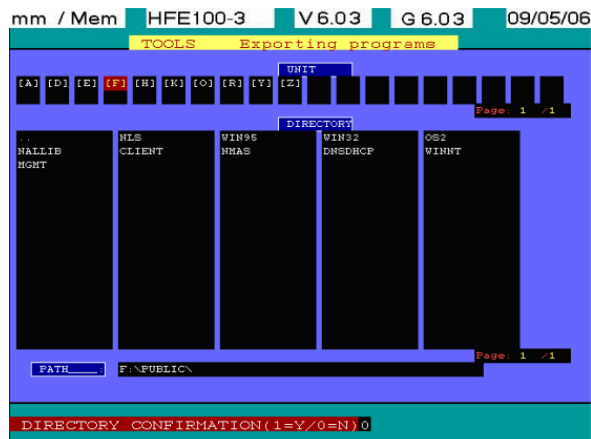
- If "0" is entered (default value), confirming it with key ⏎ or FIN/END will delete the exporting function (no program will be exported).

- If "1" is entered, confirm it with key ⏎ or FIN/END, a display will appear (identical to "LIST" ⟹ "Programs") listing all the programs contained in the NC memory. From that stage, the procedure for selecting programs to be exported is similar to the previous instructions: go to every program to be exported, then select it with key ⏎, it is then highlighted.

- Once the selection is finished, press key FIN/END to start exportation. A message will then be displayed asking whether you want to replace automatically the programs with a similar name and already existing in the exportation path (see below):

- If "1" is entered, confirm it with key ⏎ or FIN/END the programs will be exported and will automatically replace the others of the same name in the exportation path.

- If "0" is entered (default value), confirm with key ⏎ or FIN/END, the programs will be exported WITHOUT automatically replacing the others of the same name in the exportation path. Should a program with the same name be found, a display as below would be shown asking whether you want to replace or not the program existing in the exportation path.

- If "1" is selected, confirm it with key ⏎ or FIN/END, the program existing in the exportation path will be replaced by the one selected for exportation in the NC memory. If "0" is selected (default value), confirm it with key ⏎ or FIN/END, the program existing in the exportation path will be kept and the one selected for exportation in your NC memory will be ignored; the

88

exporting function will continue to the next selected program.

During importation, a display as shown below will give information regarding the importation process specifying which program is being imported as well as the importation path.
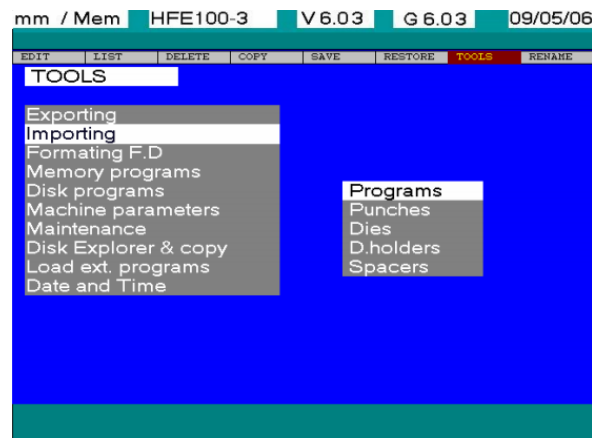

PUNCH Used to export one or several punches to the previously-selected path, this procedure is similar to the programs one.

DIE (similar).

HOLDER (similar).

SPACER (similar).

1.14.2. IMPORTING

PROGRAMS

"TOOLS" ⟹ "Import programs" is used to import program(s) contained from a drive or a directory (importation path) to your NC library.

- Use key ← or → key to select the drive for the importation of the required program(s). If the machine is not connected on line, only "[A]" and "[B]" will appear; only the external memory will be available. The NC will position directly on the previously specified path. Once the cursor has been positioned on the selected drive, confirm your choice with key, ↵ . The directories contained on this drive will be displayed as shown below:

The directories contained in the selected drive will be displayed 3 in a row (corresponding to the 3 windows in the lower part of your page) and throughout several pages (21 directories on each page). Press key ➡ to go to the next page and key ⬅ for the previous one. Use keys ←,→,↑,↓ to select the source directory of the programs to be imported. Position the cursor on the selected directory then confirm with key ↵ . Your NC screen will then display the existing sub-directories which are potentially contained in the previously selected directory. If no directory is present, ". ." is displayed in the extreme left window. Validating both dots by pressing key ↵ helps moving upward into the tree structure of the current path. You may create a directory from the current path by pressing key ◀◀ . The current path is displayed in the lower part of the screen enabling the selected importing path to be quickly checked. Once the path for the importing programs has been specified, press key FIN/END . You are then requested by the NC to confirm it as shown below:

- If "0" is entered (default value), then confirm with key [↵] or [FIN/END]; the importation function will be deleted (no program will be imported).

- If "1" is entered, then confirm with key [↵] or [FIN/END], a display will be shown with the list of the programs contained in the importation path as selected (see below):

The screen is composed of:

- In the upper part of the page, 5 windows called "PROGRAMS" showing the programs contained in the selected importation path, laid out in 5 columns, 7 rows with 35 programs max. per page. Press key [→] to go to the next page, and key [←] to go to the previous one.
- The lower part of the page contains a second set of 5 windows called "SELECTED PROGRAMS" where the programs, as selected by you, are displayed in a pattern of 5 columns, 2 rows with 10 programs selected and simultaneously displayed. Use [←], [→], [↑], [↓] to select the programs to be imported.

Go to the requested program and select with key ⏎.

It will then appear in one of the windows "SELECTED PROGRAMS". The selected programs are also displayed in video reverse in the upper windows called "PROGRAMS". Once the program(s) to be imported have been selected, a display as below will be shown:

Press ⏍ key to start the importation. A message will then be displayed asking whether you want to replace automatically the programs with the same name and already existing in the importation path (see below):

- If "1" is entered, confirm with key ⏎ or ✍END, the programs will be imported and replace automatically those with the same name already contained in your NC library.

- If "0" is entered (default value), confirm with key ⏎ or ✍END, the programs will be imported WITHOUT automatically replacing those with the same name in your NC library. Should a program with the same name be found, the display as shown below would appear, asking whether you want to replace the program existing in your NC library by the one from the importation path:

- If "1" is entered, confirm with key ⏎ or ✍END, the program existing in your NC library will be replaced by the one selected for importation.

- If "0" is entered (default value), confirm it with ⏎ or ✍END, the program existing in the NC library will be kept and the one selected for importation will be ignored. The importation function will continue by moving to the next selected program.

During importation, a display as below will be shown specifying how the importation is being carried out as well as the importation path.

PUNCH Allows you to import one or more punches to the path chosen previously, this
procedure is the same as for programs.
MATRIX (identical procedure).
HOLD (identical procedure).
RISE (identical procedure).

1.14.3. FORMATTING FLOPPY DISK

"TOOLS" ⟹ "Formatting FD (floppy disk)" enables you to prepare your new floppy disk(s) in order to be used on your NC control. (Standard formatting DOS PC 1.44MB).

Once selected, the following display is shown asking whether all the data contained in the floppy disk should be deleted or not:

- If "0" is selected (default value), confirm with key ⏎ or ✍END, the function will be deleted.

- If "1" is selected, then confirm with key ⏎ or ✍END, all the data contained on the floppy will be deleted; the formatting operation will start.

A screen (see below) is specifying how the process is progressing.
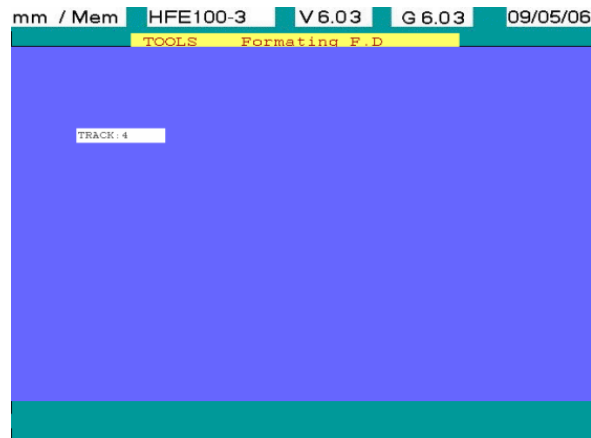
⚠ **CAUTION**: To remove a floppy disk from the drive, wait until the read / write indicator light is off. Failures to do so could damage your floppy disk or even the drive.

If an error message is displayed, check that the floppy has been correctly inserted into the drive then start the formatting operation again. Should the problem occur, your floppy disk might be faulty and should be replaced.

1.14.4. MEMORY PROGRAM

"TOOLS" ⟹ "Memory program" enables the operator to be able to work with the NC memory again once finished with a task on the external memory. "Mem" will then be displayed in the upper part of your NC.

1.14.5. F.D (FLOPPY DISK) PROGRAM

"TOOLS" ⟹ "F.D programs" enables the operator to work directly on the external memory.

NOTE A message will be displayed if the floppy is missing or cannot be read.

    - If "0" is selected (default value), then confirm with key or, the function will be deleted.

A page is then displayed specifying that the formatting is in progress.

"F.D" is shown in the upper part of your NC, to inform you that you are working from the external memory.

1.14.6. MACHINE PARAMETERS

As shown below, this function is reserved to AMADA technicians. "TOOLS" ⟹ "Machine parameters" is used to enter different parameters specific to your machine.

1.14.7. MAINTENANCE

As shown below, this function is reserved to AMADA technicians. "TOOLS" ⟹ "Maintenance" is used to diagnose some items specific to your machine for curative maintenance if necessary.

1.14.8. DISK EXPLORER AND COPY

As shown below, this function is reserved to AMADA technicians. "TOOLS" ⟹ " F.D explorer & copy" is used to browse through an external memory and if necessary copy some items from this disk to your NC memory.

1.14.9. LOAD EXTERNAL PROGRAM

As shown below, this function is reserved to AMADA technicians. "TOOLS" ⟹ "Load ext. program" is used to allow the execution of an external program to the NC.
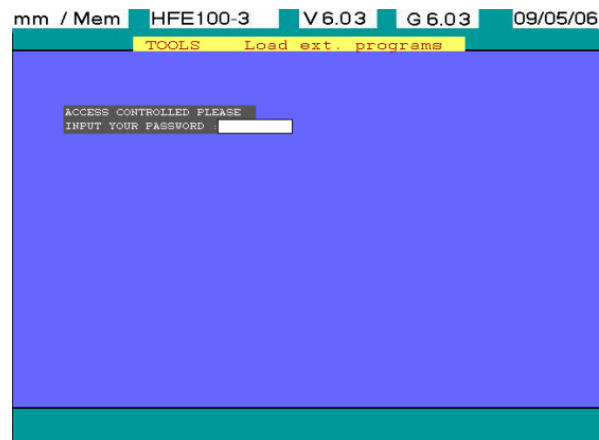
1.14.10. DATE AND TIME

"TOOLS" ⟹ "Date and time" is used to allow the setting on your NC of current date and time.

Use key ⟶ or ⟵ , to position the cursor on the value you want to change then enter the new one.
Once modified press key [FIN/END] to quit or ⟶ if the cursor is positioned in the lower right field (seconds).
Modifications will be considered if the values entered are convenient. If not, the older values will be kept.

1.15. RENAME

1.15.1. RENAME PROGRAMS

"RENAME" ⟹ "Programs" is used to rename the programs contained in your NC memory.
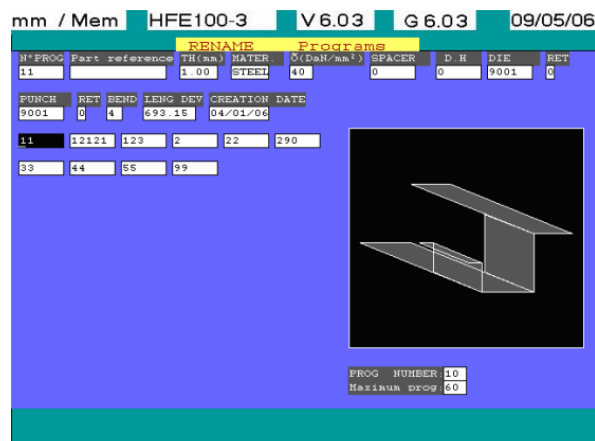
Use keys ⬅,➡,⬆,⬇ and position the cursor above the program you want to rename. Then enter directly the new program number and confirm with key ↵. Follow the same procedure for all the programs you want to rename. Once finished, press key [FIN/END] to quit. If you press key [FIN/END] before validating the name you have just entered, the function is quit without renaming the program.

This function is also used to rename the programs existing on your external memory if working directly on it (see page below):

1.15.2. RENAME PUNCHES

"RENAME" ⟹ "Punches" is used to rename the punches existing in your NC memory.
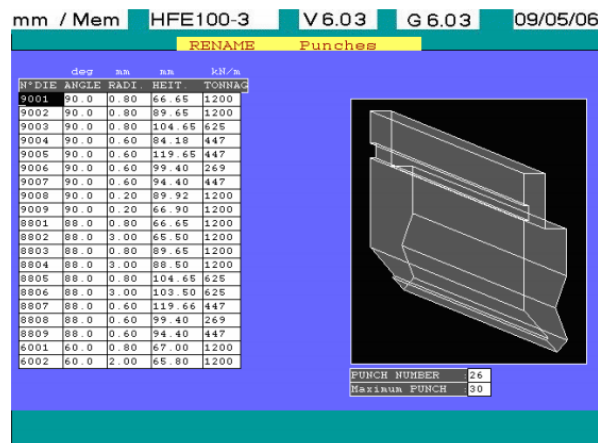
Use keys ⬆,⬇ and position the cursor above the punch you want to rename, as well as keys ➡ and ⬅ to scroll through the existing punches. Then enter directly the new punch number and confirm with key ↵. Follow the same procedure for all the punches you want to rename. Once finished, press key [FIN/END] to quit. If you press key [FIN/END] before validating the name you have just entered, the function is quit without renaming the punch. This function is also used to rename the punches existing on your external memory if working directly from it as shown below:
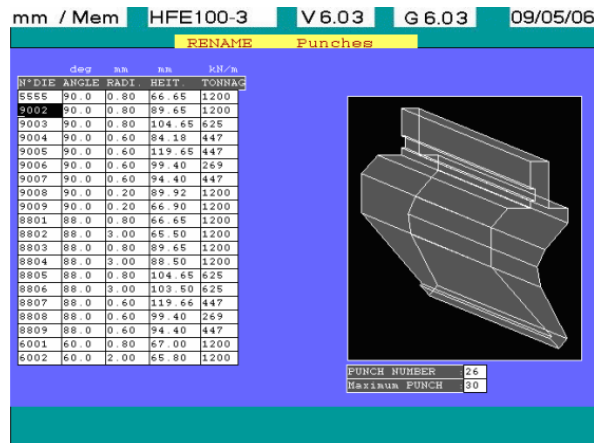
1.15.3. RENAME DIES

The procedure for renaming die(s) is similar to punches.

1.15.4. RENAME DIE-HOLDERS

The procedure is similar to the punches renaming except that all die-holders may be selected from the same page.

1.15.5. RENAME SPACERS

The procedure is similar to the punches renaming except that all spacers may be selected from the same page.

1.16. ERROR MESSAGES

On screen displayed messages (Following listed messages may change depending on the software versions).

- "RESET THE MACHINE" Reset with the pump SA1 key switch on the NC panel. SLB5 green light is lit again.

- " B.G. UNMATCHED SAFETY" This message appears when one axis value (X1 – X2) ≥ is greater than the maximum permissible value.

- "UNMATCHED V AND THICK CONFIRM (1 = Y / 0 = N)" The opening of the selected die is either too narrow or too large in respect to the material thickness programmed for bending; however you may override the warning by entering "1".

- "PUNCH/DIE UNMATCHED CONFIRM (1 = Y / 0 = N)" The punch angle is different from the die's; however you may override the warning by entering "1".

- "PROGRAM INCOMPATIBLE WITH THIS MODE" The selected program number has already been used in another operating mode.

- "BEND ANGLE SMALLER THAN TOOL ANGLE" The programmed angle is inferior to the punch angle.

- " MAX NUMBER OF BENDS ACHIEVED" You can not program any more bends as you are exceeding maximum number (25).

- "UNKNOWN PROGRAM" The requested program does not exist in the memory (NC or external).

- "UNKNOWN PUNCH" Punch does not exist in the corresponding library.

- "UNKNOWN TOOL" Die holder or spacer does not exist in the corresponding libraries.

- "EXISTING, REPLACE (1 = Y / 0 = N)" Tool already existing in the library.

- "OUT OF LIMIT VALUE" Value programmed exceeds axes limits.

- "UNKNOWN DIE" Die does not exist in the library.

- "LIBRARY FULL" Maximum number achieved.

- "UNKNOWN PROG LIBRARY" Specified program directory does not exist.

- "UNKNOWN PUNCH LIBRARY" Specified punch directory does not exist.

- "UNKNOWN DIE LIBRARY" Specified die directory does not exist.

- "EMPTY LIBRARY" Directory empty.

- "UNKNOWN DIE-HOLDER CATALOGUE" Specified die-holder directory does not exist.

- "UNKNOWN SPACER CATALOGUE" Specified spacer directory does not exist.

- "UNKNOWN MACHINE PARAMETERS" Specified machine parameter does not exist.

- "BEND PROGRAM ERROR" Can not read a bending program.

- "PRESS CLOSING SWITCH" Press closing foot pedal.

- "NO BEND IN THE PROGRAM" Program stored without any data.

- "PRESS OPENING FOOTSWITCH" Press opening foot pedal.

- "BENDING SEQUENCE ERROR" NC not getting closing signal for bending.

- "GAUGING BEND NUMBER ERROR" NC can not position axes on a given bend.

- "PROGRAM INCOMPATIBLE WITH THIS MODE" Operating mode mismatch for a given program.

- "GONE PAST THE MAX AXIS OFFSET" Axis position outside the tolerance.

- " ILLEGAL FUNCTION, INITIALIZE MACHINE " Can not use this function and machine should be reset.

- "ILLEGAL FUNCTION" Function not allowed.

- "ILLEGAL FUNCTION TOOL USED" Can not delete tool as it's used in a program. Delete the program first.

- "TONNAGE OUT OFF LIMIT" Force exceeding machine total tonnage.

- "EXISTING PROGRAM" Can not use this program number as program already exists.

- "WAITING B.G. POSITIONING" Waiting for backgauge to position after dwell time.

- "MAX DELTA AXIS EXCEEDED" Exceeding maximum permissible offset value for the axes.

- "CRITICAL MINIMUM FLANGE" Minimum flange length for a given die reached.

- "DELETE CONFIRMATION? (1 = Y / 0 = N)" Delete one or several elements confirmed or not.

- "END CONFIRMATION? (1 = Y / 0 = N)" Execution of one action confirmed or not.

- "EXIST REPLACE (1 = Y / 0 = N)" Existing element to be replaced or not.

- "UNDEFINE DIRECTORY" Selected directory not defined.

- "COM. ERROR" Communication error.

- "FILE EMPTY" File empty.
- "COMPILING PLEASE WAIT" Operation in progress.

- "ILLEGAL FUNCTION, PROGRAM USED" Program cannot be deleted because belonging to a multiple program.

- "ERASE DATA → FD CONFIRM (1 = Y / 0 = N)" Data on external memory deleted or not.

- "ERASE DATA → MD CONFIRM (1 = Y / 0 = N)" Data in the NC memory deleted or not.

- "INSERT FD (1 = CONTINUE / 0 = QUIT" Inserting external memory (FD or USB) in the drive or not.

- "ERROR MOUVMENT" Positioning error of either axis.

- "EXISTING PROGRAM" Program already existing.

- "TRANSFER (1 = PARTIAL / 2 = ALL" Type of transfer to be selected: selected elements only or all elements.

- "MULTIPLE PROGRAM" In the editing control of a multiple program, it is specified that the selected program to be part of a multiple program is already a multiple program itself.

ERROR SYSTEM

HEIGHT OVER LIMIT

WIDTH OVER LIMIT

DO YOU WANT TO REDO THE PART? (1=Y/0=N)

UNMATCHED TONNAGE CONFIRM? (1=Y/0=N

ILLEGAL FUNCTION, INITIALIZE MACHINE

ILLEGAL FUNCTION, PROGRAM USED

RETURN TO MANUAL MODE

TONNAGE > TONNAGE TOOLS OR MACHINE

UNDER COMMUNICATION DEVELOPMENT

START GRAPHIC ANGLE

MINIMUM OPENING (0-N/1-Y)

HYD. STOP SAFETY

DIRECTORY CONFIRMATION(1=Y/0=N)

DISABLE DIRECTORY

EMPTY LIBRARY

NO PROGRAM FILE FIND

AUTOMATIC REPLACE EXIST PROGRAMS(1=Y/0=N)

DELETE (1=ALL / 2=PARTIAL

DELETE. ALL PROGRAMS CONFIRM(1=Y/0=N):

DELETE. ALL DIE HOLDERS CONFIRM(1=Y/0=N):

DELETE. ALL DIES CONFIRM (1=Y/0=N

DELETE. ALL PUNCHS CONFIRM (1=Y/0=N

DELETE. ALL DIE BLOCKS CONFIRM(1=Y/0=N):

PROGRAM PAGE ALREADY CREAT

PROGRAM PAGE UNKNOWN

OBJECTS LISTED

SELECTED

NO PROGRAM LOADED ON NC

SEND FUNCT. NOT ALLOWED, OBJECT LOADED ON NC

NO OBJECT TO LIST

OBJECT USED FOR PROGRAM ON NC

TRANSMISSION ERROR

UNMATCHING INDEXES

UNKNOWN LIBRARY

COMMUNICATION IN PROGRESS

PUSH 'A' TO STORE 'S' TO LEAVE VALUES

CORRECT BENDING

OUT OF TOLERENCE BENDING

UPDATE DATABASE? (0=N/1=Y)

FLOPPY DISK ERROR

USE DATABASE? (0=N/1=Y)

HOLD START BUTTON

FACTORY PARAMETERS ALREADY SAVE

UNKNOWN FACTORY PARAMETERS

VALVE YV2.1 DOESN'T WORK PROPERLY

VALVE YV2.2 DOESN'T WORK PROPERLY

STOP TRACE PROGRAMMED

LANGUAGE INCOHERENT WITH SOFT

INITIALIZE SAVETY LASER SYSTEM

TONNAGE < TONNAGE Mini MACHINE

1.17. ALPHANUMERIC INPUT ON NC

Following data fields can take alphanumeric characters:

- Program number.
- Work-piece reference number.
- Spacer.
- Die-holder.
- Punch.
- Die.
- Reference numbers for special Punches or Dies when created.
- Directories on selected drives.

NOTE In programs numbers, following characters: [*space*] + - * / are prohibited.

If a data which does not correspond to above listed (for instance: material thickness) then it should not be entered alphanumerically, as this value will not be accepted by the NC. The cursor will be stuck in this field until a coherent value is entered.

Alphabetic characters may be entered together with the numeric keypad (for instance a job reference: ABC/123).

INPUT PROCEDURE

Select the field (for instance: workpiece reference number). Press key ☐ simultaneously with key ◄◄ to move to alphanumeric. All the characters available will be displayed in the lower part of the page (see below).

Press → and ← to select the requested character. Once the cursor is positioned on it, confirm with key ↵; follow the same procedure for the following characters. Figures may be entered at any time with your NC numerical keys; delete the lastly entered character, then press key ✎. Once entered, quit the alpha-numerical mode by pressing simultaneously keys ☐ and ◄◄.