DNC 600S

User Guide

Information in this document is subject to change without notice, and does not represent a commitment on the part of CYBELEC SA.

The software described in this document is furnished under a licence or nondisclosure agreement. The software may be used or copied only in accordance with the terms of the agreement. It is against the law to copy the software on any medium except as specifically allowed in the licence or nondisclosure agreement.

Copyright CYBELEC SA. 1991

Important:

This notice explains normal and standard programming operations for the numerical control.
In view of the fact that numerical controls can be equipped with configurable functions by the press manufacturer for his own specific purposes, please refer to the manufacturer-supplied complementary instructions regarding the programming of these functions.

Autocad$^{®}$ is a registered trade mark of Autodesk Inc..

CYBELEC$^{®}$ is a registered trademark of CYBELEC SA.

Ethernet$^{®}$ is a registered trade mark of Xerox Corporation.

IBM$^{®}$, PC/AT$^{®}$, PC Network$^{®}$, Token Ring Network$^{®}$
    are registered trade marks of the International Business Machines Corporation.

MS-DOS$^{®}$ is a registered trade mark of Microsoft Corporation.

MS-Windows$^{®}$ is a registered trade mark of Microsoft Corporation.

Novell Netware^{TM} is a registered trade mark of Novell, Incorporated.

Windows NT$^{®}$ is a registered trade mark of Microsoft Corporation.

SAFETY AND MAINTENANCE INSTRUCTIONS

- The operator must be trained for working with the machine on which the numerical control is installed.

- Improper use of the numerical control can cause heavy damage on equipment and/or injuries to people.

- Modification of machine parameters can cause important material damage or lead to irregular product quality.

- The rear panel may only be removed by a qualified technician (danger of electrocution).

- Do not expose the numerical control to excessive humidity so as to avoid any risk of electrocution and any deterioration of the equipment.

- Make sure the numerical control is disconnected from the mains power before carrying out any cleaning. Do not use liquids based on alcohol or ammoniac.

- In case of malfunction of the numerical control, call a technician.

- Do not expose the numerical control to direct sun rays or any other heat source.

- Do not place the numerical control in the neighbourhood of magnetic equipment such as transformers, motors or devices which generate interference (welding machines, etc.)

- Replace fan filters at regular intervals so as to avoid overheating.

This page has been left blank intentionally.

LICENCE AGREEMENT FOR CYBELEC SOFTWARE

GENERAL COPYRIGHT

The CYBELEC software is protected by Copyright, and all the copying rights are reserved.

The CYBELEC software may only be installed and used in authorized equipments (PC or DNC).

The user manuals are also covered by copyright, and all rights to use and to copy are reserved.

This document may not, in whole or in part, be copied, photocopied, reproduced, translated or reduced without prior consent, in writing, from CYBELEC.

SPECIAL DISKETTE COPYRIGHT

The legal users of this software product are authorized only to copy the contents of the diskette into the memory of the computer to run the program, and to make one backup copy of the original diskette for safety purposes in case of loss of the original program.

Unauthorized copying, duplicating, selling or otherwise distributing this product is a violation of the law.

SPECIAL EPROM COPYRIGHT

The CYBELEC DNC and CNC units in which the original software made by CYBELEC has been replaced by a copy not made by CYBELEC, and without written authorization of CYBELEC, will immediately lose their warranty.

WARRANTY

CYBELEC does not warrant that its software products will function properly in every computer and programming environment.

The limitations of use of a software product and its technical specifications are decided by CYBELEC only; CYBELEC solely is entitled to decide upon conformity and performance of a given software.

The CYBELEC software does not compensate for incompatibilities in operating system revisions or versions.

Running the CYBELEC software under various revisions or versions, or switching between different versions or revisions may result in loss or alteration of data.

SOFTWARE UPDATE SERVICE

Purchase of the CYBELEC software entitles the user, during one year, to delivery of software updates of the "correction" type.

During the use of a revised or corrected version of the software it may occur that data (program, parameters, etc.) is lost, or that the equipment or its connections need to be modified; these effects are not always foreseeable and do not engage CYBELEC's responsibility.

TERMINATION

This agreement shall automatically terminate upon any act of bankruptcy by or against licensee, upon any assignment for the benefit of creditors of the licensee, upon any attachment execution of judgement or process against licensee or its assets that substantially inhibits its ability to do business, or upon dissolution of licensee.

CYBELEC has the right to terminate this agreement immediately, should the licensee violate the aforementioned conditions.

Within 30 days of termination of this agreement for any reason, licensee shall at his option, either:

• return to CYBELEC or authorized dealer all existent copies of such software and related materials, or

• furnish to CYBELEC evidence satisfactory that the original and all copies of the software, in whole and in any form, have been destroyed.

LIMITATION OF LIABILITY

The foregoing warranty is instead of all other warranties, expressed or implied.

Licensee further agrees that CYBELEC shall not be liable for any lost profits, lost savings, loss of use, or other incidental or consequential damages arising from the use or inability to use the software, or for any claim or demand against licensee by any other party.

In no event shall CYBELEC be liable for consequential damages, even if CYBELEC has been advised of the possibility of such damages.

CYBELEC does not warrant that the functions contained in the software will meet the licensee's requirement or that the operation of the software will be totally error free.

Should the software prove defective, the licensee (and not CYBELEC or an authorized dealer or representative) will assume the entire cost of all necessary service, repair or correction.

CYBELEC warrants the diskettes, EPROMS or other magnetic support or cassettes on which the programs are supplied to be free of defects in material and workmanship under normal use for a period of 90 days from the date of shipment to the licensee as evidenced by a copy of the packing slip.

LIMITATION OF REMEDIES

CYBELEC's entire liability and the licensee's exclusive remedy shall be as follows:

The replacement of any diskettes or EPROMS or magnetic support media or cassettes not meeting CYBELEC's limited warranty and which materials are returned to CYBELEC or an authorized CYBELEC representative with a copy of the packing slip, or

If CYBELEC or its representative is unable to deliver replacement diskettes, magnetic support media, EPROM or cassettes which are free of defects in materials or workmanship, the licensee may terminate this agreement under the terms and conditions herein mentioned, and the purchaser's money will be refunded.

GENERAL

The licensee acknowledges that he has read this agreement, understands it and agrees to be bound by its terms and conditions.

The licensee agrees to hold CYBELEC harmless on all liability associated with licensee's breach of this agreement including, but not limited to, all reasonable attorney's fees and court costs, if any.

This licence agreement shall be governed by Swiss law; place of jurisdiction is Lausanne, Switzerland.

MAINTENANCE

CYBELEC will provide one year of software maintenance. The extent of maintenance, and response time for furnishing same, shall be at the sole discretion of CYBELEC. Maintenance shall normally include correction of errors in code, correction of errors in supporting documentation, update versions of the covered software which may be released by CYBELEC during the maintenance period.

In no event shall CYBELEC be obliged to provide technical support in attempting to resolve problems or difficulties resulting from licensee's modification of the licensed software; any such modification by licensee is entirely at licensee's own risk.

This page has been left blank intentionally.

CONTENTS

SAFETY, COPYRIGHT & LICENSE AGREEMENT

Please consult the safety instructions, copyright and license agreement on the first pages of the manual.

ABOUT THIS MANUAL

This manual is designed to help the user to familiarize with the numerical control DNC 600S and to use it with a maximum of efficiency.

To reach a maximum of comfort and productivity of the numerical control, it is recommended to read attentively the whole manual.

A table of contents and a well organized index enable you to find rapidly the searched subject.

Note: In this User Guide it is assumed that the DNC has an operational configuration (i.e. machine parameters and tools are programmed).

This manual is evolutive. You, the user, can help us to give you better assistance. If you have any comments on this document, please write us to:

> **CYBELEC S.A**.
> Dpt Communication
> Rue des Uttins 27
>
> CH-1401 **Yverdon-les-Bains**
>
> Fax ++ 41 24 447 02 01
>
> **E-mail: info@cybelec.ch**

This page has been left blank intentionally.

HOW TO…

USE EASY BEND (VERY BASIC BENDS)

☞ Go to page 6,.*EASY BEND Page*.

PROGRAM A SIMPLE PRODUCT

☞ Go to page 12, *Programming with the Sequence Page*.

PROGRAM A PRODUCT USING THE GRAPHIC DISPLAY

☞ Go to page 38, *Appendix F: Programming with L-Alpha*.

TIPS

Press this key to navigate through the *main* fields of the page from the top down.

Press this key to navigate through *all* fields of the page from the bottom up. Keeping this key pressed will navigate from field to field using the auto-repeat feature of the key (3X per second).

The navigation feature of both these keys can be activated or de-activated in the **USER PREFERENCES** page from the main menu.

Press this key combination to go to the top left field.
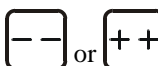
Pressing on either of these keys while the cursor is on the **P** field will toggle the display between a white and a black background.

Pressing on either of these keys while the cursor is on the **P** field will adjust the contrast.

EASY BEND PAGE

The **EASY BEND** page is accessible by hitting the ⬜ key twice.·

```
 Easy Bend
 Th:  2.00   σ  45.00 Kg/mm2  Ac_Al_In_
 p/d  1/1     CR __/ri    2.78
        --MEM--       --POS--        --COR--
 ◁α      90.0°                ◁α mes __._°

 Y1    293.11      350.92
 Y2    293.11      351.05
 L     136.0
 X1    133.9        51.8
 Lp  5000        Fp  78
 Rx   ___       Start Ext _
 Vy   ↓__% ↑__%   ΔTp   _._s
                 PCV   ___
Material thickness
```

Online Help

This page makes it possible to quickly programme a bend, and/or briefly stop production to perform a single bend without modifying or losing data in the production programme.

PROGRAMMING A BEND

- Enter the thickness and the tools if they differ from the product in the course of production.
  To speed up programming, the **p/m** fields as well as the material are taken over from sequence 1 of the product currently in hand. The modification of these data does not affect the programme that is running.

- Enter the angle of the bend in the ◁α field

- Enter the height **L** of the wing (external measurement) and the position of the X gauge will be calculated automatically.
  or
  directly enter the distance at which the **X** gauge is to be positioned.

- Enter the length of the bend.
  The system automatically calculates the pressure and bending.

- Possibly also enter,
    o how long the pressure is to be maintained (if not programmed, the default is = 0.5 s),
    o the retraction of the gauge,
    o the reduction in bending speed,
    o slow lifting, etc.

- Press the START button or the pedal and the axes will position to the desired values. The numerical command will automatically go into semi-automatic mode.
  Actuate the pedal to bend.
  When the beam returns, the DNC switches to programming mode.
  You can now enter new data or actuate the pedal for a new bend.

- Hit the $\boxed{\text{N}}$ key to return to production mode.

FIELD OVERVIEW

See *Field Overview*, page 9.

CREATING A PRODUCT

USING THE SEQUENCE PAGE

This page displays all the bending informations for the current sequence. That is the requested position of the Y and X (R, etc.) axes, the pressure, the gauge retraction, the crowning, the dwell time, etc.

Important   This page displays the result of the calculations made when programming a product in the L-alpha page with definition of the bending order.
The operator can modify these values if necessary (especially the retraction, the dwell time, the top dead center).
However he must avoid to directly modify the Y and X values calculated by the DNC.

Hint   An operator can directly program a product in this page, without "passing" through the L-alpha page and definition of the bending order.
This simplifies the manipulations a great deal, since all the informations required for the bending are on this page.

To reach the sequence page, press the [N] key.

Pressing once will display the page in large characters, pressing twice [N] will display all dimensions and functions of the current sequence in small characters.

Pressing the key 3 times [N] will display the main sequence data in large characters, with a graphic of the part underneath.

This page is displayed during the powering-on of the DNC.

To change from one sequence to another, press the [➡□] or [□⬅] key.

In order to add a new sequence with the same content as the last sequence at the end of the program, press [➡□], then **1** when the warning message displays.

In order to insert after the current sequence a sequence with the same content as the current one, press [⤵] when the cursor is on the **N** field.

The [⬇] key is used to move the cursor from one significant field to another.

The [⬆] key is used to move the cursor across all the fields. This function can be deactivated in the user preferences.

If a product is removed, the data contained in the tool fields, the thickness field and the material information fields continue to be displayed. This saves the operator from having to reprogramme unchanged data.

```
 P   1  N  1   CY __
 Th:  2.00     σ  45.00   Kg/mm²
 p/d  1/ 1     CR __/ri    2.78
      --MEM--        --POS--      --COR--
 ◁α      90.0°
 Y1      293.11        350.92     ____.__
 Y2      293.11        351.05     ____.__
 X 1     133.91         51.85     ____.__
 Q       ____          ____
 Lmat    5000      TON    78
 Rx _    ____      Ext start _
 Vy  ↓__%  ↑__%   ΔT    _._s
 TDC     ____     SP    ___
 F1:___  F2:___  F3:___  F4:__  F5:__  Cr: 41
```

FIELD OVERVIEW

P	Number of the product in the work memory.
P+	Number of the following product when there is a sequence of programmes.
N	Bending order (sequence) number to be made and the total number of bends for the product.
CY	Number of sequence repetitions. (programmed 0, the sequence is jumped)
Th	Material thickness.
σ	The Greek symbol "Sigma" indicates the tensile and rupture strength. This field, as well as those showing the type of material, can be hidden (see the **preferences** page in the machine parameters). If this field is hidden, the default value is **steel 45 kg/mm²**, but other values can be defined in the *ad hoc* field in the machine parameters.
p/d	Number of the punch/die pair associated with the bend to be made. In the **P** or **D** field press ⟦➡⟧ to display the list of tools.

Select the tool in the list then press ![key icon] again to return.

CR/ri	Number of bends requested when working with step bending (min. 4), and internal bending radius.
--MEM--	This column contains either the values calculated by the system when simulating, or the values programmed by the operator.
--POS--	This column indicates the real axes' positions.
--COR--	This column contains all the sequence and product corrections introduced on the CORRECTION page.

The ◁α, **Y1**, **Y2**, etc. fields are dependant on the axes programmed in the numerical control.

L	Height of the wing after bending. The DNC will automatically calculate the position of the back gauge of axis X.

L

Q	Quantity of products to be produced. The **Q** field is displayed only if the parameter in the **PREFERENCES** page is activated.
Lmat	Bending length.
TON	Bending force.
Rx	Back gauge retraction. The operator indicates in mm the retraction distance. If nothing is programmed, the beam doesn't stop at the PP. No retraction. If value 0 is introduced, the beam stops at the PP, then executes immediately the bend without carrying out a retraction.

The field between the Rx and the retraction value serves to define the retraction mode.
_ = the beam stops at the PP and waits till the retraction is made, in order to carry out the bend.
1 = the beam stops at the PP, then continues immediately, at the same time as the retraction is carrying out.

Ext start	If nothing is programmed in this field, the axes start automatically from the BDC, from the PP or from the TDC, depending on the choice made in the machine parameters.

If **1** is programmed, the start will be made as a function of the configuration made by the

constructor.
- In most of the configurations, it will be sufficient to give a down command. At this command, the axes take position, then a second down command has to be given, in order to move the beam.
- In other configurations, the provided start button must be pressed.
- When the message TOL ZONE appears, that means that a down command has been given and that the axes are not positionned in the current sequence. Give a start with the provided button or on the front panel of the DNC.

Vy ↓	Beam bending speed. Programmed at 0, the speed is 1 mm/s. Programmed at 9, the speed is 10 mm/s. (If the machine is capable of reaching this speed)
Vy ↑	Rising speed from BDC to PP. Programmable from 0 (slow) to 9 (fast). Allows to vary the speed at which the beam rises to the PP. (Function depending on the machine's hydraulics).
ΔT	Dwell time.
TDC	Distance from top dead center.
SP	Distance from the beam speed changing point. (Switch Point).
F1: - F5:, Cr	Value of the auxiliary functions F1 to F5 and crowning.

These fields can also be re-programmed by the user.
However, certain fields are related, that is modifying one provokes a modification of the other.

PROGRAMMING WITH THE SEQUENCE PAGE

In this guide, it is considered that the DNC is configured so as to be operational (i.e. machine and tool parameters programmed).

The operator has 2 methods available for programming.

The fastest

Direct programming (or programming in the sequence page)
Direct programming is the faster method for an experienced operator, because all the programming of the product is done on the same page. This makes it possible to programme the positions of the axes directly, and it calculates the depth of bending according to the angle entered.

With calculation of the flat pattern

The L-Alpha method.
The L-alpha method makes it possible to enter a profile to bend with the external dimensions and angle of each face. The DNC will calculate the developed length:
After having programmed the sequence of bends, the DNC will calculate the position of the gauges.
Explanation of this method in *Appendix F: Programming with L-Alpha* on page 38.

DIRECT PROGRAMMING

Programming Example

The following product is to be realized:

20.0

50.0

200.0

60.0
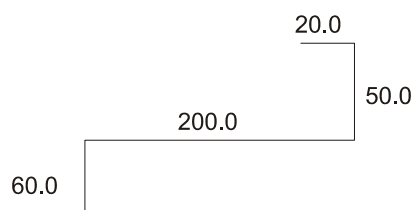
Th: 2.0 mm
Lmat: 1000 mm

The bending order is the following:

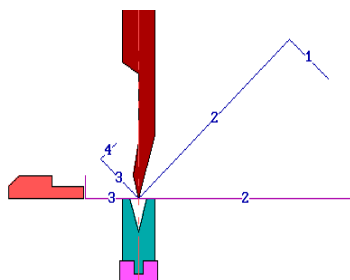- Bend 1 on FACE 1 and LEG 0
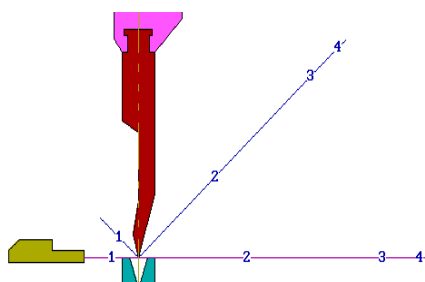
- Bend 2 on FACE 3 and LEG 4
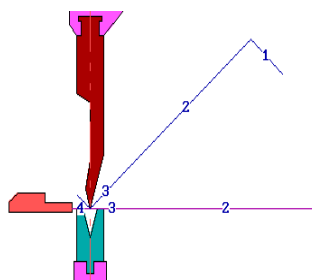
- Bend 3 on FACE 2 and LEG 3

Clear the memory
- Call the sequence page with the [N] key.

Clear the work memory.
In the event of the removal of a product, the data contained in the Tools fields, the Thickness field and the Material Information fields remain (if displayed). This avoids the operator having to reprogramme unchanged data.

Place the cursor in the **N** field (reminder: [++] + [⇧] to position the cursor on the top of the page).

```
P    125   N  99    CY __
 Th: __.__         σ   __.__   Kg/mm²
 p/d  _/_     CR __/ri    __.__
       --MEM--         --POS--      --COR--
 ◁α      __.__°
 Y1      ___.__         350.92      ___.__
```

- Introduce **99** and press the ⌨️ key.

1st sequence

- Introduce the thickness, the sigma, the tools as below:

- To select a tool, enter the tool number or press ➡️ when the cursor is in the **P** or **M** field to display the tool list.

  Select the tool in the list and press ➡️ again to go back to the sequence page.

```
P    0   N  1    CY __
  Th: 2.00        σ  37.00   Kg/mm²
  p/d 1/1     CR __/ri    __.__
```

- Program **90** in the ◁α field.
  By leaving the field, the system calculates the Y1 and Y2 axes' values.

- Introduce **58.0** in the **X** field (stop position for this first bend).

- Introduce the bending length **1000**.
  The system calculates the pressure and the crowning automatically.

- If necessary, introduce the dwell time (by default non-programmed = 0.5 s), the gauge retraction, the bending speed reduction, the slow raising, etc.

- Press ➡️ and then 1, in order to copy this first sequence in sequence 2.
  The message **COPIED** is displayed, the **N** field passes on **2**.
  The LED of the key remains lighted, indicating that you are on the last sequence.

2nd sequence

The following screen is displayed:

Note: The values **Y1**, **Y2**, **Fb** and **Cr** will vary, for they are calculated as a function of the programmed tools. The values **POS Y1**, **Y2** and **X** will also be different, for they display the real position of the axes.

```
P   0  N  2   CY __
Th:  2.00      σ  37.00  Kg/mm²
p/d  1/ 1     CR __/ri  __.__


      --MEM--        --POS--      --COR--
 ◁α     90.0°
Y1     229.90        350.92       ____.__
Y2     229.90        351.05       ____.__
 L      ___.__
 X       58.00        51.85       ____.__

Lmat   1000        TON   19
Rx      ___        Ext start _
Vy    ↓_   ↑_      ΔT   _._s
TDC      ___        SP    ___
F1:___  F2:___  F3:___  F4:__  F5:__  Cr: 41
```

Modify the values which are different for this sequence, in this case enter 20.0 in the **L** field or **18** in the **X** field.

```
P   0  N  2   CY __
Th:  2.00      σ  37.00  Kg/mm²
p/d  1/ 1     CR __/ri   __.__


      --MEM--        --POS--      --COR--
 ◁α     90.0°
Y1     229.90        350.92       ____.__
Y2     229.90        351.05       ____.__
 L       20.0
 X       18.00        51.85       ____.__

Lmat   1000        TON   19
Rx      ___        Ext start _
Vy    ↓_   ↑_      ΔT   _._s
TDC      ___        SP    ___
F1:___  F2:___  F3:___  F4:__  F5:__  Cr: 41
```

• Press the sequence forwards key [➡🔲], in order to copy this second sequence in sequence 3.
The message **COPIED** is displayed, the **N** field passes on **3**.
The LED of the key remains lighted, indicating that you are on the last sequence.

The following screen is displayed:

```
P    0   N  3    CY __
Th:  2.00        σ  37.00   Kg/mm²
p/d  1/ 1    CR __/ri    __.__


       --MEM--            --POS--        --COR--
◁α      90.0°
Y1      229.90           350.92        ____.__
Y2      229.90           351.05        ____.__
 L       20.00
 X       18.00            51.85        ____.__
Lmat   1000         TON    19
Rx       ___        Ext start _
Vy   ↓_   ↑_        ΔT    _._s
TDC      ___        SP    ___
F1:___  F2:___  F3:___  F4:__  F5:__  Cr: 41
```

- Modify the values which are different for this sequence, in this case enter 50.0 in the **L** field (**X** is automatically calculated) or the value **48** in the **X** field.

The programming of the product is terminated.

If you wish to save:

- Place the cursor on the **P** field.

  (**++** and **⇧** to position the cursor on the top of the page.)

- Introduce the number you wish to give to this product.

- Press **⇲** (if the system replies **EXISTS**, choose another number for saving this product).

  The product will be saved in the internal memory with the number which you have given it.
  It will, however, remain present in the work memory.

- Pass to semi-automatic mode **⮐**.

- Call the sequence 1.

- Execute the first bend.

CORRECTIONS

It can happen that the angle bent is slightly greater or less than the angle programmed.

The operator has then the possibility to introduce the real value of the angle bent. The DNC will correct the Y1 and Y2 axis values accordingly.

The corrections can be entered into two different pages:
- Page corrections (see explanation on page 71).
- Sequence page.

```
 P   0   P+__   N 1 / 4   CY __
 Th:  2.00      σ  37.00  Kg/mm2
 p/d  1/1       CR __/ri    __.__

        --MEM--         --POS--       --COR--
  ⊲α     90.0°           ⊲α  mes   91.20
 Y1     229.60          350.92       ____.__
 Y2     229.60          351.05       ____.__
 L       60.00
 X       58.00           51.85       ____.__
 Lp     1000        Fp   19   TON
 Rx      ___        Start ext _
 Vy   ↓_  ↑_        ΔTp  _._s
 PMH     ___        PCV   ___
 F1:___  F2:___  F3:___  F4:__  F5:__  Bo: 41
```

The following working method is recommended:

- Programme or call the product.

- Switch to semi-automatic mode.

- Execute first bend.

- Measure the result.

- Switch to programming mode.

    Enter the measured angle into the **COR** field (see above). The DNC automatically calculates the corrections to be made to Y1 and Y2.

- If necessary, make the corrections to the gauge axes.

- Bend a second product on the first bend to check the corrections.

- Move to the following sequence and then proceed in the same way for all subsequent sequences.

- When all the bends are correct, switch to automatic mode and proceed normally.

The correction entered above is applied only to the current sequence. Every other sequence must be corrected individually.

Product-by-product correction is possible in the **CORRECTION** page (see explanation on page 71).

PRODUCT MANAGEMENT

This chapter indicates how to manage the products (programs) stored in the numerical control.

PRODUCT LIST

- Press the ⌊L⌋ key.

```
ACTUAL PIECE                      NEW_P
CODE  623-42.15
SEQUENCES PROGRAMMED       42
          AVAILABLE       306


     -LIST OF PRODUCTS IN MEMORY-
  1    2  11 222 997 ___  ___  ___  ___  ___
___  ___  ___  ___  ___  ___  ___  ___  ___  ___
___  ___  ___  ___  ___  ___  ___  ___  ___  ___
___  ___  ___  ___  ___  ___  ___  ___  ___  ___
```

The upper part of the screen displays the number of the product currently in the work memory as well as the number of the drawing of that product.

Note:

The drawing number is optional. It may be activated in the User Preference page.

The DNC also displays the number of sequences programmed, as well as the number of sequences still available in the internal memory.
Then are displayed, in ascending order, the numbers of all the products stored in the internal memory.

The **NEW_P** field enables the operator to create a new product directly, starting from this page.

- Position the cursor on **NEW_P** and press the ➡️◻ key.

SEARCHING FOR A PRODUCT BY CRITERIA

- Press the [L] key again.

```
ACTUAL PIECE  28      New_P
          -- CRITERIA --
CODE
p/d      __/__
Th       __.__
Lmat  _____
DevL   ____.__
     -- LIST OF PRODUCTS IN MEMORY--
   P              CODE        p/d   Th
  ____                       __/__ ___.__
  ____                       __/__ ___.__
  ____                       __/__ ___.__
  ____                       __/__ ___.__
```

This page allows to display a list of searched products according to one or more criteria like the code number, the thickness, the bending length and/or the unfolded length.

Under the heading **--CRITERIA--** are displayed the different criteria according to which it is possible to search for a product.
These criteria can be combined, that is it is possible, for example, to ask for the list of all the products stored using a specific tool pair **and** having a particular material thickness.

The products meeting the defined selection criteria are displayed on the second half of the screen.

To make a search by criteria, proceed as follows:

- Introduce one or more selection criteria in the fields provided.

- Press [➡️▯].

If the list contains 7 or more products, the following page or pages can be displayed by pressing [➡️▯].

CALLING A PRODUCT

This operation searchs a product in the internal memory and place it into the work memory.

Calling a product is possible from all pages on which the cursor can be placed on the **P** or **CODE** field.

Caution: The called product "erases" the product already in the work memory. Pay attention to save, if necessary, the current product before calling another product.

- Introduce the product number in the **P** field or the code number in the **CODE** field.

- Leave the cursor on the field which has been programmed just now.

- Press ➡️◻️.
  The product requested is then copied into the work memory (the original of this product is, of course, conserved in the internal memory).

SAVING A PRODUCT

This operation is used to permanently save a product located in the work memory.

Saving a product is possible from all pages on which the cursor can be placed on the **P** field.

It should be noted that after saving the product remains present in the work memory.

To save a product proceed as follows:

- If you wish to give a drawing number or name to the product, fill in the **CODE** field. This operation is optional.

- Enter the number of the product in the **P** field.

- Leave the cursor in the **P** field.

- Press the ⬇️ key.
  If the system displays the message **EXISTS**, choose another number.

If you wish to save a product under a number which already exists (e.g. after modifying a product), the product bearing that number must first be deleted in the internal memory.

NB: Caution, the N° 998 and 999 are reserved for special functions.

- 998 for the temporary storage of the work memory contents during PC <-> DNC transfer.

- 999 for **total** deletion of the internal memory (see page 21).

DELETING A PRODUCT

- Call the list of products page.

- Place the cursor on the **P** field and introduce the number of the product to be deleted.

- Press .

It should be noted that this action has no effect on the internal memory.

DELETING ALL PRODUCTS

Caution

This operation deletes the totality of the products stored in the internal memory without the possibility to cancel this command.

- Call the list of products page.

- Introduce the value **999** in the **P** field.

- Press .

APPENDIX A: CONVENTIONS

As a general rule, in this manual we will not repeat how to validate a field, select a tool, call a page or any other basic manipulations.
These informations are described at the beginning of this manual.

TYPOGRAPHICAL CONVENTIONS

Arial bold	Quotations of text as seen on the screen.
Arial bold italic	Used to indicate the name of a DNC input or output.
Italic	Reference to a written element, a paragraph or a manual. For example: See ///ADD CROSS REFERENCE.

ABBREVIATIONS / GLOSSARY

Explications of the abbreviations which are not visible fields in the pages of the numerical control.

TDC	Top dead centre.
SWP	Switch point of speed, i.e. the change from approach speed in bending speed in the descent phase.
PSS	Safety point. This point is calculated in function of the tools height and the material thickness.
PP	Pinch point (see page 60).
BDC	Bottom dead centre
LED	Light Emitting Diode. Small red light serving as luminous indicator.

This page has been left blank intentionally.

APPENDIX B: USER INTERFACE

USING THE SCREEN

The screen displays the products, tools and machine parameters as well as all other useful information for programming and machine work.

The keys situated on the front of the DNC are used for selecting the screen pages and introducing data.

A cursor indicates where the user can intervene.

On all pages, when first displayed the cursor is located on the programmable field on which it was placed during the last intervention on that page.

The cursor can be moved to the previous or following field by pressing the ⬇ or ⬆ key.

Faster

Regardless of the cursor position on the page, pressing the ⊞ and the ⬆ keys simultaneously moves the cursor to the first programmable field of the page.

USING THE KEYBOARD

The keyboard is divided into six zones:

- Numerical keyboard
- Screen page selection keys
- Working mode zone
- The commands
- The cursor keys
- The manual mode keys

NUMERICAL KEYBOARD

The numerical keys as well as the ⊟ and ⊡ keys are used to introduce numbers or values into the different fields.

SCREEN PAGES SELECTION KEYS

[ ]
Main menu key
A dual-function key
Hitting the key once opens the **MAIN MENU** page.

Hitting the key a second time displays the
///ADD CROSS REFERENCE

[L]
Product list key
Double function key.
Pressing this key once displays the list of products in the DNC memory.
Pressing the key again displays the search for products by criteria page.

[P]
Product key
Triple function key.
Pressing this key once displays the angle length (L-alpha) values.
Pressing a second time displays the bending order with their respective legs.
Pressing a third time displays the bending values which are calculated as a function of the values introduced on the previous page.

[N]
Sequence key
Triple function key.
Pressing the key once displays all the bending values in large characters of the current sequence.
Pressing a second time displays all the values and functions of the current sequence.
Pressing a third time displays the Graphical Bend page.

[C]
Correction key
This key displays the correction page which allows to make corrections to the current sequence, as well as to the entire product.

WORKING MODES

Programming mode
Allows to introduce, modify and read data as well as to store, search for and transfer programs.

Manual mode
Authorizes axis movement and auxiliary functions using the ++ + − −− keys.

Semi-automatic mode
Allows a machine cycle with the current sequence values **without** automatic sequence change.
This mode is used for adjustments when realizing the first product, as well as for products with only one bend.

The sequence change is made using the keys.

NB: The product counter Q does not function in this mode.

Automatic mode
Normal mode for product production.
The DNC automatically changes sequence after each bend.
Changing directly from "programming" mode automatically forces the first sequence of the program.
Changing from "programming " mode to "semi-automatic" mode and then to "automatic" mode, conserves the current sequence for execution.
This allows to take up again a product in production without having to make "empty" cycles.

COMMANDS

Delete key
Allows to delete a sequence or a program when the cursor is placed on the **PRODUCT** or **N** fields.
Only functions in programming mode.

Clear key
This key allows to delete data indicated by the cursor.
Caution: pressing this key twice, on most pages, deletes all the data on the page.

Search key
According to the cursor position allows:
- to search for a product
- to search for a sequence
- to search for a tool
- to search for a screen page
- to start a product feasibility calculation
- to transform L-alpha values into machine values
- to calculate the unfolded length of the sheet
- to execute a transfer
Only functions in programming mode.

Store / insert / teach key
Depending on the cursor position, allows to store the current program in the DNC internal memory, or create (insert) a new sequence in the middle of an existing program, or copy (teach) the value of an axis positioned manually in the current sequence.
This key is only valid in programming (memorization and insertion) mode and in manual (teach) mode.

CURSOR KEYS

Sequence forwards / Page forwards key
This key allows to scroll pages of the same type. Also allows to pass to the next page when a series of information occupies several pages.

Eg.: program sequence
       punch-die pages
       production pages
       parameter pages

In programming mode also allows to create an identical sequence (copy function) to the previous one as long as this is the last sequence of the program.
The incorporated LED indicates whether the sequence is the last of the program.

Page backwards / Sequence backwards key

Inversed function of the ⬛ key.
Allows to scroll backwards through pages of the same type.
Functions in all modes except automatic mode.

Cursor upwards, to the next accessible field. Holding this key down causes an auto-repeat to move the cursor back until the key is released.

Cursor downwards, to next accessible field. Holding this key down causes an auto-repeat to move the cursor forwards until the key is released.

This key combination positions the cursor in the upper part of the current page.

"MANUAL" ZONE

Start key
Commands:

- The displacement of the axis and auxiliary
  functions. During movement the LED lights
  up (except for the beam).

- An indexation at the beam (on option
  depending on the configuration on other
  axes) when this key is first pressed after
  switching on the DNC.

To indicate that an index is being taken the
LED remains constantly alight, until the
indexes are received. Indexes can only be

taken in [▣] or [▣] modes.

Stop key
Depending on the working mode chosen, stops
the axes or auxiliary functions, or stops the
calculation being executed.
When the DNC is in transfer mode (RS232)
the STOP key allows to terminate the transfer.
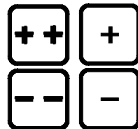
Manual displacement keys of the selected axes.
Allows, in the machine parameters, to
configure the axis type, the inputs/outputs and
to select the display of the N2X axes.

APPENDIX C: MAIN MENU

By pressing the ⬜ key on the top right of the keyboard, the main menu appears.

```
           CHOICE ?  __
  1  NEW PRODUCT
  2  EASY BEND
  3  LIST OF PRODUCTS /CRITERIA
  4  LIST OF PUNCHES
  5  LIST OF DIES
  6  PROGR. PUNCHES / DIES
  7  PRODUCT PUNCHES / DIES
  8  L-ALPHA PRODUCT
  9  BEND SEQUENCE
 10  USER PREFERENCES (KEY 2)
 11  PROGR.AXES DATUM
 12  TRANSFER DNC <-> PC
 13  MACHINE PARAMETERS VIEWER
```

Regardless of which screen page you are on, you can always gain access to

the main menu by the ⬜ key, which allows you to circulate in the different pages.

The option desired is chosen by entering the option number in the **CHOICE**

field on the first line of the screen and pressing the search ⬜ key.

MAIN MENU OPTIONS

1	**NEW PRODUCT**	Create a new product and display the page sequence, ready for entering the data. Access ///ADD CROSS REFERENCE
2	**EASY BEND**	Allows for quickly programming a bend and/or stopping production to quickly do another job (bend) without loss or modification of the program in hand.
3	**LIST OF PRODUCTS /CRITERIA**	This double page displays the list of parts held in DNC memory and allows for selectively seeking parts.
4	**LIST OF PUNCHES**	Displays the list of the punches.
5	**LIST OF DIES**	Displays the list of thedies.
6	**PROGR. PUNCH / DIE**	From this page, it is possible to recall, program or delete a die or a punch.
7	**PRODUCT PUNCHES / DIES**	This page allows to modify, for the product only, the reference of a tool.
8	**L-ALPHA PRODUCT**	Generally called L-alpha page, this page displays and allows the programming of products in "lengths and angles" mode.
9	**BEND SEQUENCE**	Displays the page allowing to program or modify the bending order.
10	**USER PARAMETERS (KEY 2)**	Lets you edit user parameters and materials preferences.
11	**PROGR. AXES DATUM**	Allows to program the axis position counter.
12	**TRANSFER DNC <-> PC**	Allows the global or partial two way transfer of products, tools or machine parameters between the fixed internal memory and a PC.
13	**MACHINE PARAMETERS VIEWER**	Displays all machine parameter pages. Caution: you cannot edit machine parameters using this menu option.

APPENDIX D: WORKING MODES

3 working modes are generally available at machine level.
The functioning is described hereinafter.

Depending on the manufacturers and the safety standards in force in the country, the functioning can be different.

Adjustment mode
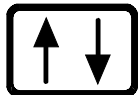
Sensitive mode
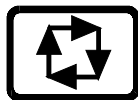
Automatic mode

These modes are independent of the DNC modes and can be combined. Exception: the adjustment mode.

ADJUSTMENT MODE

This mode only functions with the DNC in mode.

If this is not the case, the press refuses to descend.

In this mode the table only functions in **bending speed** with the pressure and speed programmed in the current sequence.

The top dead center of the current sequence is active, it is thus not possible to raise the beam above this point.

On the other hand, the bottom dead center programmed in the sequence is not active. Thus only stopping the descent command, or a mechanical stop (punch in the die), or the lower limit switch will stop the beam in the descent phase.

SENSITIVE MODE

DNC in [icon] Mode

The functioning is identical to adjustment mode, except that the return to the TDC is made in one movement.

DNC in [icon] or [icon] Mode

The press works in the conventional way, that is:

- High speed approach
- Deceleration
- Safety stop (if the mode requires it)
- Bending
- Bottom dead center

It is at the bottom dead center (BDC) that the differences appear.

The DNC remains under pressure at the bottom dead center until the descent command disappears, **and this indifferently to the dwell time at BDC.**

When the descent command is suppressed, the DNC remains at the BDC without pressure or descent command, until reception of a rise command which it executes immediately.

AUTOMATIC MODE

This mode functions in a similar way to sensitive mode, except that the raising of the beam is made automatically once the dwell time programmed in the current sequence has elapsed.

This even if the descent command remains active.

NB: **In all these modes the ascent command has priority. It is executed immediately on its reception.**

APPENDIX E: TOOL PROGRAMMING

The DNC 600S has several pages which allow consultation of the list of existing tools, to display them and / or program new ones.

The DNC 600S memory can hold a maximum of 20 punches and 30 dies.

PUNCH LIST

- From the main menu choose option **LIST OF PUNCHES**.
  The system displays the list of punches existing in the memory, with the number and characteristics of each punch:

If your DNC contains more than 4 punches, you can call the following page

or pages by pressing [⬛].

Die List

From the main menu, choose option **LIST OF DIES**.

The consultation of the list of dies can be done in the same way as explained for the punches.

PROGRAMMING PUNCHES / DIES

TOOL PROGRAMMING OVERVIEW

The programming of tools is done by introducing their values and dimensions on the page as represented below.

- Call the **PROGRAMMING PUNCH / DIE** page (via the main menu).
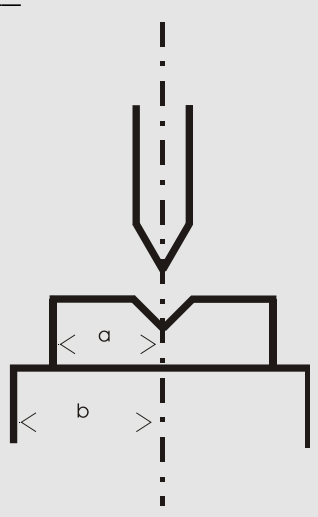
α_p	Punch angle.
Hp	Height of the punch between the lowest part of the beam and the point of the punch.
rp	Punch radius.
TON/M	Maximum force supported by the punch per linear meter.
Vd	V width of the die.

α_d V angle of the die.

hd   Height of the die from the surface of the table.

rd   Die radius.

TON/M   Maximum force supported by the die per linear meter.

REF Y   Sum of the hp and hd.
                        This value is automatically calculated during programming of pairs. See *Erreur ! Résultat incorrect pour une table.*.

SAF X   X Safety distance.
                        (Not programmed = half V opening)

a: and **b:**   Table and die dimensions according to the drawing displayed on the screen.

PROGRAMMING A PUNCH

- On the **PUNCH / DIE PROGRAMMING** page, delete the values on this page by pressing the ⬜ key twice.

- Fill in the different fields.

- If you wish to save this new punch proceed as follows:

  a) Place the cursor on the **PROGR. p/d** field and introduce the number you wish to give to this punch as indicated below.

     Caution: The die N° must not be programmed.

  b) Press ⬜.

  c) If the system displays the message **EXISTS**, choose another number.

```
        PROGR. p/d  1/__
PUNCH
Xp      90°
hp     100.00
rp       1.50
TON/M  100
DIE
Vd      15.00
Xd      90°
hd     100.00
rd       1.50
TON/M  100
REF Y  200.00
SAF X   10.00
a:      20
b:      30
```

Modifying a Punch to Create a New One

On the **PUNCH / DIE PROGRAMMING** page.

- Call the punch which will serve as a basis.
  Place for that the cursor in the **p** field, introduce the desired number and press the ⬜ key.

- Modify the values as necessary.

- Save the new punch by introducing its number in the **p/** field and press the ⬜ key.

MODIFYING A PUNCH

On the **PUNCH / DIE PROGRAMMING** page.

- Call the punch which will serve as a basis.
  Place for that the cursor in the **p** field, introduce the desired number

  and press the [➡️◻] key.

- Modify the values as necessary.

- To be able to save this punch under the same number, the former
  version must first be deleted from the internal memory.

  Place the cursor on the punch number and press the [⬥] key.

- Leave the cursor at the same place and press [⬥].

  Caution: The die N° must not be programmed.

  The former version of your punch has now been replaced by the new
  one which has the same number.

DELETING A PUNCH

- Call the **LIST OF PUNCHES** page (via the main menu).

- Introduce the number of the punch to be deleted in the **p/** field.

- Press [⬥].

PROGRAMMING A DIE

The programming of a die is done in the same way as for a punch, but in the
/m field.

APPENDIX F: PROGRAMMING WITH L-ALPHA

In this manual, it is being assumed that the DNC is configured in such a way to be operational (i.e. the tools and machine parameters have already been programmed).

2 programming methods are accessible by the operator.

The L-alpha method allows to enter a profile to be bended with the external dimensions and angles of each face. The DNC calculates the unfolded length. After having programmed the bending order, the DNC will calculate the stop positions.

Before starting to program a product, make sure that the tools necessary to make the product exist in the DNC memory and write down the numbers of the punches and dies to be used.

To do this proceed as described below:
If you are already familiar with the tools present in the DNC, you may go directly to *Appendix F: Programming with L-Alpha*, page 38).

Looking up the Punch List

To display the tools list in the **P** or **M** field, hit the [➡️▢] key.

Select the tool from the list then press [➡️▢] again to return

If you don't find a punch which corresponds to your needs, you can create one.

In this case refer to chapter *Appendix E: Tool Programming*, page 34.

It is possible, if more than four punches are programmed to see them by pressing the [➡️▣] key.

Looking up the Die List

Proceed in the same way as described above for punches; access from the main menu **LIST OF DIES**.

PROGRAMMING USING THE L-ALPHA PAGE

The **L-ALPHA PRODUCT** page is often called L-alpha page for its
programming method.
On this page, a product is programmed by defining its profile by the length
and the angle of each face.

Call the L-alpha page using the [P] key, or by passing via the main menu
choosing **L-ALPHA PRODUCT**.

This page presents as follows:

```
P   0   P+ ____   N  1    I/mm
 CODE                      CAL_/__.__
 p/d       __/__          St:_ Al:_ SS:_
 Th:       __.__          σ  Kg/mm²   ____.___
 Lmat      _____          Dev L       ____.__
-N-    -L-        - α -   -p/d-     -ri-     CR
 1 ____.__    ____.__°   __/__    ____.__    __
 2 ____.__    ____.__°   __/__    ____.__    __
 3 ____.__    ____.__°   __/__    ____.__    __
 4 ____.__    ____.__°   __/__    ____.__    __
```

Note: On the following pages, you will find a systematic explication of all the
             fields which figure on the L-alpha page.
             If you wish to just follow the procedure, you may pass directly to chapter
             L-alpha Method, page 43.

L-ALPHA PAGE: FIELD OVERVIEW

P   Product number
                            - existent in the work memory or
                            - to be created or
                            - to be searched for

                            The product number must be a number of a
                            maximum of 3 digits between 1 and 997.
                            (998 and 999 are reserved numbers)

                            If the product is created from an empty page,
                            the field contains the number 0.
                            The field keeps this value as long as the
                            product has not been stored.

P+	Number of the next product which will be executed automatically. If this field is left empty, when the last sequence of the product is executed, the program will return to the first sequence of the current product. If it contains a product number, it will be executed immediately at the end of the current product. This allows to follow on several programs one after another. Do not forget to program in the last program the number of the first program in order to close the loop.
N	Number of the current sequence, whose axe's values are displayed on the sequence page.
I/mm	Allows to choose the unit of length. Programmed at 1, the face length values are expressed in Inches. Not programmed or programmed with a value other than 1, the data is displayed in millimeters.
CODE	Code name or number (facultative), allowing to associate supplementary information with the product number in order to facilitate product management and searching. For the introduction of alphanumerical characters, see *Alphanumerical Characters*, page 72.
CAL	Allows automatic compensation of variations in sheet metal thickness. For programming this field, refer to chapter *Calibration*, page 65.
p/d	Punch and die of the current product.
St: **Al**: **S.S.:**	Choice of product material, steel, aluminium or stainless steel. Program 1 in the field next the material used. If no field is programmed, steel is used by default.
Th:	Material thickness.
σ Kg/mm²	(Sigma) Tensile strength.
Lmat	Bending length.
Dev L	Unfolded length of the product calculated by the system according to DIN 6935 standard.

L-alpha Page: Field Overview

Each line (except the last) of the table on the **L-ALPHA PRODUCT** page represents data related to one bend.

```
-N-    -L-      - ∢α -    -p/d-   -ri-   CR TOL
 1 ____.__    ____.__°__/__ ____.__   __ ___
 2 ____.__    ____.__°__/__ ____.__   __ ___
 3 ____.__   __
```

-N-

Automatic numbering of the faces.
A "face" is defined as being that part of the sheet metal situated between two bends or between the edge of the sheet and a bend. The maximum number of faces which can be programmed for a product is 14. If more sequences are needed, use the chaining of product function (see previous page, field **P+**).

-L-

Face length. Designates the distance between two bends or the distance between the edge of the sheet and the first bend.

| 80.00 | 60.00 | 80.00 | 85.00 | 120.00 |

Length

- ∢α -
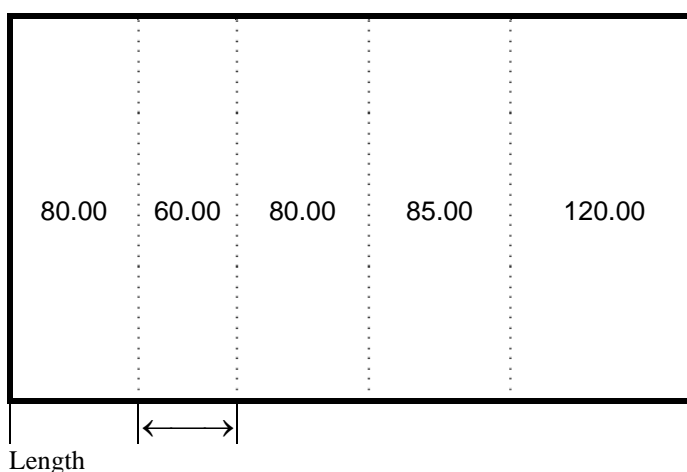
Bending angle.

p/d

Allows, for a given bend, to define a special tooling pair different from the one specified for the product.

-ri-

If the adjacent field CR (step bending) is empty, when the [➡️▯] key is pressed, the **ri** field indicates the internal bending radius calculated taking into account the angle, the material and the tools.

The operator can enter the value of the internal radius which he considers to be correct, then introduce the value 1 in the **CR** field.

During the calculation, the **TOL** (tolerance)

field indicates the distance defined according to the TOL explication below.

When performing step bending, this field indicates the theoretical radius requested by the operator. (See *Ideal Curve*, page 67).

CR   Number of bends requested when working with step bending. (See *Ideal Curve*, page 67). This number must be between 3 and 99. The value 2 makes it impossible to change modes.

> **Note:**   This number must be such that the length of the step bend segments are greater than half the length of the die V opening.

TOL   Tolerance. If the value in the CR field is equal to 1, the tolerance indicates the value according to the illustration below.
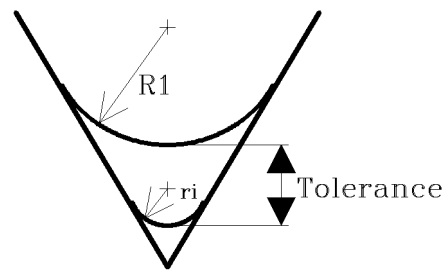
R1 =   Bending radius introduced by the operator.

ri =   Bending radius calculated by the system.

If the value in the CR field lies between 4 and 99, indicates the difference between the chord formed by two bends and the arc of the theoretical circle, that is the difference between the theoretical radius ri and the apex A of the chord.
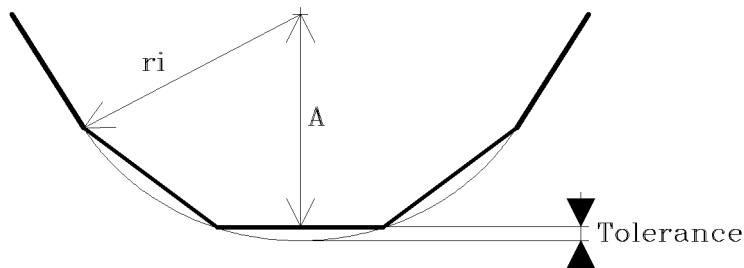
ri =   Theoretical bending radius requested by the operator.

A =   Apex.

For an example of ideal curve (also called step bending), see *Ideal Curve*, page 67.

L-ALPHA METHOD

1. If you have it not already on screen, call the **PRODUCT L-alpha** page.
    The screen displays the data concerning the product in the work memory at present.

2. In order to program a new product, the work memory must be cleared by deleting the product already there.

    If you do not wish to loose that product, you can transfer it to the internal memory by following the instructions below; if the current product is not important or has already been saved, you can pass directly to point 4.

3. To save the product:

    - Place the cursor on the **P** field.

      (➕➕ and ⬆ to position the cursor on the top of the page.)

    - Type the number which you wish to give to this product.

    - Press ⮆ (if the system replies **EXISTS**, choose another number for saving this product).

      The product will be saved in the internal fixed memory with the number which you have given it.
      It will, however, remain present in the work memory.

4. Delete the product from the work memory:

    - Place the cursor on the **N** field.

    - Introduce the value **99**.

    - Press the 🖋 key.

      The work memory is now empty.

      Note that this operation only acts on the work memory and don't destroy the data contained in the internal memory.

5. The product in our example is defined as follows:
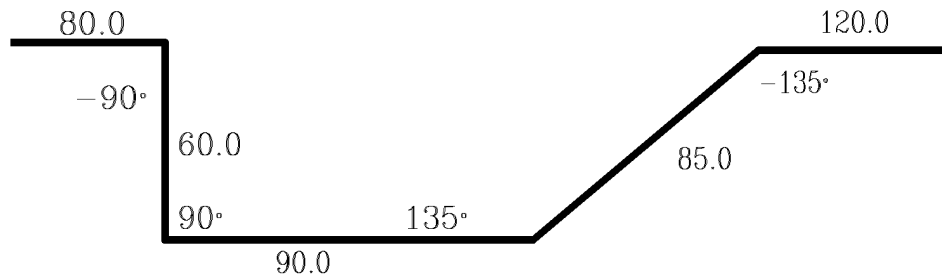
Thickness:	2 mm
Sigma:	45 Kg/mm²
Material:	Steel
Bending length:	250 mm

6. Fill in the fields on the screen according to the model below, taking in to account the following particularities:

   CODE: Optional field.

   p/d   When the cursor is over the **P** or the **M** field, press [➡️▯] to display the tool list. Select the tool you wish to use and press [➡️▯] to go back to the **PRODUCT L-alpha** page**.**

   Then fill in the table columns.

   Your screen should present as follows (but with the punch and die numbers corresponding to your tools):


```
P    0   P+ ___   N   1    I/mm
 CODE EXAMPLE-01                  CAL_/__.__
 p/d        1/ 1         St:1 Al:_  SS:_
 Th:        2.00          σ  Kg/mm²    45.000
 Lmat       250.000      Dev L        ____.__
-N-    -L-       -◁α-    -p/d-     -ri-     CR
 1   80.00    -90.0°   __/__   ____.__    __
 2   60.00     90.0°   __/__   ____.__    __
 3   90.00    135.0°   __/__   ____.__    __
 4   85.00   -135.0°   __/__   ____.__    __
```

7. After introducing the last length, press the [➡️▯] key.
   The system calculates the radius of each bend as well as the unfolded length of the product and displays these values on the screen.

Save this product in the internal memory by proceeding as follows:

- Introduce the number you wish to give the product in the **P** field, in this case the number **1** for this product which will be used later as an example.

- Press the $\boxed{\Rightarrow}$ key.
  If the system replies with the message **EXISTS**, choose another number.

MODIFYING A PRODUCT

If you wish to modify a product programmed in L-alpha, call the product from the **PRODUCT L-alpha** page.

Deleting a Bend

- Place the cursor on the **N** field and enter the number of the face to be deleted.

- Press the $\boxed{\varnothing}$ key; the face containing the bend is deleted.

Adding a Bend

- Place the cursor on the **N** field and enter the number of the face **in front** of which you wish to insert a new face.

- Press $\boxed{\Rightarrow}$. The system inserts a line and copies the data of the current bend. Modify the **LENGTH** and **ANGLE** fields.

- Start the calculation by pressing $\boxed{\rightarrow\square}$.

- Continue in the next chapter. The definition of the bending order has to be reprogrammed.

DEFINING THE BENDING ORDER

Press the [P] key, to display the bending order page (Choice **9**, **BEND SEQUENCE** of the main menu).

```
P    1  N   1          p/d   1/ 1

BEND         1  2  3  4  5  6  7  8  9 10
◁α          __ __ __ __ __ __ __ __ __ __
LEG         __ __ __ __ __ __ __ __ __ __
```

Field Overview

N   Number of the sequence.

FACE   Face number.
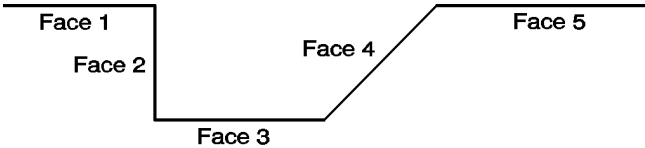
LEG   Number of the face which will press against the stop.
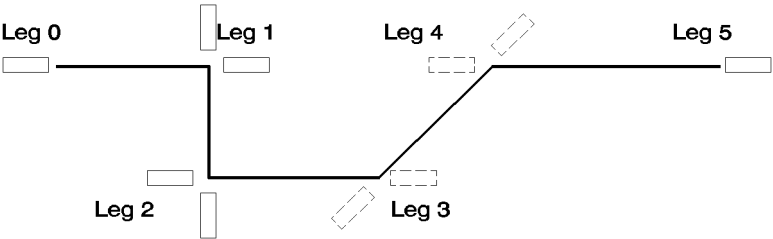
The orientation of the legs is made as a function of the bending order.

CR   Number of bends requested when working with step bending .

p/d   Definition of a tooling pair for a particular
                                           bend (if different from the pair specified in the
                                           p/d field at the top of the page).

Example a

For this example we will use the previously created product.
It was stored under the number 1. (If you cannot find it, create it by
following the instructions of page 43).

- Call the product number 1 by entering its number in the **P** field, and

  then pressing the ⬛➡ key.

The following illustrations show you the desired bending order.
For each sequence, the illustration shows the sheet metal with its bend before
and after bending.

- Sequence N 1: Face 1 and Leg 0
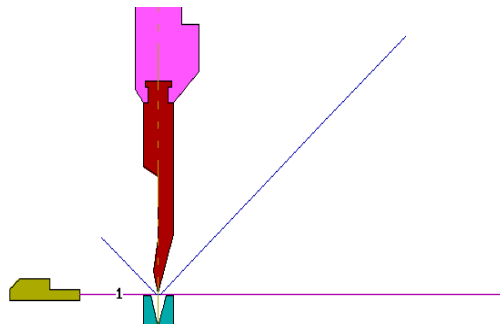
- Sequence N 2: Face 3 and Leg 1
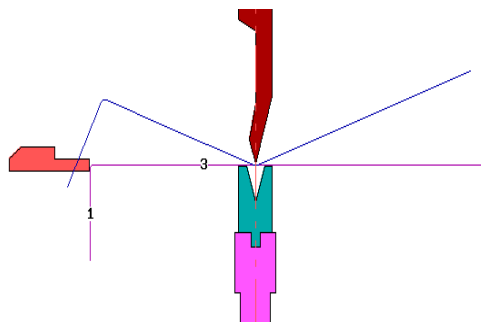
- Sequence N 3: Face 2 and Leg 1
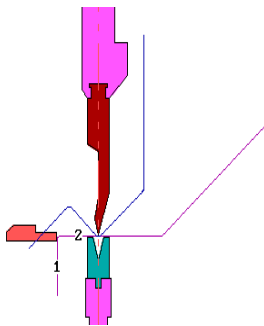
- Sequence N 4: Face 4 and Leg 5
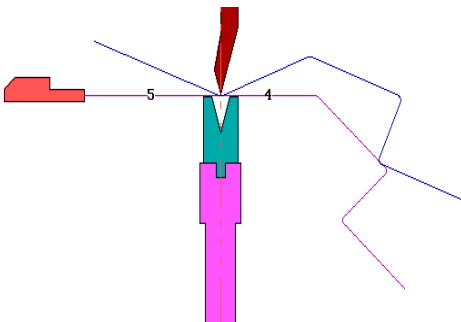
To introduce this bending order:

- Fill in the **FACE** and **LEG** fields as follows:

```
    P    1  N   1           p/d   1/ 1

BEND          1   2   3   4   5   6   7   8   9  10
◁α            1   3   2   4  __  __  __  __  __  __
LEG           0   1   1   5  __  __  __  __  __  __
```


- Press the key.

The system calculates the entire machine program (axes, functions, etc.).

```
 P   1   N   2        p/m   1/ 1    RETURNS

BEND        1   2   3   4   5   6   7   8   9  10
◁ α         1   3   2   4  __  __  __  __  __  __
LEG   0   1   1   5   __  __  __  __  __  __
```
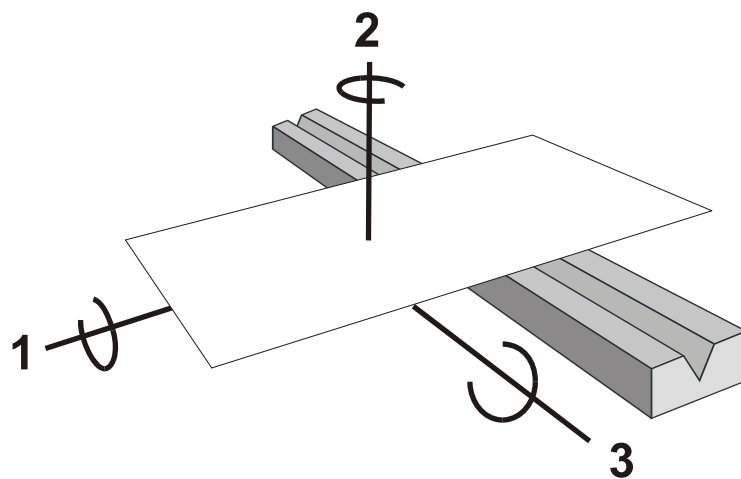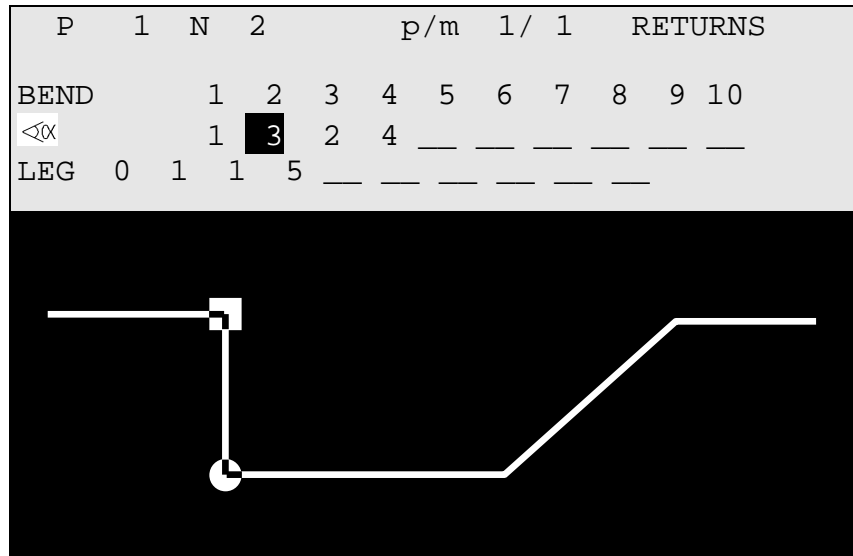

Fig. Sheet metal manipulation

1   **Return**

2   **Swing**

3   **Swivel** (= Swing 180° + return)

- Press the ⃞N key 3 times to view the bend sequence and all information on how to move the metal sheet between bends.


```
N 2 / 5  CR _ _

Pu  1  Di   3  RETURNS
```
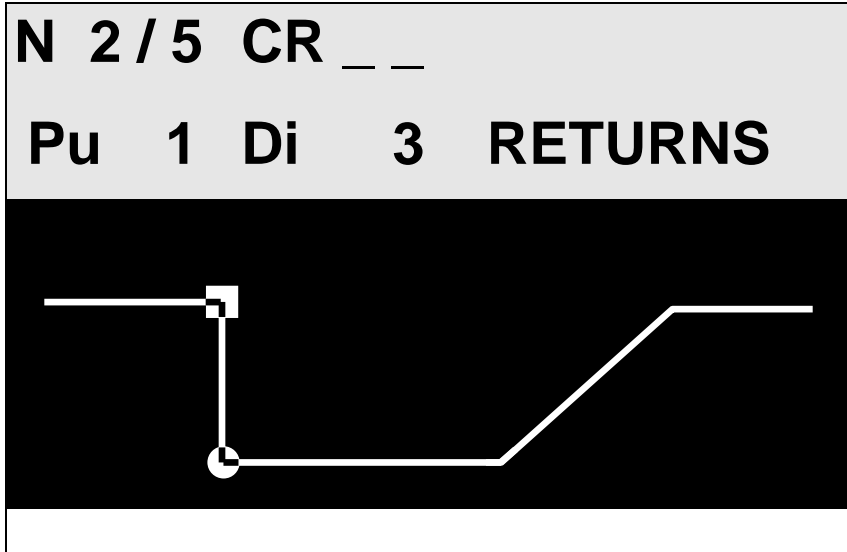
Example b

Here is an other bending order.

For this new bending order, the following illustrations show each sequence before and after bending.

- Sequence N1: Face 4 and Leg 0
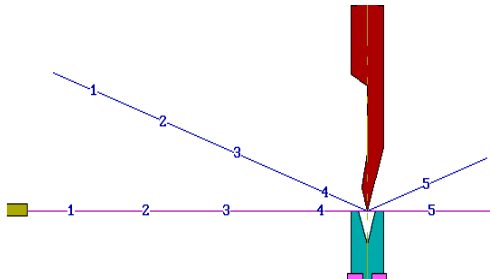
- Sequence N2: Face 1 and Leg 0
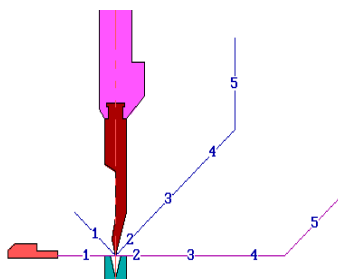
- Sequence N 3: Face 2 and Leg 1
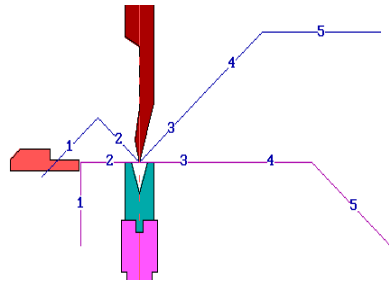
- Sequence N 4: Face 3 and Leg 2
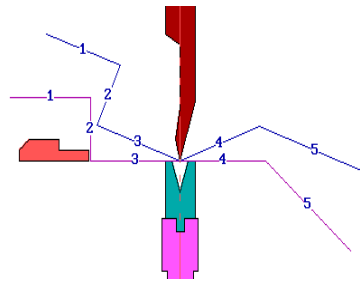
- Call the bending order page by pressing the [P] key twice.

- Fill in the ◁α and **LEG** fields as below.
  You will remark that the **p/d** fields are already programmed since we have already made a calculation before.

```
  P   1  N  2           p/d  1/ 1

BEND          1   2   3   4   5   6   7   8   9  10
◁α            4   1   2   3  __  __  __  __  __  __
LEG           0   0   1   2  __  __  __  __  __  __
```


- Press the [➡️] key.

The system calculates the entire machine program (axes, functions, etc.).

```
   P   1   N   2        p/d  1/ 1

BEND       1   2   3   4   5   6   7   8   9 10
◁α         4   1  [2]  3  __ __ __ __ __ __
LEF        0   0   1   2  __ __ __ __ __ __
```
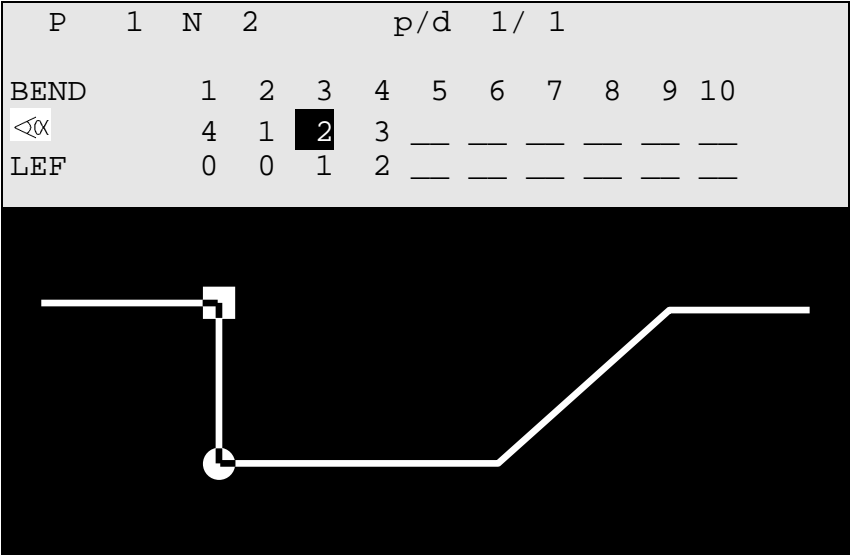

You may navigate between sequences in order to display each one with its

related face and leg by pressing the ⬆ and ⬇ keys (legs=

bend=.)

- Press the [N] key 3 times to view the bend sequence and all information on how to move the metal sheet between bends.

```
N 3 / 5  CR _ _

Pu   1  Di   3   RETURNS
```
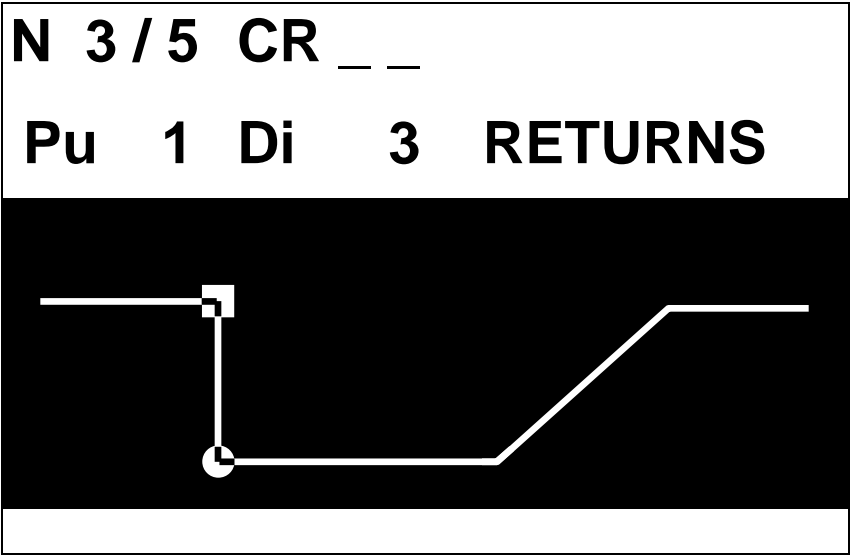

- Go to sequence 1.

- Switch to semi-automatic mode by pressing [⊡].

- Perform the first bend.

- To view the axes' position, go to the Bend Numerical page (large or small characters) by pressing the [N] key.

This page has been left blank intentionally.

APPENDIX G: MISCELLANEAOUS

INITIALIZATION PAGE

This page appears if a calculation is impossible, due to unprogrammed data or an incorrect value.

It can also appear as a result of pollution of one of the memory zones which will also give impossible calculations.

```
         INITIALIZATION DNC PS



   _   Clear variables zone
   _   Clear punches-dies zone
   _   Clear machine parameters zone


           EXECUTION CODE  ____
```

It can be necessary to display this page intentionally in order to clear completely the data of the DNC and to re-start the programming on a "clean" basis.

To display the initialization page:

Switch on the machine by pressing the [++] and [−−] keys.

In case of unexpected apparition of the init page, proceed as follows:

If point n° 1 does not re-establish a normal situation, go on to the next point and so on.

The following operations clear successively all the data in the selected memories.

It is therefore recommended to re-introduce the data.

1)   Press the [ ] key twice.

2)   Enter:

value **1** in the field **Clear variables zone**

and value **817** in the field **EXECUTION CODE**.

Press the [➡️] key.

Press the [ ] key twice.

3)   Enter:

value **1** in the field **Clear variables zone**,

value **1** in the field **Clear punches-dies zone**

and value **817** in the field **EXECUTION CODE**.

Press the [⮕] key.

Press the [ ] key twice.

4)   Enter:

value **1** in the field **Clear variables zone**,

value **1** in the field **Clear punches-dies zone**,

value **1** in the field **Clear machine parameters zone**

and value **817** in the field **EXECUTION CODE**.

Press the [⮕] key.

Press the [ ] key twice.

The clearing of the machine parameters deletes also all the input / output configuration defined by the operator.

It is therefore advisable to re-configure the DNC 600S before the next use.

Caution: If you use an external system for the data re-introducing (CYBACK via RS232) and the problem persists, then the informations re-entered by this system have to be called into question.
The best to do in this case is to delete again the whole contents of the DNC and to re-program the DNC manually via the keyboard (not the RS232 line).

TOOL REFERENCE

The machine has been adjusted according to the following principle:

When, theoretically, the punch's leg face on the beam is in contact with the surface of the table, the value of the Y1 and Y2 counters is 000.00.

In practice this adjustment is made using two calibrated blocks, whose value has been introduced into the counters.

Definition:
The reference value (REF Y) is theoretically the sum of the heights of both tools; punch and die.

Y1 = 000.00
Y2 = 000.00

a or b = punch bearing face
The bearing face of the punch varies according to the type of machine
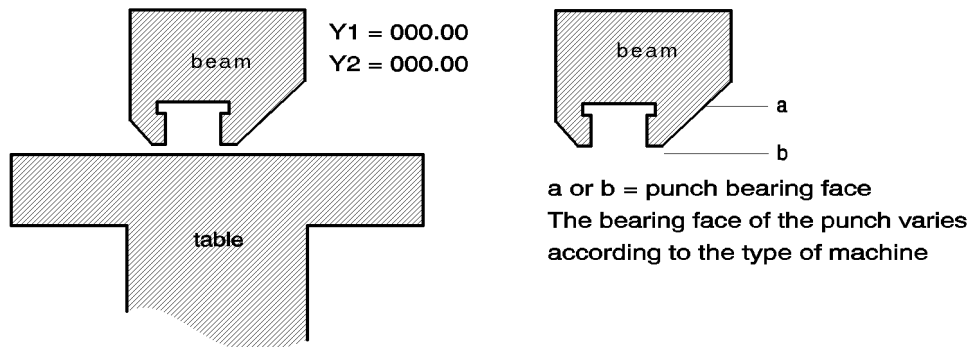
The height of the punch is the distance between the "point" of the punch and its leg face on the beam.

height of punch
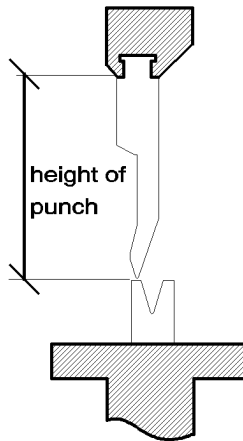
The precision of all bends realized by the machine depends on the reference value **REF Y**.

The numerical control calculates the bending depth using:

- the **REF Y** value,

- the geometrical values of the tools,

- the sheet metal thickness.

If the resulting bends differ greatly from the programmed value, the reference and the values above must be controlled.

ref. =
Y value
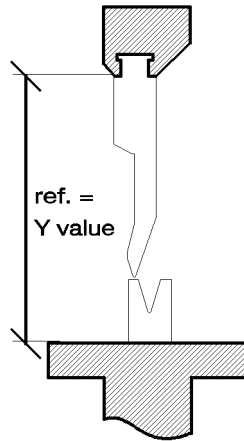
To control the reference, simply, in [hand icon] mode, place the point of the punch on the flat of the die (be careful with the pressure of the active sequence) and read the value indicated by the **POS Y1** and **Y2** counters.

This value must be identical, for both counters and also correspond to the **REF Y** reference value displayed on the **PUNCH / DIE PROGRAMMING** page.

If it is not possible to move the die to obtain a situation as shown above, it is possible using a die with a small opening, to place on the die a thick sheet (which will not bend under the weight of the beam) or blocks of a known precise thickness, (see figure below).

The value of the material introduced between the punch and the die must then be subtracted to obtain the reference value **REF Y**.

ref. =
Y value - sheet th.

min. sheet th. 10mm
ev. more depends of Ve
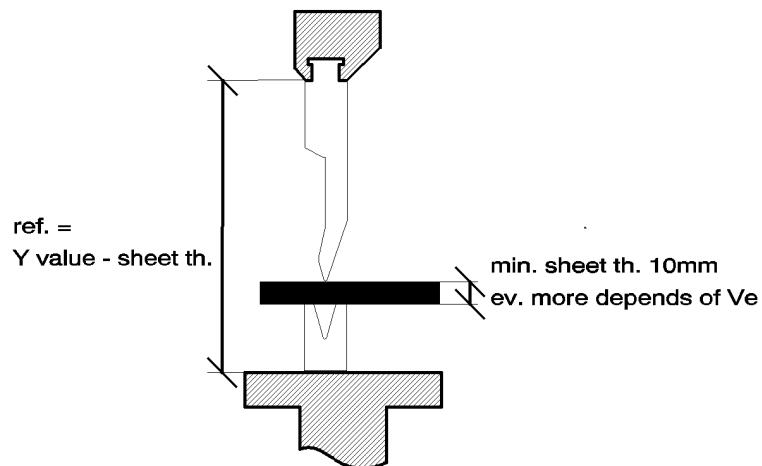
Y1 and Y2 Counters Differ Greatly

If the counters are not identical and present a difference superior to what is normally tolerated (the press in this case furnishes an angle which varies from one side to the other of the machine), a test must be made with another tool in order to determine if the difference comes from the tool itself (tool badly finished presenting a difference in height along its length) or if the press is badly calibrated. In the latter case, you must contact the manufacturer.

Y1 and Y2 Counters Identical, but Different from REF Y

If the difference between the counters Y1 and Y2 is within the acceptable tolerance, but the Y1-Y2 value is greatly different from the **REF Y** value, you must determine if the difference comes from the height of the die or that of the punch.

In this case the tool at fault must be corrected so that the **REF Y** result is identical to the value of the Y1 Y2 counters.

Modifying the Reference for a Given Product

The possibility exists to modify the reference for each tool used for the product. This modification is stored with the product.

It is in some ways a reference modification per product tool. It is thus possible in this case for the same tool pair used for two distinct products, to have a reference modified positively in the product XXX and to have a reference modified negatively for the product YYY.

The principle of measure / control is identical to that which is described above.

To modify the reference in a particular product, you must access the PRODUCT PUNCHES / DIES page via the main menu, choice 7, and program the new reference in the REF Y field which is accessible by the cursor.

All the sequences having the same tooling will be modified.

PINCH POINT

Definition

The pinch point (contact point) is the position of the beam which corresponds to the moment when the point of the punch makes contact with the sheet metal placed in the die.

This point is not programmable, but is calculated by the DNC using the **REF Y** value and the sheet thickness.

The position is determined by the following calculation:

PP = REF Y + Th (sheet thickness).

The PP is used by the DNC cycle to maintain the sheet when an X backgauge retraction is programmed.

As we have seen at the beginning of this chapter, the PP is closely linked to the Y reference and the sheet thickness.

The following cases may occur:

a)   the sheet is not pinched during X backgauge retraction

b)   the sheet is bent too much during X backgauge retraction

To remedy these problems you must:

- verify the sheet thickness and if necessary correct it
- verify the reference and if necessary correct it
- add a PP correction in parameter 37.

A simple way of verifying the position of the PP is to program a sequence with a very large retraction (approx. 400 mm) and make a descent of the beam.

During the execution of the retraction, the beam stops on the sheet, it is thus easy to control if the sheet is pinched sufficiently or too much.

Simply interrupt the descent command before the end of retraction to prevent the beam from bending the sheet. An ascent command frees it immediately for another try.

For the softwares having the "PP correction" parameter, if the reference, the tooling dimensions and material thickness are correct you can modify this parameter in order to correct the PP (adjustment of the material clamping) without interfering with the other points.

GAUGE AXES

The addition of supplementary axes (to X) does not present any difficulties for the operator, the fields being clearly defined on the screen.

The safety factors concerning the supplementary axes such as **R, X1, X2, Z1** and **Z2** are described below.

X-R Safety Factors

When the R axis is a digital axis, an anti-collision safety factor prohibits the positioning of the gauge probes in the die.

A safety zone encircling the die provokes a particular type of positioning when the probes position themselves in this zone.

The safety zone encircling the die is defined by the data introduced in the machine parameters, and are defined by the manufacturer.

The zone where all positioning is prohibited, is defined using the values introduced for each set of tools.
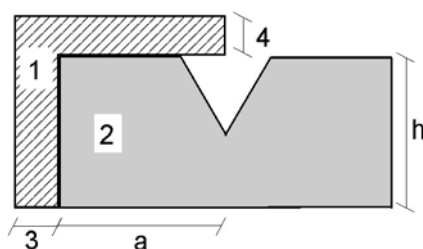
1   Safety zone

2   Prohibited zone

3   Value **Safety die X** parameter 50

4   Value **Safety die R** parameter 50

X-R Safety Positioning Diagram
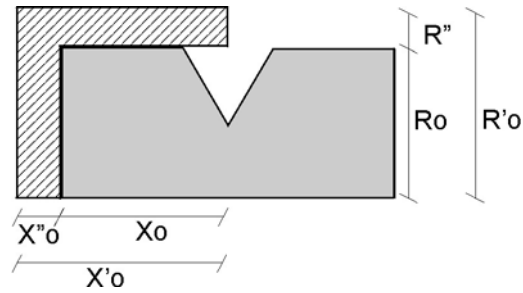
$X''o$ $R''o$ = machine parameters value 50
Xo = "a" die value
Ro = "hm" die value
$X'o$ = Xo + $X''o$
$R'o$ = Ro + $R''o$
Xc = X axis current value
Rc = R axis current value
Xv = X axis target value
Rv = R axis target value

Two Examples of Positioning

These two examples show that it is possible to position in the safety zone.

If the positioning requested is situated in the prohibited zone, the DNC

refuses to go into ⬛ or ⬛ mode and the cursor moves to the X field.

1 Departing position
2 Temporary stopping position
3 Destination position

1 Departing position
2 Temporary stopping position
3 Temporary stopping position
4 Destination position

X1-X2 Safety Factor

The safety factors offered for X1 X2 are identical to the ordinary X axis.

The nearest axis (X1 or X2) to the die is taken into account for the safety factor.

Z1-Z2 Safety Factor

An anti-collision safety factor prevents the passage to and modes, and the interactive message "Z1-Z2" is obtained if the positioning is impossible (see parameter 51).

Normally the origin of the Z axes is on the left of the machine when looking at the front of the machine.

PROGRAMMING THE AXES DATUM

Select option 11 of the main menu, the following page appears:

```
AXES DATUM
  --SET--     --POS--
X _____.__      0.00
Y1 ____.__  ____.__
Y2 ____.__  ____.__
```

- Introduce in the SET column the values for the axes datum.

- Leave the field for validation.
  At this moment, the **POS** field takes the new value.

Dangerous Operation

- This operation can only be executed by an experimented operator who knows the consequences of his modification.

- If you modify the position of an axis in this way, make sure that it corresponds to the reality. After modifying, the electronic, electrical and mechanical limit switches are to verify. Generally the axis in question is displaced in manual mode slow speed up to the negative/positive limit switch.

- It is recommended not to modify Y1 and Y2.
  If this should be the case however, a particular caution has to be taken. In fact, if you modify here the position and an error is committed, the damages caused by this modification can be dangerous and/or harmful for the functioning of the machine.
  Furthermore this modification get lost at each restarting of the machine.

CALIBRATION

The **CAL** (calibration) field on the **L-ALPHA PRODUCT STATUS** page is used for the automatic compensation of variations in sheet metal thickness.

To ensure the good working of this function, it is necessary that the beam can "set down" on the sheet metal with minimum pressure without bending it.

The principle of functioning is as follows:

The beam approaches the sheet at minimum speed and pressure, the resistance of the sheet will stop the beam. This stop allows to measure the thickness of the sheet under the punch, and in the case of a difference with a reference sheet, to make a correction.

It is thus essential that the sheet is rigid enough to stop the beam without flexing, otherwise the measure will be false.

Calibration (procedure)

Calibration is only possible on sequence N° 1.

- Measure a reference sheet with a micrometer.

- Introduce this value in the **Th** field.

- Program the **CAL** field at **3**.

- Verify that the DNC is on the sequence **N1**.

- Change to mode.

- Place the reference sheet in the tools and bend.

- Once the bend is finished, the **CAL** field changes automatically to the value **4**, which confirms that the calibration has been made.

The calibration is only done once for a series of products of the same thickness.

Use

After calibration, it is possible to work in mode. The measurement of the sheet is made for each product on sequence **N1**.

To leave this work mode, clear the **CAL** field using the key.

If any modifications are made to the tools, it will be necessary to repeat the calibration.

IDEAL CURVE

The ideal curve functionality (also called *step bending* or *big radius*) allows to make a bend with a large internal radius.

Ideal curve can be programmed on one or the other of these pages.
Ideal curve is defined by programming from **4** to **98** (programming 0, 2 or 3 will give an error).

Note: For the result to be coherent, the number of bends to realize the angle must be such that the length of each segment is greater than half the length of the V opening of the die.
If this is not the case a message informs the operator and the result will be an angle too open.

In ideal curve mode, the software calculates different bends for the first, second, second last and last bend according to the drawing below.
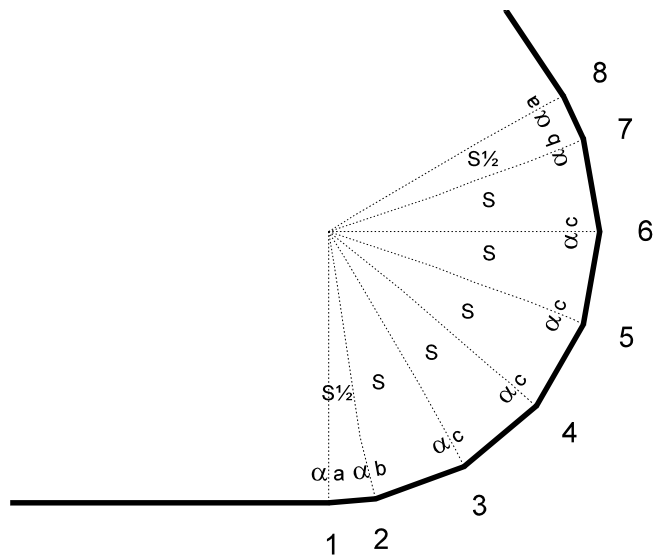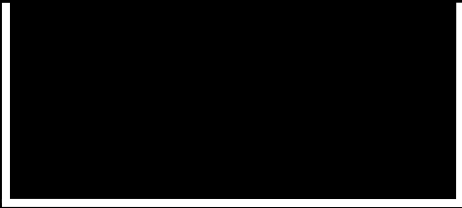
Procedure to Perform an Ideal Curve

In this example, an ideal curve with an internal radius of 50 mm is to be made.

- Call the **L-ALPHA PRODUCT** page and fill in the fields in order to make the bend illustrated below.


```
P    0   P+ ___    N  1    I/mm
CODE                        CAL_/__.__
p/d        1/ 1        St:1 Al:_ SS:_
Th:        2.00        σ  Kg/mm²   45.000
Lmat       2000        Dev L      ___.__
-N-    -L-       - ◁α - -p/d-    -ri-    CR
 1  100.00      90.0°__/__    50 00   15 ___
 2  200.00      90.0°__/__ ____.__   __ ___
 3  100.00     ____._°__/__ ____.__   __ ___
 4  ____.__    ____._°__/__ ____.__   __ ___
 5  ____.__    ____._°__/__ ____.__   __ ___
    ____.__    ____._°__/__ ____.__   __ ___
```
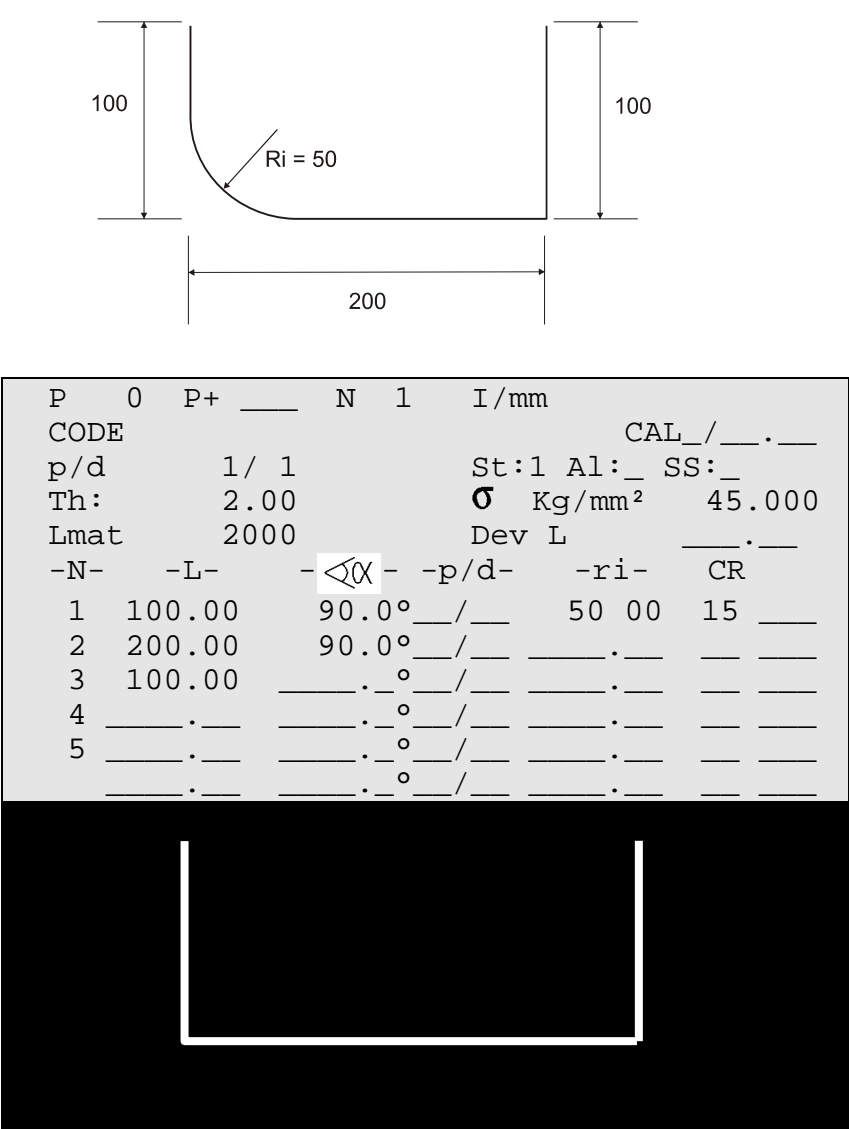
- Press the ➡️▯ key.

The system fills in the **Dev L**, and **ri** fields.

```
 P    1  N   2          p/d   1/ 1

BEND         1   2   3   4   5   6   7   8   9  10
◁α           1   2  ███       __ __ __ __ __ __
LEG          0   1            __ __ __ __ __ __
```

☞   Note: bends with a big radius are not supported in the graphical display.

The total number of steps (**CR**) in the big radius must be such that the X distance is greater than half the V-width. Otherwise, the final resulting angle is incorrect.

CR = 4

CR = 8
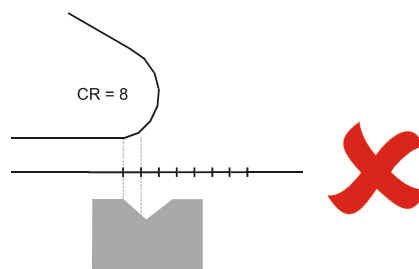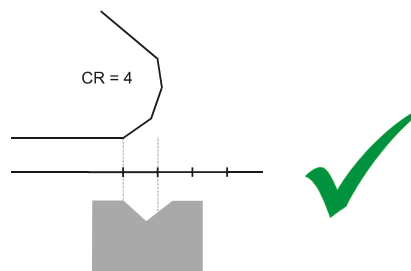
- Press the [P] key and fill in the **LEG** and ◁α fields as below.

```
 P    1  N   2          p/d   1/ 1

BEND         1   2   3   4   5   6   7   8   9  10
◁α           1   2  ███       __ __ __ __ __ __
LEG          0   1            __ __ __ __ __ __
```

Caution   On the sequence page, the **X** values in the **TARGET** column do not correspond to the position actually aimed for by the axis during ideal curve.

CYCLE WITHOUT BEND

The cycle without bend is used to move the axes and auxiliary functions without moving the beam.

To program a cycle without bend:

- Delete the ⊲α field.

- Program Y1 and Y2 at a value greater than that of the pinch point (PP).

The cycle proceeds as follows:

- Descent command
- TDC output cut
- End of bend impulse
- Sequence forward
- Start of axes and auxiliary functions depending on the mode
- TDC output activated
- Control synchro impulse
- Wait descent command

This cycle has been defined for re-arming the electrical cabinet for a new bend.

PUNCHING

To use punching tools on the pressbrake, it is essential to know the exit value of the punch.

Then the DNC is to program in such a way that the bottom dead center corresponds to this exit point.

If necessary, a fictitious tools pair can be programmed in order to adjust the reference height such that the clamping point is just above the pressing up of the punching tools.

To find these values:

- Make a descent with the machine in adjustment mode and the DNC in mode.

- When the punch reaches its exit point, stop the descent and note the position of the beam.

- Delete the ⊲α field.

- Introduce this value in Y1 Y2.

- Regulate the tool reference value as a function of the tools used.

BOTTOMING

As for most work under pressure, it is preferable to do this type of bending in the center of the press.

- Delete the $\triangleleft\!(\!\!\alpha$ field.

- Delete or leave the **Y1** and **Y2** fields unprogrammed.

The press makes the descent in the conventional way until the beam stops in the bottom of the die with the pressure of the current sequence.

The DNC notes that the beam has stopped and the ascent cycle is then switched on.

ANGLE CORRECTION

It can happen that the angle bent is slightly greater or less than the angle programmed.

The operator has then the possibility to introduce the real value of the angle bent. The DNC will correct the Y1 and Y2 axis values in consequence.

This page allows to introduce and display the corrections on the calculated axis values for the current product and bend.

Call the **CORRECTION** page by pressing the ⬜C key.

The passage to the following or previous bend is made by using the ➡⬜ or ⬜⬅ keys.

```
N   1   CY ___
             -COR N-  -COR P-
◁(α          90.0°
◁(α MES     ____._°    ____._°
   Y1       ____.___   ____.___
   Y2       ____.___   ____.___
   X        ____.___   ____.___
```

COR N	This column allows to make a correction for the current sequence only. (The wording **COR N** may possibly not appear according to the machine configuration, however it concerns always the fields of the left column).
COR P	This column allows to make corrections on the whole product in one single operation.
$\sphericalangle \alpha$	Value of the angle programmed in the sequence. This field is not programmable on this page. It simply repeats the desired value.
$\sphericalangle \alpha$ **MES**	Value of the angle measured after bending. By leaving the field the DNC calculates automatically the correction for Y1-Y2. According to whether the operator will apply this correction to the current sequence or to all the sequences having the same angle, the measured value is to program in the **COR N** or **COR P** column.
Y1, Y2, X, etc.	Direct corrections on the axes.

The following working method is recommended:

- Program or call the product.

- Pass to semi-automatic mode.

- Execute the first bend.

- Measure the result.

- Enter the measured angle in the **COR N** or **COR P** field (see above). The DNC calculates automatically the corrections to apply in Y1 and Y2.

- If necessary, make the corrections on the gauge axes.

- Bend a second product on the first bend, in order to control the corrections.

- Pass to the next sequence and proceed in the same way for the following sequences.

- When all the bends are correct, pass to automatic mode and produce normally.

Axes Y1-Y2   If the operator wishes to intervene directly on the values of the Y1 and Y2 axes, he has to do it on this page directly in the Y1-Y2 fields, and not in the sequence page $\boxed{\text{N}}$.

ALPHANUMERICAL CHARACTERS

The CODE field allows the introduction of alphanumerical characters.

To introduce these characters, the [+] [−] and [--] keys are used, combined with the numerical keys of the keyboard.

Example: To introduce the character A, press the [+] key and keeping the latter held down, press the **0** key.

Here is the correspondence of these keys:

	0	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**
[+]	A	D	G	J	M	P	S	V	Y	\
[−]	B	E	H	K	N	Q	T	W	Z	]
[--]	C	F	I	L	O	R	U	X	[	^

MESSAGES / ERRORS

When passing from programming or manual mode into semi-automatic or automatic mode, the safety systems are verified and any error is mentioned by the impossibility to pass to the chosen mode.

The DNC forces the [⊡] mode; the cursor positions itself on the field concerned and the interactive field indicates **ERROR**.

The error must then be corrected and the passage to [⊡] or [⊡] mode be tried again. If another error exists the process repeats itself until all the errors have been corrected.

By error we understand the programming of a value not realizable by the DNC (further than the limit switches, collision, etc.).

INTERACTIVE MESSAGES

CASSETTE	Indicates that the "cassette" (internal memory) - has not been initialized - is polluted - has changed the format (e.g. by changing the software version). Just "clean" the internal memory by placing the cursor on the **P** field, entering **999** and pressing the [key icon] key.
COMPLETE	Displayed in [mode icon] mode when the buffer is full. (No more space for an additional sequence).
COPIED	Displayed in [mode icon] mode when a new sequence has been created at the end of the program using the [key icon] key.
DELETED	Displayed in [mode icon] mode when a product or sequence is deleted using the [key icon] key.
END	Displayed in [mode icon] or [mode icon] mode, indicates the last sequence of the program.
Err - axes	Following error (theoretical trajectory).
Err - FC	Limit switch error.
ERROR	This message is indicated when a calculation is impossible, or would create out-of-range values. When security measures are in force (in other words when an axis is told to move beyond its authorized limits). This information is followed by a return to [mode icon] mode and the cursor positions itself automatically over the field to be corrected.
ERROR	In the case of a transmission exterior ->DNC, this message indicates that the data received are not valid, and thus impossible to read.
EXECUTED	The DNC has made the calculation correctly (corrections, transformations, etc.).
EXISTS	Indicates that the product or tool number chosen for storage already exists.
FREE	Displayed in [mode icon] mode when the buffer is empty (no

sequences programmed).

IGNORE   Displayed when the key used cannot (must not) be used at that time.

INSERTED   Indicates that the insertion has been made correctly.

Jitter   Error caused by the application of simultaneous edges on the input encoder signals (can be provoked by a parasite, a fault in the encoder or a short-circuit on the encoder phases).

LAST   Displayed in ▢ mode when the sequence shown is the last of the product.

NEXT   Is displayed during successive display (▢ key) of the sequences which exist in the buffer (except for the last sequence which indicates END).

NO ACTIVE   Indicates an attempt to change to ▢ or ▢ mode on a sequence where the CY is programmed at 0.

NO PLACE   Indicates that the product list memory has no more space available.

PAUSE   Axis pause.

PREVIOUS   Same as NEXT but with the ▢ key.

RUN   Indicates that the transmission is being made correctly. This message flashes slowly for the duration of the transmission.

STOP   Indicates that the transmission has been interrupted by means of the DNC STOP key.

STORED   Indicates that the storage operation has been made correctly.

synchro   The maximum authorized separation (X1<->X2) has been exceeded.

TEACH OK   The TEACH operation has been made correctly.

TIME OUT   Indicates that the DNC has not received any valid information since the beginning of the transmission, and thus returns the command to the operator. This message appears after about 30 seconds.

TOL ZONE   Displayed in ▢ or ▢ mode when the axes are positioned outside of their tolerances. Generally, just press the start of the front pannel to position the axes will do.

UNDEFINED   Refusal to change to ⊡ or ⊡, if the parameters vital to execution are not or only partially programmed and thus prevent the verification calculations.

Var KO   One of the DNC inputs "SERVO DRIVER OK" is not (no longer) active (+24 VDC).

This page has been left blank intentionally.

APPENDIX H: TRANSFER

For transfer to a PC by serial line the DNC 600S must be equipped with the RS232 option.

Products

The PC must be equipped with the CYBELEC PC1200 software with option LINK7000. This combination allows to create products in the bending software on PC and to transfer them to the DNC for execution.

Backup

The PC must be equipped with the CYBACK storage software. CYBACK allows to make a backup of the data in the DNC without intervention on the data on the PC's level.

Updating

Easy, practical and very rapid, this possibility facilitates in case of necessity the updating of the DNC software in the FLASH memory. The PC must be equipped with the updating software.

- Call the **TRANSFER DNC <--> PC** page via the main menu.

```
    DNC <-> PC   STATUS TEST
  TRANSFER    _  1,2 Cyback
                 3,4 DATA LINK TEST 1 2
                 7 FLASH
                 8 EXT KEYB


                 11 Internal backup
                 12 Internal restore
Quantity of free backup memory    96%
          -- COMPUTER LINK --
Baud rate 57600              Stop 1.0
Parity odd (1) EVEN (0)  _   Bits 8
```

Who commands ?

This page will only be used to initialize the transmission type. It is not possible to start the transmission from the DNC, this will always be done from the PC.

TRANSFER	Allows to select the type of transmission.
Cyback	Will initialize a data transfer between the PC utility Cyback and the DNC. For details, refer to page 80, *CYBACK*.
DATA LINK TEST	Selects the test mode of the serial ports. See the following chapter.
FLASH	Prepares the DNC to receive an updating.
EXT KEYBD	Puts the DNC in external terminal mode. See the concerned chapter.
Internal backup	Launches a backup of all machine parameters and/or tools onto the Flash memory For details, refer to page 98, *Internal Backup*.
Internal restore	Restores all machine paramters and/or tools previously backed up in the Flash memory. For details, refer to page 100, *Internal restore of PMs and tools*.

TESTSING THE SERIAL PORTS

The 2 serial ports of the DNC 600S are SUB-D 9P, plugs J5 and J6.

- Introduce and verify that the transmission parameters are programmed as follows:

```
PARAMETERS RS232 / DIVERS

 . . .

      -- COMPUTER LINK --
106 BAUD RATE  4800         STOP  1.0
    PARITY ODD(1) EVEN(2) _  BITS  8
    PROTOCOLE _             BCC  ___
```

- Set a test loopback connector on the serial line connector RS232 to be tested.

- Press the [⇥▢] key to start the test.
  The message **RUN** must flash.

- To stop the test, press the [◉] (STOP) key.

If you wish as well to test the cable:

- Connect the transmission cable to the DNC and fit a short circuiting loopback connector on the other end of the cable to be tested.

Test Loopback Connector

Pin 2 (RXD) wired to pin 3 (TXD)
Pin 4 (DTR) wired to pin 6 (DSR)
Pin 7 (RTS) wired to pin 8 (CTS)

RS 232 Transmission Cable

Max: 15 m
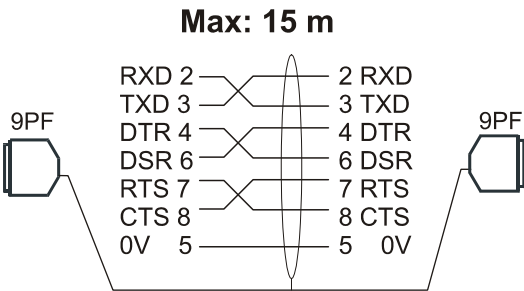
The shield must be connected on the metallic hood of the Sub-D plugs.

CYBACK

For this program, the RS cable must be connected on the RS232 port of the J5 plug.
The transmission parameters must be programmed on the DNC with the same values as for the test of the serial ports (see paragraph above). These same values must be programmed on the PC.

If the transmission runs at 4800 baud, you can try to increase the transmission speed at 9600 baud.
The RS 232 norm gives as limit 15 m. By using a more important distance, there is a risk of bad transmission and it will be necessary to diminue the transmission speed.

Reminder: The transmission command can be given only from the PC side.

To permit the serial transmission with CYBACK:

Pass to programming mode.

- Call the **TRANFER DNC <-> PC** page.

- Introduce **2** in the **TRANSFER** field.

- Press the ⬛ key.
  The display indicates **STATUS RS232**.

Programming mode

From this instant on, the DNC can receive data sent from the PC, provided that the DNC is in programming mode.
In the other modes (auto, semi-auto or manual) the DNC doesn't respond, the transmission is not accepted.
During the transmission the operator can leave the DNC on any page. It is not necessary to display the transfer page.

The transfer mode remains memorized, even if the DNC is disconnected from power supply. It has to be re-programmed if the mode has been changed (for instance FLASH).

This page has been left blank intentionally.

APPENDIX I: PRINTING THE CURRENT SCREEN

The DNC 600S authorizes, if the RS232 port is included (optional), to print the content of a page displayed on the screen (connector J5).

This allows, through a hardcopy, to retain information regarding the product(s), tools or machine parameters.

To be able to send this information to the printer, **it is compulsory that you use a serial printer (RS232).**

You have to program the DNC 600S transmission parameters to the **printer's required specifications**.

Baud rate:	4800 max.
Bits:	7 or 8
Stop bits:	1, 2 etc.
Parity:	1 or leave blank
Protocol :	this field defines how many DNC's pages you want to have on one printer's page. After the relevant number the DNC sends a form feed character.
	Example: 1 = 1 DNC's page per printer'spage.
	2 = 2 DNC's pages per printer's page, etc.
BCC:	this field defines the left margin in cm.
	left margin = 2 + (BCC x 2)
Valid range:	0 (2 space characters) to 9 (20 space characters)
Example:	BCC = 4, then 2 + (4 x 2) = 10

TO PRINT FROM THE DNC

- Select the relevant page.

- Simultaneously press the stop and correction or menu keys.

HYPERTERMINAL

It is possible to enter all the pages of a DNC600S on a PC in a txt file, which allows for a hardcopy backup of the machine parameters, programs or other DNC 600S elements.

CONNECTION

Connect an inverted (non-modem) RS 232 cable between the J5 connector of the DNC and the RS 232 port of the computer. In the case of a portable computer without an RS 232 port, a USB-RS232 converter has to be used (e.g. an Aten brand converter).
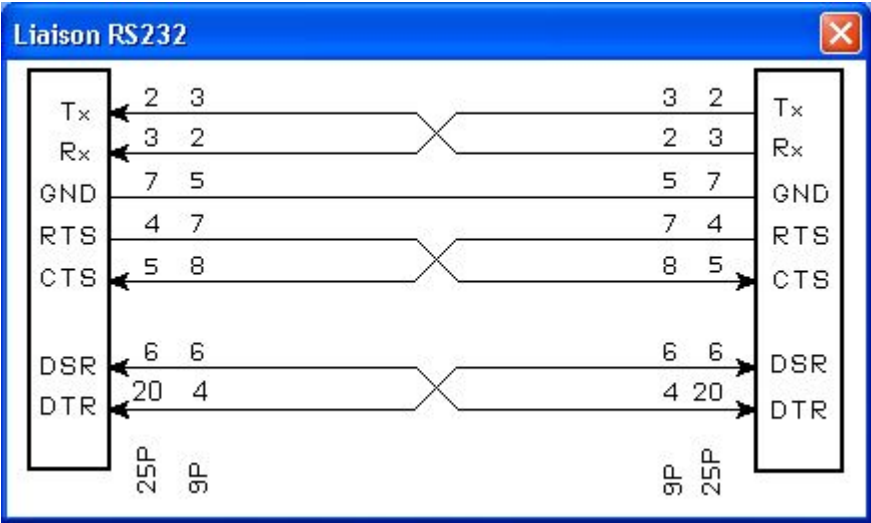
PREPARATION

The connection parameters must be the same on the DNC as on the HyperTerminal.

DNC 600S Parameters

```
 RS232/STOPS/DIVERSE


104 DWELL TIME IN MINUTES                    _


     ---- COMPUTER LINK ----
106 BAUD RATE 57600            STOP 1.0
    PARITY ODD (1) EVEN (0)  BITS 8
    PROTOCOLE_                BCC
```

Protocole: The number of DNC pages printed on a page is indicated in this field.
According to the indicated number, DNC sends a page-feed to the printer.

Example:
1 = 1 DNC page per printer page.
2 = 2 DNC pages per printer page.
If nothing is programmed, the pages are sent one after the other.

BCC: this field defines the left margin in characters (spaces).
left margin= 2 + (BCC x 2)

Field: 0 (2 characters) to 9 (20 characters)
Example: BCC = 4, i.e. 2 + (4 x 2) = 10

(i) After having modified the parameters:
- Go to the **TRANSFER** page.

- Select **1** then hit the [➡️] key to confirm the transmission with the new parameters, or turn the DNC off and on again.

Configuring HyperTerminal

- Enter a name for the file which will contain the parameter setting for this numerical command and choose an icon. You will call up this file every time you connect to the numerical command without having to reset the Hyper Terminal parameters.
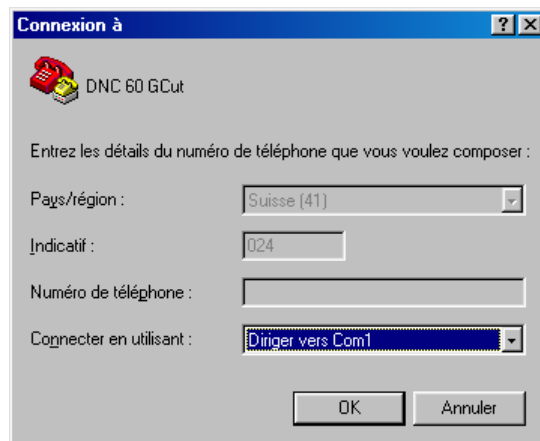
- Connect via a serial port.
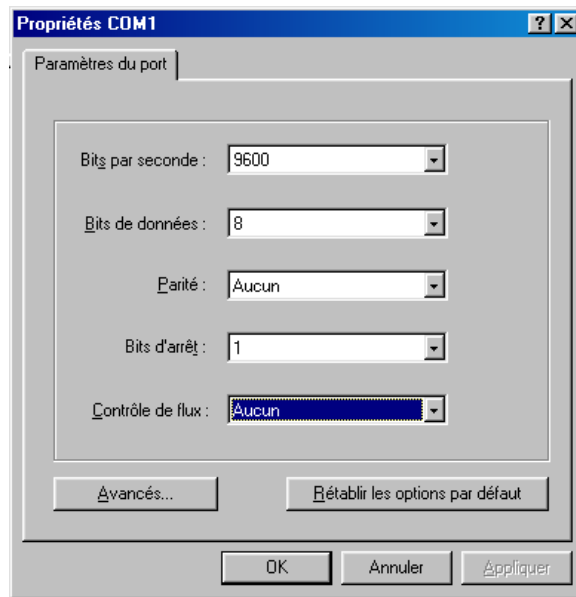
- • - Port parameters:
  Bits per second: 9600
  Data bits: 8
  Parity: None
  Stop bits: 1
  Control flow: None
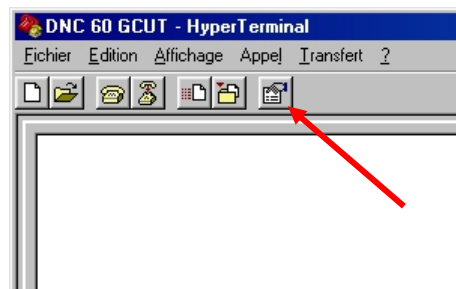
- • Select properties:
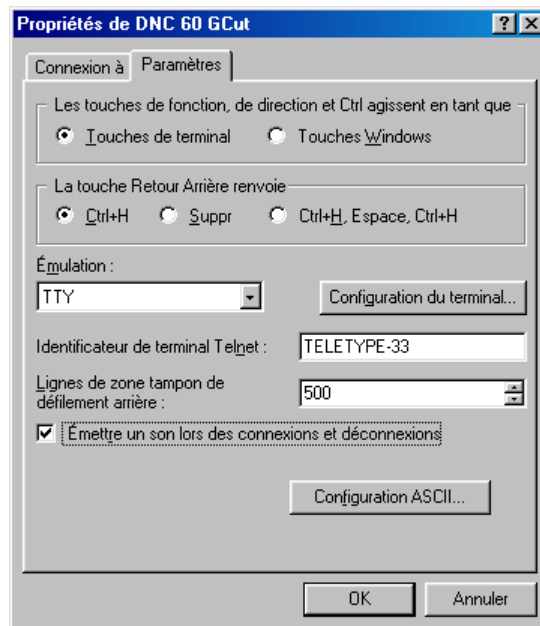
- Enter the parameters as above.
- Click the **ASCII Configuration** button.
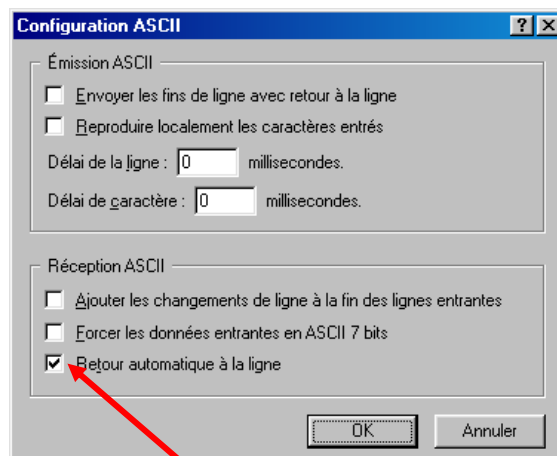
- Check the **Line Wordwrap** box. Then **OK**
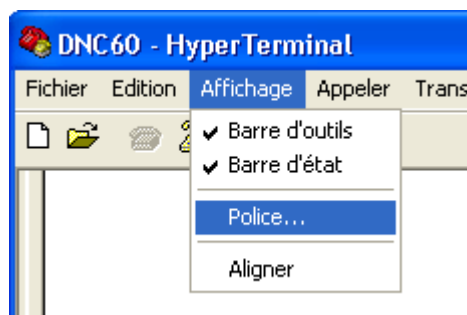
- Select **Display** then **Font**
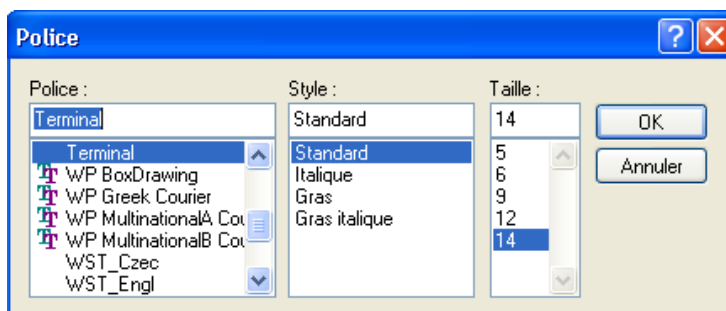
- Choose **Standard Terminal 14**

ENTERING THE SCREENS

The parameters have now been set on both the DNC and the Hyper Terminal, which have been interconnected are ready to transmit.
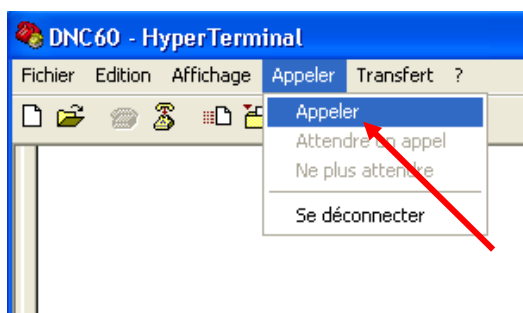
- Open the **Call** menu,
  - Select **Call**
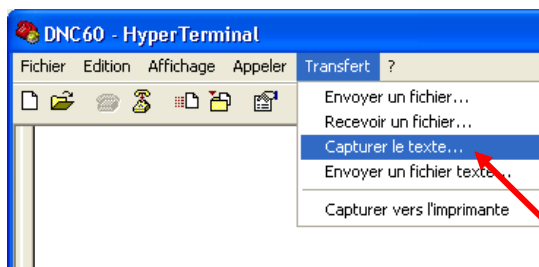
- Open the **Transfer** menu.
  - Select **Text Capture.**
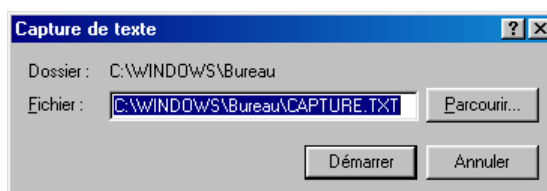
- Select the location and name of the file which will contain the DNC screen pages.

- On DNC 600S, select the page to be printed

- Press the ⊚ and ☐ keys simultaneously.

- Repeat for each desired page.

- Disconnect

- The captured pages are visible in the previously selected txt file.

Printing out all Machine Parameters

With the DNC 600S P, PC and PS, starting from the SIXFDAx version, it is possible to print all the machine parameters in a single operation. To do this:

- Go to the machine parameters

- Press + +, − − and ☐ simultaneously

APPENDIX J: PROTECTING THE ACCESS LEVELS

GENERAL INFORMATION

Depending on the version, the DNC 600S can or can not be equipped with a 4-positions physical key.
However the protection levels 0-1-2 and 3 still exist.
For the case where the physical key doesn't exist, the access is made by password.

CAUTION : these passwords will be reset in the default values at each initialization of the machine parameters (817 on INIT page and Choice Init Mach Par = 1).

Thus, in this manual we always will speak of a (virtual) key position like e.g.: *"Key in position 3"*.

Levels

There exist 4 access levels, 0 to 3.

   0 = Programming prohibited.

   1 = Creation, correction, modification, saving, deleting, transfer of one (or more) product(s).

   2 = Creation, correction, modification, saving, deleting, transfer of the tools.

   3 = Programming, modification and transfer of the machine parameters.

Access

These levels are accessed by pressing the ⊚ keys during two seconds minimum.
A pop-up is displayed.

Press the ⓪ ① ② or ③ keys.
The key position is displayed as a number on the upper right part of the screen (after the interactive field).

When passing to a higher, not authorized level, a password modification will be requested.
When the password has been introduced, you can "navigate" levels inferior or equal to the authorized one without reintroducing the password.
Passing to level 0 resets password validity.

Users

A number of different users are predefined. A user is not a physical person in particular, but can be e.g. all the operators having the authorization to work on the machine.
Each predefined user possesses his own password and a maximum level which he can reach. See further under *Table of users, access and passwords*.

Password

Certain users can modify their own password. For the others, the password can only be changed by a user having a superior access.

Loss of the password

In case of loss of the password, a user of a superior level has to reprogram the password.

USERS

Table of users, access and passwords

Level	Names of predefined users	Changing of the personal password	Changing of passwords of the subordinateds	Level virtual key	Password by default	User generally attributed to:
1	EUL1	NO	NO	**1**	111	Operators having the access authorization of level 1
2	EUL2	NO	NO	**2**	222	Operators having the access authorization of level 2
3	EUL3	NO	NO	**3**	333	Operators having the access authorization of level 3
4	WSSUPER	OK	OK	3		Workshop supervisor
5	MACHMAN	NO	OK	3		Machine manufacturer's Service technicians
6	MACHMAN0	OK	OK	3		Responsible of the technicians at the machine manufacturer's

A predefined user is just a role.
Many physical persons can have the same role. E.g., many physical operators can be a **EUL1** (level **1**).

After installing the machine it is advised to modify the password by default of level **4** (WSSUPER = Workshop supervisor) and of level **3** (EUL3 = Operators with authorization level 3), because the passwords are in this manual.

PASSWORD PROTECTED ACCESS

By starting the software, the virtual key is always positioned at **0**.

- Press the [⊙] key during two seconds minimum.

- Choose the level to access by pressing one of the [1], [2] or [3] keys.

- The message **VALUE ?** appears.

- Introduce the password and press the [→] key to validate the password.

- The message **OK** and the selected level are displayed if the password is accepted, or **KO** if it is refused.

Once the authorization acquired, the operator can change the level among those authorized to him without reappearing of a new password request. For instance, a user with access on level 3 can navigate between levels 1, 2 and 3 without having to give his password again.

If level 0 is activated, the access on any other level will require to introduce the password again.
This request will also appear when the user passes to a superior level (from 0 to 1, from 1 to 2, from 2 to 3, etc.) and he has no access authorization.

Advice   If you have accessed to level **3**, access to level **0** after your intervention. This will avoid to make undesired changings by inadvertence.

ACCESSING LEVELS HIGHER THAN 3

Certain users can access to levels superior to 3, which enables them, among other things, to modify the passwords.
In order to know the authorizations, see *Table of users, access and passwords*.

- Press the [image] key during two seconds minimum.

- A pop-up is displayed.

- Press the [4] key.

- The message **LEVEL ?** appears.

- Type the level number you want to log in.

- The message **VALUE ?** appears.

- Introduce the password corresponding to the requested level and validate with the [image] key.

- The DNC passes to level **1**. The operator can "navigate" between levels **1** and **3** without reintroducing his password.
  If his access level enables him, he can call the procedure of password modification (see next paragraph).

- At the end of the intervention, don't forget to pass to level **0** in order to leave the current level.

CHANGING A PASSWORD

It is possible to modify the passwords attributed by default.

Certain users can do it for themselves, others not. In order to know the authorizations, see *Table of users, access and passwords* on page 91.

To change a password:

- Press the [⊙] keys during two seconds minimum.

- A pop-up is displayed.

- Press the [4] key.

- The message **LEVEL ?** appears.

- Type the level number you want to log in.

- The message **VALUE ?** appears.

- Introduce the password corresponding to the requested level and validate with the [➔] key.

- Then press the [⊙] keys during two seconds minimum.

- A pop-up is displayed.

- Press the [5] key.

- The message **LEVEL ?** appears.

- Program the level on which you want to modify the password.
  The message **IGNORE** appears if the requested level is superior to the "logged" level, in the contrary, the message **VALUE ?** appears.

- Introduce the desired new password and validate with the [➔] key.

- The message **CONFIRM** appears.

- Introduce the new password again and validate with [➔].

- If the 2 registered passwords are identical, the message **OK** is displayed.

- If the 2 registered passwords are different, the user is invited to introduce the 2 passwords again, the message **VALUE** is displayed.

- At the end of the intervention, don't forget to pass to level **0** in order to leave the current level.

MANAGING ACCESS LEVELS WITH AN EXTERNAL KEY

If the use of the external key is defined in the machine parameters, the access levels are managed by the external key. The passwords are no more active.

The use of the external key is defined by the parameter **Key = 1** and by the use of the inputs *KEY 0* and *KEY 1*.

Input KEY 0	Input KEY 1	Pos. key
OFF	OFF	**0**
ON	OFF	**1**
OFF	ON	**2**
ON	ON	**3**

MAINTENANCE

LCD SCREEN

Given its design the screen can only function correctly within a certain range of ambient DNC temperatures, namely from +10°C to +40°C.

"WHITE" SCREEN

Low temperature   At low temperature (less than 10°C), the screen will remain temporarily white (transparent) until the temperature range mentioned above is attained.

If the temperature around the DNC is below 10°C, then it will suffice to switch on a few minutes before operation starts. The duration of this warming up period depends on the ambient temperature, and it can be of the order of 15-20 minutes if the temperature is very low.

"BLACK" SCREEN

High temperature   At high temperature (above 40°C) the screen will remain temporarily "black" (opaque) until the working temperature returns to a value within the range mentioned above. If the DNC is fixed within a heat generating enclosure (such as an electrical cabinet), the solution consists of cooling the whole unit to bring down the temperature to within the limits quoted here.

ADJUSTMENT

Adjusting the contrast   In the two cases above it is possible to adjust the screen contrast as soon as the temperature approaches the prescribed limits. To adjust the screen viewing angle (contrast), take the following steps

- Select the programming mode .
- Go to a page where access to the **P** field (Program) is possible.
- Position the cursor on the **P** field.
- To increase the contrast (lighten), press the [+] key.
- To decrease the contrast (darken), press the [–] key.

IN CASE OF FAILURE

INITIAL CHECKS TO CARRY OUT

- Switch the machine off then on again.
- Check the program (safety factors).
- Check that the axes can be displaced in manual mode.
- Check the machine parameters against the written list made during the installation.
- Check power supply voltages (to do this please refer to the technical manual).
- Verify that the keyboard responds.
- Check that front panel lamps light up.

POSSIBLE MEMORY POLLUTION

After having made the initial checks above, without finding a fault, it is then likely that you have a memory pollution problem. To localize the fault, take the following steps:

- Switch off the DNC.
- Turn the key to position **3**.
- Switch on the DNC.
- Follow the instructions in the chapter "Initialization page", page 55.

BEFORE CALLING FOR TECHNICAL HELP

Please have ready the following information:

- Type of numerical control.
- Serial Nr of the DNC.
- DNC and axes (N2X) software numbers.
- Name of the machine tool's manufacturer.
- List of machine parameters (have on hand when the technician comes).
- The state of the numerical control:
  - which LEDs are on?
  - which message is on the screen ?
  - in which position is the key ?
- Details of what caused or is causing the problem.

INTERNAL BACKUP

A machine parameter and tools backup is possible in the internal memory, as well as a restore since this same internal memory.

This functionality allows the manufacturer to memorize at the time of the delivery of the machine, the parameters and tools state.
This will allow thereafter, in the event of problems, the operator to easily reload the original parameters and/or tools delivered. This without the necessity to connect the DNC to a PC. Only the last compatible backup can be restored.

The memory quantity is limited. Each backup take some memory place (no rewrite possible) and the number of backup depends of the software version and the number of tools.

If this memory is full, it can be only deleted by reinstalling the boot software. (See further). Operation which must imperatively be done by a qualified and trained person.

The backup quantity possible (generally 1 to 2) is enough for correct use of this function.

Internal Backup of the PMs and Tools

For save the machine parameters and tools, go in the **TRANSFER RS232** page from the main menu.

```
    DNC <-> PC   STATUS TEST
  TRANSFER    _  1,2 Cyback
                 3,4 DATA LINK TEST 1 2
                 7 FLASH
                 8 EXT KEYB


                 11 Internal backup
                 12 Internal restore
Quantity of free backup memory    96%
          -- COMPUTER LINK --
Baud rate 57600              Stop 1.0
Parity odd (1) EVEN (0)  _   Bits 8
```

- Select **11 INTERNAL BACKUP**

```
   DNC <-> PC    STATUS TEST
TRANSFER  1    1, 2 Cyback
               3,4 TEST LIAISON 1 2
               7 FLASH

   Internal backup

       0 = Abort
Qu     1 = Save machine parameters   %
       2 = Save tools

      --  RS232 PARAMETERS  --
Baud rate  57600            Stop  1.0
Parity odd (1) EVEN (0) _  Bits  8
```

- Select the operation to do by pressing the 0, 1 or 2 key.
  The pop-up is closing.

- The message **value ?** is displayed at the top right of the screen (as to change to level 2 or 3).
  Introduce the level 4 code for the machine parameters or the 217 code for the tools.

- Press the ⟳ key.

```
   DNC <-> PC    STATUS
TRANSFER  1    1, 2 Cyback
               3,4 TEST LIAISON 1 2
               7 FLASH
               8 EXT KEYBOARD
              11  INTERNAL BACKUP
              12  INTERNAL RESTORE
Quantity of free backup memory   93 %


      --  RS232 PARAMETERS  --
Baud rate  57600            Stop  1.0
Parity odd (1) EVEN (0) _  Bits  8
```

When the backup has finished, the EXECUTE message and the new free memory percentage are displayed.

INTERNAL RESTORE OF PMS AND TOOLS

This functionality is to use only, in the event of problems, by the operator on request of the manufacturer.

The parameters restoration will replace the current contents of DNC by the last parameters backup (generally by the manufacturer).

The tools restoration will replace all tools data which have the same number. The tools which have a different number as these in the backup are not affected by the restoration.

Example: The punches number 1 and 2 has been saved.
The user has modified the data of the punches 1 and 2 and he has added the punches 12, 14 and 15.
At the moment of the restoration, the value of the data of the punches 1 and 2 will be replaced by these of the saved tools.
The punches 12, 14 and 15 aren't affected.

Only the last compatible backup can be restored.

For restore the machine parameters and tools, go in the **TRANSFER RS232** page from the main menu.

```
   DNC <-> PC    STATUS
 TRANSFER  1     1, 2 Cyback
                 3,4 TEST LIAISON 1 2
                 7 FLASH
                 8 EXT KEYBOARD
                11 INTERNAL BACKUP
                12 INTERNAL RESTORE
Quantity of free backup memory    93 %


    -- RS232 PARAMETERS  --
Baud rate  57600           Stop  1.0
Parity odd (1) EVEN (0) _  Bits  8
```

- Select **12 INTERNAL RESTORE**

```
   DNC <-> PC    STATUS TEST
TRANSFER  11   1 2 Cyback
               3,4 DATA LINK TEST 1 2
               7 FLASH

   Internal restore

        0 = Abort
Qu      1 = Restore machine parameters
        2 = Restore tools

       --  RS232 PARAMETERS  --
Baud rate  57600              Stop  1.0
Parity odd (1) EVEN (0) _  Bits  8
```

- Select the operation to do by pressing the 0, 1 or 2 key.

- The pop-up is closing.

- The message **value ?** is displayed at the top right of the screen (as to change to level 2 or 3).
  Introduce the 718 code for restore the machine parameters or the tools.

- Press the ⬦ key.

```
TRANSFER  11    1 2 Cyback
                3,4 DATA LINK TEST 1 2
                7 FLASH
                8 EXT KEYBD

Restore successfully done

         Press any key to reboot


Baud rate  57600              Stop  1.0
Parity odd (1) EVEN (0) _  Bits  8
```

When the backup has finished, a pop-up is displayed.

ACCESSORIES

CYBACK   Software for storage in PC.
Allows the storage and recall of data (products, tools, machine
parameters) for the DNC 7000, DNC 70, DNC 30, DNC 60 and DNC
600S CNCs in a PC via the serial line.
Up to 10 different CNCs can be selected.
Ask for our "Product information" about CYBACK.

PC1200   PC1200 is a Windows program. This software allows to program on a PC
in 2 and 3 dimensions the products to be realized on the numerical
control.
This software executes the simulation and graphic display of the bending
order taking into account the characteristics of the machine, the tools and
the material.

This page has been left blank intentionally.

INDEX

W

Warranty, III