DA-56

Reference Manual
Operation
of Version V1

PREFACE

This manual describes the operation of the Delem controller type DA-56 and is meant for operators who are instructed for operation of the total machine.
Only authorised people should be able to create new or edit existing programs, as well as programming or editing of the tooldata. Therefore the controller has a keyswitch to prevent uncontrolled programming. With the keyswitch in the off position the operator can only execute a selected program. The operator cannot change the parameters to wrong, unwanted values.

Version history

The control software is updated regularly to increase performance and add new functionality. This manual is also updated as a result of changes in the control software. The following overview shows the relation between software and manual versions.

Software version	Manual version	Description
V1.1	V0505	first issue V1
V1.2	V0606	update

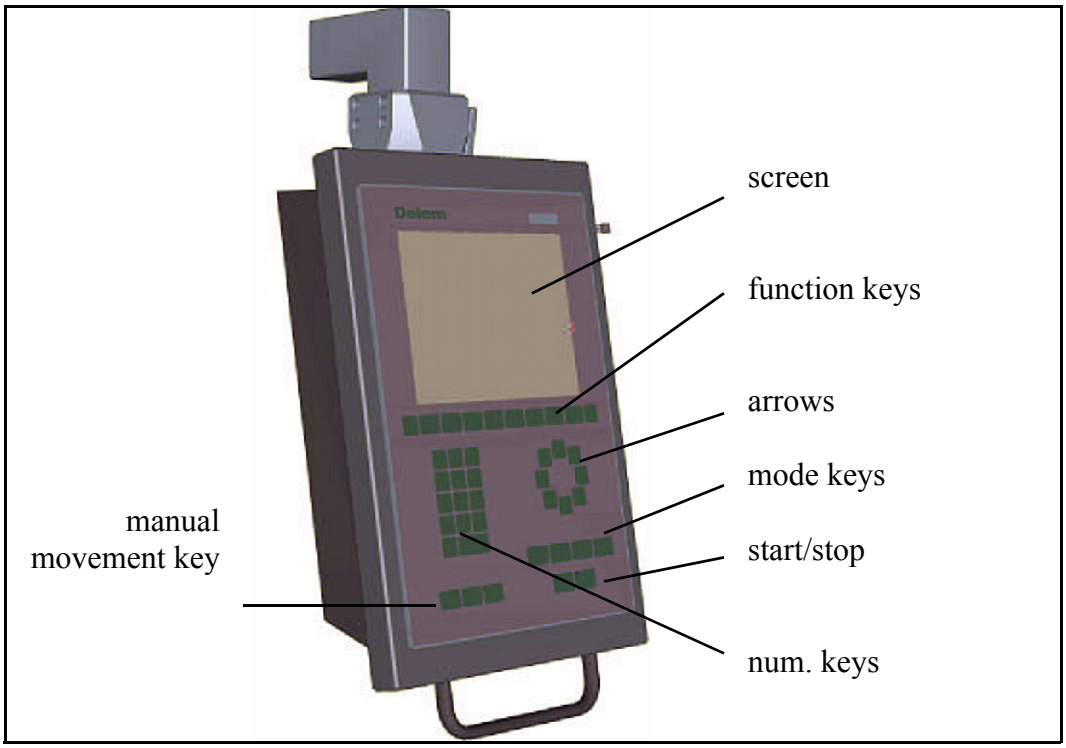
This manual is valid for software version 1.2.

Table of contents

1. Operation overview and general introduction

1.1. The control unit

The control looks as follows:



1.a

The precise outfit of your control may vary.
Operation of the control is done with the various keys on the front panel. A description of all keys and their functions is given in the next section.
Beside the front panel keys, an external USB mouse can be used as a pointing device to select menu items, parameters or softkeys. It depends on your configuration whether such a device is available. In this manual the phrase ‘mouse’ is used to describe any of these possible pointing devices.

1.2. Operation modes

The control has the 4 following modes:

Manual mode In this mode it is possible to program all parameters of just one bending. After pushing the start button all parameters are active and the backgauge will go into position. It is also possible to move the axes manually.

Programming mode In this mode bend programs can be made or edited and also be written to or read from disk.

Automatic mode The selected program can be executed automatically.

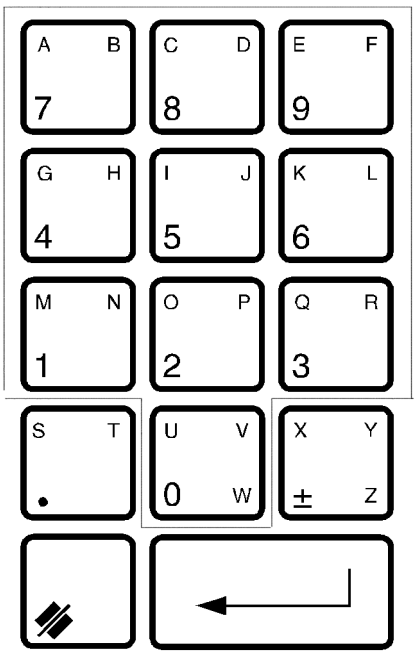
Step by step mode The selected program can be executed bend by bend.

Each mode can be selected by pressing the relevant push button. A LED in the push button indicates whether or not this mode is active.

1.3. Frontpanel

The frontpanel, besides the 4 operation mode buttons, consists of the following items :

Keyboard:



10 numerical keys (0-9) incl. alphanumeric input

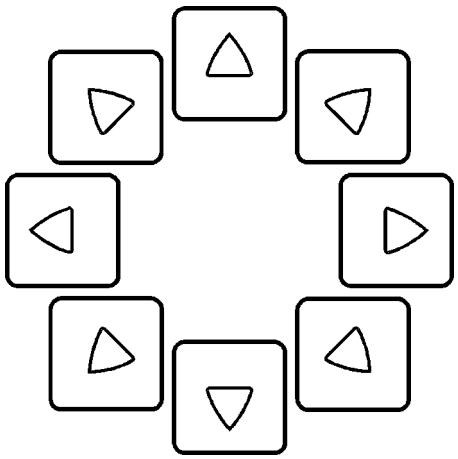
decimal point

plus/minus toggle

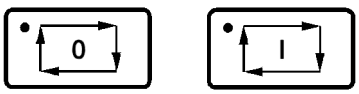
clear key: Clearance of the input data field in the bottom left corner on the monitor screen

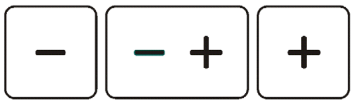
enter key, to confirm a programmed value

Cursor path control:



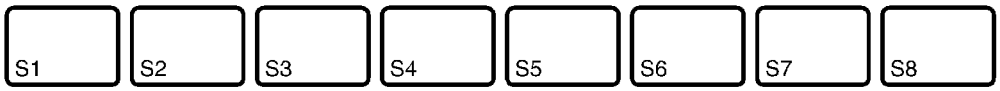
Stop button Start button



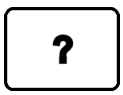


Manual movement of any axis (Y + backgauge axes)

Softkeys; The function of these keys is stated at the bottom side of the monitor screen.



End of menu program.
It is also possible to leave a menu by pressing <ESC> on an external keyboard or clicking with the mouse on the menu symbol in the upper left corner.



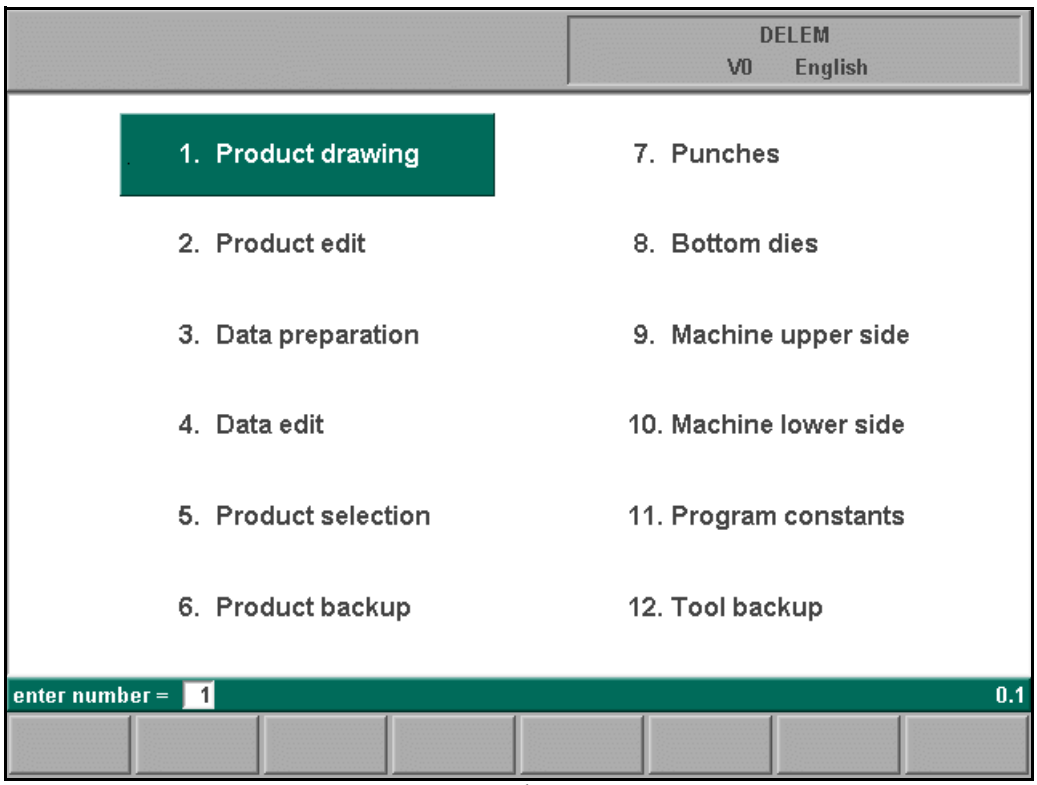
On the screen pages where you find the "?" symbol you can press the "?" to get an explanation of the respective function or parameter to program.

1.4. Programming mode

1.4.1. The main menu

Programming mode Select the programming mode.

The main menu in programming mode looks as follows:



1.b

Each of these menu items can be selected in several ways:
- enter the menu number and press ENTER
- use the arrow keys to scroll to the desired menu item and press ENTER
- click with the mouse once on the desired menu item

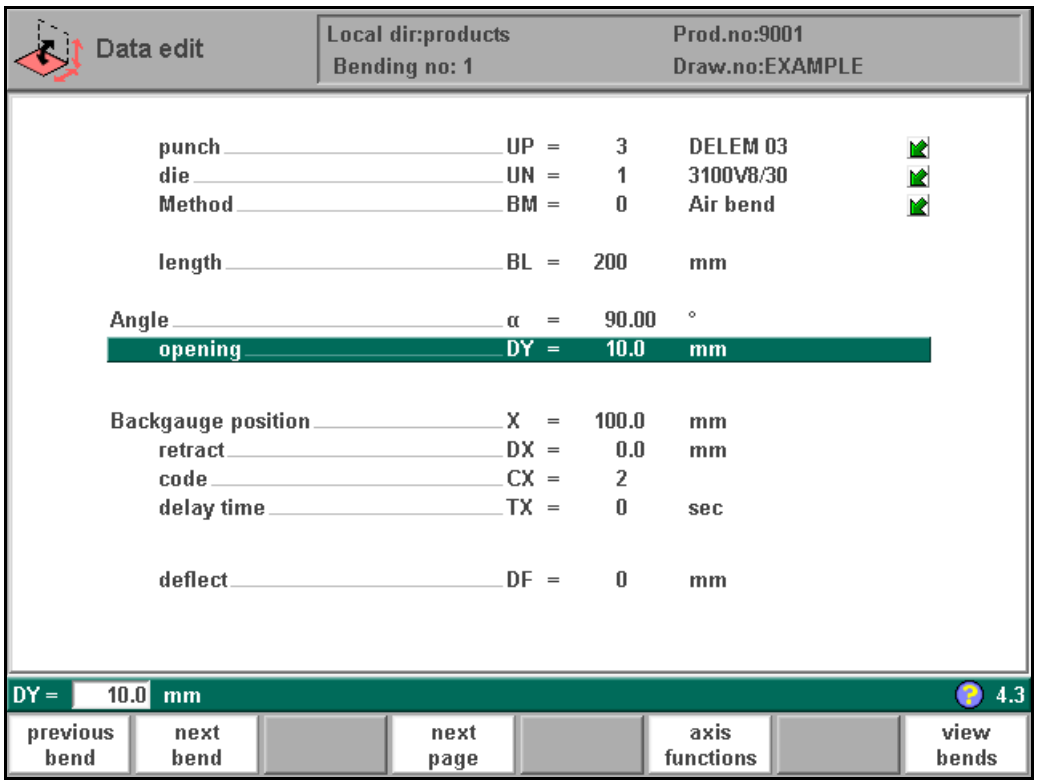
Explanation of menu items:

1	To draw your product on the screen and compute the bendsequence (graphical).
2	To edit your product drawing and compute the bendsequence (graphical).

3	Creation of a new CNC-program by data entry (numerical).
4	To edit an existing CNC-program by data entry (numerical).
5	To select a program out of the product library.
6	Write programs to or read programs from a back-up medium.
7	To program the dimensions of the upper tools.
8	To program the dimensions of the under tools.
9	To program the dimensions of the upper side of your machine.
10	To program the dimensions of your machine table.
11	To program specific programming data.
12	Write or read tooling data and machine shapes to or from a back-up medium.

1.4.2. Help text

This control is equipped with an on-line Help function. This help text is available when the question mark '<?>' appears on the screen (lower right corner).

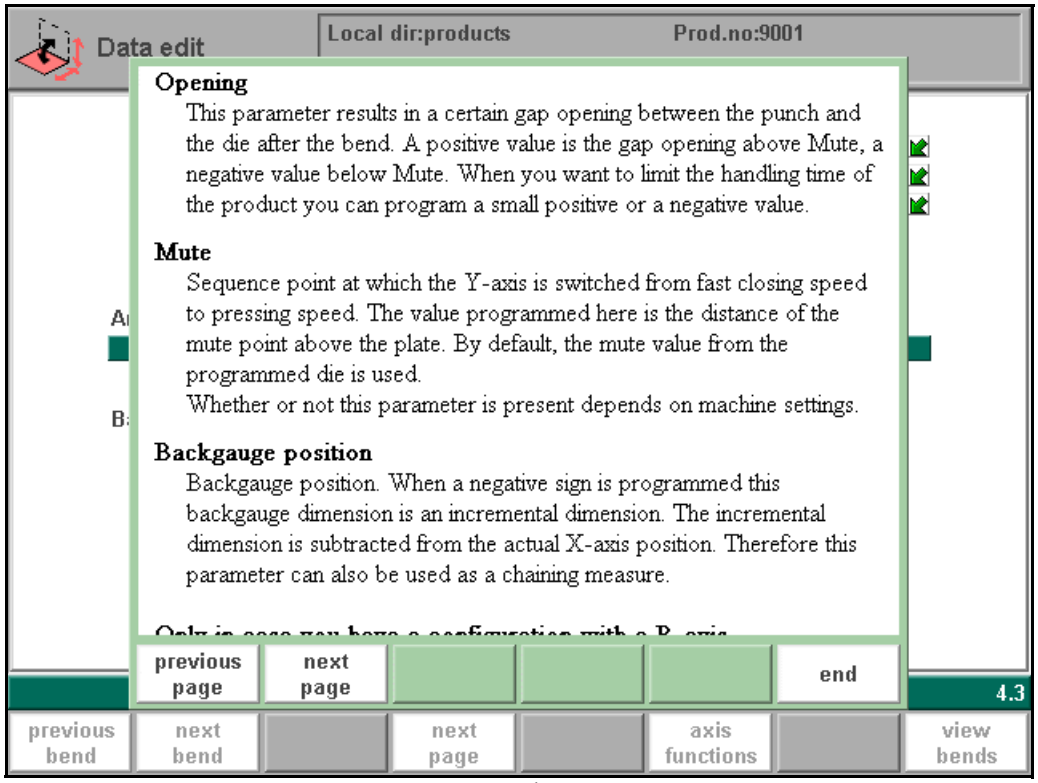


1.c

To activate a help window for a parameter:

- press the question mark key ('?') on the frontpanel or
- click with the mouse on the help symbol.

A pop-up window appears with information on the active parameter.

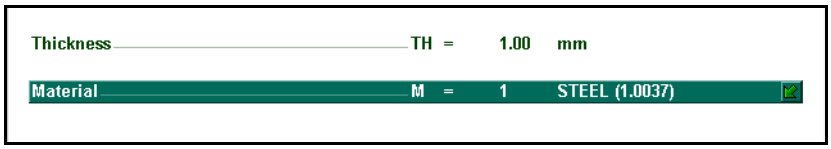


1.d

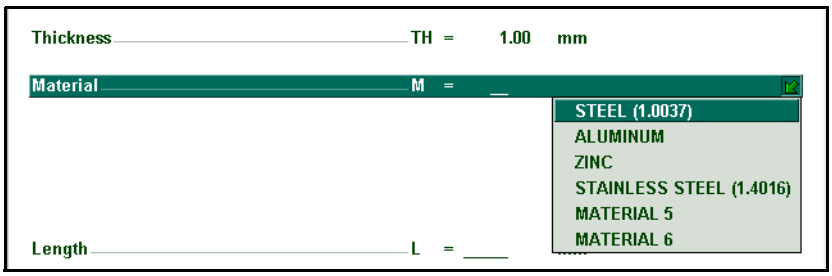
This Help window contains the same information as the Operation manual.
The help window can be used as follows:
Use the arrow keys (up/down) to browse through the help text or use the function keys 'previous page' and 'next page'.
Press the function key 'end' or the END key to close the Help window.

1.4.3. Listbox functionality

Several parameters on the control have a limited number of possible values. Beside such parameters, the sign ⌂ appears.



This means that the  key can be pressed to get a listbox with possible values of this parameter.



This listbox can also be opened by clicking with the mouse on the ⌂ symbol.

1.5. Graphical programming

From the main menu you can select one of the programming possibilities.
In order to design or edit a new product, the graphical programming facility is present.
This facility is a complete product design tool that allows you to draw the profile of your product. It consists of a machine-and tool-library which allows you fast automatic, interactive or manual bending sequence computations with display of possible product/tool/machine collisions and developed length.
The graphic design tool provides a way of selecting the most optimal bend sequence for a product, keeping in mind the minimum production time and manipulation possibilities of the product.

Features of the design tool

- Graphical design of product shapes in 2D
- Auto scaling
- Horizontal and vertical projected dimensions can be entered
- Blank length computation
- Real scale tool design
- 10 different upper side shapes and 10 different under side shapes (tables)
- Changing of lengths and angles
- Bumping (big radius)
- Adding or deleting of bends
- Existing products can be copied, changed and stored as a new product
- Production time indication
- Closing dimension or highest precision tolerance selection
- Connecting 2D programs for 3D-production

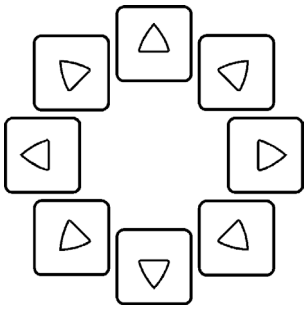
Axes according to the machine configuration

The 2D-programming will be explained in chapter 2.

1.5.1. Control keys

The drawing software uses several function keys on the front panel.

- S1 thru 8
- Drawing cursor control with:



- Zoom function:

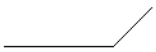
At bendsimulation overview in case of 2D-products (bendsequence menu and automatic mode):

Enlargement

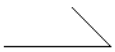
Reducement

- Drawing cursor control for 2D-products

In case you are drawing the profile of your product or tools the cursor keys can be used to give directly multiples 45 degree angles. e.g.:



a = 135°



a = 45°

a = -90°

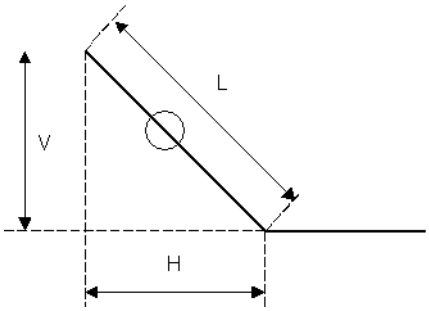
a = 90°

- Horizontal or vertical projections for 2D-products

After you entered the length of the line interval you can specify if this line interval is either the nominal length, horizontal or vertical projection. The given length dimension in the enter field is the line length L if the drawing cursor is in the concerning line.

Horizontal projection with  or  key

Vertical projection with  or  key

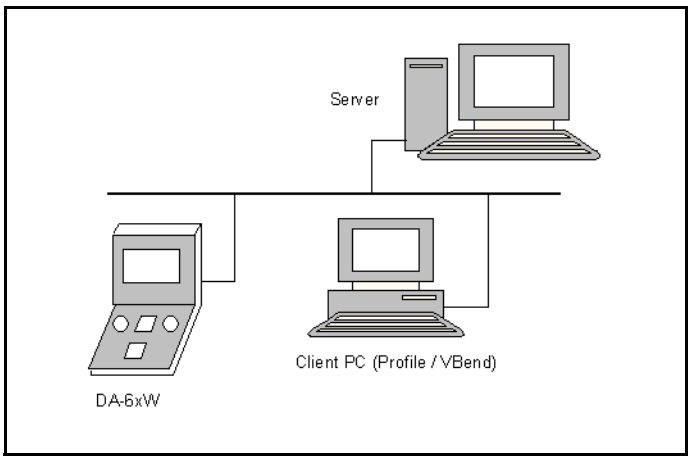


L is normal entered line length
V is vertical projected line length
H is horizontal projected line length

It will be noted on the screen if projection is not possible.

1.6. Network option

The CNC control is equipped with a network interface. If activated, the network function offers the operators the possibility to import product files directly from the network directories or to export the finished product files to the required network directory.



1.e

Chapter 6 contains more information about networking possibilities.

1.7. Software versions

The version of the software in your control is displayed at the upper side of the menu screen in the programming mode.

Example of version number:

V 1.2
V stands for version
1 is version number
2 is version level

The version number is increased when new features are added to the software, the level number is increased when minor corrections are needed in the existing version number.

1.8. Delem Limited warranty

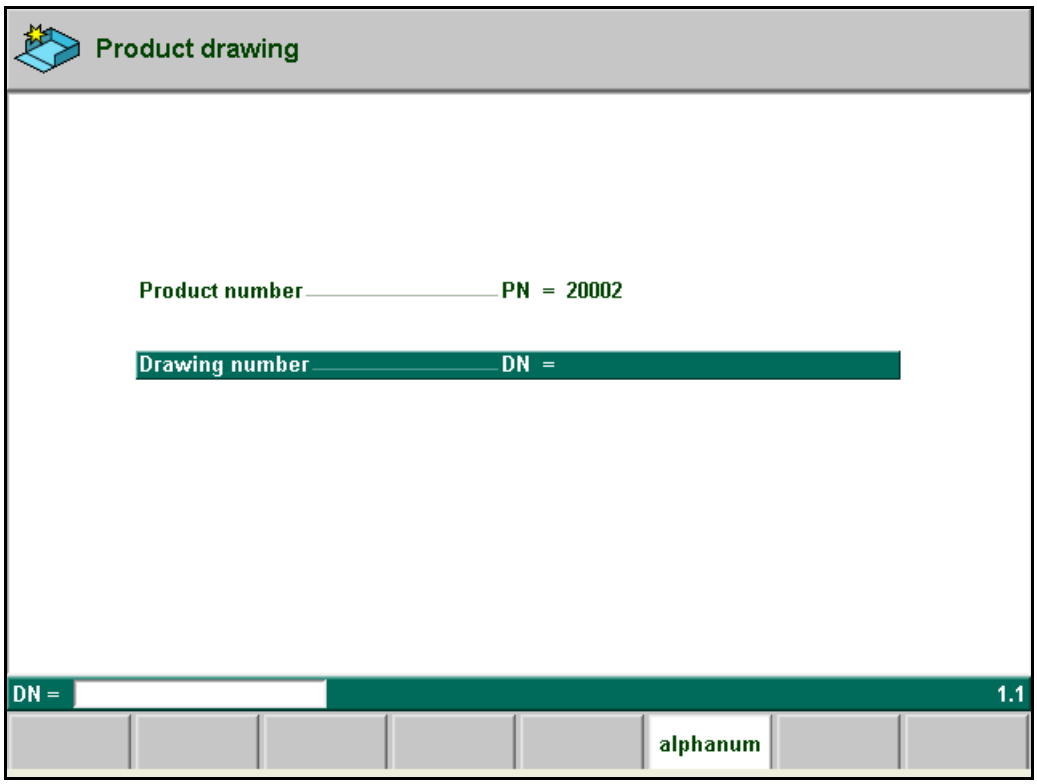
- This manual does not entitle you to any rights. Delem reserves the right to change this manual without prior warning.
-

2. Product drawing/Product edit for 2-dimensional products

2.1. Introduction

1. To start a new product drawing, choose **product drawing** in the main menu.
2. To edit an existing product drawing, choose **product edit** in the main menu.

On this page you have to enter first the product number and then the drawing number. The drawing number may also contain alphanumeric characters, which can be entered with the help of the function key ‘alphanum’ (S6).



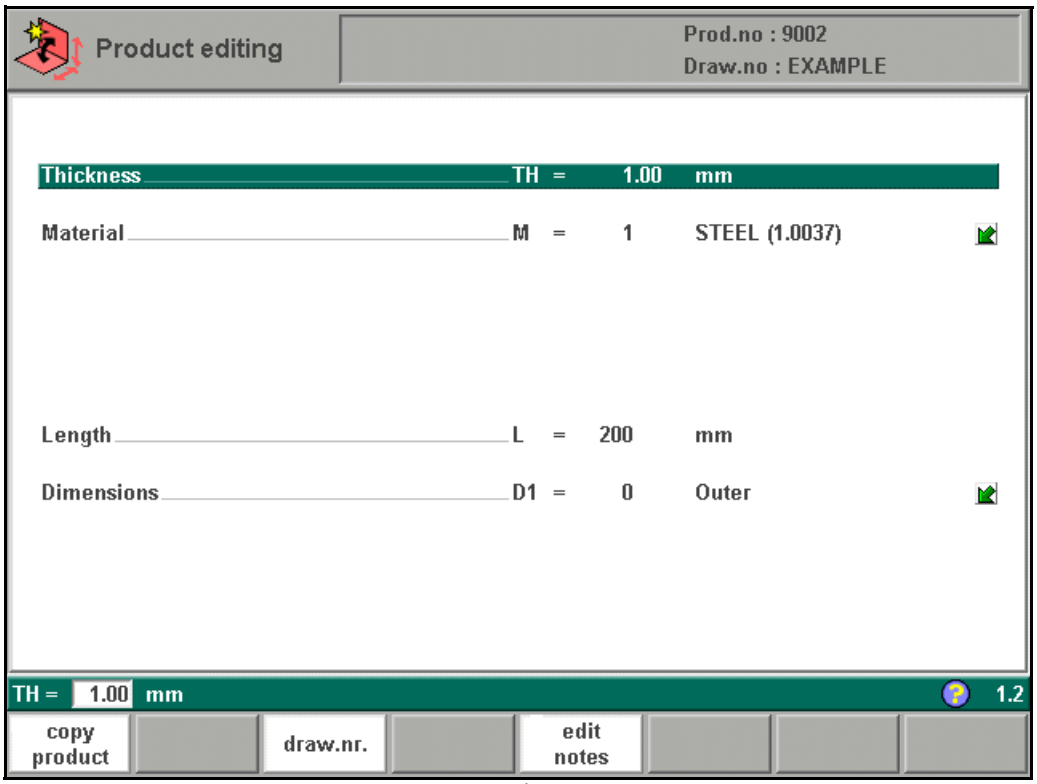
2.a

Product number . . . . . .PN
A unique number to identify a product program. The maximum length is 7 digits.

Drawing number. . . . . .DN
A name or description of this program. The maximum length is 20 characters.

If an existing product number is entered, a warning appears that this product already exists. You are asked whether to replace that existing product with the new product or not. If you choose ‘yes', the existing product is erased. If you choose ‘no', you must enter a new number. The "±" key prompts a "-" character and the "." key prompts a "/" character in the drawing number.

After finishing this input you have to enter specific product data. Then you can start drawing the product.
In the edit mode you also have the possibility to make a copy of the active product. A special function key ‘copy product’ appears.



2.b

Function keys

S1 **copy product** Copy the current product. When pressed, you must enter a new product number for the new copy.

S3 **draw. nr.** Go to the screen with product number and drawing number, in order to edit the drawing number.

Thickness . . . . . .TH=
Thickness of the material of the plate in millimeters (mm).

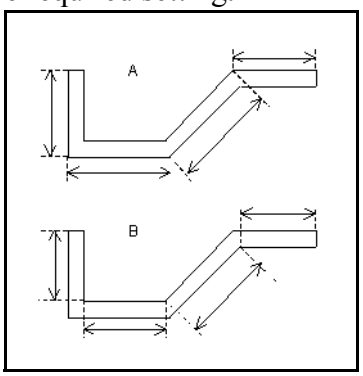
Material. . . . . . .M=
Selection of the material type. The control contains 4 preprogrammed materials. In total, 99 materials can be programmed on the control. See chapter 8 how to program materials.
Press the key ⌂ to select the required setting.

Length . . . . . .L=
The Z-length of the plate in millimeters (mm).

Dimensions . . . . . .D1=

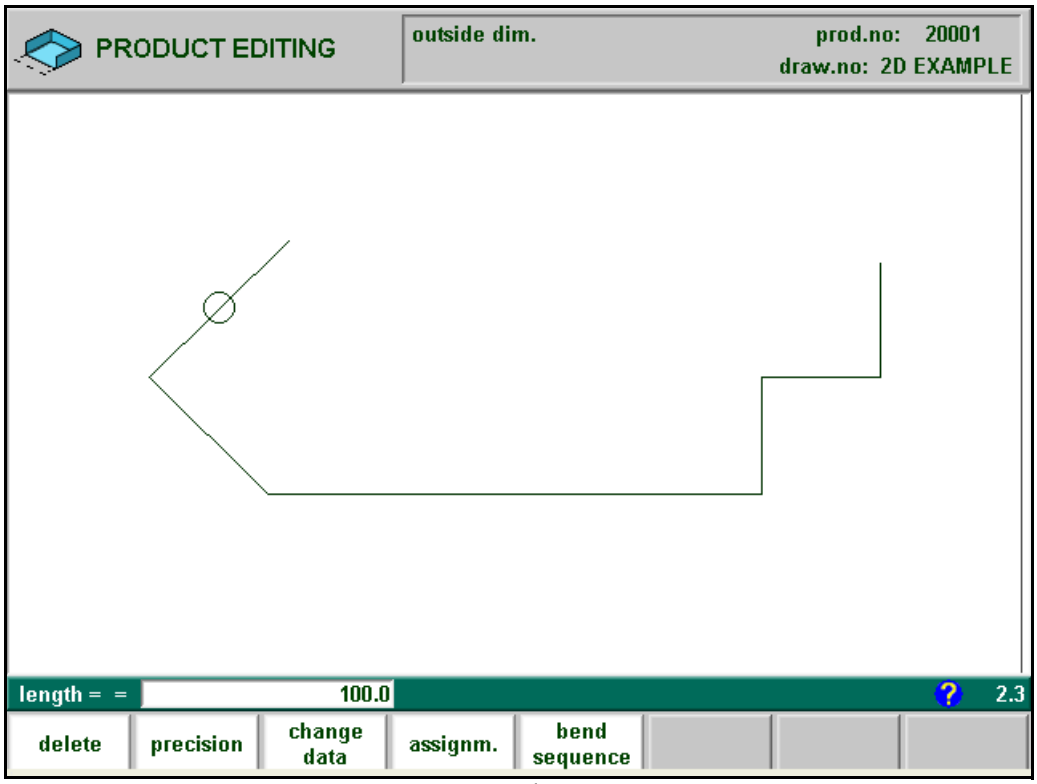
Determine the use of the outside(A) or the inside(B) dimensions in the product drawing. Figure 2.c gives the definition of both the dimensions.

Press the key ⌂ to select the required setting.



2.c

After entering the general product data the drawing screen appears. In the upper information row you will find the information about product number, drawing number and inside/outside dimensions selection.
You create now the bending profile of the product. First you enter the value of the basic length of the product. Then you enter the angle of the next side followed by the length of that side. This procedure continues until the product has the desired profile. A circle indicates the actual position. With the cursor control keys you move this circle to an other position (angle or length). During the drawing of the profile of the product the graphical software always displays the product at relative scale.



2.d

For creating the product drawing you have to enter the length of a line and the angle to bend or you can use the cursorkeys for angles of multiple 45 degrees.

In ‘product drawing’ or ‘product edit’ of a 2D product you can program up to a maximal of 25 bendings per product (graphical programming).

Function keys:

S1	**insert / delete**	Delete an angle/line or insert an angle, depending on the drawing cursor position (section 2.2).
S2	**precision**	To define selected line segment, with round cursor, for high precision or if it is to be a "closing" dimension (section 2.3).
	bumping	Bumping: when the cursor is on an angle you can create an angle with a big radius (section 2.4).
S3	**change data**	To page with product data.
S4	**assign-ments**	To select assignments for bending sequence computations (section 2.5).
S5	**bend sequence**	Program bend sequence (section 2.6).
END		Return to main menu

These functions will be explained in the sections as indicated in the abovementioned overview.

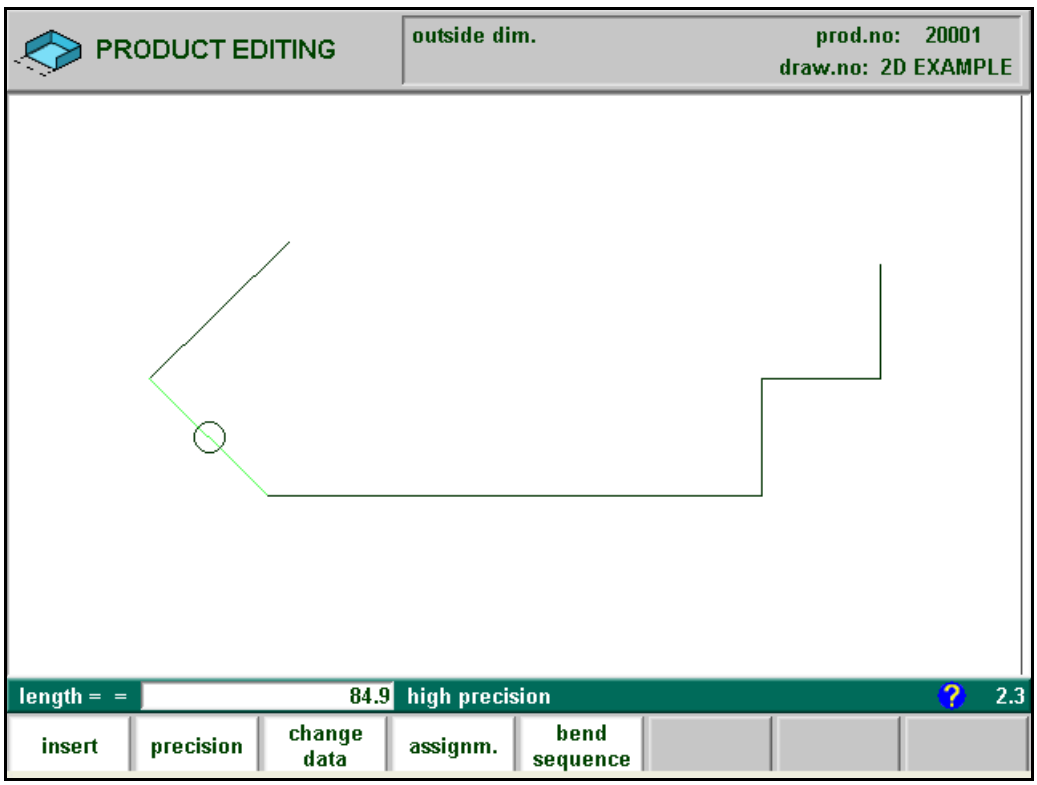
2.2. Delete an angle/line or insert an angle

The function of softkey (S1) depends on the position of the drawing cursor.

- If the cursor is within a line segment, it is possible to insert a new angle to bend, in combination with the enter key.
- If the cursor is positioned on a bend, it is possible to delete that bend.
- If the cursor is at an end line of the product, the line can be deleted.

2.3. Precision selection

When the drawing cursor (small circle) is on a line segment, with S2 high precision or closing dimension can be selected. With S2 these functions will be toggled giving 3 possibilities (high precision - closing dimension - normal situation).

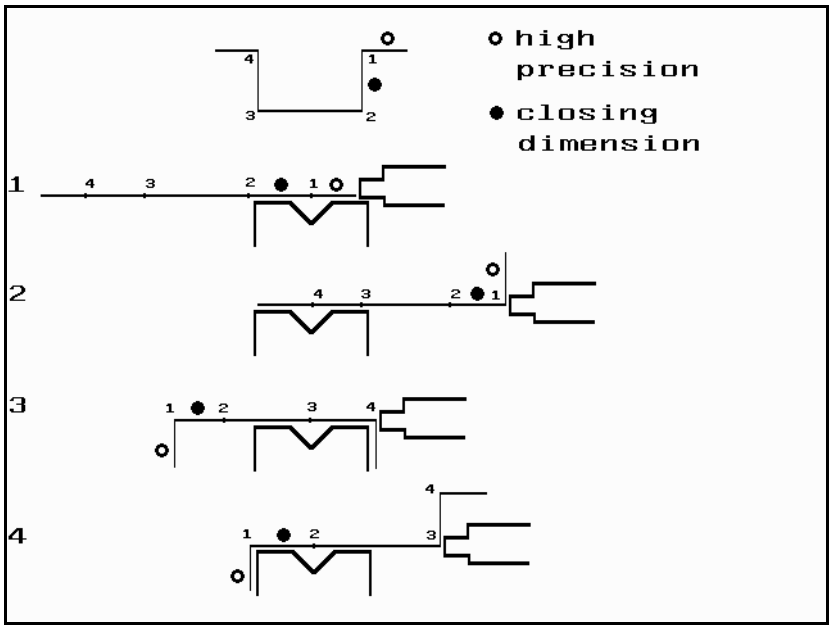


2.e

- High precision:
At bend sequence computation the backgauge stop position will be chosen to get the highest possible precision of this line interval.

- Closing dimension:
At bend sequence computation the backgauge stop position will be chosen to get the resulting tolerances in this line interval.

Example:



2.f

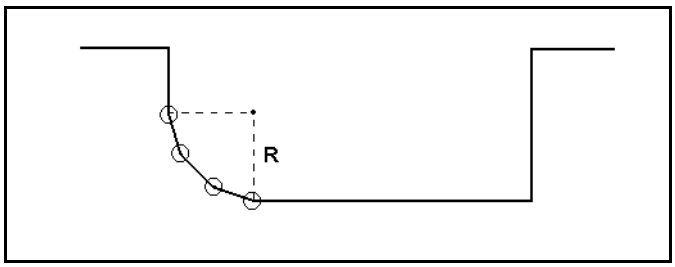
Line interval marked with the open circle should be, if possible, directly placed between back stop and the centre of the die.

Notes:

Specifying line intervals with high precision and closing dimensions may result in longer production time.
Also it will have priority over the "front extend ratio", if that is set to "comply if possible". See section 2.5.

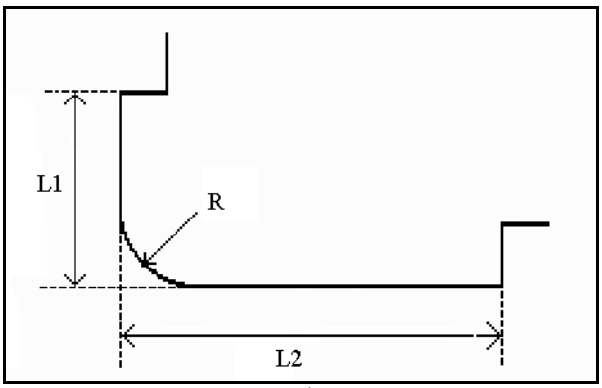
2.4. Large Radius: Bumping

To make a bend with a large radius, the control uses the bumping method. With this method, a large radius in a product is obtained by a series of slight bends in succession.



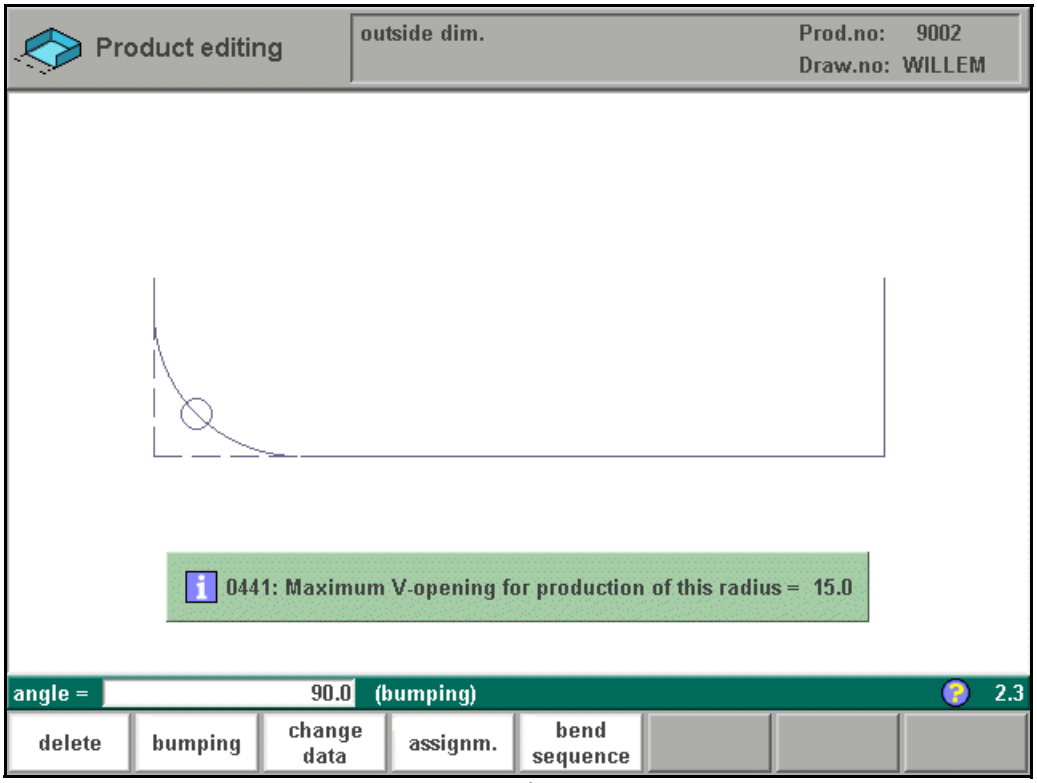
2.g

To program such a bend, you must program the following parameters:
radius = the desired radius
segments = number of segments
The number of bends in this radius is the number of segments plus 1.
The more segments you select, the more bendings will be used to create the programmed radius within a smaller tolerance. With a high number of segments you will need a smaller V-die opening to be able to bend in a proper way. Which value is acceptable as maximum for the V-opening of the die is calculated and displayed on the screen.
For the definition of the line lengths to be programmed in the part connected to a bump radius segment, see figure 2.h.



2.h

Lengths L1 and L2 must be equal or bigger than the radius R.
When you must program such a bend, first program a standard angle with adjacent sides. Then put the cursor back on the angle and press the function key ‘bumping’ (S2). You will be prompted to program the radius and the number of segments. After programming these parameters the radius is drawn in the product and the maximal V-opening which can be used is displayed on the screen. This is shown in figure 2.i.



2.i

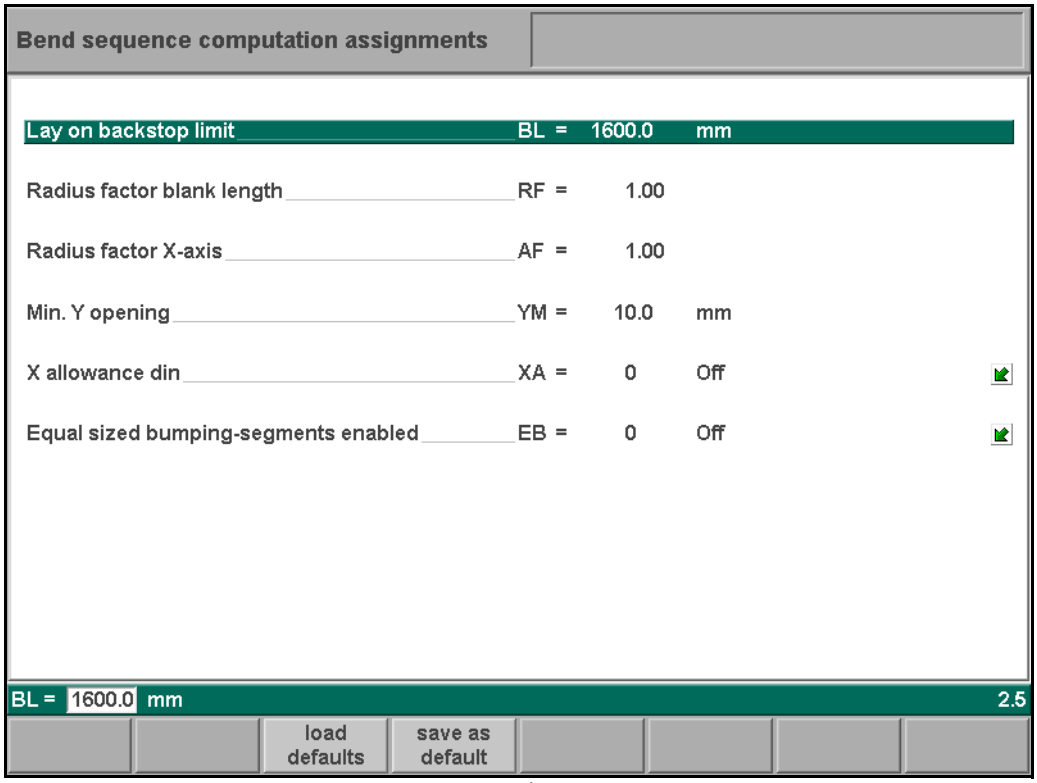
Specification items:

Radius input:
min. value = 0.1 mm
max. value = 2500.0 mm

In the ‘assignments’ menu, it is possible to modify the way the radius bend is divided into segments. See section 2.5 for more information.

2.5. Assignments

Pressing ‘Assignments’ results in the parameter pages.
If Variants programming is active (indication 'Variants On' in the title bar), a warning is given that corrections will be lost. Press 'Yes' to proceed with the assignments or 'No' to go back to the edit screen.
See section 2.8 for more information about variants programming.
Some of these parameters are machine related, axis speeds a.o. and some are related to handling possibilities and turn times.



2.j

Function keys:

S1 **previous page**

S2 **next page**

S3 **load defaults** To load default assignment settings. It is possible to determine a set of assignments which have the most optimal values to work within your surrounding. This set can be stored into the internal memory by pressing softkey S4 (save as default). While programming another product you can recall this previous fixed set by loading the values via softkey S3.

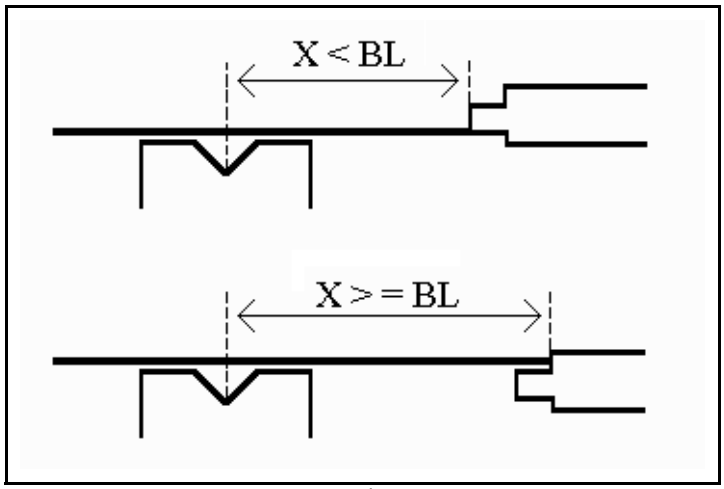
S4 **save as defaults** Save as default assignments settings

END **end** To go back to the drawing.

• Parameter explanation

Lay-on backstop limit . . . . . .BL=

This parameter (mm) is usefull in case the pressbrake has been equiped with back-gaugefingers on a moving R-axis, having a so-called “lay-on” construction.
When the length of the plate at the backside of the machine is greater than this limit, the X-axis and R-axis positions will be corrected automatically so the plate will rest on the backgauge finger. (0 - 3200.0 mm)
This is only possible if an automatic R-axis is enabled.



2.k

After postprocessing, the controller calculates the developed length of your product and the bend allowance.
Important for the developed length calculation and the bend allowance is the inner radius of the bends.
For each of these computations a correction factor can be programmed (RF and AF).

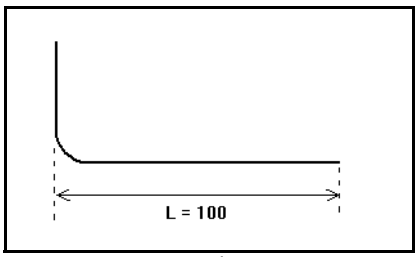
Radius factor blank length . . . . . .RF=

The computed inner radius is multiplied by this factor to correct the total developed length of the product. The initial value of RF is 1.

Radius factor X-axis position . . . . . .AF=

The computed inner radius is multiplied by this factor to correct the X-axis position in order to have a correct product dimension after each specific bend (bend allowance).
This factor can be checked by making a product with just one bend and a certain product dimension e.g. 100 mm (outer dimension). See figure 2.s.

The controller computes the X-axis position necessary to obtain L=100 as shown in figure 2.s. The accuracy of the length L is dependent on the material parameters like thickness, strength and kind of material. In order to have a correction possibility with the radius factor AF you can optimise this computation.
RF and AF do not influence each other. It is recommended that first you optimise the AF factor for your product and thereafter find the correct value for the developed length correction RF.



2.1

Minimum Y-axis opening. . . . . .YM=

During postprocessing of the programmed product, the control always computes an optimal opening of the pressbeam to handle your product. Here you can program a minimum required opening. The programmed value is the distance above the speed change point MUTE.

X-allowance DIN. . . . . .XA =

For the calculation of the developed length and bend-allowance during the postprocessing of a graphical 2D product, a Delem formula is used. It is also possible to select the standard DIN-formula (DIN6935).
0 = Delem formula (OFF)
1 = DIN formula (ON)

Default setting is zero for the Delem formula. Press the key ⌂ to select the required setting.

Equal sized bumping segments enabled . . . . . .EB=

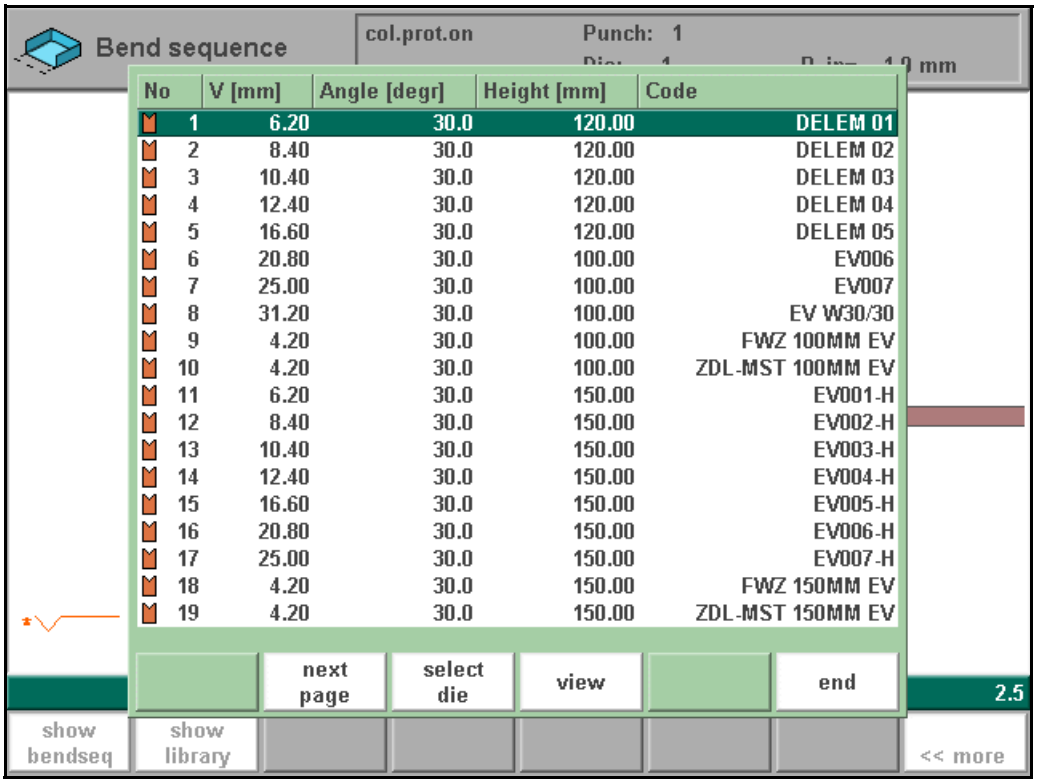
When a product has a radius bend, the segment size is computed from the number of segments, which has been defined by the user. Standard the first and last segment are calculated half the size of the mid segments to obtain a better result. However, it can be a problem selecting a die suitable to bend these small segments. Therefore the control can calculate an equal size for all segments. This can be defined with this parameter.
0 = disabled (no equal sizes)
1 = enabled (equal sizes)

Press the key ⌂ to select the required setting.
When this parameter is set to 1, all segments will have an equal size.
If it is set to 0 the calculation is as before, including half size segments. If in this case a problem with the size of the V die is detected in the bend sequence menu, the user is asked whether or not to select a re-calculation with equal size segments.

2.6. Bend sequence

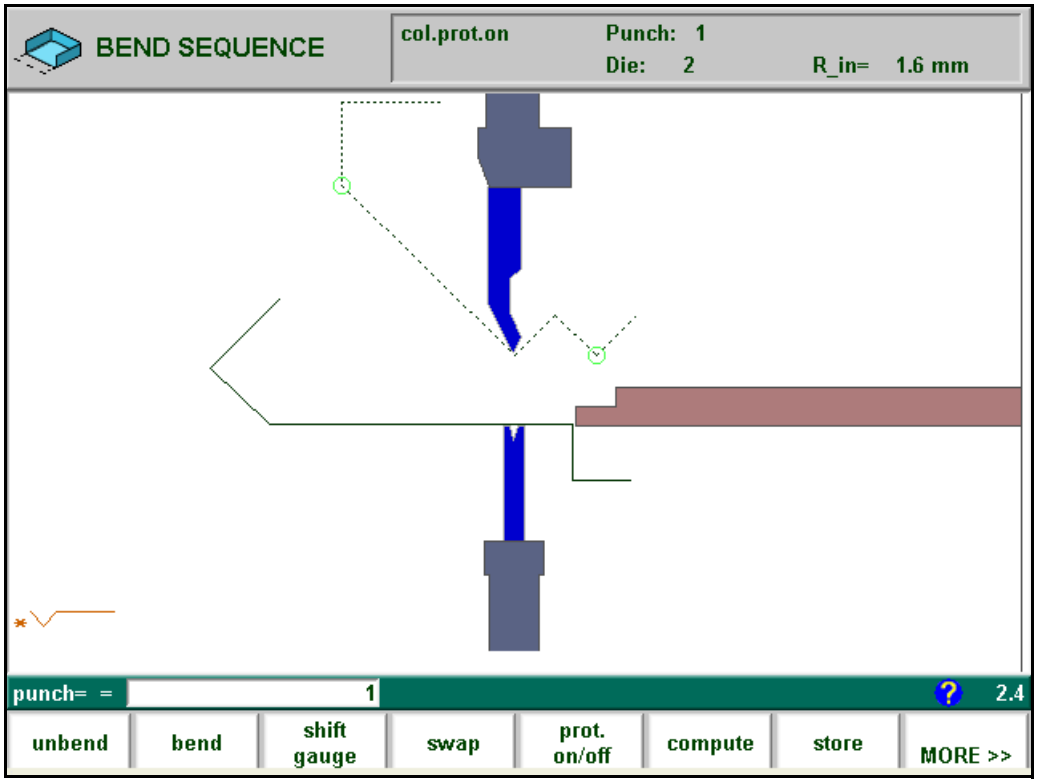
After completion of your drawing, with function key S5 you select the "bend sequence" mode. You will have to enter the number of the machine lowerside, the die, the machine upperside and the punch first. This number corresponds with the number in the respective tool libraries (menu number 7, 8, 9 and 10).
If the number entered is not known, the controller prompts the message "not programmed". You have to program the machine parts and tools before you can compute the bend sequence. Programming can be done via the respective selections in the program menu. The several tools as available in the respective tool libraries can be shown on the screen with S2 (show library).

This selection will give an overview of the tools including the main properties of each tool.



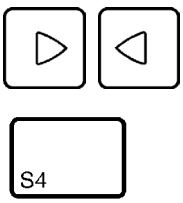
2.m

In your machine and tool library you can program 1 machine uppersides, 1 machine undersides (tables), 30 different dies and 30 different punches. One of each must be selected, and can also be changed during the bending sequence determination. These selections are prompted in the lower left corner of the screen. Within the bend sequence screen, the softkey ‘show library’ will remain available to get a graphical overview of the tool library as shown above.
After specifying machine part and tools the product and machine will be drawn on the screen.



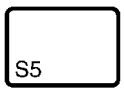
2.n

The product, as it was drawn, is placed directly under the punch, at one of the last bending positions possible.
The shape of your product before this last bend is placed on top of the die. A bigger circle on an angle indicates that this bending is also possible without collision.
A complete overview of available softkeys in this screen is given on page 2.25. The function key 'MORE>>' serves to switch between the primary and secondary row with function keys.



With these keys you can select any of the other bendings which you prefer to be the last bending. The bends possible are indicated with the round cursor.

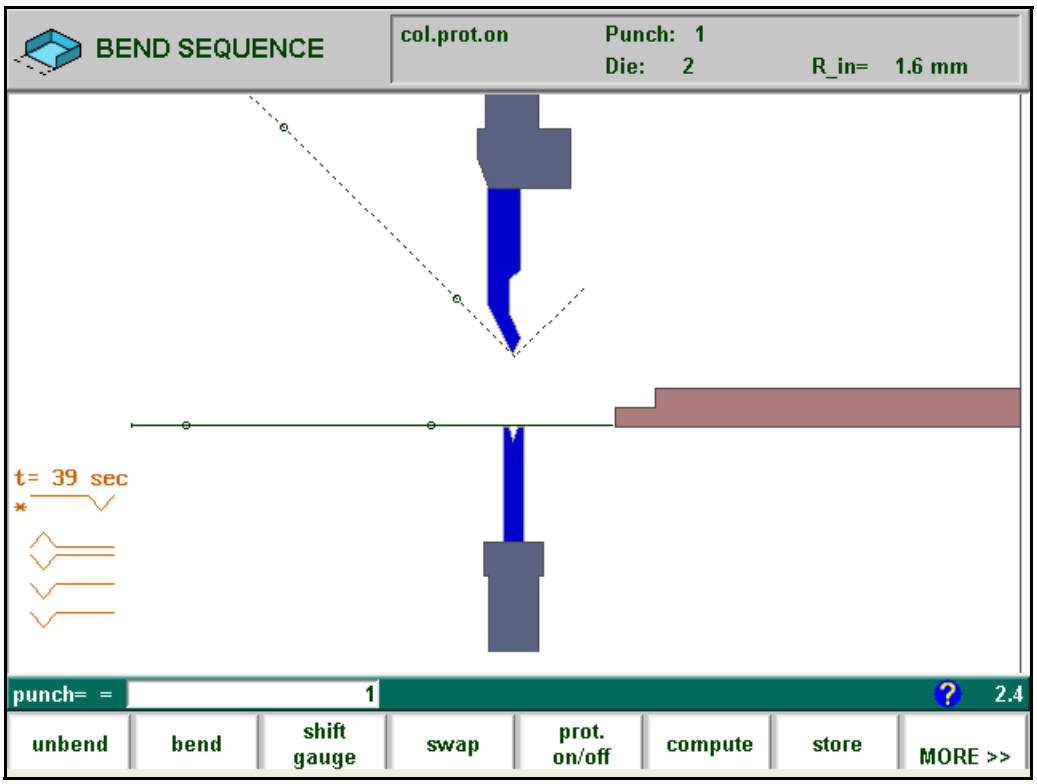
If the product has collision with the tools or machine it will be indicated via a warning message on the screen.



With this function key you can switch the collision protection check on or off.
This is indicated at the top of the screen. When you have selected collision "off" you can also select other bends which will give collision with the tools or machine parts.
The inner radius resulting after this bend is displayed at the top of the screen.

To enlarge or reduce the complete drawing.

After pressing "unbend" the position of the bending is indicated by a small circle, see figure 2.w.



2.o

Note:

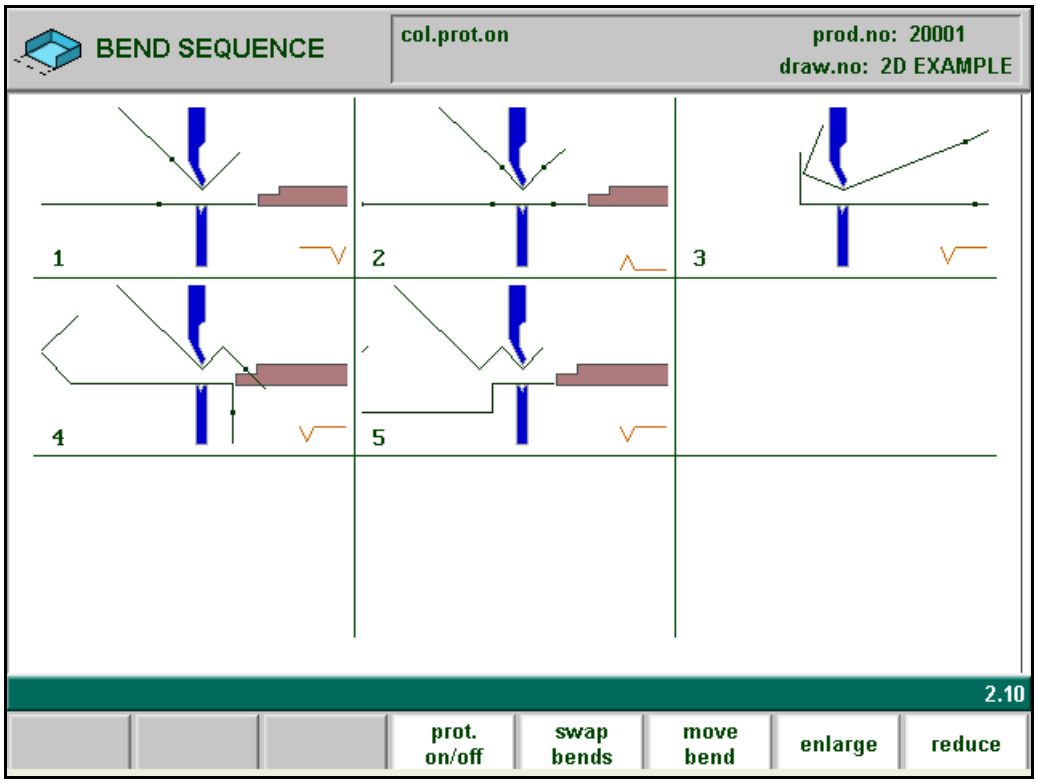
A big radius (bumping) is shown during bend sequence computation but is treated as a single bending.
The CNC-data of the bendings necessary to produce the radius are computed during postprocessing.

S7

When a bendsequence has been determined, a CNC program can be calculated and stored. This process is called postprocessing. When postprocessing is finished, the ‘blank length’ of the product is displayed.

S1

(second row) Open a graphical overview of the bend sequence, in order to have a visual check.



2.p

The images in this graphical overview can be zoomed-in or zoomed-out with the cursor- up and cursor- down key. The function keys S7 and S8 enlarge or reduce the number of images that is displayed on the screen at once (minimal 4 and maximal 25).

Function keys:

S5

Swap bends Let two bends exchange place in the bend sequence.

S6

Move bend Move a certain bend to another place in the bend sequence.

Function keys:

S7 **Enlarge** Enlarge all bend pictures, thereby decreasing the number of bends shown on the screen.

S8 **Reduce** Shrink the bend pictures, thereby increasing the number of bends shown on the screen.

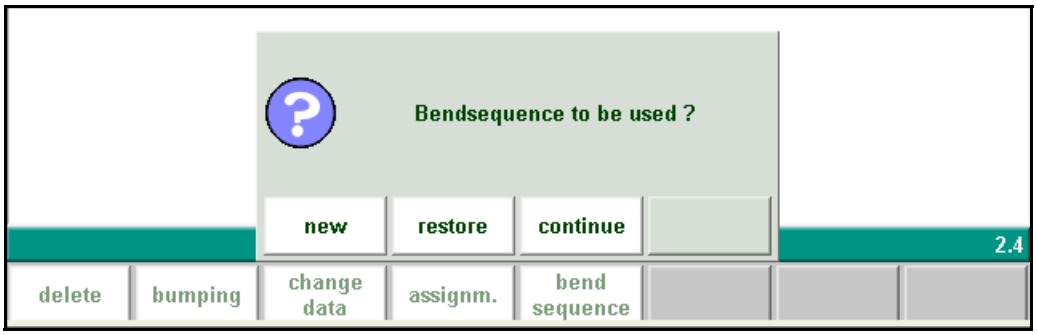
Move bend
In the graphical overview of the bendsequence, it is possible to change the order of bends simply by moving a bend to another place. Press the button 'Move bend' and the number of the first bend is highlighted. Use the arrow keys to move this cursor to the bend that must be moved. When the correct bend number is highlighted, press the enter key to select this bend. Now use the arrow key to move the bend to the right place in the sequence. If the bend is on the correct place, press 'enter' to confirm.

Swap Bends
With this command, two bends can change place in the bend sequence. Press the 'swap bends' button. Move the cursor to one of the required bends and press the 'swap' button or press enter. Then move the cursor to the bend with which it must be swapped and press the enter key or the 'swap' button. Now the bends have been swapped. If for any reason the action must be cancelled, press the softkey 'abort swap' during the procedure.

• Restoring a bend sequence

A (partly) determined bend sequence is not automatically reset after leaving the bend sequence menu.
After you re-enter the bend sequence menu it is possible to continue with the existing bend sequence.

If you re-enter the bend sequence menu you have the following options:



2.q

'new' start new bend sequence, the existing bend sequence is reset

‘continue’	continue with current bend sequence, the existing (partial) bend sequence is restored and shown on the screen

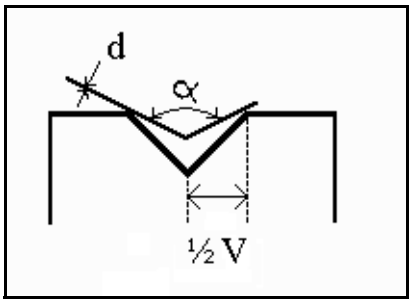
If you have loaded a drawing from which a postprocessed bending program already exists, there is a third option when you enter the bend sequence menu:

‘restore’	restore bend sequence from postprocessed program, the bend sequence is restored from this postprocessed bending program and shown on the screen

If you have drawn a new product in the menu "Product drawing" and you enter the bend sequence menu for the first time, there is no existing bend sequence and postprocessed program yet, these previously mentioned selections are not shown.

• Minimum bending length

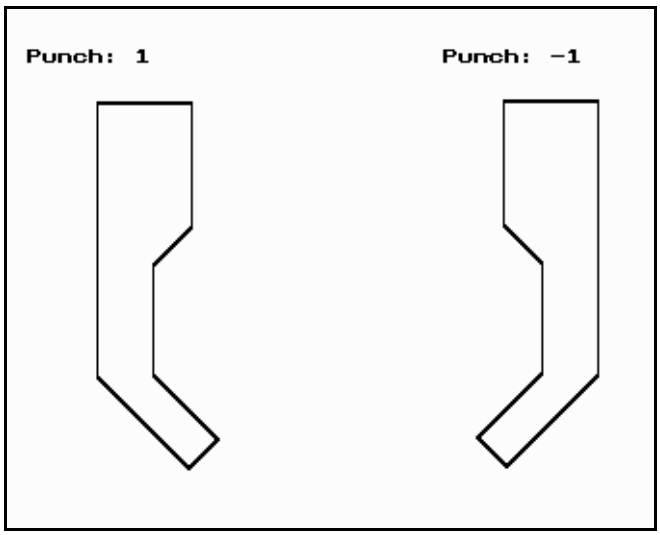
The minimum possible bending length will be calculated from the V-opening parameter (½ V), but also depends on the angle to bend and the thickness of the material to be bend, see figure 2.z. If the programmed value for the wanted product does not correspond with the minimum possible length, you will get a warning message on the screen.



2.r

• Machine/Tool selection

At each bend you can select one of the tools or machine shapes which are present in your library. The displayed number is the number of the tool and machine part which is presently selected.
Entering a new number will select another tool or machine part which will be drawn directly on the screen.
With the "enter" key you can toggle between punch-, die-, machine upper- and lower part number.
It is also possible to turn the punch and die with respect to the machine shapes. To achieve this, the tool number must be programmed negative.

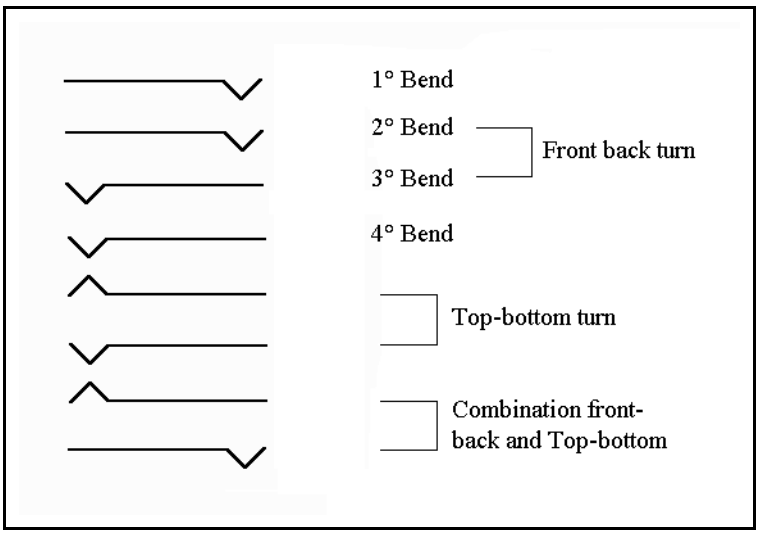


2.s

In order to get a graphical overview of the available tools while you are in the bend sequence menu, press the function key 'show library' (S2). If pressed, an overview appears of available tools.
Which type of tools is shown depends on which type was prompted in the bend sequence screen: punch, die, machine upper- or lowerside.

- **Turn indication**

In the lower left corner a bend turn indication is displayed at each bend of bend sequence.



2.t

The asterisk indicates which bend is presently displayed.

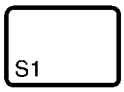
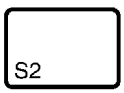
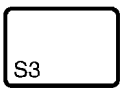
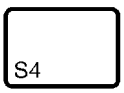
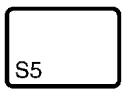
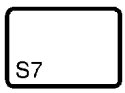
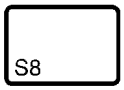
- **Screen data**

On top of the screen some general important data is displayed.

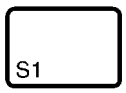
Col.prot.on	Collision and backgauge checks on or off
Punch: 1	Punch number selected
die: 1	Die number selected
R-in = 1.3	Inner radius of the bend to be obtained with selected die
Variants On	Variants programming enabled: the product has a valid CNC program that can be modified from the graphical menu.

• Function- and control keys summary

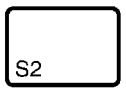
Function keys:

	**unbend/ next**	To unbend the graphical product
	**bend/previous**	To bend the graphical product
	**shift gauge**	Select backgauge position
	**swap**	Front to back turn of the product.
	**protection on/off**	To switch the collision detection with collision warning On or Off.
	**store**	Automatic computation of all axes positions (complete CNC program) and storage in the product library of the postprocessed product.
	**more>>**	The softkeys for this mode are divided in two rows. Press this key to switch from one row to another.
	**end**	To go back to the drawing.

Function keys secondary row:



show bend-seq. To show a mosaic screen with a step-by-step graphical overview of the bendsequence.



show library Softkey to get a graphical overview of the tool library.

Control keys:

Zoom enlargement

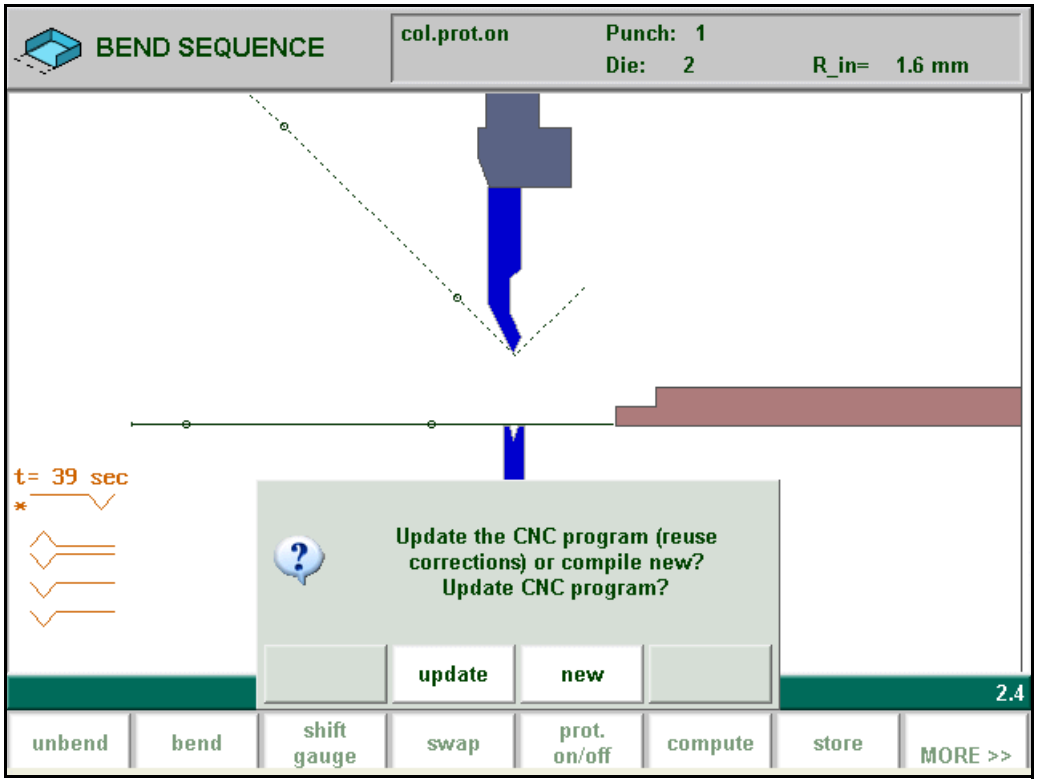
Zoom reducement

- **Store**

After you have completed the bend sequence a CNC-program can be generated. This generated program will be stored in the controller program memory.

2.7. Variants programming

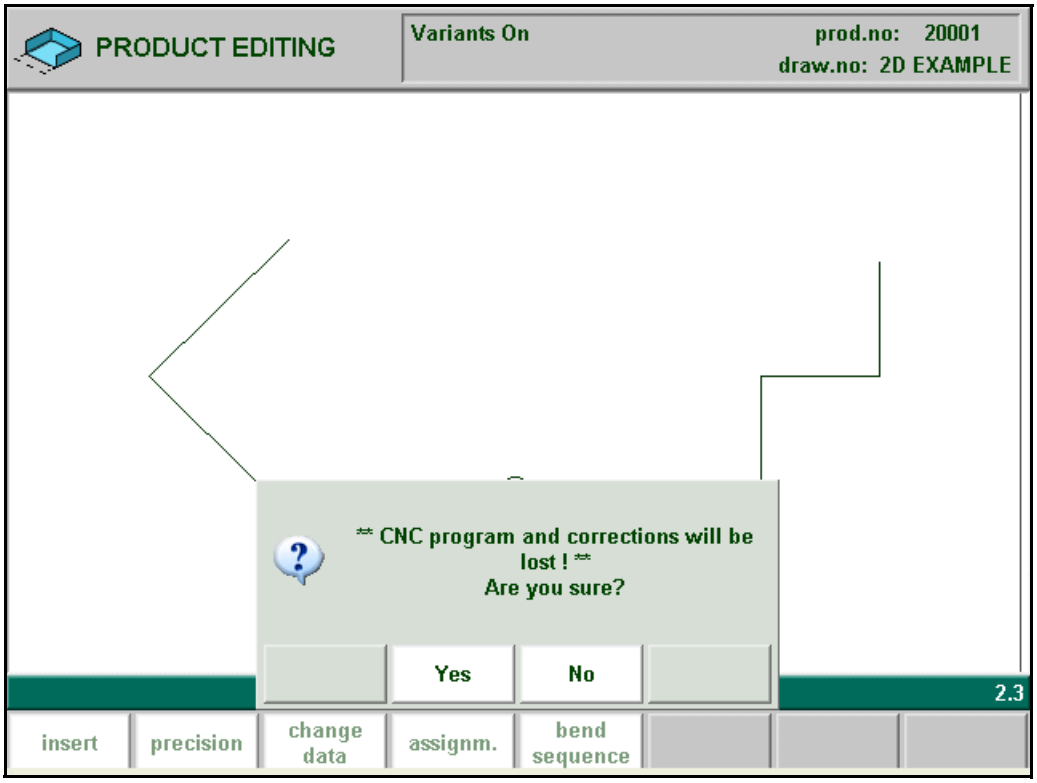
The term ‘Variants programming’ means that the operator can make modifications to an existing program in the graphic menu, without having to build a new CNC program from scratch. An existing program can be altered without loss of existing corrections and bendsequence. If such an existing program contains proper axis values and product positions then this information can stay intact, only the latest changes (angle, sheet length) are recalculated and entered into the program.
For instance, when a side length has been altered in the graphical bend overview and the ‘store’ command is given, then a question appears how to compute a CNC program.



2.u

When update (S4) is chosen, the existing CNC program is changed with regard to modified values (angle, length), but other values of bend parameters stay the same. The expression ‘re-use corrections’ refers to the corrections that may have been entered in one of the production modes. These production modes are explained in chapter 10.
When new (S5) is chosen, a new CNC program is calculated. Corrections in the existing CNC program are lost.

In order to keep the existing program, check that the indication ‘Variants On’ remains in the title bar. When you start an action or command that would seriously change the program, a warning is issued by the control:



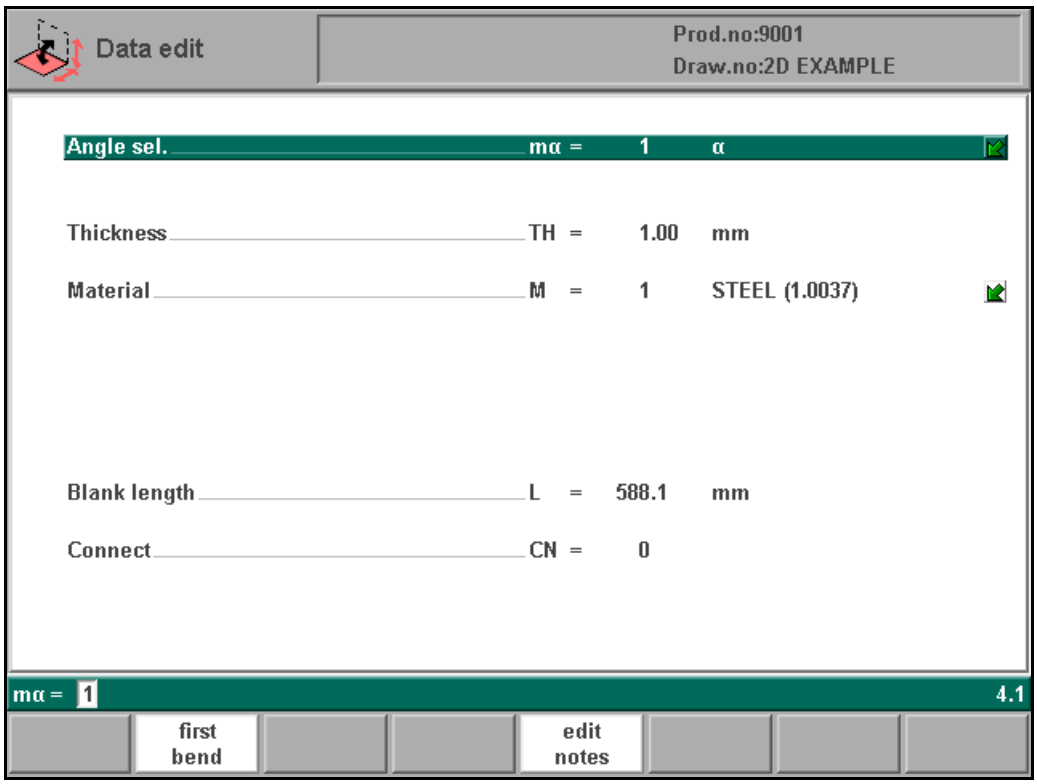
2.v

If you choose 'yes' (S4), the indication 'Variants On' will disappear from the title bar. This means that the next time when a new 'postprocess' command is started, a new CNC program is built and old corrections will be lost. When 'No' (S5) is chosen, the action is cancelled and 'Variants on' remains.
The message "CNC program and corrections will be lost" is not literally true. The CNC program is still present, but it cannot be updated anymore from the graphical menu. If the old CNC program must be stored, it is still possible to go to ‘data edit’ mode and store the program under a different product number.

3. Data preparation/Data edit

3.1. Introduction

To create a new CNC program, choose ‘Data preparation’ in the main menu.
To edit an existing CNC program, choose ‘Data edit’ in the main menu.
In both cases, a screen as shown below should appear. Programming and changing data is done in the same way for both modes.
To edit a program that is not currently loaded, choose ‘Product selection’ in the main menu. In the offered list, you have to select the wanted program number.

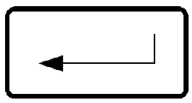


3.a

This page gives all data which are the same for every bending of the program (main data of program).

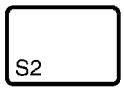
For placing the cursor at one of the parameters. The value entered for the selected parameter appears in the entering field, situated in the lower left corner.

The input can be cleared for entering new input data. Data has to be given to be able to continue programming.



After pressing the enter key the programmed value will be placed at the corresponding parameter.

Function keys:



first Bend The cursor jumps to the page with the first bend information.

END Finishing the data preparation and returning to the program menu.

• Parameters explanation

Angle selection . . . . . .mα

Selection of the programming mode for the Y-axis.
0 = absolute: program the absolute Y-axis position for a bend.
1 = α: program the angle to bend. The required Y-axis position is computed.
Depending on this parameter, either the parameter 'angle' or the parameter 'bend position' will appear.

Thickness . . . . . .TH

Thickness of the plate in millimetres.

Material. . . . . .M

Selection of one of the programmed materials, which are used to calculate the bending depths. The control contains 4 preprogrammed materials. In total, 99 materials can be programmed on the control. See the chapter about programming constants how to program materials.

Press the key ⍓ to select the required setting.

	E-MODULE (N/mm²)	TENSILE STRENGTH (N/mm²)
1 = Steel	210.000	400
2 = Aluminium	70.000	200
3 = Zinc	94.000	200
4 = Stainless steel	210.000	700

Blank length L

The required length of the original sheet from which the product is bent. If the program

has been processed from a 2D drawing, this value has been calculated.
This parameter is also required for 2D visualisation in a production mode (Automatic or Step).

Connect . . . . . .CN

The parameter "connect" is to have a possibility to connect certain programs to one another. This option can be used to produce 3-dimensional products. (See also the next page).

• Connecting CNC programs

With the parameter Connect it is possible to create a 3-dimensional product. The control automatically executes the bend sequences in the different directions in succession. You program the control as follows:

Instruction for 3-dimensional product programming (9 steps)

1 Create the product in one direction.
2 Create the product in the other direction.

There are now two bendprograms of one product in two directions. You connect these programs as follows:

3 Select the program with the bendsequence in the direction which you want to execute in the first place. You select the program of the product via menu 3, product selection.
4 Go to menu 2, data editing. Select the parameter CONNECT. Enter the program number of the product in the other direction.
5 Select the second program as in step 3. Repeat step 4. If you want to connect two programs, as in this example, you enter the program number of the first program. The cycle is closed.

When you want to execute more than two programs in succession (not necessarily to create a 3-dimensional product) the second program must refer to the third. The third program to the fourth and so on. The final program of the cycle must always refer to the first program.

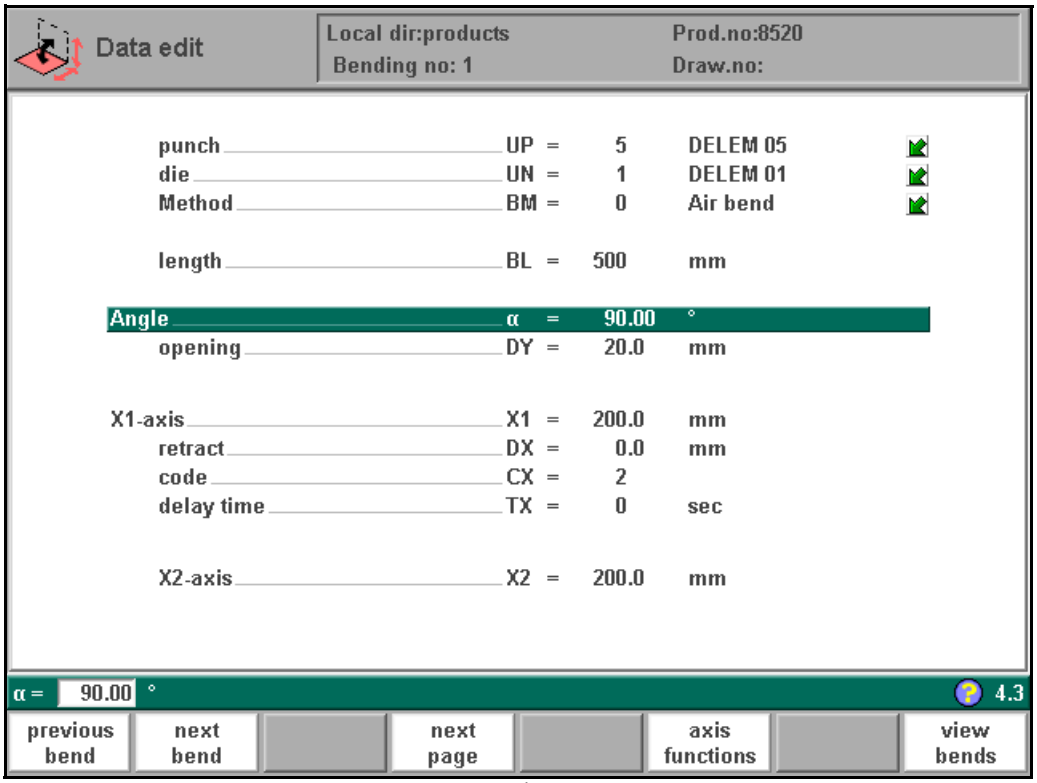
To produce products with connected programs the next four actions are necessary.

6 Select the first program
7 Select the Automatic mode
8 Program the amount of products you want to produce with the ‘stock count’ parameter.
9 Push the Start key.

When the first program has been finished the second program starts automatically. The program counter indicates the remaining amount of repeats.

3.2. Bend programming

The parameters of one bend are divided over 2 screen pages.



3.b

The bend number, product number and drawing number are displayed in the top row on the screen, the function of the "softkeys" in the bottom row.

Function keys:

S1 **prev. bend** select previous bend

S2 **next bend** select next bend

S3 **show library** Open a window with an overview of the tool library. From here, a tool can be selected. Only available if the program cursor is positioned on a tool.

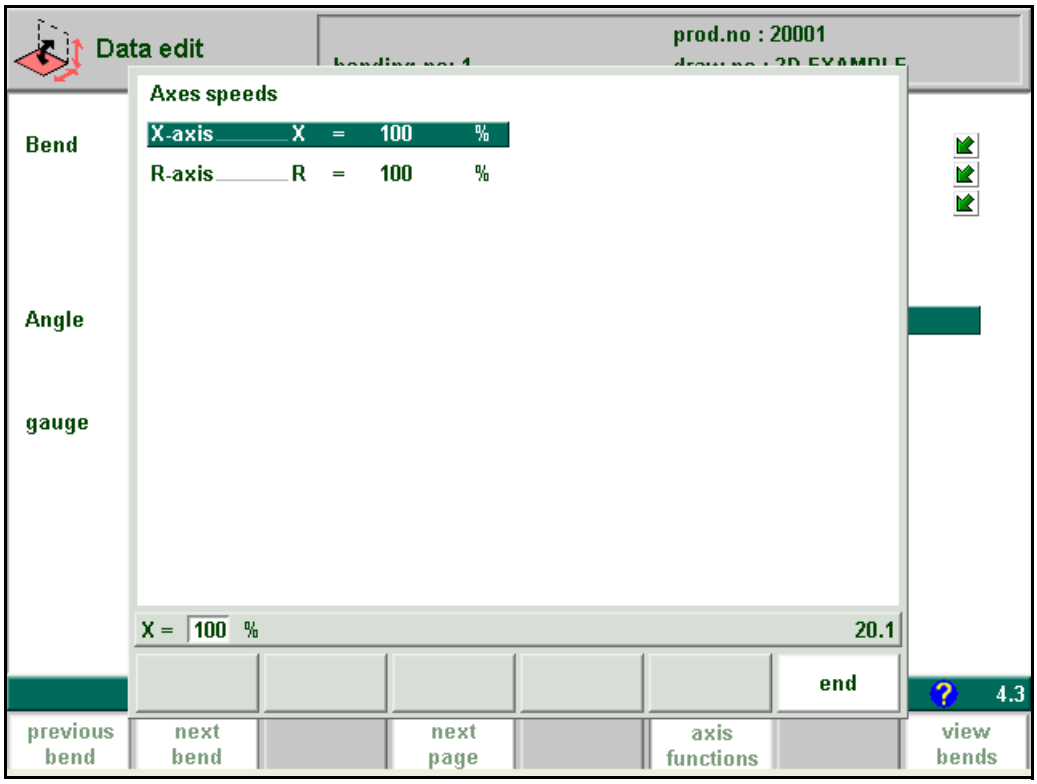
S4 **next page** next page, to select the second page of the parameters to be programmed for this bending.

Function keys:

S6	**axis functions**	axis functions, to change speed and retract distance of available axes in the current bend. This function is machine-dependant.
S8	**all bends**	to the listing of all bends of this program (only possible in data-editing).
END	**END**	To end the data prep./edit mode and return to the main menu. In the dialogue that appears, enter a new product number or confirm the existing number.

• Axis functions

With this command, speed and retraction of the available axes on the machine can be programmed. This function is machine-dependant. The function is activated by pressing S6 in the 'Data edit' screen:



3.c

A pop-up window appears, in which all programmable dimensions for the axes are shown. Which axes are shown here depends on the machine configuration.
Press the 'END' button to leave this window.

Axis speeds
Speed of the selected axis in the current bend. Speed can be programmed in a percentage of the

maximum possible speed.

Retract
Retract distance of the selected axis in the current bend. The distance is always programmed in millimeters. If the retract distance of the X-axis is altered in this window, the Retract parameter of the X-axis is automatically adjusted.

• Parameter explanation

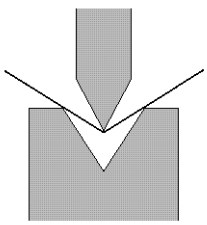
Punch. . . . . .UP
Number of punch in library.

Die . . . . . .UN
Number of die in library.

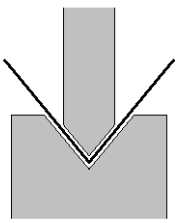
Program the required tool numbers or press the key ⌂ to select a tool from the list. Use the function key 'Show library' to check the available tools.

Bendmethod . . . . . .BM
Select the required bending method. The control supports 4 methods:
Air bend
Bottoming
Flattening
Flattening & bottoming.
Press the key ⌂ to select the required setting.

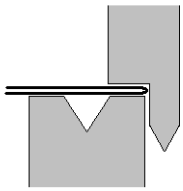
Bend methods:



air bend — The sheet is bent to the programmed angle by bringing the punch to the required depth. The control calculates the required Y-axis position to obtain the programmed angle.

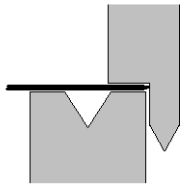


bottoming — The sheet is bent by squeezing the sheet between the punch and the die. The control assumes the bottom of the die as required Y-axis position.



flattening

The sheet is folded in two. This is possible after the sheet has been bent into a sharp angle in a previous bend. The control calculates the precise Y-axis position for this action: the surface of the die plus twice the sheet thickness.
Y-axis positioning can be adjusted by programming a 'flattening offset'.



flattening and bottoming

The same as flattening, but now the control assumes the top of the die as required Y-axis position. The folded sheet is squeezed between punch and die.

Note 1:
The flattening bends are shown here with a special flattening punch, but this is not required.

Note 2:
When bottoming operation is selected, the end of bend position of the Y-axis beam is depending on the working tonnage. If however the force is sufficient for the beam to go to the calculated Y-axis end of bend position, the beam stroke will be limited by the position value.

Length . . . . . .BL
Length of plate between tools

Angle . . . . . .α
The required angle of this bend. This parameter only appears if angle programming is selected with the parameter 'Angle sel.' and the bend method is an air bend.

Bend position. . . . . .Y
The required Y-axis position for this bend. This parameter only appears if absolute programming is selected with the parameter 'Angle sel.' This parameter also appears if the bend method is bottoming and/or flattening.

Opening . . . . . .DY
This parameter results in a certain gap opening between the punch and the die after the bend. A positive value is the gap opening above Mute, a negative value below Mute. When you want to limit the handling time of the product you can program a small positive or a negative value.

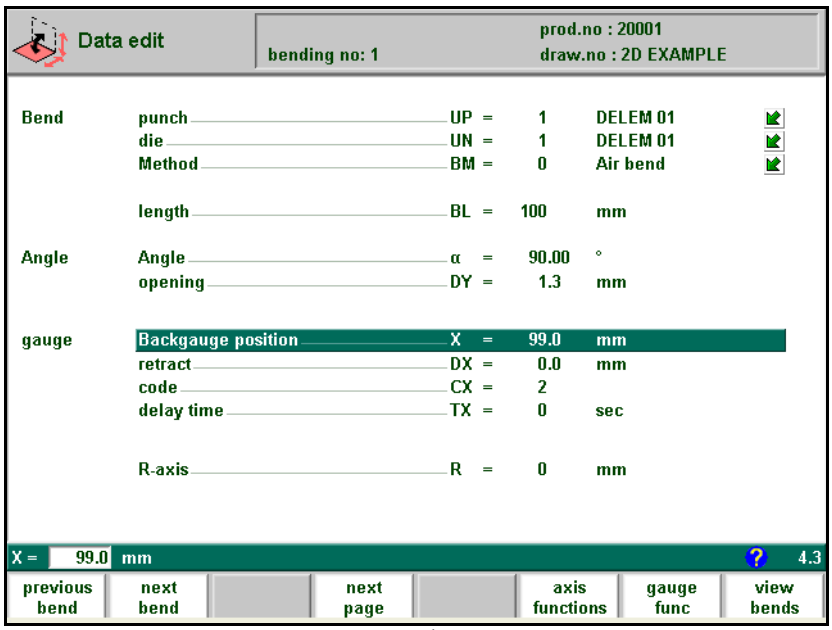
Mute. . . . . .M
Sequence point at which the Y-axis is switched from fast closing speed to pressing speed. The value programmed here is the distance of the mute point above the plate. By default, the mute value from the programmed die is used.
Whether or not this parameter is present depends on machine settings.

Backgauge position X

Backgauge position. When a negative sign is programmed this backgauge dimension is an incremental dimension. The incremental dimension is subtracted from the actual X-axis position. Therefore this parameter can also be used as a chaining measure.

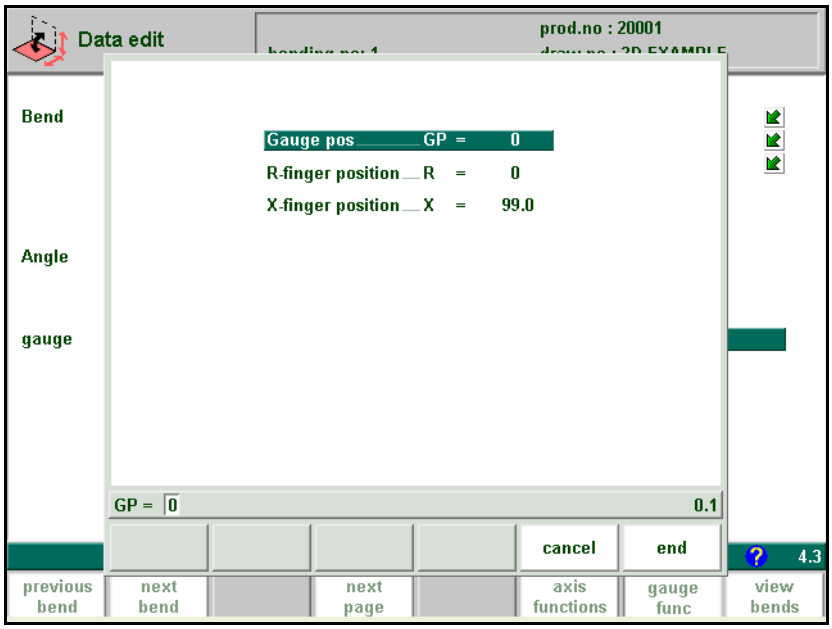
Only in case you have a configuration with a R-axis.

The function key (‘gauge func’) appears when you put the cursor bar on this X-axis parameter.



3.d

If you press ‘gauge func’ a window appears with three programmable parameters.

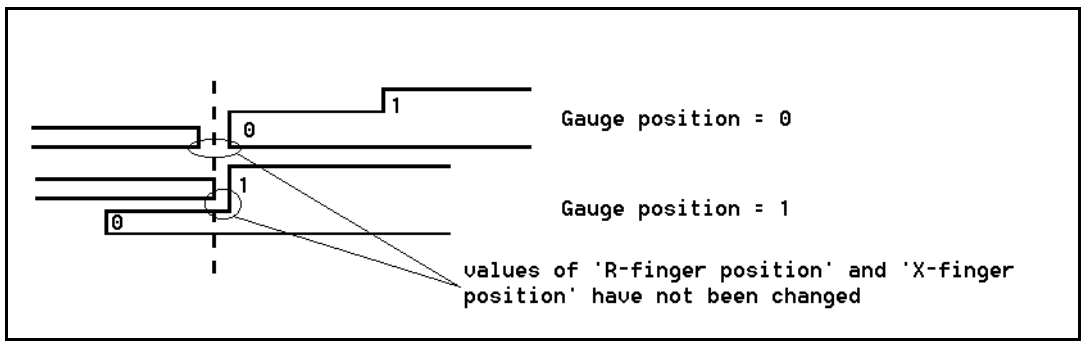


3.e

Gauge pos.. . . . . GP

With this function you can program another ‘gauge position’ for this specific bend. The

gauge position (parameter GP) you can program depends on the backgauge dimensions. Backgauge dimensions are programmed in the menu ‘program constants’.
Default gauge position = 0 (no lay on)
Other possibilities are:
gauge position = 1, lay on first level
gauge position = 2, lay on second level (only possible when number of gauge positions (GN) = 3)
gauge position = 3. lay on third level (only possible when number of gauge positions (GN) =4)
When you program another ‘Gauge position’ the backgauge position changes, but the gauge position for the plate does not change. This means that the next two parameters ‘R-finger position’ and ‘X-finger position’ remain the same. See also example below.



3.f

R-finger position . . . . . .R

Default value = the height (R direction) of the gauge position of the plate. This value can manually be adjusted in this window, if required.

X-finger position X

Default value = the value in X-direction of the gauge position for the plate. This value can manually be adjusted in this window, if required.
With ‘END’ you leave the window and the new values have been saved. If you press ‘cancel’ you leave the window without changes.
When you have changed the ‘Gauge position’ then the axes positions in X- and R-direction have also been changed. Therefore the parameters ‘backgauge position...X’ and ‘R-axis....R’ have been changed. The new values depend on the dimensions of the backgauge finger which you have programmed in the menu ‘program constants’. When you have changed the values of ‘R-finger position’ and ‘X-finger position’, then these changes are also included.

Retract . . . . . .DX

Retract distance of backgauge during the bending. The "backgauge retract" is started at the pinching point of the plate.

code CX

Programmable parameter which determines when the parameter values for the next bending will be active. The possibilities are:

0 = Bending number change (step change) at end of decompression (next bend parame-

ters active).

1 = Step change at muting position when the beam moves in opening direction.

2 = Step change at upper dead point.

3 = Step change at upper dead point without movement of any axis and the control goes to "stop".

4 = Step change if C-input signal becomes active, without movement of the beam. When you still have a beam movement there will be no retract function of the backgauge performed. See also code 5.

5 = Step change if C-input signal becomes active and the beam is in the upper dead point. Now you may move the beam and the retract function of the backgauge will be performed.

10= Step change at end of decompression with the Y-axis beam waiting at the pinching point until the backgauge is at the retract position.

11= Step change at mute position when the beam moves in the opening direction, with the Y-axis beam waiting at the pinching point until the backgauge is at the retract position.

12= Step change at upper dead point, with the Y-axis beam waiting at the pinching point until the backgauge is at the retract position.

13= Step change at upper dead point without movement of any axis. The Y-axis beam will wait at the sheet clamp point until the backgauge is at the retract position. After the bending with this code, the control goes to "stop".

15= Step change if C-input signal becomes active and the beam is in the upper dead point. The beam may be moved but not necessarily. The Y-axis beam will wait at the sheet clamp point until the backgauge is at the retract position.

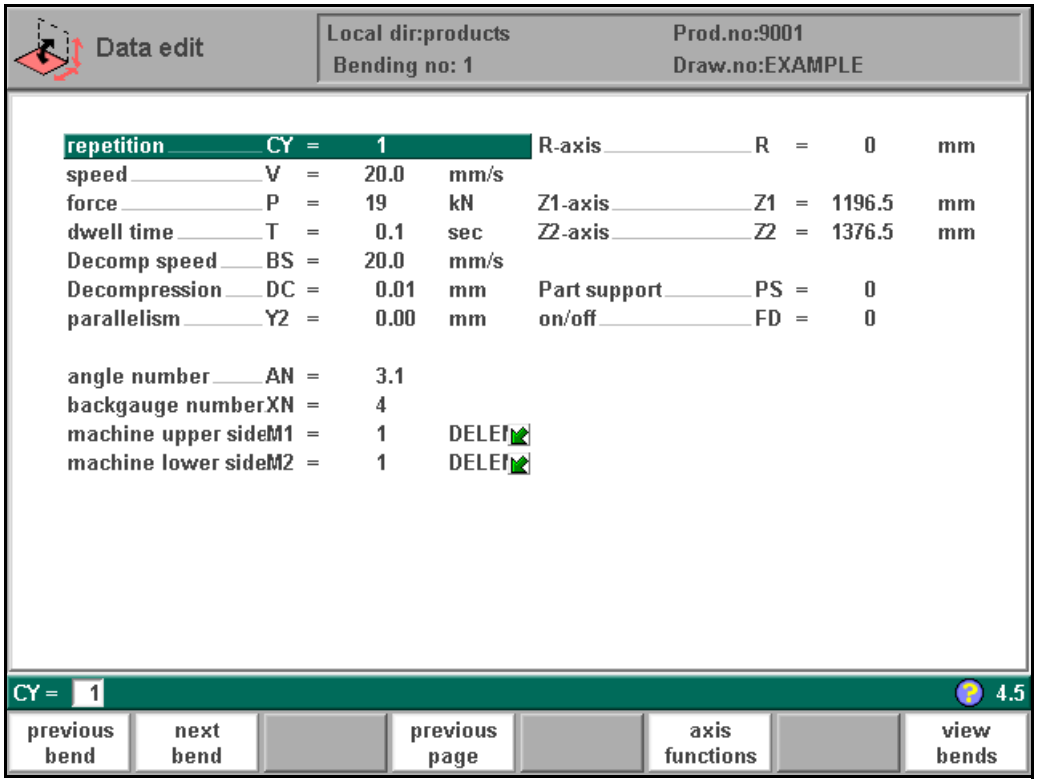
The not mentioned CX-values have no meaning.

Delay time . . . . . .TX

Programmable delay time before step change (0-30sec)

• Parameter explication (second page)

Auxiliary functions of the bending can be programmed on this page.



3.g

Repetition .....CY

0 = bending is skipped.
1 through 99 = the number of times this bending will be repeated.

Speed V

Working speed (pressing speed). Initially, the value for this parameter is copied from the parameter ‘pressing speed’ in the programming constants menu.

Force .....P

Max. adjusted tonnage during pressing (auto calculated).

Dwell time .....T

Holding time of punch at bending point.

Decomp speed .....BS

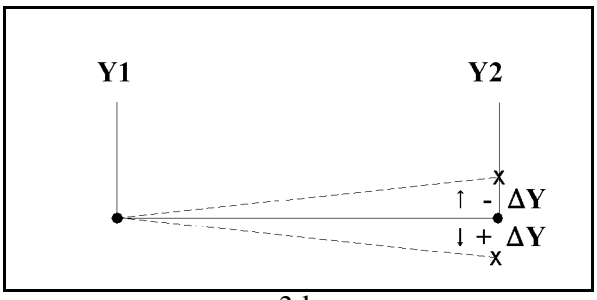
The decompression speed is the programmable speed which is active during the programmed decompression stroke.

Decompression .....DC

Decompression stroke after bending to release the working pressure

Parallelism.....Y2

Difference of left- and right hand side cylinder (Y1 and Y2). When positive, right hand side lower. When negative, right hand side higher. The programmed value is active below the clamping point.



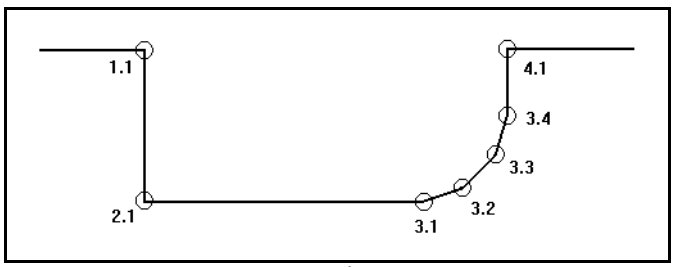
3.h

Auxiliary axis . . . . . .R/Z/Aux.

If you have one or more auxiliary axes (for instance a R-axis, Z-axis or part support) the parameters of these axes appear here.

Angle number . . . . . .AN

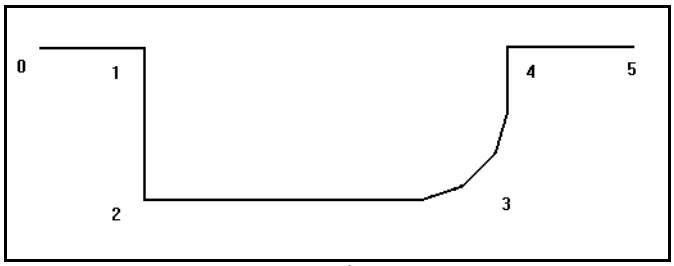
The rank number of this angle, when counted from left to right through the product profile (figure 4.j). The digit before the dot describes this rank number, the digit after the dot descibes the rank number of this bend in case of a bumping sequence.
In case of a normal angle, the bumping number should always be 1. In the case of a radius bend, all bends in the bumping sequence should have the same angle number but a different bumping number. Programming a radius bend in this manner will allow for bumping correction in the production mode.



3.i

Backgauge number . . . . . .XN

A backgauge number refers to a certain spot of the product, that can be placed against the backgauge during bending. Such a spot is either a product end or an angle. The backgauge numbers in the product are counted from left to right, starting with 0 (see figure 4.k). For each bend you must program here which spot must be placed against the backstop. If the plate must rest on the backgauge finger, add 100 to the XN value. For each next lay-on position, add another 100 to the backgauge value (200, 300, etc.).



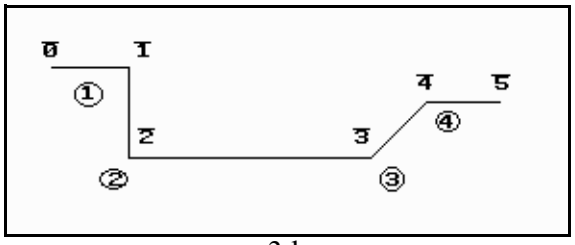
3.j

Note:
The last 2 parameters are used for graphic simulation of the product in a production mode (Automatic mode or Step mode). This visualisation does not require a product drawing, only these parameters must be programmed correctly. Furthermore, the parameter ‘Blank Length’ must be programmed correctly in order to obtain a visualisation.

Note:
After selecting a new bend this will be a copy of the preceding one; you only have to program those parameters which are different from the preceding bend.

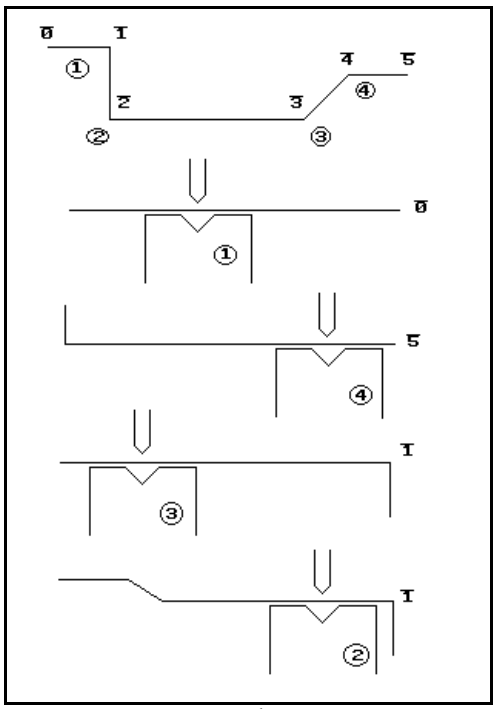
Example:

Product to make:



3.k

0 to 5 are possible backgauge positions.
1 to 4 are angle numbers.



3.l

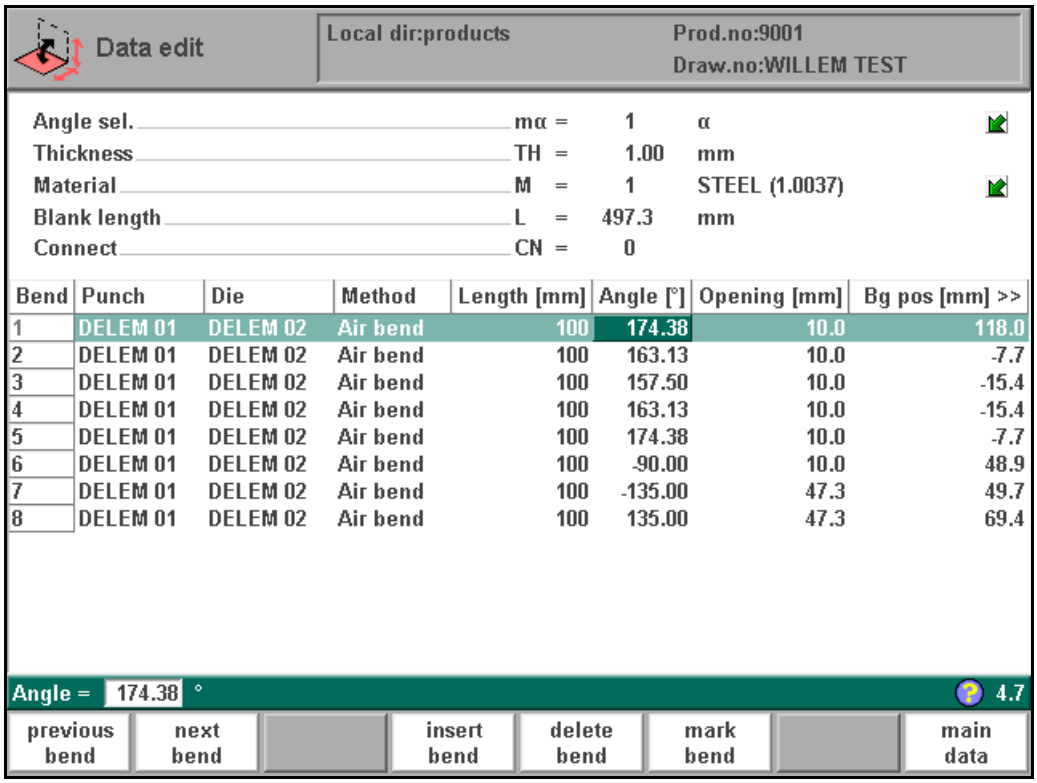
Bend	Angle (AN)	Backgauge (XN)
1	1	0
2	4	5
3	3	1
4	2	1

When in, for example, the 1st bend the plate must rest on the backgauge finger, see table below for programming values.

Bend	Angle (AN)	Backgauge (XN)
1	1	**100**
2	4	5
3	3	1
4	2	1

- **List of bendings of the prepared program**

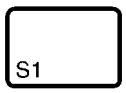
Pressing this softkey in the data editing mode will result in a complete overview of the bendings programmed. After pushing END the page will be restored from which this page was selected, with the cursor on the parameter selected before.



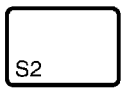
3.m

A specific bend can be selected on the screen by putting the highlighted bar on that bend, then pressing END.
The view is split in two parts. At the top of the parameter screen, the general parameters of the product are shown. In the main section of the screen, the bend parameters are shown in a spreadsheet view.
From within this screen, the complete CNC program can be edited. All bend parameters can be edited within the spreadsheet and bends can be swapped, moved, added and deleted.

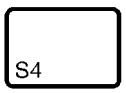
Function keys:



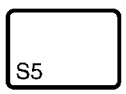
prev. bend To go to a **previous bend** when available.



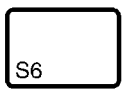
next bend To go to a **next bend** when available.



insert bend To insert a new bend between one of the bends, select with the cursor keys first the bend after which a bend must be inserted. Then push S1 for inserting, results in a new bend which is a copy of the previous one.



delete bend To delete a bend of the program first select this bend with the cursor (highlighted bar).

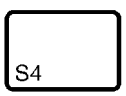


mark bend Mark the current bend, in order to prepare it for another action, like move or swap. See description below.

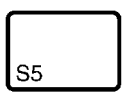
END Back to bend parameters.

When a bend has been marked with the function key 'mark bend' several other function keys become available:

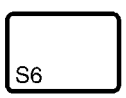
Function keys:



move bend With this command the bend sequence can be edited in this screen.



swap bends Let two bends exchange places.



abort mark Remove the mark from the currently marked bend.

Move bend
In the spreadsheet overview of the bendsequence, it is possible to change the order of bends simply by moving a bend to another place. Use the arrow keys to move the cursor to the bend

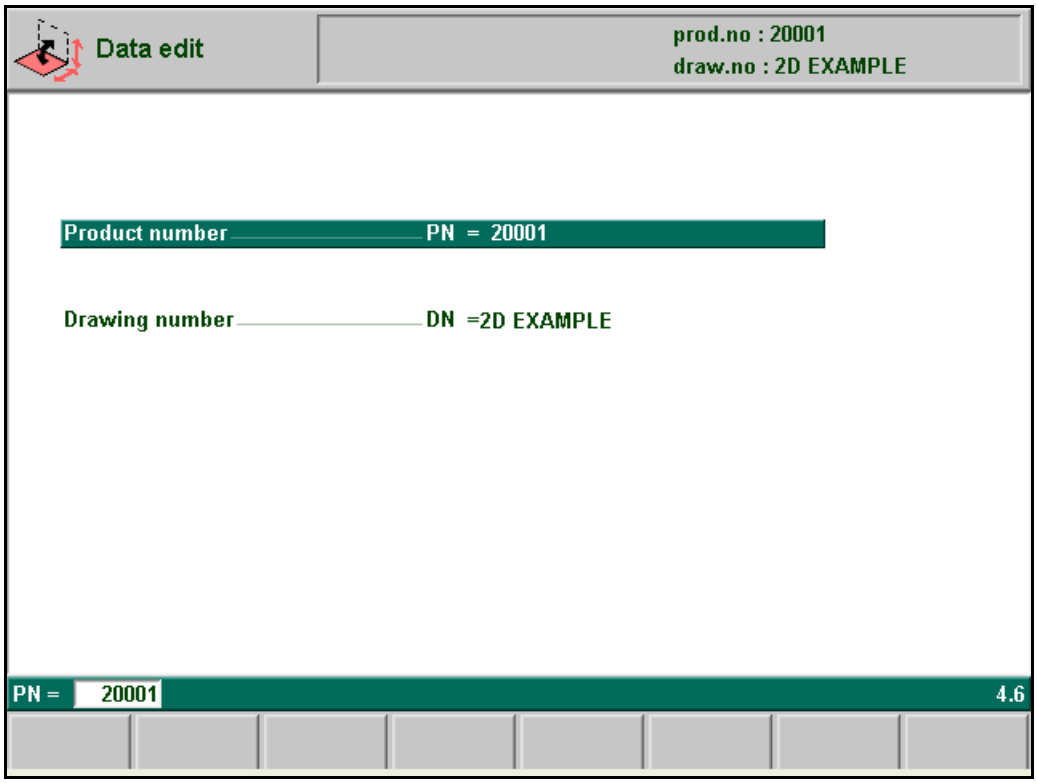
that must be replaced. Then press the button ‘mark bend’ and the bend is highlighted. Now use the arrow key to move to the right place in the sequence. When the correct bend number is highlighted, press ‘move bend’. The bend will be inserted on the current place.

Swap bends
With this command, two bends can change place in the bend sequence. Move the cursor to one of the required bends and press the ‘mark bend’ button. Then move the cursor to the bend with which it must be swapped and press ‘swap bends’. If for any reason the action must be cancelled, press the softkey ‘abort mark’ during the procedure.

A bend is no longer marked when the mark is aborted, when an action is finished or when this menu is left.

• Ending data preparation/data editing

After pushing the END key in the Data edit screen, a new page with product number and drawing number appears.



3.n

At this page the program number and the drawing number must be given to the new prepared program, or can be changed in the edit mode.
If either way the number is changed into an existing product number, a warning appears that this product already exists. You are asked whether to replace that existing product with the new product or not. If you press ‘yes’ (S4), the existing product is replaced by the current product. If you press ‘no’ (S5), you have to enter a new number.
After typing the product number, press ‘enter’ to confirm it and to return to the main menu.

3.3. Special edit function

When a new program has been made and several tests are required, the operator can switch from "editing"-mode directly to the "auto"-mode and vice versa by pressing the particular mode buttons.
When in "data editing" a specific bending is selected, the operator can now select the "automatic mode" or "step by step mode".
In case one wants to edit the program, the edit mode can be selected directly with the programming key. In this way many key operations are saved in order to make program changes quickly.

Note:
This special edit function can not be used with connected programs.

3.4. Edit notes

After changing program data in the menu ‘Data edit’ the control will not automatically calculate:

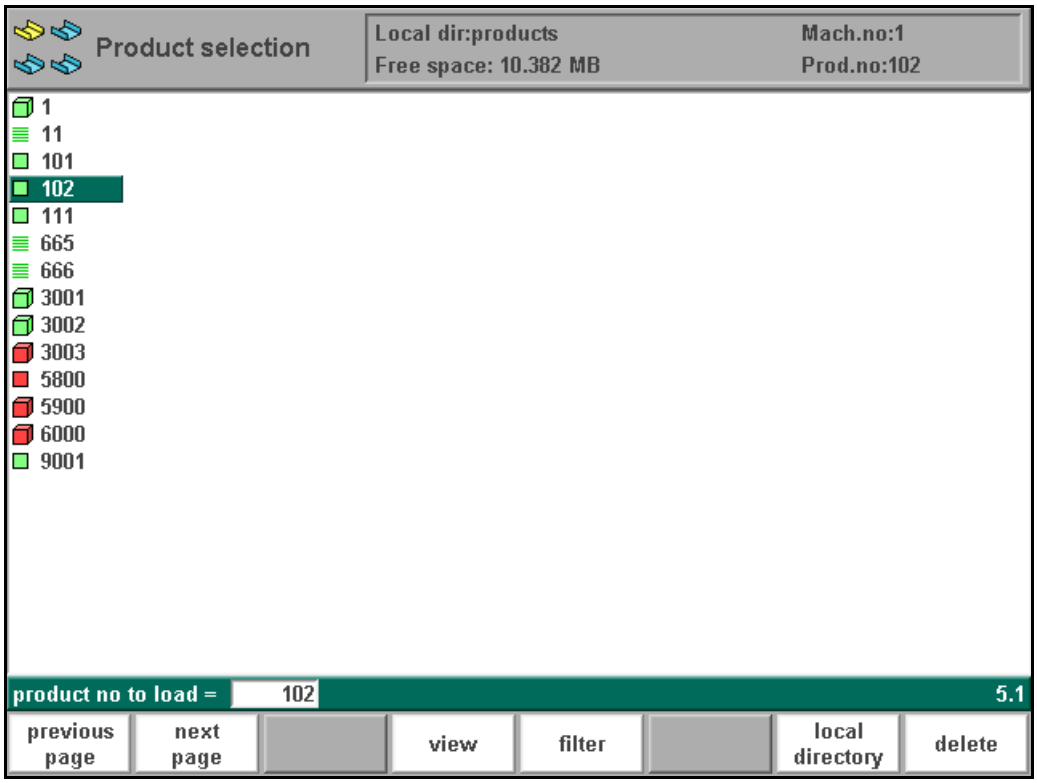
1 Force
2 Decompression
3 Crowning device setting
4 Z-axis position offset
5 X-axis position correction

Parameters 1 through 4 are only automatically recalculated if the parameter **Auto computations edit** (see the menu ‘program constants’) has been activated (value = 1).
Parameter 5 is only automatically recalculated if the parameter Bend allowance (see the menu ‘program constants’) has been activated (value =1). Additional corrections on the X-axis position can be made with the parameter **corr. X** (per bend) and **G-corr.** (for all the bends of the active program) in the automatic and step-by-step mode.
For detailed information upon both parameters see the chapter about Automatic mode and Step-by-step mode.
There is one exception: when the parameter bendmethod is changed the “force” and “Decompression” will be adjusted automatically.
Also you may find that the graphical simulation will not go properly when you have changed tools and/or X-axis dimensions.

4. Product selection

4.1. Introduction

This chapter describes the menu ‘product selection’.



4.a

In the "product selection" overview, a listing of all the programs which are present in the main memory is given. In the enter field a new product number or drawing number can be entered to select (load) a new program from the mainmemory. The loaded product can directly be executed in the automatic mode. Each item in the list consists of a symbol and a number. The number is simply the product number, the symbol can have the following meanings:

	the product has a CNC program, there is no drawing
(red)	the product consists of a 2D drawing, there is no CNC program
(green)	the product has a 2D drawing and a CNC program

Functions keys:

S1	**previous page**	Previous page with products overview
S2	**next page**	Next page with products overview
S4	**view**	Button to select one out of three possible viewing modes: - normal dir. - dir. expand
S5	**filter**	Function key to open a new button bar with extra function keys for product selection functions: - prod. nr. - draw nr. - search
S7	**local directory**	Via 'local directory' you get an overview of the manually created product directory structure.
S8	**delete**	Delete programs from the memory
END		End the product selection

When the function key 'View' has been pressed a new, temporary button bar appears with additional softkeys:

4.b

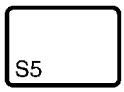
Function keys:

S4	**normal dir.**	Select a normal view of the product collection. This is the standard setting.
S5	**dir. expand**	To obtain a detailed overview of the product collection, with additional information for each product

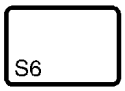
The other function keys cannot be selected here, until one of these three function keys has been chosen. The temporary bar will disappear again.
When the function key ‘filter’ has been pressed a new, temporary button bar appears with additional softkeys:

4.c

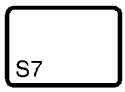
Function keys:



prod. nr. Products are shown in columns with symbols and product number, they are sorted on product number. This is the default view.

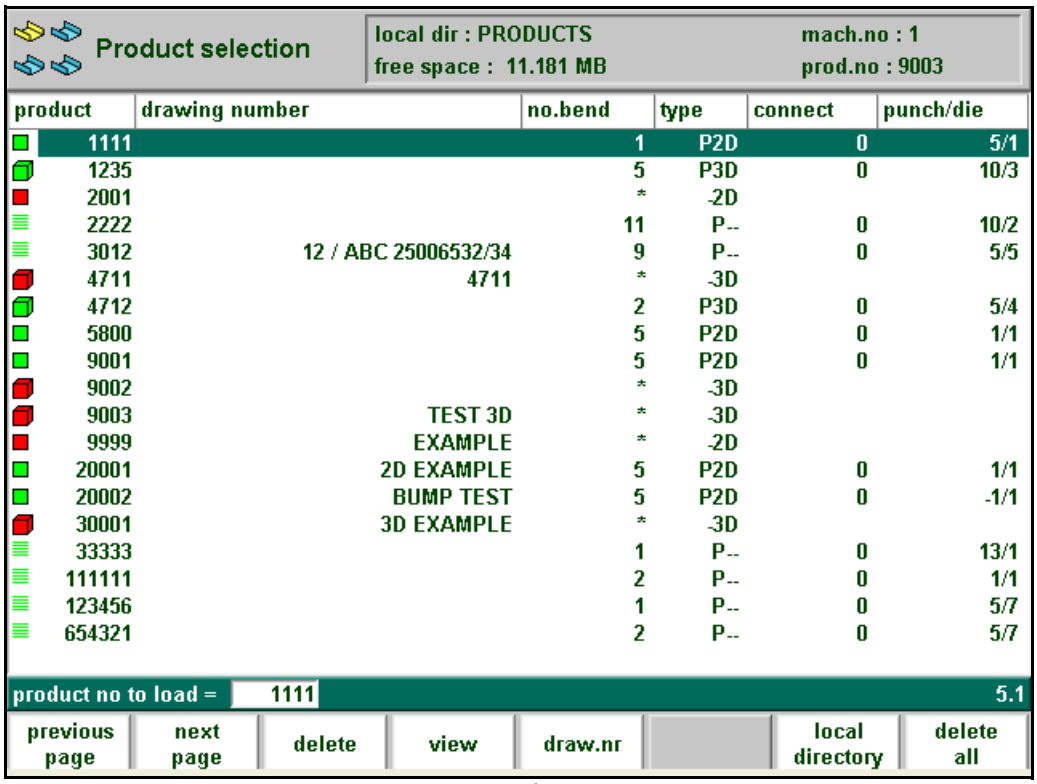


draw nr. If this key is pressed in this screen the list of products is automatically sorted on the alphabetical order of drawing numbers.



search Start the search function, to search for a specific name or number.

- ## Expanded directory



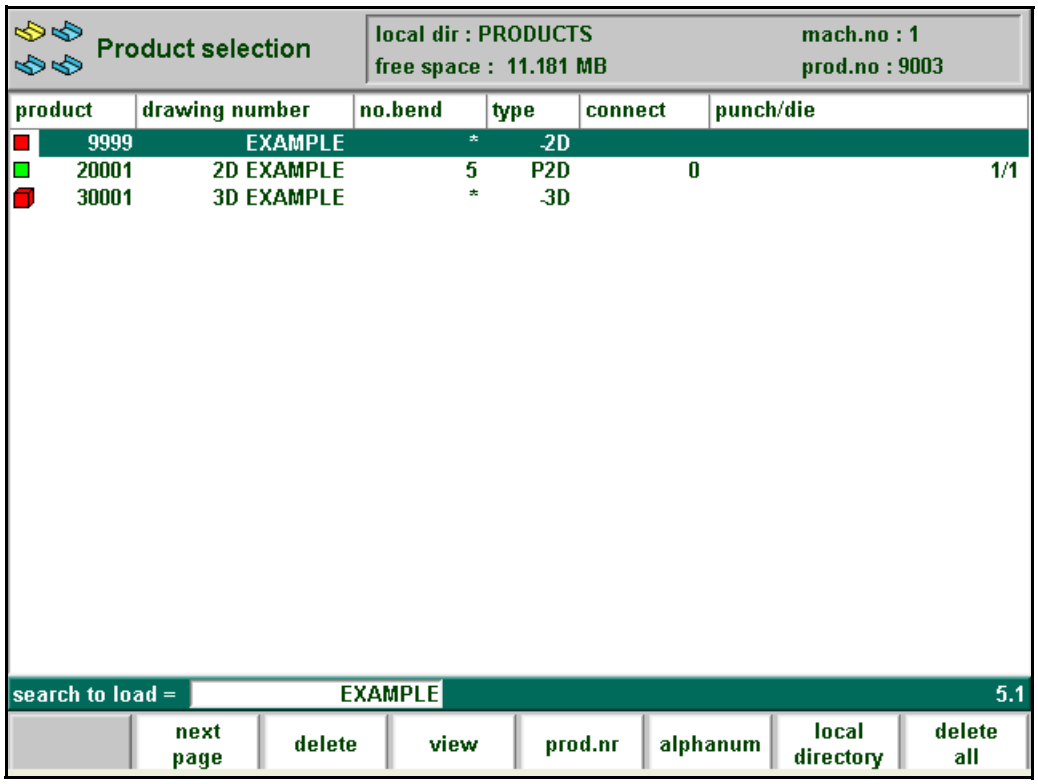
4.d

Expanded directory, a product overview which contains the following information for each product is displayed:

- Product number
- Drawing number
- Number of bends
- Type indication
- Connected program (0 = no connection)
- Selected tools (in first bend)

• Search function

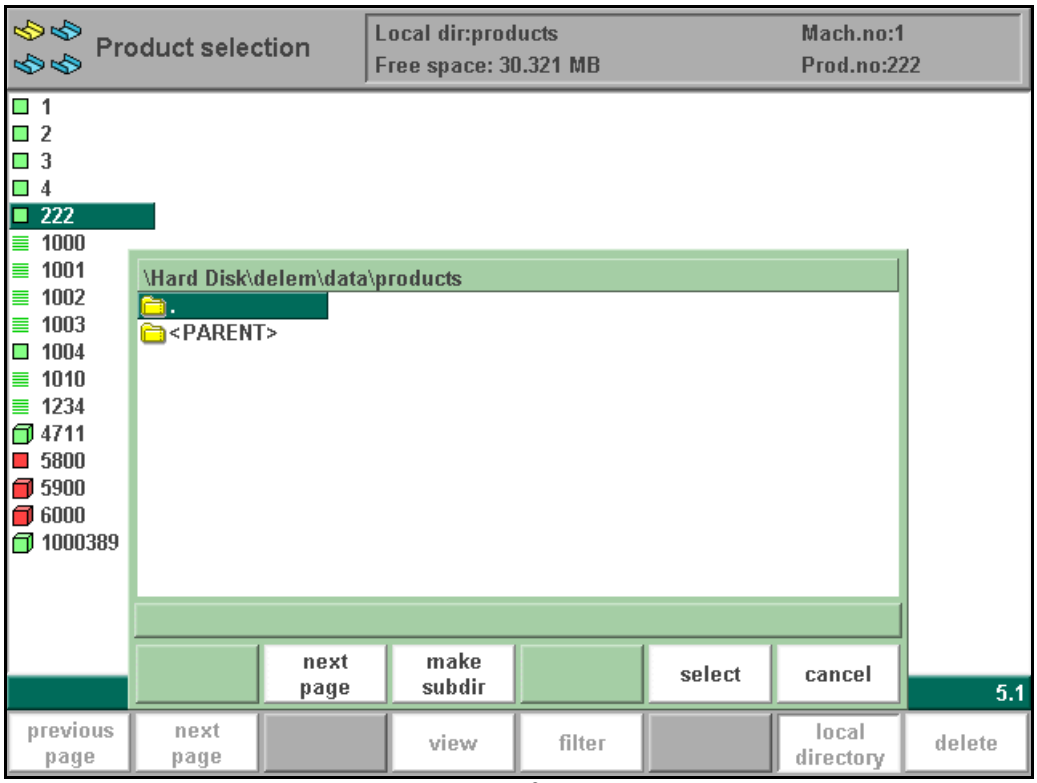
When the search function is selected, you can search for products with a certain number or name. At the bottom of the screen the text “search to load” is displayed, to indicate that you may enter a string the control will search for. You can either enter a complete name or number or only a part of it. Press ‘enter’ to start a search.
If you enter part of a name and this part occurs in several product names, the control will show all product names that contain the given characters. It is also possible to enter a combination of name and number.



4.e

4.2. Directories

Bend programs on the control can be stored in different directories. This directory can be used for storage of the products. The active local directory name is displayed in the header.
When the function key ‘local directory’ has been pressed in the product menu, a new window appears with an overview of directories on the control.



4.f

Function keys:

S4 — **make subdir** Add a new subdirectory. The name of the subdirectory can be any alpha numeric string of maximal 8 characters. (Avoid the use of slashes ‘/’).

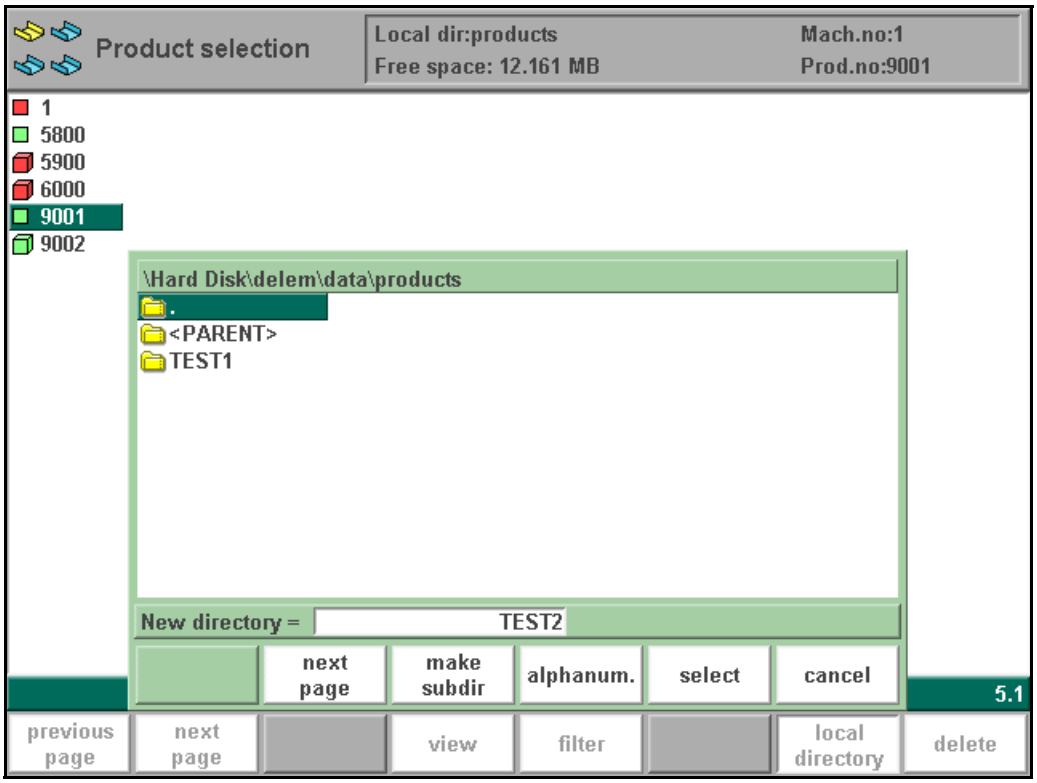
S5 — **remove sub-dir** Delete a subdirectory. If the subdirectory is not empty, the control returns the message ‘UNABLE TO REMOVE DIRECTORY’. An empty subdirectory is deleted without warning. The default directory “PRODUCTS” can also be deleted. When all the subdirectories have been removed the control creates automatically the default directory

S6 — **select** To select the directory on which the cursor is placed.

Function keys:

△	**Cursor key Up/Dn**	Select another subdirectory.
▽		

Use the function key 'select' to make a subdirectory active. Press ENTER to look inside a subdirectory. To move up, go to the <PARENT> map and press ENTER. To select the directory you are currently in, go to · and press ‘select’.
In this menu it is also possible to remove existing subdirectories (only an empty one) and to make new subdirectories. Press ‘make subdir’ and enter the new name. Subdirectories are called subdirectories because these directories reside under the local directory ‘\PRODUCTS’.



4.g

It is not possible to copy products from one subdirectory to another subdirectory. A product must be restored in a subdirectory via the menu ‘product backup’. You cannot change the name of the subdirectory.
When you leave the product selection menu the control remembers the active subdirectory and the active product (if a product was selected) until another directory or product is selected.

5. Programming of tools

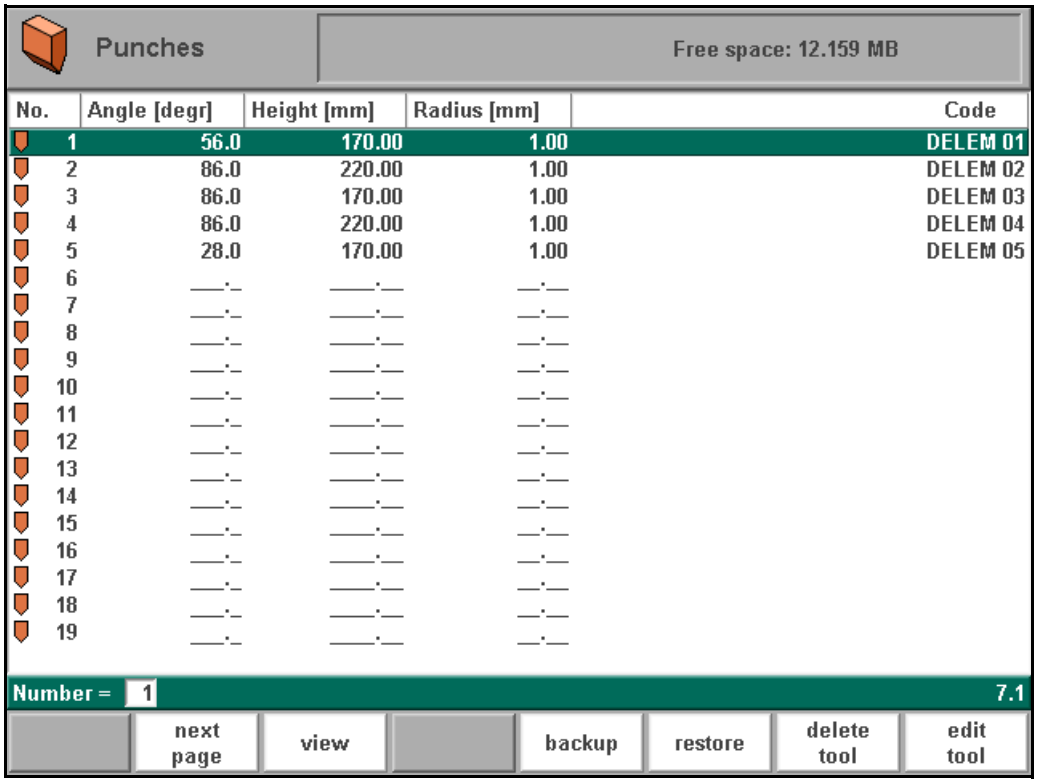
5.1. Introduction

This chapter describes the programming of the tools and machine shapes.

5.2. Programming of Punches

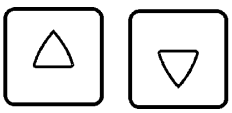
5.2.1. The punch menu

The programming of punches is started by choosing ‘Punches’ in the main menu. The punch profile can be created with help of the programming facilities of the control.



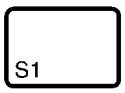
5.a

A listing of punches programmed into the memory is given.

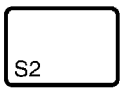


The punch is selected by moving the highlighted cursor bar resulting in the drawing of the upper tool.

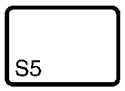
Function keys:



previous page To select a previous page within the several pages of the punch library.

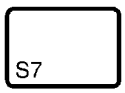


next page To select the next page within the several pages of the punch library

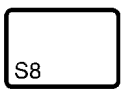


backup Save tools from library to disk.

restore Load tools from disk into the control.



delete tool Remove tool from library.



edit tool To edit the tool drawing or edit the punch data.

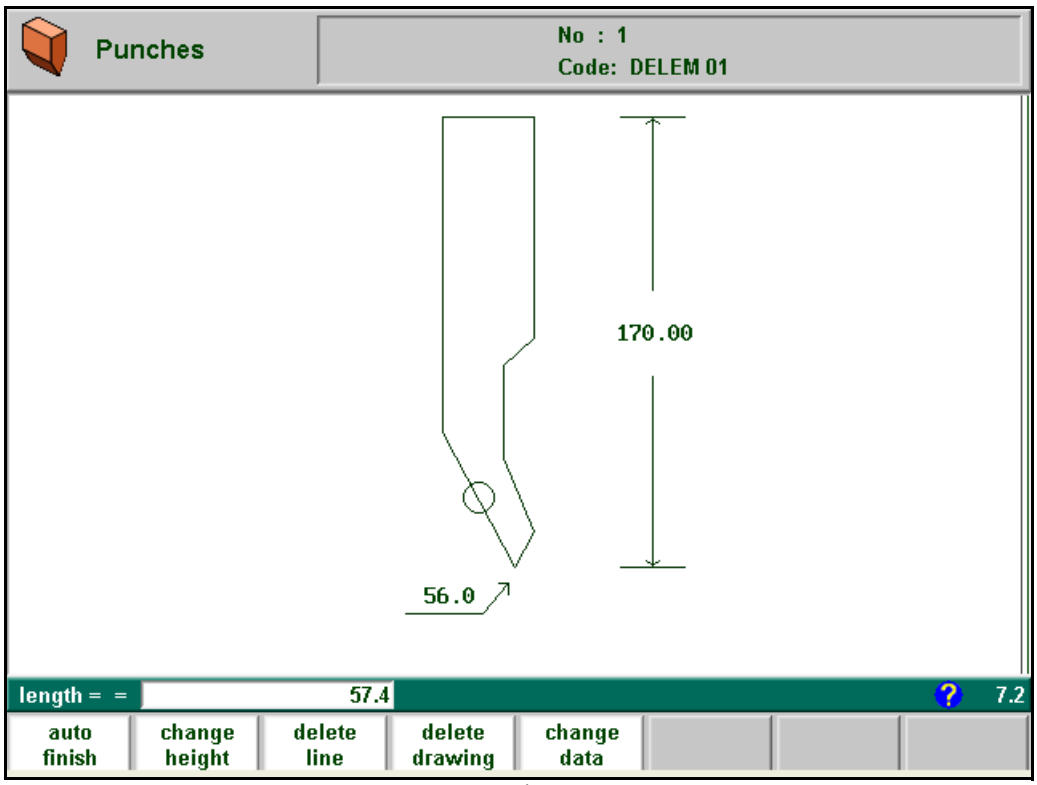
• Edit punch drawing

The display starts prompting the initial dimensions of your punch in the case of programming a new punch.

Height . . . . . .H
The height of the tool. Important: this height value will be used in bend depth calculation.

Angle . . . . . .a
The angle of the punch tip.

Hereafter you can enter angle and length dimensions as in the product drawing facilities.



5.b

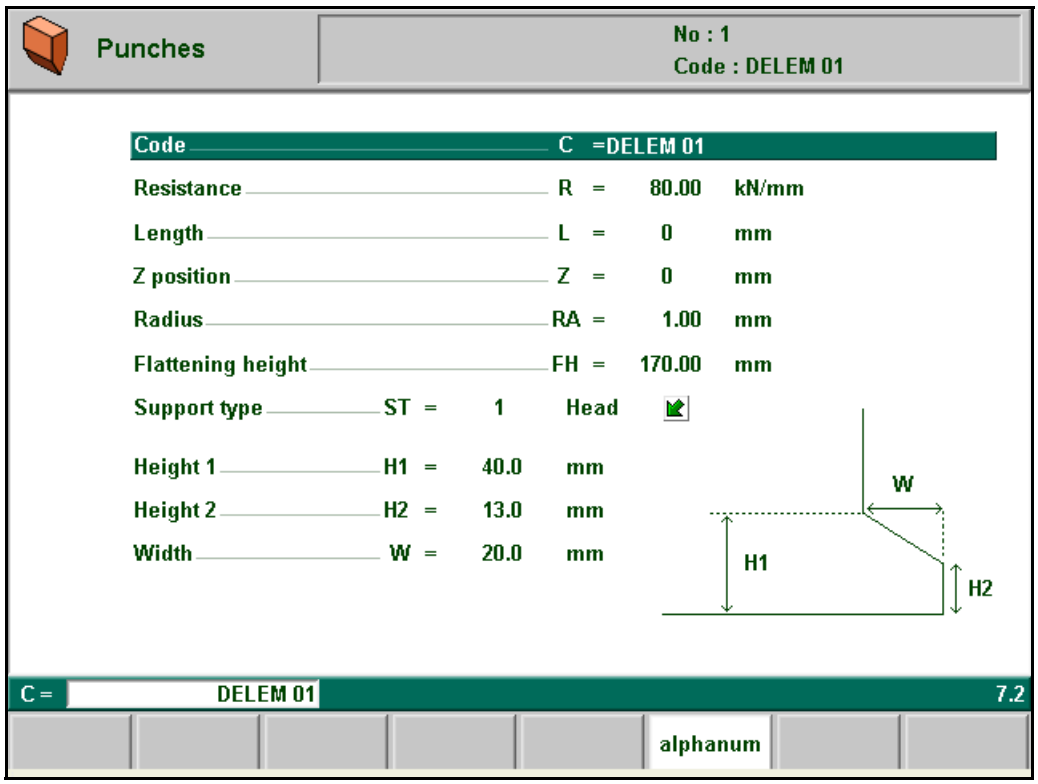
Function keys:

S1	**auto finish**	Finishing the tool outline to the top of the tool automatically
S2	**change height**	To change the height dimension of the tool
S3	**delete line**	To delete a line segment
S4	**delete drawing**	To delete the complete tool from the library
S5	**change data**	To change specific tool data and code (see section 6.2.2.)
END		To return to the directory overview

- **Drawing orientation of the punch on the screen**

The right hand side of the tool is the backgauge side.
The bottom point of the punch will be placed on the center line of the press brake shape.

5.2.2. Specific Punch Data



5.c

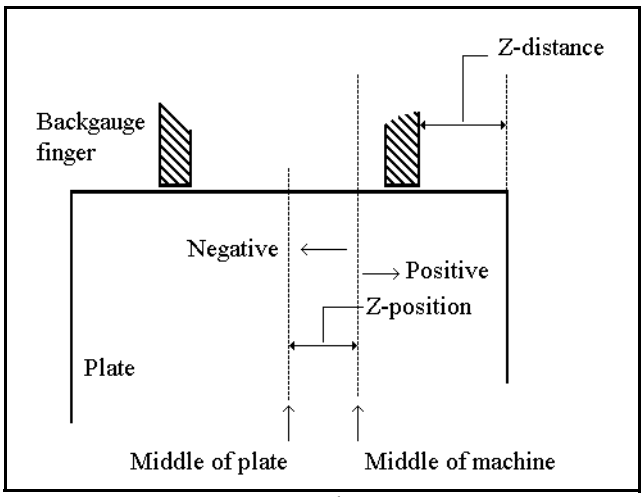
Code C
Alphanumeric tool name with a maximum of 20 characters

Resistance . . . . . .R
Maximum allowable force on punch in kN/mm

Length L
Length of the punch to be used

Z-position . . . . . .Z
Offset position of the middle position of the punch in relation to the middle of the press brake.
(Default value is 00 mm)
In case automatic Z-axis are used the Z-axis value is computed from the: plate length, Z-distance (menu 'Program constants') and Z-position.

Top view



5.d

Radius . . . . . .RA

The radius of the punch.

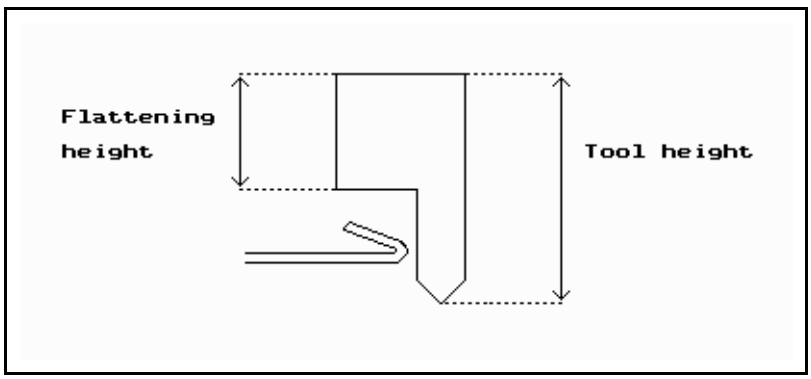
This parameter has to be used as selection criterium, the value will not be used in computation formulas.

Flattening height. . . . . .FH

The flattening height need to be programmed when you make use of a special flattening punch to flatten your product.

The programmable height is used to calculate the Y-axis value in case of "flattening". The default-value for this parameter is the standard tool height, which is already programmed. Every time that this standard height is changed, the flattening height will be made equal to this tool height (due to safety reasons). See figure 6.g.

The total toolheight will be used to calculate the speed change point mute and the flattening height will be used to calculate the end position of the Y-axis beam to flatten the product. The calculation will take 2x sheet thickness into account. Also you can program a flattening offset in the menu ‘Program constants’. This value will be added to the value of 2x sheet thickness.



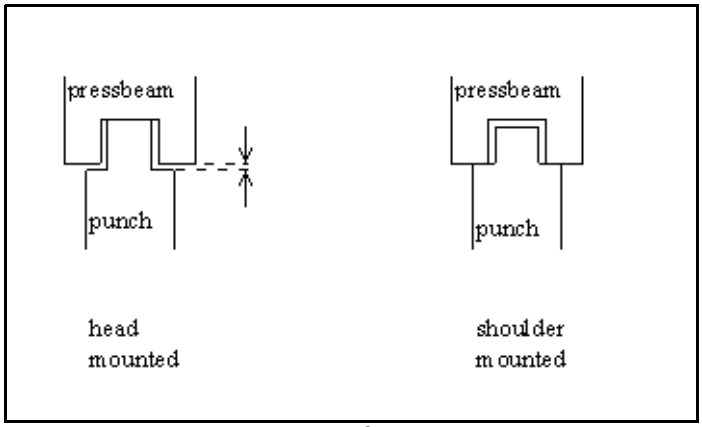
5.e

Support type . . . . . .ST

Switch parameter, to account for differently mounted punches. Such punches could

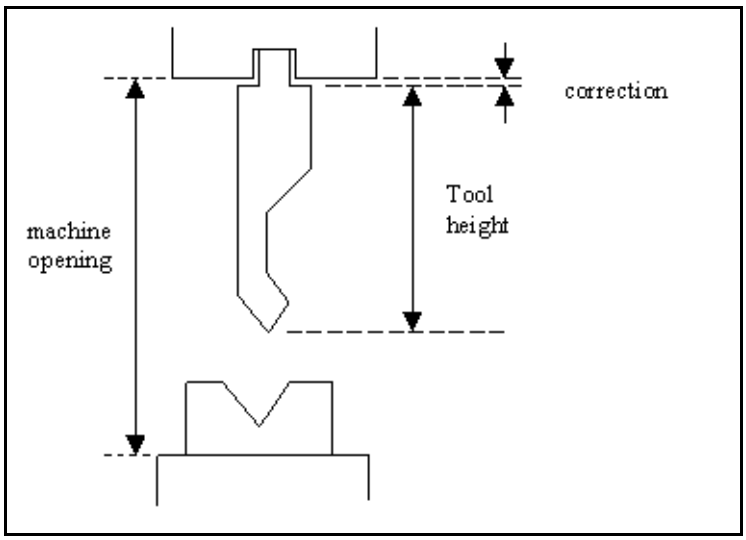
cause inaccuracies in the tool height and the resulting Y-axis position. The control makes a distinction between two settings, 'head mounted' and 'shoulder mounted'.
0 = shoulder mounted (default setting)
1 = head mounted

Press the key ⌂ to select the required setting.



5.f

If 'shoulder mounted' is chosen, the Y-axis position is calculated from the standard tool height. This is the default setting.
If 'head mounted' is chosen, a correction is made for Y-axis computation.

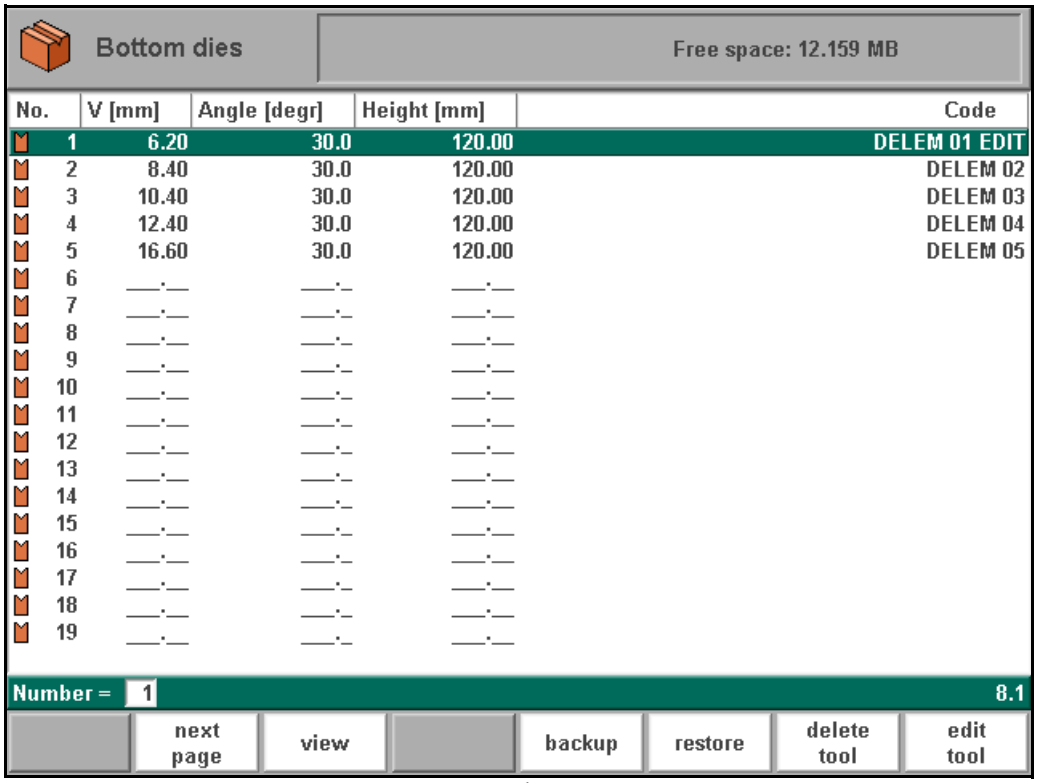


5.g

5.3. Programming of bottom dies

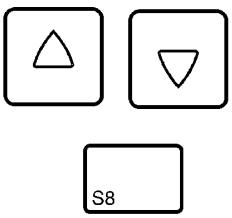
5.3.1. The die menu

The programming of dies is started by choosing ‘Bottom dies’ in the main menu.



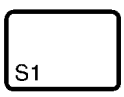
5.h

A listing of the programmed dies is given.



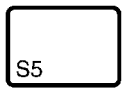
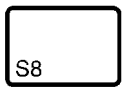
A die can be selected with the highlighted cursor.

Function keys:



previous page To select a previous page within the several pages of the available dies in the library.

Function keys:

	**next page**	To select the next page within the several pages of the die library.
	**backup**	Save tools from library to disk.
	**restore**	Load tools from disk into the control.
	**delete tool**	Remove tool from library.
	**edit tool**	To edit the tool drawing or check the die data.
		To return to the program menu.

• Edit die drawing

When starting the die drawing, the display prompts the main dimensions of the die:

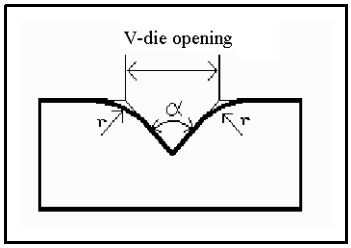
Height . . . . . .H
The height of the tool. Important: this height value will be used in bend depth calculation.

V-opening . . . . . .W
The V-opening of the die.

V-Angle . . . . . .α
The angle of the die.

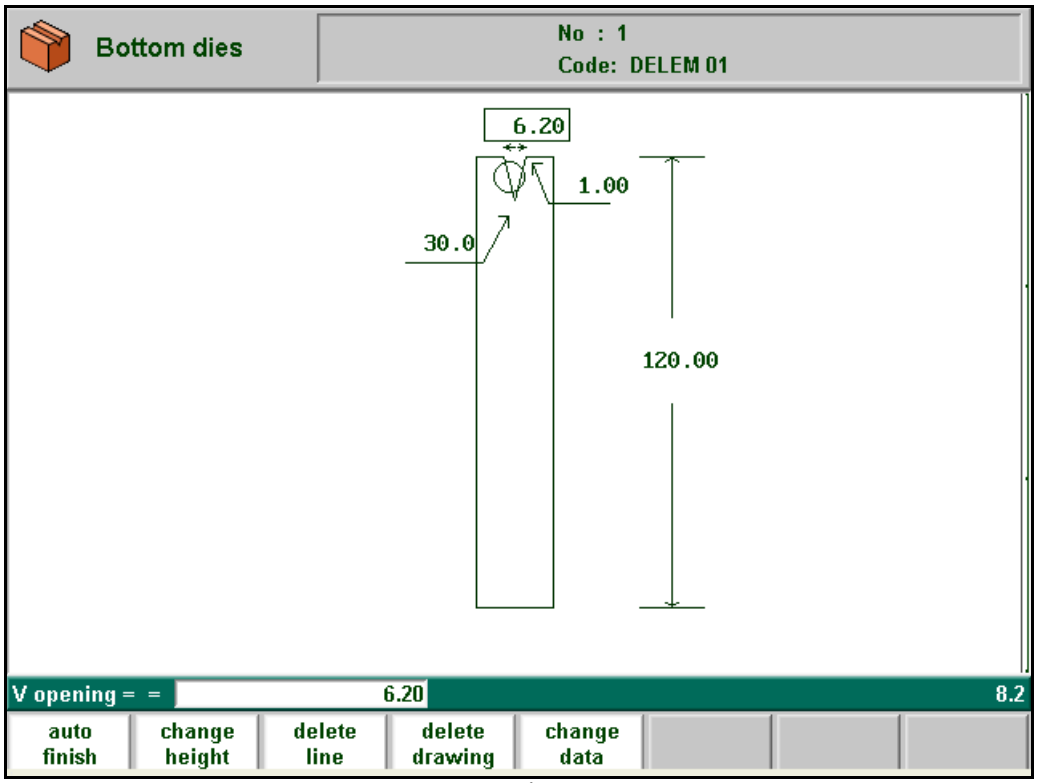
Radius . . . . . .RA
The radius of the edges of the V-opening.

V-die opening:



5.i

The width V is the distance between the touching lines crossing.
After entering these parameters you are ready to draw the important outer dimensions of your die. The right side of the die is the backgauge position. The mid-position of the V-opening will be placed on the centre line of the press brake shape.



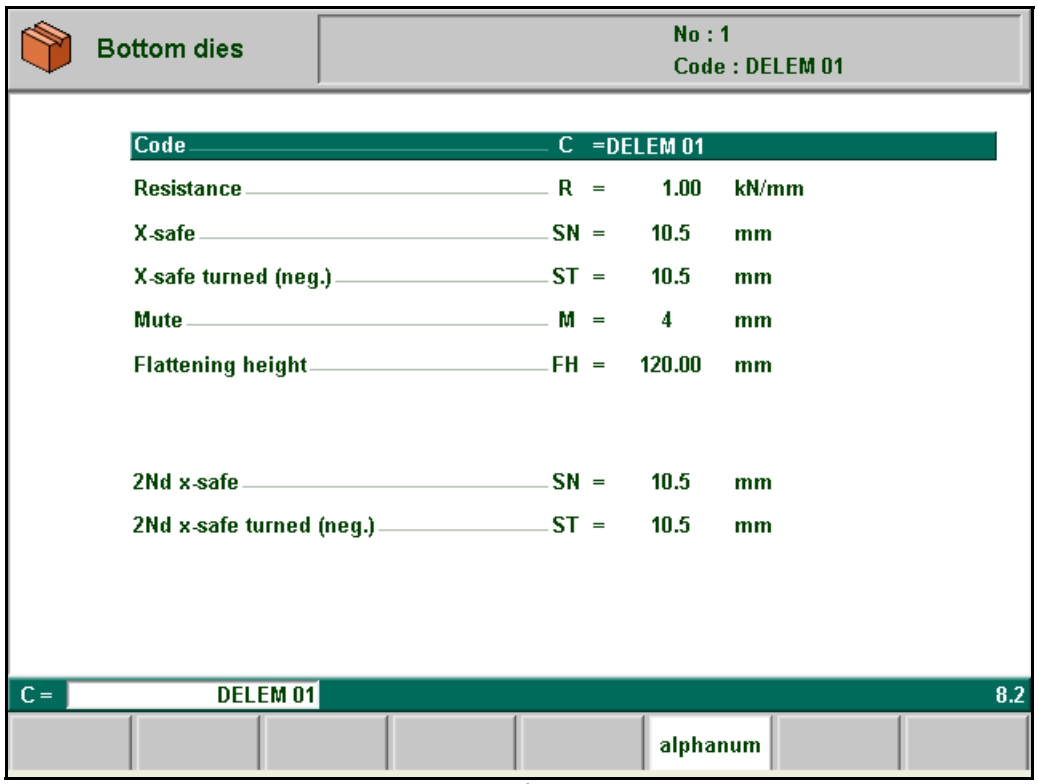
5.j

The length of the flat section on the right hand side of the V-grove will be used in order to calculate the minimum X-axis position for the R-axis collision detection with die (X-safe).

Function keys:

The softkey functions are exactly the same as for punch programming.

5.3.2. Specific die data



5.k

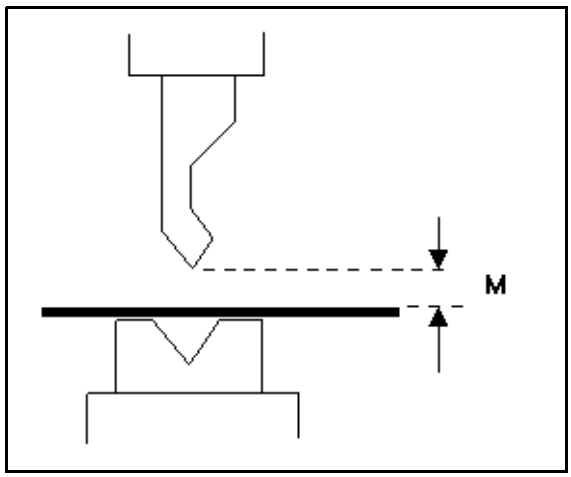
Code C
Alphanumeric tool name with a maximum 20 characters.

Resistance . . . . . .R
Maximum allowable force on the die in kN/mm.

X-safe . . . . . .SN
Calculated safety zone (minimum X-axis value), which will be used in the case a R-axis is mounted. This to prevent finger to die collision. The indicated minimum value is computed automatically from the die dimensions as follows:
X-SAFE = FS + ½ V in which:
FS = flat section on the right hand side of the V-grove
V = opening value
In this formula also an extra small safety value has been added.

X-safe turned (negative) . . . . . .ST
This second safety value will be active if the die is used in the turned position. This can be achieved by programming the die in the bendsequence computation with a negative number. For instance: Program for die number 2: "-2" if you want to use it in the turned position. In this situation parameter ST will be used as safety zone for the backgauge.

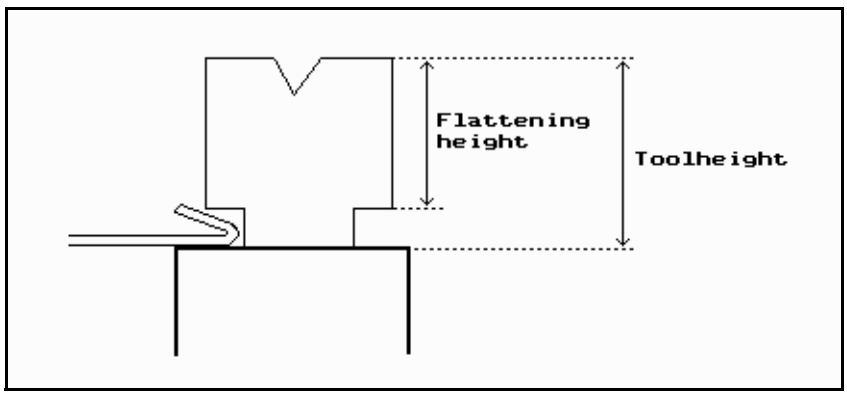
Mute . . . . . .M
Muting distance. Distance above the plate at which the speed change takes place.



5.l

Flattening height. . . . . .FH

The flattening height need to be programmed when you make use of a special flattening die to flatten your product.
The programmable height is used to calculate the Y-axis value in case of "flattening". The default-value for this parameter is the standard tool height, which is already programmed. Every time that this standard height is changed, the flattening height will be made equal to this tool height (due to safety reasons). See figure 6.r.



5.m

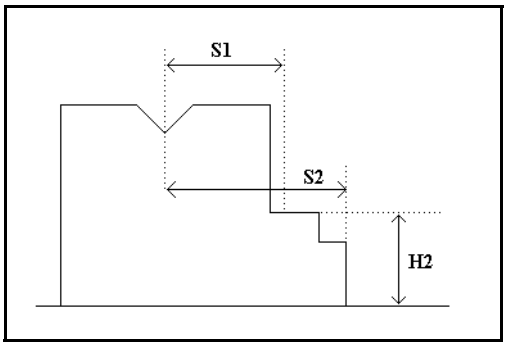
The total height will be used to calculate the speed change point mute and the flattening height will be used to calculate the end position of the Y-axis beam to flatten the product. The calculation will take 2x sheet thickness into account. Also you can program a flattening offset in the menu ‘Program constants’. This value will be added to the value of 2x sheet thickness.

2nd X-safe . . . . . .SN=

2nd X-safe turned (negative) . . . . . .ST=

Second automatically calculated safety zone (minimum X-axis value) to be used in the case a R-axis is mounted. The value is the distance from the middle of the V-opening to the most outside point of the drawing of the die (on the right side). Also an extra safety tolerance of 1mm is added. The situation for the die in turned position the distance to the

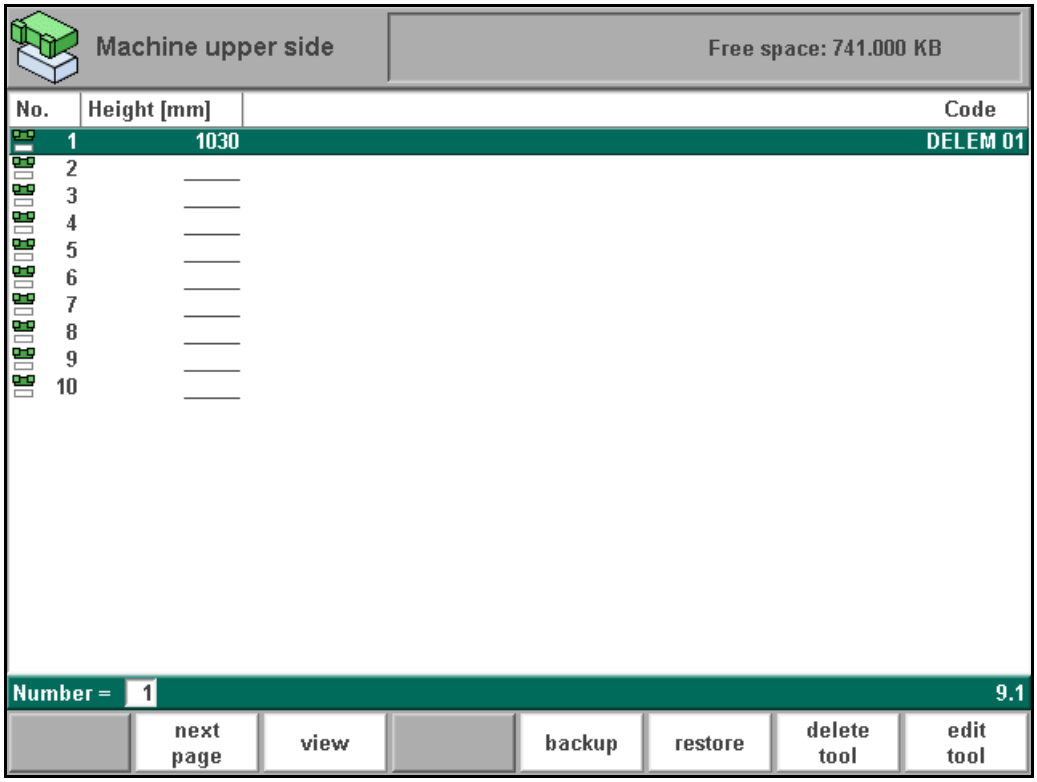
left side is used. These values are calculated each time the die drawing changes. These 2nd-values cannot be changed manually.



5.n

H2 is the height of the 2nd-values. H2 is not visible on the screen. See for explanation upon the safety zones the example drawing.

5.4. Machine upper side and lower side



5.o

Function keys:

S5 **backup** Save tools from library to disk.

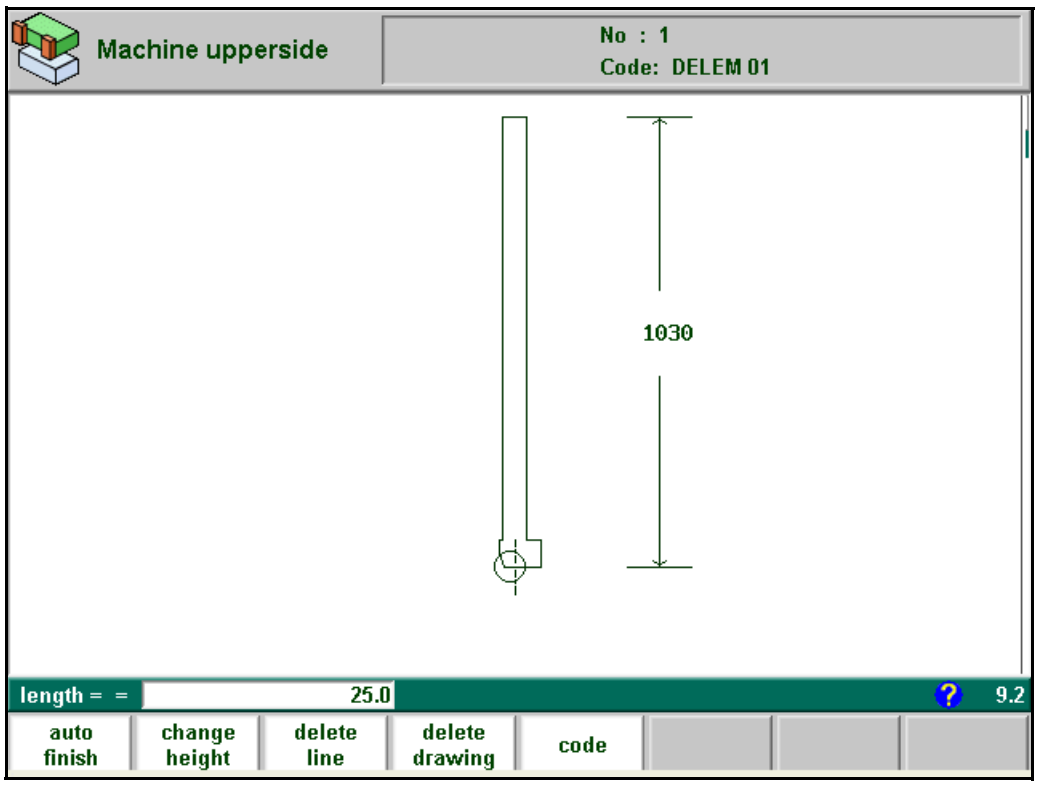
S6 **restore** Load tools from disk into the control.

S7 **delete tool** Remove tool from library.

S8 **edit tool** To edit the machine shape drawing or check the dimensions.

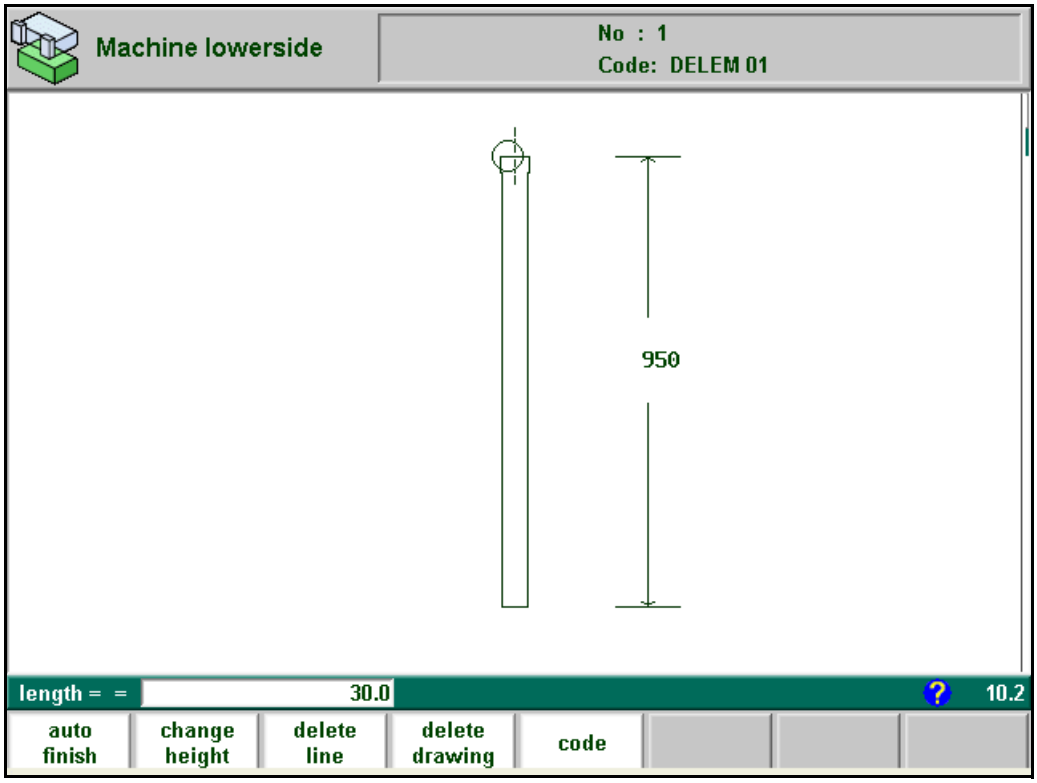
The shapes of your machine are drawn in the same way as for the punches and dies. Like with the tools the right hand side of the drawing is the backgauge position of the machine.
Before starting drawing the machine shape upper or lower side you must enter the press-beam height of the upper part when the moving part of the upper side is in its **mechanical top dead**

centre.
The length prompted after entering the height of the machine shape is the distance from the center line to the front side or the back side of the machine.



5.p

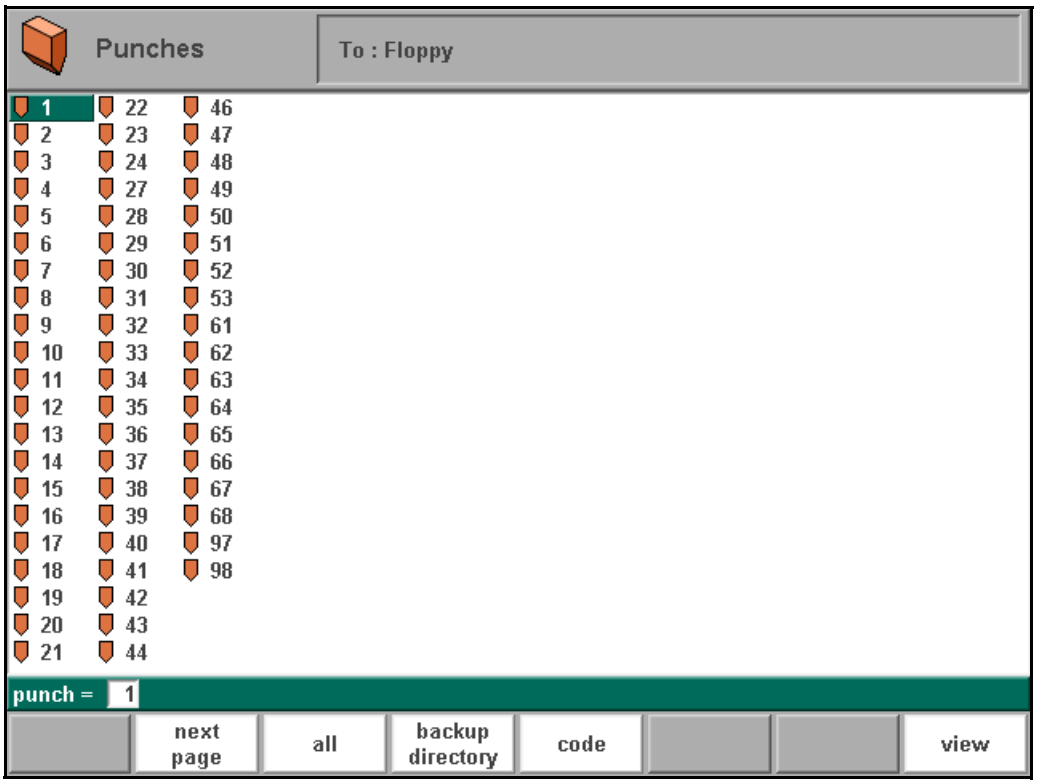
For the lower side of the machine the height is defined from the table surface to the floor level.


Machine lowerside
No : 1
Code: DELEM 01
950
length = =
30.0
10.2
auto finish
change height
delete line
delete drawing
code

5.q

5.5. Saving/loading of tools

With the function key 'save to disk' (in any of the tool menus) the dialogue 'Save Punches' is opened. From this menu, it is possible to save tools to a back-up location. This procedure works the same for all tools and machine shapes and regardless of the back-up location (floppy disk, network).
When the function key 'save to disk' has been pressed, a screen like shown below should appear.



5.r

At the top of the screen, the back-up location is shown. If this location is not correct, you can change the subdirectory with the softkey 'backup directory' or you should go to the program constants menu, to change the device where tools are saved to. See the chapter about the program constants.
To save a tool, enter its number from the list and press 'enter'.

Function keys:

S3 **all** Save all tools.

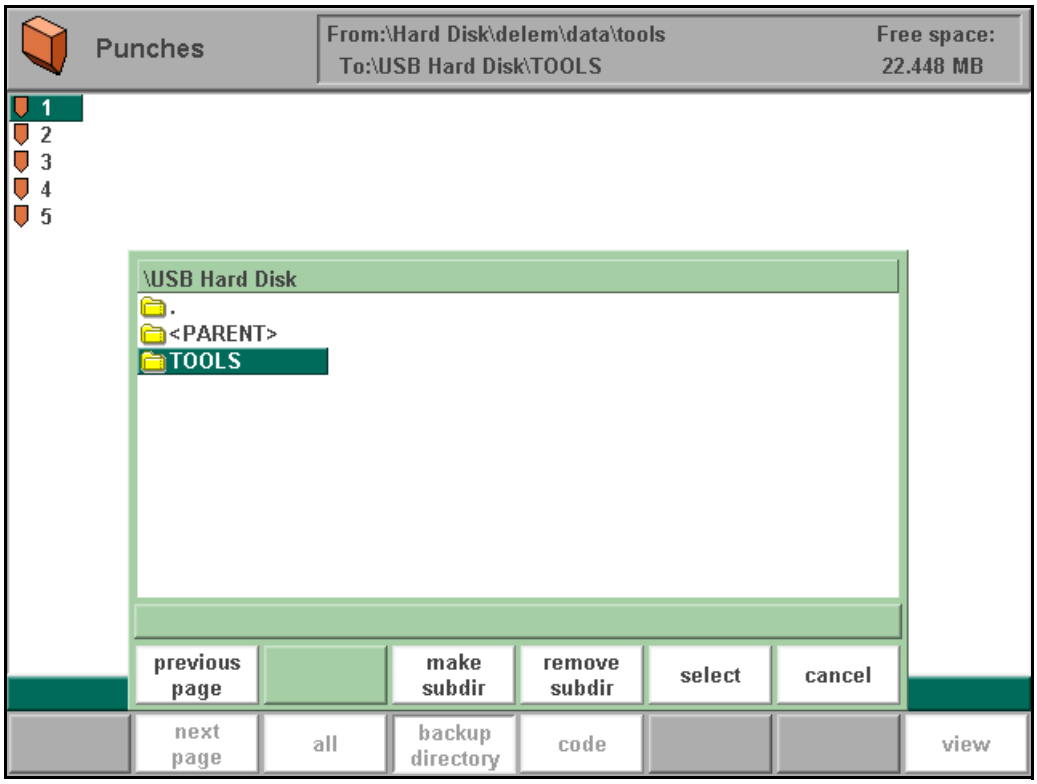
S4 **backup directory** Change, make or remove a subdirectory where tools will be saved.

Function keys:

S5 **code** Select on code instead of number.

S8 **view** Button to select one out of two possible viewing modes:
- normal dir.
- dir. expand

For backup operations, it is possible to make use of subdirectories. Subdirectories can be created in this menu. When the softkey 'backup directory' is pressed, a new window appears with a list of available back-up directories. A selected directory can contain products and tools, so each user can keep his own set of products and tools, if necessary.



5.s

You can make new subdirectories or delete existing ones with the softkeys 'make subdir' and 'remove subdir'. If there are subdirectories present, use the arrow keys to scroll to the required directory and presss 'enter' to select it.

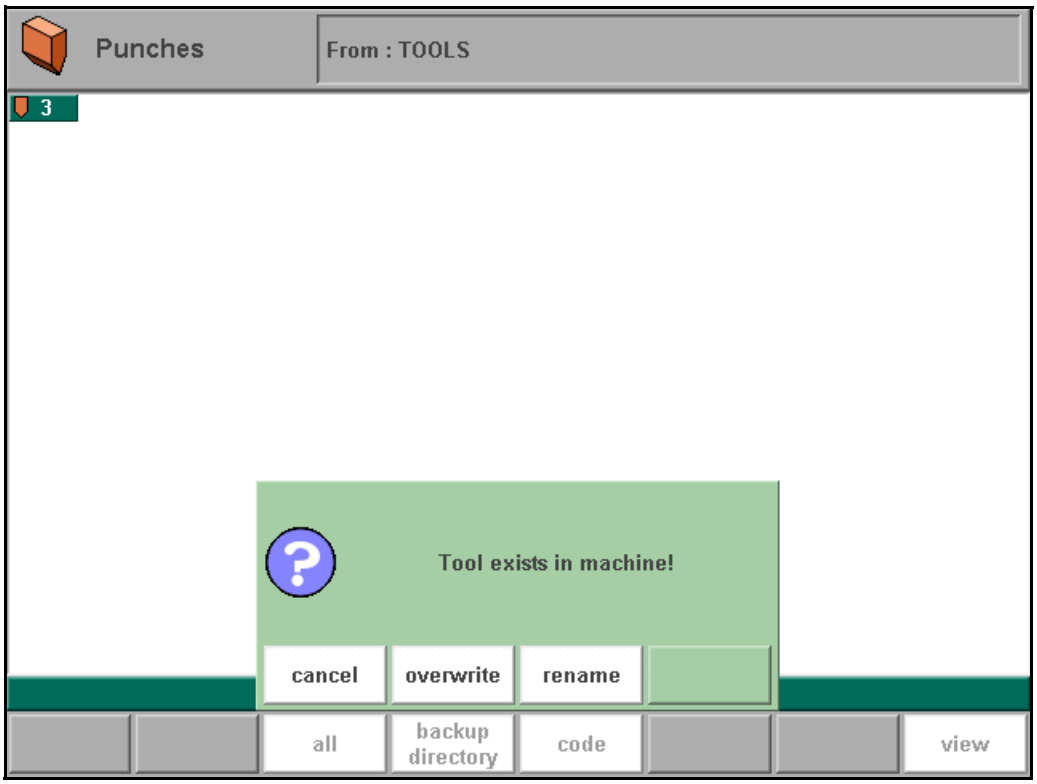
Function keys:

S4 **make subdir** Add a new subdirectory. The name of the subdirectory can be any alpha numeric string of maximal 8 characters. Avoid the use of slashes ('/')

Function keys:

S5	**remove subdir**	Delete a subdirectory. If the subdirectory is not empty the controller returns the message 'unable to remove directory'.

The 'load tools' procedure is different from the 'save tools' procedure. When you select 'load tools' the control checks if the toolnumber already exists in the control. If not, then the tool is automatically copied from disk to the control. If the toolnumber already exists then you have three options:
-cancel (S3), the toolnumber is not loaded.
-overwrite (S4), the tool in the control is replaced by the tool from disk.
-rename (S5), the tool number of the tool on the disk is first renamed before it is loaded from disk.
This means (in case of rename) that a new toolnumber is proposed. This new toolnumber is the first free tool number in the tool library. Accept the proposed tool number with the enter key or enter the number of another free tool number and press the enter key.



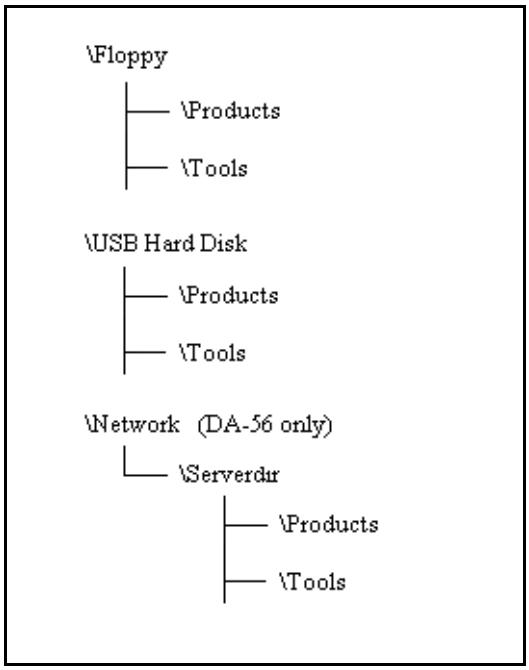
5.t

Note:
Tools are linked to the control by their machine number. When a 'restore' operation is started, the control will only show those tools, which have the same machine number as the control. If for any reason the machine number is changed, a new tool back-up operation should be executed to save the control tools with their new machine number.

6. Products and tools back-up

6.1. Introduction

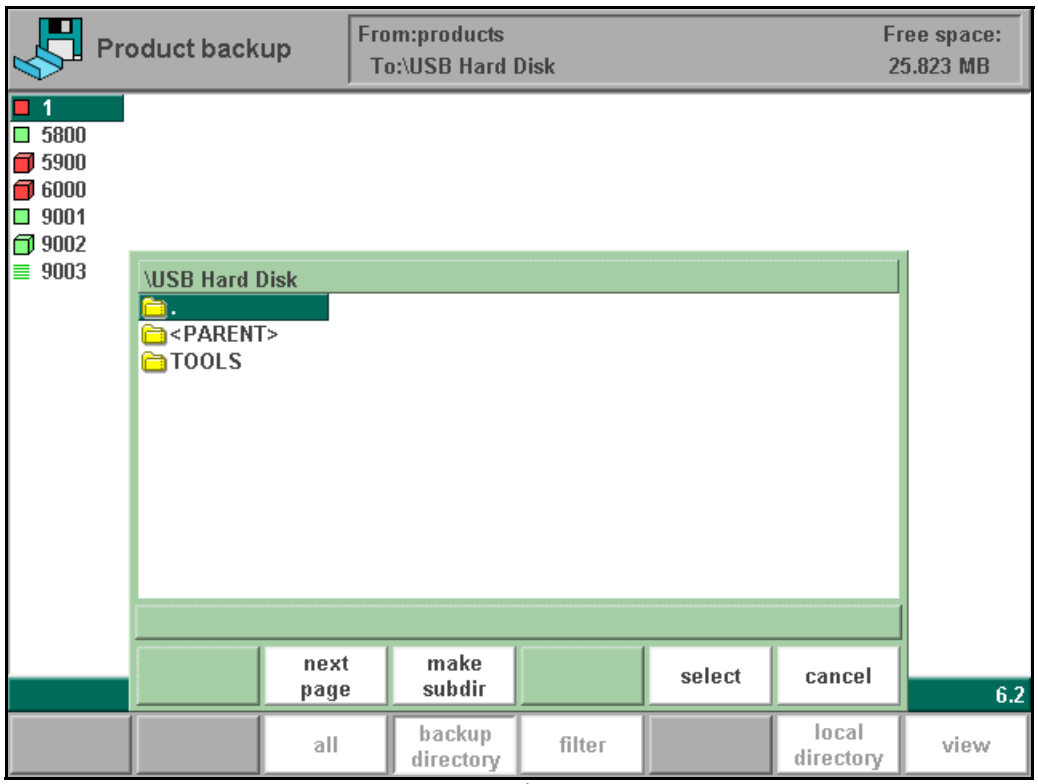
This chapter describes the procedures to make a back-up of your products and tools. The procedures for saving or reading data are similar for all types of back-up media: floppy disk, network or USB disk.
The actual back-up directory consists of a device (USB disk, network) and a directory. The choice of devices depends on which devices are connected to the control. If necessary, directories can be created and selected. The back-up locations for storage of products and tools are set independently.



6.a

6.2. Directory navigation

When softkey ‘backup directory’ is pressed, a new window appears with a list of available back-up directories.



6.b

In this window you can browse through the directory structure of your back-up device. Press ENTER to look inside a subdirectory. To move one level up, go to the <PARENT> map and press ENTER. To select the directory you are currently in, go to . and press 'select'.
To change from one device to another, press ENTER on the <PARENT> symbol a number of times until the highest level is reached. From there, select the proper device and choose the correct subdirectory.
If a network connection is available you must first select 'network' and subsequently one of the offered network volumes. After that it works similar to other devices.
You can make new subdirectories or delete existing ones with the softkeys ‘make subdir’ and ‘remove subdir’. If there are subdirectories present, use the arrow keys to scroll to the required directory and presss ‘select’ to select it.

Function keys:

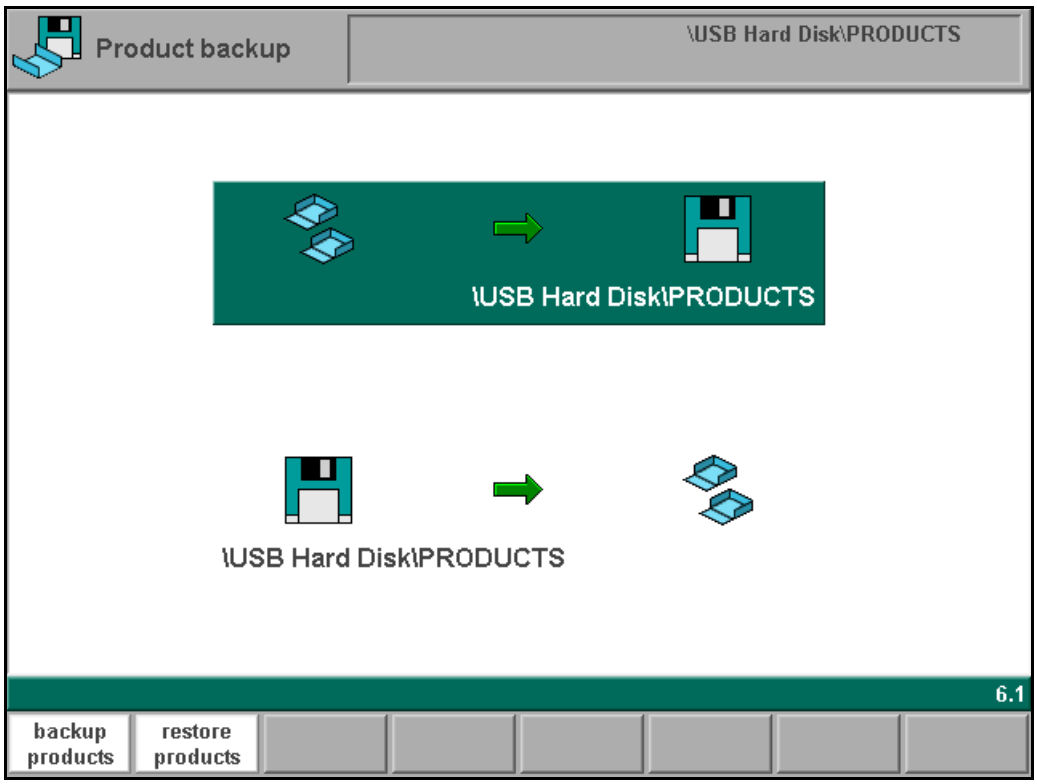
S4	**make subdir**	Add a new subdirectory. The name of the subdirectory can be any alpha numeric string of maximal 8 characters. Avoid the use of slashes (‘/’).

Function keys:

S5	**remove subdir**	Delete a subdirectory. If the subdirectory is not empty the controller returns the message 'UNABLE TO REMOVE DIRECTORY'.

6.3. Product backup

To make a back-up of programs to disk, choose 'Product backup' in the main menu.



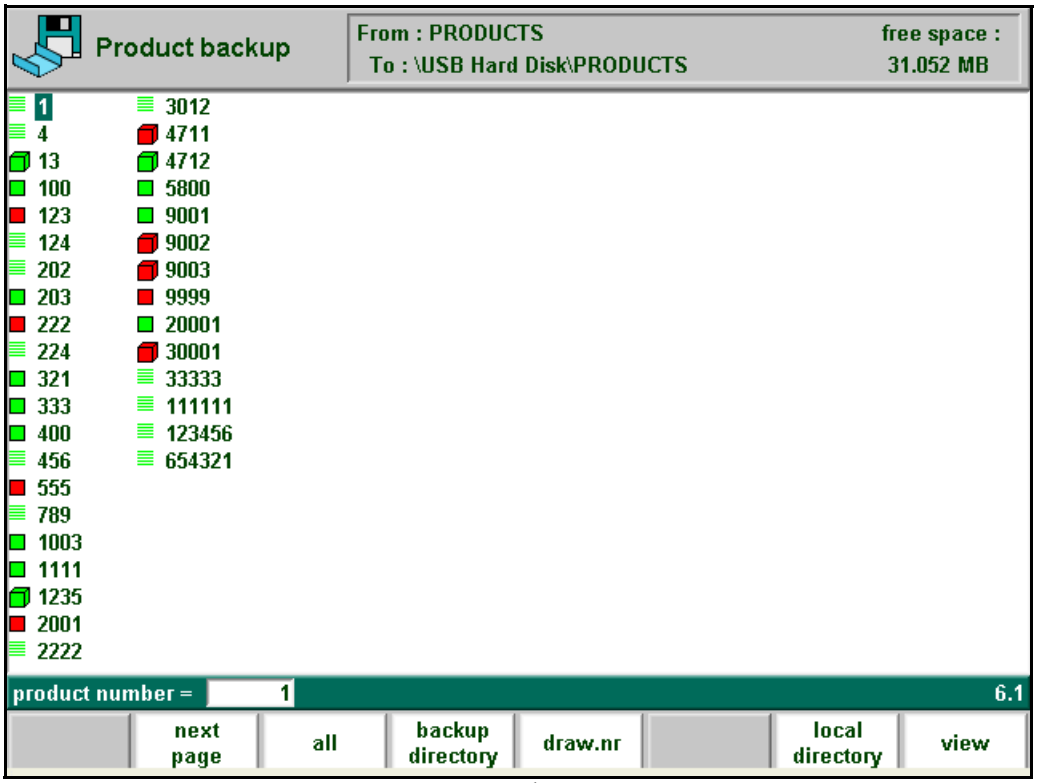
6.c

From this screen, a backup or a restore function is available. It is also indicated to which location a backup is made (floppy disk, network, etc.).

Function keys

S1	**backup products**	Open the back-up menu to save products to disk.
S2	**restore products**	Open the restore menu to load products from disk.

When ‘backup products’ has been selected, a screen like shown below should appear.



6.d

At the top of the screen, the back-up location is shown. If this location is not correct you can change the subdirectory with the softkey ‘backup directory’ or you should go to the program constants menu to change the device where products are saved to.
To save a product, enter its number from the list and press ‘enter’.

Function keys:

S3	**all**	load all products
S4	**backup directory**	Change, make or remove backup directory
S5	**draw. nr.**	Toggle drawing number / product number
S7	**local directory**	get an overview of the manually created product directory structure on the control disk.

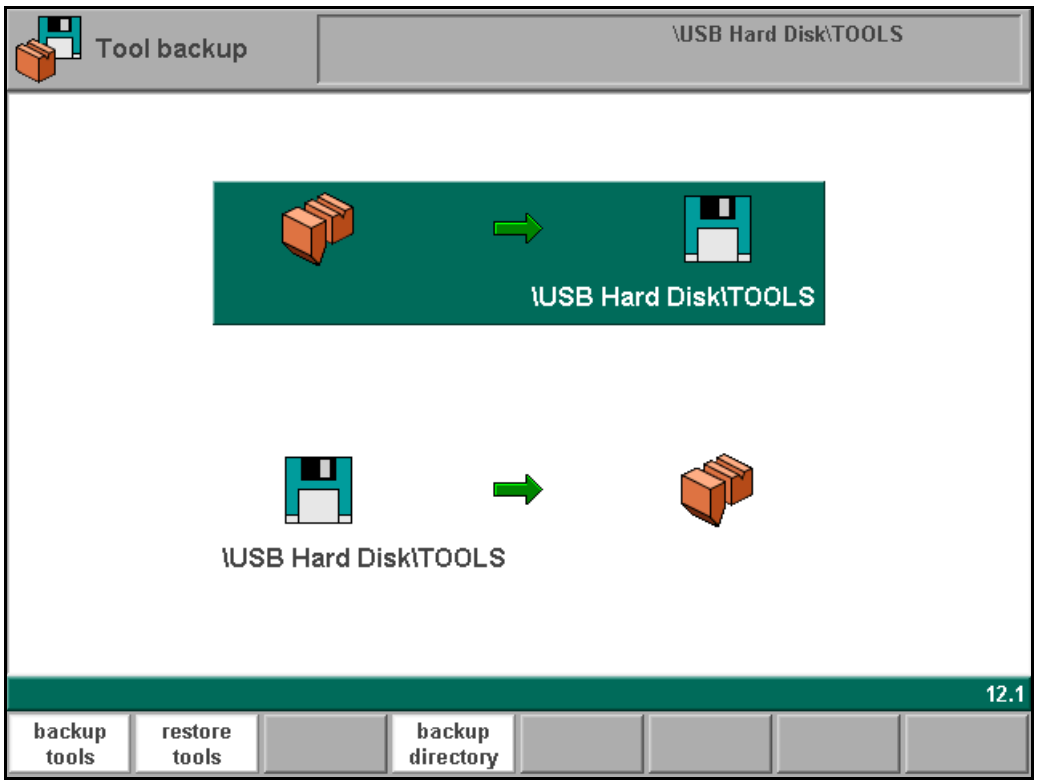
Function keys:

S8	**view**	Button to select one out of two possible viewing modes: - normal dir. - dir. expand - graph. dir.

A back-up operation is carried out unconditionally. When a restore operation is started, the control checks whether the product is already present on the control. If it is, you are asked whether to replace the existing product or to cancel the restore operation.

6.4. Tool back-up

To make a back-up of tools to disk, choose 'Tool backup' in the main menu.



6.e

With this menu a back-up of all programmed tools on the control is made: punches, dies and machine shapes. When activated, one warning is given that existing tools with the same number on the destination disk will be replaced.

Function keys:

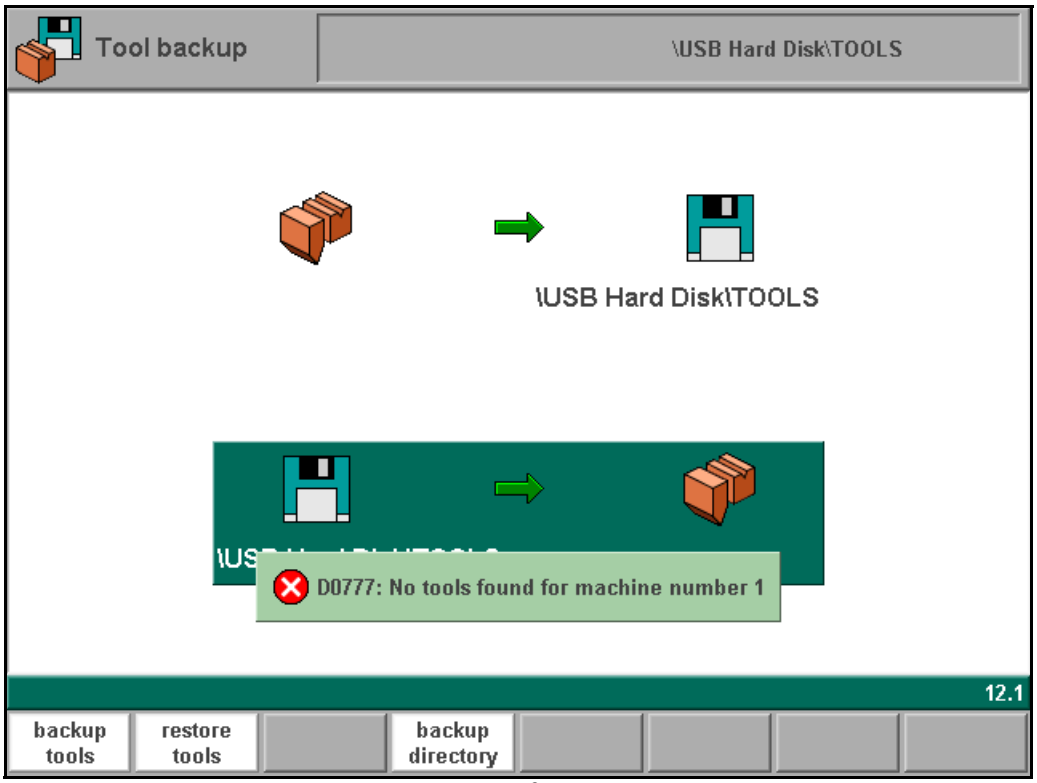
S1	**backup tools**	save all tools to disk.

Function keys:

S2 **restore tools** restore all tools from disk.

S4 **backup directory** change, make or remove backup directory

To load or save only a few tools, go to the specific menus of those tools and save or load each tool individually. This has been explained in detail in the chapter about tools.
A restore operation runs similar to a back-up operation, when it is started a warning is given that the existing tools on the control will be replaced by the tools that are imported.
Tools are linked to the control by their machine number. When a restore operation is started, the control will only import those tools, which have the same machine number as the control. If for any reason the machine number is changed, a new tool back-up operation should be executed to save the control tools with their new machine number. If this has not been done, the control will detect no tools at the next restore action.



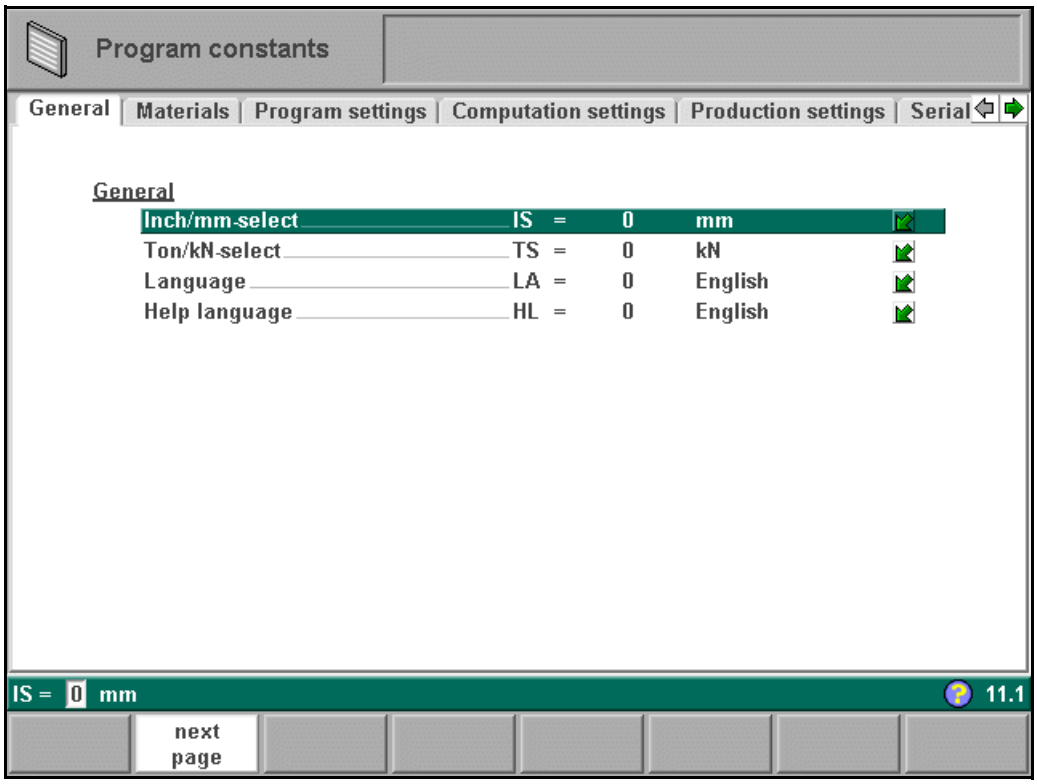
6.f

7. Program Constants

7.1. Introduction

Choose ‘Program constants’ in the main menu to program the programming constants. The program constants are divided across several pages. They are discussed in the following sections.

7.2. General



7.a

Use the keys 'next page' (S2) and 'previous page' (S1) to browse through the various pages with parameters. Use the arrow keys up/down to select individual parameters.

Inch/mm-select . . . . . .IS

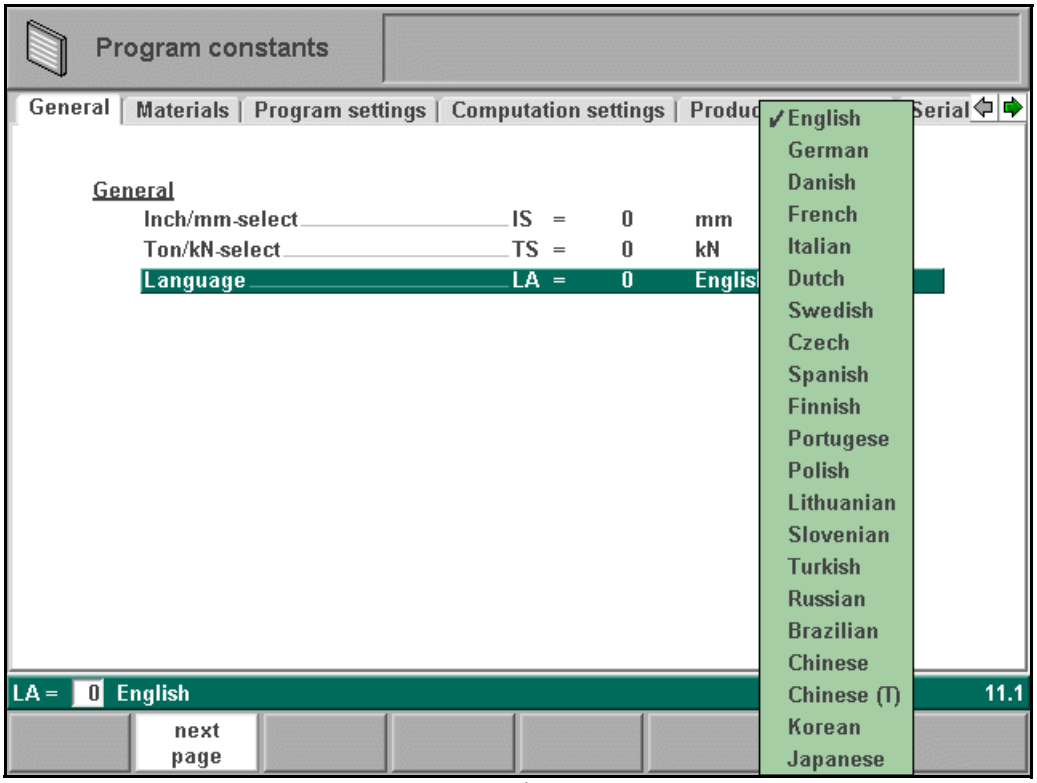
1 = dimensions in inches
0 = dimensions in millimeters

Press the key ⌂ to select the required setting.

Ton/kN select. . . . . .TS

1 = Ton
0 = kN
It is possible to select the units for all Force data to be expressed in Tons or kN.

Press the key ⌂ to select the required setting.



7.b

Language. . . . . .LA

Select the language for the user interface. The following languages are supported:

0 = English	11 = Polish
1 = German	12 = Lithuanian
2 = Danish	13= Slowenian
3 = French	14= Turkish
4 = Italian	15= Russian
5 = Dutch	16= Brazilian
6 = Swedish	17= Hungarian
7 = Czech	21= Chinese
8 = Spanish	22= Chinese Traditional
9 = Finnish	23= Korean
10= Portugese	24= Japanese

It depends on the machine configuration which languages are available.

Press the key ⌂ to select the required setting.

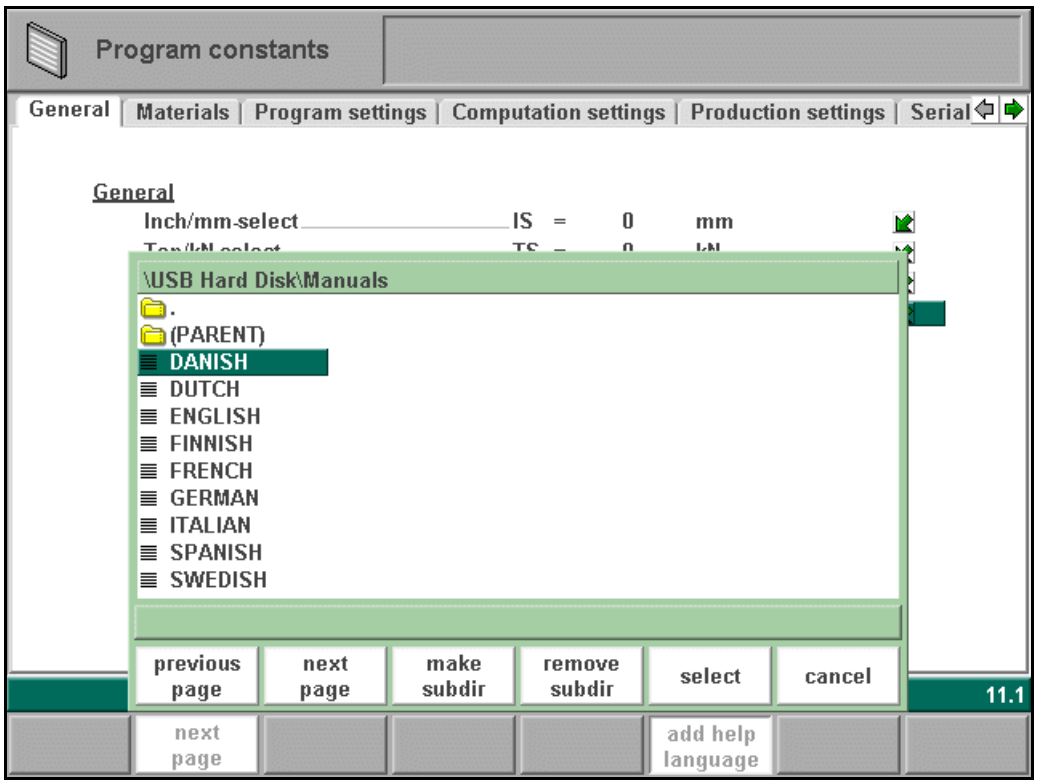
Help language . . . . . .HL

Select the language for the on-line Help function. By default, the Help language is set to the same language as the user interface. If that language is not available as on-line Help, English is chosen.

The Help function is available in the following languages:

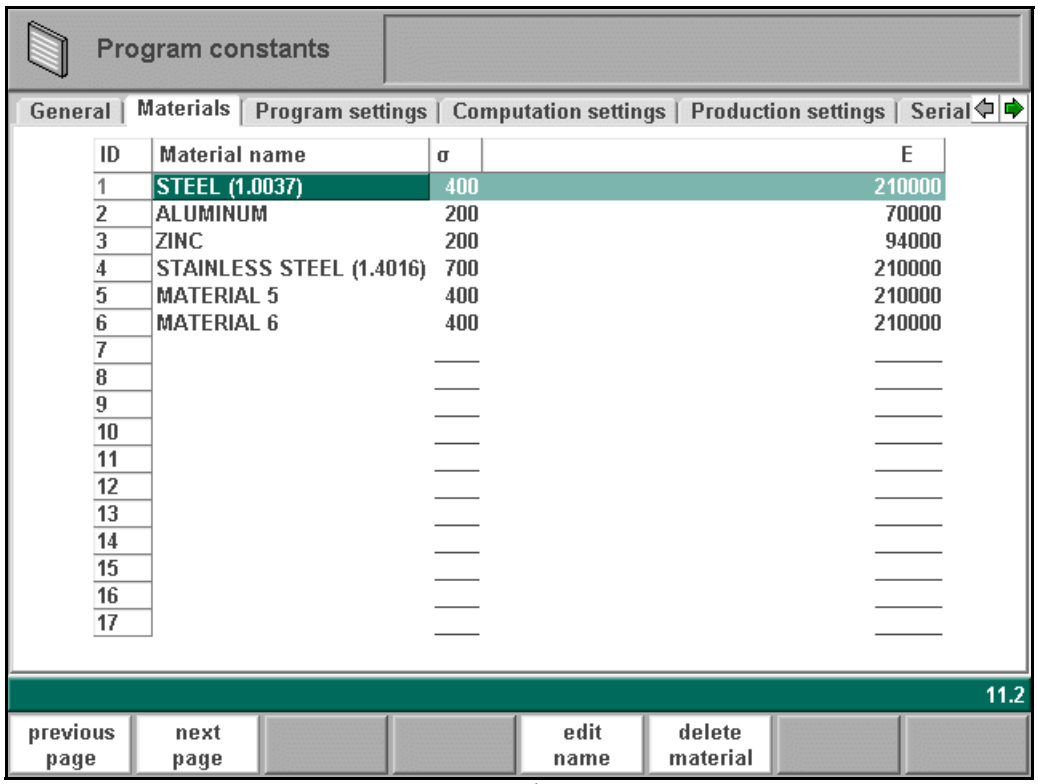
0 = English
1 = German
3 = French
4 = Italian
5 = Dutch
8 = Spanish

When the cursor is on the parameter 'Help language', the function key 'add help language' appears. With this function key you can install a new help language on the control. Make sure the required help file is present on the control disk or another accessible location (network, USB disc).



7.c

7.3. Materials



7.d

In this window, material properties can be programmed. You can edit existing materials, program new materials or delete existing materials. A maximum of 99 materials can be programmed on the control.
For each material, three properties are present and can be viewed and edited.

Material name . . . . . .NA =
Name of the material. The maximum allowed length of the material name is 25 characters, the name must begin with a character (not a numeral).

Tensile strength. . . . . . .S =
Tensile stength of the selected material.

E module . . . . . .E =
E- module of the selected material.

Function keys

S5

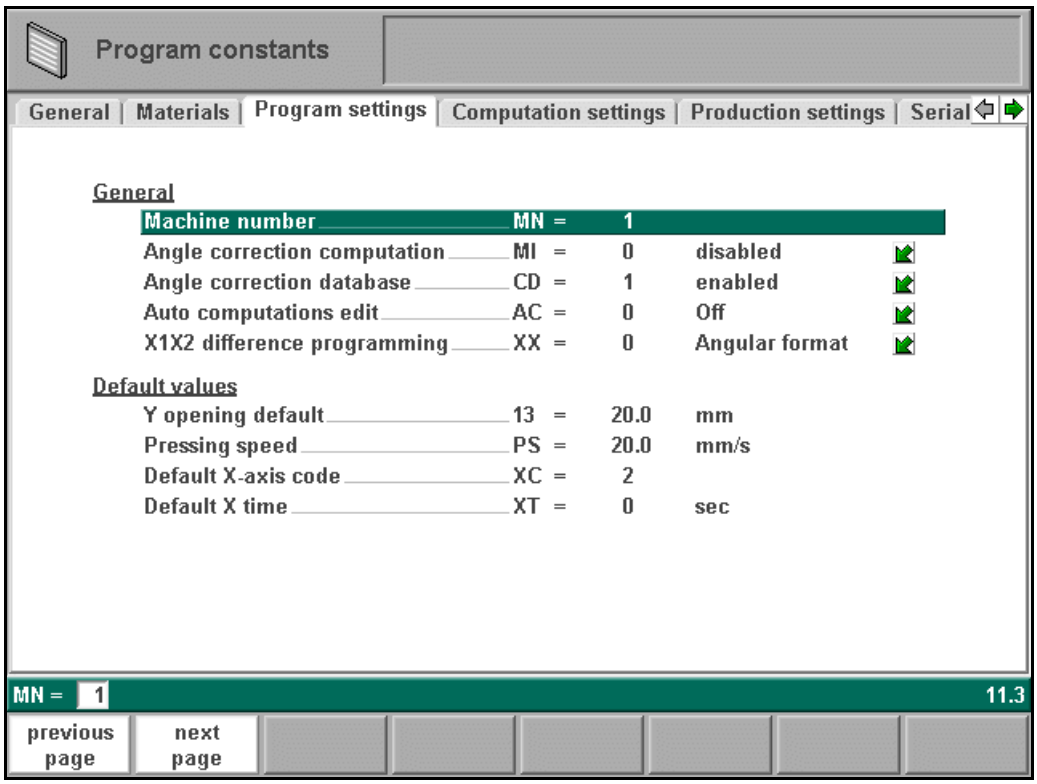
edit name To change the name of the currently selected material

Function keys

S6	**delete material**	To delete the material that is currently shown on the screen.

The materials are initially listed according to their material number, which is shown in the first column (ID).
The list can be sorted according to the different properties. If you have a mouse available, click on the title of a column. The materials will be sorted in ascending or descending order of that property.
To change an existing material, go to the relevant line and change the values as you see fit. Use the ENTER key to confirm a typed value.
To delete an existing material, move the cursor to the relevant line and press the function key 'delete material' (S6) to erase the values.
To program a new material, move the cursor to an empty space and start programming its values.

7.4. Program settings



7.e

Machine number. . . . . .MN =

When there are several bending machines in a factory, it can be useful to give the control on each machine a unique machine number.
The selected machine number will be stored with the bending program. This number will be checked when a program is read from floppy or a network. When the machine number does not match you must confirm to read it anyway or not. If you do not confirm the

question the action will be aborted.
The machine number is also stored in the filenames of tools. When the machine number is changed in this screen, you are prompted to make a new backup of your tools because the filenames of all tools have been changed according to the new machine number.

Angle correction computation. . . . . . MI =
Parameter to switch computation of angle corrections on or off.
When this parameter is set to enabled, the operator can enter the measured angle of a bend and have angle corrections calculated.
With this parameter is set whether or not the field 'measured angle' is available in the corrections window. See chapter 10 for more information about the corrections window.

Angle correction database . . . . . . CD =
Toggle function, to switch the database with angle corrections on or off.
Angle corrections are entered in production mode (automatic/step by step). These corrections are stored in the product program. Beside this, it is also possible to store these corrections in a general database with angle corrections. This way corrections that have once been entered for certain bends remain available for future use in other products.
When this setting is switched on, the control checks during production whether corrections for similar bends are present in the database. If corrections for certain bends are available, then they will be offered. On other occasions, corrections can be interpolated and offered.
The correction database is adjusted by entering new corrections during production.
When the database is enabled with this parameter, all new-entered corrections are stored in the database.
When searching for similar bends, the control searches for bends that have the same properties as the active bend. The following properties of a bend are compared:
- Material properties
- Thickness
- Die opening
- Die radius
- Punch radius
- Angle

The first five properties of a bend must be exactly the same as the active bend to start a comparison. If the angle is the same as the angle of the active bend, the correction is offered. If the angle of the active bend has a maximum difference of 10° with two adjacent bends, a correction is interpolated from these two bends. If the resulting correction has a difference of more than 5° with one of the two other bends, there will be no correction offered.

Auto-computations edit . . . . . . AC
In the ‘data edit’ mode you can change the value of the programmed parameters. This can also influence the value of other parameters. With this parameter you can choose to have the other values automatically recomputed. Program ‘auto computations edit’ to 1.
There are three cases in which automatic computation can be used.
1. In case you change the value of the parameters ‘material type’ or ‘material thickness’

then the following parameters are automatically recomputed and changed by the control:
- Pressure
- Decompression distance
- Crowning device setting

2. In case you change the value of the parameter 'length' then the following parameters are automatically recomputed and changed by the control:
- Pressure
- Decompression distance
- Crowning device setting
- Z-axis position

3. In case you change the parameter 'punch' or 'die' and the height of the new punch or new die is different AND ONLY in case the 'bend method' is 'bottoming' or 'flattening' then the following parameter is automatically recomputed and changed by the control:
- Y-axis position ('bend position')

(When the 'bend method' is 'air bend' (normal situation) the Y-axis position is calculated in the automatic mode!)

If the automatic recomputation is switched off (default situation) then these parameters stay the same. However, when you go to these parameters with the cursor keys the recomputed value is displayed in the input row. When you press 'enter' the recomputed value replaces the old value after all. So, you can choose to change the values.

X1X2 difference programming . . . . . .XX =

0 = angular format
1 = projection method

When you have two independent X-axes (X1 and X2) and the option OP-X1X2 has been installed, you can program the values for X1, X2 in a certain relation. This means that you program X1 and an angle value (angular method) or in a projection measure.
Please ask the option description from your supplier when this option has been installed.
Only available when this option is installed.

Y opening default . . . . . .13 =

Default Y-axis opening value.
The value programmed here is used as initial value for the parameter 'Y-axis opening' in the Data Preparation menu of the programming mode.

Pressing speed . . . . . .PS =

Default pressing speed, used as initial value for the parameter 'speed' in a new program.

Default X-axis code. . . . . .XC =

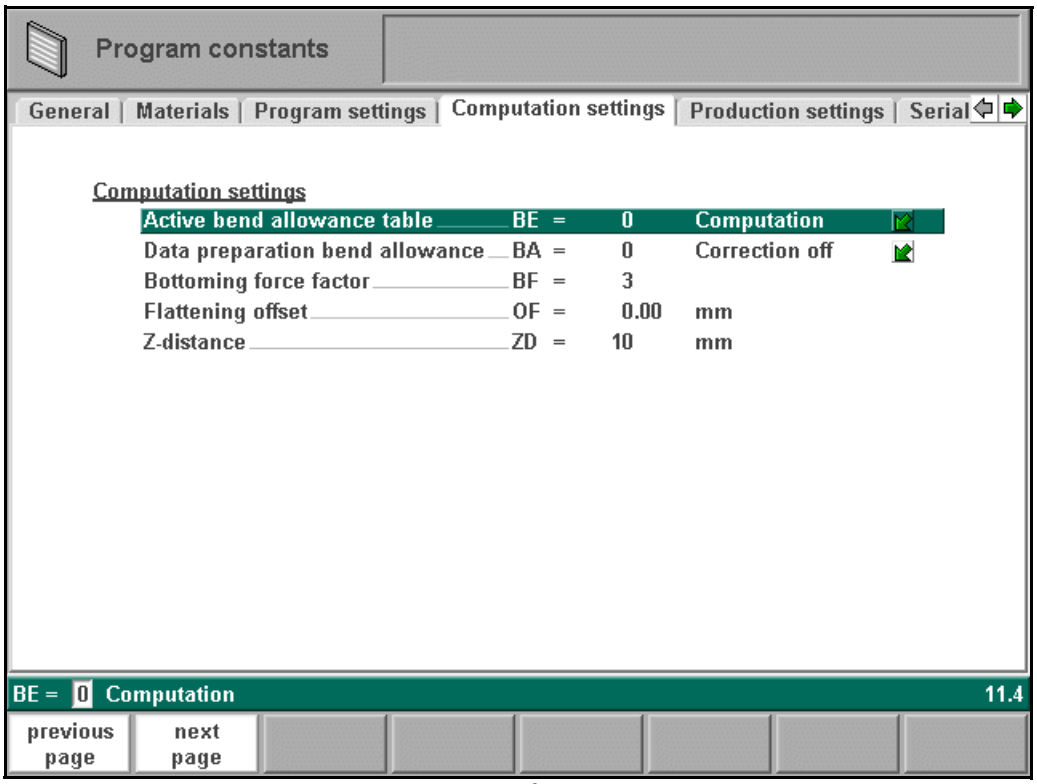
Default value for the parameter 'code' in a bend program. This parameter determines the moment of step change in a bend program. Initially the code parameter is set to the value programmed here during postprocessing and during programming.
The default value of this parameter = 2.
See chapter 'data preparation/data edit' for more information about the code parameter.

Default X time. . . . . .XT =

During the postprocessing, the waiting time of the X-axis at step change is put to zero. With this parameter you can preset a longer waiting time when needed for product han-

dling.

7.5. Computation settings



7.f

Active bend allowance table . . . . . .BE =

0 = Internal / 1 = bendallw.tab

Bend-allowance: correction of the X-axis due to plate shortening after bending. With this parameter the method for bend-allowance calculation is chosen. 'Internal' means the standard formula of the control is used to calculate the bend-allowance. 'Bendallw.tab' means a bend-allowance table with correction values can be used. The choice 'Bendallw.tab' is only available if a bend-allowance table is present on the control.
If a table has been chosen, the name of that table (if it has a name) is shown on the screen.

Press the key ⌂ to select the required setting.

Data preparation bend allowance. . . . . .BA =

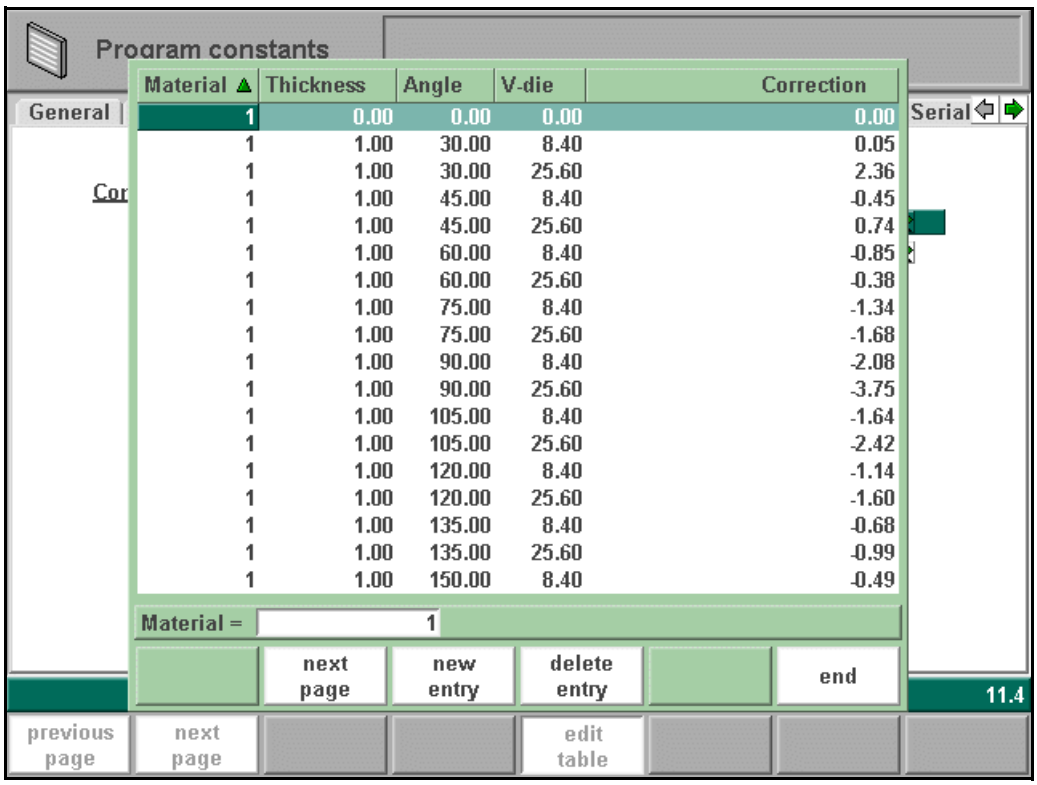
0 = correction off / 1 = correction on

With this parameter you can choose whether or not you wish to have programmed values corrected for bend-allowance. This on/off setting only refers to corrections during product programming in the menu 'data preparation'. If a numeric program has been entered with corrections on, the axis corrections are calculated and stored in the program. These corrections can be viewed and edited in production mode (see 'Automatic mode').

Press the key ⌂ to select the required setting.

This choice has no effect on the postprocess function in the drawing mode. When the control

uses the postprocess command to calculate a CNC program from a drawing with bendsequence, the control will allways take bend-allowance into account.
When a bend-allowance table is present and the cursor is on the parameter ‘active bend allowance table’ then an extra function key is available: ‘edit table’. With this softkey the table can be modified to meet user requirements. The table appears in a new window, with its own set of function keys.



7.g

Each line contains a table entry, with several parameters. In this screen, the following function keys are available:

Funtion keys:

S4 **new entry** Enter a new line in the table.

S5 **delete entry** Delete the current line.

Use the arrow keys to scroll to the field you wish to change. Then type the new value and press ENTER. It is not possible to create a table through this menu. Only when a table has been loaded into the control is it possible to edit its contents.
For more information about bend-allowance tables, we refer to the Delem manual of the bend-allowance table.

Bottoming Force Factor. . . . . .BF =

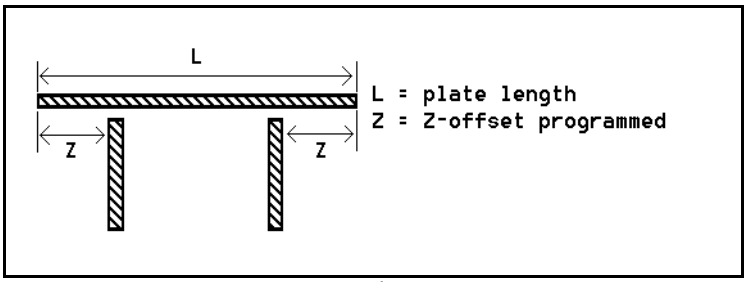
The pressure needed for an airbend multiplied by this factor in order to obtain the coining pressure.

Flattening offset . . . . . .OF =

The calculated depth position for the Y-axis beam in case of flattening your product depends on the programmed flatten height of your special tool and the sheet thickness (x2) and the here programmed offset to realise a flattened product as you require.

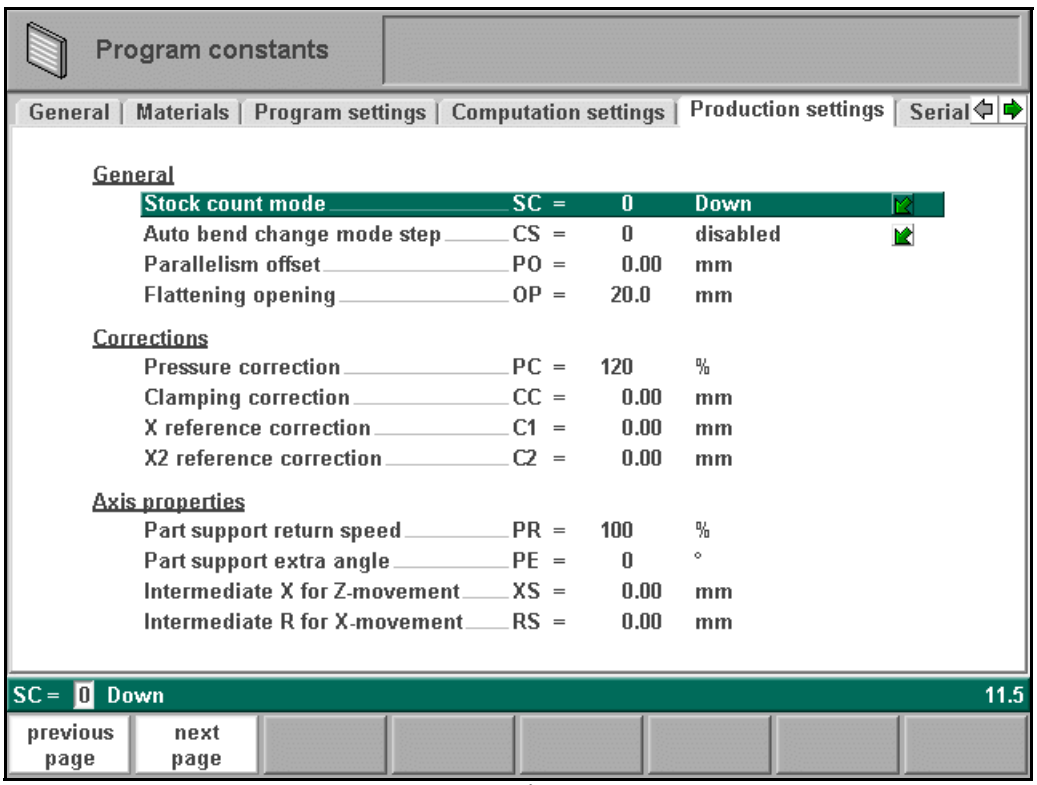
Z-distance . . . . . .ZD =

When automatic Z-axes are installed, the distance of the fingers on the backgauge are automatically calculated with respect to the end of the plate.



7.h

7.6. Production settings



7.i

Stock count mode . . . . . .SC=

Setting for the stock counter in production mode, to have the stock counter (product counter) count up or down.
When downcounting is selected, the stock counter in production mode is decremented after each press cycle. When the counter has reached zero, the control is stopped. On the next start action, the stock counting value is reset to its original value.
When upcounting is selected, the counter is incremented after each press cycle.
Downcounting can be useful if a pre-planned quota must be produced. Upcounting could be used to give a report on production progress.

Press the key  to select the required setting.

Auto bend change mode step. . . . . .CS =

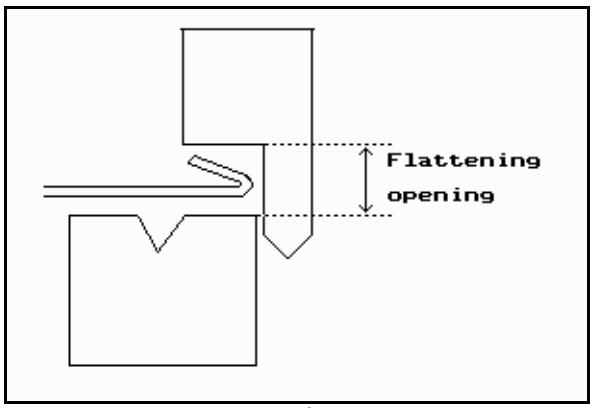
This parameter can be used to have automatic step change in the bending process with the "step by step"-mode. To be programmed 0 or 1.
When programmed 0:
No automatic step change (next bending parameters active) will take place. To perform the next bending you must select the new bending and press the start button.
When programmed 1:
The next bending parameters are loaded automatically but the axes will start positioning after the start button has been pressed.

Parallelism offset . . . . . .PO =

An overal parallelism, valid for the complete Y-axis stroke, can be programmed with this parameter. The programmed value will be checked against the maximum allowed value during production. The parallelism which can be programmed for each bending (Y2) is only active below the clamping point. The parallelism below the clamping point is the sum of the two parameters (Y2 + Par. offset).

Flattening opening . . . . . .OP =

Depending on the construction of your machine you can program here an openings position for your punch at which position you can put in your product to flatten the particular bend. A second mute output will become active which can stop the beam depending on the machine interfacing. See figure below. The opening position will also take 2x sheet thickness into account.



7.j

Pressure correction . . . . . .PC =

Percentage of calculated tonnages which actually controls the pressure valve.

Clamping correction . . . . . .CC =

The position of the beam at which the plate is clamped, is calculated. In order to have a firm clamped plate it is possible to offset the calculated pinch point with the value here programmed. A positive value will result in a deeper position, a negative value in a higher position of the beam.

X-reference correction . . . . . .C1 =

When the actual, mechanical X-axis position is not corresponding with the displayed value than is it possible to correct the position with this parameter. Program the calculated difference.

Example:

- When the programmed and displayed value = 250 and the actual, mechanical position value = 252 the XR parameter = -2.
- When the programmed and displayed value = 250 and the actual, mechanical position value = 248 the XR parameter = +2.

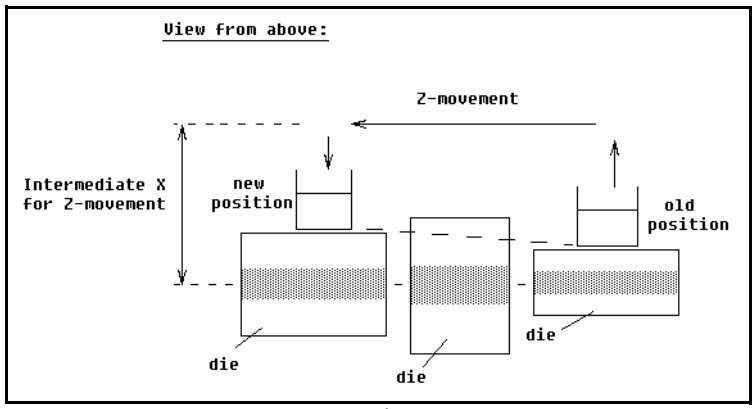
X2-reference correction C2

Same function as the previous parameter but now for the X2-axis.

Intermediate X for Z-movement. . . . . . .XS=

Temporary safe value for the X-axis, to avoid collision as a result of movement along the Z-axis. With this parameter a standard safety zone for the X-axis is defined, which is valid for all programs. The value 0 disables this functionality. This parameter should not be confused with the parameter 'X-safety' of each die.

This parameter is especially useful when several dies of different sizes are placed on the machine. In that situation, this intermediate X-value should be larger than the safety zone of the largest die that is installed.



7.k

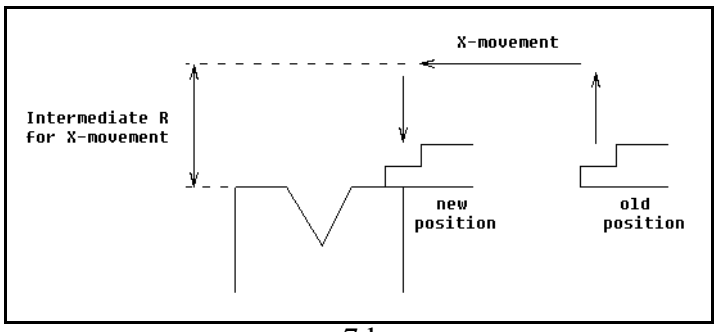
When the backgauge has to move to a different Z-position, it is checked whether the current X-position is safe. We can distinguish the following situations:

- Old X-axis position as well as new position outside the zone: X- and Z-axis movements happen at the same time, no change.
- Old X-axis position outside, new position inside the zone: backgauge is positioned on Z-axes first, the X-movement starts when the Z-axes are in position.
- Old X-axis position inside, new position outside the zone: backgauge starts along X-axis, Z-movement

is started when X-axis is outside the zone.
- Old X-axis position as well as new position inside the zone: backgauge moves to the intermediate X-axis position, then the Z-movement is started. When the Z-axes are in position the X-movement is started to move the backgauge to its new position (see figure 8.j.).

Intermediate R for X-movement. . . . . .RS=

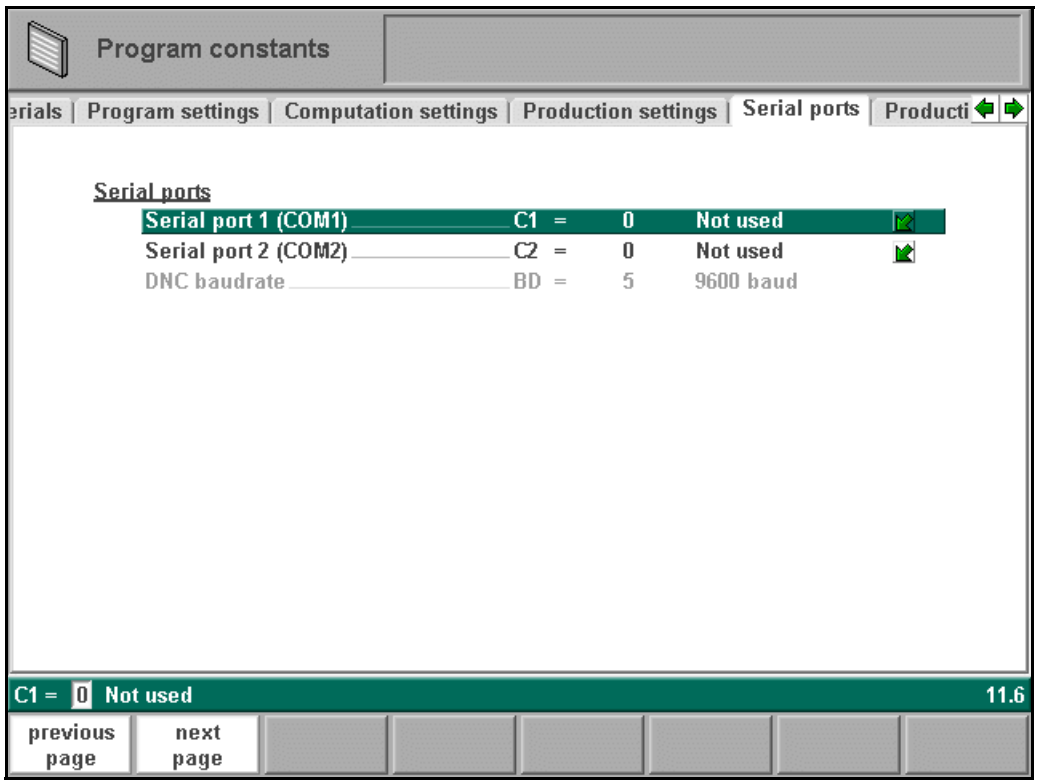
Temporary position for the R-axis, to avoid collision as a result of movement of the X-axis. The value 0 disables this functionality. When programmed not equal to zero this position will be active when the X-axis has to move inside the safety zone of the die.



7.1

The sequence will be as follows:
- The R-axis is moved to the intermediate position;
- then the X-axis is moved to its position;
- finally the R-axis is moved to its position.

7.7. Serial ports



7.m

Serial port 1/2 . . . . . .C1/C2

The control is equipped with two serial ports (RS-232). With these parameters you can assign certain applications to these serial ports. To change these settings, contact your machine supplier. The following settings are available:

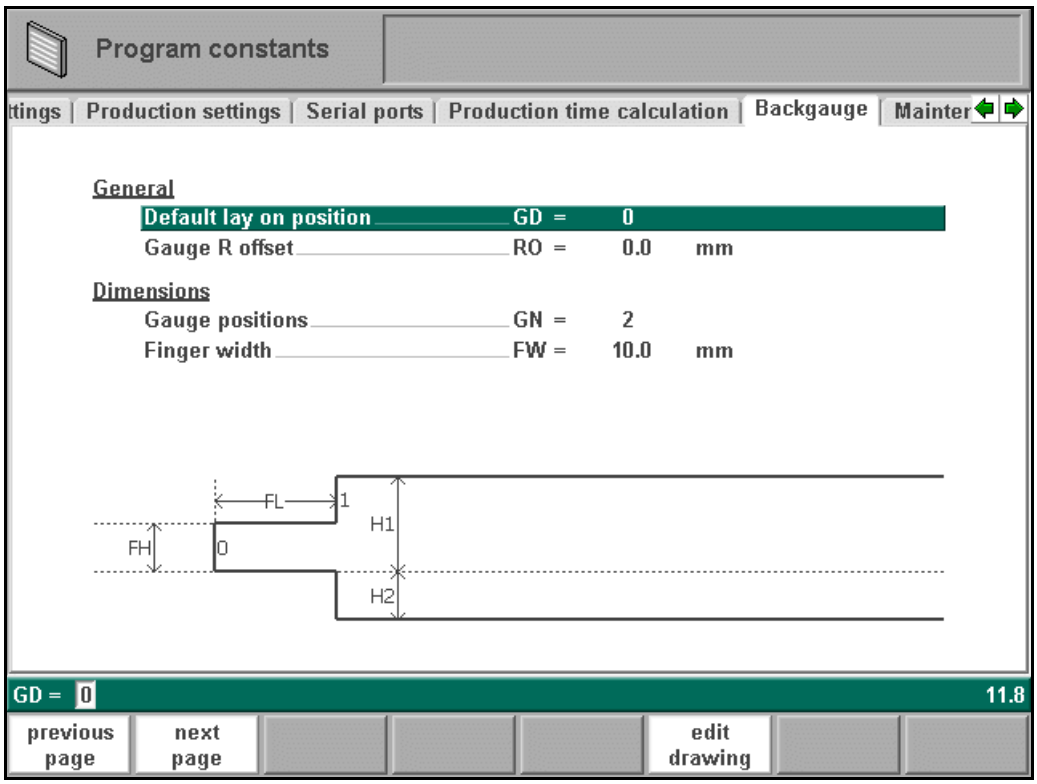
DNC

The serial port is used for DNC operation, which means the control can be accessed by a computer through a serial connection according to RS-232.

Other settings are machine-dependent. If a serial port is already programmed to another setting, it is not available for DNC anymore.

7.8. Backgauge dimensions

With these finger dimensions the R-axis movement and work/backgauge collision can be computed.



7.n

Gauge positions. . . . . .GN =

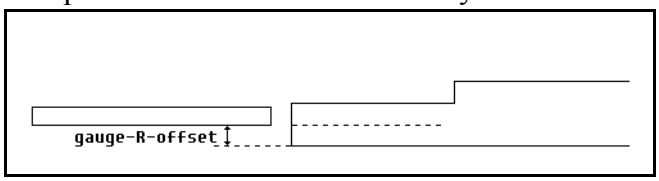
The number of possible gauge positions (max. 4). When this parameter is changed, a new pop-up window with finger geometry appears. There the finger dimensions can be programmed.

Default lay on position . . . . . .GD =

Default gauge lay on position when lay on is selected during the graphical bendsequence programming.
Default value=0, not lay on.

Gauge R offset. . . . . .RO =

An offset value for the R-axis can be set if the backgauge is positioned against the sheet edge and the X-axis position is outside the die safety zone.

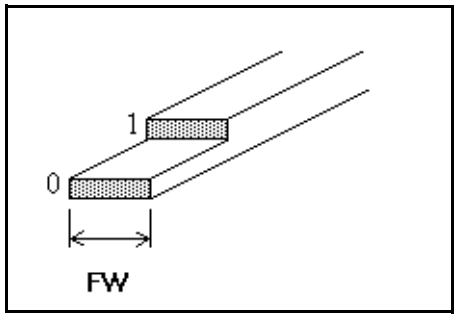


7.o

A negative value gives a lower backgauge position. This offset is only valid for gauge position 0.

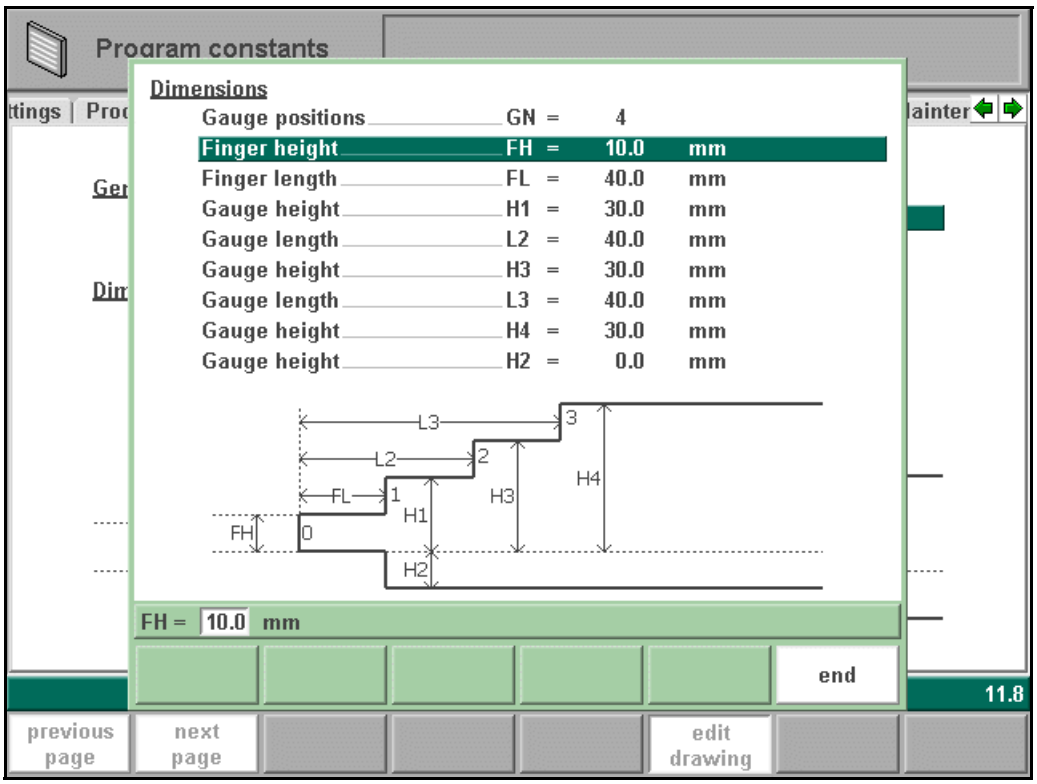
Finger width . . . . . .FW

The width of the backgauge finger.



7.p

When the function key ‘edit drawing’ has been pressed, a new window appears wherein the dimensions of the backgauge finger can be programmed.



7.q

The following parameters describe the dimensions of the backgauge and the lay-on positions. The number of parameters that has to be programmed depends on the number of gauge positions. If more gauge positions are programmed, more parameters are offered. The illustration at the bottom of the screen will change according to the number of gauge positions.

Finger height. . . . . . .FH

The height (or thickness) of the first backgauge finger tip.

Gauge height H1/H2/H3/H4
The height of the different finger levels.

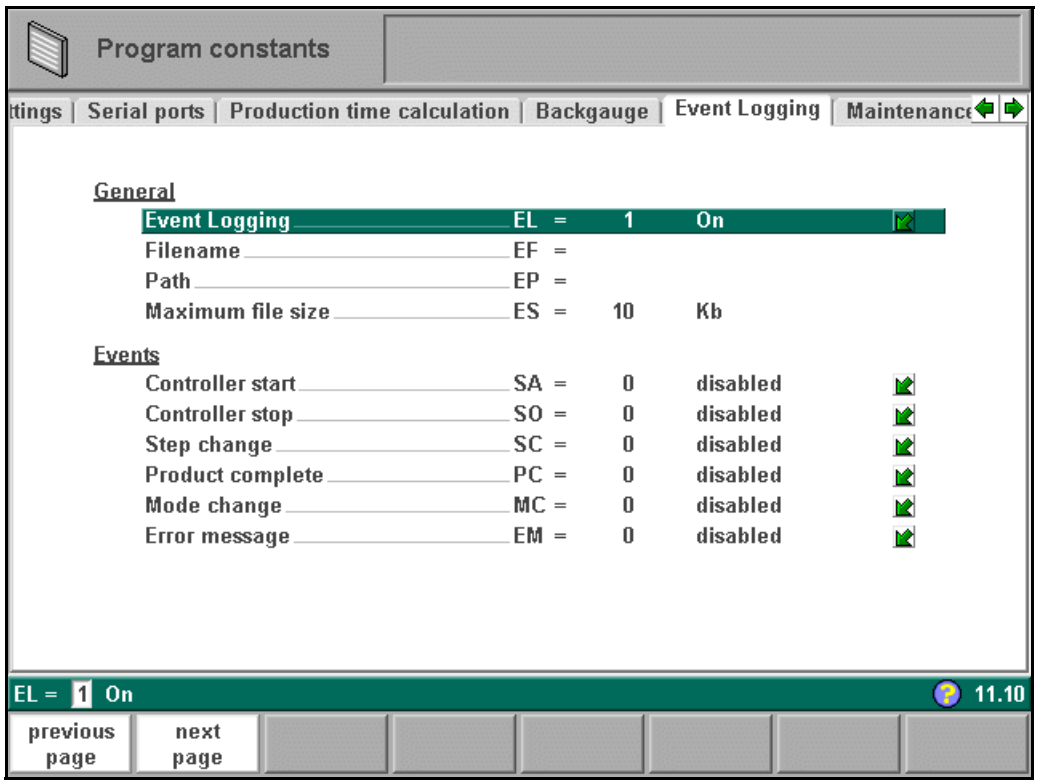
Finger length . . . . . .FL
The length of the first finger.

Gauge length L2/L3
The length of additional lay-on levels.

7.9. Event logging

7.9.1. Parameters

With this feature it is possible to register certain events on the control. These events are stored in text files that can be studied afterwards. The log files that are created with this feature can be used for production management.
This page contains all parameters for the event logging function. The events that are logged can be assigned with the parameters on this page.



7.r

Event logging . . . . . .EL
To switch the event logging function on or off.

Filename . . . . . .EF
The name for the log file. Enter the required name, the extension '.txt' will be added to the filename automatically.

Path . . . . . .EP
The folder (directory) where the log file will be stored. This path can be on the internal memory of the DA-control (\Hard disk), a USB memory stick or even a network drive, if connected.

Maximum file size . . . . . .ES
The maximum size of the log file in Kilobytes. If the current log file reaches this size it will be closed and renamed automatically. Immediately a new file with the programmed

name is created and opened to continue logging.

The following parameters show the events that can be logged. With each parameter it is possible to program if the event should be logged or not. At least one of these parameters should be switched on for logging.

Controller start . . . . . .SA
Log event: the control has been started.

Controller stop . . . . . .SO
Log event: the control has been stopped.

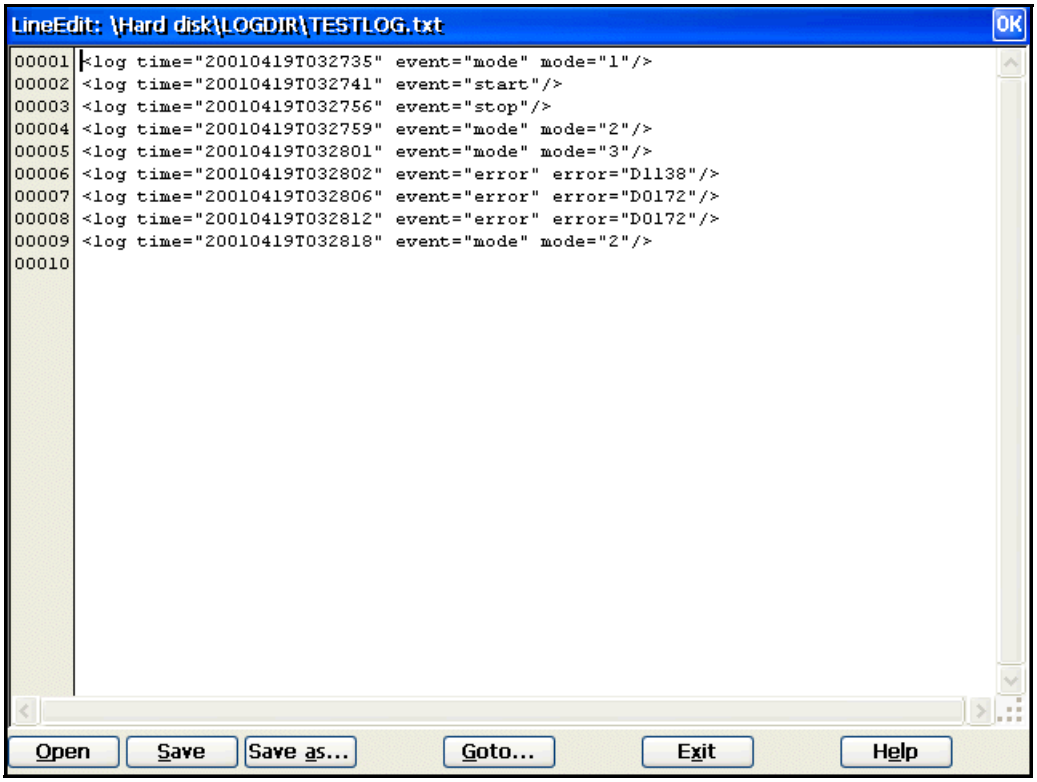
Step change . . . . . .SC
Log event: the control has switched to the next bend step.

Product complete . . . . . .PC
Log event: the last step of a bend program has been finished.

Mode change . . . . . .MC
Log event: a different mode has been selected.

Error message . . . . . .EM
Log event: an error message has occurred.

7.9.2. Explanation



7.s

One line in the log file can look as follows:

<log time=20060407T160712 event="mode" mode="1"/>

or:

<log time=20060407T160712 event="error" error="D0089"/>

Each line is one event, with a few possible attributes. The time is always listed, followed by the nature of the event.
The time is listed as follows:

time=<date>T<time>

The date is shown in the order year-month-date, the time is shown in the order hours-minutes-seconds. These two fields are separated by the T character.
The nature of the event is indicated by a keyword. Depending on its nature, one or more attributes are listed.

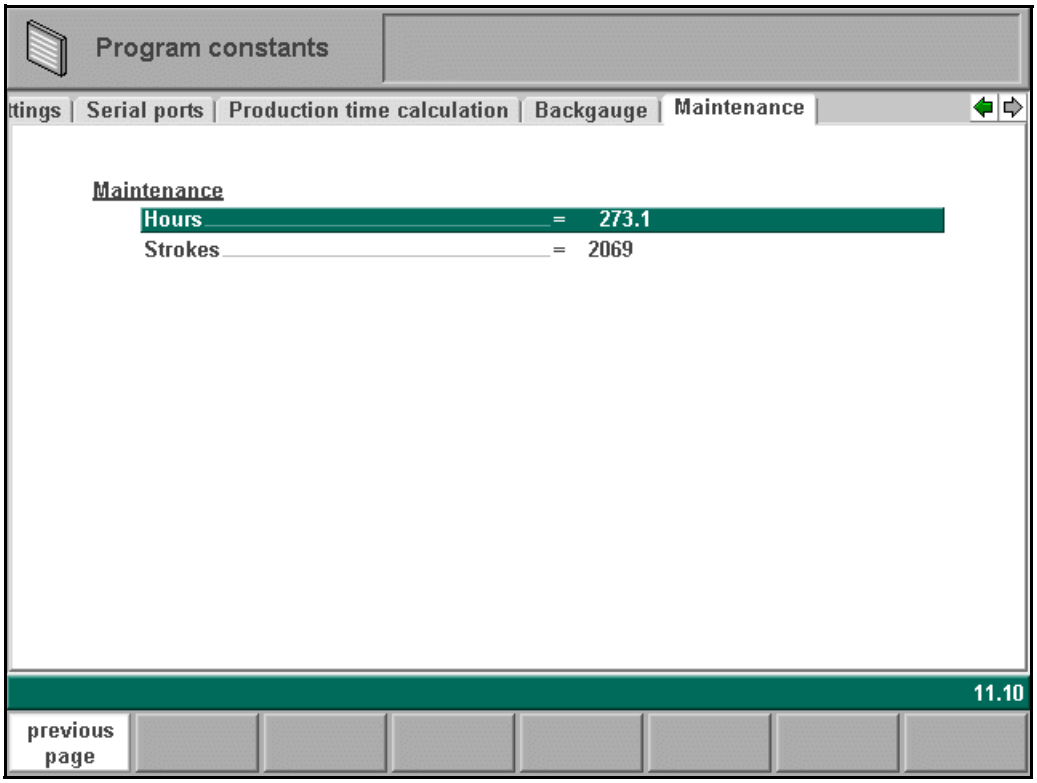
Event type	Keyword	Possible attributes
Mode change	mode	mode number: 1 = manual 2 = programming 3 = automatic 4 = step by step
Step change	step	product number, step number
Control start	start	product number, step number and stock counter
Control stop	stop	product number, step number and stock counter
Product complete	prod_rdy	product number, stock counter
Error message	error	error number

Each attribute is indicated by a keyword. The following keywords are possible:

Attribute	Keyword
Mode number	mode
Bend step number	step
Product number	prod
Stock count number	stock
Error number	error

One line in the log file takes approximately 50 bytes, so with a file size of 10 Kilobytes it is possible to log about 200 events. This can vary, since the description length of one event can vary.
If a large file is necessary to record a lot of events, it could be wise to use an external USB key or a network location to store the file. The maximum file size supported by the control is 1 Megabyte (1000 Kilobytes).

7.10. Maintenance



7.t

Hours . . . . . .

The number of hours the machine is running.

Strokes . . . . . .

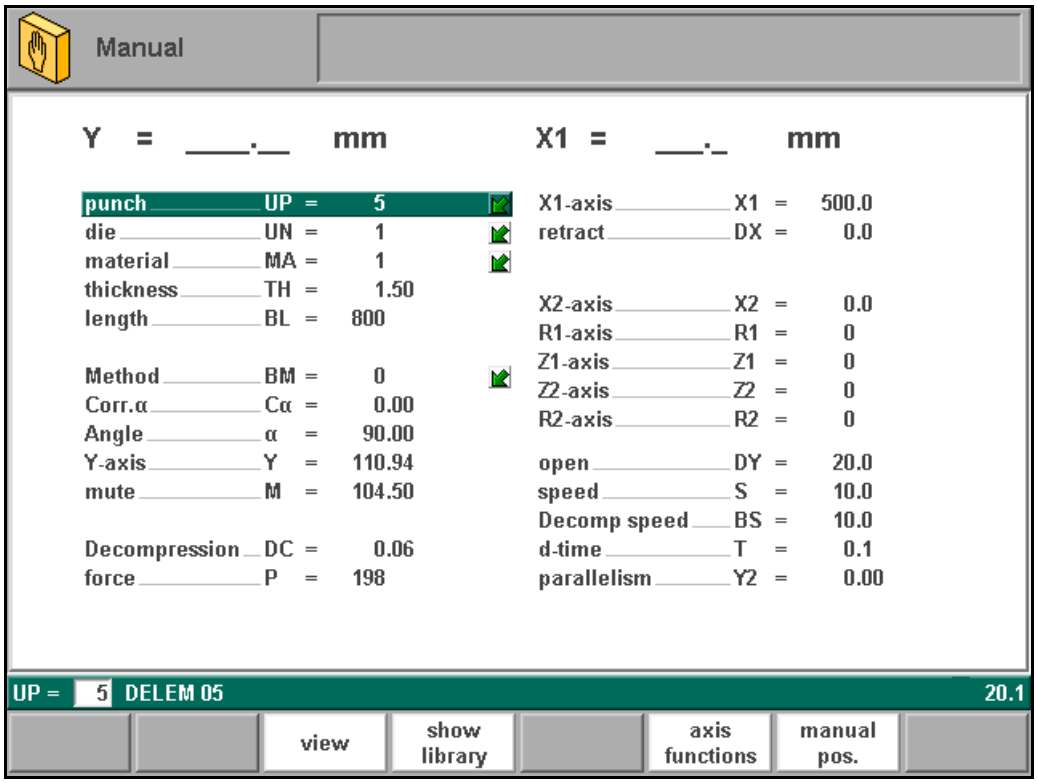
The number of strokes the pressbeam has executed.

8. Manual mode

8.1. Introduction

Manual mode By pushing this key the CNC is in manual mode.

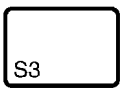
In manual mode you program the parameters for one bending. This mode is useful for testing and for calibration.



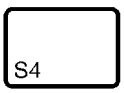
8.a

All the parameters can be programmed independently of the programs in memory.

Function keys



view Button to select one out of three possible viewing modes:
- axes
- aux. axes
- zoomed values



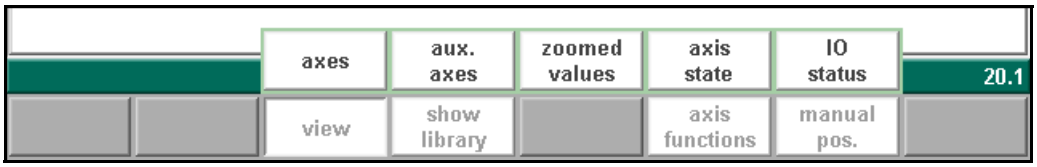
show library Open a window with an overview of the tool library. From there, a tool can be selected.

Function keys

S6 — **axis functions** — To change speed and retract distance of available axes in the current bend. This function is machine-dependant and is described in detail in the chapter ‘Data preparation’.

S7 — **manual pos** — Start the function to move axes manually with the hand-wheel. See section 9.2.

When the function key ‘View’ has been pressed a new, temporary button bar appears with additional softkeys:



Function keys:

S3 — **axes** — Switch to default parameter view.

S4 — **aux. axes** — Switch to second view with parameters of auxiliary axes.

S5 — **zoomed values** — Switch to view with axes values.

S6 — **axis state** — Switch to view with state of the available axes. Available for service purposes.

S7 — **IO status** — Switch to view with status of system I/O. Available for service purposes.

• Parameter Explanation

Punch . . . . . .UP
Number of selected punch in the library.
Press the key ⍓ to select the required setting.

Die . . . . . .UN
Number of selected die in the library.
Press the key ⍓ to select the required setting.

Material . . . . . .MA
Selection of one of the programmed materials, which are used to calculate the bending depths. The control contains 4 preprogrammed materials. In total, 99 materials can be programmed on the control. See the chapter about programming constants how to program materials.
Press the key ⍓ to select the required setting.

Thickness . . . . . .TH
Thickness of the plate in millimeters.

Length . . . . . .BL
Length of the plate between the tools in millimeters.

Method . . . . . .BM
Select the required bending method. The control supports 4 methods:
Air bend
Bottoming
Flattening
Flattening & bottoming
Press the key ⍓ to select the required setting. See chapter 3 for more information about possible bend methods.

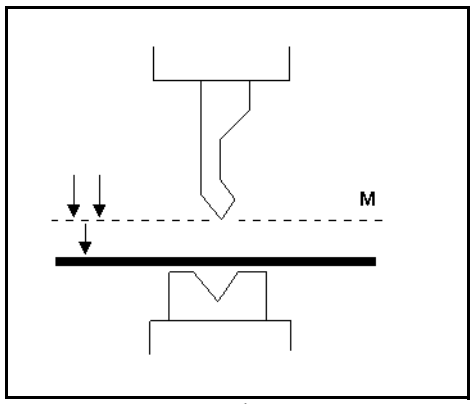
Corr. α . . . . . .Cα
Correction on angle to bend
The angle correction should be entered as follows:
- Programmed value of 90 degrees.
- Measured value of 92 degrees.
Then it is required to program Corr._ with -2.
- Programmed value of 90 degrees.
- Measured value of 88 degrees.
Then it is required to program Corr._ with +2.

Angle . . . . . .α
Angle to bend

Y-axis . . . . . .Y
The programmed or calculated Y-axis value to realise a certain angle.

Mute . . . . . .M

Sequence point where the Y-axis is switched from fast closing speed to pressing speed. It is programmed here as a Y-axis position value.



8.b

Decompression . . . . . .BP

Decompression stroke after the bending to release the working pressure. The same parameter as DC in the data preparation mode.

Force . . . . . .P

Max. adjusted tonnage during pressing.

X-Axis X

The programmed value in X- direction to get a certain backgauge position.

Retract. . . . . . .DX

Retract distance of backgauge during the bending. The "backgauge retract" is started at the pinching point of the plate.

Auxiliary axis . . . . . .R

If you have one or more auxiliary axes (for instance a R-axis, Z-axis or part support) the parameters of these axes appear here. When you have a R1-axis and a R2 axis the programmed R1 value is automatically copied to the R2-axis value. The R2-axis value can, if necessary, be changed afterwards.

Open . . . . . .DY

This parameter results in a certain gap opening between the punch and the die after the bend. A positive value is the gap opening above Mute, a negative value below Mute. When you want to limit the handling time for the product you can program a small positive or a negative value.

Speed . . . . . .S

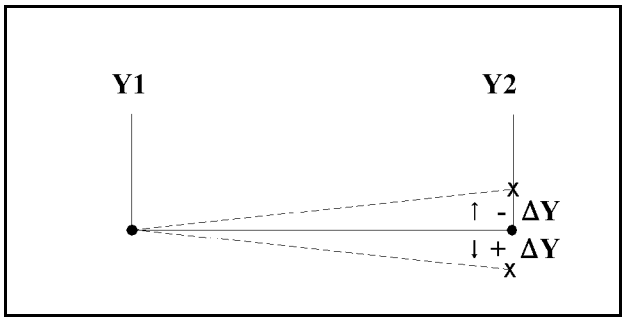
Pressing speed

Decomp speed . . . . . .BS

The decompression pressure speed is the programmable speed of the beam during the decompression stroke.

D-time . . . . . .T
Dwell time of punch at the bending point.

Parallelism. Y2
Difference of left- and right hand side cylinder (Y1 and Y2). When positive right hand side lower. When negative right hand side higher. The programmed value is active below the clamping point.

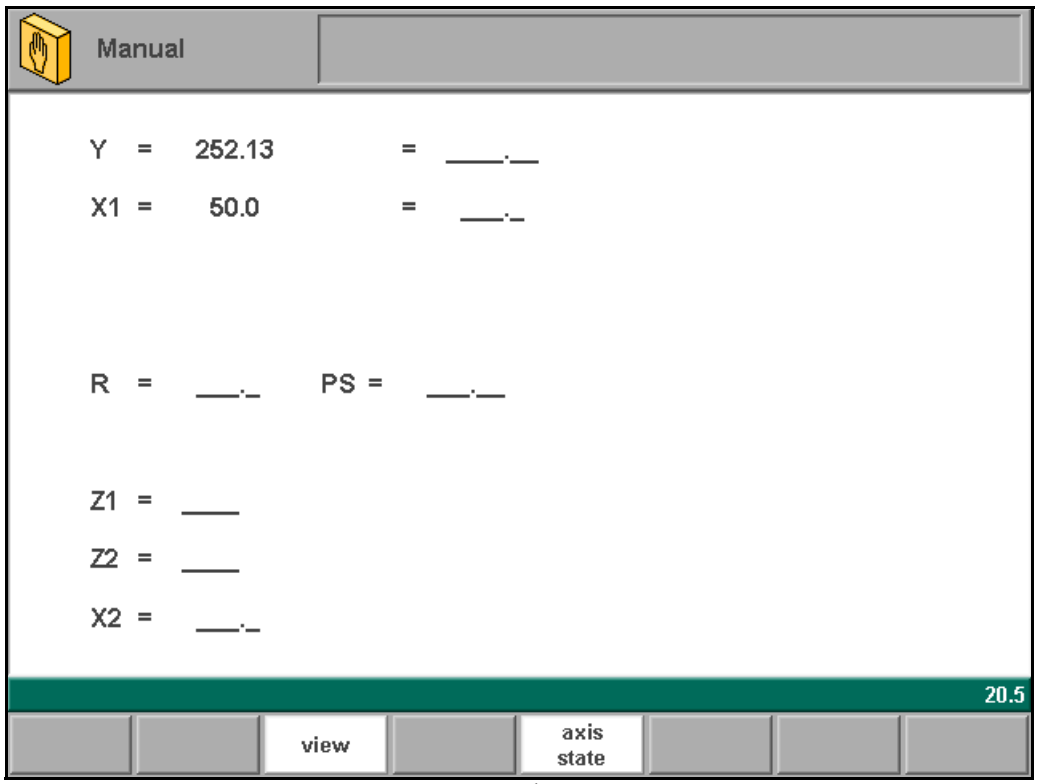


8.c

After pushing the start button all the programmed parameters are active.

- **Zoom function**

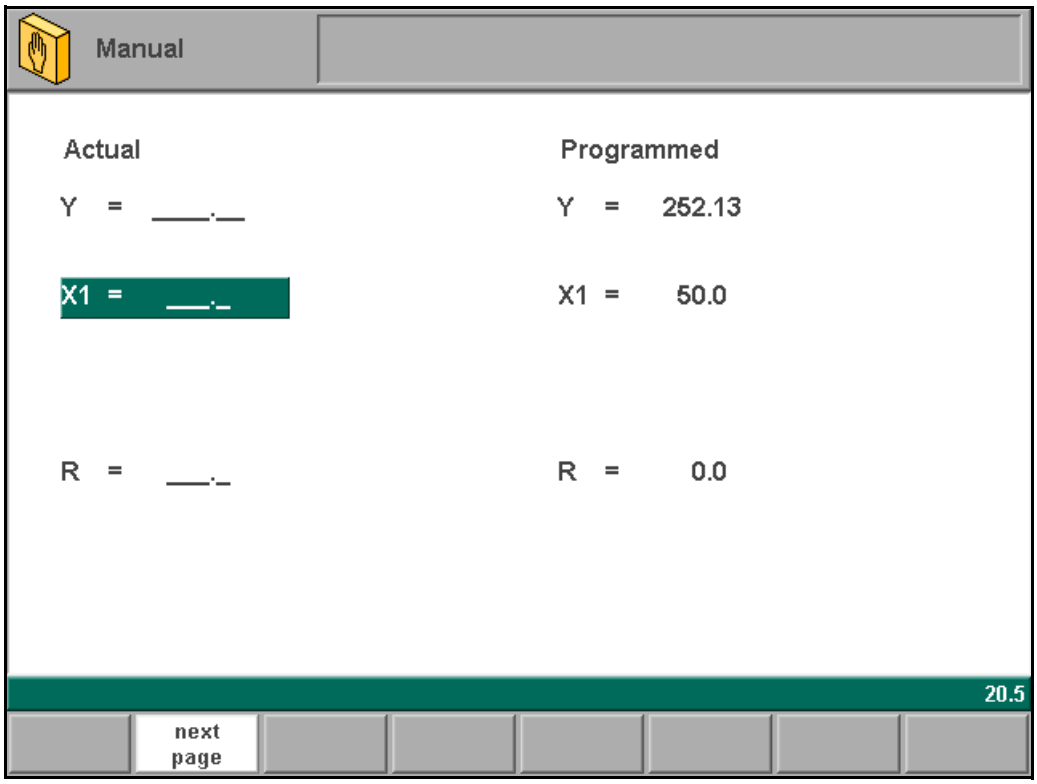
When the function key ‘zoomed values’ has been pressed, the control switches to a new view with only axes values on the screen.



8.d

8.2. Manual operation of the axes

It is possible to move an axis by turning the handwheel on the front panel of the control. After pressing the function key ‘manual pos’ in the main screen of manual mode, the following screen appears:



8.e

Put the cursor bar on the axis you wish to move with the handwheel. Then simply turn the handwheel and watch the movement of the intended axis. The procedure for moving the axis depends on the axis you wish to move.

- Auxiliary axes:

With the handwheel the backgauge can be positioned manually. This operation is only possible in "Stop" + "Manual Mode".
First you select the respective backgauge axis with help of the "Manual positioning"-key, so that you will see the cursor bar at the required axis. Then you can move the axis with help of the wheel.

- Y-axis:

With the handwheel the punch can be positioned manually in the same way as for the auxiliary axes. This operation is only possible in "Start" + " Manual Mode". Furthermore, the following conditions must be met:

- The ‘adjust’ function must be active, indicated on the screen by "Adjust" in the lower righthand corner.

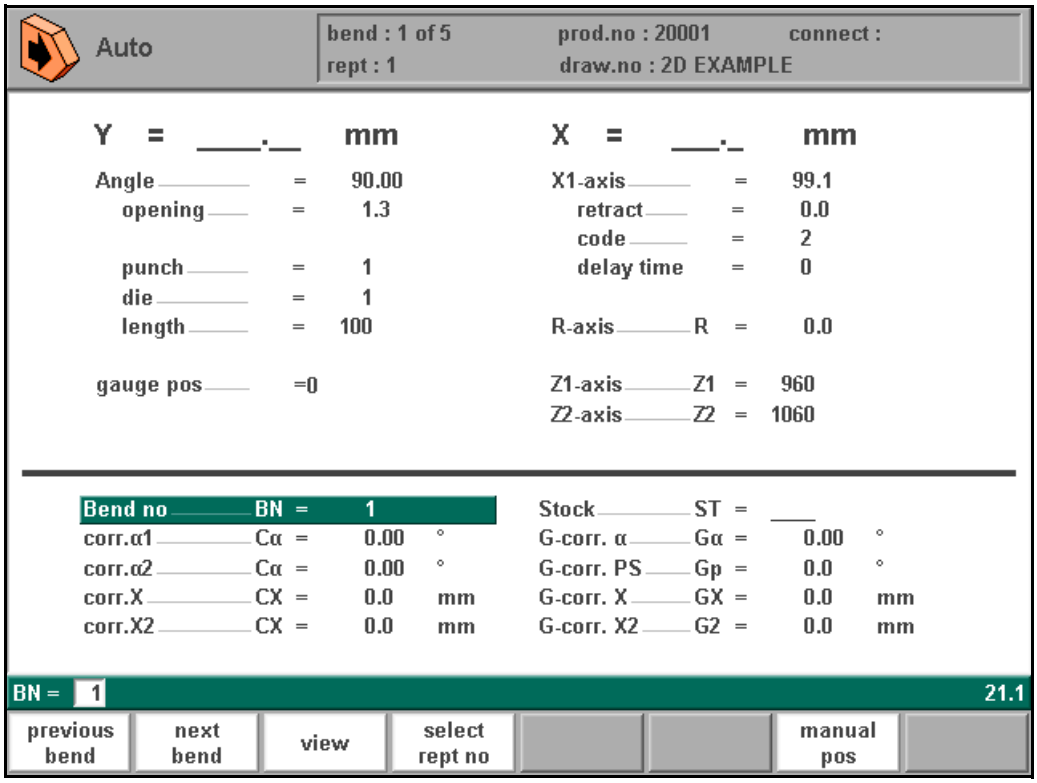
- The Y-axis must be below mute-point.
- A pressing command must be given to the CNC.

• To teach

You can teach an axis a correct position within the manual positioning mode. When you have moved an axis to a certain position with the handwheel, you may want to store this position. To do so, press the ENTER key within this screen.
The actual axis value (left side) will appear in the programmed axis field (right side).

9. Automatic / step by step mode

9.1. Introduction



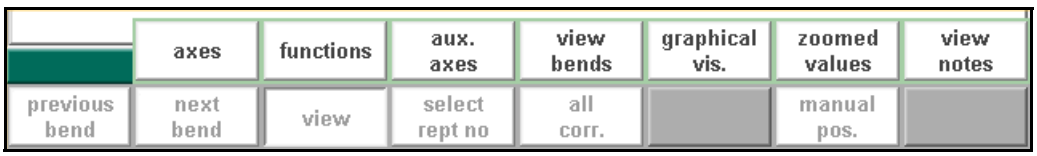
9.a

The automatic mode executes the program (programming mode) automatically bend by bend after pushing the ‘start’-key.When a new bending program is selected you must check your tools and tool positions in your machine. This is also indicated with a ‘check tools’ warning message when you enter the automatic mode.
In the header information is displayed on the number of bends, the repetition of a bend, the product number, the drawing number and the connected program (the latter two do not have to be present).
Above the horizontal line the programmed and computed parameters are displayed. These parameters are divided over two screens: ‘functions’ and ‘axes’. The parameters below the horizontal line can be programmed.

Function keys:

S1	**previous bend**	Switch to previous bend of the program.
S2	**next bend**	Switch to next bend of the program.
S3	**view**	Button to activate pop-up bar with additional functions.
S4	**select rept no**	Switch between display of bend number or the number of repetitions of this bend.
S5	**all corr.**	Open window to edit corrections of all bends of the product. See section about corrections.
S7	**manual pos**	Start function to move an axis manually with the hand-wheel and change the axis positions with the teach function like in manual mode.

When the function key ‘View’ has been pressed a new, a temporary button bar appears with additional softkeys:

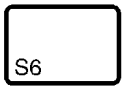


9.b

Function keys:

S2	**axes**	Switch to default view of automatic mode.
S3	**functions**	Switch to view with additional bend parameters.

Function keys:

	**aux. axes**	Switch to second view with parameters of auxiliary axes.
	**view bends**	Switch to a table view in which all bends are shown.
	**graphical vis**	Switch to graphical visualisation of the program.
	**zoomed values**	Switch to view with axes values.

• Parameters

BEND no . . . . . .BN =

Selection of a bend of the active program.

Repetition . . . . . .CY

Selection of one of the repeated steps of one bend. Useful if a bend has a repetition value larger than 1. This parameter becomes visible when the function key 'select rept. no' is pressed.

Corr α1 . . . . . .Cα=

Corr α2 . . . . . .Cα=

Corrections on angle values (Cα) in this bending.

Angle corrections can be programmed for both sides of the machine, Y1 and Y2. When correction α1 is entered for one side, this value is automatically copied to the α2 correction for the other side. The correction for the other side can then be changed. When both angle corrections have been entered, the resulting corrections for Y-axis and parallellism are calculated. The corrections will be saved in the active bending program.

The angle correction should be entered as following examples indicate:

1) Programmed value of 90 degrees.
Measured value of 92 degrees.
Then it is required to program Corr. α with -2.

2) Programmed value of 90 degrees.
Measured value of 88 degrees.
Then it is required to program Corr. α with +2.

In case the angle correction database has been switched on, the control checks whether a correction exists for this type of bend in the database. The result of this check is prompted in the entry field:

No stored correction. No correction has been found for this bend

Stored correction. A correction that matches the current bend has been found

Interpolated correction. A correction has been calculated (interpolated) based on other existing corrections
If a correction is entered, it will be stored in the database. At each next bend with the same properties, this same correction will be offered.
See the chapter 'Program constants' for more information on the angle correction database.

Corr X . . . . . . Cx =

Corrections on X-axis positions (Cx) in this bending. In case bend allowance is activated (see PROGRAM CONSTANTS) and a program has been entered in data preparation, the X-axes correction values are the result of bend allowance calculation. The corrections will be saved in the active bending program.
The X-axis correction should be entered as following examples indicate:
1) Programmed value of 200 millimeters.
Measured value of 202 millimeters.
Then it is required to program Corr.x with -2
2) Programmed value of 200 millimeters.
Measured value of 198 millimeters.
Then it is required to program Corr.x with +2

Stock . . . . . . ST =

The stock counter is incremented or decremented after each end of a program cycle.

G-Corr α . . . . . . Gα =

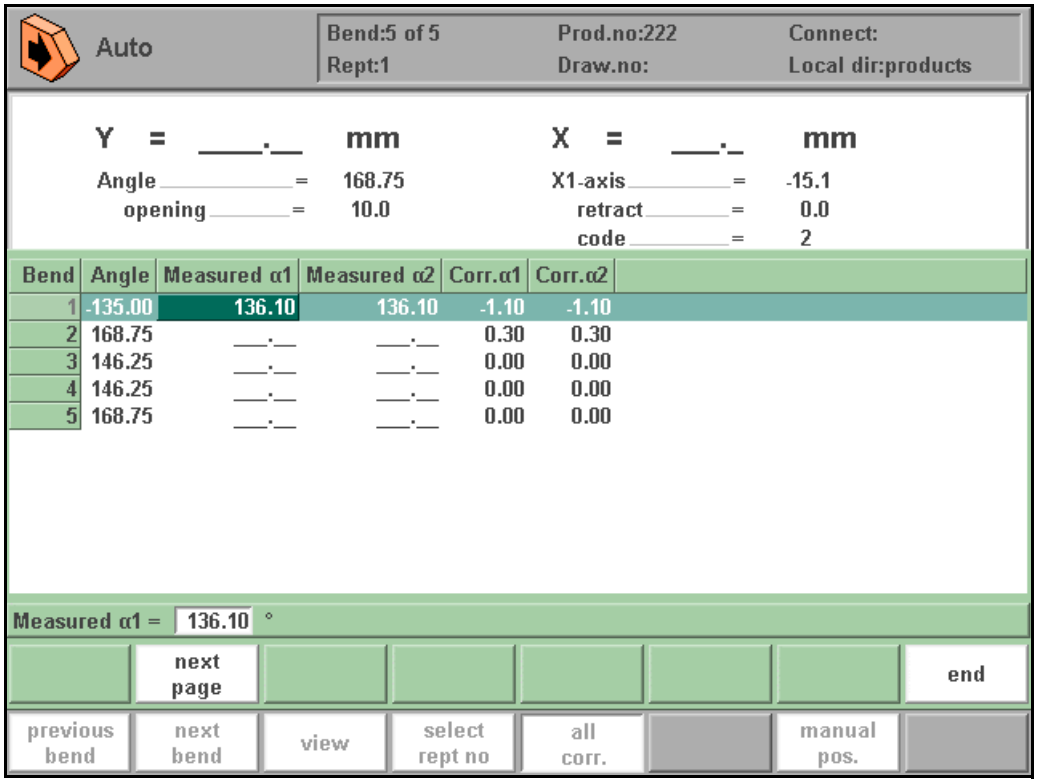
General correction of the angle, valid for each bend of the program. The value should be programmed in the same manner as for the correction per bend.

G-Corr X. . . . . . Gx =

General correction of the X-axis position, valid for each bend of the program. The value should be programmed in the same manner as for the correction per bend.

• Corrections

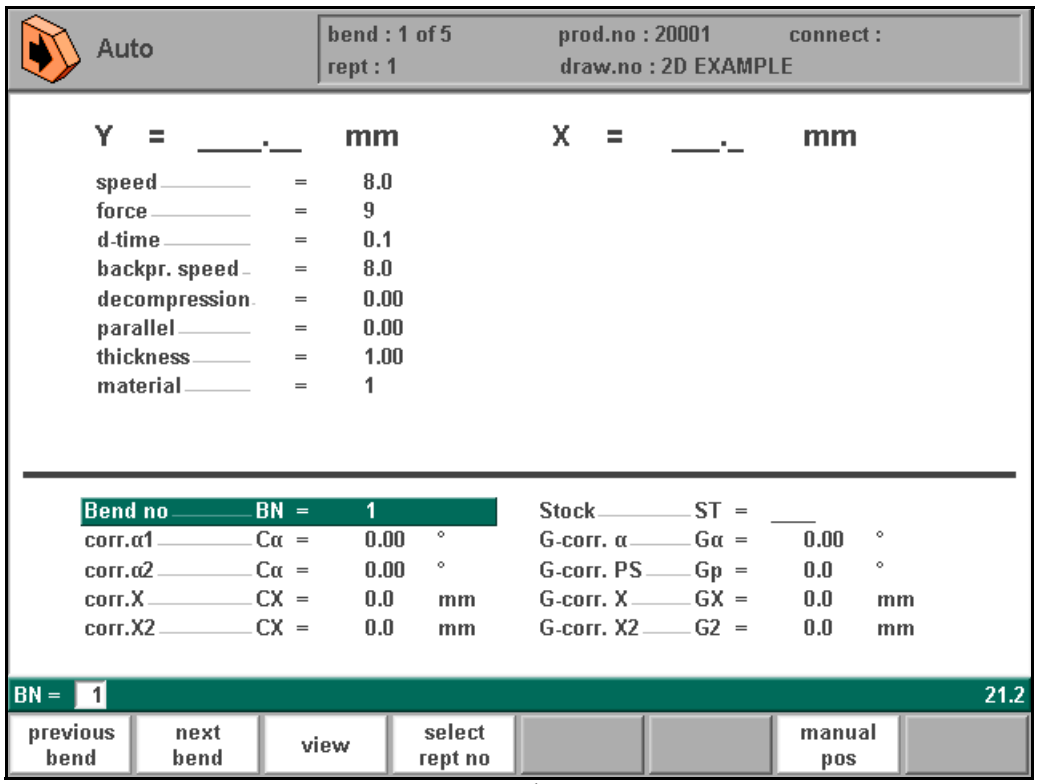
Angle corrections can be edited in a separate window. This window can be activated by pressing the function key 'all corr.' in the main screen. The following window appears:



9.c

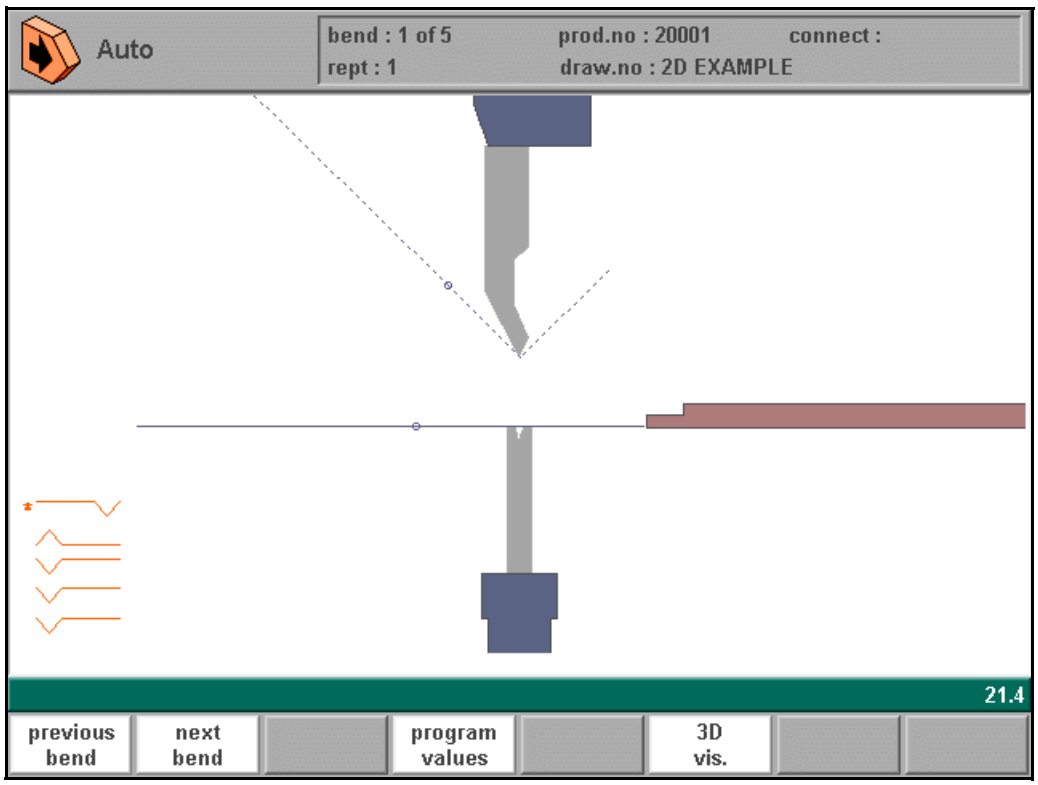
In this window the corrections of all bends are shown. You can browse through all corrections and change them as you see fit. If a correction for α1 is entered then this value is copied to the correction for α2. Different corrections for α2 can be entered in the field itself.
The columns 'measured α1/2' are present if they are enabled through the parameter 'angle correction computation' in the programming constants.
In the columns 'measured α1/2' the real, measured angle of the completed bend can be entered. On entry of this measured angle, the control will automatically compute a necessary correction to reach the programmed angle. If there is already a correction present, the control will adjust this correction to account for the latest discrepancy.
The column ‘Stored correction’ is only available when the Angle correction database has been activated. When activated, the column ‘Stored correction’ shows for each bend the correction value that is present in the database. A blank entry in this column means the database does not have a correction value for this type of bend. When a new correction is entered, it will be copied to the database automatically.
The markers ‘>’ indicate bends that have the same value.
The function key ‘all from stored’ serves to copy corrections in the database to the current program: corrections in all bends are adjusted according to database values.

- **Functions screen**



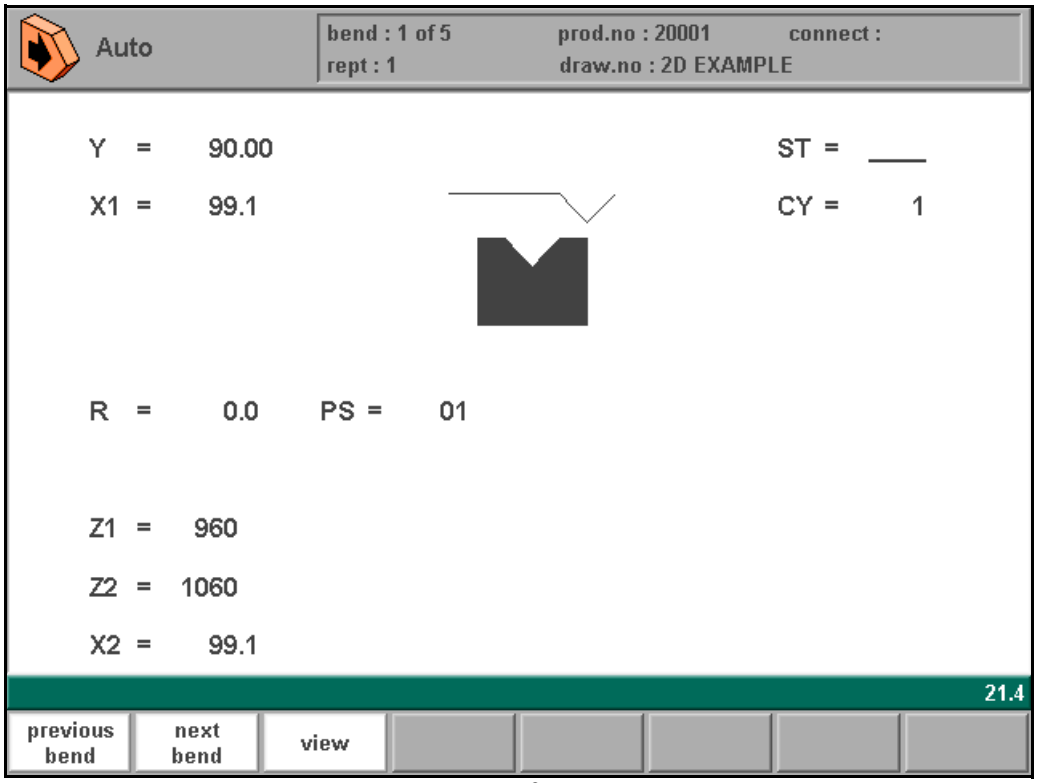
9.d

- **Graphical visualisation**



9.e

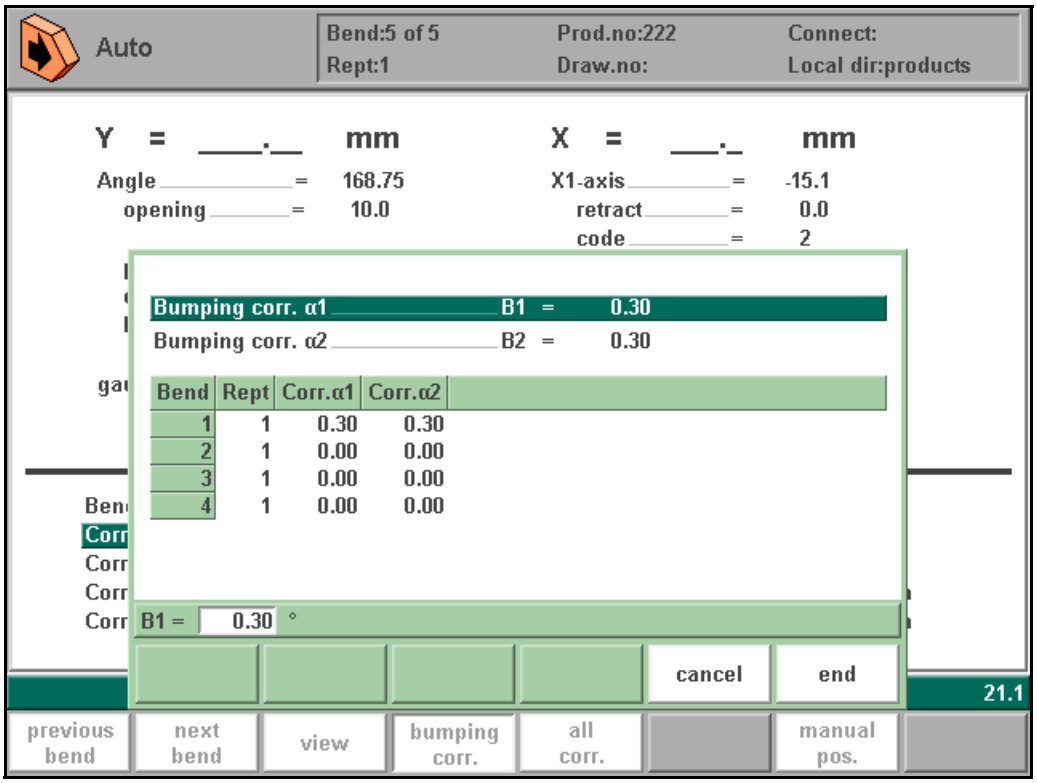
- **Zoomed values**



9.f

• Bumping correction

With this function a general correction for a radius bend can be entered. This function can be activated when the cursor is on the parameter for angle correction (‘corr. α1/α2’). It is only available if a product is loaded that contains a radius bend.
When the function is activated, a new window appears in which the correction can be entered.

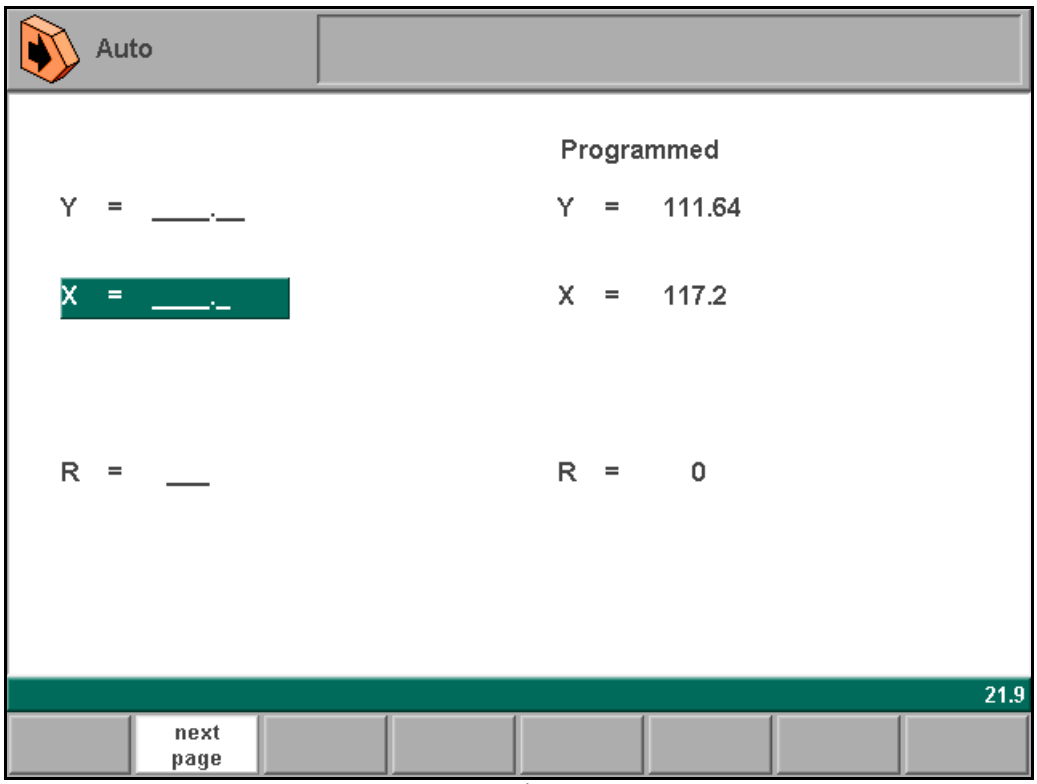


9.g

When the general correction of an angle is altered, all individual corrections are recalculated. When any of the individual corrections is altered, the general correction is recalculated.
Bumping corrections can be programmed independently for both sides, α1 and α2.
When the general correction α1 is changed, it is automatically copied to α2 and as a result, all separate corrections for α2 are recalculated. To change correction values of α2, move to correction α2 or one of the separate corections of α2.

- **Manual positioning**

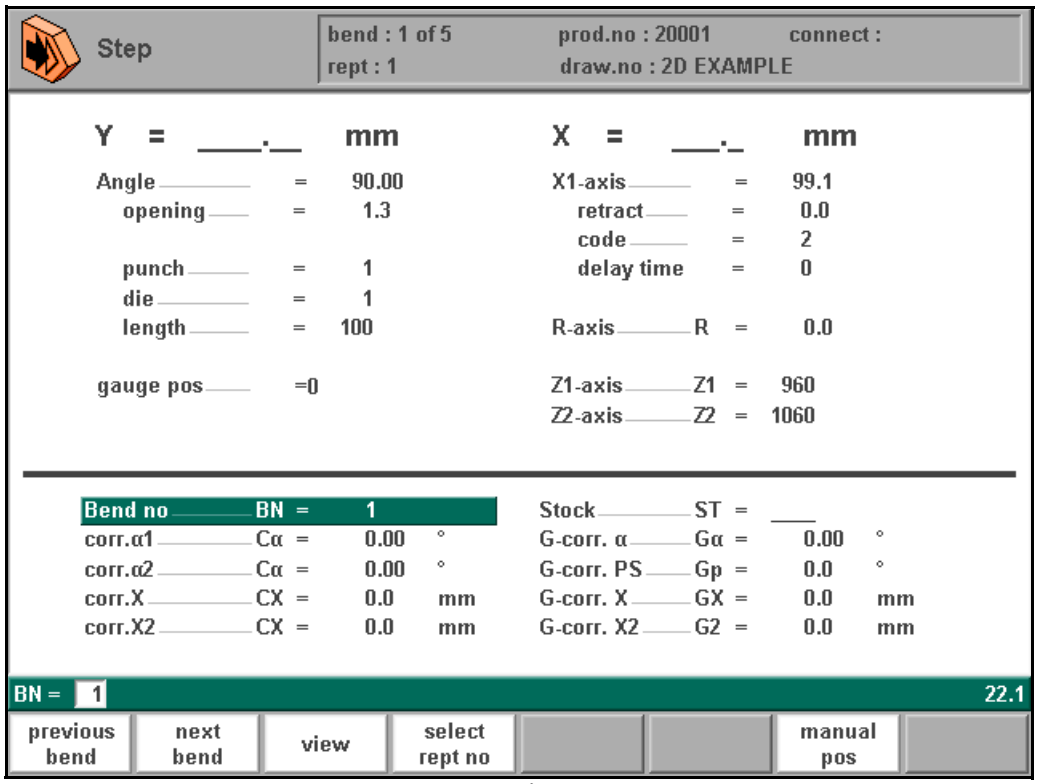
In the automatic production mode it is possible to move the selected axis manually with the handwheel and change the axis-positions with the teach function like in manual mode.



9.h

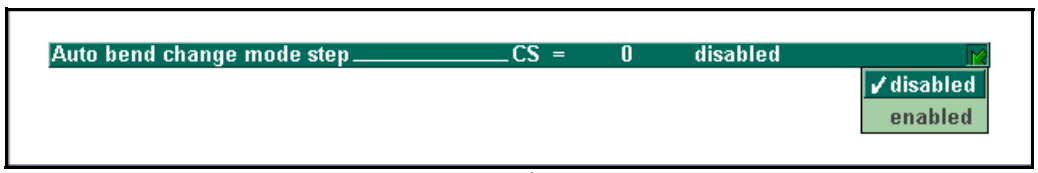
9.2. Step mode

In the step-by-step mode you have the same possibilities as in the automatic mode. There is only one difference. After each bend cycle, the control will stop. To continue working, you must start the control again by pressing ‘start’ on the front panel of the control.



9.i

After each bending step, the control can stay in the current bend or jump to the next bend. This depends on a parameter in the programming constants:



9.j

Disabled: when a step is completed, the control will stop and stay in the current step.
Enabled: when a step is completed, the control will load the next step and stop.
See also the chapter about programming constants for more information.

A. Parameter index

This appendix contains a list of all parameters described in this manual, in alphabetic order.