Delem

DA-65W

Reference Manual
Operation of Version 3
English

Manual version V1108

Preface

This manual describes the operation of the Delem controller type DA-65W and is meant for operators who are instructed for operation of the total machine.

Delem Limited warranty

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

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
V3.1	V1107	first issue V3
V3.2	V1108	update

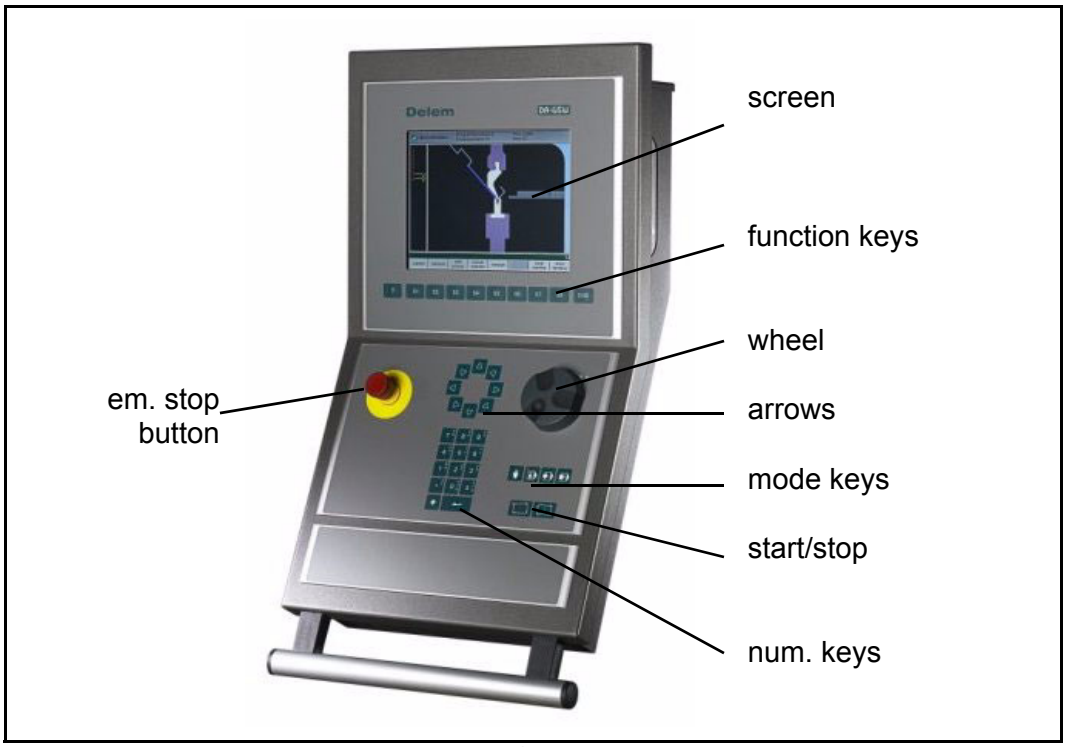
This manual is valid for software version 3.2.

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, a built-in touch pad or external USB mouse can be used as a pointing device to select menu items, parameters or function keys. 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 following four operation modes:

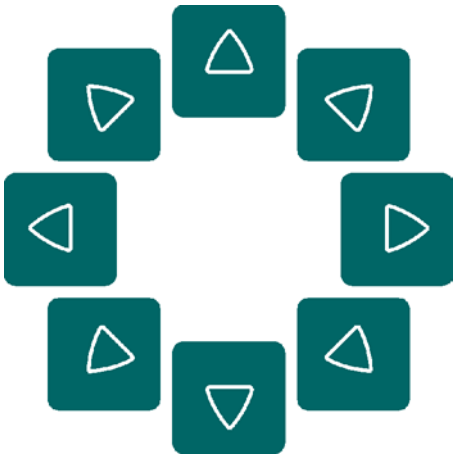
	**Manual mode**	In this mode it is possible to program all parameters of just one bending. After pushing the start button all parameters are active and the backgauge will go into position. It is also possible to move the axes manually.
	**Programming mode**	In this mode bend programs can be made or edited 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, beside the 4 operation mode buttons, consists of the following items:

arrow keys:



keyboard:

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

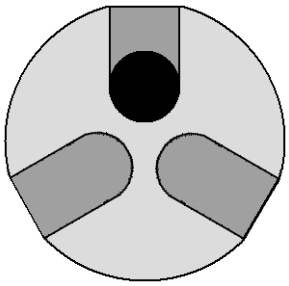
decimal point

plus/minus toggle

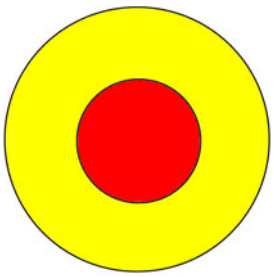
clear key: clearance of the input data field in the bottom left corner on the screen

enter key, to confirm a programmed value

Stop button Start button



handwheel; manual control of any axis (Y + backgauge axes)



emergency stop button, to be implemented by machine manufacturer.

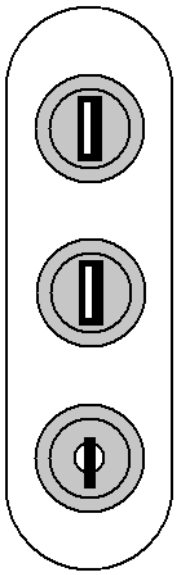
Function keys; the function of these keys is stated at the bottom of the screen.

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. Side connectors

At the right side of the control unit, the following connectors are available.



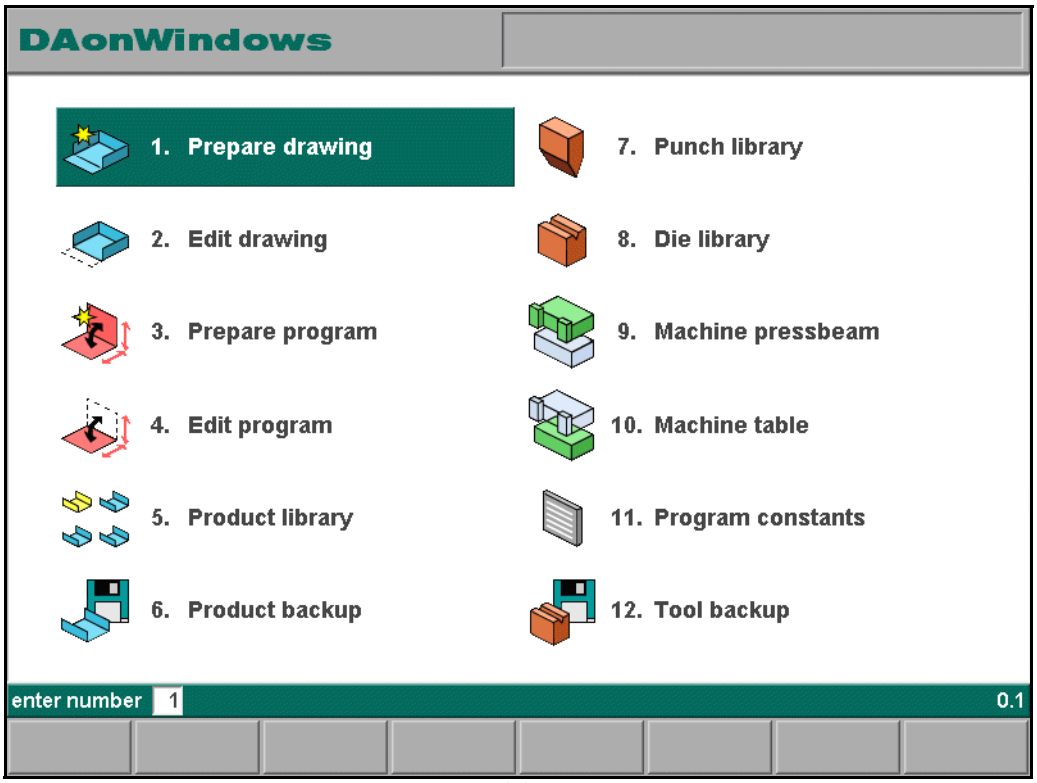
USB ports (2x)
Two USB ports are available for connection of external devices, such as a memory stick or an external keyboard.

Keyswitch
The control has a keyswitch to prevent uncontrolled programming. With the keyswitch in the off position only a selected program can be executed. The parameters cannot be changed to wrong, unwanted values.

1.5. Programming mode

Programming mode Select the programming mode.

The main menu in programming mode looks as follows:



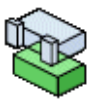
1.b

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 create a product drawing and compute the bend sequence (graphical).
2		To edit a product drawing and compute the bend sequence (graphical).

3		Create 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 punches.
8		To program the dimensions of the bottom dies.
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.

1.6. Getting started

1.6.1. Introduction

In order to obtain a bend program for a product, the control offers the possibility to create a product drawing and calculate a valid bend sequence for the product. With this information, a product program is generated.
This is done with the following steps:

1	Enter product properties	chapter 2
2	Draw 2D product profile	chapter 2
3	Program tool set-up	chapter 3
4	Determine bend sequence	chapter 3
5	Generate bend program (CNC)	chapter 3
6	Edit CNC program	chapter 4

1.6.2. Preparations

Before product programming can be started, the following preparations must be made.
- The correct material properties must have been programmed in the Program constants menu. See chapter 8 about the program constants.
- The correct tools must be programmed in the tool libraries. Tools are necessary to create a CNC program. See chapter 6 about programming tools.

1.6.3. Create a drawing

The control offers the functionality to create a drawing of the intended product. With this drawing application a 2D profile or 3D product drawing is created. At this stage, there is no calculation of bends or dimensions: any profile or drawing can be created.
More information about this can be found in chapter 2.

Features of the drawing tool:

- Graphical design of product shapes in 2D and 3D (if available)
- Scaled sheet thickness
- Auto scaling
- Horizontal and vertical projected dimensions can be entered
- Real scale tool design
- Various machine shapes (pressbeams and tables)
- Changing of lengths and angles
- Big radius
- Adding or deleting of bends
- Existing products can be copied, changed and stored as a new product
- Closing dimension or highest precision tolerance selection
- Connecting 2D programs for 3D-production

1.6.4. Determine bend sequence

When the product drawing is correct, the control offers a simulation menu to determine the bend sequence. First, a tool configuration menu is offered to program the exact tool set-up as it is organised on the machine.
In the bend simulation menu, the control shows the product, the machine and the tools. In this menu the bend sequence can be programmed and checked visually. When a bend sequence has been determined, a CNC program can be generated.
More information about this can be found in chapter 3.

Bending sequence computation:

- Automatic computation for minimum production time
- Interactive bend sequence determination
- Manual bend sequence determination
- Collision visualisation of product with tools and machine
- Free tool and machine shape selections
- Assignments of turn times, backgauge speed etc.
- Blank length computation
- Production time indication
- Bending sequence simulation
- Programmable finger positions

1.6.5. CNC program

There are two possibilities to create a CNC program:
- enter a numerical program, step by step (menu 3 & 4: data preparation);
- generate the program from the graphical bend simulation (menu 1 & 2; product drawing).

If the program is entered by hand, there is no collision check. All program values must be entered manually. The program depends on operator experience.
If the program is generated from a graphical bend sequence, the program can be visualised during production. A generated program can be edited according to operation needs.
More information about this can be found in chapter 4.

Post processing of drawings

The post process facility computes:
- Necessary force
- Machine adjustments such as:
  - Y-axis position
  - Decompression
  - X-axis position
  - X-axis retract
  - Y-opening
  - R-axes
  - Z-axes

Axes positions are calculated according to the machine configuration.

1.6.6. Production

A product program can be executed in Automatic mode or Step mode. In Automatic mode, a

complete program can be executed bend after bend. In Step mode, each bend must be started separately.
The manual mode of the control is an independent production mode. In this mode, one bend can be programmed and executed. It is typically used to test the behaviour of the bend system.
More information about this can be found in chapter 9 and 10.

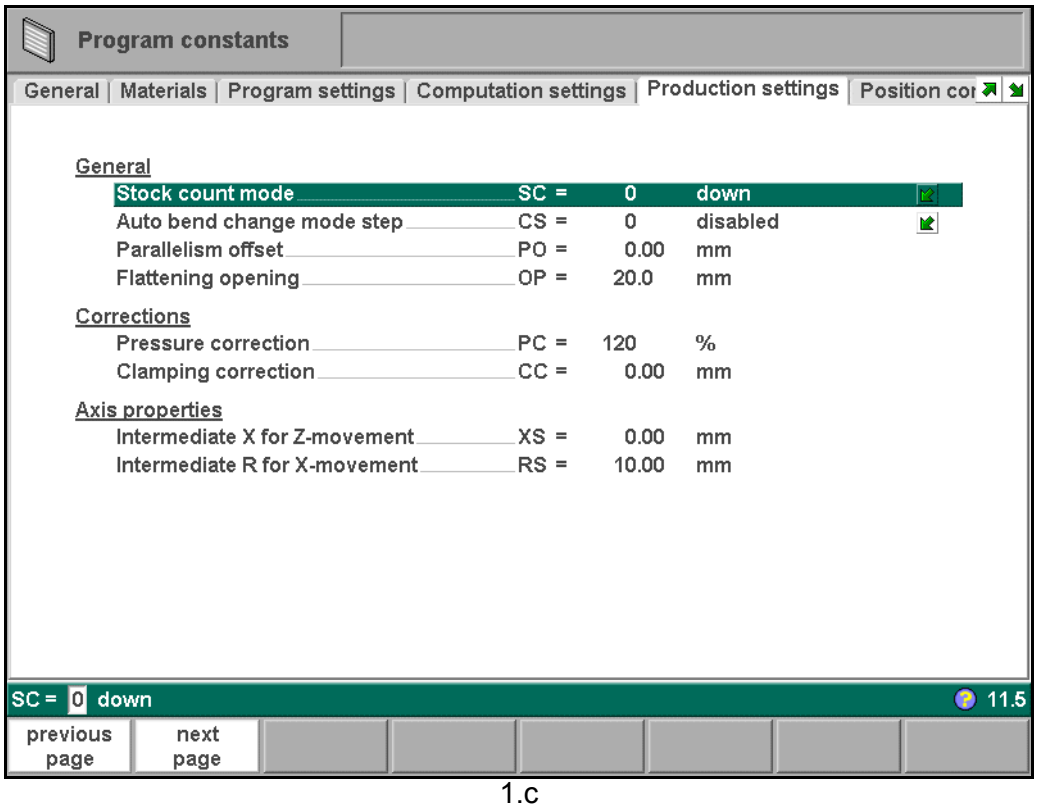
1.6.7. Back-up data

Both product and tool programs can be stored externally. Depending on the configuration, these files can be stored on a network or on a USB storage device. This facilitates a back-up of important data and the possibility to exchange files between Delem controls.
More information about this can be found in chapter 7.

1.7. Programming aids

1.7.1. Help text

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

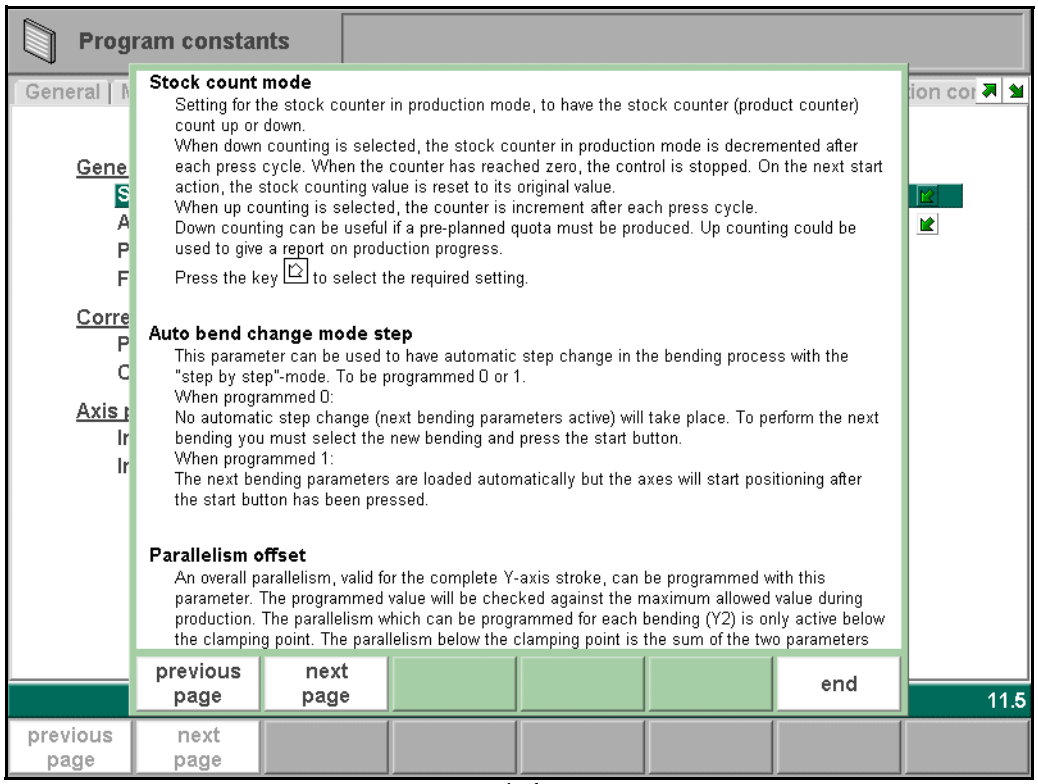


1.c

To activate a help window for a parameter:

- press the Help key ('?') on the front panel or
- click with the mouse on the help symbol.

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

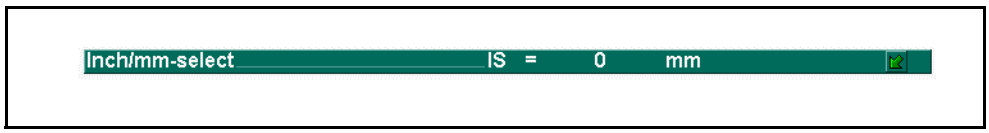


1.d

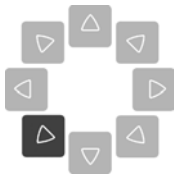
This Help window contains the same information as the Operation manual.
The help window can be used as follows:
Use the arrow keys (up/down) to browse through the help text or use the function keys 'previous page' and 'next page'.
Press the function key 'end' or the END key to close the Help window.

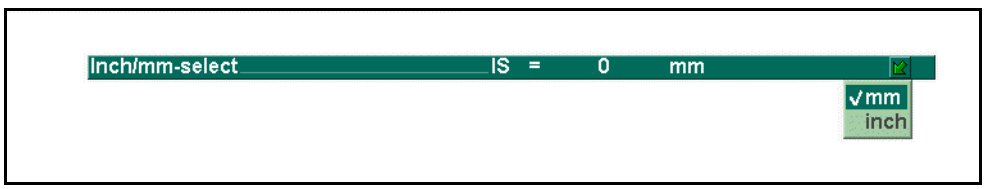
1.7.2. Listbox functionality

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



This means that an arrow key (down left) 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.
When opened, use the arrow keys (up/down) to scroll to the desired value and press ENTER.
Press END to exit the listbox.

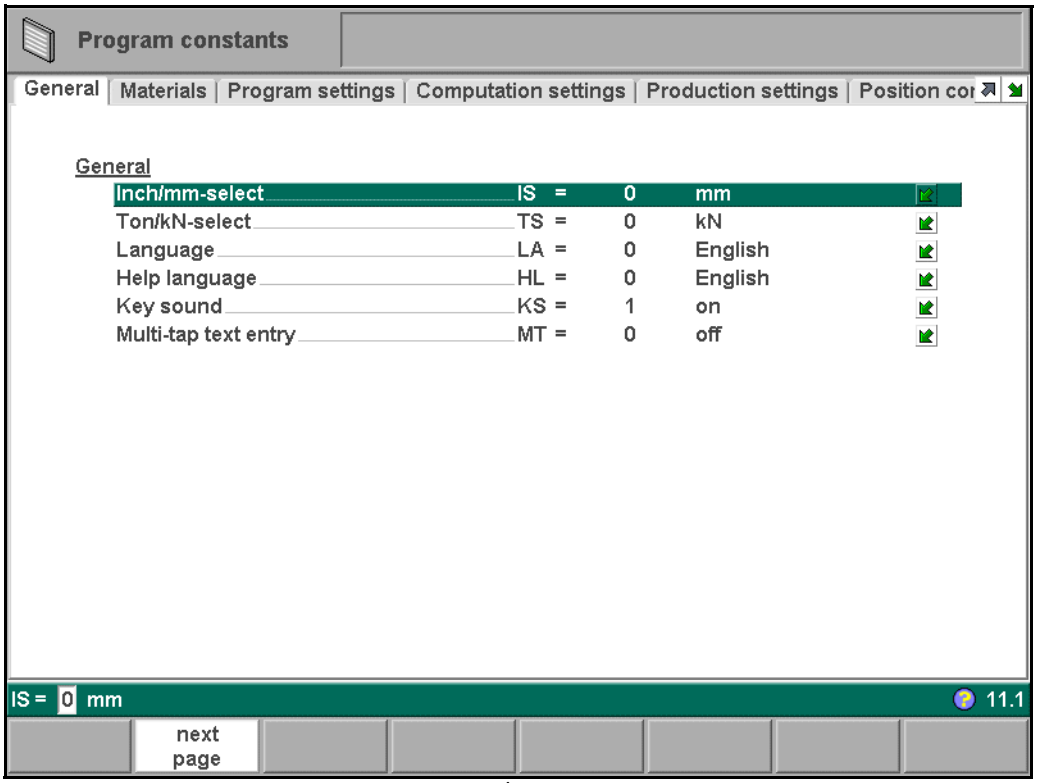
1.7.3. Live search

In some menus a list of entities (products, tools, etc.) is offered. An example of such a menu is menu 5 (Product selection). To search a particular product or tool, type a part of the ID in the enter field. Automatically, the list is limited to those items that contain the typed part.

Example of a product list:	type:	shown list:
product123 product 456 exampleproduct01 exampleproduct02	1	product123 exampleproduct01
	ex	exampleproduct01 exampleproduct02
	ex 1	exampleproduct01

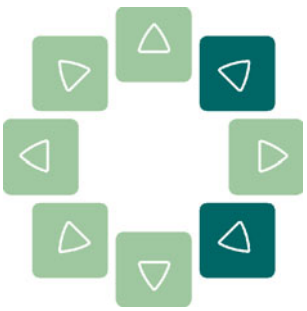
1.7.4. Navigation

Within some menus, the program screens are divided into tabs.



1.e

The following arrow keys can be used to browse between pages.



next page

previous page

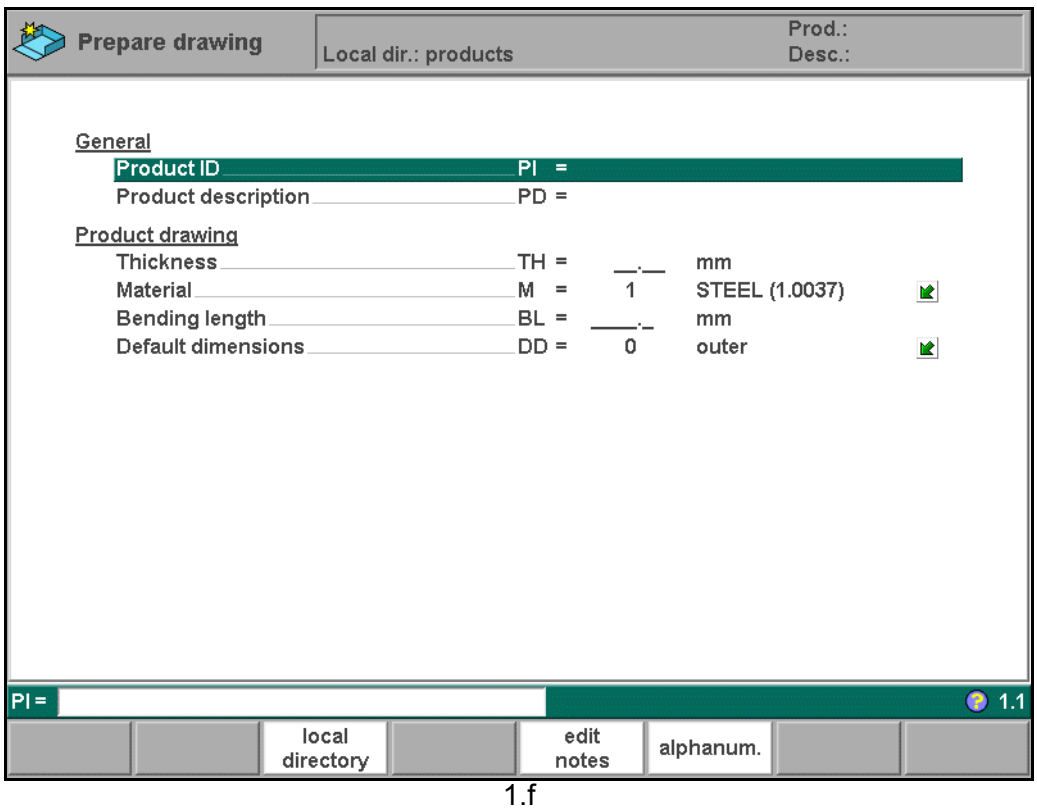
This browse functionality is also indicated with the two arrow symbols in the upper right corner of the screen.

1.7.5. Typing alphanumeric characters

When necessary, it is possible to type alphanumeric characters on the control by means of the front panel keys. There are two procedures available to enter characters.

1. The alphanum. key

When the cursor is placed on a parameter that contains alphanumeric characters, the function key ‘alphanum.’ appears in the function key bar at the bottom of the screen.



1.f

To type a character:
- press the key on the front panel that contains the character;
- press the function key ‘alphanum.’ and check the character that appears;
- if necessary, press ‘alphanum.’ again to select another character.

When the correct character has been selected, repeat this procedure to type another character or press the ENTER key to confirm the entered string.
The available characters are displayed on the keys on the frontpanel. See also section 1.3 for an overview of these keys.

2. Key repetition

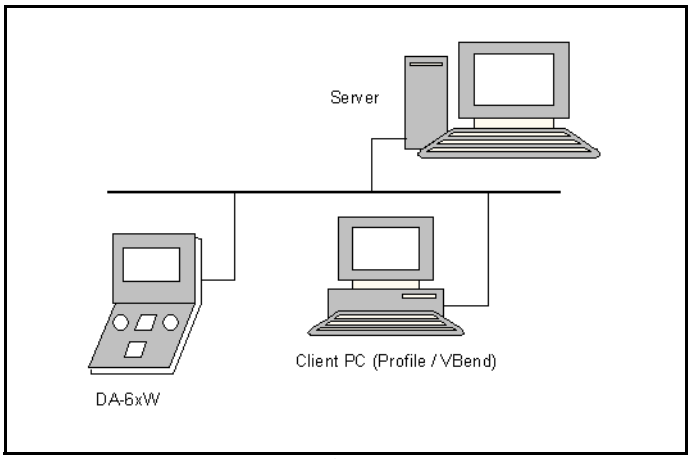
A character can also be entered by pressing a key repeatedly:
- simply press the required key repeatedly until the desired character appears
- when the correct character is shown, wait for two seconds to keep the character.

Now repeat this procedure to type another character or press ENTER to confirm the entered string.
This procedure is only possible if the parameter ‘multi-tap text entry’ is switched ON in the

programming constants. See also chapter 8.

1.8. Network

The CNC control is equipped with a network interface. 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.g

Chapter 7 contains more information about networking possibilities.

1.9. 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 3.1
V stands for version
3 is the major version number
1 is the minor version number

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

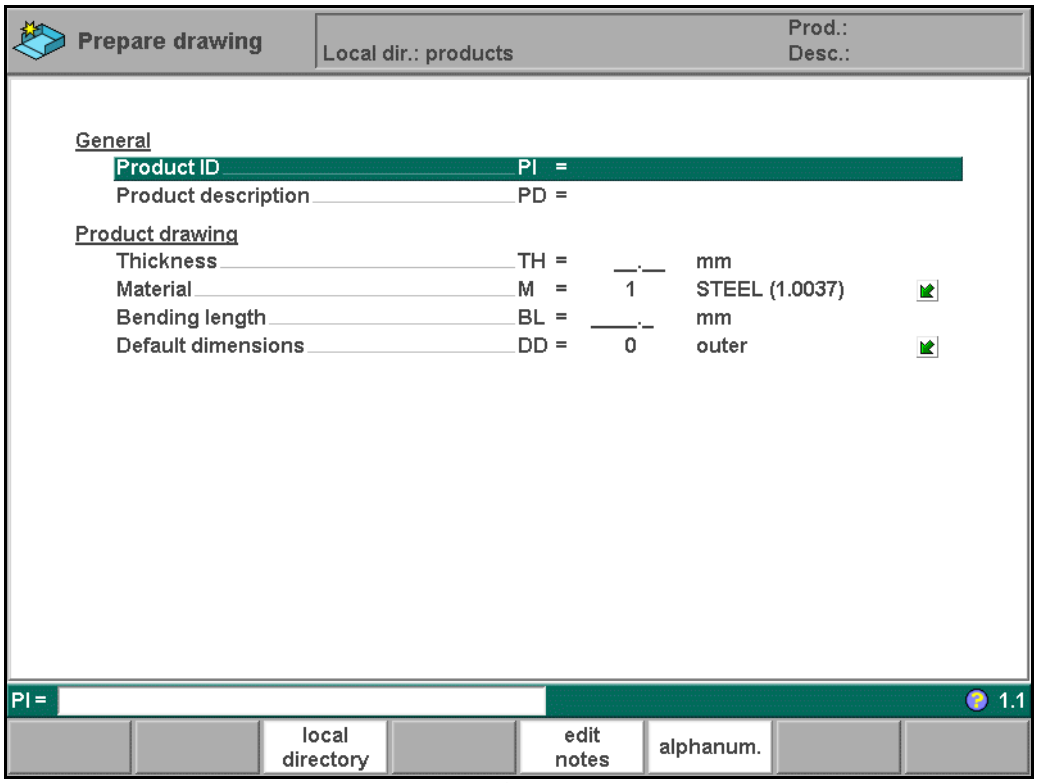
2. Product drawing

2.1. General product properties

To start a new product drawing, choose **prepare drawing** in the main menu.

To edit an existing product drawing, choose **edit drawing** in the main menu.

When a new product drawing is started or an existing one is opened, a screen with general product properties appears.



2.a

Product ID . . . . . .PI

A unique name to identify a product program. The maximum length is 25 characters. The product ID may contain letters and numbers. Letters can be entered with the help of the function key ‘alphanum.’ (S6).

Product description. . . . . .PD

A name or description of this program. The maximum length is 25 characters. The product description may contain letters and numbers. Letters can be entered with the help of the function key ‘alphanum.’ (S6).

If an existing product ID 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 ID.

The "±" key prompts a "-" character and the "." key prompts a "/" character in the product description. Use the key “4” to obtain a space.

Thickness . . . . . .TH

Thickness of the sheet.

Material . . . . . .M

Selection of the material type of the sheet. The control contains 4 pre-programmed 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.

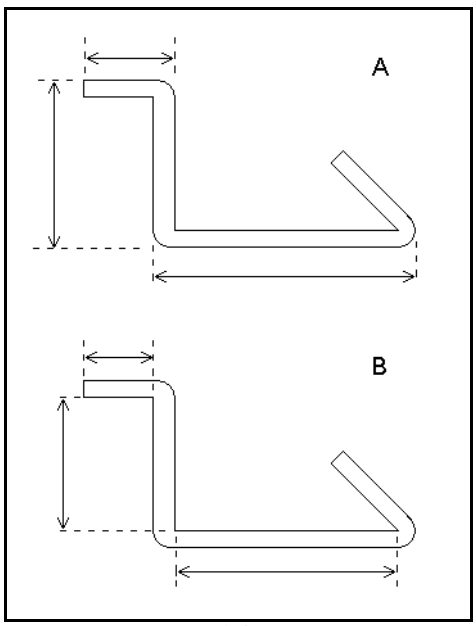
Bending length . . . . . .BL

The Z-length of the sheet.

Default dimensions. . . . . .DD

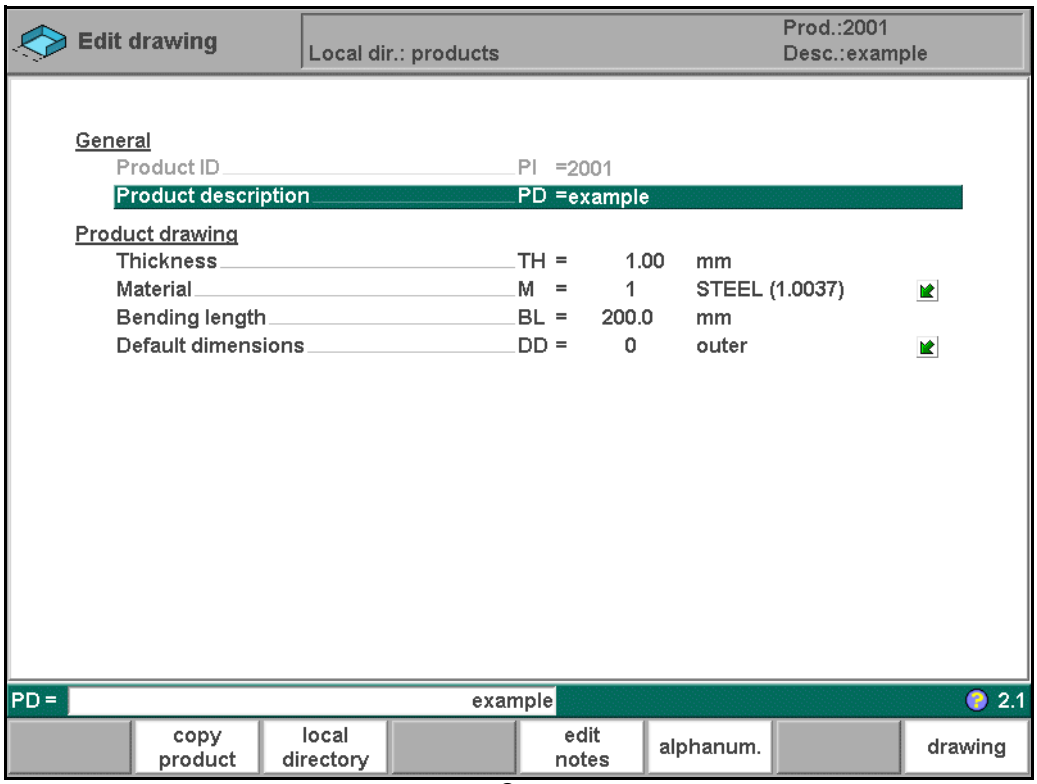
Determine the use of the outside (A) or the inside (B) dimensions in the product drawing when new sides or surfaces are added. Figure 2.b shows the definition of both the dimensions.

Press the key ⌂ to select the required setting.



2.b

If an existing drawing is opened with ‘edit drawing’ in the main menu, the same screen with general product properties appears. At the bottom, more function keys are available.

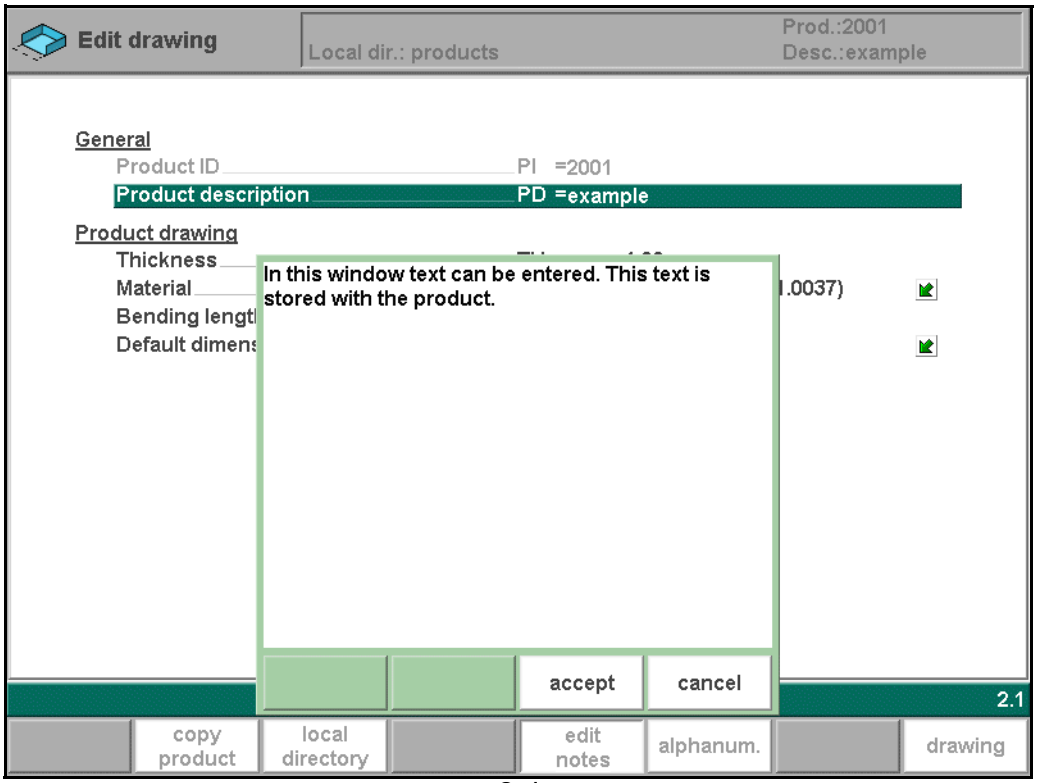


2.c

Function keys

S2	**copy product**	Copy the current product. When pressed, you must enter a new product ID for the new copy.
S3	**local directory**	Change the active directory on the control disk. The current product is automatically copied to the new directory.
S5	**edit notes**	Open a window with notes about the current product.
S6	**alphanum.**	Select an alphanumeric character after typing. The possible characters are displayed on the frontpanel key. Only possible at relevant parameters.
S8	**drawing**	Switch to the product drawing.

When the function key ‘edit notes’ has been pressed, a new window appears in which you can edit the text about the current product. See the chapter ‘Product programming’ for more information about product notes.

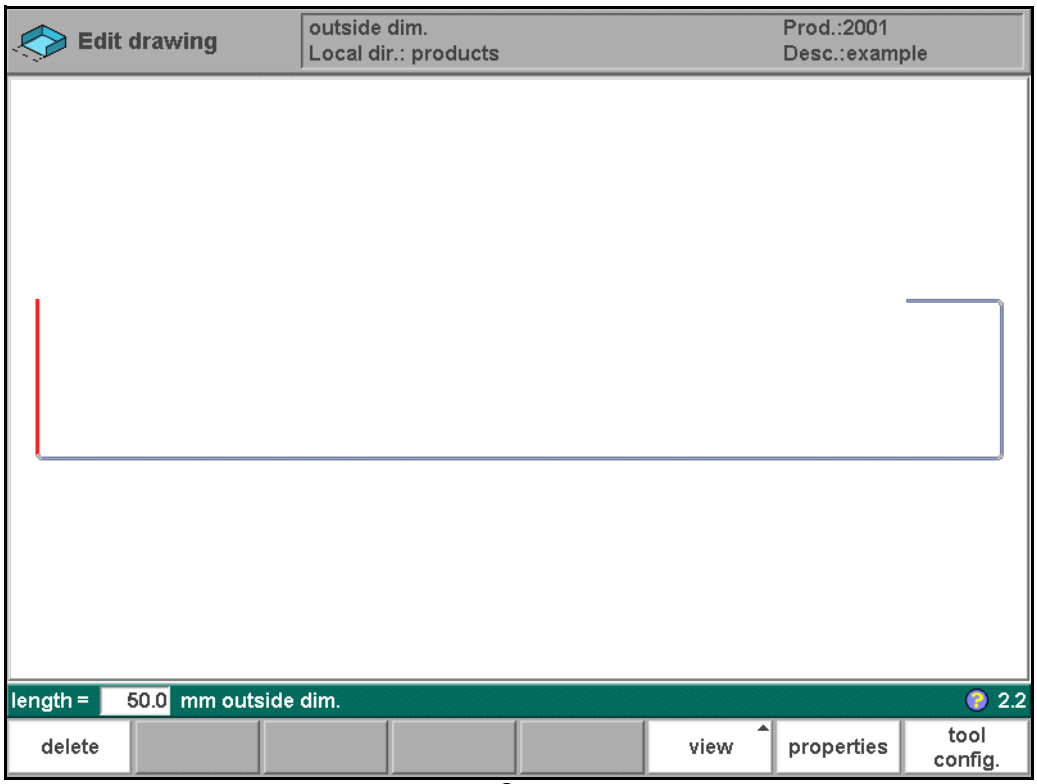


2.d

2.2. 2D product drawing

- **Introduction**

After entering the general product data the drawing screen appears. In the upper information row you will find the information about product ID, product description and inside/outside dimensions selection.
Now you can create the profile of the product. First enter the value of the basic length of the product. Then enter the angle of the next side followed by the length of that side. These properties are prompted in the lower left corner of the screen. This procedure continues until the product has the desired profile. It is also possible to use the cursor keys to quickly draw new sides, see also the section about cursor control keys.
The currently active element (line or angle) is highlighted. With the cursor control keys you move the drawing cursor to another position (angle or length).
In a product drawing you can program up to a maximum of 99 bends per product (graphical programming).



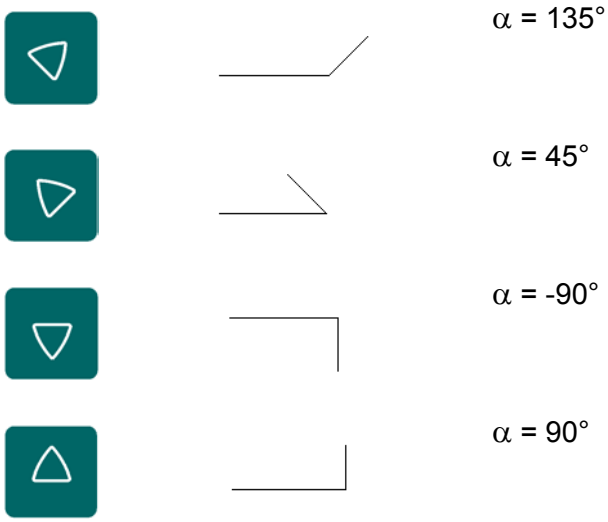
2.e

Function keys:

S1	**delete**	Delete the currently selected line. Only available if the cursor is on the first or last line of a profile.
S7	**properties**	Open a window with properties of the currently selected bend.
S8	**tool config.**	Program the tool configuration and start the simulation. See chapter 3.
END		Exit the drawing and return to the product properties.

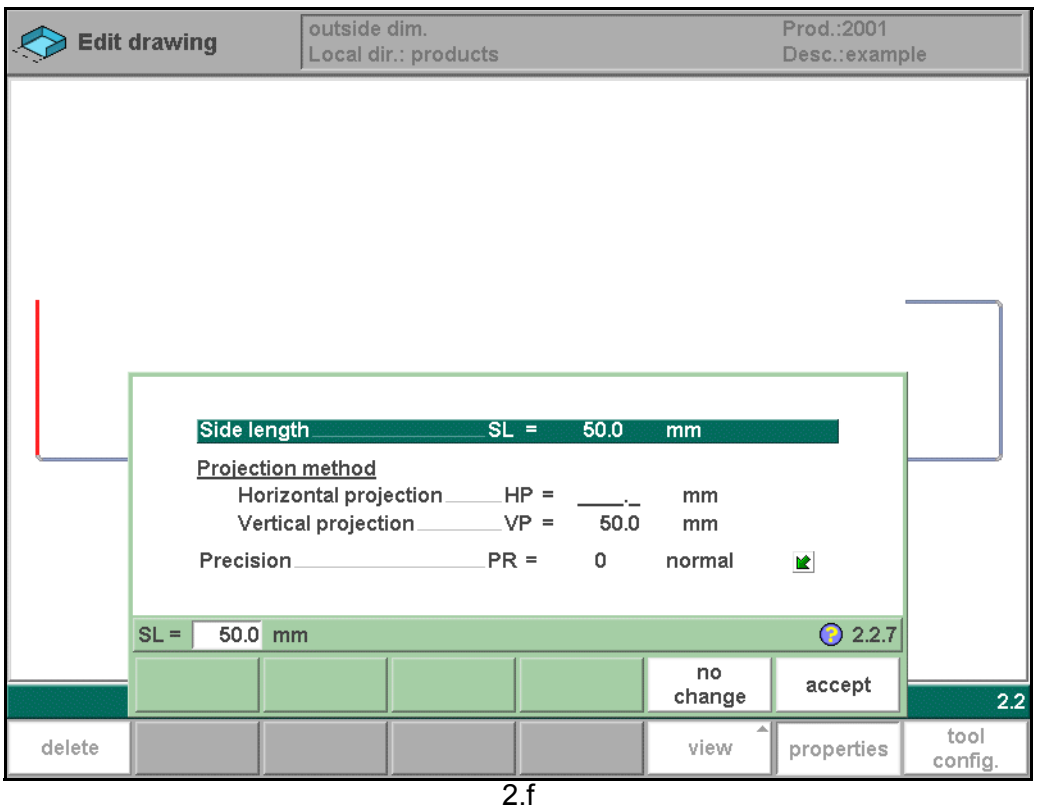
- **Drawing cursor control for 2D-products**

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.:



2.3. Line properties

2.3.1. Introduction



2.f

For each line (surface) several properties can be programmed:
Line properties:

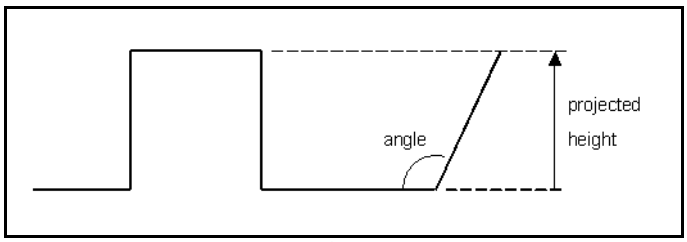
- length
- projection method
- precision

2.3.2. Projection

Inside the window with line properties, the following projection properties can be programmed:

Horizontal projection. . . . . ..HP
The horizontal distance a line must measure, regardless of its angle value.

Vertical projection . . . .VP
The vertical distance a line must measure, regardless of its angle value.

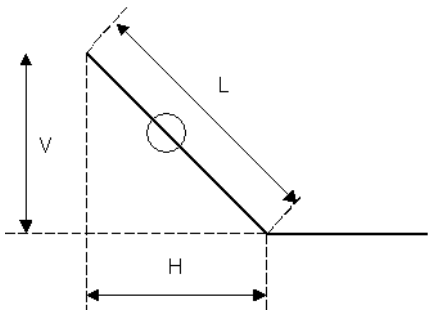


2.g

The projection function is a useful aid for drawing diagonal lines between points without having to compute the precise side length. When a line is selected, simply enter a horizontal or vertical projection distance and press ENTER. The required line length is computed and applied to the selected segment.
Projections can also be applied during programming. After programming an angle for a new side, program the projection distance (horizontal or vertical) and press one of the arrow keys for the required direction.

Horizontal projection:  or 

Vertical projection:  or 



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.

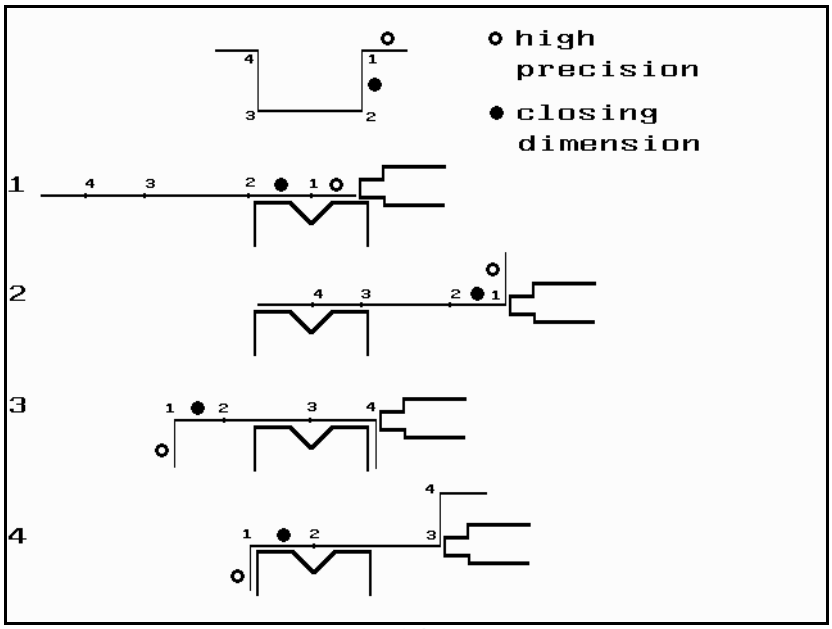
2.3.3. Precision selection

When the drawing cursor is on a line segment, you can select the level of precision for this line. Enter the properties and go to the parameter ‘precision’.

Precision . . . . . .P

Select the level of precision for a line.
Normal: achieve normal accuracy for this segment.
High: at bend sequence computation the back gauge stop position will be chosen to get the highest possible precision for this line interval.
Closing dimension: at bend sequence computation the back gauge stop position will be chosen to get the resulting tolerances in this line interval.

Example:



2.h

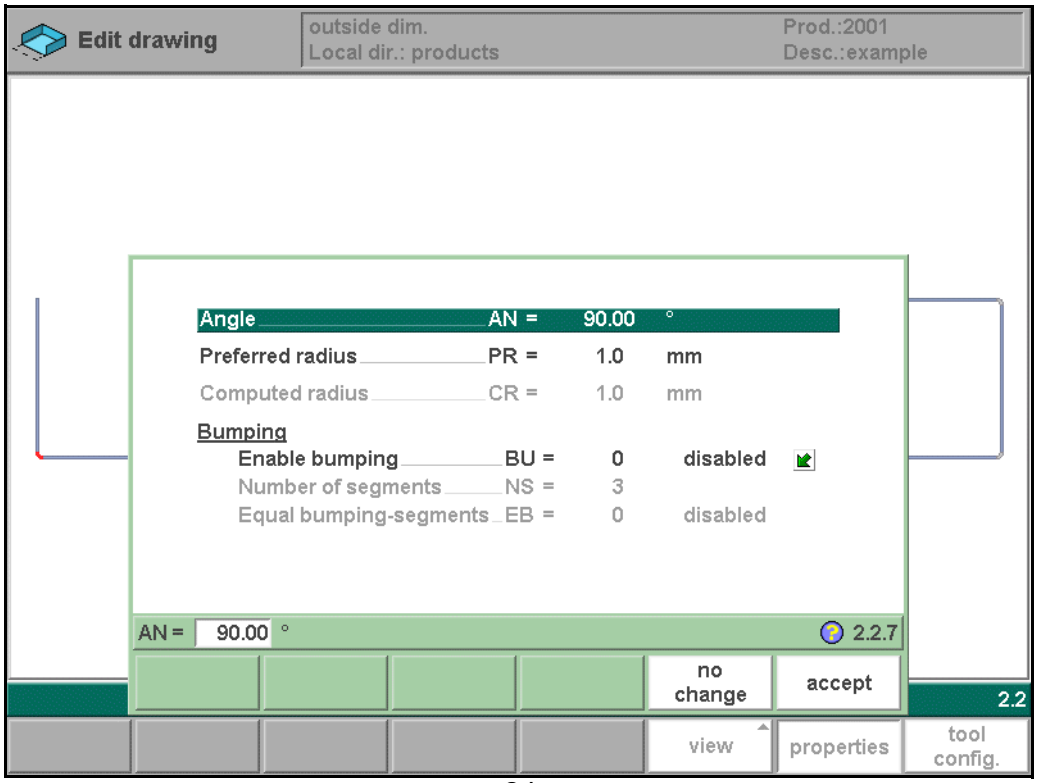
Line interval marked with the open circle should be, if possible, directly placed between back stop and the centre of the die.

Notes:

Specifying line intervals with high precision and closing dimensions may result in longer production time.
The precision parameter will have priority over the "front extend ratio", if that is set to "comply if possible". See section 3.3.

2.4. Bend properties

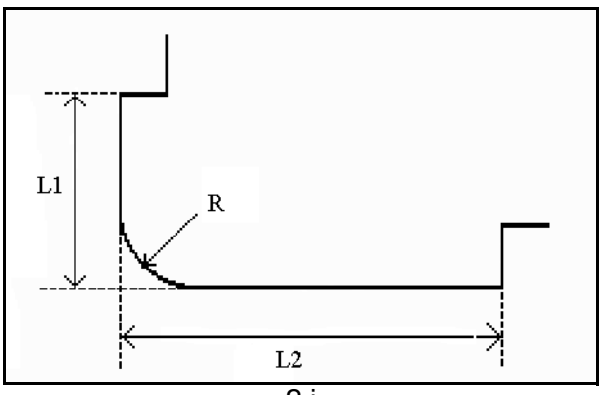
2.4.1. Large radius



2.i

Bend properties:
- angle value
- radius properties

A bend with a large radius is programmed in the properties window. The radius value may not exceed the length of any of the adjacent sides.
For the definition of the line lengths to be programmed in the part connected to a radius bend, see figure 2.j.



2.j

Lengths L1 and L2 must be equal or bigger than the radius R.

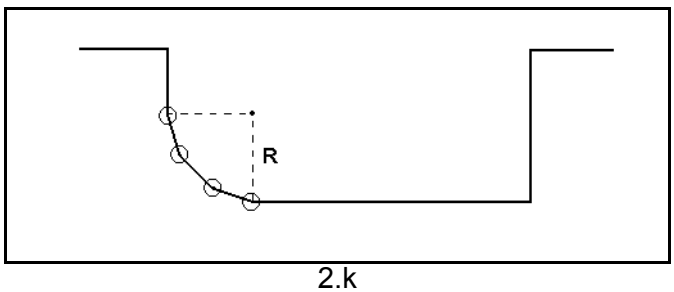
Preferred radius . . . . . .PR
The intended radius which is programmed.

Computed radius . . . . . .CR
The resulting radius as computed from the current control settings.

A large radius is meant to be bent with a special punch with large radius. If such a punch is not available, the bumping method can be selected.

2.4.2. Large Radius: Bumping

If a tool with large radius is not available, the bumping method can be chosen. With this method, a large radius in a product is obtained by a series of slight bends in succession.



2.k

To apply the bumping method, the following parameters must be programmed:

Enable bumping . . . . . .BU
Switch the bumping method on or off.

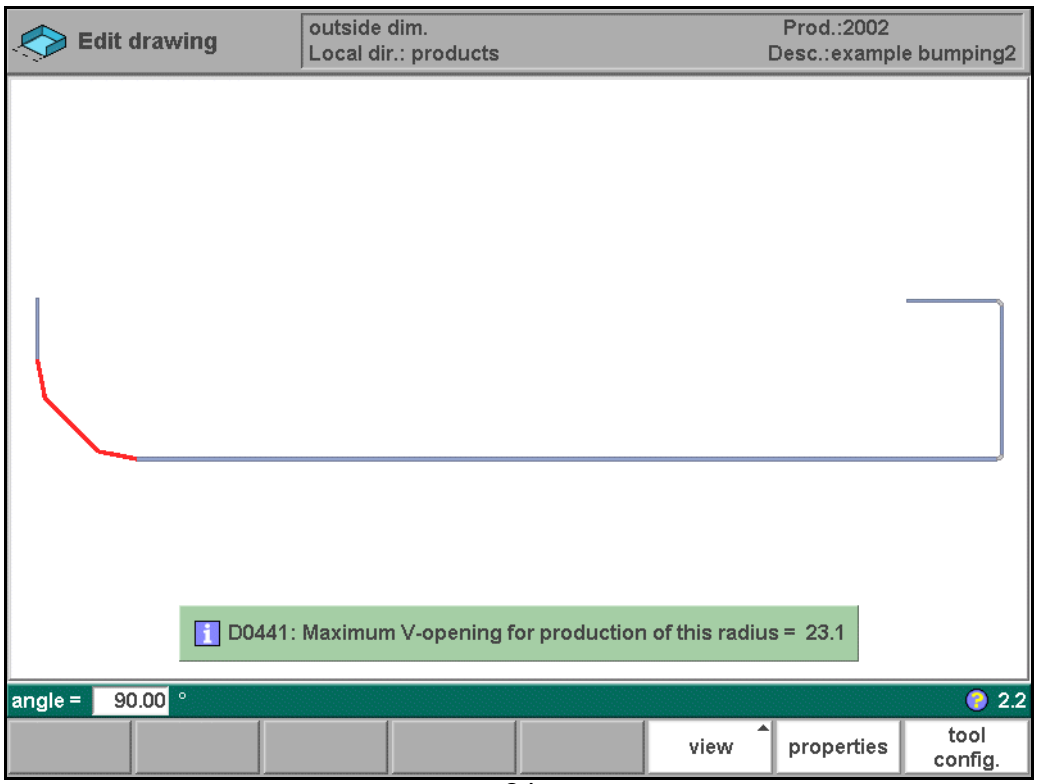
Number of segments . . . . . .NS
The number of segments in which the radius will be divided. The number of bends in this radius is the number of segments plus 1.
The more segments you select, the more bends will be used to create the programmed radius within a smaller tolerance. With a high number of 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.

Equal bumping-segments . . . . . .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 equally sized segments.

When you must program such a bend, first program a standard angle with adjacent sides. Then put the cursor back on the angle, press the function key ‘properties’ and set the parameter ‘Enable bumping’ to enabled. 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

3. Tool configuration and bend sequence

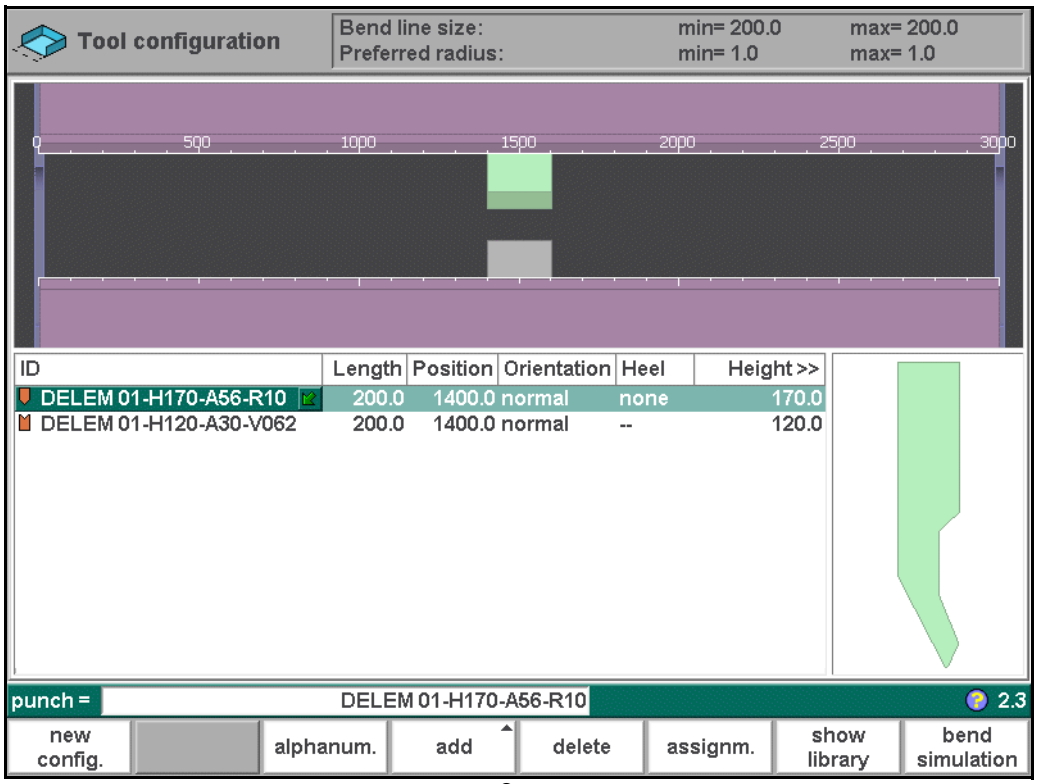
3.1. Introduction

This chapter describes how to program a tool configuration and a bend sequence for a product. When finished, a product program can be generated with this information.

3.2. Tool configuration

3.2.1. Standard procedure

When the function ‘tool configuration’ has been activated, the screen shows a front view of the machine set-up in the upper half of the screen. In the lower half of the screen, the tool data is displayed. In this screen, the placement of tools in the machine can be programmed.

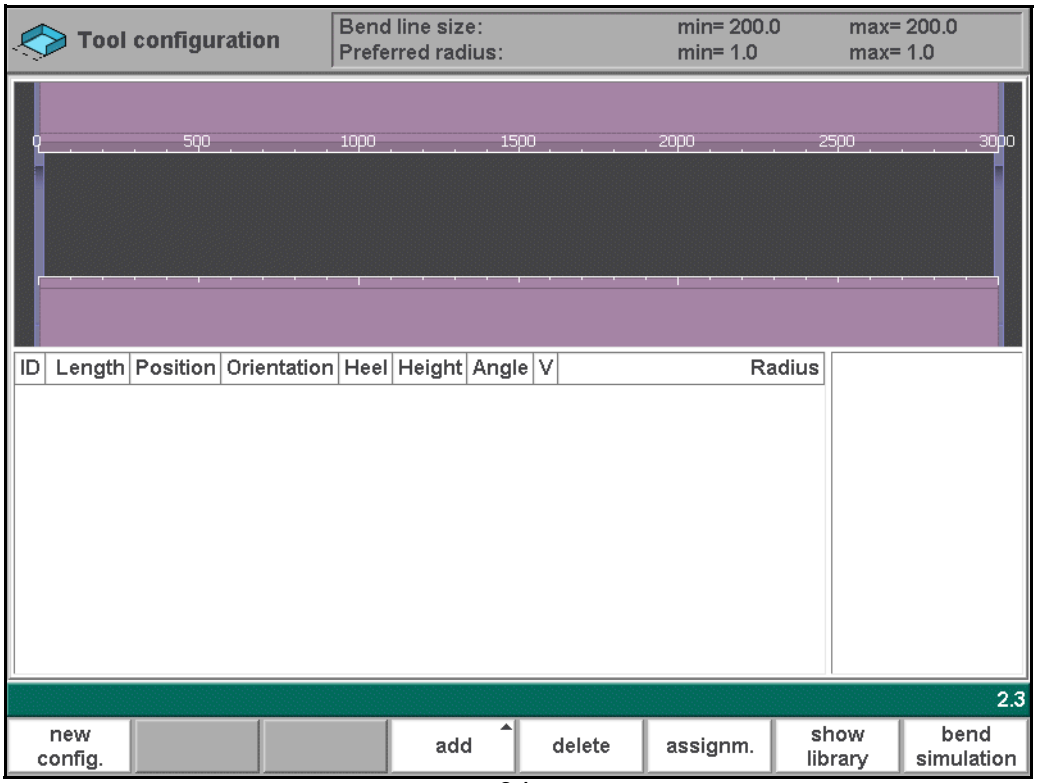


3.a

In the front view, the following machine elements are shown, from top to bottom:

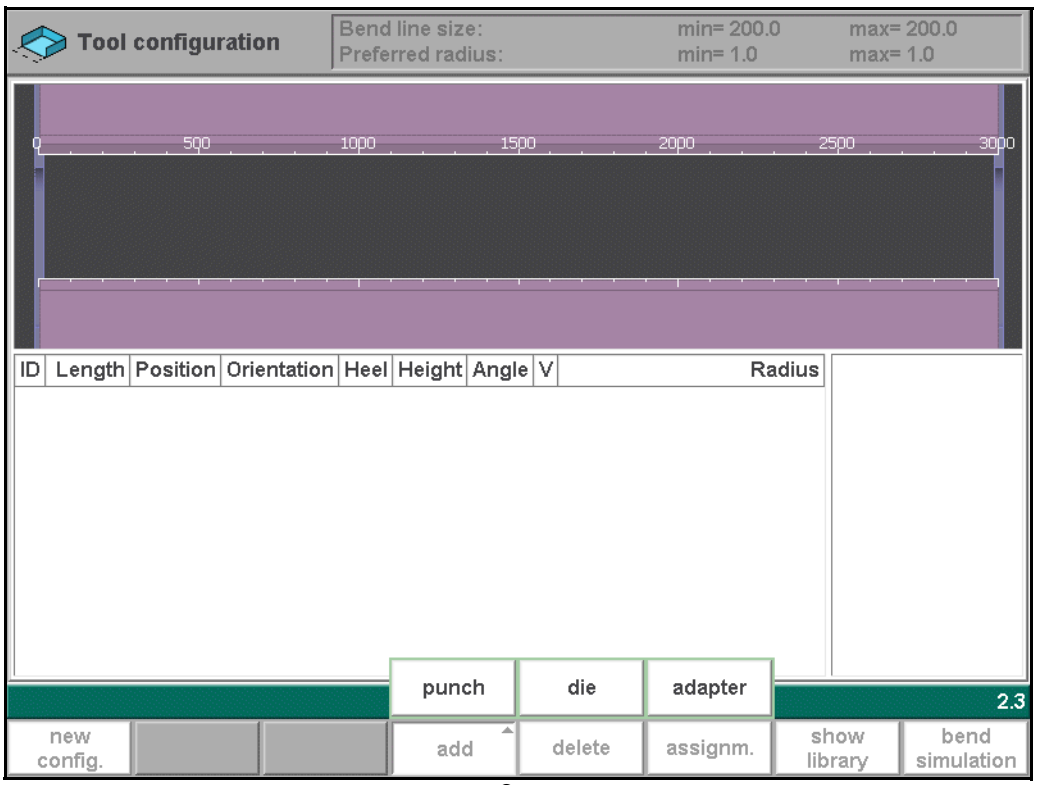
- machine upper side (pressing beam);
- adapter for punch (if an adapter is programmed);
- punch;
- die;
- machine lower side (table).

The machine parts have already been pre-selected in the menu ‘program constants’. Normally these parts will not change. Whether an adapter can be programmed depends on the parameter ‘enable adapters’ in the programming constants.
When starting a new tool configuration, the machine opening is empty.



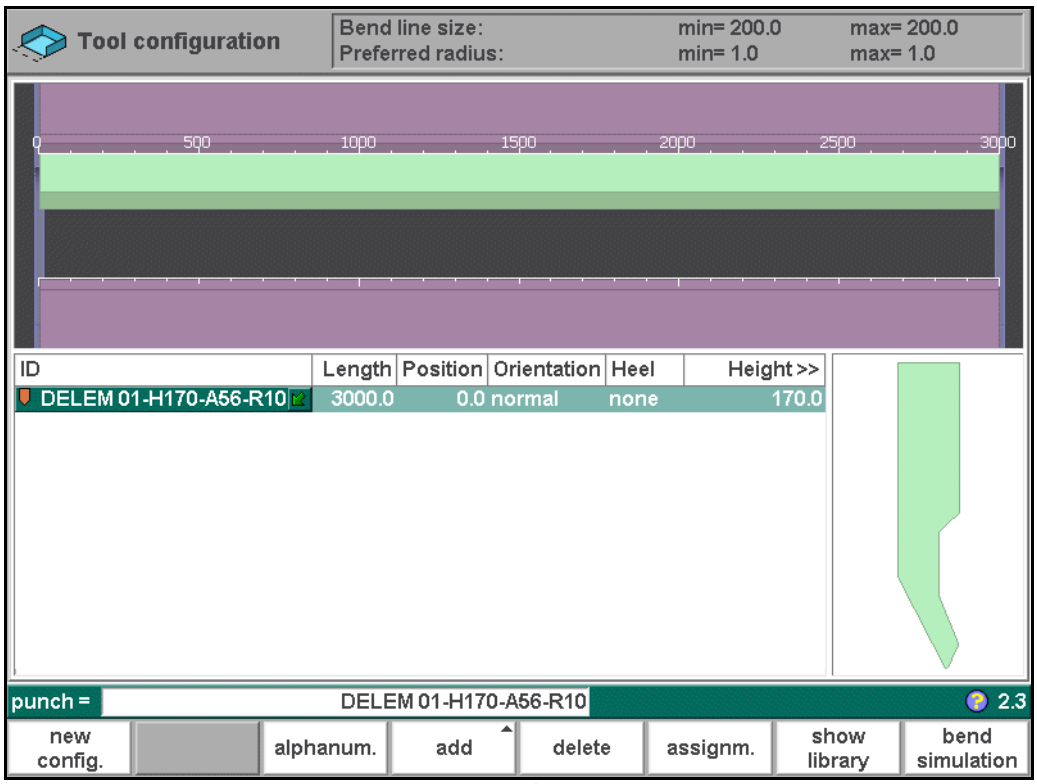
3.b

Press the function key ‘add’ to add a tool to the configuration: punch, die or adapter (if enabled).



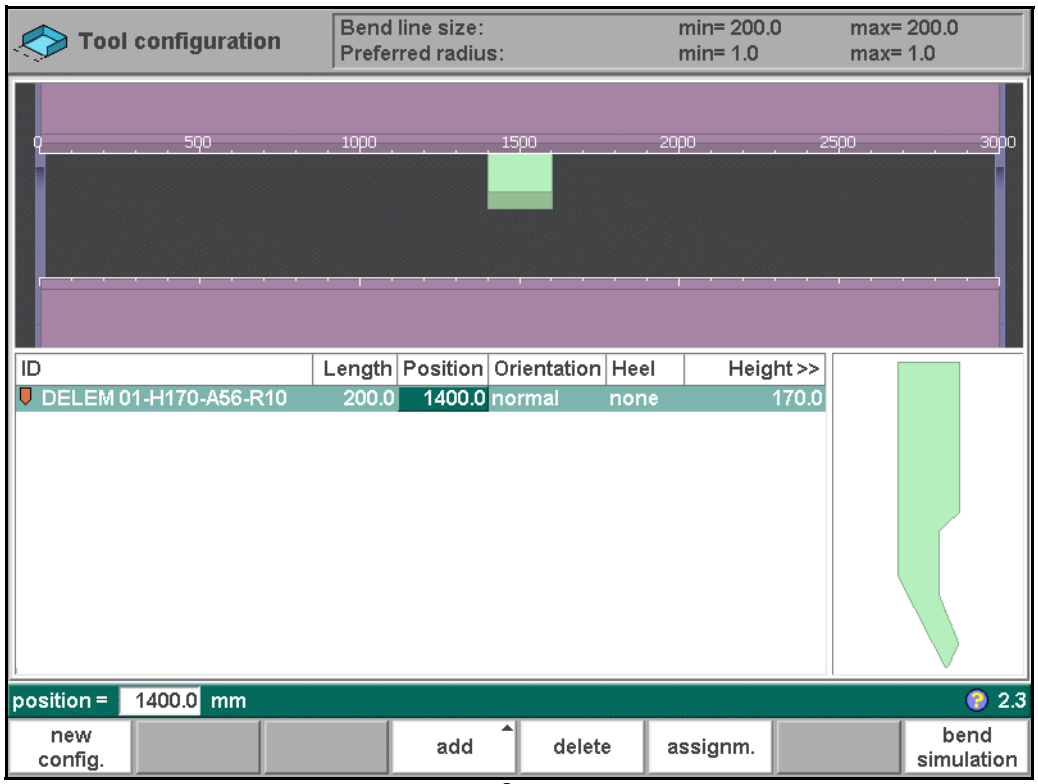
3.c

When a tool has been chosen (e.g. a punch), it is placed in the machine with maximum available length.



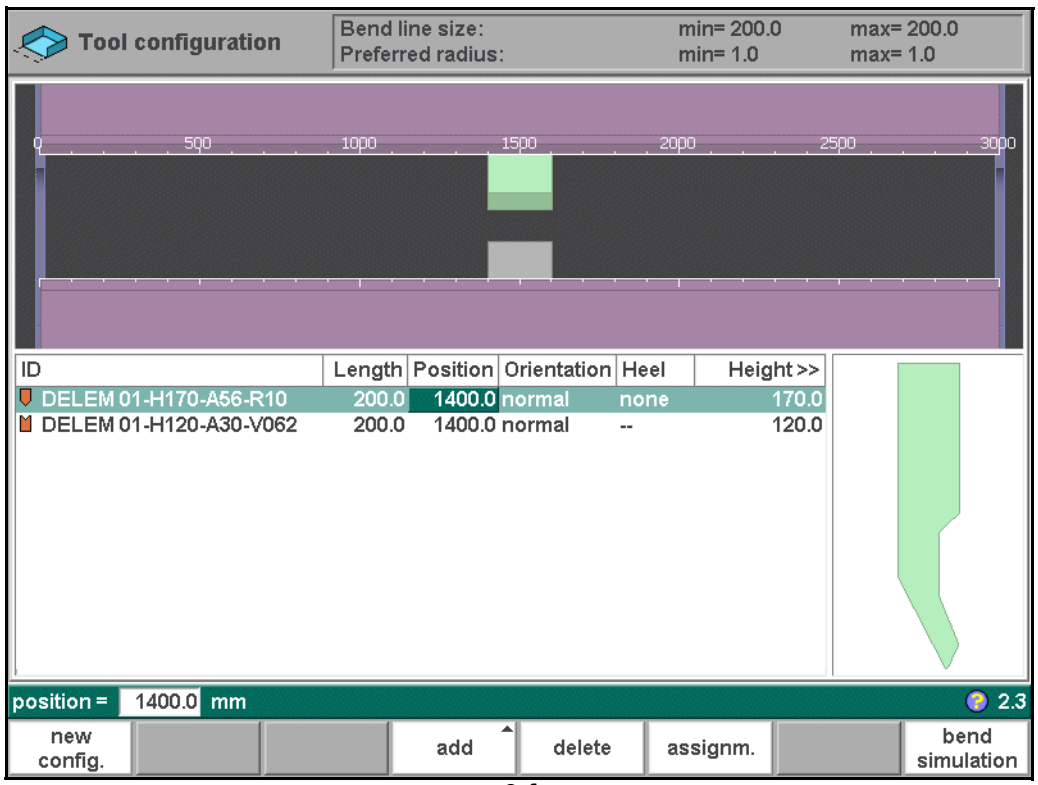
3.d

After a tool is placed, the tool ID can be changed by typing an ID, by pressing the ⌂ key to get a short list of tools or by pressing the function key ‘show library’ to get an overview of available tools. If only a part of the tool ID is typed, the control automatically offers a list of tools with the typed characters.
To change length and position of a tool, move the cursor to the appropriate field, type a new value and press ENTER. When the length and position are changed, the tool is ready.



3.e

After the punch is finished, a die with the default ID, length and position is put below the punch.



3.f

The tool set-up can be modified with the available function keys. Punches and dies can be added or deleted, the existing tools can be moved to a different position or their length can be modified.

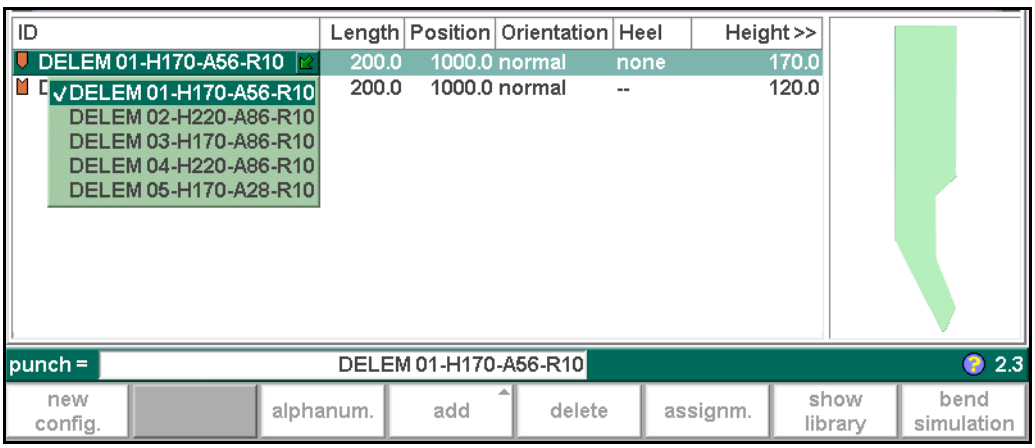
Function keys:

S1	**new config.**	Start a new tool configuration; the existing configuration is deleted.
S4	**add**	Add a new tool to the tool configuration. When pressed, some new functions keys are offered to choose the tool: punch, die or adapter (if enabled).
S5	**delete**	Delete the currently selected tool.
S6	**assignm.**	Open a screen with parameters for bend sequence computation.
S7	**show library**	Open a window with an overview of the tool library.
S8	**bend simulation**	Start bend simulation.

3.2.2. Tool selection

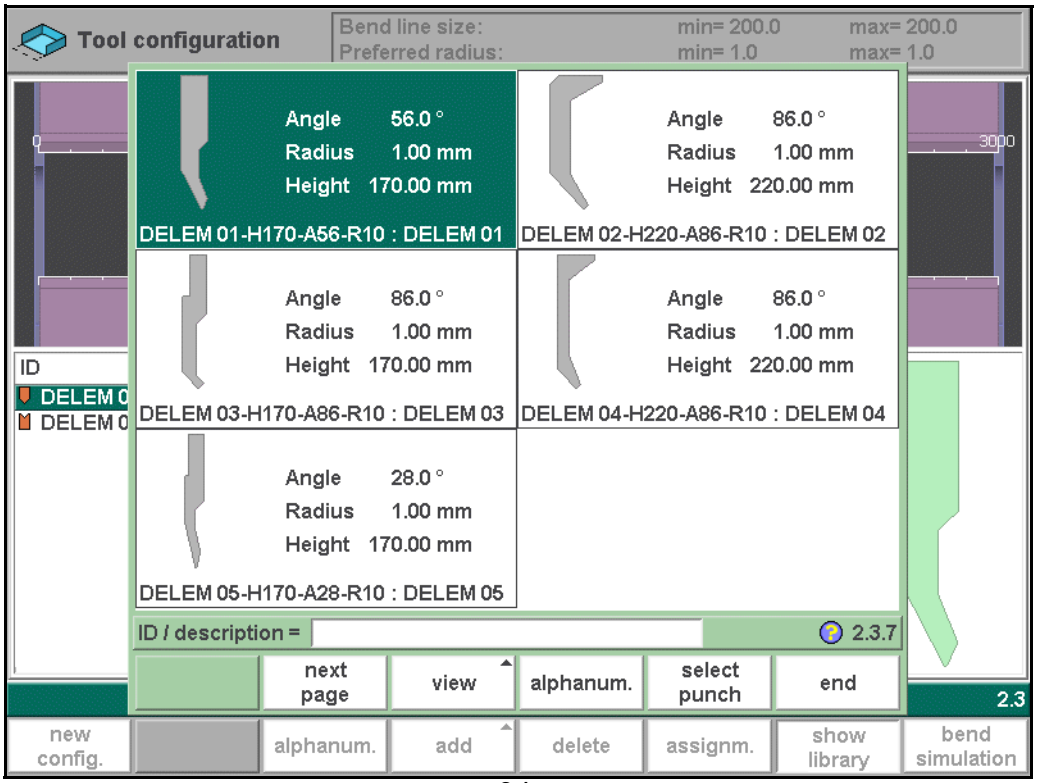
When the control prompts for a tool (or when the cursor is placed on one), there are several ways to select one.

- If the ID is known, type the ID of the required tool.
- Press the key ⌂ to open a pop-up with available tool IDs. Then move to the required ID and press ‘enter’.



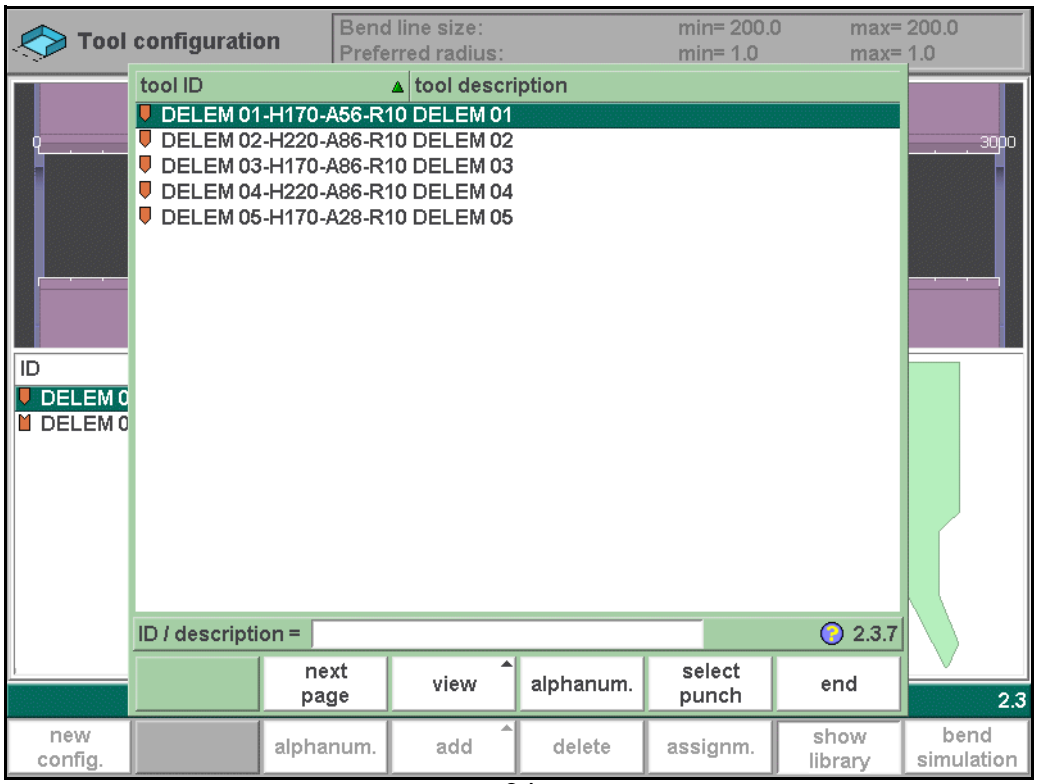
3.g

- Press the function key ‘show library’ to get a new window with an overview of available tools.



3.h

If necessary, press the keys ‘view’ (S4) and ‘expanded dir.’ (S4) to get a basic overview of the available tools.

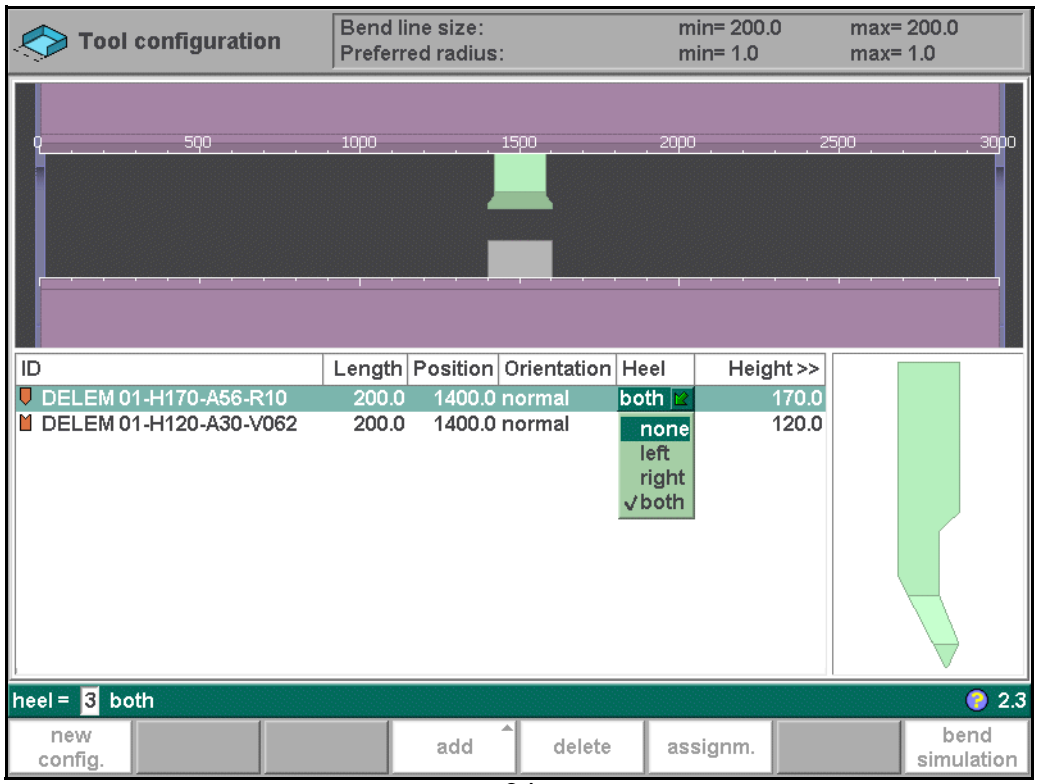


3.i

In any of these windows, move the cursor to the required tool and press ‘enter’ or the function key ‘select’ (S6) to select the tool.

3.2.3. Heel programming

To program a punch with heels, move the cursor to the ‘Heel’ field. Press the key ⌂ to select the required setting.



3.j

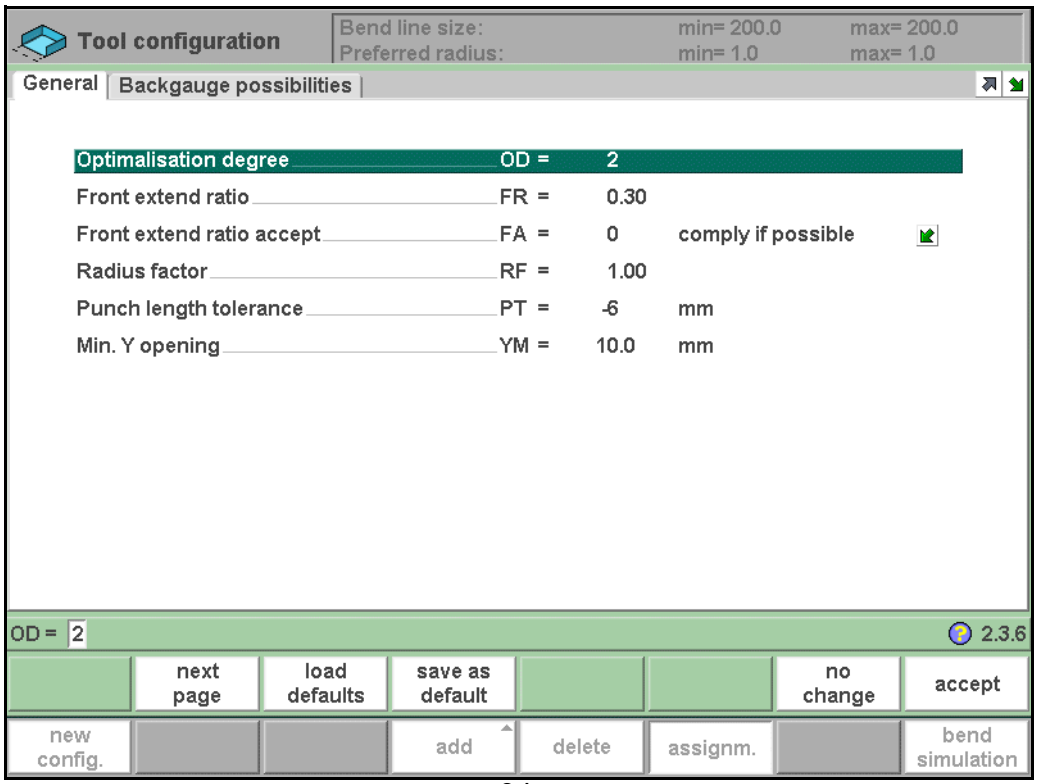
Heels kan only be assigned to a punch if the punch has heel properties. See also chapter 6 for more information about punches.

3.3. Assignments

3.3.1. Introduction

The Assignments are parameters with which the bend sequence computation is controlled. The Assignments screen is opened from the tool configuration screen with the function key ‘assignm.’.

Automatic bend sequence computation works with several conditions in order to find an optimum between a minimum production time, handling possibilities without product/machine and product/tool collision.
In order to find one of the optimums you must program several computation parameters with which the bend sequence can be computed.
Some of these parameters are machine-related and some are related to product accuracy, handling possibilities and turn times.



3.k

Function keys:

S3	**load defaults**	Load default assignment settings. It is possible to determine a set of assignments which have the most optimal values for your situation. This set can be stored by pressing function key S2 (save as default). While programming another product you can recall this previous fixed set by loading the values via function key S1.
S4	**save as default**	Save the current settings as default assignments settings.
S7	**no change**	Leave the current screen without saving changes. This can also be done by pressing END.
S8	**accept**	Save the changes and leave the current screen.

3.3.2. Assignments - general

Optimisation degree . . . . . .OD

Range 1-5.

The number of alternatives to be computed for each bend must be entered here. The higher this number the more alternatives are to be examined by the control, so the longer the computing time will be.

Front extend ratio . . . . . .FR

Range 0.0 - 1.0.

This is the ratio of the minimum allowable length of your product which extends in front of the press to the total blank length of the product. You must have a minimum length of your product in front of the press to be able to handle the product.

Front extend ratio accept . . . . . .FA

0 = comply if possible:

This means that when possible the computer tries to comply to the front extend ratio and only when this will result in no solutions to be found it will accept that the length in front is smaller than the specified ratio.

1 = comply always:

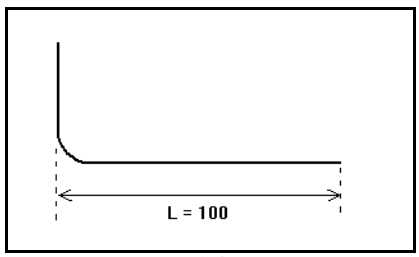
The computer will always comply to the front extend ratio. This may result in no solutions to be found.

Press the key ⌂ to select the required setting.

Radius factor . . . . . .RF

The computed radius of a bend is multiplied by this factor for correction purposes. This correction affects the X-axis position and the computed blank length. The initial value of RF is 1.

The controller computes the X-axis position necessary to obtain L=100 as shown in figure 3.I. 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 RF you can optimise this computation.

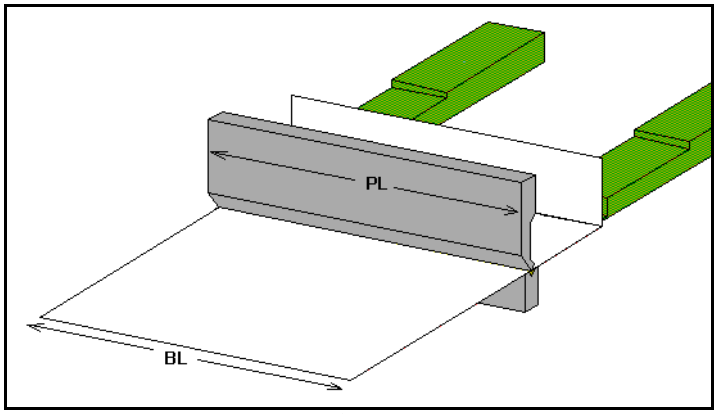


3.I

Punch length tolerance . . . . . .PT

The punch length may be shorter than the length of the bendline.

The maximum allowed difference between the punch length and bend length can be programmed here. Punch length tolerance may influence your bend sequence: if the length of a punch is shorter than the tolerance permits, it will not be accepted for a bend. See figure below in which the Punch length has been indicated by PL and the bending length by BL. The tolerance value is the difference between PL and BL.

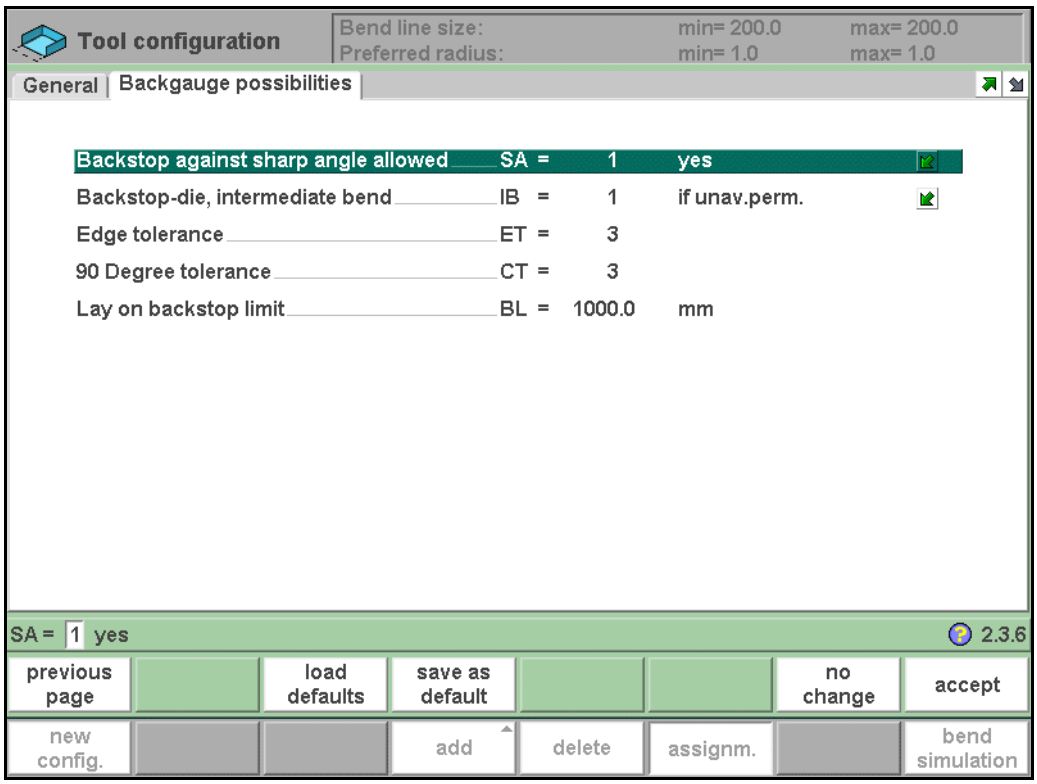


3.m

Minimum Y 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).

3.3.3. Assignments - Backgauge possibilities

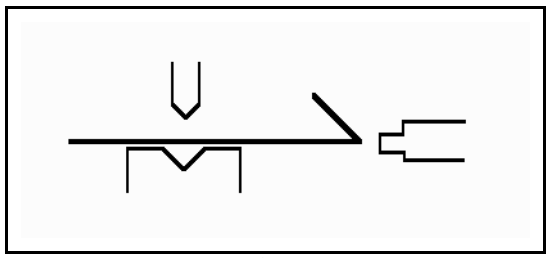


3.n

Backstop against sharp angle allowed. . . . . .SA

Specify if backstop may be placed against an angle smaller than 90°.
0 if not allowed
1 if allowed

Press the key ⌂ to select the required setting.

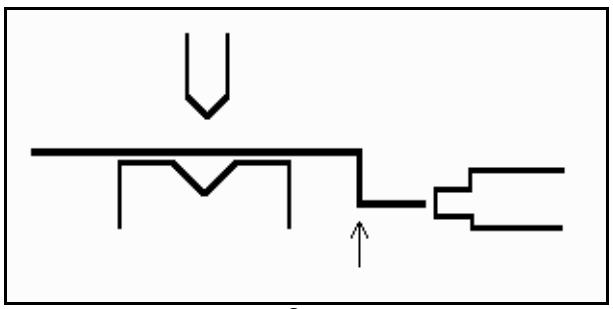


3.o

Backstop-die, intermediate bend . . . . . .IB

Set to allow if there may be a bend between the die and backstop.
Selection possibilities:
0 = permitted
1 = if unavoidable permitted: if it results that no solutions are to be found, than it is permitted
2 = prohibited: never allowed.

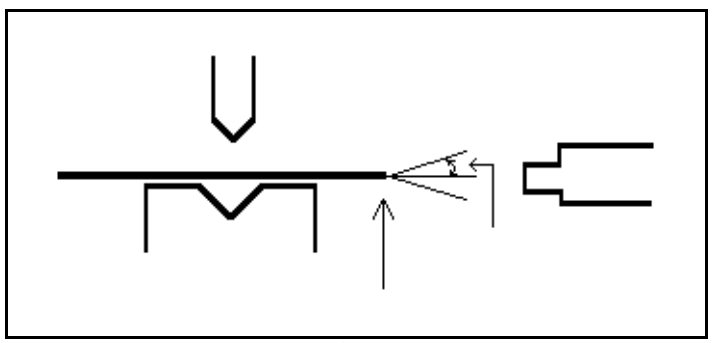
Press the key ⌂ to select the required setting.



3.p

Edge tolerance . . . . . .ET

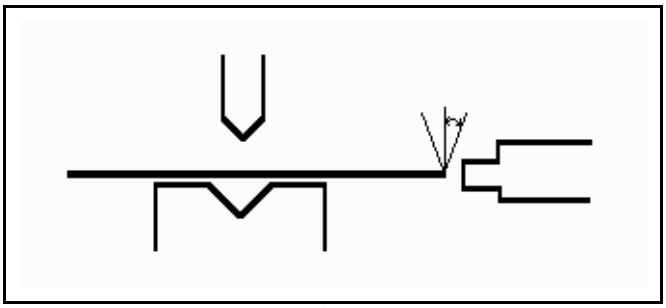
In case backstop is against flat sheet an angle tolerance is allowed (deviation from horizontal).
To be programmed in degrees of tolerance (0 - 90° input).



3.q

90 Degree tolerance . . . .CT

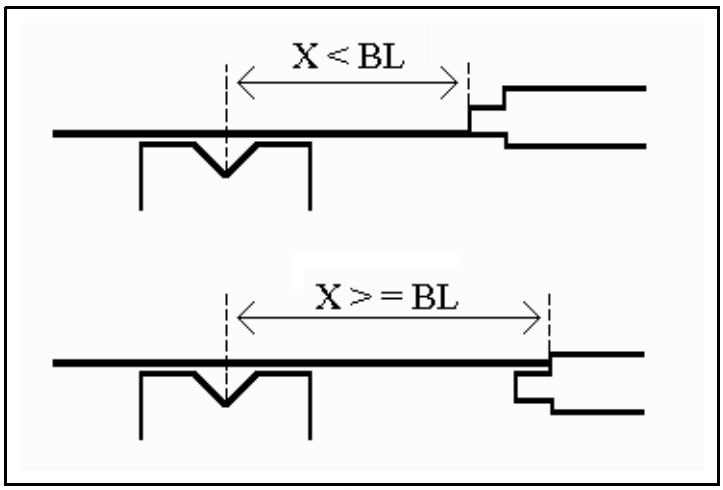
The maximum allowed deviation from vertical (90°), when the backgauge is against a bent angle which is not 90°.



3.r

Lay-on backstop limit . . . .BL

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

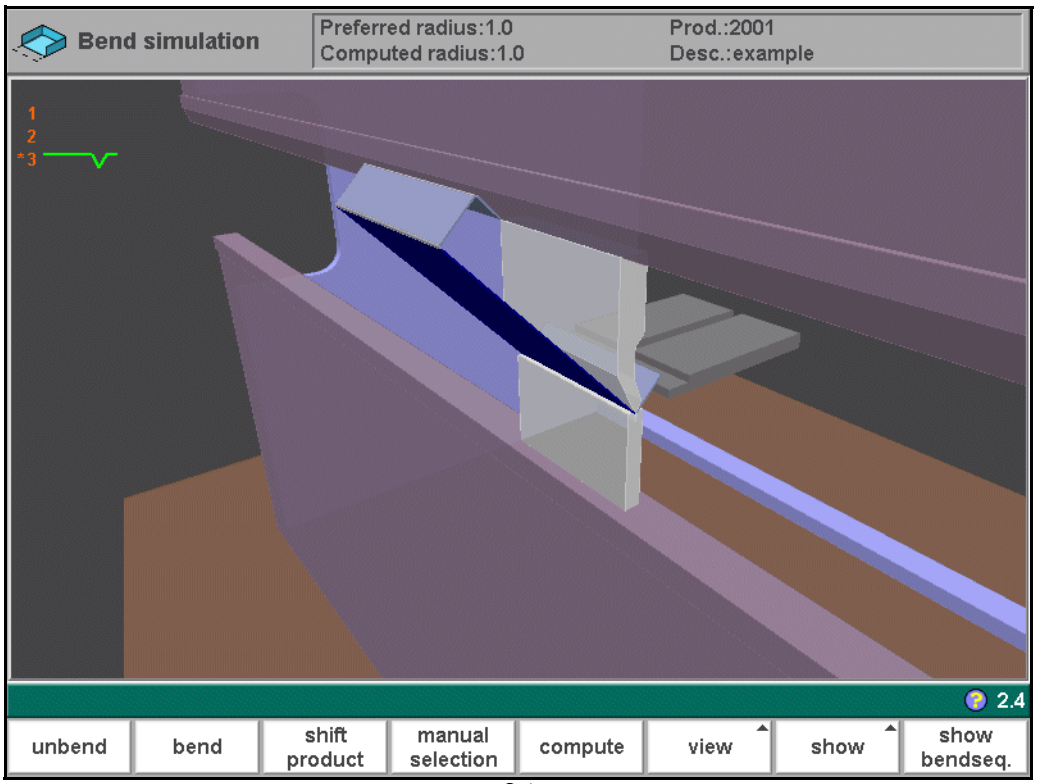


3.s

3.4. The bend sequence in 3D

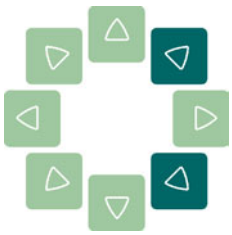
3.4.1. Introduction

When a tool configuration is available, the bend simulation can be started to determine a bend sequence for the active product. The bend simulation is started by pressing the function key ‘bend simulation’ in the tool configuration screen.



3.t

On this screen the product appears between the tools in a possible last bend position. When starting the simulation, the product is shown in its final state. In order to obtain a bend sequence, the product must be unfolded from the last bend to the first. This can be done with the available function keys.



The product can be zoomed in or out with the cursor keys.

Function keys:

unbend Unbend the currently shown bend or start searching for the next feasible bend to unfold.

bend Bend the product in the simulation screen or switch to the next bend step.

Function keys:

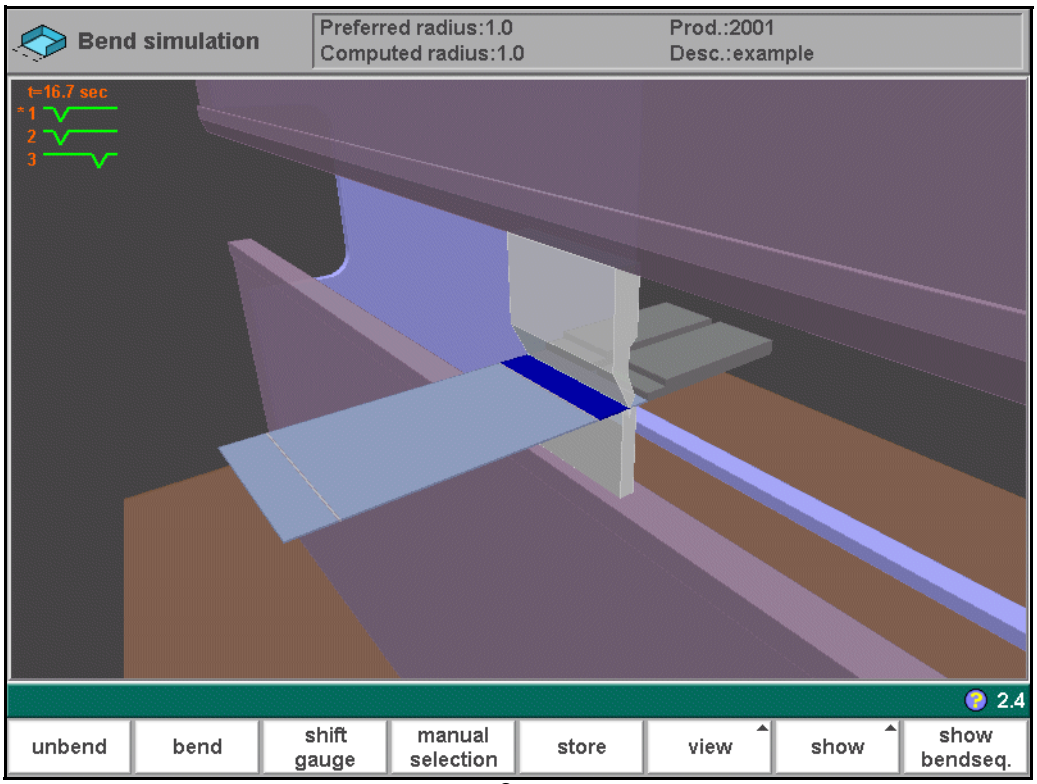
S3	**shift gauge / shift product**	Shift the product manually (if product is bent) or shift the gauge manually (if product is unbent).
S4	**manual selection**	Manual selection of a bend line. Additional possibility to determine the bend sequence.
S5	**compute**	Compute a valid bend sequence.
	/	
	store	Compute a numeric program and save it on disc. The resulting CNC program contains all necessary axis positions and tool numbers. This command can only be executed when a complete bend sequence has been determined.
S6	**view**	Button to select a possible viewing mode: - 2D - 3D
S7	**show**	Toggle between some possible ways of displaying the product/tool configuration: - product - product/tools - all
S8	**show bendseq**	Show bend sequence. To show a mosaic screen with a step-by-step graphical overview of the bend sequence.
END	**END**	Return to the drawing.

3.4.2. Unbend product

In order to generate a CNC-program the bend sequence must be known. There are two ways to achieve this:

- Press the function key ‘compute’ (S5). The control will automatically compute the quickest possible bend sequence for this product.
- Press the function key ‘unbend’ (S1) repeatedly, until the product is completely unbent.

When the product is completely unbent, press the function key ‘store’ (S5) to generate and store the CNC-program.



3.u

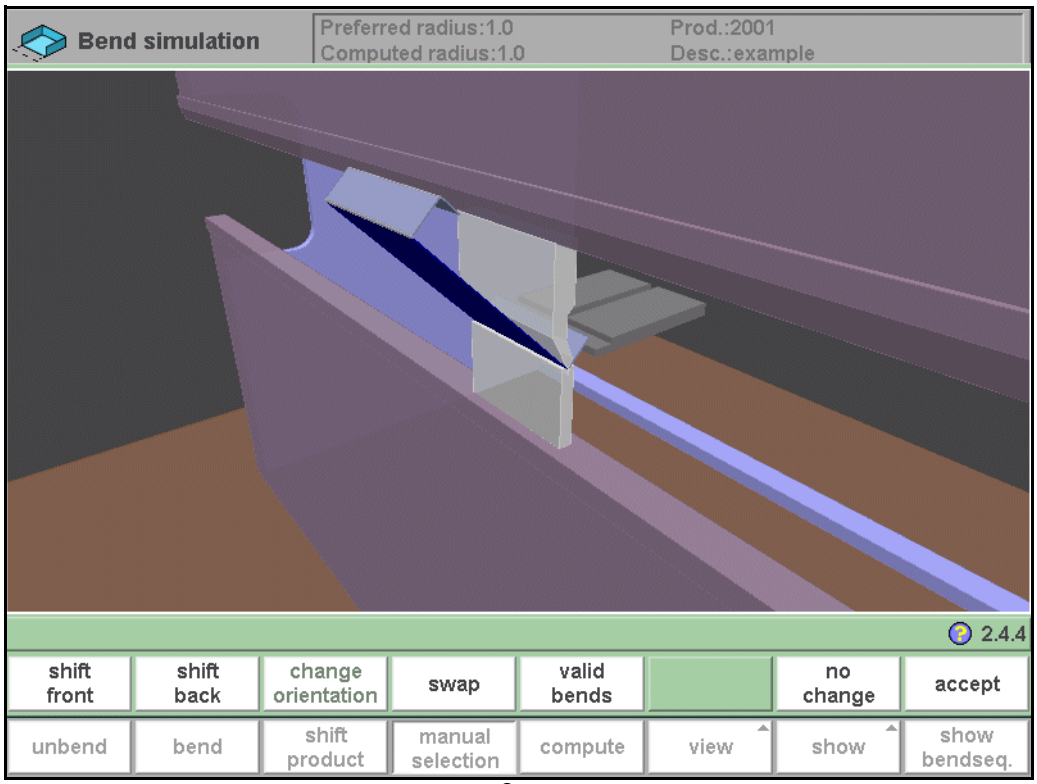
It is possible that no bend sequence has been found for various reasons:

- The installed tools are not correct. Return to the tool configuration menu to change the tool configuration.
- The assignments are incorrect. Return to the assignments menu to modify the assignments.
- A collision has been detected during unbending. It is possible to manually adjust the bend sequence with the function keys. This is explained in the following sections.

When a bend in the bend sequence is adjusted manually with one of the available functions, it must be confirmed by pressing ‘unbend’. Otherwise, it will not be stored in the sequence.

3.4.3. Manual selection of bends

Normally the control proposes the next (un)bend in a sequence. This is computed by the control depending on the programmed assignments and of course the product shape and applied tools.
For various reasons it can be necessary to choose another bend line for the bend sequence. The bend sequence can be changed/determined through the function ‘manual selection’. When the function key ‘manual selection’ (S4) has been pressed, a new window is opened.



3.v

Function keys:

S1	**shift front**	Shift the product to the front.
S2	**shift back**	Shift the product to the back.
S3	**change orientation**	Turn the product 90 degrees. Only possible for a 3D product.
S4	**swap**	Turn the product between the tools (back to front).
S5	**valid bends**	Indicate other possible bends that can be selected. When pressed, other possible bends are highlighted.
S7	**no change**	Leave the current screen without saving changes. This can also be done by pressing END.

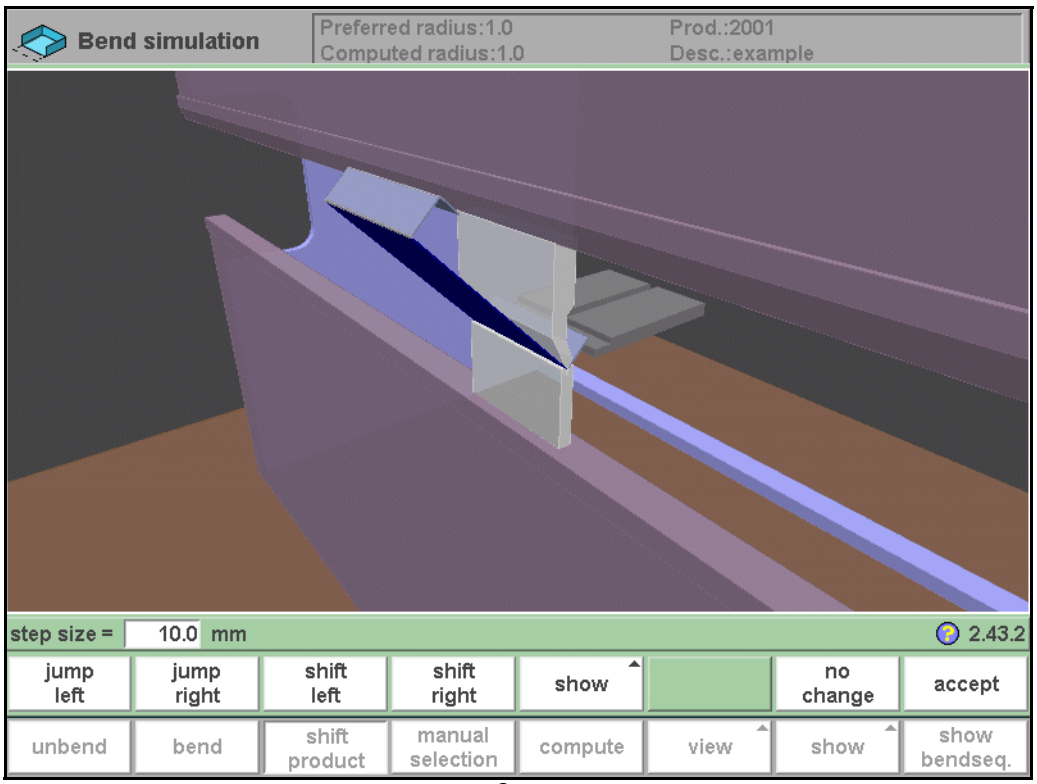
Function keys:

S8 **accept** Save the changes and leave the current screen.

Note:
When a bend in the bend sequence is adjusted manually with one of the available functions, it must be confirmed by pressing ‘unbend’. Otherwise, it will not be stored in the sequence.

3.4.4. Move product

In the bend simulation menu, the control computes the next possible bend to unbend. The product is placed between the tools, where there is no collision with the tools or the machine. In case you want to shift the product under the tool set (which is mounted), you can move the product by pressing the function key ‘shift product’ (S3). A new window appears.



3.w

Function keys:

S1 **jump left** Move the product to another toolset combination.

S2 **jump right** Move the product to another toolset combination.

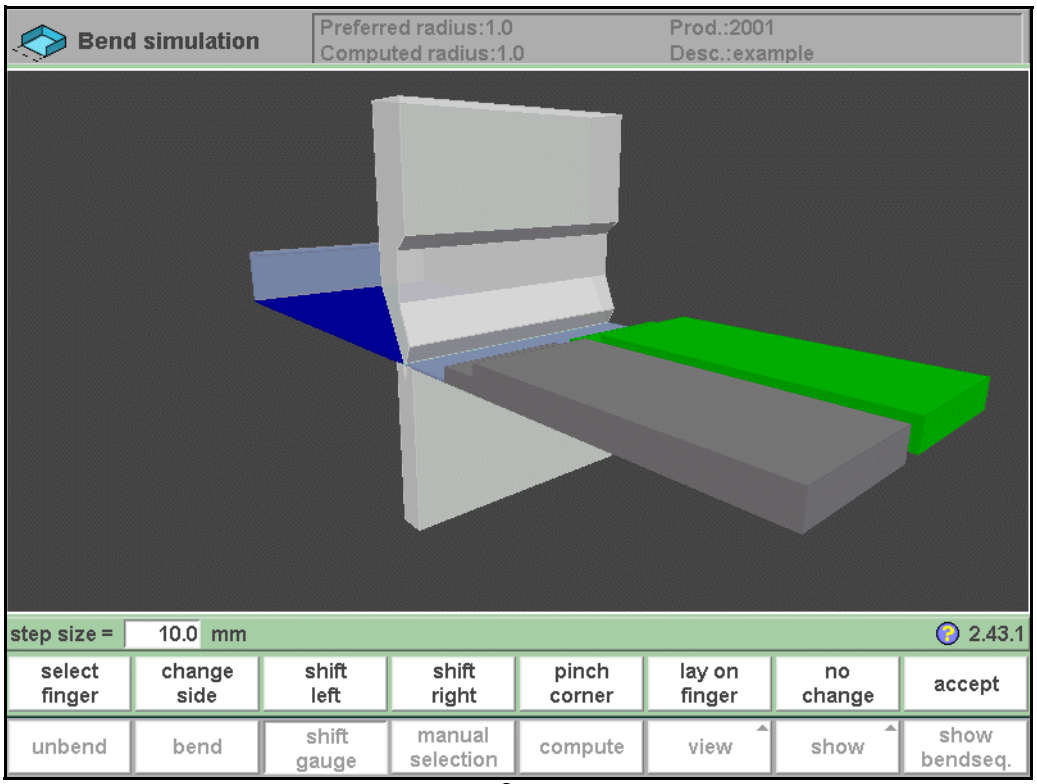
Function keys:

S3	**shift left**	Shift product to the left within the same toolset. The step size is displayed at the command line prompt and can be changed.
S4	**shift right**	Shift product to the right within the same toolset. The step size is displayed at the command line prompt and can be changed.
S5	**show**	Select a possible view of the simulation: - product - product/tools - all
S7	**no change**	Leave the current screen without saving changes. This can also be done by pressing END.
S8	**accept**	Save the changes and leave the current screen.

Note:
When a bend in the bend sequence is adjusted manually with one of the available functions, it must be confirmed by pressing ‘unbend’. Otherwise, it will not be stored in the sequence.

3.4.5. Move fingers

The control automatically computes at each bend the X-axes, R-axes and Z-axes positions. It takes into account the values of the option assignments and searches for a solution without collision of the fingers with the product. In order to be able to choose alternative positions, you can move the fingers manually.
When the product is unbent, press ‘shift gauge’ (S3). A pop-up window displays the backgauge fingers with one finger highlighted.



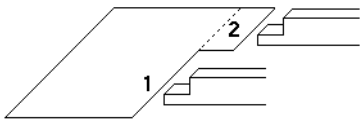
3.x

The fingers can be moved by means of the function keys S2 through S4. The step size of the finger can be changed by entering the desired value. The higher the value of the step size, the bigger is the displacement.

Function keys:

S1 **select finger** Select the finger to move.

S2 **change side** Change side. Move the selected finger to another side of the product behind the machine. In the example there are two possible finger positions to choose.



This way is only possible and will only be accepted if the required axes are enabled in the control (like X1 and X2)

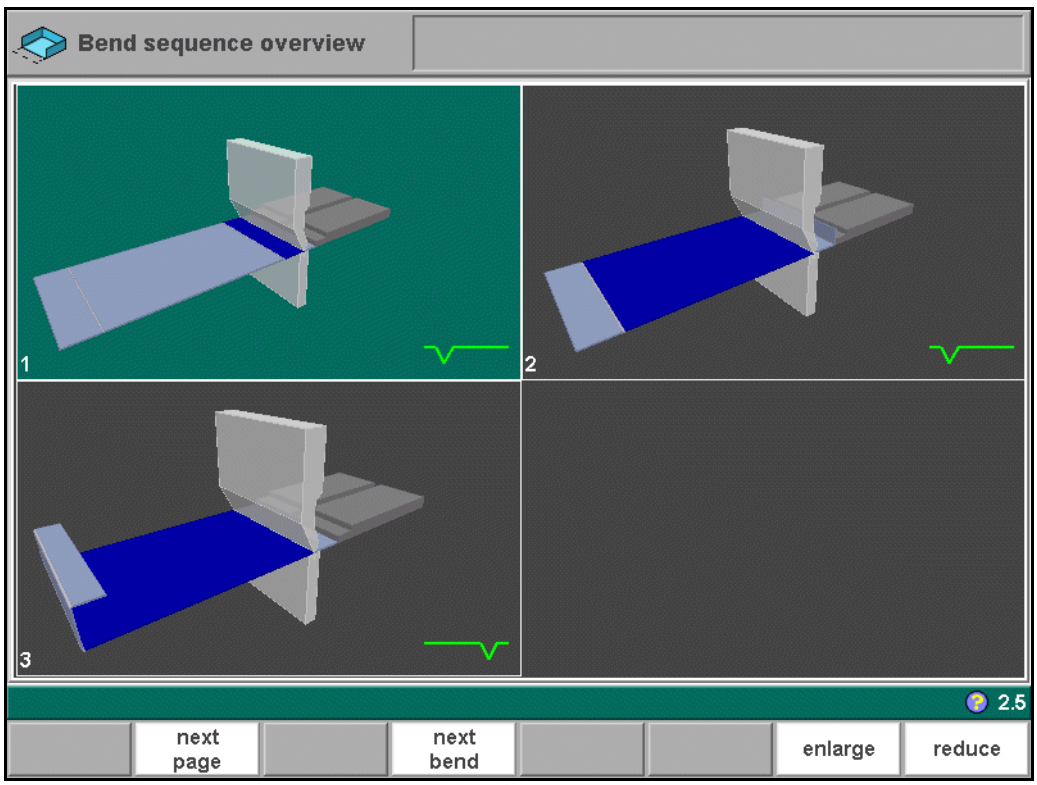
S3 **shift left** Shift the selected finger to the left. The step size is displayed at the command line prompt and can be changed.

Function keys:

S4	**shift right**	Shift the selected finger to the right. The step size is displayed at the command line prompt and can be changed.
S6	**lay on finger**	Toggle between lay product on selected finger or not. This option is only selectable in case you have R-axes in your machine.
S7	**no change**	Leave the current screen without saving changes. This can also be done by pressing END.
S8	**accept**	Save the changes and leave the current screen.

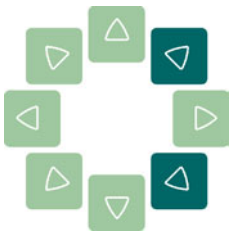
3.4.6. Show bend sequence

Before the product is completely unbent you can enter the ‘show bendsequence’ menu.



3.y

This menu option can be called at any time after the first unbend has been made. The graphical overview displays the determined bends as well as the not yet determined bends (question mark sign).
The total overview can be enlarged or reduced with the function keys ‘enlarge’ and ‘reduce’. As a result, more or less bends will be shown on one page.

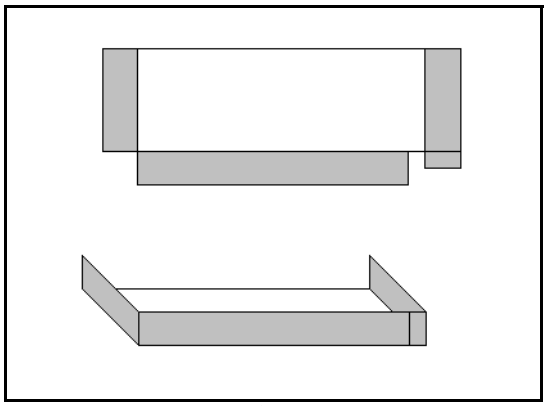


Each image in the overview can separately be enlarged or reduced with the zoom keys.

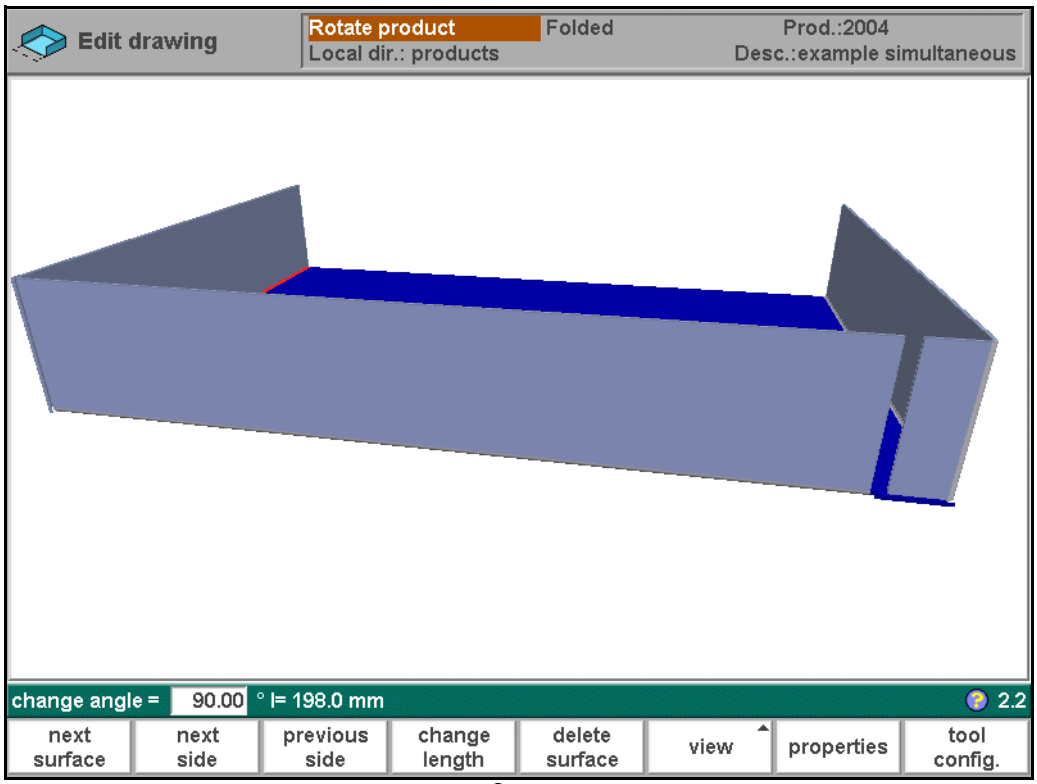
They can also be rotated with the arrow keys.

3.4.7. Simultaneous bends

When several bends extend each other (the bends run in one line, see figure below), the control will try to execute these bends simultaneously. This can only happen when all the tools have the right length and all bends have the same angle.

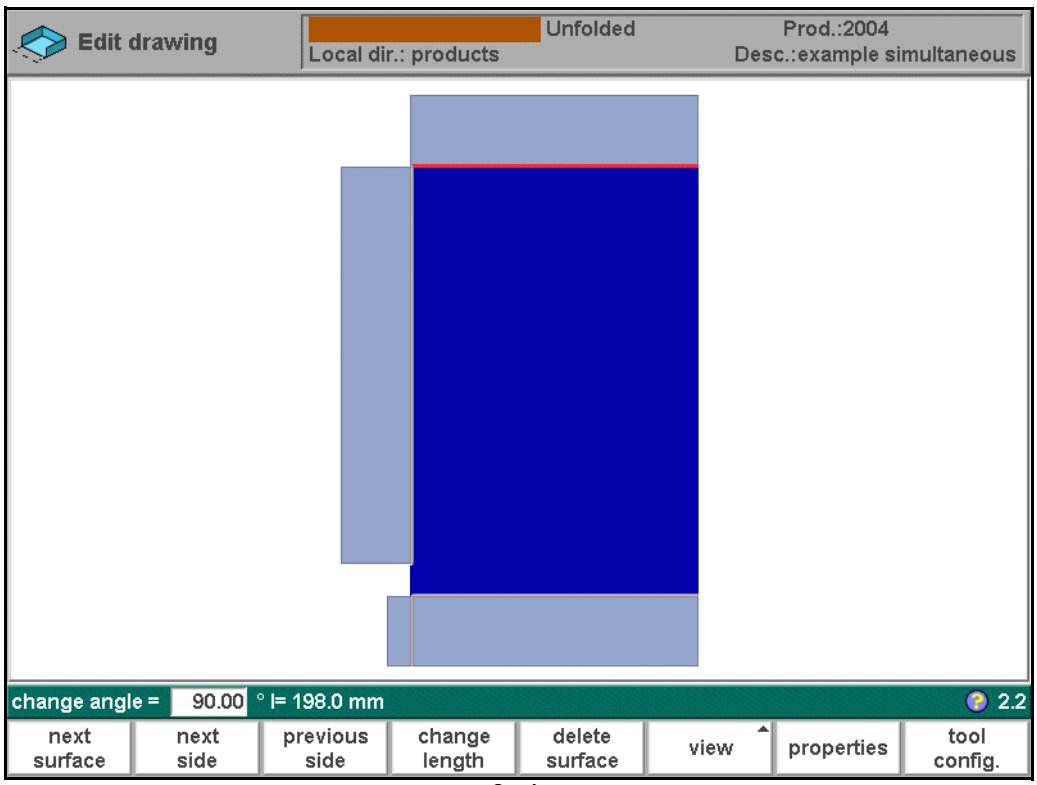


3.z



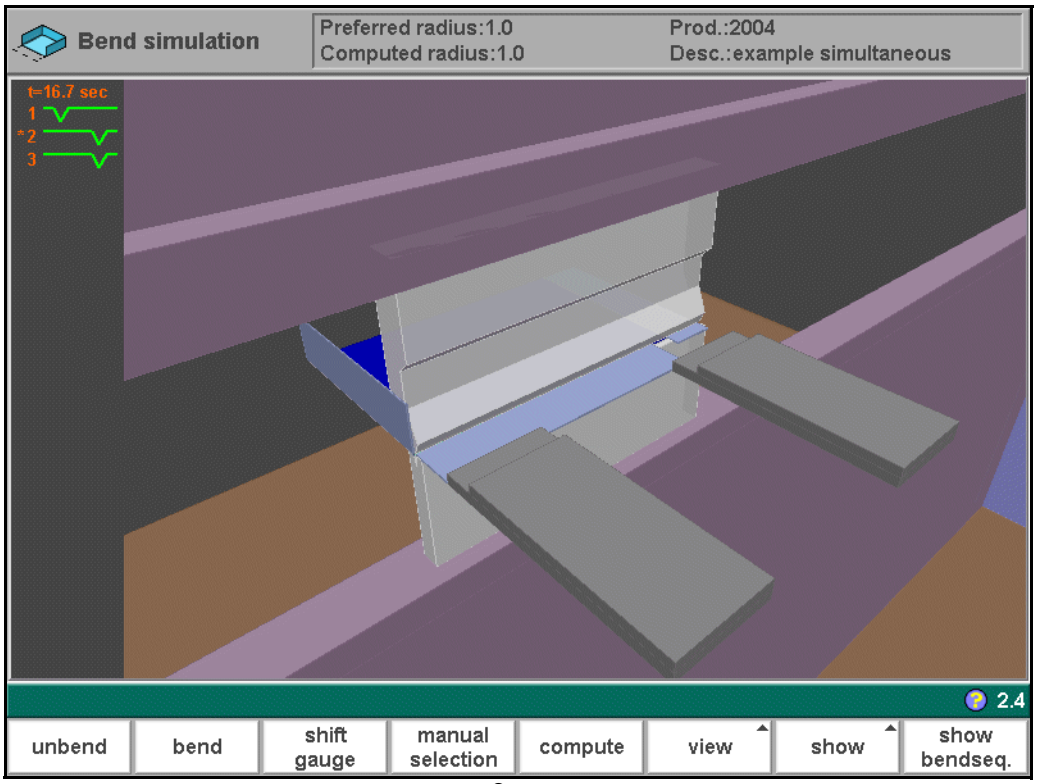
3.aa

This example shows two combined bends, which are clearly visible when the product is unfolded:



3.ab

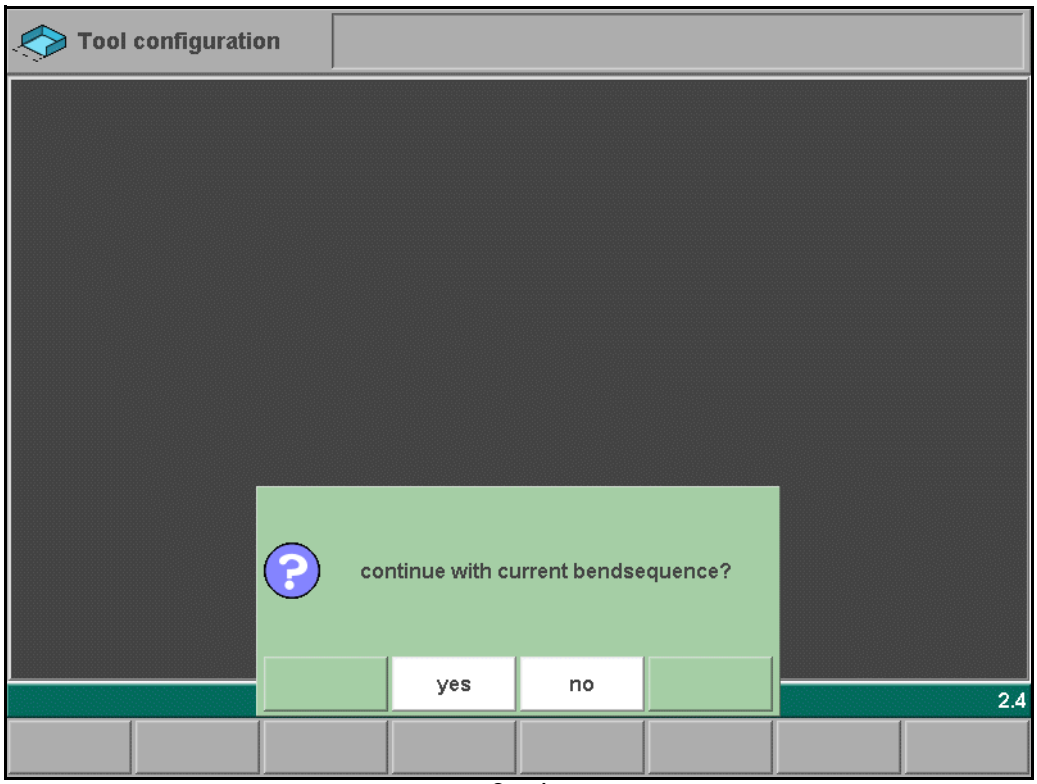
In the bend sequence menu, the two bends are combined in one bend step.



3.ac

3.4.8. Opening an existing bend sequence

When the simulation mode is entered and the product already has a bend sequence, the control will ask what to do with the existing information.
If the bendsequence has been computed during the current session, the control simply asks whether or not to proceed with the current information.

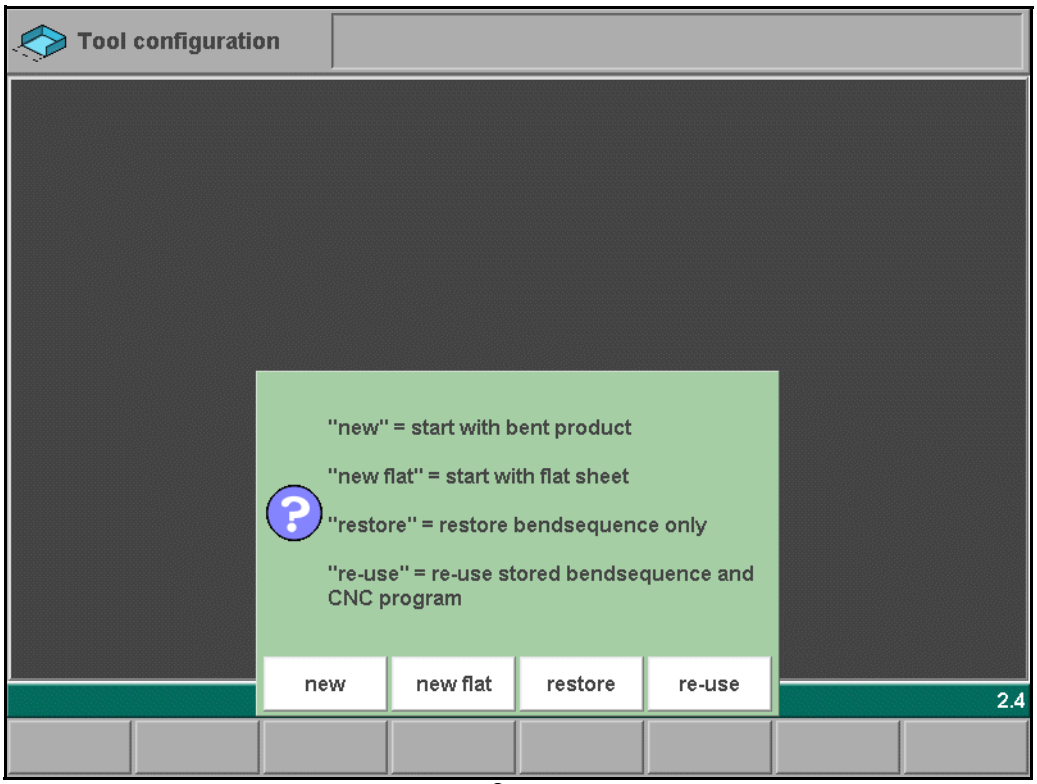


3.ad

Function keys:

S4	**yes**	The control switches to the simulation mode, where the bendsequence can be edited.
S5	**no**	Do not continue with the currently active bendsequence.

If ‘no’ has been selected, a new window appears to ask how you wish to proceed.



3.ae

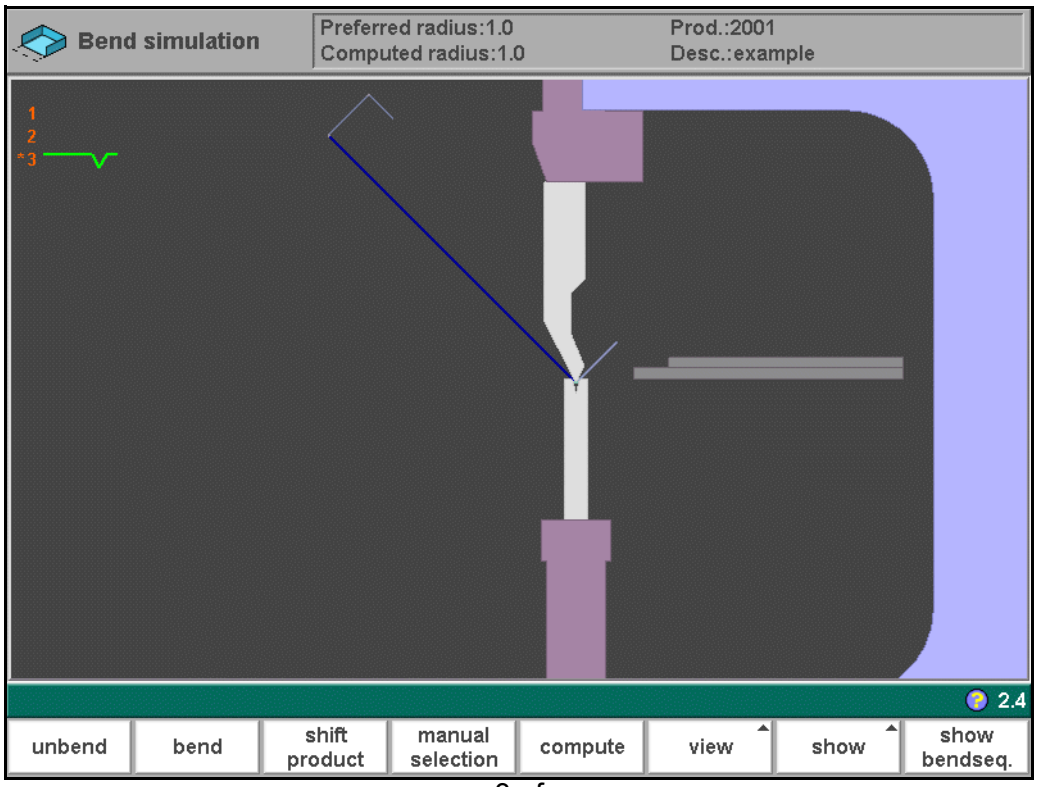
Function keys:

S3	**new**	Start a new bend sequence, any existing sequence is removed.
S4	**new flat**	Start a new bend sequence from a flat sheet.
S6	**restore**	Restore an existing bend sequence from disk, but disregard any related CNC program.
S7	**re-use**	Restore an existing bend sequence from disk, including the related CNC program.

If a product with bendsequence information has just been loaded, the control will open this window when the simulation is started. This would be the case if a product has just been selected in menu 5 (Product selection) or when the control has been restarted.

3.5. The bend sequence in 2D

The bend sequence can be also be viewed and programmed in a 2D view. This may work faster for simple 2D products. If a product has been drawn in 2D mode, the simulation screen starts in 2D mode.



3.af

In the 2D mode the same functions are available as in the 3D mode, with some exceptions:

- The product cannot be turned 90 degrees.
- The backgauge fingers cannot be moved in the Z-direction.

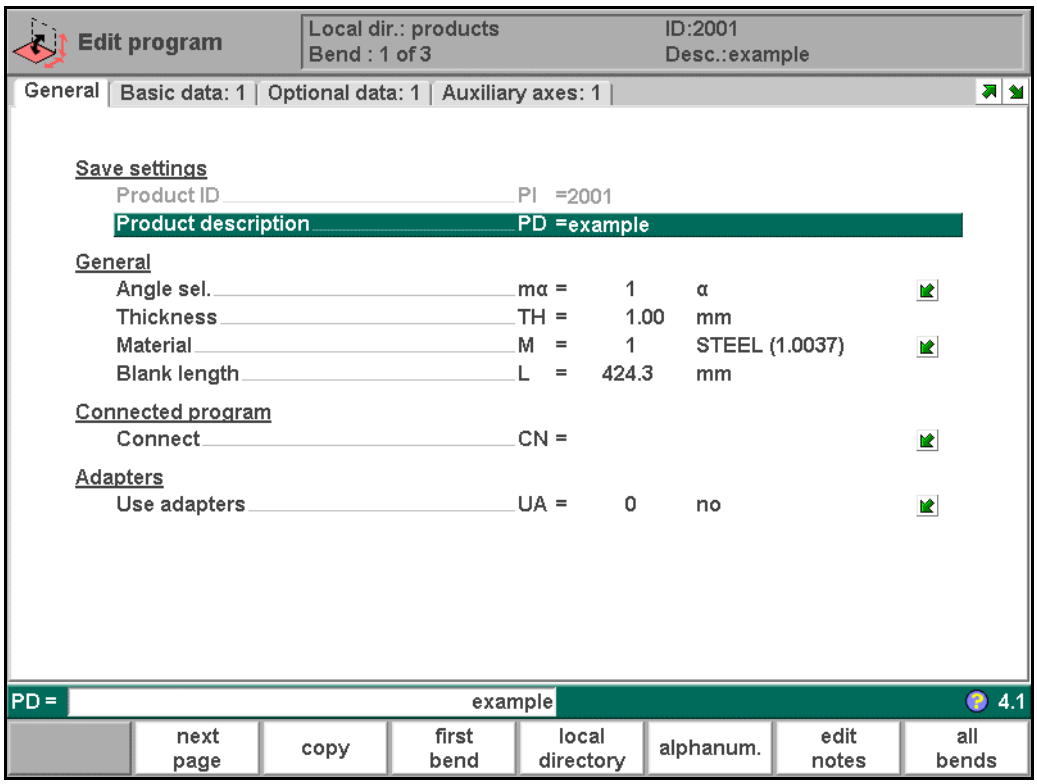
4. Product programming

4.1. Introduction

To create a new CNC program, choose **prepare program** in the main menu.

To edit an existing CNC program, choose **edit program** 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 library’ in the main menu. In the offered list, you have to select the wanted program number.



4.a

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

Function keys:

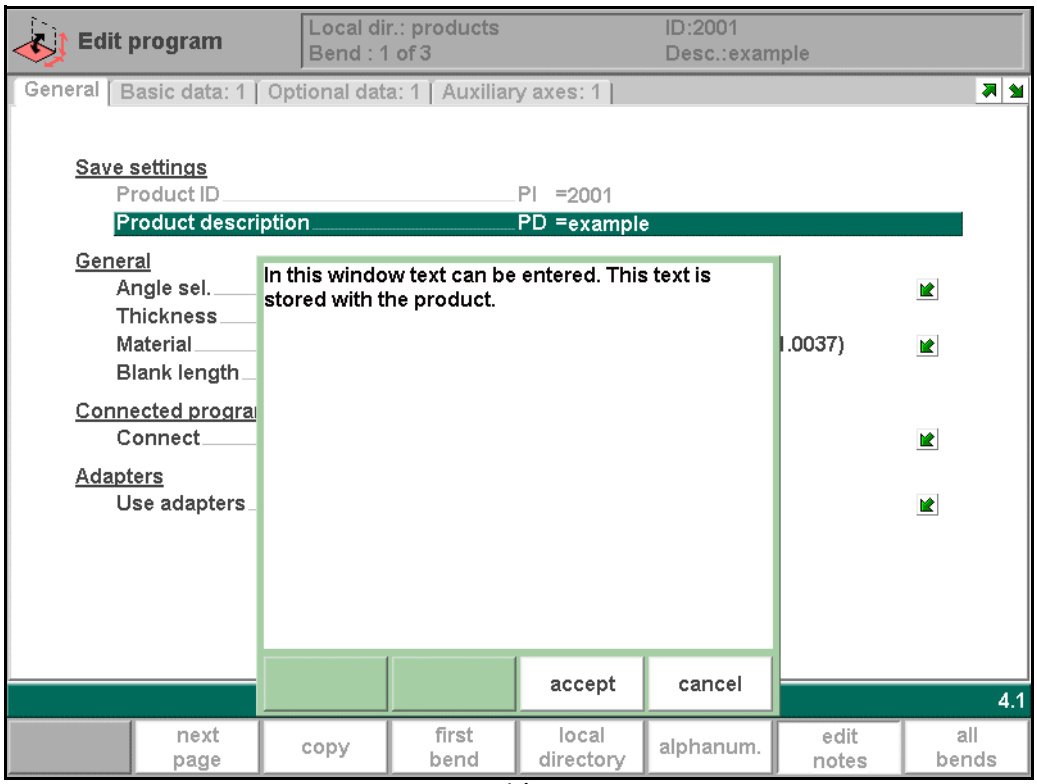
next page Switch to the next page with bend information.

Function keys:

S3	**copy product**	Copy the current product. When pressed, you must enter a new product ID for the new copy.
S4	**first bend**	The cursor jumps to the page with the first bend information.
S5	**local directory**	Choose another location (directory) on the local disk to store the current product. The product is automatically copied to the new location.
S6	**alphanum.**	Select an alphanumeric character after typing. The possible characters are displayed on the frontpanel key. Only possible at relevant parameters.
S7	**edit notes**	Open a window with notes about the current product.
S8	**all bends**	Open a new window, in which all bends are shown in a table.
END	**END**	Finish the data preparation and return to the program menu.

- **Edit notes**

It is possible to add a note to your product, in order to store comment or background information about the current product.



4.b

The note is a simple text field, it has no influence on product values or bend sequence calculations. The maximum allowed length of the note is 1023 characters.
The possible characters in a note are the following:
a-z
A-Z
0-9
?!%*/\+-:;,._
To edit a note, a keyboard is needed. Depending on the control type, you can connect an external USB keyboard or use an integrated keyboard option.
To exit the note and save the changes, press the function key ‘accept’.
To exit the note and ignore the changes, press the function key ‘cancel’.

- **Parameter explanation**

Product ID . . . .PI

A unique name to identify a product program. The maximum length is 25 characters. The product ID may contain letters and numbers. Letters can be entered with the help of the function key ‘alphanum.’ (S6).

Product description. . . . . ..PD

A name or description of this program. The maximum length is 25 characters. The product description may contain letters and numbers. Letters can be entered with the help of the function key ‘alphanum.’ (S6).

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 in a bend step.

Press the key ⌂ to select the required setting.

Thickness . . . . . .TH

Thickness of the sheet.

Material . . . . . .M

Selection of one of the programmed materials, which are used to calculate the bending depths. The control contains 4 pre-programmed 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.

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 out of two programs.

Use adapters . . . . . .UA

Program whether or not a tool adapter is used for this product. The use of an adapter affects the calculated Y-axis position for all bends. Whether an adapter can be programmed depends on the parameter ‘enable adapters’ in the programming constants.

Press the key ⌂ to select the required setting.

• 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:

1 Create the product in one direction.

2 Create the product in the other direction.

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

3 Select the program with the bend sequence in the direction which you want to execute in the first place. You select the program of the product via menu 5, ‘product library’.
4 Go to menu 4, ‘edit program’. 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.

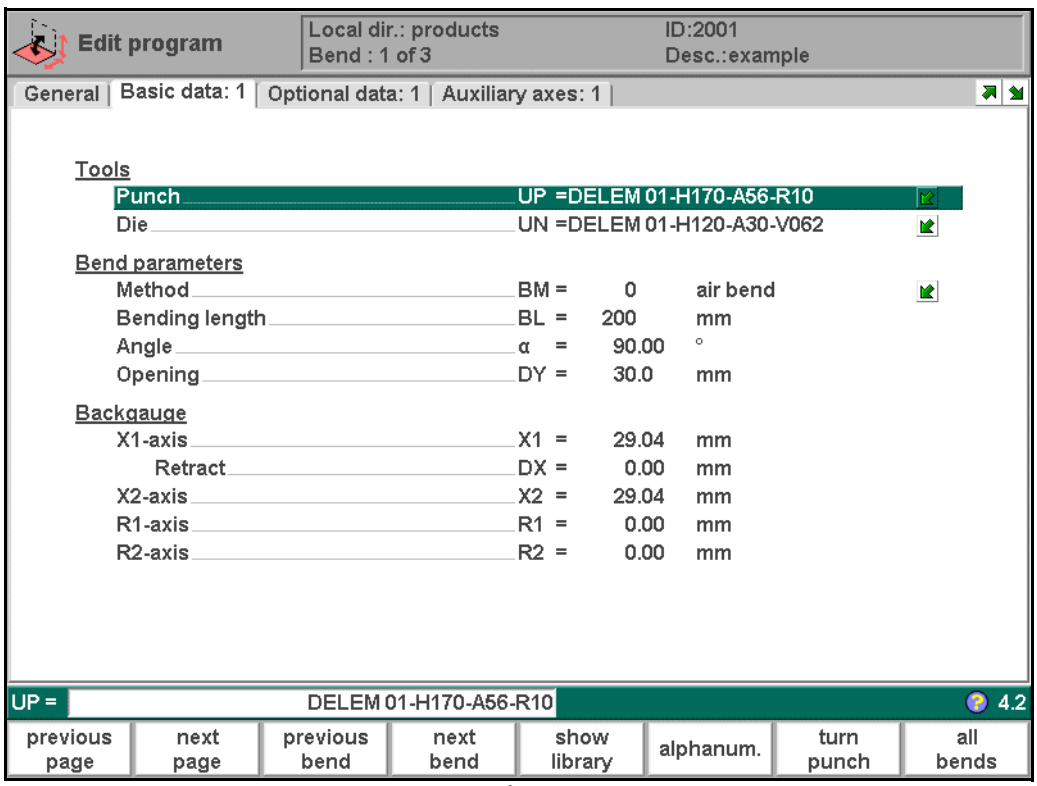
To produce products with connected programs the next four actions are necessary.

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.

4.2. Bend parameters - basic data

The parameters of one bend are divided over several pages.



4.c

The bend number, product ID and product description are displayed in the top row on the screen.

Function keys:

S1	**previous page**	Select the previous page with parameters of this bend.
S2	**next page**	Select the next page with parameters of this bend.
S3	**previous bend**	select previous bend
S4	**next bend**	select next bend
S5	**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.
S6	**alphanum.**	Select an alphanumeric character after typing. The possible characters are displayed on the frontpanel key. Only possible at relevant parameters.
S7	**turn punch / turn die**	Turn around the applied tool (back to front). Only available if the cursor is placed on a tool parameter.
S8	**all bends**	Open a new window, in which all bends are shown in a table.
END	**END**	To end the data prep./edit mode and return to the main menu.

- **Parameter explanation**

Punch . . . . . .UP
The name (ID) of the applied punch.

Die . . . . . .UN
The name (ID) of the applied die.

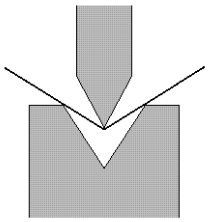
Program the desired tool numbers or press the key ⌂ to select a tool from the list. Use the function key ‘Show library’ to check the available tools.

Method . . . . . .BM

Select the required bending method. The control supports 4 methods:
Air bend
Bottoming
Flattening
Flattening & bottoming.

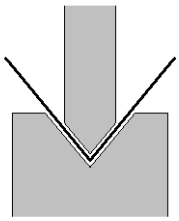
Press the key ⌂ to select the required setting.

Bend methods:



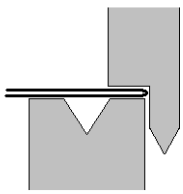
air bend

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.



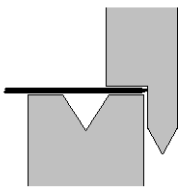
bottoming

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 depends on the working force. 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.

Bending length . . . . . .BL

Length of the sheet 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.

X-axis X

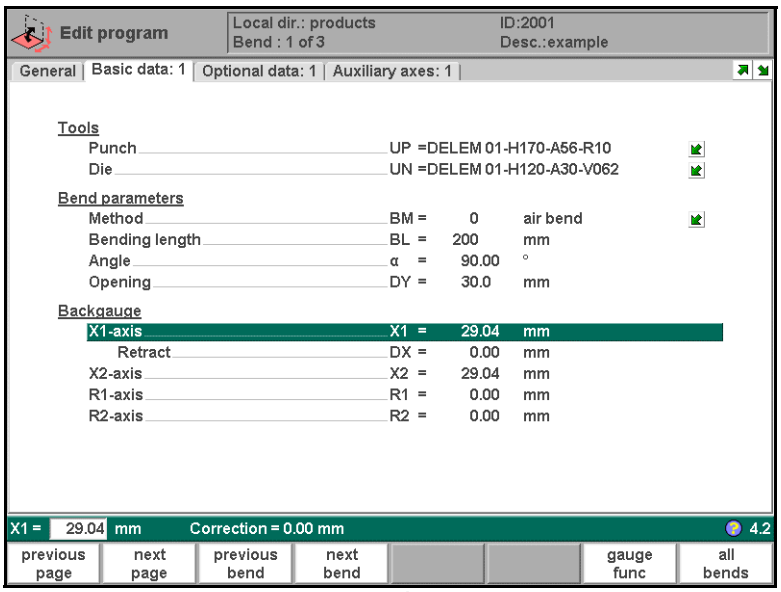
The position of the X-axis, also known as back gauge position.
When a negative value is programmed this back gauge value is interpreted as a relative value. This value is subtracted from the actual X-axis position. Therefore this parameter can also be used as a chaining measure.

Retract . . . . . .DX

Retract distance of back gauge during the bending. The "backgauge retract" is started when the beam is pinching the sheet.

- ## Gauge function

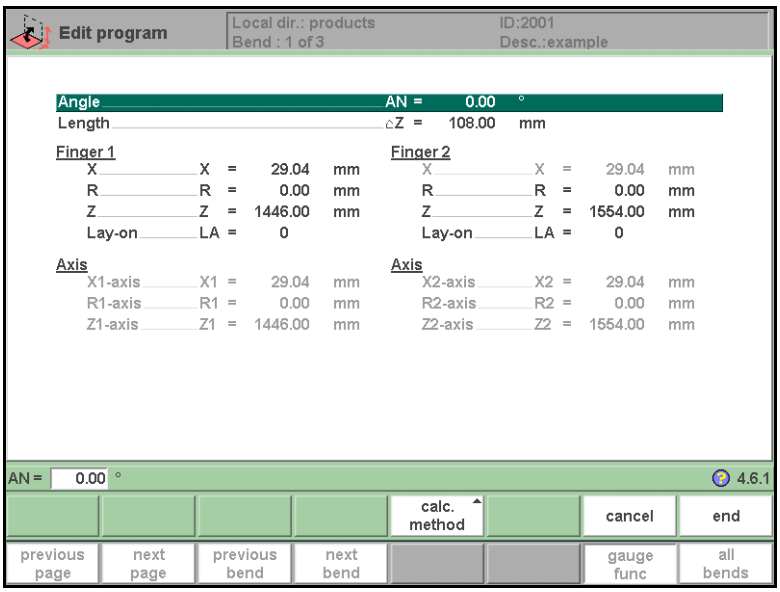
The function key ‘gauge func’ (S7) appears when the cursor bar is on an axis parameter.



4.d

If ‘gauge func’ is pressed, a window appears with several programmable parameters. These parameters serve to program the desired finger positions for a certain bend. The necessary

axis positions that are necessary for this bend are calculated from the programmed finger positions.



4.e

X X

The value in X-direction of the finger position for the sheet. This value can manually be adjusted in this window, if required.

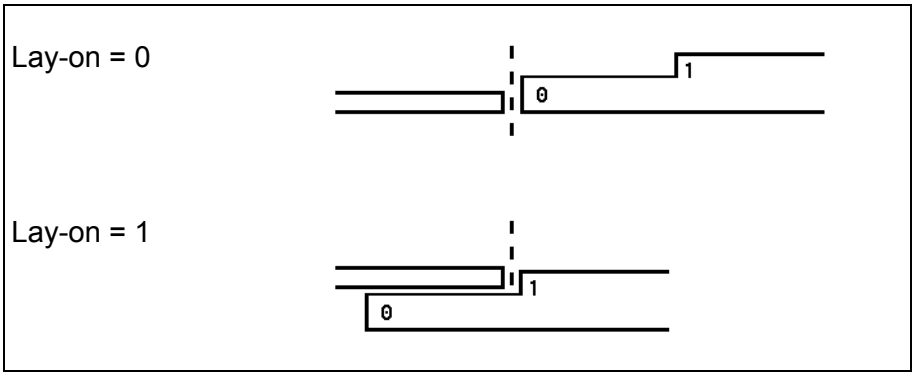
R . . . .R

The height (R-direction) of the gauge position of the sheet. This value can manually be adjusted in this window, if required.

Lay-on. . . . . .GP

With this parameter you can program another finger position for this specific bend. The finger position you can program depends on the back gauge possibilities. Back gauge dimensions are programmed in the menu ‘program constants’.
Lay-on = 0: no lay on (default)
Other possibilities are:
Lay-on = 1: lay on first level
Lay-on = 2: lay on second level
Lay-on = 3: lay on third level

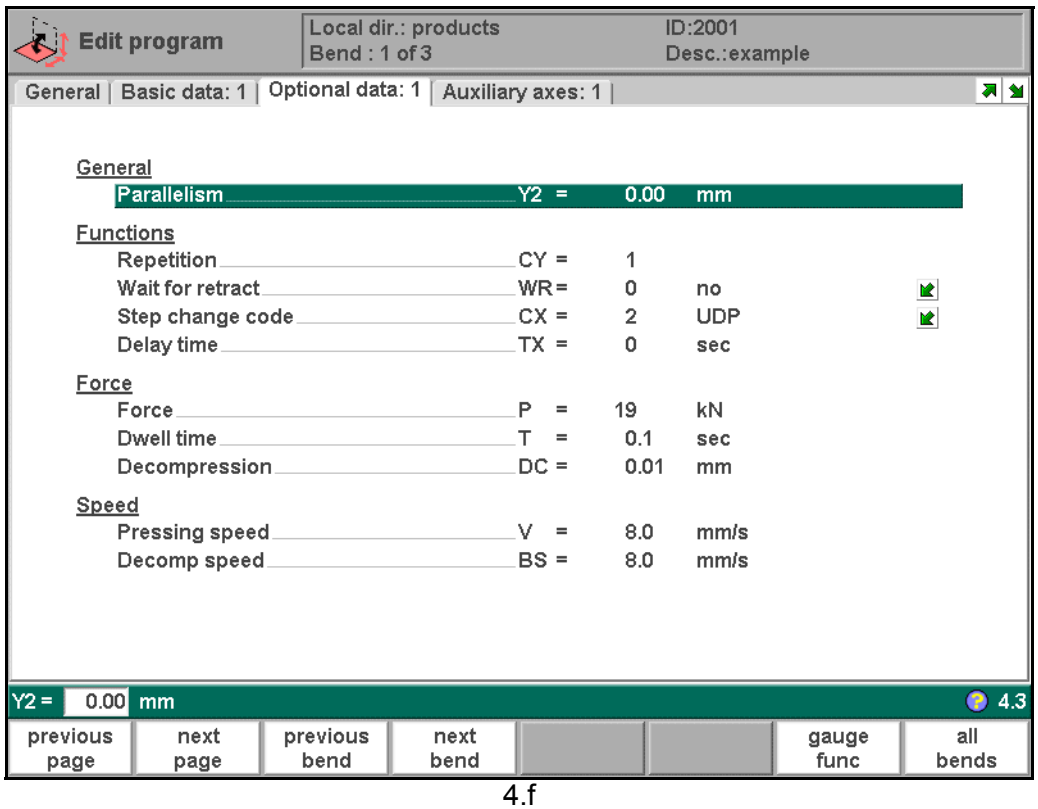
When you program another ‘Lay-on’ the axis positions change, but the gauge position for the sheet does not change. This means that the previous two parameters ‘R’ and ‘X’ remain the same. See also example below.



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 ‘Lay-on’ then the axes positions in X- and R-direction have been changed. The new values depend on the dimensions of the back gauge finger which you have programmed in the menu ‘program constants’. When you have changed the values of ‘R’ and ‘X’, then these changes are also included.

- **Bend parameters - optional data**

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



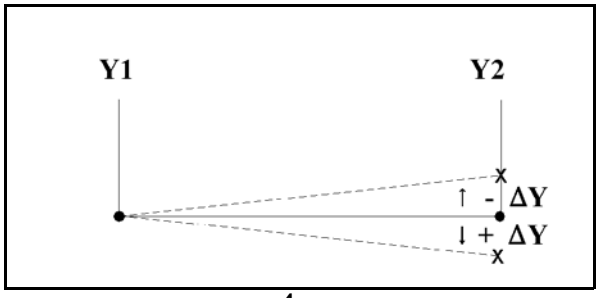
4.f

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 sheet.
By default, the mute value from the programmed die is used.
Whether or not this parameter is present depends on machine settings.

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.



4.g

Repetition . . . .CY

0 = bending is skipped

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

Wait for retract . . . .WR

In case of a retract, let the Y-axis wait until the retract is finished, yes or no.

0 = No: the retract is started when the Y-axis passes the clamping point, the Y-axis does not stop.

1 = Yes: when the Y-axis reaches the clamping point, the Y-axis is stopped and the retract is started. When the retract is completed, the Y-axis moves on.

Step change code CX

Programmable parameter which determines when the parameter values for the next bending will be active. The following settings are possible.

0 = ER: Bending number change (step change) at end of decompression (next bend parameters active).

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

2 = UDP: Step change at upper dead point.

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

4 = EXTERNAL: 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 back gauge performed. See also code 5.

5 = UDP EXTERNAL: 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 back gauge will be performed.

Delay time. . . . . ..TX

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

Force. . . . . ..P

Max. adjusted force during pressing (automatically computed).

Dwell time. . . . . ..T

Holding time of punch at bending point.

Decompression . . . . . .DC

Decompression stroke after bending to release the working pressure

Pressing speed V

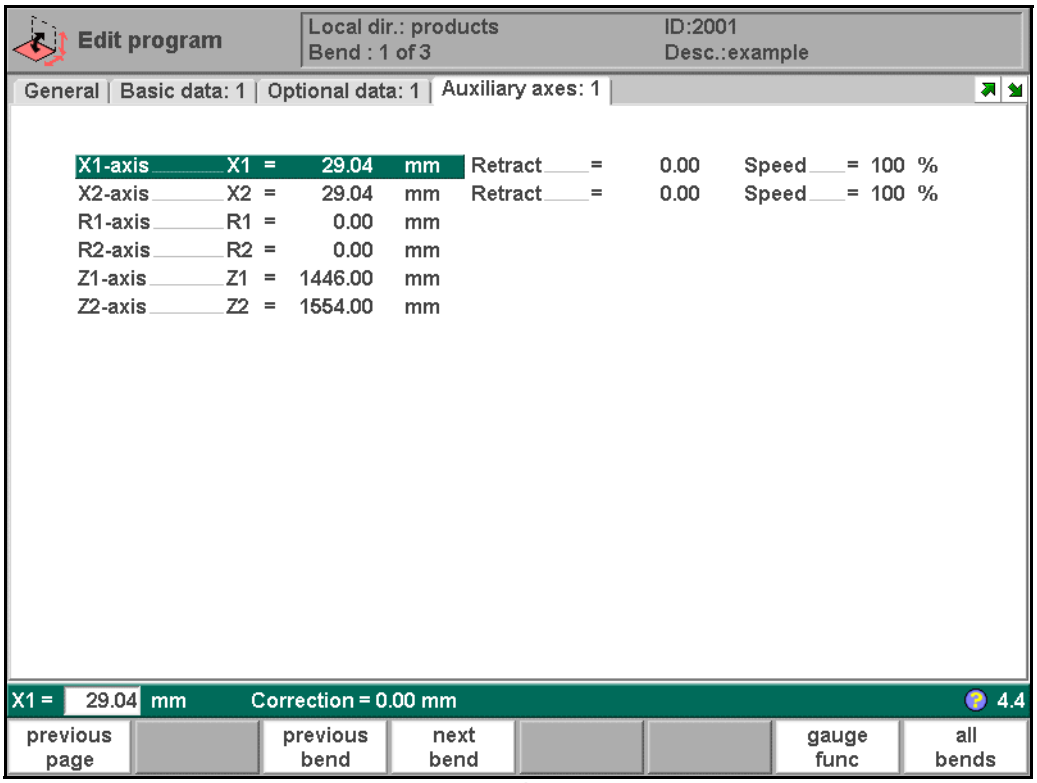
Working speed (pressing speed). Initially, the value for this parameter is copied from the parameter ‘pressing speed’ in the programming constants menu.

Decomp speed . . . . . .BS

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

- **Bend parameters - auxiliary axes**

On this page, all programmable axes are shown. This page serves to have all axes positions combined on one page in case they need to be changed. The axes shown here depend on the machine configuration.



4.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.

Axis speeds

Speed of the selected axis in the current bend. Speed can be programmed in a percentage of the maximum possible speed.

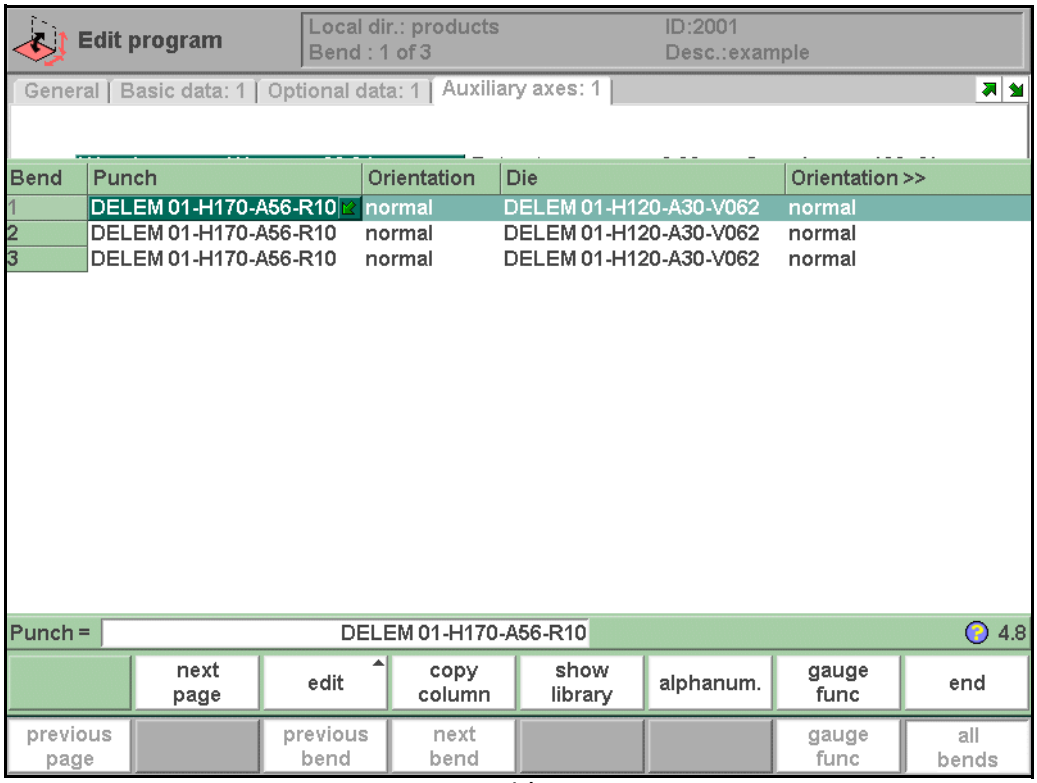
Retract

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

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

- **Bend parameters - all bends**

When the function key ‘all bends’ has been pressed, a complete overview of the bends appears. After pushing END the page from which this page was selected will be restored, with the cursor on the parameter selected before.



4.i

A specific bend can be selected on the screen by putting the highlighted bar on that bend, then pressing END.
From within this screen, the complete CNC program can be edited. All bend parameters can be edited within the table and bends can be swapped, moved, added and deleted.

Function keys:

S1	**previous page**	To go to a **previous page** when available.
S2	**next page**	To go to a **next page** when available.

Function keys:

S3	**edit**	Edit the program with one of the following commands: - insert bend - delete bend - mark bend
S4	**copy column**	Copy the value of the currently selected parameter in the current bend to all other bends.
S5	**show library**	Open a window with an overview of the tool library.
S6	**alphanum.**	Enter alphanumeric characters. See also section 1.7.5.
S7	**gauge func**	Program the gauge positions, as already explained earlier in this chapter.
S8	**end**	Return to the bend parameters.
END	**END**	Return to the bend parameters.

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

Function keys:

S3	**insert bend**	To insert a new bend between one of the bends. When pressed, the current bend is copied and added after the current bend.
S4	**delete bend**	To delete the bend that is currently selected.
S5	**mark bend**	Mark the current bend, in order to prepare it for another action, like move or swap. See description below.

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

Function keys:

S3	**move bend**	Move a bend to another place in the program. See also description below.
S4	**swap bends**	Let two bends swap places. See also description below.
S5	**abort mark**	Remove the mark from the currently marked bend.

Move bend
In the table overview of the bend sequence, 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 number 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 function key ‘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.

4.3. Special edit function

When a new program has been made and several tests are required, the operator can switch from "edit"-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.

4.4. Remarks

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

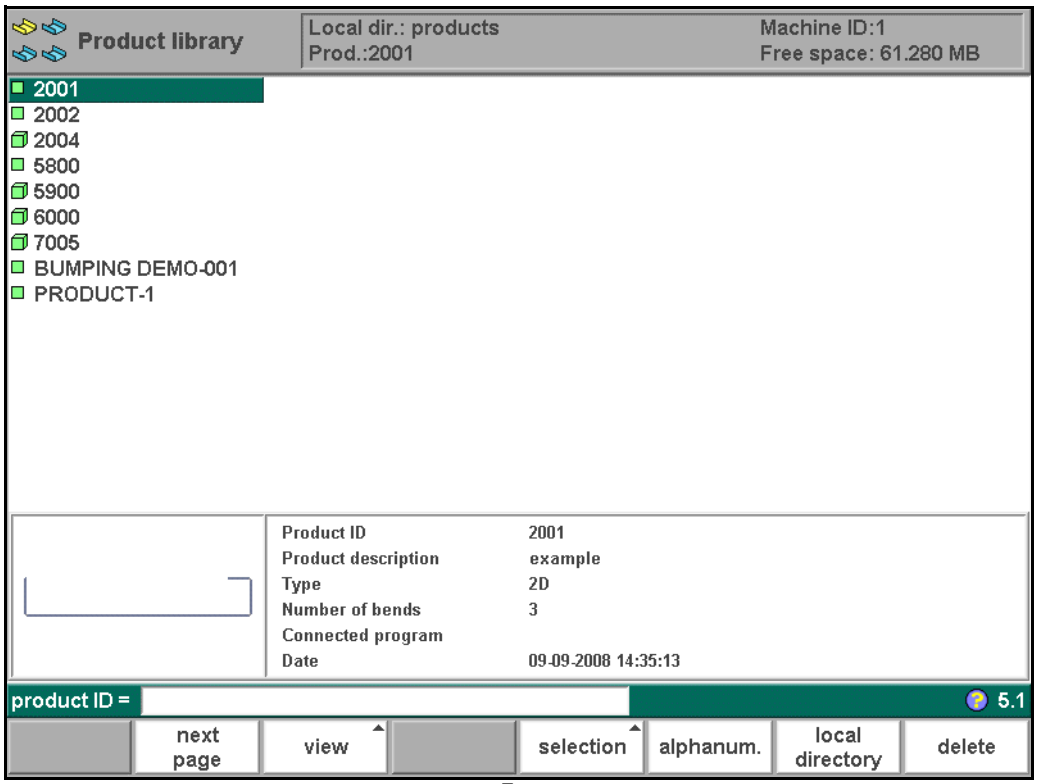
Parameters 1 through 4 are only automatically recalculated if the parameter **Auto computations edit** (see the menu ‘program constants’) has been activated (value = 1). Parameter 5 is only automatically recalculated if the parameter Bend allowance (see the menu ‘program constants’) has been activated (value =1). Corrections on the X-axis position can be edited with the parameter **corr.X** (per bend) and **G-corr.X** (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 bend method is changed the “force” and “Decompression” will be adjusted automatically.

5. Product library

5.1. Introduction

5.1.1. The main view

In this chapter the menu **product library** is described.



5.a

The "product library" menu gives an overview of the program library on the control. In this menu a product program can be selected (loaded). A program can be modified or executed. Each item in the list consists of a symbol and an ID. The ID is simply the product ID, 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
(red)	the product consists of a 3D drawing, there is no CNC program
(green)	the product has a 3D drawing and a CNC program

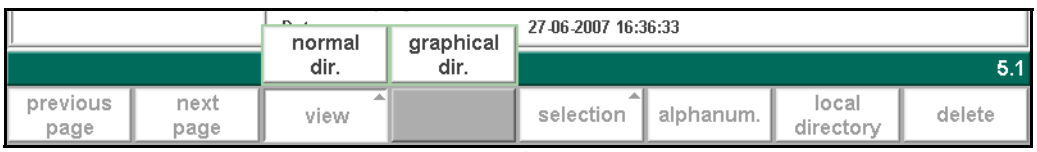
If a product program is already active its ID is shown in the top of the screen. A program can

be loaded by scrolling to the correct ID and pressing enter or by typing the product ID in the enter field.

Functions keys:

S1	**previous page**	Previous page with products overview
S2	**next page**	Next page with products overview
S3	**view**	Select one of the possible viewing modes: - normal dir. - graphical dir.
S5	**selection**	Function key to open a new button bar with extra function keys for product selection functions: - product ID - product description - type - date
S7	**local directory**	Open an overview of the manually created product directory structure.
S8	**delete**	Erase the selected program.
END		End the product selection.

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



5.b

Function keys:

S3	**normal dir.**	Select a normal view of the product collection. This is the standard setting.
S4	**graphical dir.**	Select a graphical overview of the product collection.

The other function keys cannot be selected here, until one of these three function keys has been chosen. The temporary bar will disappear again.

5.1.2. Selection

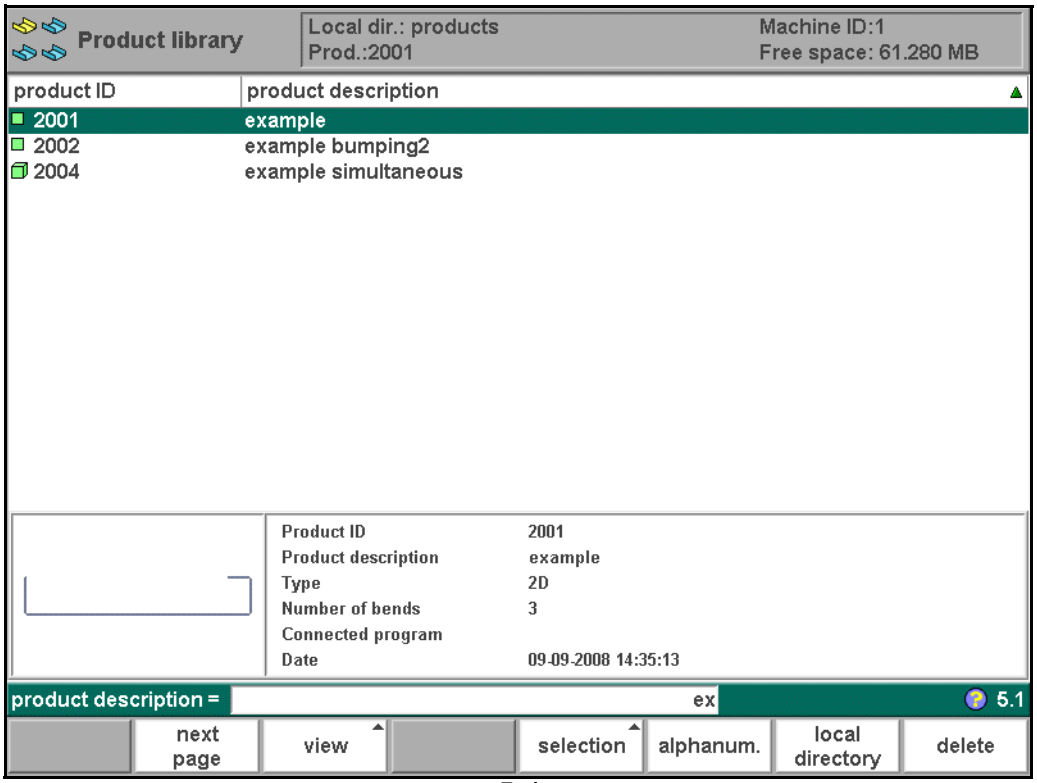
When the function key ‘selection’ has been pressed a new, temporary button bar appears with additional function keys:

5.c

Function keys:

S5	**product ID**	Products are shown in columns with symbols and product ID, they are sorted on product ID. This is the default view.
S6	**product description**	Products are shown in columns with product ID and product description, they are sorted in the alphabetical order of product description.
S7	**type**	Products are shown in columns with product ID and product type, they are sorted on product type.
S8	**date**	Products are shown in columns with product ID and date, they are sorted on date.

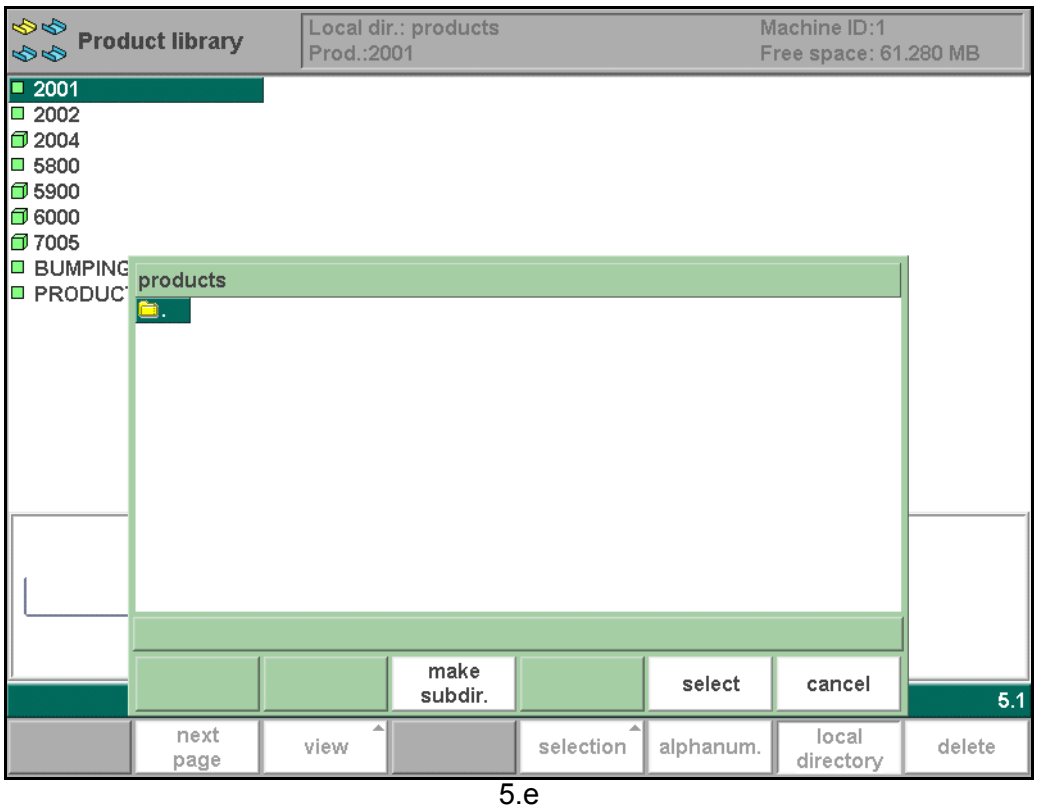
You can search for products with a certain number or name. At the bottom of the screen you may enter a string the control will search for. You can either enter a complete name or number or only a part of it.
If you enter part of a name and this part occurs in several product names, the control will show all product names that contain that part. It is also possible to enter a combination of name and number. See also section 1.7.3 about ‘live search’.


Product library
Local dir.: products
Prod.:2001
Machine ID:1
Free space: 61.280 MB
product ID
product description
2001
example
2002
example bumping2
2004
example simultaneous
Product ID
2001
Product description
example
Type
2D
Number of bends
3
Connected program
Date
09-09-2008 14:35:13
product description =
ex
5.1
next page
view
selection
alphanum.
local directory
delete

5.d

5.2. Directories

Bend programs on the control can be stored in different directories. 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.



5.e

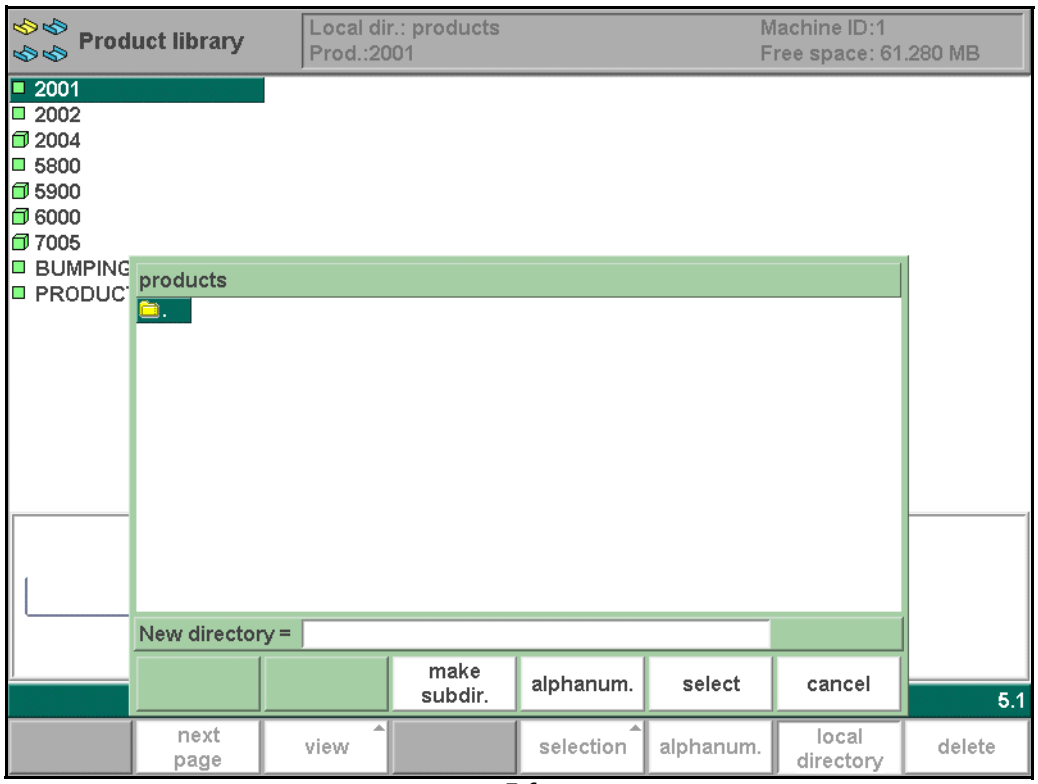
Use the arrow keys to move to a directory. Depending on the cursor position, several functions are available.

Function keys:

S4	**make subdir**	Add a new subdirectory. The name of the subdirectory can be any alphanumeric string of maximal 24 characters. (Avoid the use of slashes ‘/’).
S5	**remove subdir**	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” cannot be deleted.
S6	**select**	To select the directory on which the cursor is placed.

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’. The name of the subdirectory cannot be changed.



5.f

In this menu it is not possible to copy products from one subdirectory to another subdirectory, this can be done in the menu ‘product edit’ or ‘data edit’.
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.

6. Programming of tools

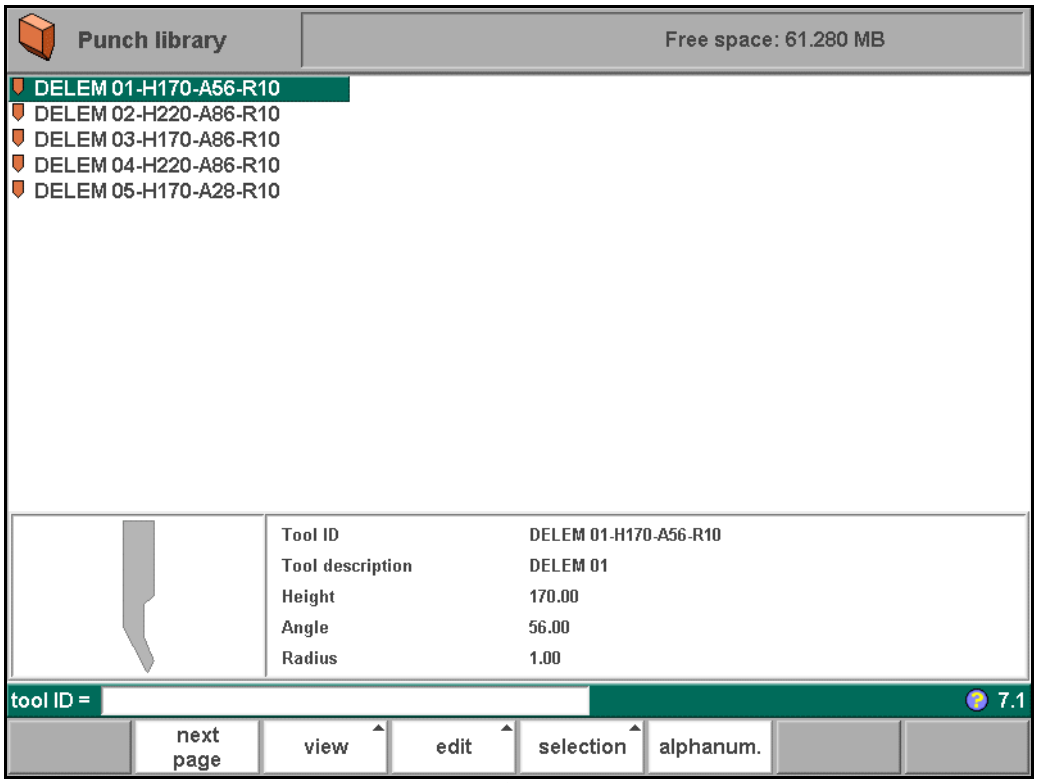
6.1. Introduction

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

6.2. Programming of Punches

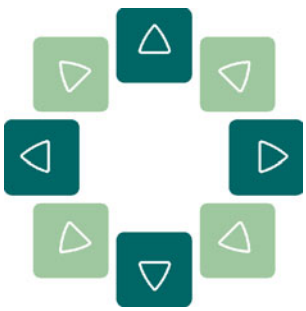
6.2.1. The punch menu

The programming of punches is started by choosing **Punch library** in the main menu. The punch profile can be created with help of the programming facilities of the control.



6.a

A listing of punches programmed into the memory is given.

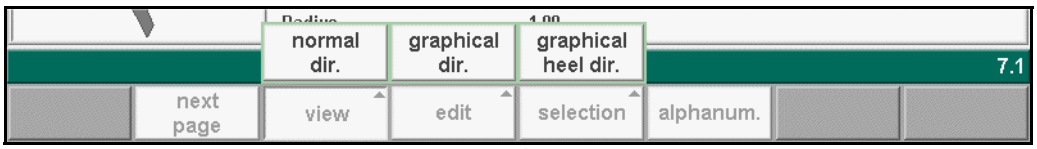


Use the arrow keys to move the cursor bar to the desired tool. Then press the ENTER key to edit this tool or press the function key ‘edit’ to choose another function.

Function keys:

S1	**previous page**	To select a previous page within the several pages of the punch library.
S2	**next page**	To select the next page within the several pages of the punch library
S3	**view**	Select one of the possible viewing modes: - normal dir. - graphical dir. - graphical heel dir.
S4	**edit**	Edit the tool library with one of the following commands: - edit - add - copy - delete.
S5	**selection**	Button to select one of the possible viewing modes: - tool ID - tool description
S6	**alphanum.**	Select an alphanumeric character after typing. The possible characters are displayed on the front panel key. Only possible at relevant parameters.

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



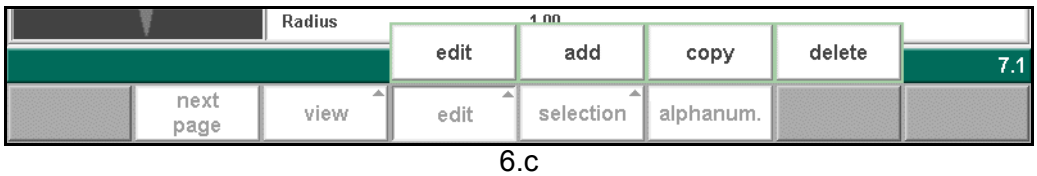
6.b

Function keys:

S3	**normal dir.**	Select the normal view of the tool collection. This is the standard setting.
S4	**graphical dir.**	To get a graphical overview of the tools as available in the library including main dimensions of the tool.

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 ‘edit’ has been pressed a new, temporary button bar appears with additional function keys:

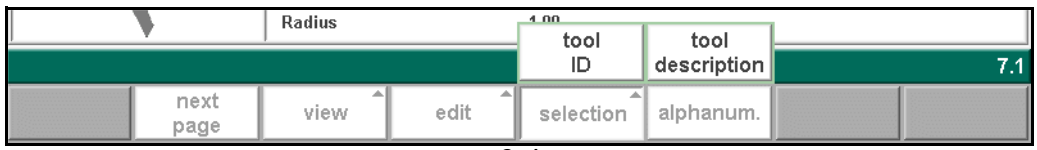


6.c

Function keys:

S4	**edit**	Modify the currently selected tool.
S5	**add**	Create a new tool and add it to the tool collection.
S7	**delete**	Delete the currently selected tool.

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 ‘selection’ has been pressed a new, temporary button bar appears with additional function keys:



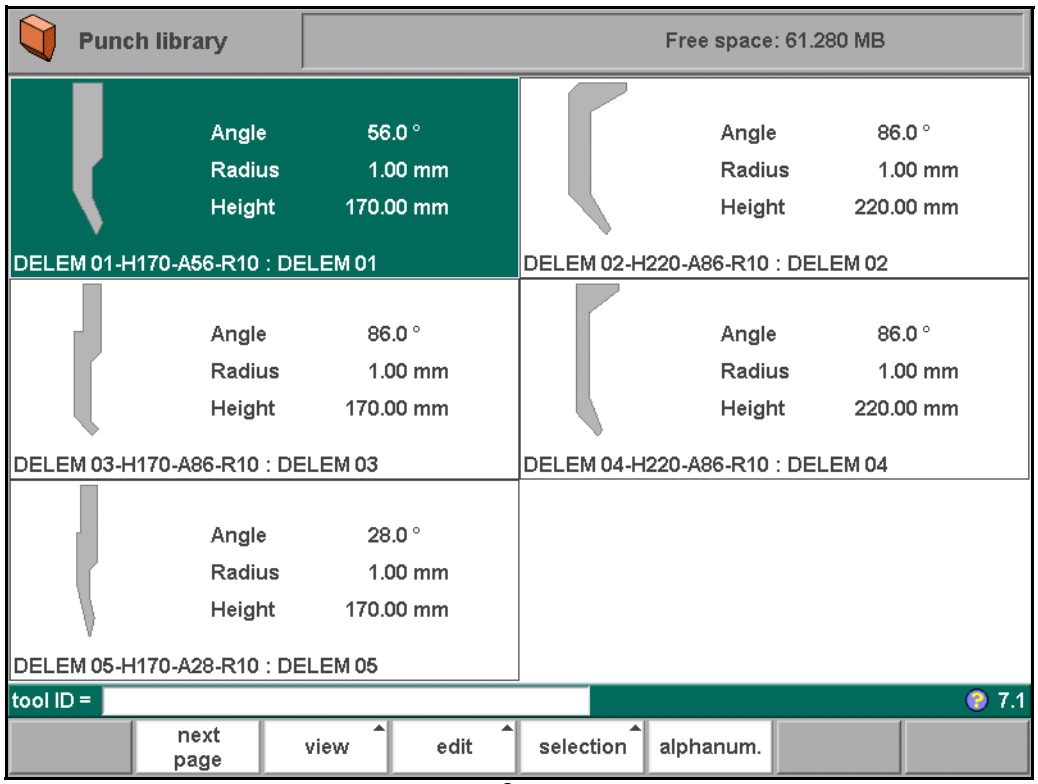
6.d

Function keys:

S5	**tool ID**	Tools are shown in columns with symbols and tool ID, they are sorted on tool ID. This is the default view.
S6	**tool description**	Tools are shown in columns with tool ID and tool description, they are sorted on the alphabetical order of tool description.

- **Graphical directory**

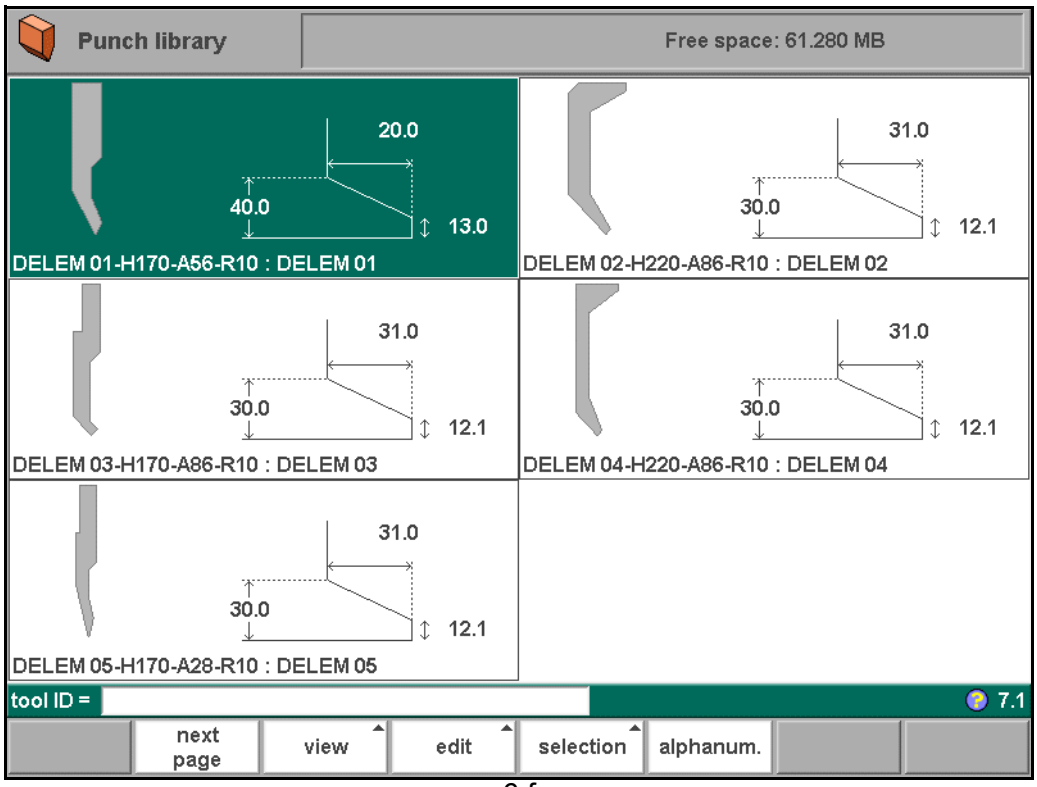
After the function key ‘graphical dir.’ has been pressed, a graphic overview of the tool library is shown. Each tool is shown with the main properties.



6.e

- **Graphical directory punches with heels**

After the function key ‘graphical heel dir.’ has been pressed, a graphic overview of the tool library is shown. Each tool is shown with the programmed heel properties.

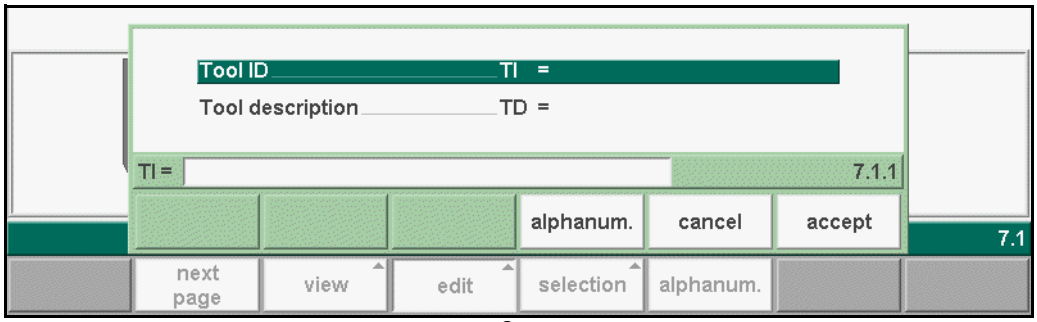


6.f

6.2.2. Punch drawing

- **Create a new punch**

To create a new tool, press the key ‘edit’ in the punch library and subsequently the key ‘add’. The control will start by asking for a tool ID and a description.



6.g

Tool ID. . . . . ..TI
A unique name or number to identify the tool. The maximum length is 25 characters.

Tool description. . . . . ..TD
A name or description of this tool. The maximum length is 25 characters.

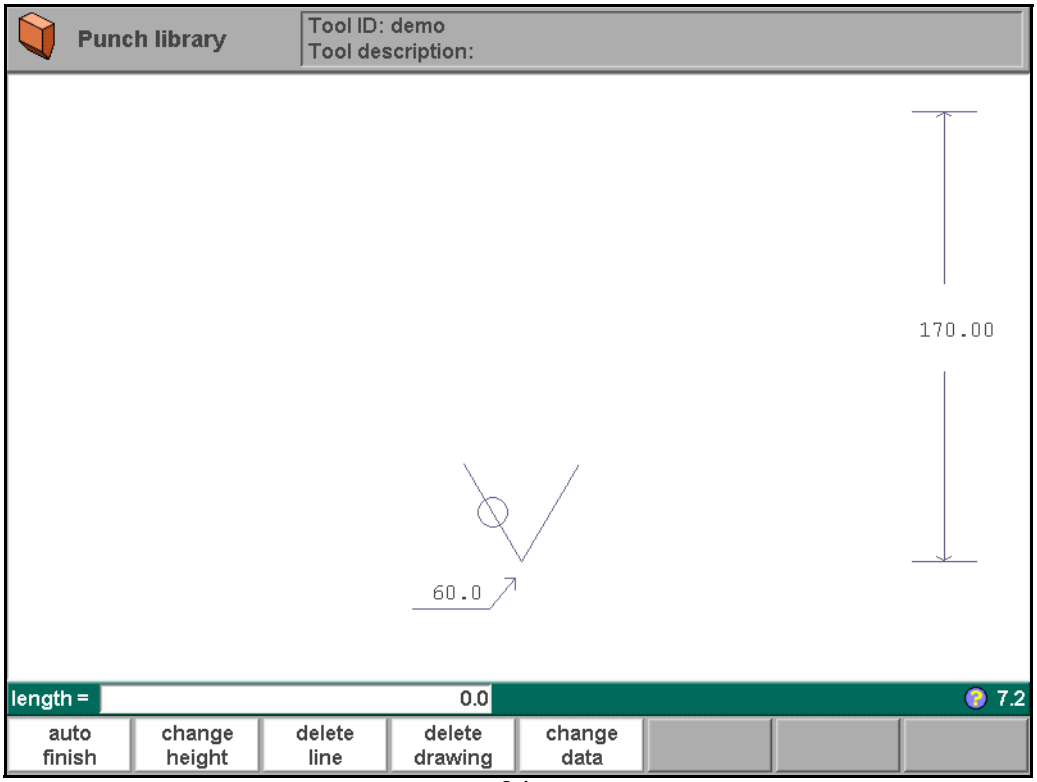
These parameters may also contain alphanumeric characters, which can be entered with the help of the function key ‘alphanum’ (S5).
When finished, press ‘accept’ to leave this window and start with the tool drawing.

The control will start by prompting the initial dimensions of the tool.

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.

After entering these typical values you can create the tool drawing with the drawing facilities.



6.h

Drawing a tool profile is done by entering angle values and line length values. These values are prompted in the lower left corner of the screen.

Function keys:

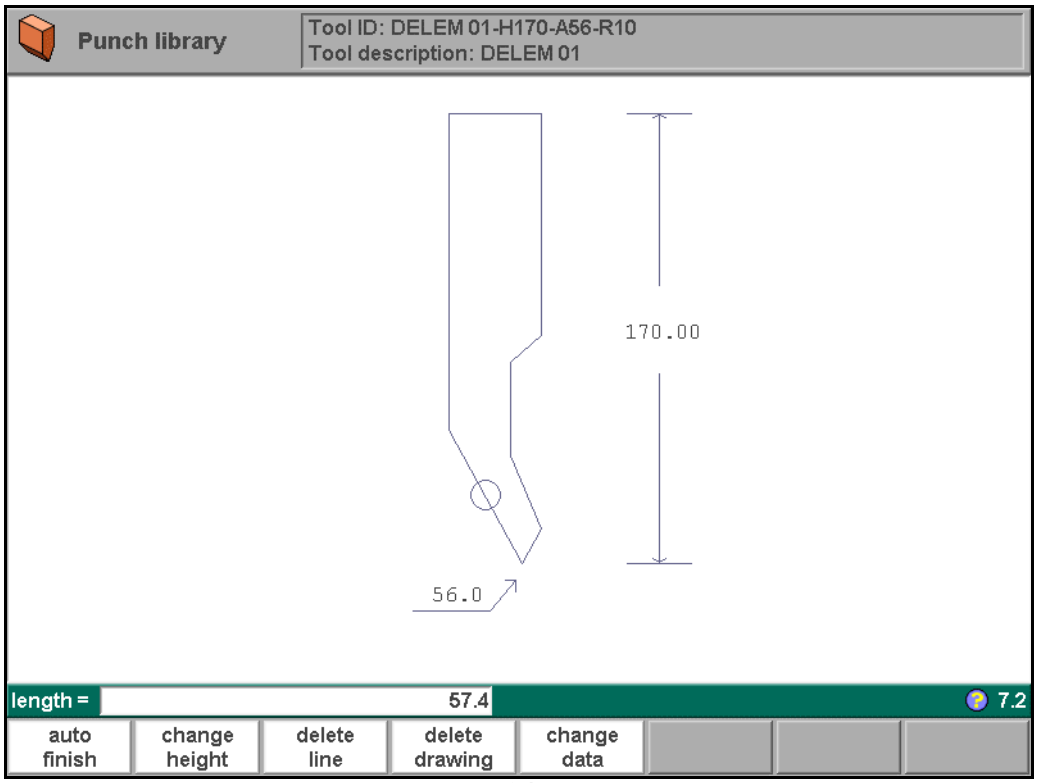
S1	**auto finish**	Finishing the tool outline to the top of the tool automatically.
S2	**change height**	To change the height dimension of the tool.
S3	**delete line**	To delete a line segment.

Function keys:

S4	**delete drawing**	To delete the complete tool from the library.
S5	**change data**	To change specific tool data and description.
END		To return to the directory overview.

- **Edit punch drawing**

To edit an existing tool, press the key ‘edit’ in the punch library and subsequently the key ‘edit’. The tool appears on the screen and can be edited with the drawing facilities.

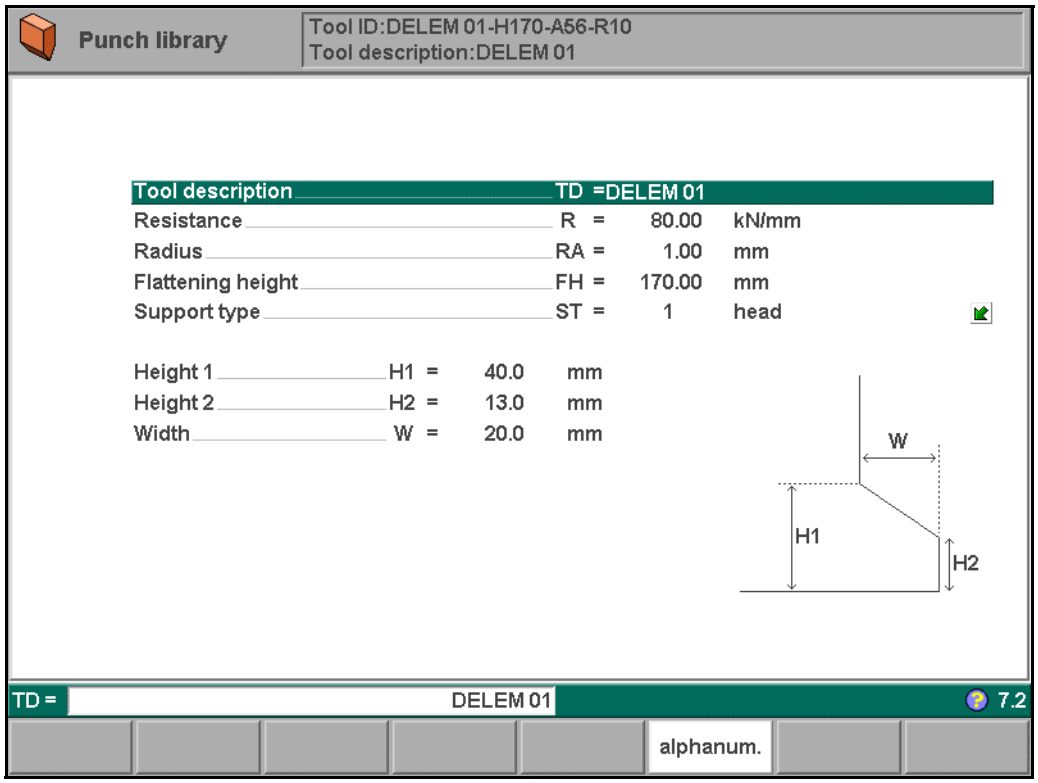


6.i

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

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

6.2.3. Specific Punch Data



6.j

Tool description . . . . . .TD
A name or description of this tool. The maximum length is 25 characters.

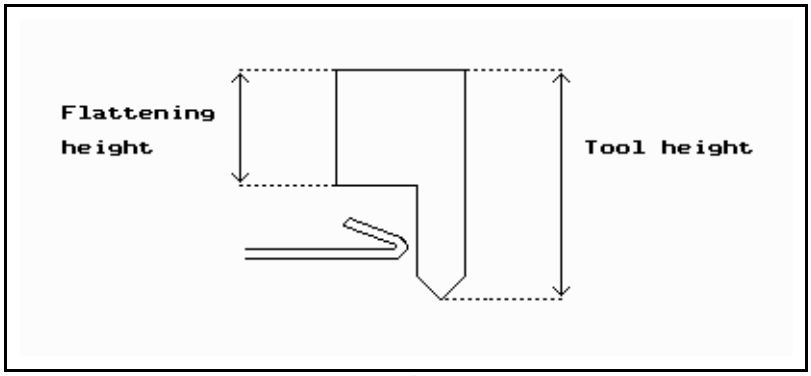
Resistance . . . . . .R
Maximum allowable force on punch in kN/mm.

Radius . . . . . .RA
The radius of the punch.

Flattening height . . . . . .FH
The flattening height needs 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 (for safety reasons). See figure 6.j.

The total tool 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 twice the 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 twice the sheet thickness.

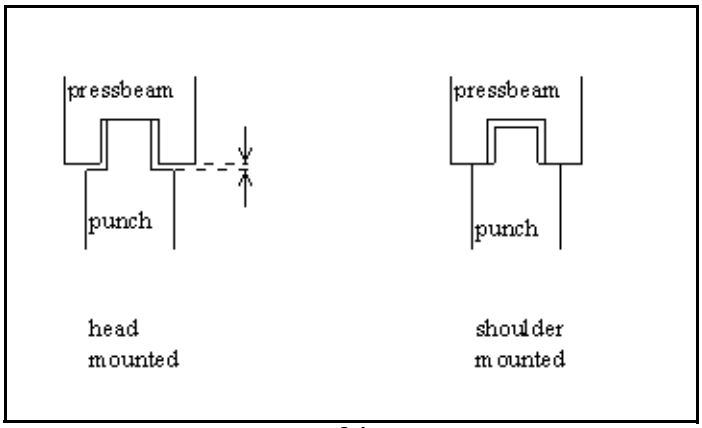


6.k

Support type . . . . . .ST

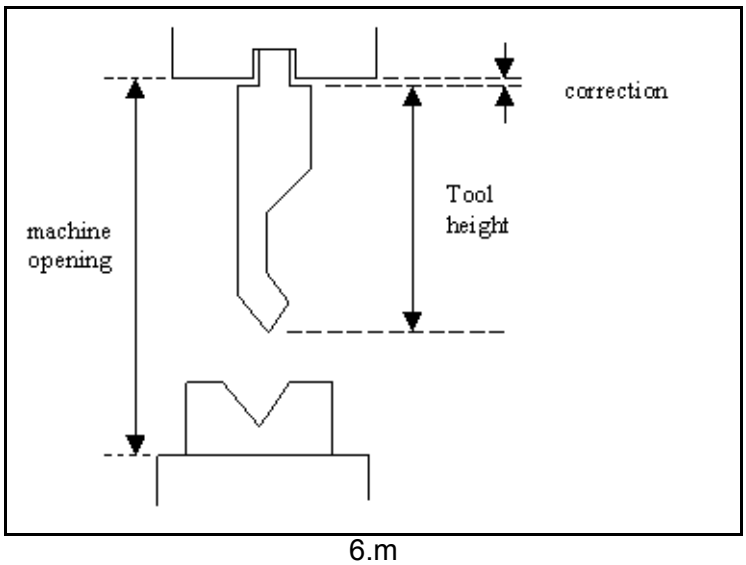
Switch parameter, to account for differently mounted punches. Such punches could cause inaccuracies in the tool height and the resulting Y-axis position. The control makes a distinction between two settings, 'head mounted' and 'shoulder mounted'.
0 = shoulder mounted (default setting)
1 = head mounted

Press the key ⌂ to select the required setting.



6.l

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.



6.m

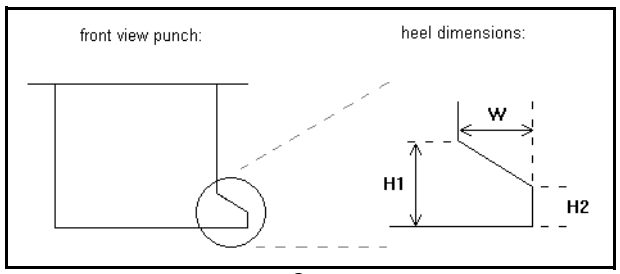
Heel dimensions:

Height 1 . . . . . .H1
Height1 of the heel.

Height 2 . . . . . .H2
Height2 of the heel.

Width . . . . . .W
Width of the heel.

Height1 and height2 do not influence the tool height you have programmed before.

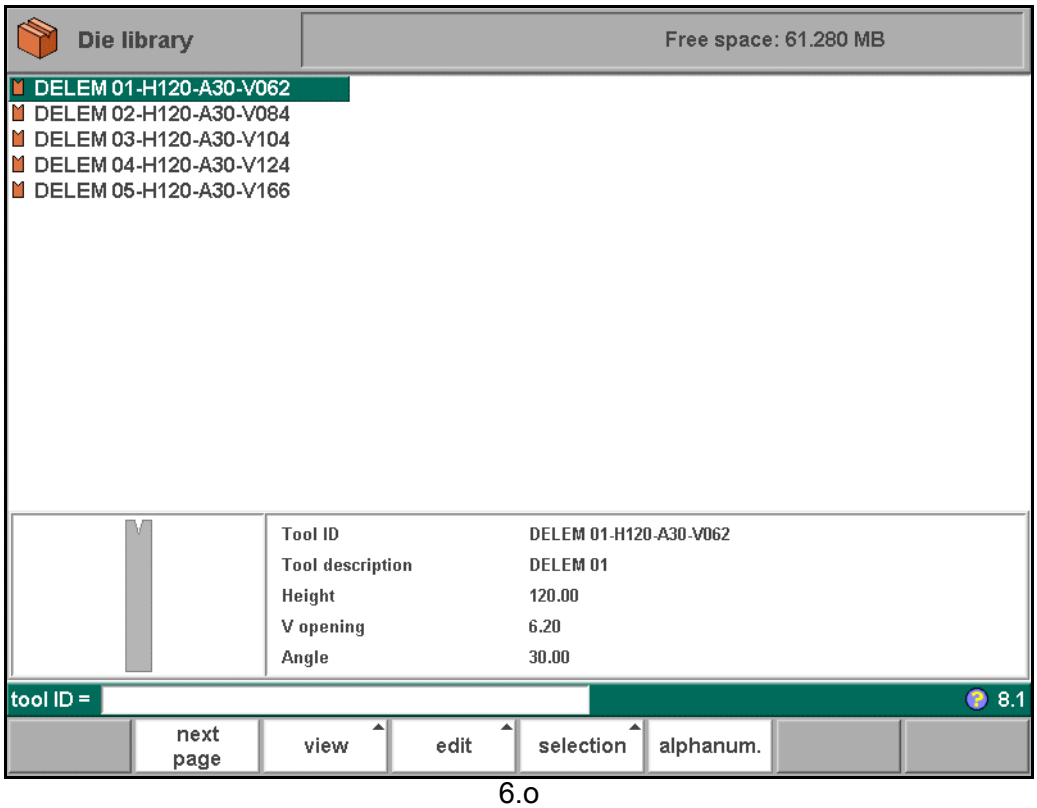


6.n

6.3. Programming of bottom dies

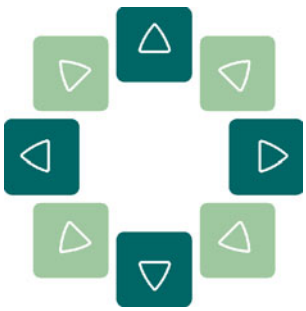
6.3.1. The die menu

The programming of dies is started by choosing **Die library** in the main menu.



6.o

A list of the programmed dies is given.



Use the arrow keys to move the cursor bar to the desired tool. Then press the ENTER key to edit this tool or press the function key ‘edit’ to choose another function.

Function keys:

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

Function keys:

S2	**next page**	To select the next page within the several pages of the die library.
S3	**view**	Button to select one of the possible viewing modes: - normal dir. - graphical dir.
S4	**edit**	Edit the tool library with one of the following commands: - edit - add - copy - delete.
S5	**selection**	Button to select one of the possible viewing modes: - tool ID - tool description
END		To return to the program menu.

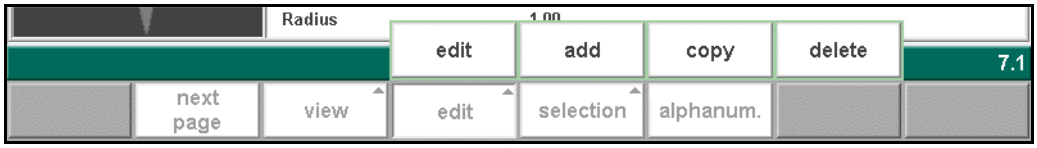
When the function key ‘View’ has been pressed a new, temporary button bar appears with additional function keys:

6.p

Function keys:

S3	**normal dir.**	Select the normal view of the tool collection. This is the standard setting.
S4	**graphical dir**	To get a graphical overview of the tools as available in the library including main dimensions of the tool.

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 ‘edit’ has been pressed a new, temporary button bar appears with additional function keys:

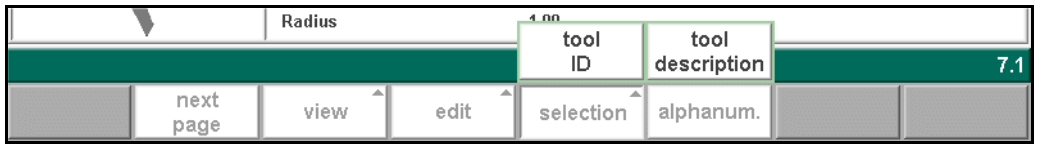


6.q

Function keys:

S4	**edit**	Modify the currently selected tool.
S5	**add**	Create a new tool and add it to the tool collection.
S7	**delete**	Delete the currently selected tool.

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

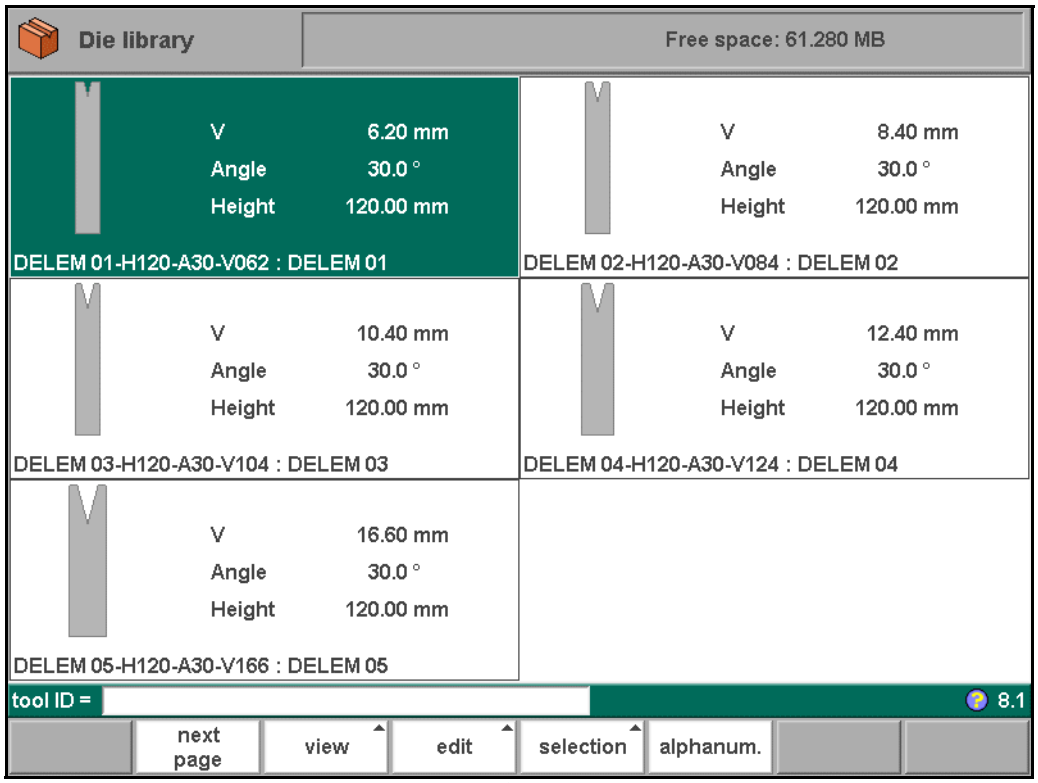


6.r

Function keys:

S5	**tool ID**	Tools are shown in columns with symbols and tool ID, they are sorted on tool ID. This is the default view.
S6	**tool description**	Tools are shown in columns with tool ID and tool description, they are sorted on the alphabetical order of tool description.

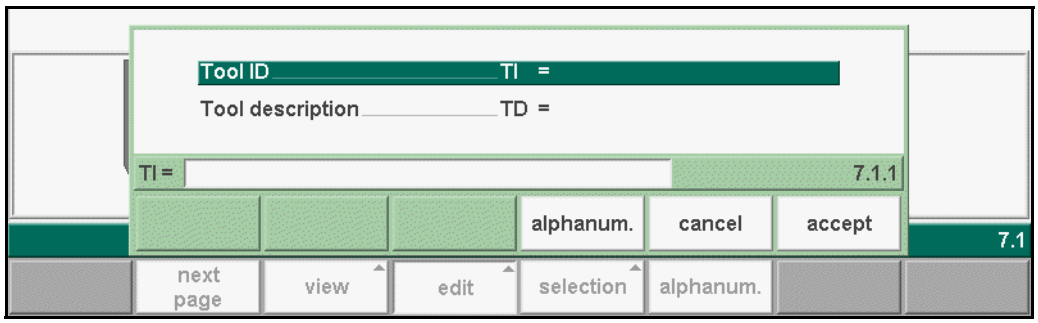
- **Graphical directory dies**



6.s

- **Create a new die**

To create a new tool, press the key ‘edit’ in the die library and subsequently the key ‘add’. The control will start by asking for a tool ID and a description.



6.t

Tool ID . . . . . .TI

A unique name or number to identify the tool. The maximum length is 25 characters.

Tool description . . . . . .TD

A name or description of this tool. The maximum length is 25 characters.

These parameters may also contain alphanumeric characters, which can be entered with the help of the function key ‘alphanum’ (S5).
When finished, press ‘accept’ to leave this window and start with the tool drawing.
The control will start by prompting the initial dimensions of the tool.

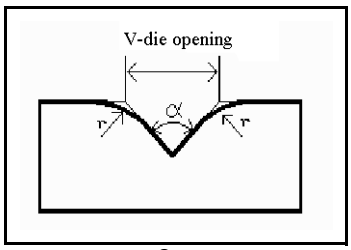
Height . . . .H
The height of the tool. Important: this height value will be used in bend depth calculation.

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

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

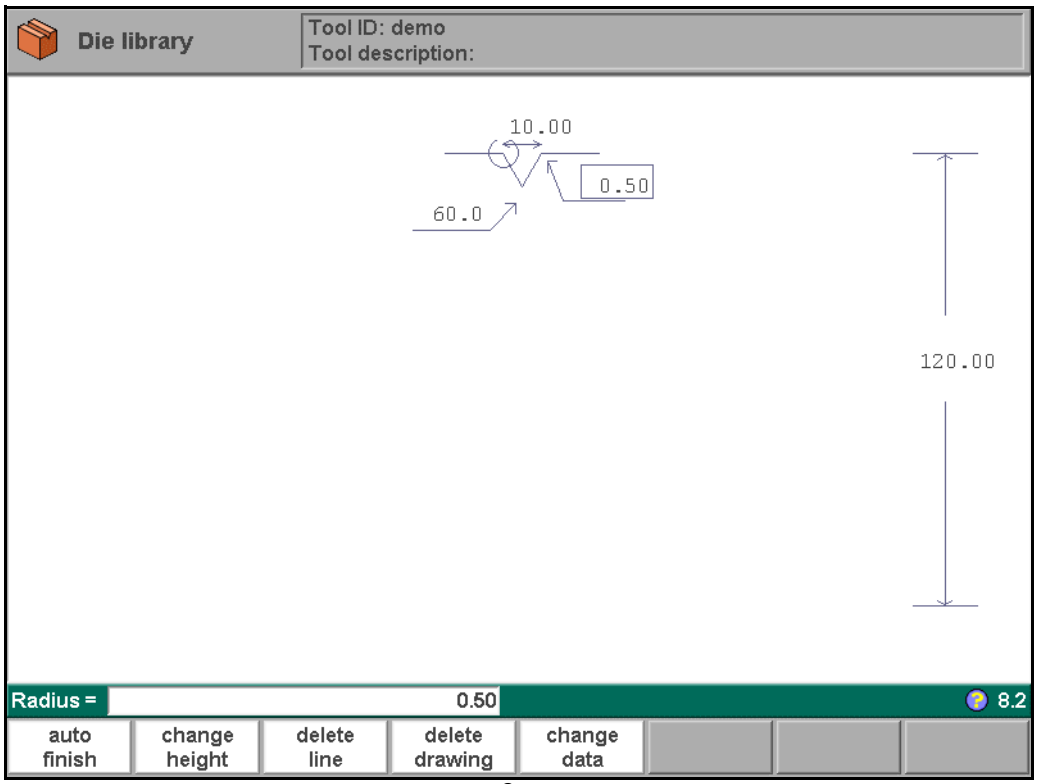
Radius. . . . . ..RA
The radius of the edges of the V-opening.

V-die opening:



6.u

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 back gauge position. The mid-position of the V-opening will be placed on the centre line of the press brake shape.



6.v

- **Create a new die with U-shape**

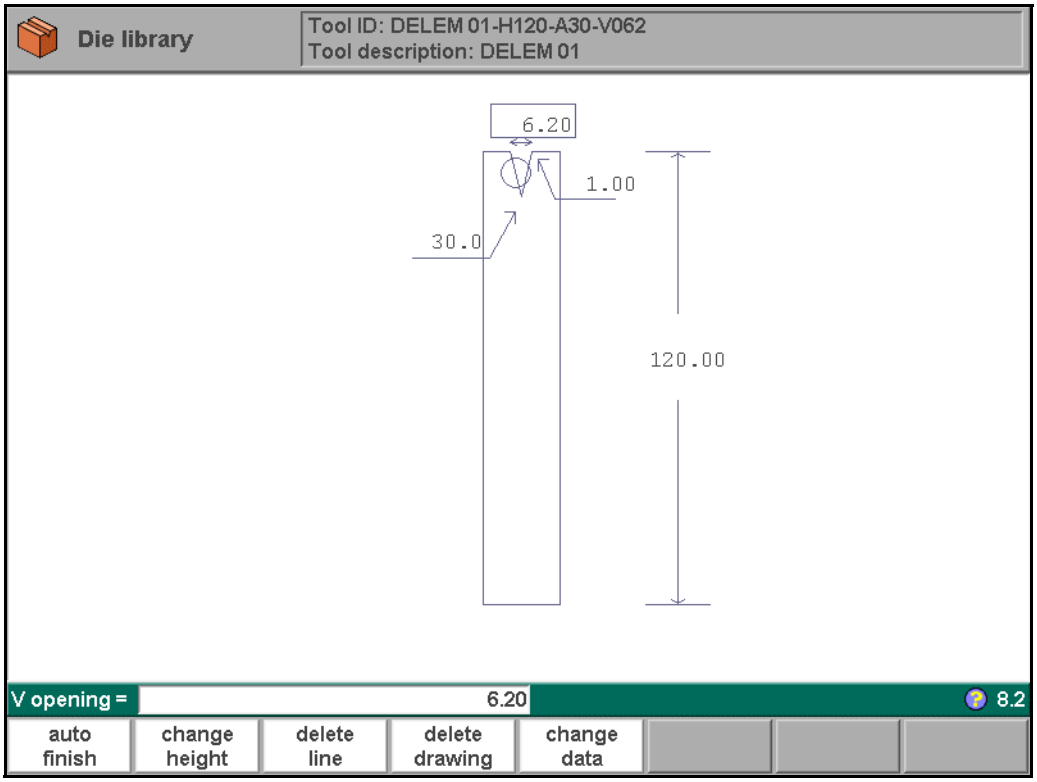
To create a die with a U shape, follow the same procedure as described for creating a V-shaped die. Except program 180 degrees for the V-angle. When entered, the control prompts for the specific U-height:

U-height . . . . . .U
The height of the U-opening in case of a U-shaped die.

The other properties of the die are the same as for a V-shaped die.

- **Edit die drawing**

To edit an existing tool, press the key ‘edit’ in the punch library and subsequently the key ‘edit’. The tool appears on the screen and can be edited with the drawing facilities.



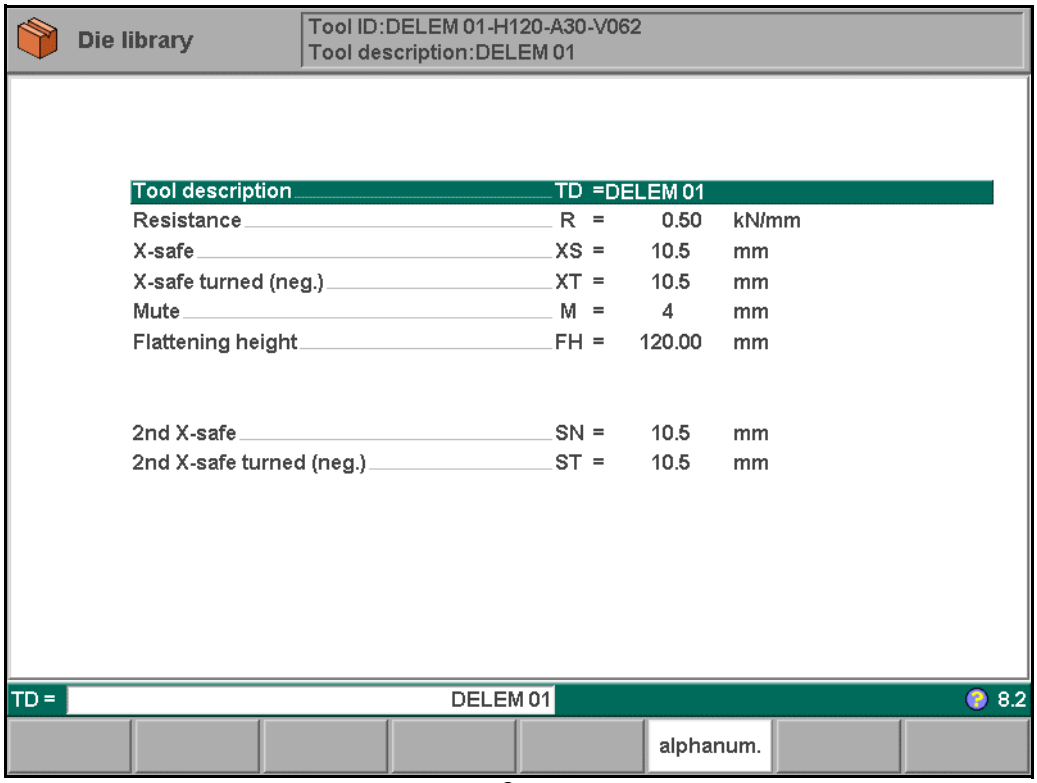
6.w

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 function keys are exactly the same as for punch programming.

6.3.2. Specific die data



6.x

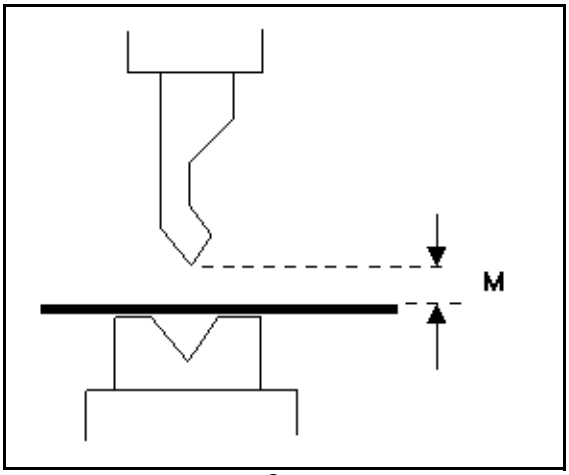
Tool description . . . .TD
A name or description of this tool. The maximum length is 25 characters.

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 back side of the V-grove
V = opening value
In this formula also a small additional safety value (0.5 mm) has been added.

X-safe turned (negative) . . . .ST
This safety value will be active if the die is used in the turned position. In this situation parameter ST will be used as safety zone for the backgauge.

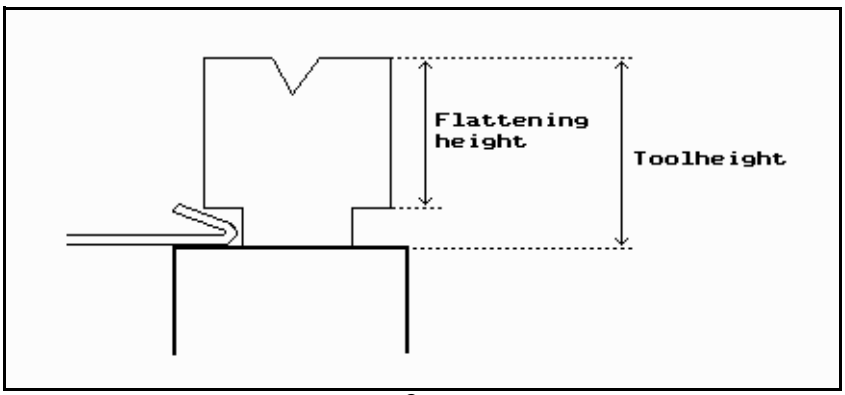
Mute . . . .M
Muting distance. Distance above the sheet at which the speed change takes place.



6.y

Flattening height . . . . . .FH

The flattening height needs 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 (for safety reasons). See figure 6.y.



6.z

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 twice the 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 twice the sheet thickness.

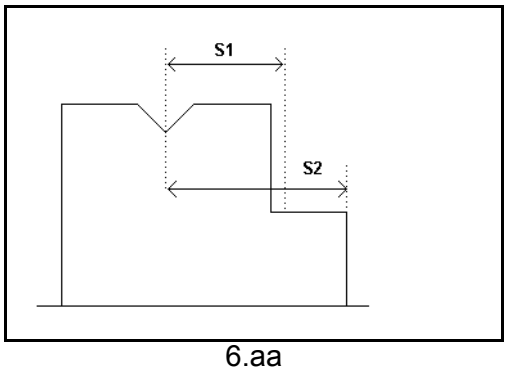
2nd x-safe . . . . . .SN=

Second automatically computed safety zone (minimum X-axis value) to be used in case a more complex die is used. 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 back side). Also an extra safety tolerance of 0.5 mm is added.

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

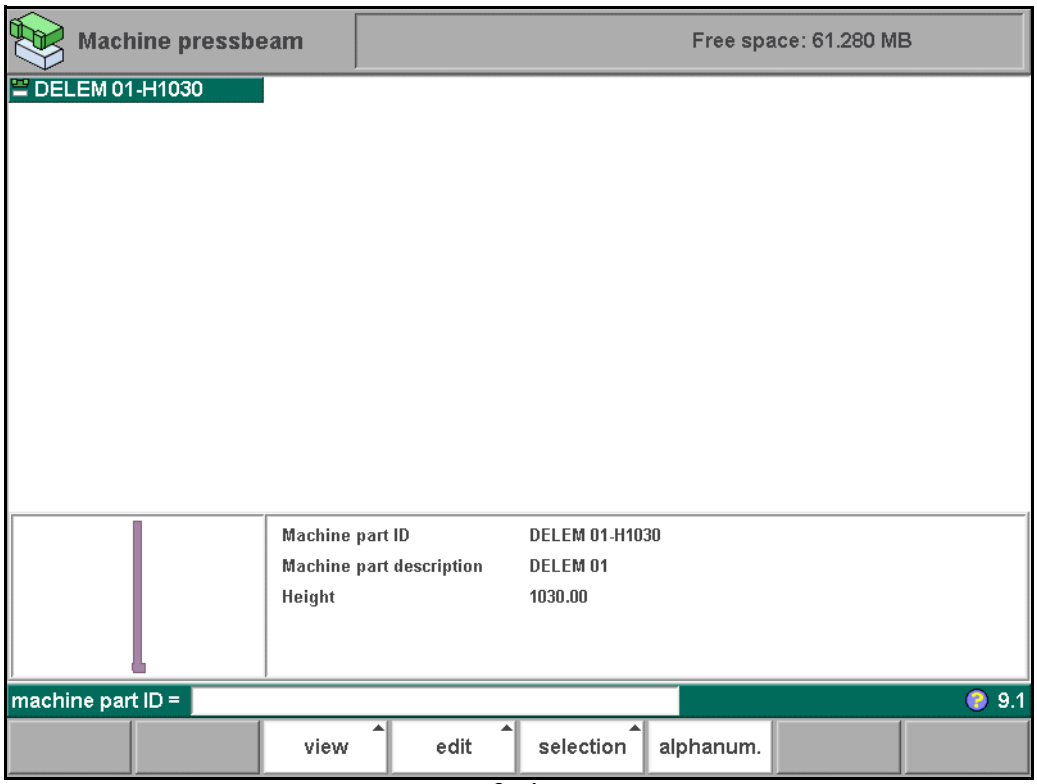
Second automatically calculated safety zone (minimum X-axis value) if the die is used in the turned position.

These values are calculated each time the die drawing changes. These 2nd-values cannot be changed manually.
See for explanation on the safety zones the example drawing.



6.aa

6.4. Machine upper side and lower side



6.ab

Function keys:

S3	**view**	Button to select one out of two possible viewing modes: - normal dir. - graphical dir.

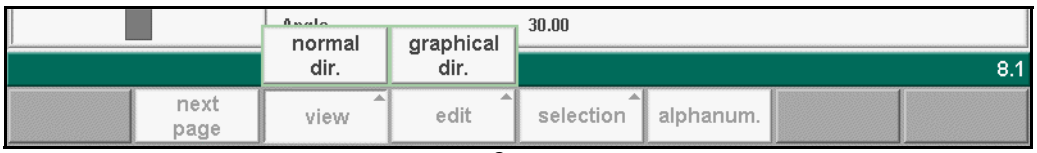
Function keys:

S4

edit

Edit the tool library with one of the following commands:
- edit
- add
- copy
- delete.

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



6.ac

Function keys:

S3

normal dir.

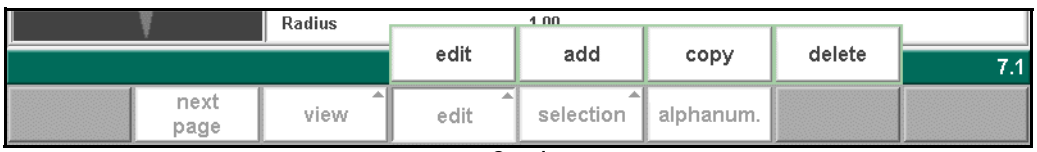
Select the normal view of the tool collection. This is the standard setting.

S4

graphical dir.

To get a graphical overview of the shapes as available in the library including main dimensions.

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 ‘edit’ has been pressed a new, temporary button bar appears with additional function keys:



6.ad

Function keys:

S4

edit

Modify the currently selected tool.

S5

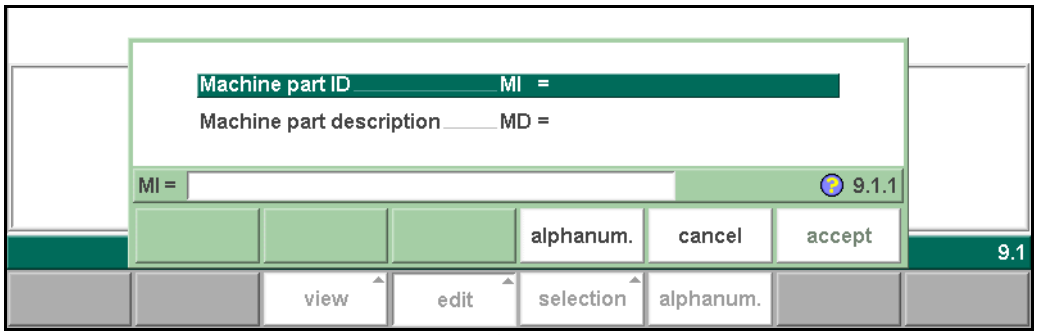
add

Create a new tool and add it to the tool collection.

Function keys:

S7 **delete** Delete the currently selected tool.

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 back gauge position of the machine. To create a new machine part, press the key ‘edit’ in the library and subsequently the key ‘add’. The control will start by asking for an ID and a description.



6.ae

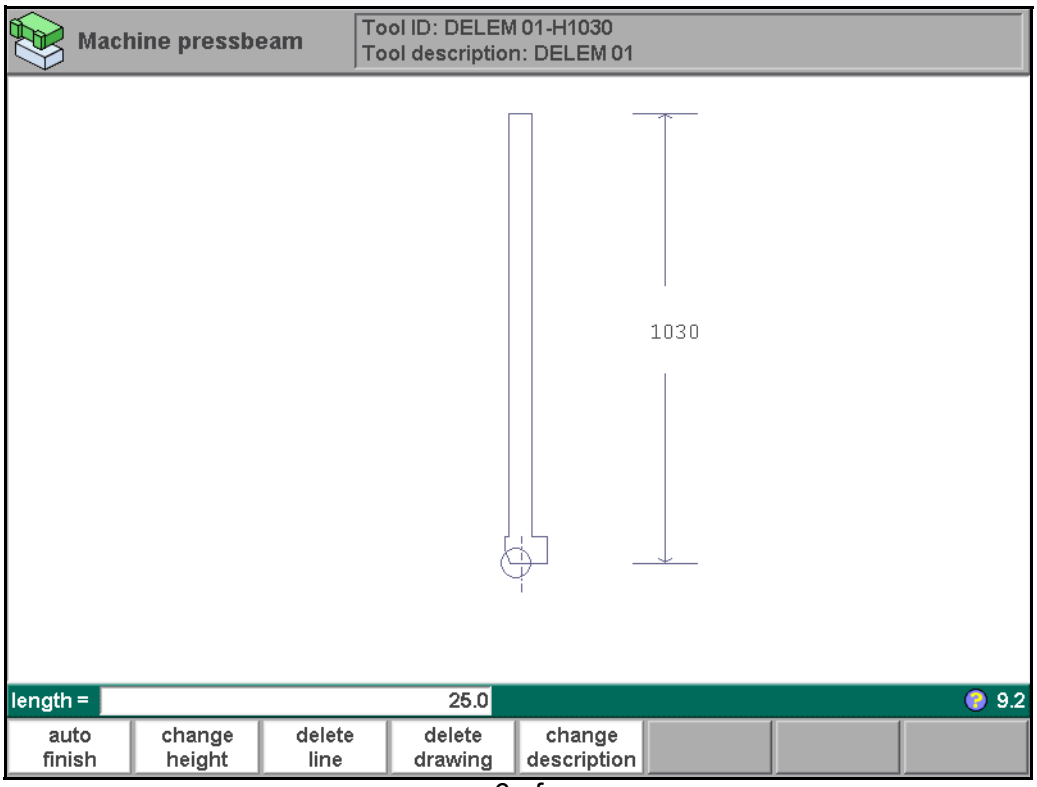
Machine part ID . . . .TI

A unique name or number to identify the machine part. The maximum length is 25 characters.

Machine part description . . . .TD

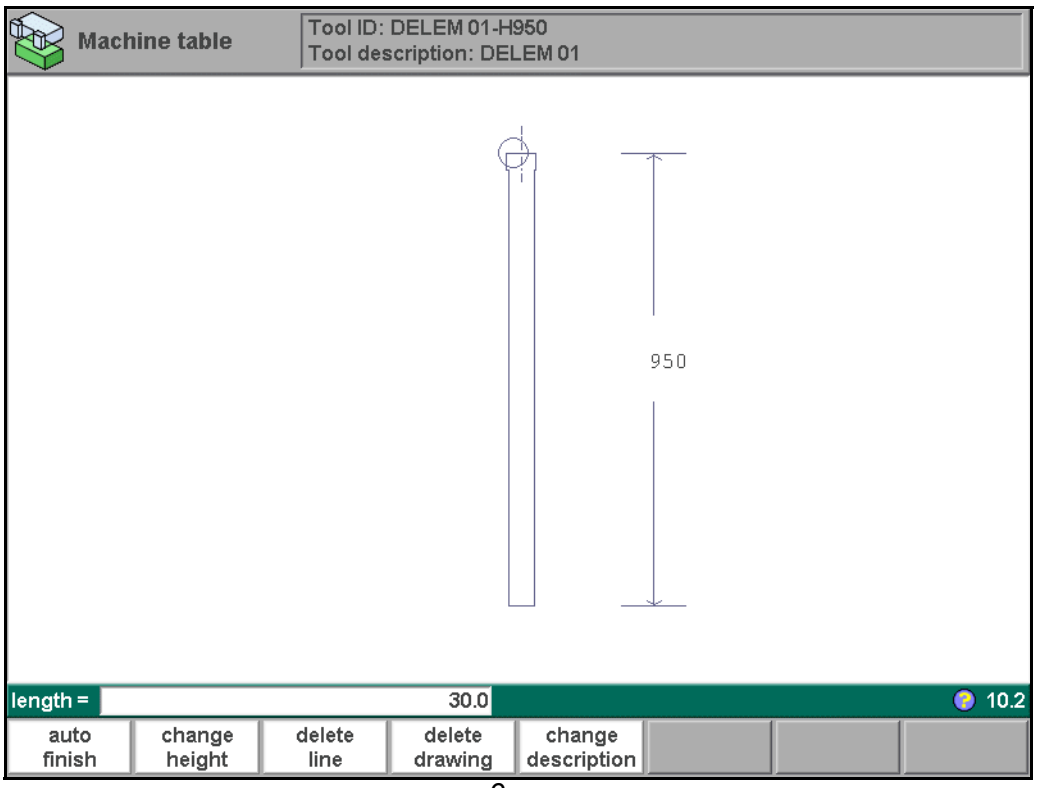
A name or description of this machine part. The maximum length is 25 characters.

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.



6.af

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

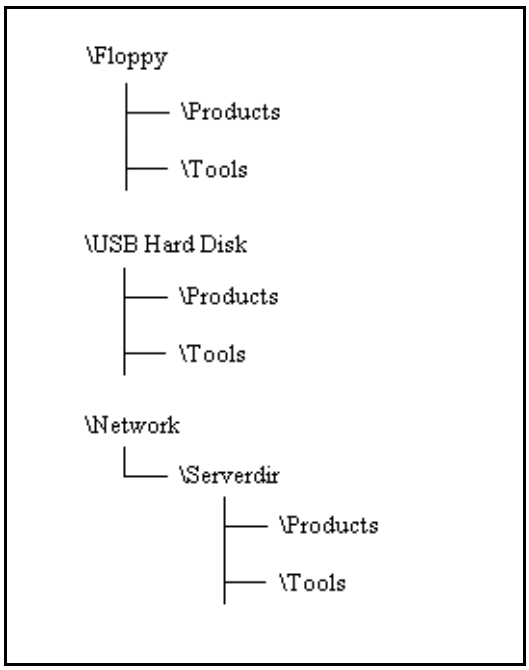


6.ag

7. Products and tools back-up

7.1. Introduction

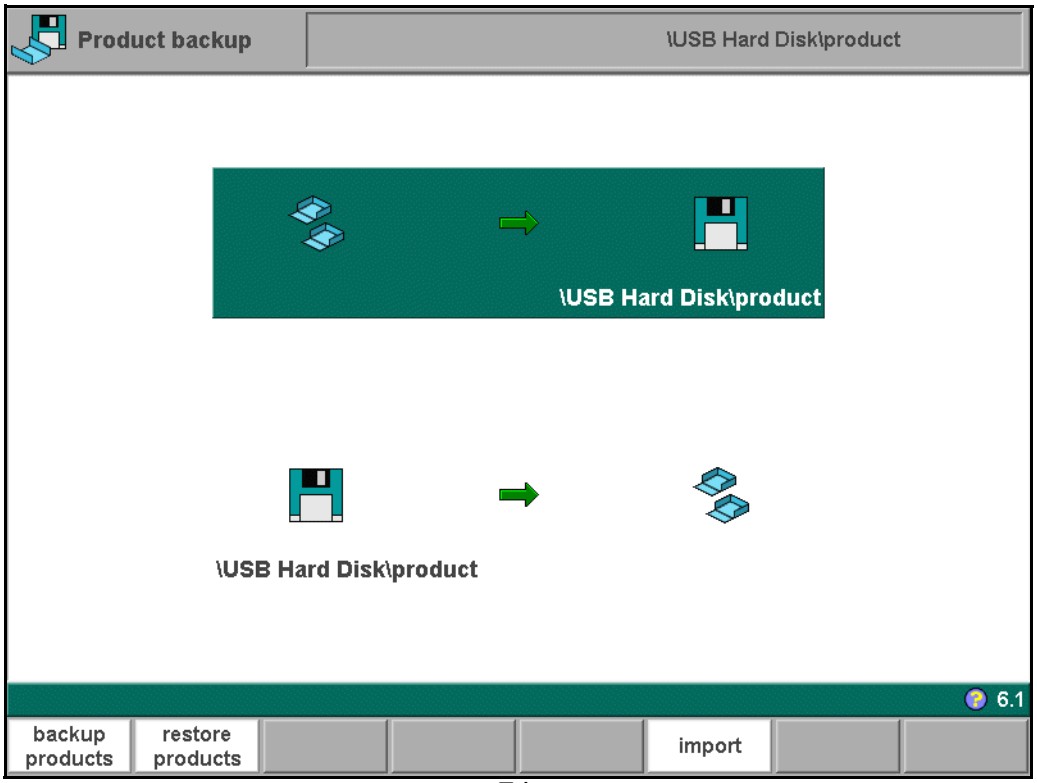
This chapter describes the procedures to make a back-up of your products and tools. The procedures for saving or reading data are similar for all types of back-up media: floppy disk, network or USB disk.
The actual back-up directory consists of a device (USB disk, network) and a directory. The choice 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.



7.a

7.2. Product backup

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



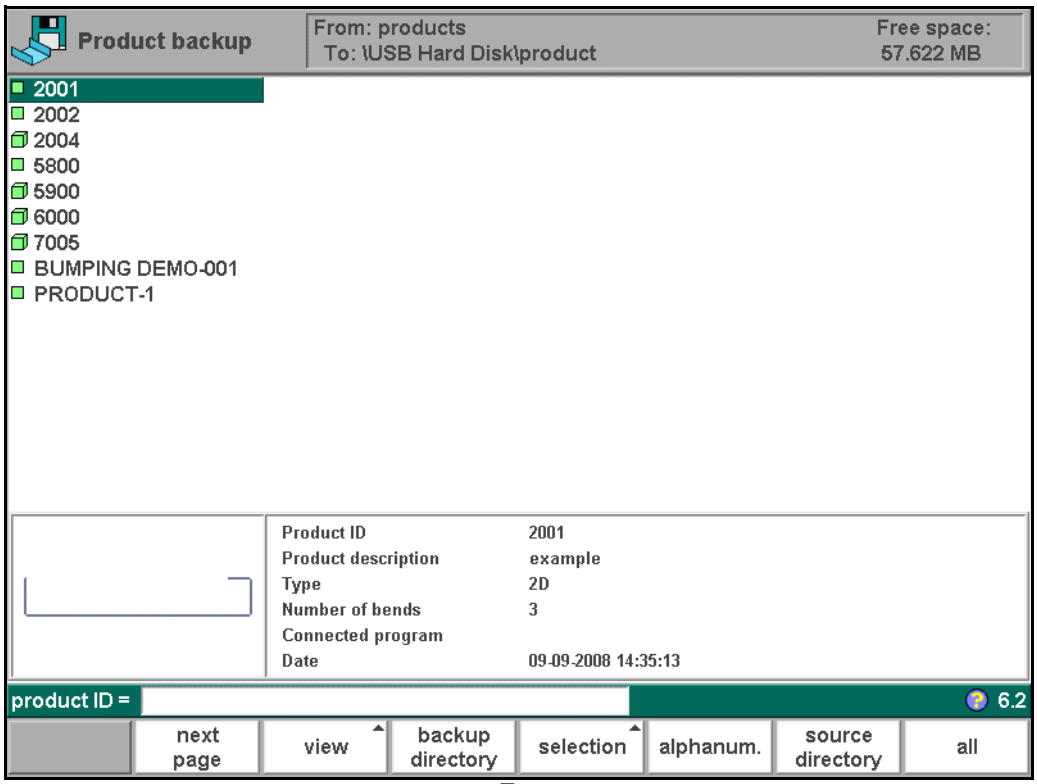
7.b

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.
S6	**import**	Import product programs from previous control software versions.

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



7.c

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 function key ‘backup directory’ or you should go to the program constants menu to change the device where products are saved to.
To save a product, select its number from the list and press ‘enter’. If a product file with the same name is present on the back-up location, a question is offered whether or not to replace that file.

Function keys:

S3	**view**	Button to select one out of two possible viewing modes: - normal dir. - graphical dir.
S4	**backup directory**	Change, make or remove backup directory
S5	**selection**	Function key to open a new button bar with extra function keys for product selection functions: - product ID - product description - type - date
S6	**alphanum.**	Select alphanumeric character during typing.

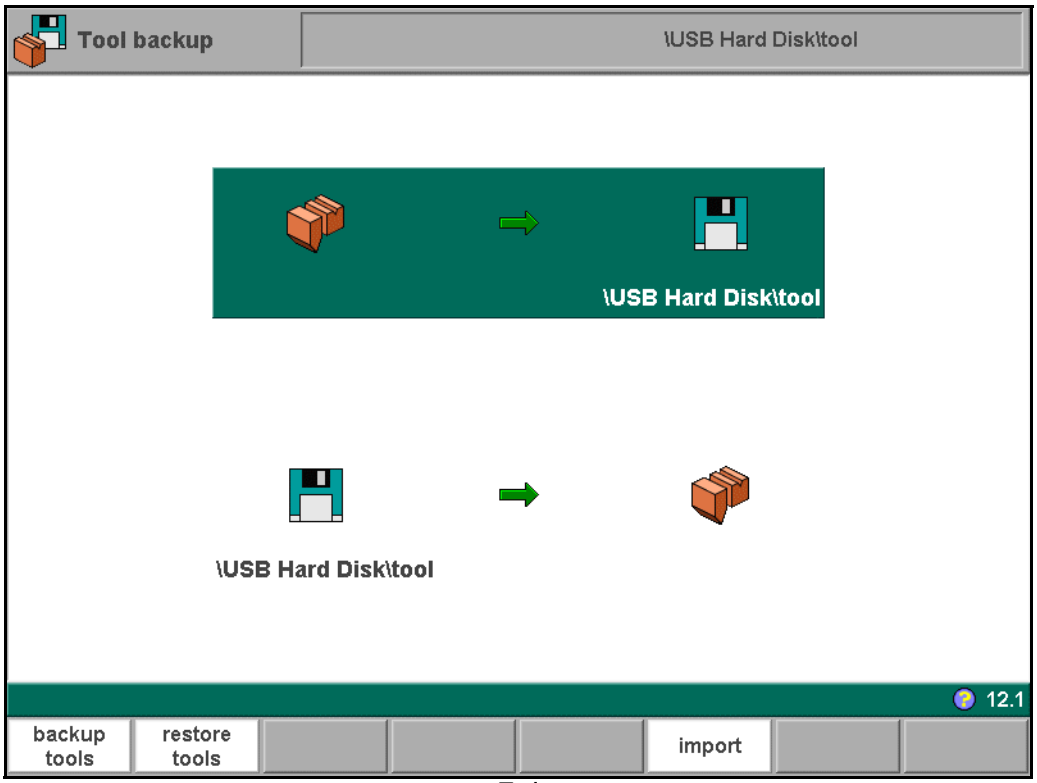
Function keys:

S7	**local directory**	get an overview of the manually created product directory structure on the control disk.
S8	**all**	make a backup of all products

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.

7.3. Tool back-up

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

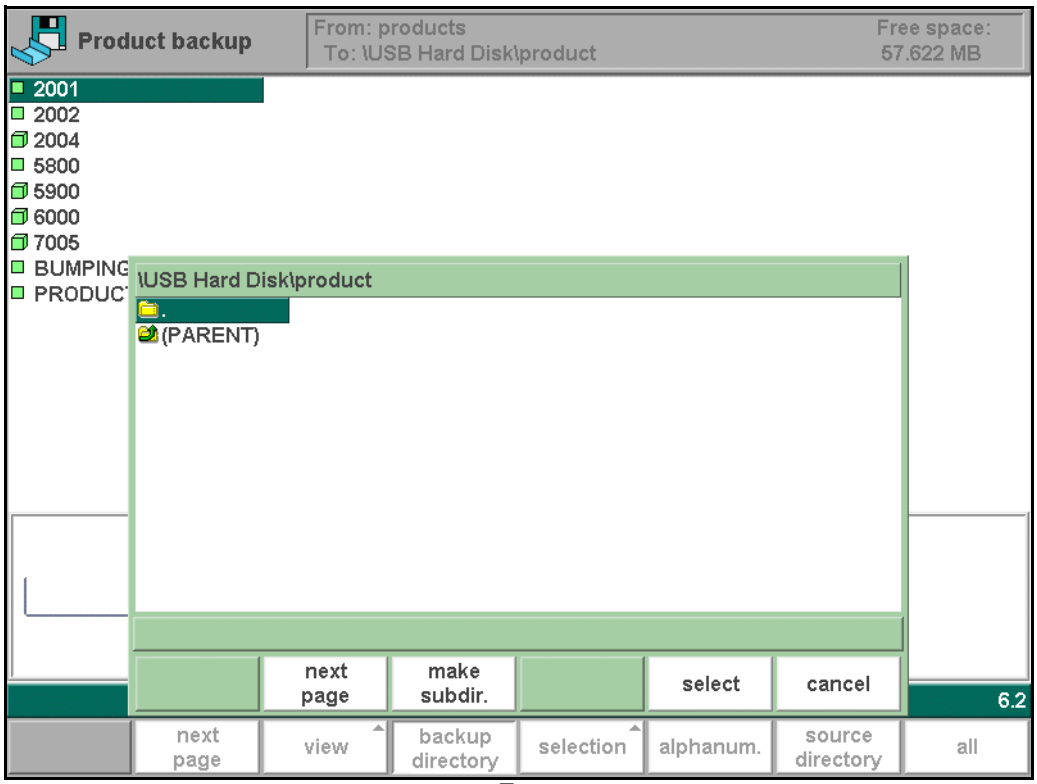


7.d

With this menu a back-up of tools on the control can be made: punches, dies or machine shapes. The procedures for a tool back-up run similar to the procedures for a product back-up.

7.4. Directory navigation

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



7.e

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 function keys ‘make subdir’ and ‘remove subdir’. If there are subdirectories present, use the arrow keys to move to the required directory and press ‘select’ to select it.

Function keys:

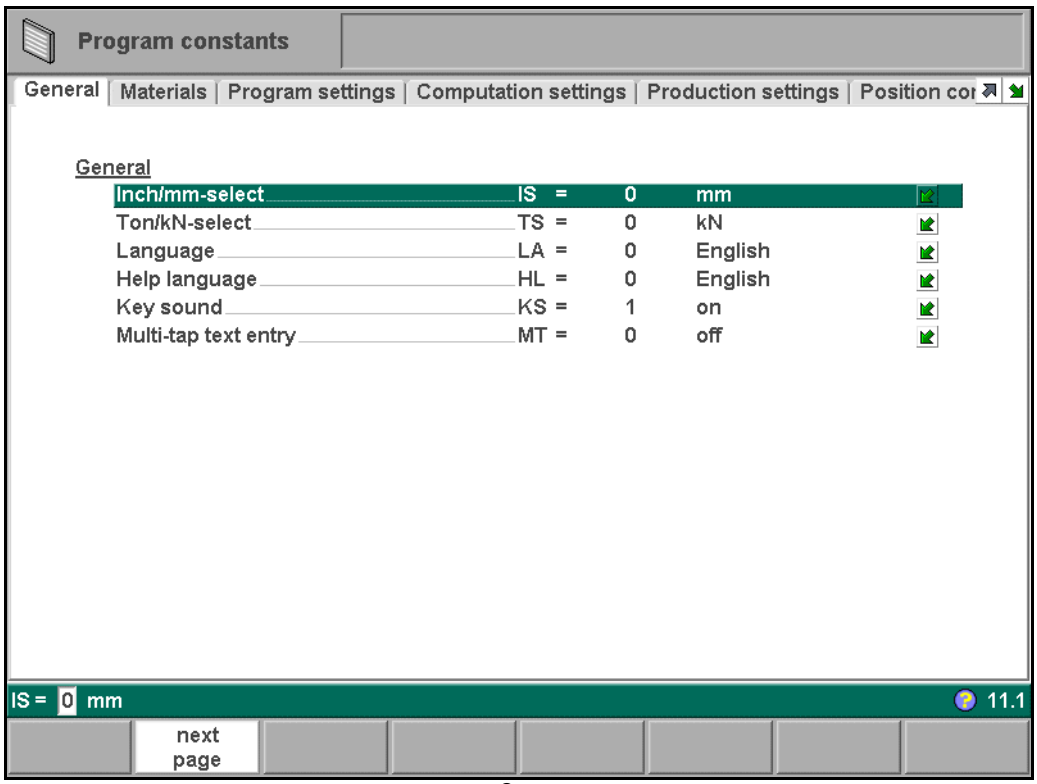
S4	**make subdir**	Add a new subdirectory. The name of the subdirectory can be any alpha numeric string of maximal 24 characters. Avoid the use of slashes (‘/’).
S5	**remove subdir**	Delete a subdirectory. If the subdirectory is not empty the controller returns the message ‘UNABLE TO REMOVE DIRECTORY’.

8. Program Constants

8.1. 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.

8.2. General



8.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

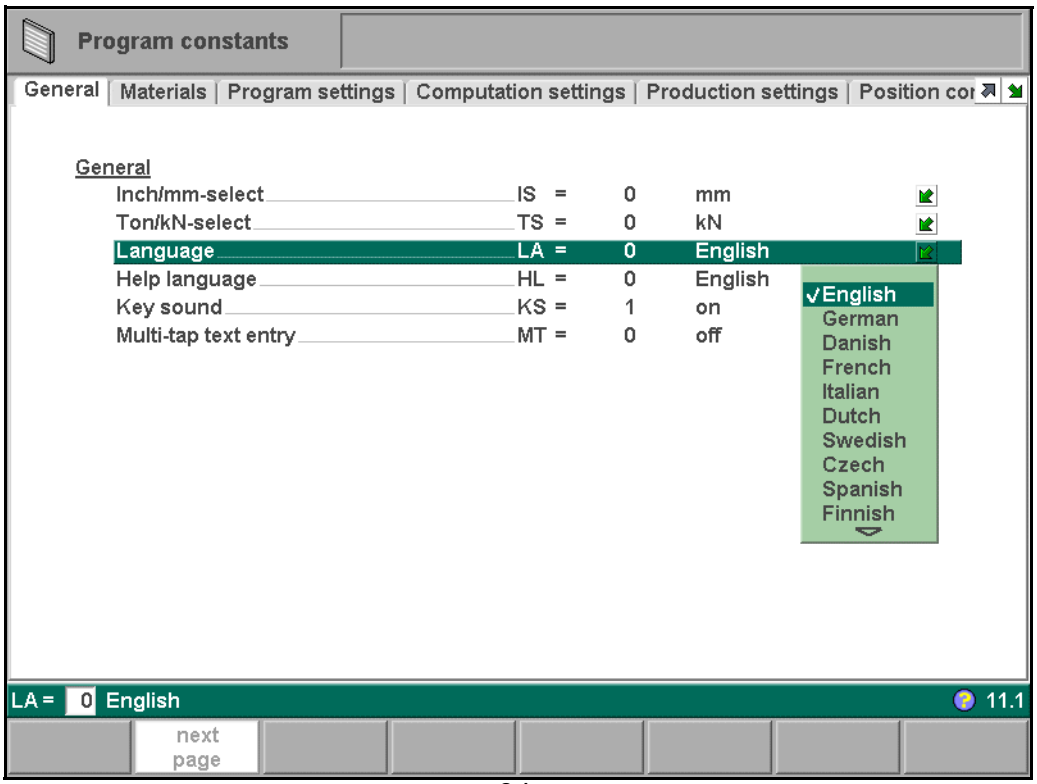
1 = dimensions in inches
0 = dimensions in millimetres

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.



8.b

Language . . . . . .LA

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

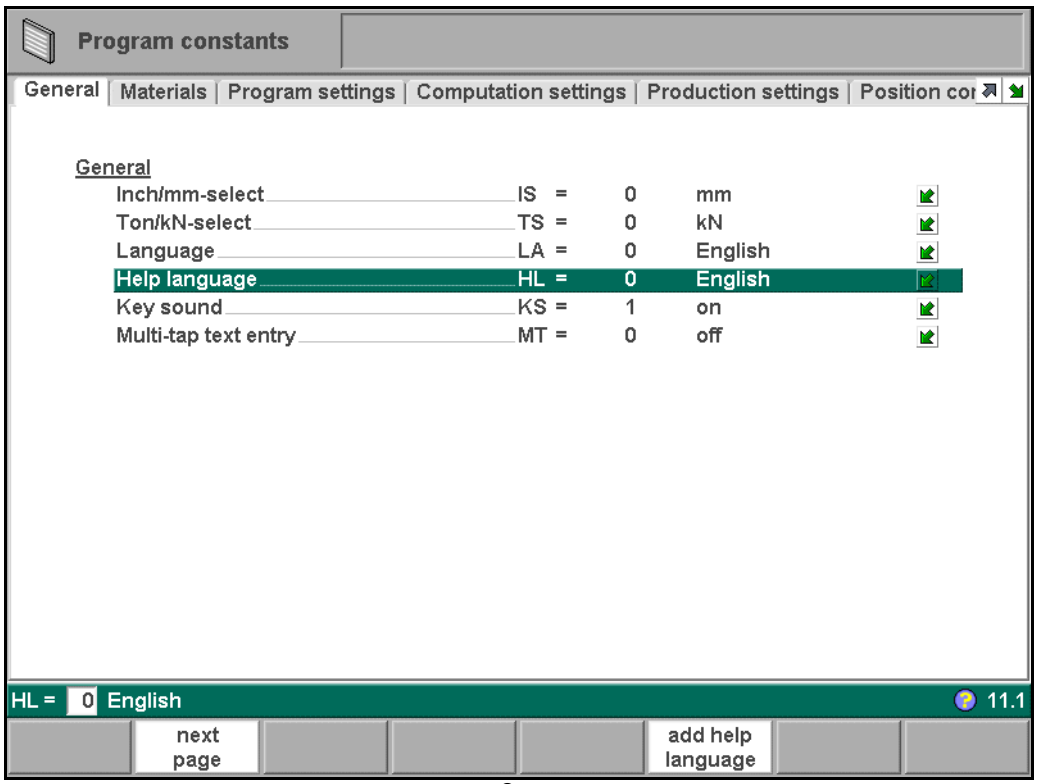
0 = English	12 = Lithuanian
1 = German	13= Slovenian
2 = Danish	14= Turkish
3 = French	15= Russian
4 = Italian	16= Brazilian
5 = Dutch	17= Hungarian
6 = Swedish	18= Romanian
7 = Czech	19= Greek
8 = Spanish	21= Chinese
9 = Finnish	22= Chinese Traditional
10= Portuguese	23= Korean
11 = Polish	24= Japanese

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
2 = Danish
3 = French
4 = Italian
5 = Dutch
6 = Swedish
7 = Czech
8 = Spanish
9 = Finnish
11 = Polish
14= Turkish
17= Hungarian
18 = Romanian
21 = Chinese

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).



8.c

Key sound . . . . . .KS

Switch the sound function of the input panel on or off.
0 = off: no sound when a front panel key is pressed.
1 = on: a sound is issued when a front panel key is pressed. This is the default setting.

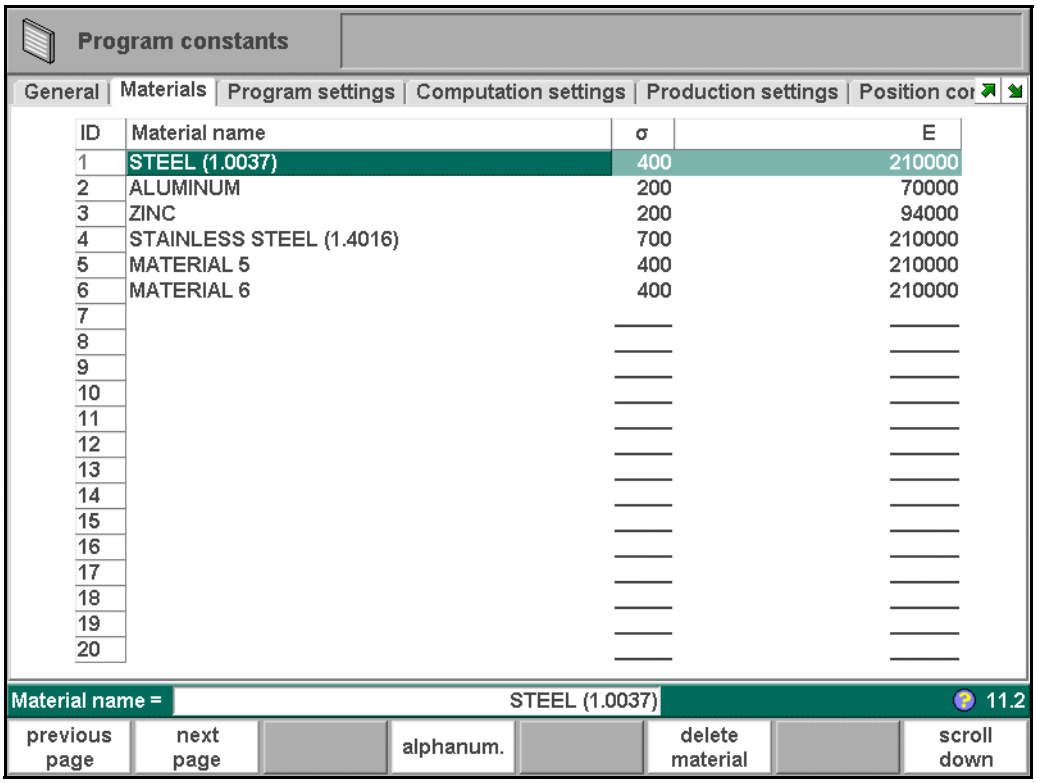
Press the key ⌂ to select the required setting.

Multi-tap text entry . . . . . .MT

Choose the manner of text entry through the front panel.
0 = off: text can be entered by pressing a front panel key and the function key 'alphanum.' for character selection. This is the default setting.
1 = on: text can be entered by pressing a frontpanel key repeatedly until the correct character appears. This works similar to a cell phone.

Press the key ⌂ to select the required setting.

8.3. Materials



8.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 strength of the selected material.

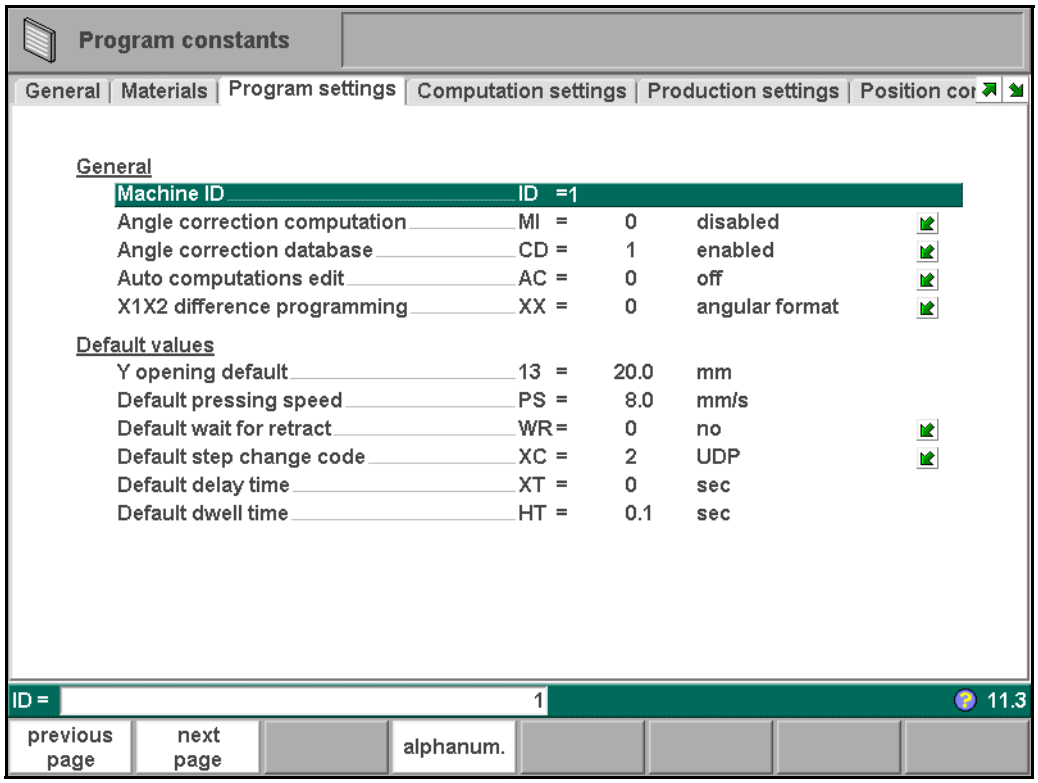
E module . . . .E
E- module of the selected material.

Function keys

	**delete material**	To delete the material that is currently shown on the screen.
	**scroll up**	To flip upwards through the material pages.
	**scroll down**	To flip downwards through the material pages.

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.

8.4. Program settings



8.e

Machine ID . . . . . .ID

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

Angle correction computation . . . . . .MI

Parameter to switch computation of angle corrections on or off.
0 = disabled: no correction computation from measured angles.
1 = 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:
- Force
- 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:
- Force
- 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.

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

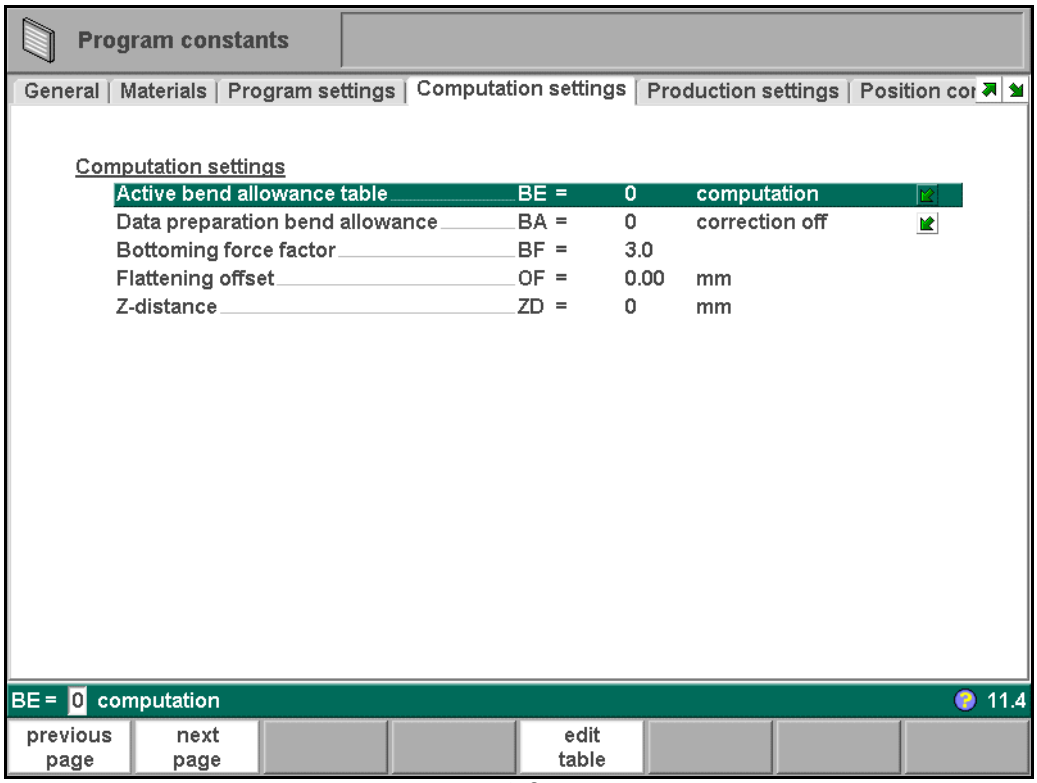
Default wait for retract . . . . . .WR
Default value for the parameter ‘wait for retract’ in a bend program. This parameter determines the control behaviour in a bend program. Initially the parameter ‘wait for retract’ is set to the value programmed here during postprocessing and during programming.
The default value of this parameter = 0.

Default step change code XC
Default value for the parameter ‘step change 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 is 2 (UDP).
See chapter ‘data preparation/data edit’ for more information about the ‘step change code’ parameter.

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

Default dwell time . . . . . .HT
Default value for the parameter ‘dwell time’ in a bend program.

8.5. Computation settings



8.f

Active bend allowance table. . . . . .BE

0 = computation
1 = table
Bend-allowance: correction of the X-axis due to sheet shortening after bending. With this parameter the method for bend-allowance calculation is chosen. ‘Computation’ means the standard formula of the control is used to calculate the bend-allowance. ‘Table’ means a bend-allowance table with correction values can be used. The choice ‘table’ is only available if a bend-allowance table is present on the control.

Press the key ⌂ to select the required setting.

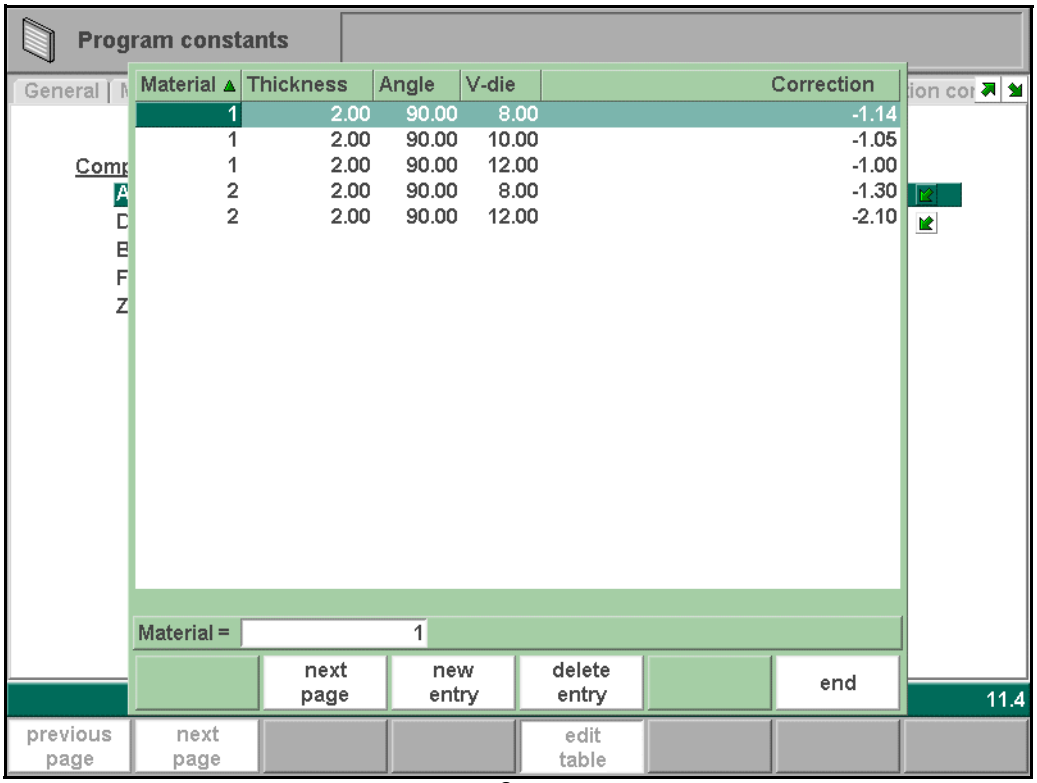
Data preparation bend allowance . . . . . .BA

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 post process function in the drawing mode. When the control uses the post process command to calculate a CNC program from a drawing with bend sequence, the control will always 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 function key the table can be modified to meet user requirements. The table appears in a new window, with its own set of function keys.



8.g

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

Function 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

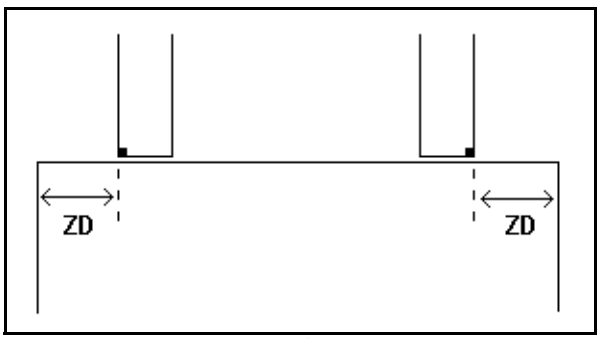
The force needed for an air bend multiplied by this factor in order to obtain the bottoming force.

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.

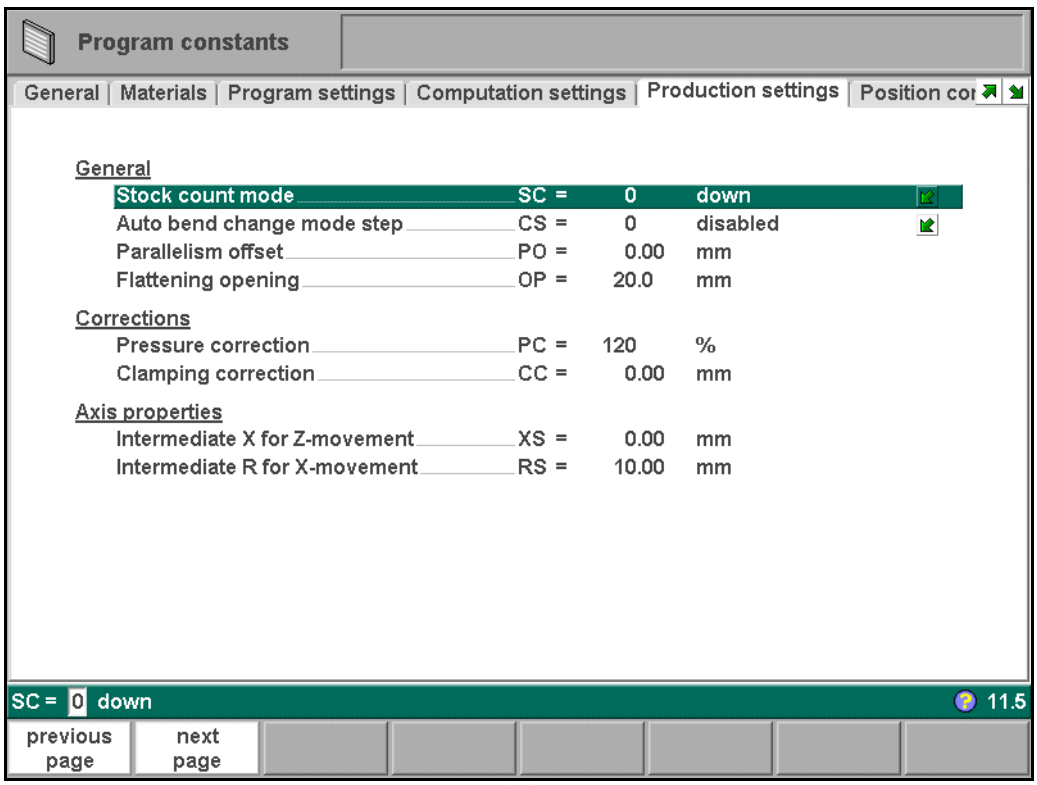
Z-distance. . . . . .ZD

The distance from the edge of the finger to the corner of the sheet.
When automatic Z-axes are installed, the position of the fingers is automatically calculated with respect to the end of the sheet.



8.h

8.6. Production settings



8.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 down counting 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 up counting is selected, the counter is increment after each press cycle.
Down counting can be useful if a pre-planned quota must be produced. Up counting could be used to give a report on production progress.

Press the key to select the required setting.

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

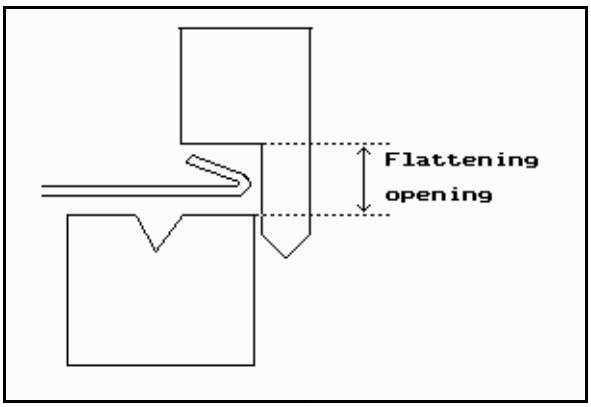
This parameter can be used to have automatic step change in the bending process with the "step by step"-mode. To be programmed 0 or 1.
When programmed 0:
No automatic step change (next bending parameters active) will take place. To perform the next bending you must select the new bending and press the 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 overall 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. See figure below. The opening position will also take 2x sheet thickness into account.



8.j

Pressure correction . . . . . .PC

Percentage of calculated force which actually controls the pressure valve.

Clamping correction. CC

The position of the beam at which the sheet is clamped, is calculated. In order to have a firm clamped sheet 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.

Part support return speed . . . .PR=

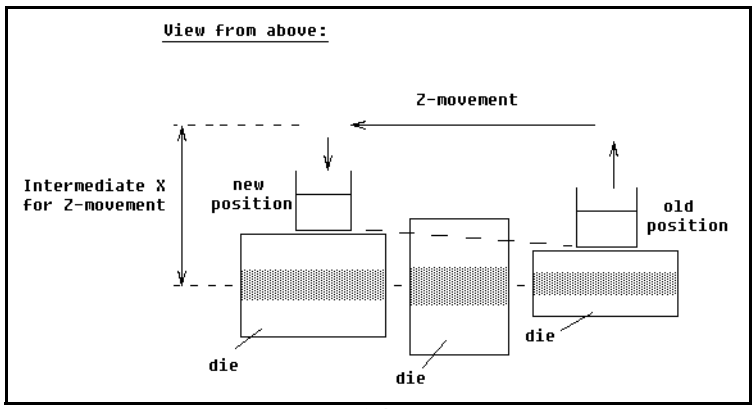
Return speed of the part support after a bending. The speed value is programmed as a percentage of the maximum speed.
Only available when there is a part support on the system.

Part support extra angle . . . .PE=

Extra angle for the part support unit. This angle value is added to the part support angle position at the end of decompression. So while the pressbeam moves up to its upper dead point, the part support moves to this angle. When the beam has reached upper dead point, the part support will move back to zero.
Only available when there is a part support on the system.

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.



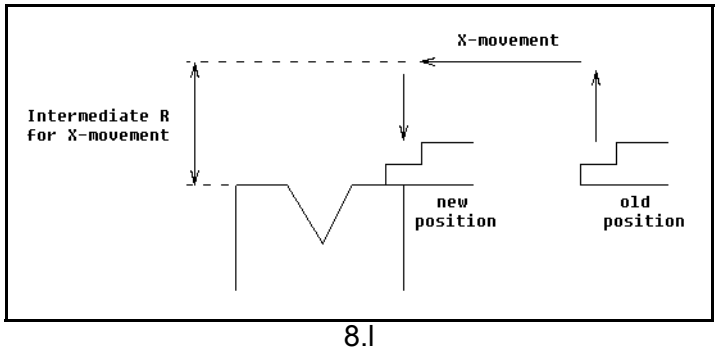
8.k

When the back gauge 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: back gauge 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: back gauge 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: back gauge 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 back gauge to its new position (see figure 8.k.).

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.



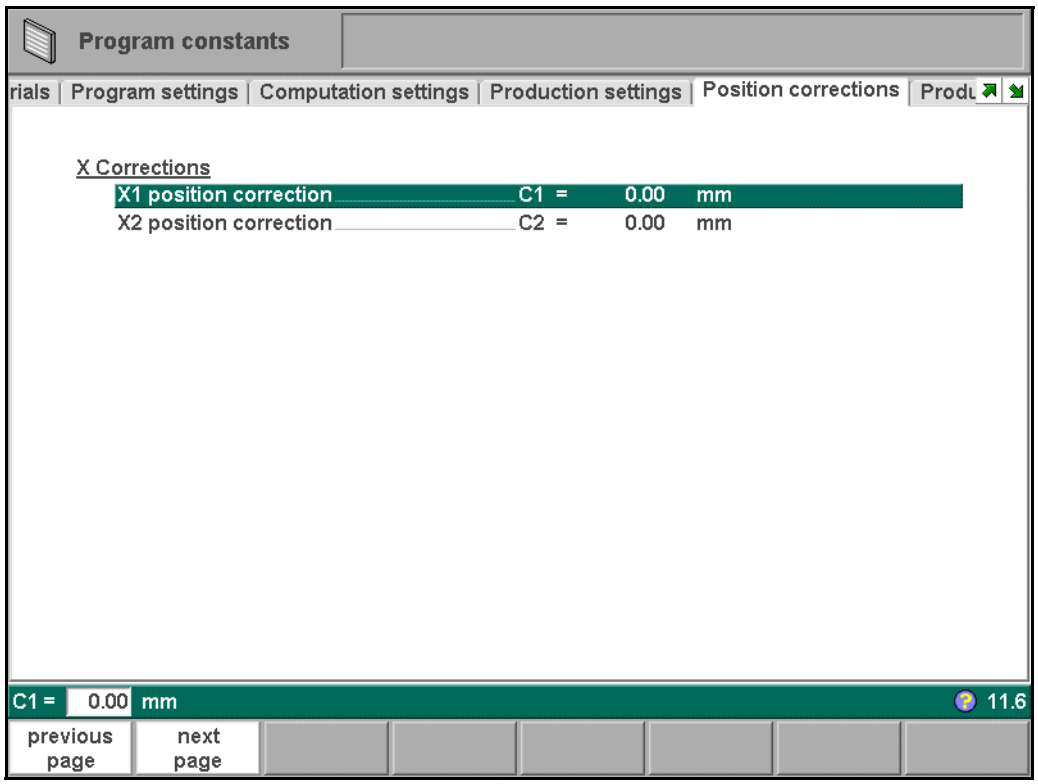
8.l

The sequence will be as follows:

- The R-axis is moved to the intermediate position;
- then the X-axis is moved to its intended position;
- finally the R-axis is moved to its intended position.

The safety zone of the die is defined as follows:
SZ = X-safe + SD
Explanation:
SZ = safety zone
X-safe = safety zone of the die
SD = safety distance, defined by machine supplier

8.7. Position corrections



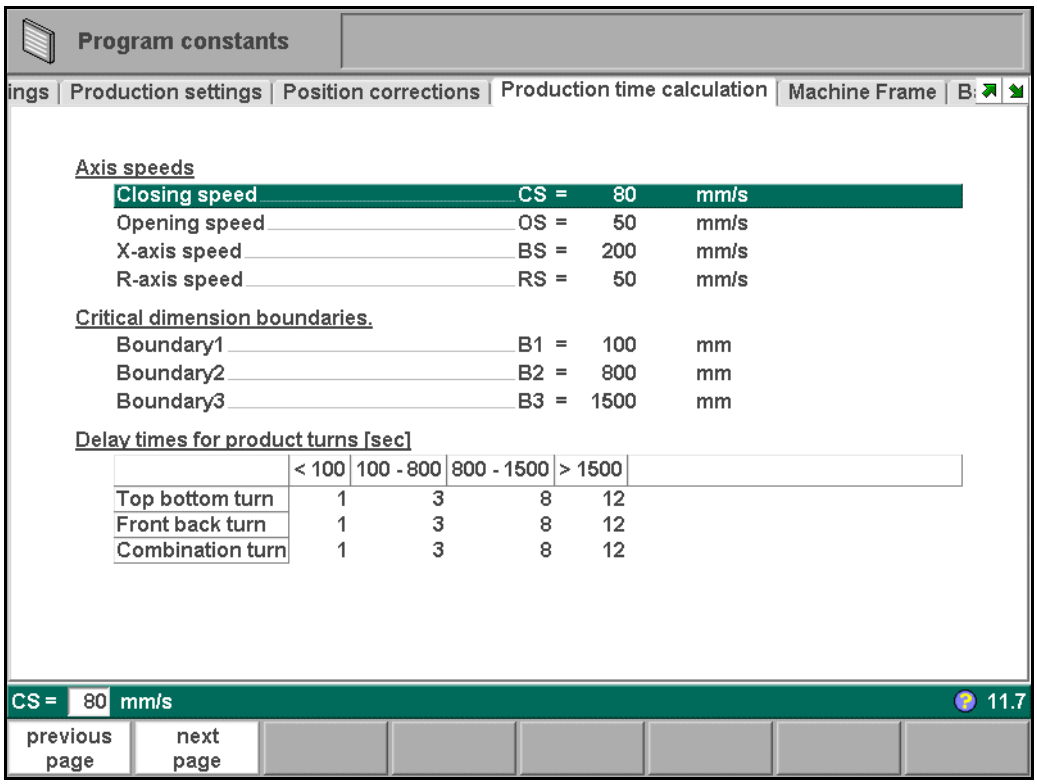
8.m

X position correction CX

When the actual, mechanical 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 CX parameter = -2.
- When the programmed and displayed value = 250 and the actual, mechanical position value = 248 the CX parameter = +2.
In case there are several X-axes installed, a separate parameter is available for each X-axis.

8.8. Production time calculation



8.n

The parameters on this page are used to calculate the production time for a product. This production time depends on the positioning speed of the axes and the product handling times.

Closing speed . . . .CS=
Speed of the Y axis during fast closing.

Opening speed. . . . . ..OS=
Speed of the Y axis during opening.

X-axis speed. . . . . ..BS=
Working speed of the back gauge of your press brake.

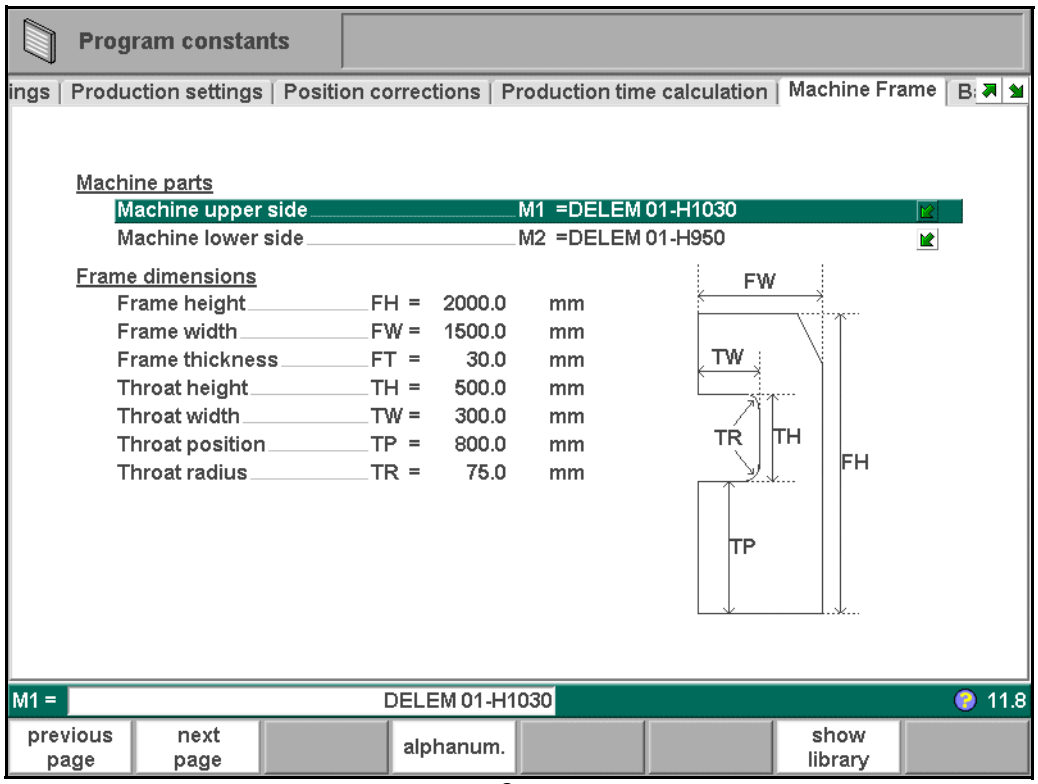
R-axis speed . . . .RS=
Working speed of the R-axis fingers (only in case you have an R-axis in your machine)

Turning of a product takes production time. This time depends upon the length and width of your product.

For a relative small product (in Z-direction) a top-bottom turn can be done quickly.
But a relative small product which is long (in x-direction) needs some longer time to turn from front to back or in a combination turn.
The turn time can be set in a table in seconds. For this purpose there are 4 length intervals (3 boundaries) each with a specific turn time depending on the type of turn. Like the turn times you also can set the length limit boundaries.

Boundary 1/2/3 . . . . . .B1/2/3
Threshold values for product size. For values between these boundary values different product turn times can be programmed.

8.9. Machine frame



8.o

On this page the dimensions of the side frame of the machine are programmed. The programmed machine will be used in the simulation screen during graphical programming. Watch the on-screen picture for the parameter meaning.
At the top of the screen the possible machine parts can be selected. The selected machine parts are used when a new tool configuration is programmed in the graphical programming menu. See chapter 3 for more information about the tool configuration.

Machine upper side M1
Select the relevant machine upper side.

Press the key  to select the required setting. To search for the correct machine parts, press the function key ‘show library’.

Machine lower side M2
Select the relevant machine table.

Press the key  to select the required setting. To search for the correct machine parts, press the function key ‘show library’.

Frame height . . . .FH
Total height of machine frame.

Frame width . . . .FW
The width (or depth) of the machine frame.

Frame thickness . . . .FT
The thickness of the side frame.

Throat height . . . .TH
The height of the workspace (the ‘throat’) in the machine.

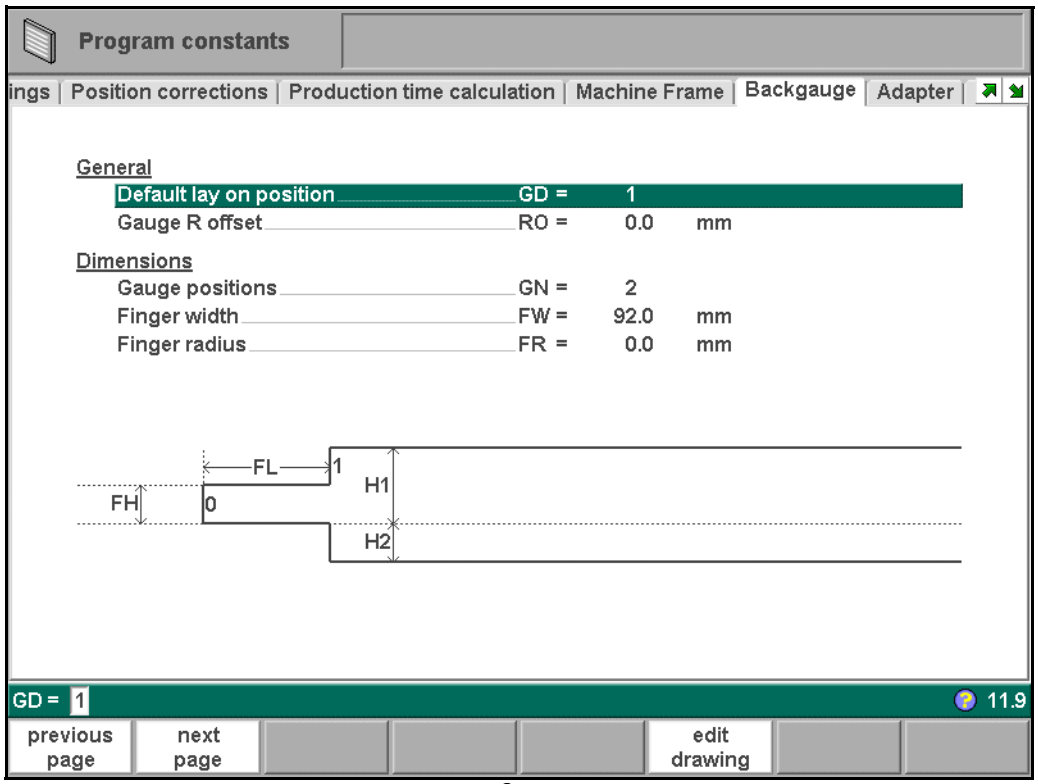
Throat width . . . .TW
The width (or depth) of the workspace.

Throat position . . . .TP
The position of the workspace in the machine.

Throat radius . . . .TR
The radius of the workspace corner.

8.10. Back gauge dimensions

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



8.p

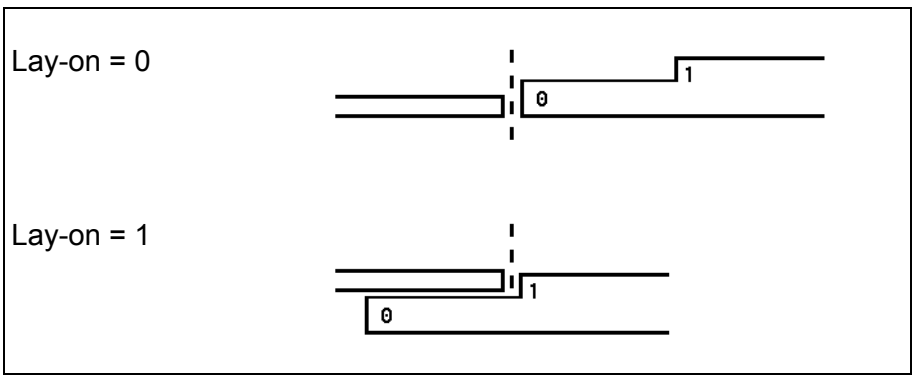
Gauge positions . . . . . .GN

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

This is the default lay-on position in case a lay-on position must be used during automatic bend sequence computation.

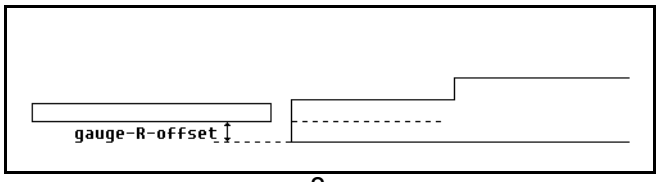
Meaning of different lay-on numbers:



Gauge R offset . . . . . .RO

An offset value for the R-axis can be set if the back gauge is positioned against the

sheet edge and the X-axis position is outside the die safety zone.

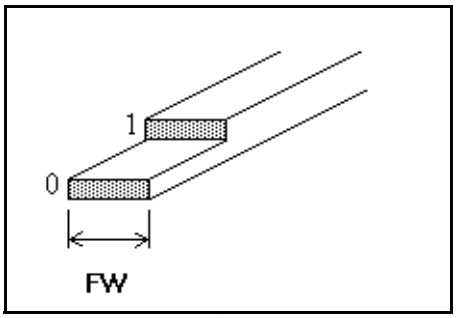


8.q

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

Finger width . . . .FW

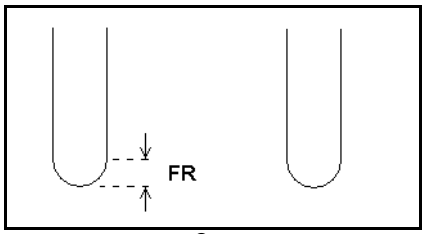
The width of the back gauge finger.



8.r

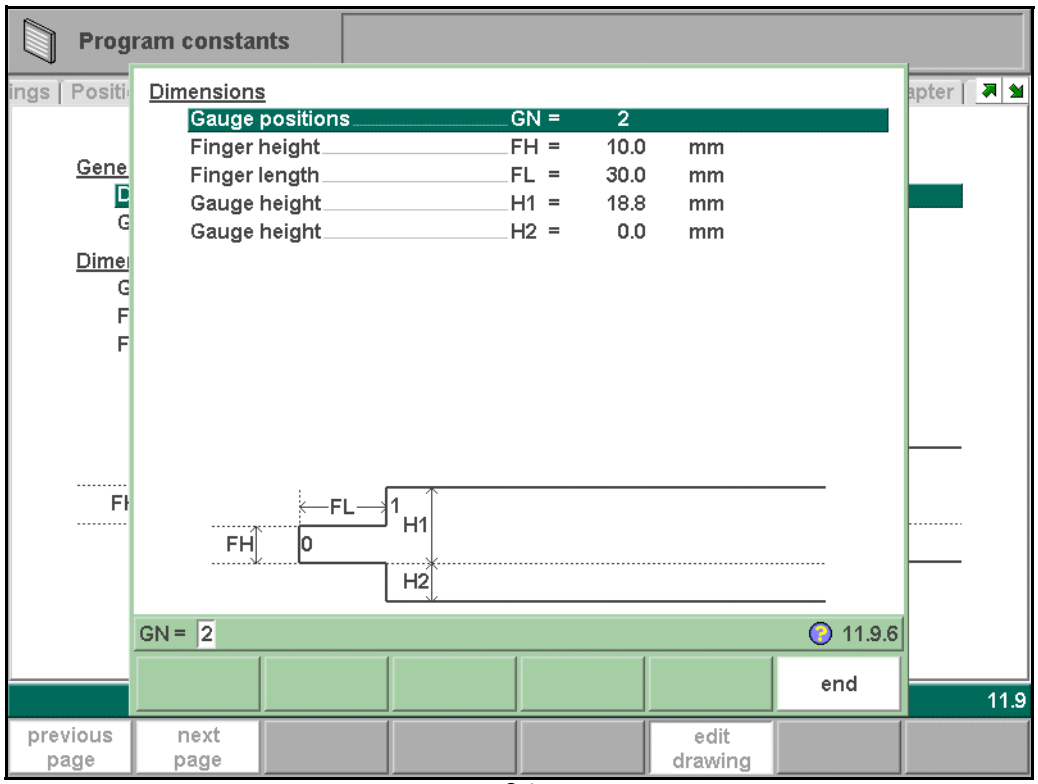
Finger radius . . . .FR

The radius of the back gauge finger.



8.s

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



8.t

The following parameters describe the dimensions of the back gauge 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 back gauge finger tip.

Gauge height H1/H2/H3/H4

The height of the different finger levels.

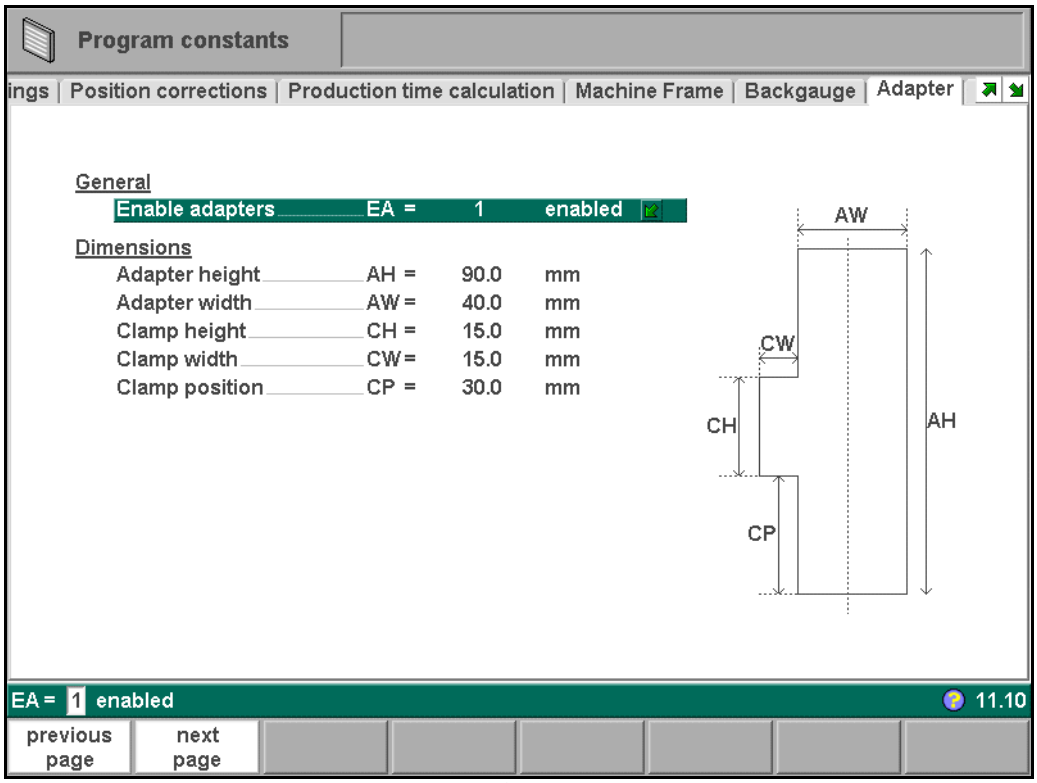
Finger length . . . . . .FL

The length of the first finger.

Gauge length L2/L3

The length of additional lay-on levels.

8.11. Adapter



8.u

On this page a tool adapter can programmed. The programmed dimensions are used for collision warnings.

Enable adapters. . . . . ..EA
Enable or disable the possibility to use an adapter.
Disabled(0): adapter cannot be programmed.
Enabled(1): in each program can be chosen whether or not to use an adapter.

Adapter height . . . .AH
The adapter height. Important: this height value will be used in bend depth calculation.

Adapter width. . . . . ..AW
The width of the adapter.

Clamp height . . . .CH
The height of the clamp of the adapter.

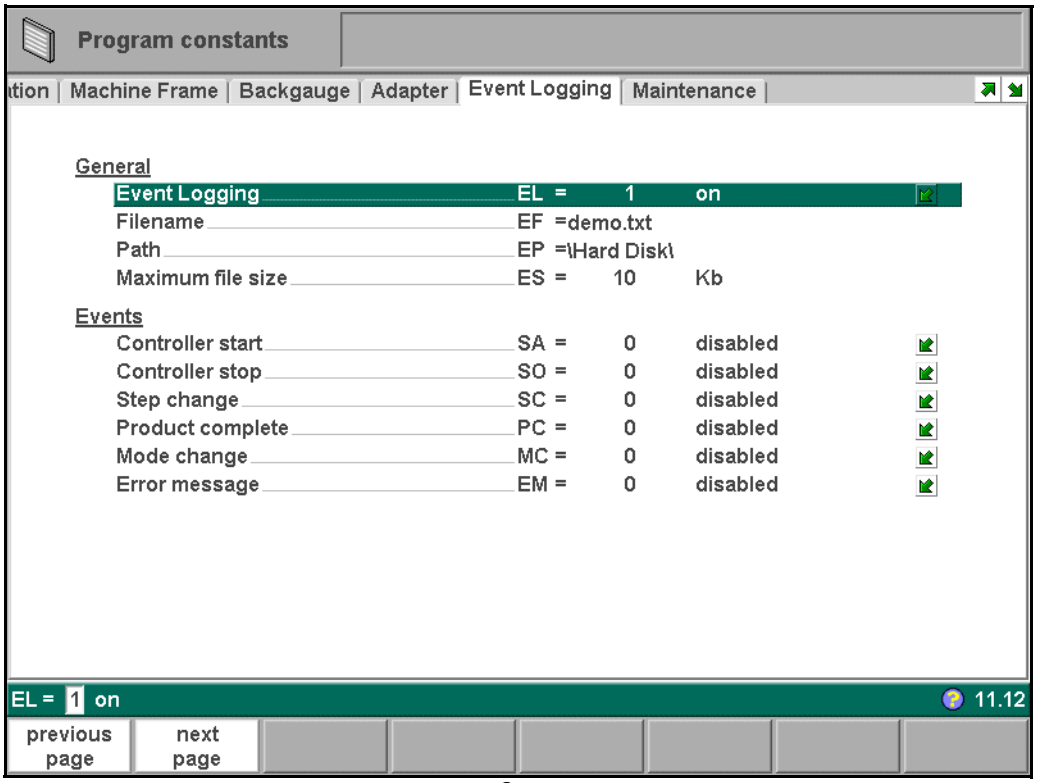
Clamp width . . . .CW
The width of the clamp.

Clamp position. . . . . ..CP
The position of the clamp, related to the adapter.

8.12. Event logging

8.12.1. Parameters

With this feature it is possible to register certain events on the control. These events are stored in text files that can be examined 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.



8.v

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.
When the cursor is on this parameter, use the function key ‘select path’ to select the desired location for the log file.

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.

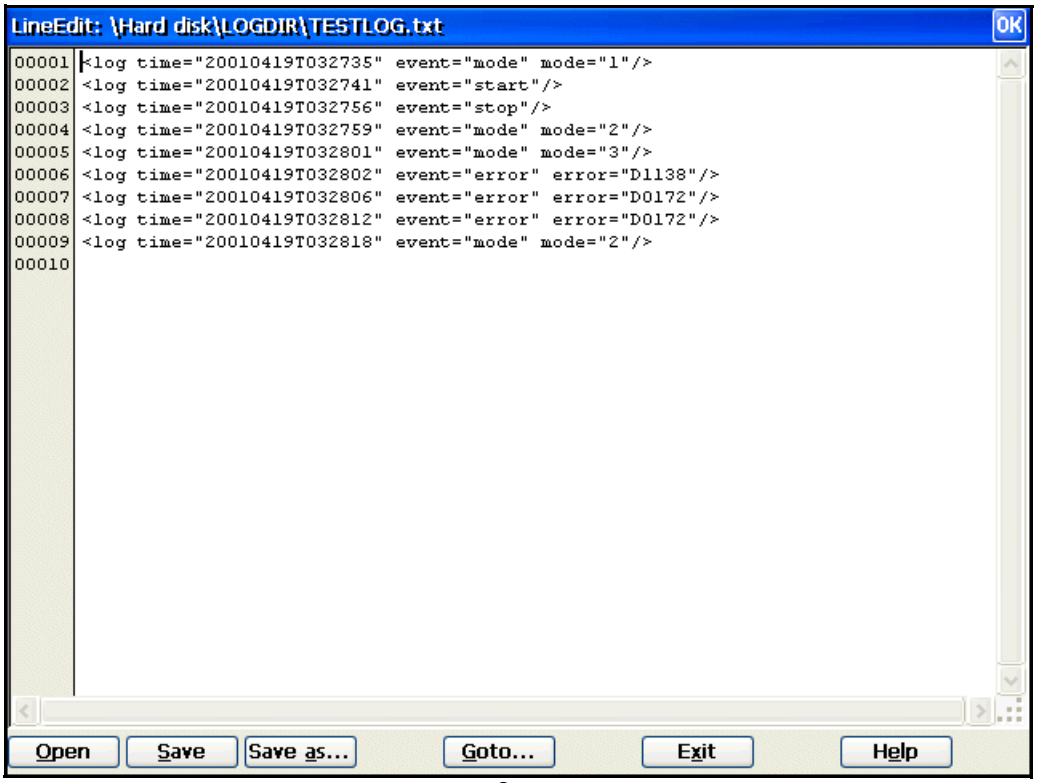
Step change . . . .SC
Log event: the control has switched to the next bend step.

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.

8.12.2. Explanation



8.w

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:

log 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.

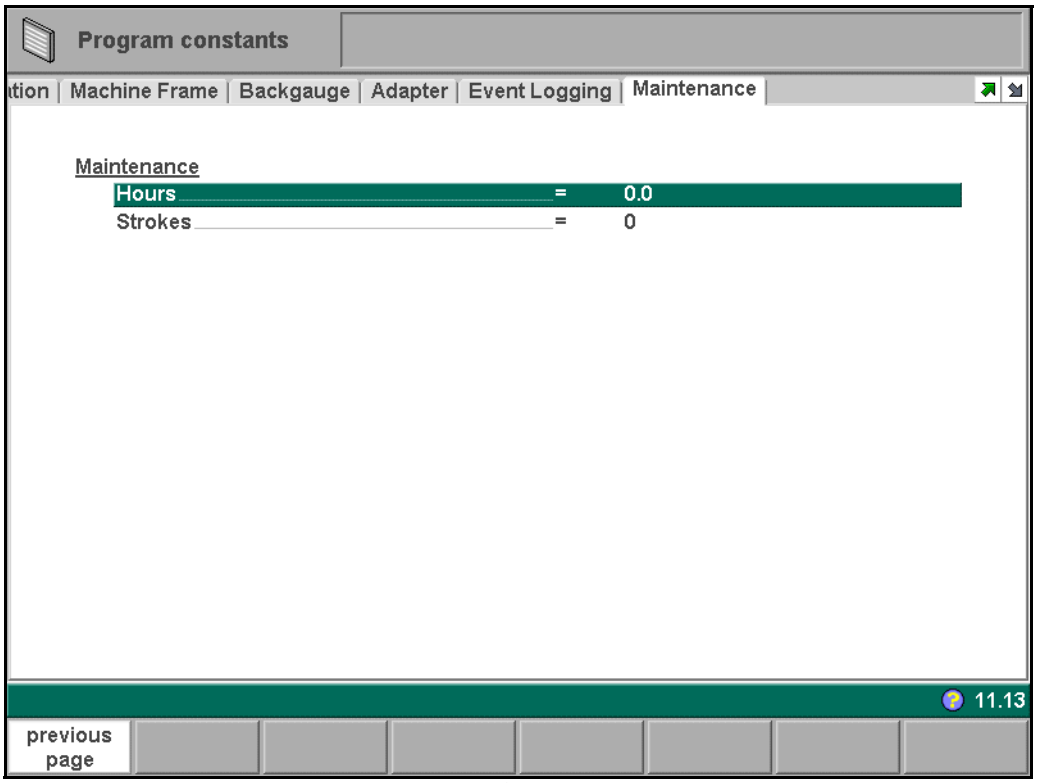
Event type	Keyword	Possible attributes
Mode change	mode	mode number: 1 = manual 2 = programming 3 = automatic 4 = step by step
Step change	step	product ID, step number
Control start	start	product ID, step number and stock counter
Control stop	stop	product ID, step number and stock counter
Product complete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

8.13. Maintenance



8.x

Hours . . . . . .
The number of hours the machine is running.

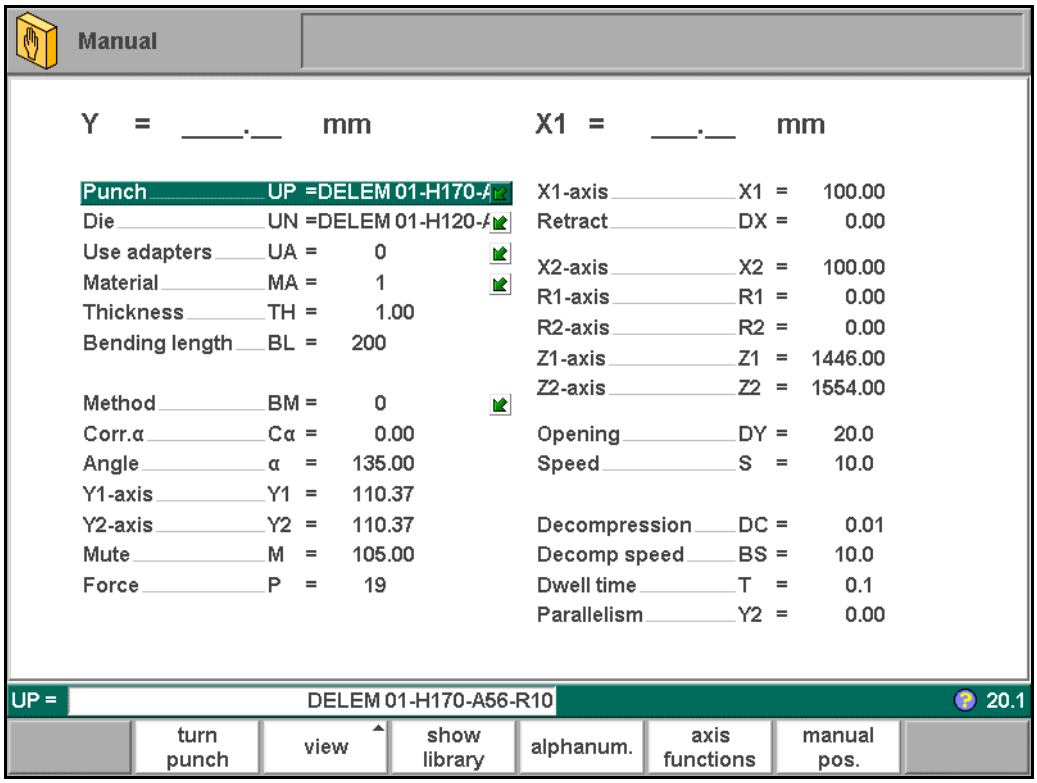
Strokes . . . . . .
The number of strokes the pressbeam has executed.

9. Manual mode

9.1. 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.



9.a

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

Function keys

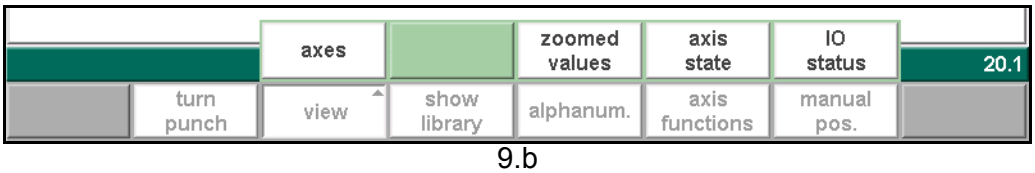
S2 **turn punch / turn die** Turn around the applied tool (back to front). Only available if the cursor is placed on a tool parameter.

S3 **view** Button to select one of the possible viewing modes:
- axes
- aux. axes
- zoomed values
- axis state
- IO status

Function keys

S4	**show library**	Open a window with an overview of the tool library. From there, a tool can be selected.
S5	**alphanum.**	Select an alphanumeric character after typing. The possible characters are displayed on the frontpanel key. Only possible at relevant parameters.
S6	**axis functions**	To change speed and retract distance of available axes in the current bend.
S7	**manual pos**	Start the function to move axes manually with the handwheel. See section 9.2.

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



9.b

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

The name (ID) of the applied punch.

Press the key ⍓ to select the required setting.

Die . . . .UN

The name (ID) of the applied die.

Press the key ⍓ to select the required setting.

Use adapters . . . .UA

Program whether or not a tool adapter is used for this product. The use of an adapter affects the calculated Y-axis position for all bends. Whether an adapter can be programmed depends on the parameter ‘enable adapters’ in the programming constants.

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 pre-programmed 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 sheet.

Bending length . . . .BL

Length of the sheet between the tools.

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 4 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

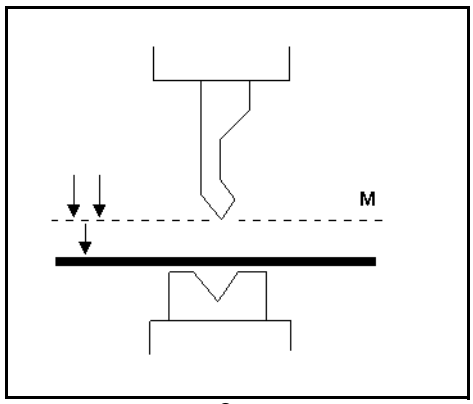
Angle to bend.

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.



9.c

Force . . . . . .P

Max. adjusted force during pressing.

X-Axis. X

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

Retract . . . . . .DX

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

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.

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 for the product you can program a small positive or a negative value.

Speed . . . . . .S

Pressing speed

Decompression. . . . . .DC

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

Decomp speed . . . . . .BS

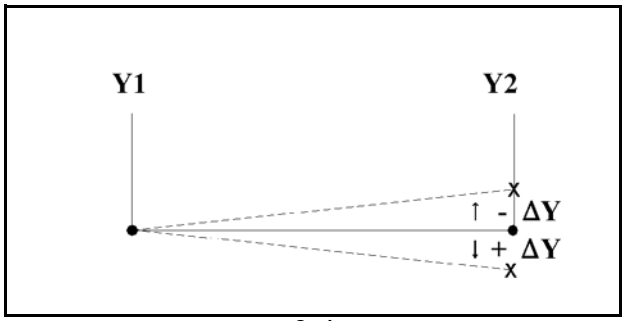
The decompression pressure speed is the programmable speed of the beam during the decompression stroke.

Hold time . . . .T

Hold 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.



9.d

G-corr. PS . . . .Gp=

This is a general correction on the angle position of the part support. It is valid for all bends.

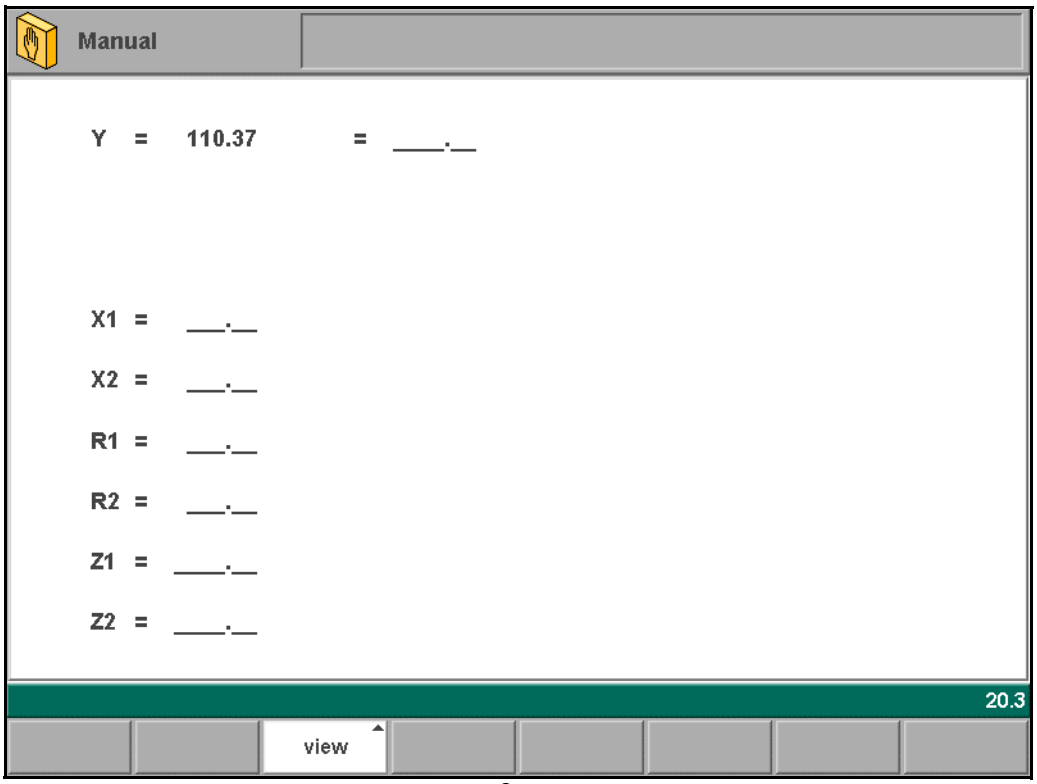
The correction is programmed here as an angle value, relative to the part support angle of the current bend. The correction is remembered as a percentage of the current angle. If another bend is programmed or selected with a different angle, the correction value will be adjusted accordingly.

Only available if a part support is present.

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

- **Zoom function**

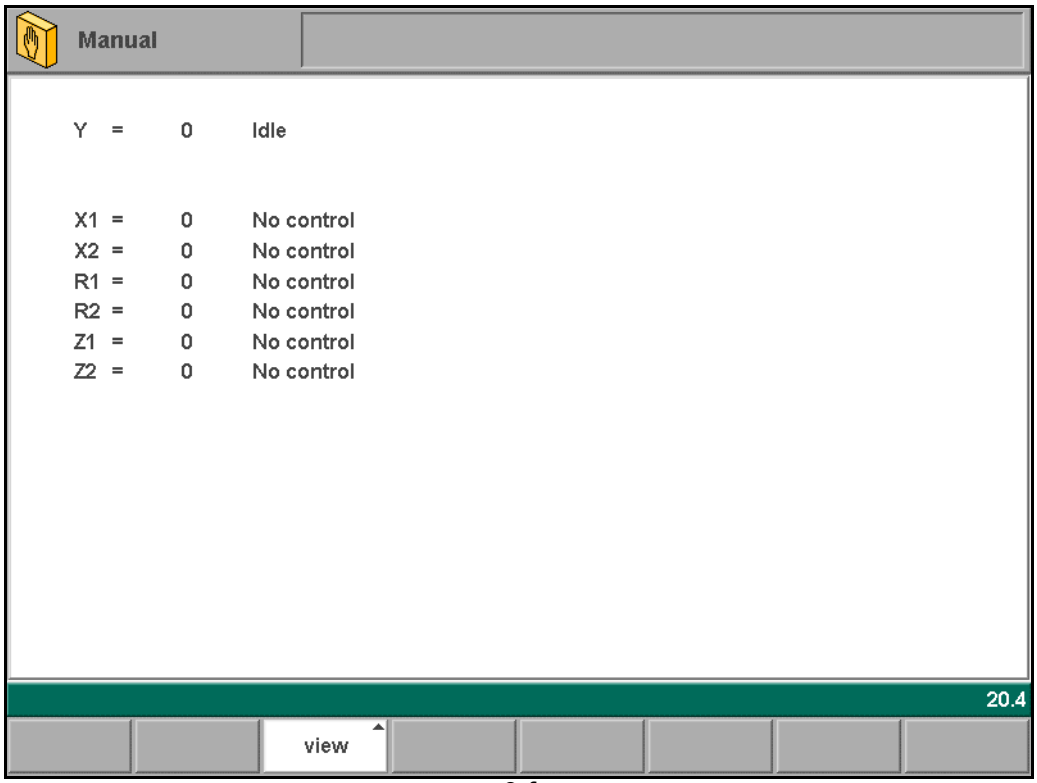
When the function key ‘zoomed values’ has been pressed, the control switches to a new view with only axes values on the screen.



9.e

- **Axis state**

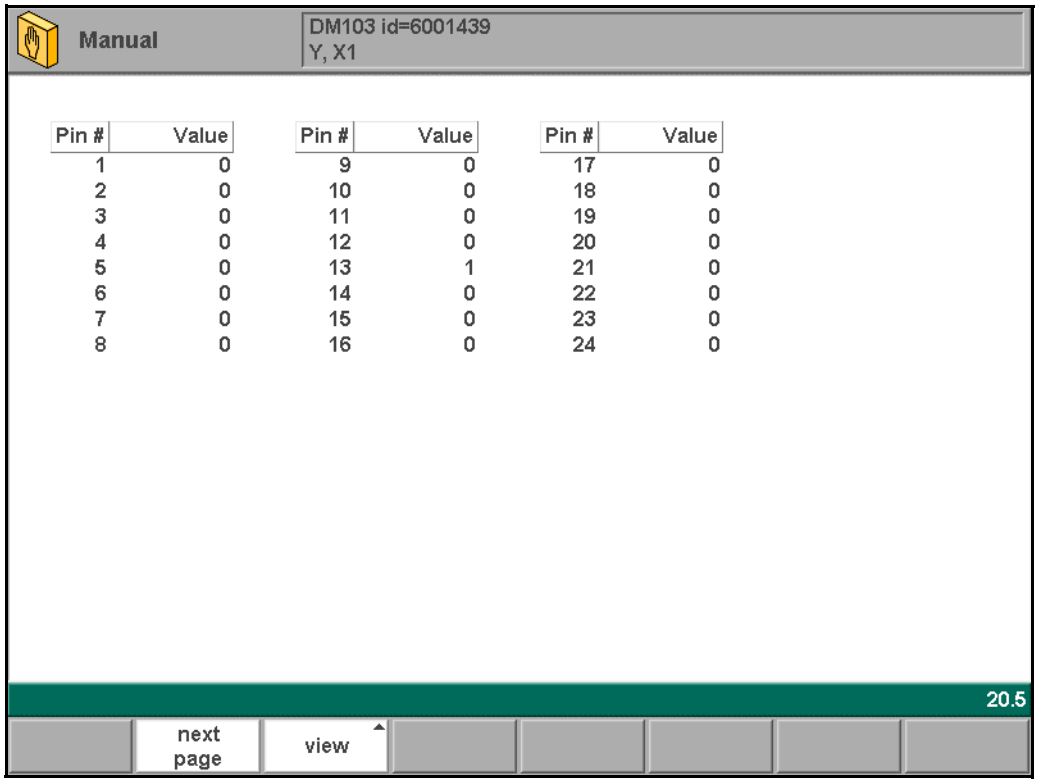
When the function key ‘axis state’ has been pressed, the control switches to a new view with axes states. In this window, the current state of available axes can be observed.
This screen can also be active while the control is started. As such, it can be used to monitor the control behaviour during a bend cycle.



9.f

- **IO status**

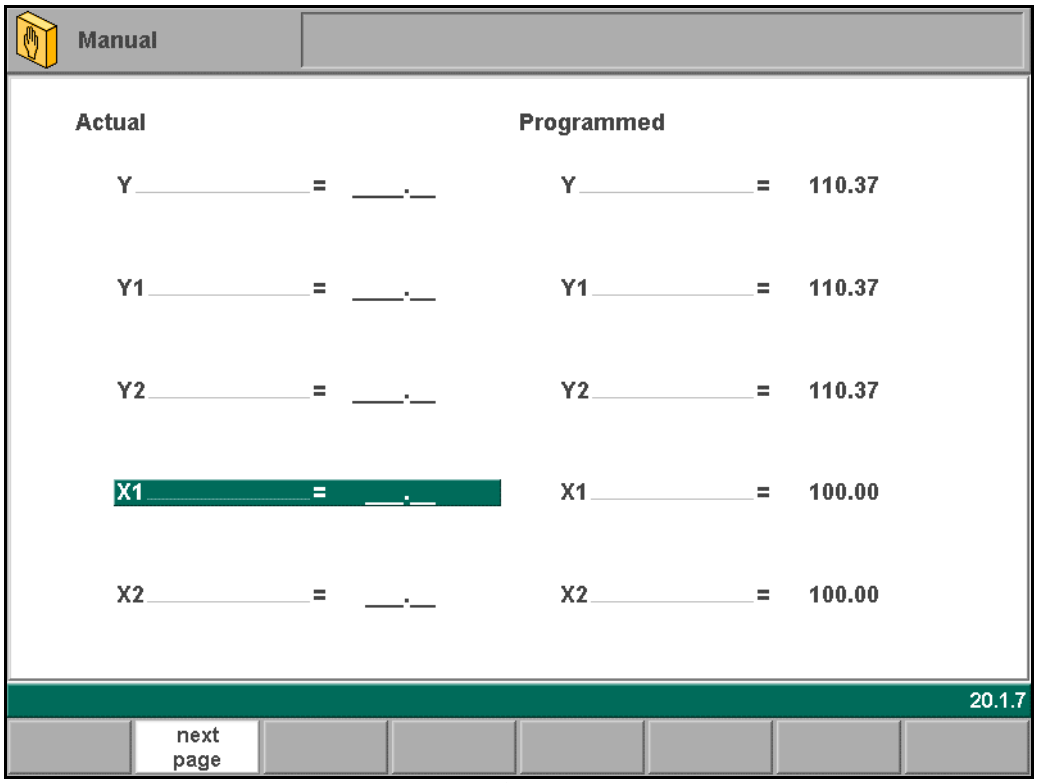
When the function key ‘IO status’ has been pressed, the control switches to a new view with the state of inputs and outputs. In this window, the current state of inputs and outputs can be observed.
This screen can also be active while the control is started. As such, it can be used to monitor the control behaviour during a bend cycle.



9.2. Manual operation of the axes

9.2.1. Movement procedure

It is possible to move an axis by turning the hand wheel 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:



9.g

Within this mode, any of the shown axes can be moved by turning the hand wheel. The procedure for moving the axis depends on the axis you wish to move.

- Auxiliary axes:

The control must be stopped (the LED on the ‘Stop’ button is on).
First select the desired back gauge axis with the arrow keys, 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:

The pressbeam can be positioned manually in the same way as the auxiliary axes. However, for the Y-axis several conditions must be met:

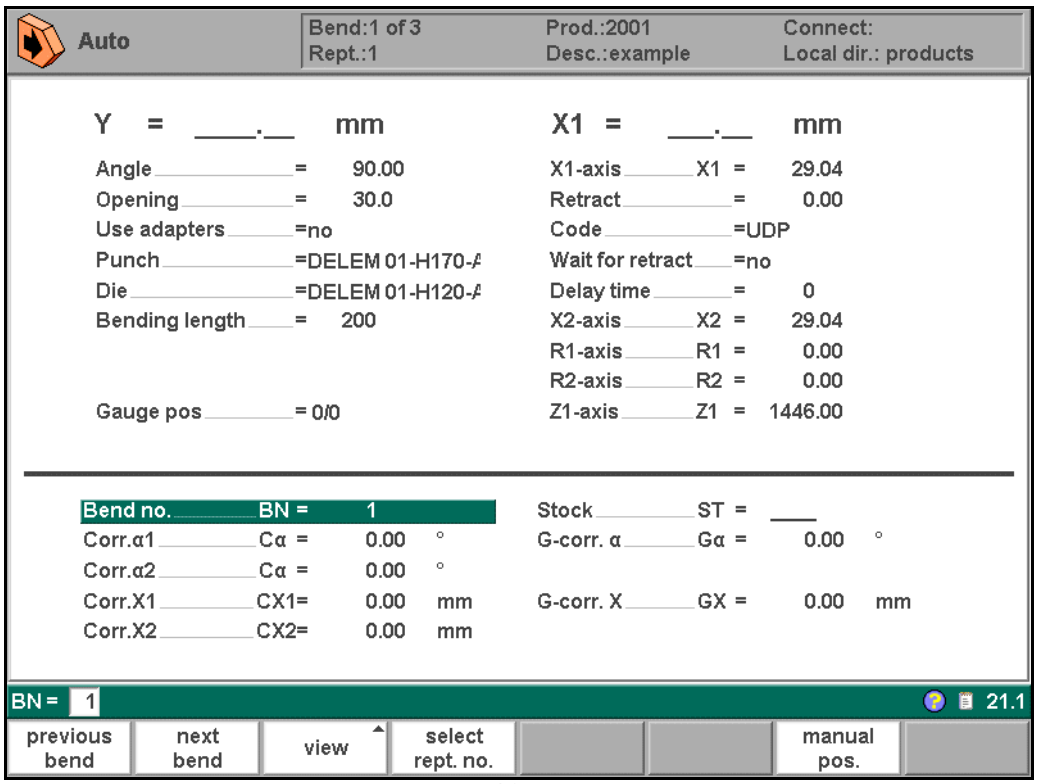
- The control must be started (the LED on the ‘Start’ button is on).
- The ‘adjust’ function must be active, indicated on the screen by "Adjust" in the lower right-hand corner.
- The Y-axis must be below mute-point.
- A pressing command must be given to the CNC.

9.2.2. 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 hand wheel, 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). When you return to the standard screen of manual mode, the axis parameter will have the recently taught value.

10. Automatic / step by step mode

10.1. Introduction



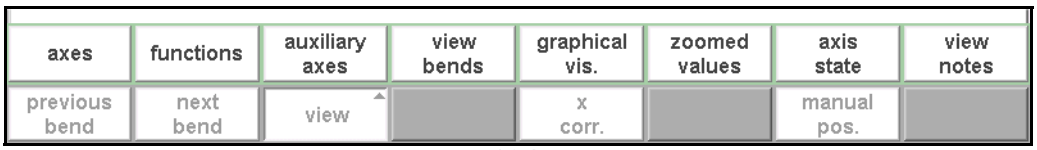
10.a

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 ID, the product description 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	**alpha corr. / X 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 handwheel 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 function keys:



10.b

Function keys:

S1	**axes**	Switch to default view of automatic mode.
S2	**functions**	Switch to view with additional bend parameters.
S3	**auxiliary axes**	Switch to second view with parameters of auxiliary axes.

Function keys:

S4	**view bends**	Switch to a table view in which all bends are shown.
S5	**graphical vis.**	Switch to graphical visualisation of the program.
S6	**zoomed values**	Switch to view with axes values.
S7	**axis state**	Switch to view with state of the available axes. Available for service purposes.
S8	**view notes**	Open a window with notes about the current product.

- **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 parallelism 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.Y . . . . . .CY=
Correction on the Y-axis position, in case absolute programming is used or bottoming is selected for a bend.

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 millimetres.
Measured value of 202 millimetres.
Then it is required to program Corr.x with -2
2) Programmed value of 200 millimetres.
Measured value of 198 millimetres.
Then it is required to program Corr.x with +2

Stock . . . . . .ST =
The stock counter is increment 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.

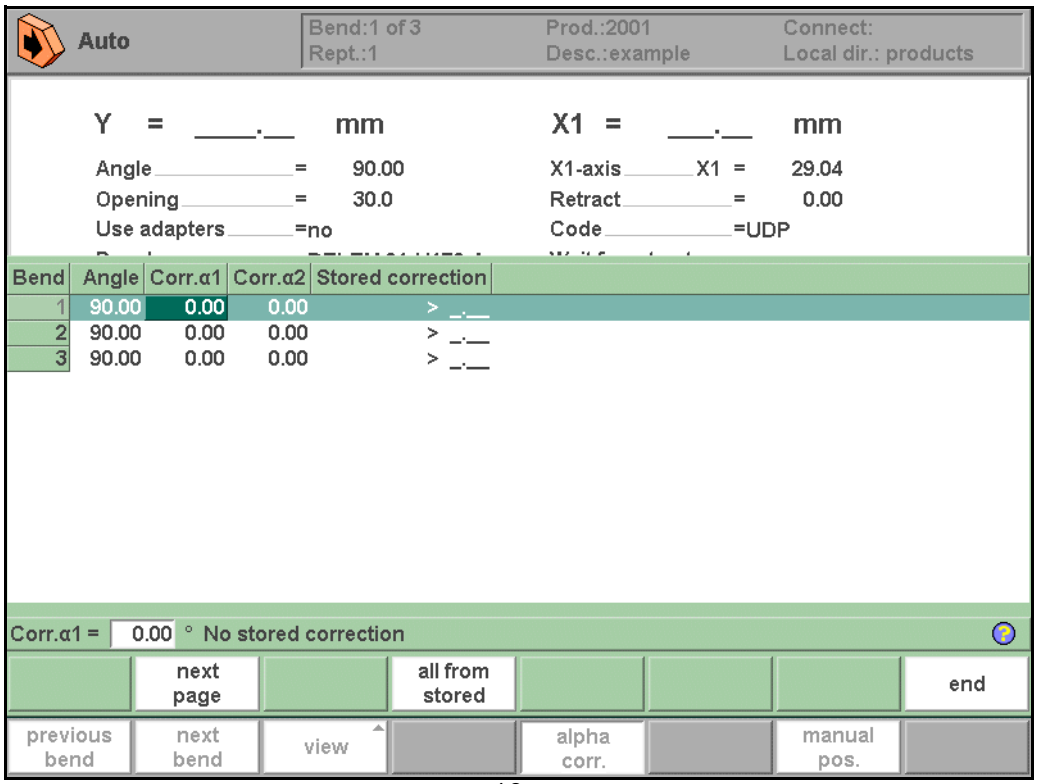
G-Corr.Y . . . . . .CY=
General correction on the Y-axis position, in case absolute programming is used or bottoming is selected for a bend. This correction is valid for each bend of the program.

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.

G-corr. PS . . . . . .Gp=
This is a general correction on the angle position of the part support. It is valid for all bends.
The correction is programmed here as an angle value, relative to the part support angle of the current bend. The correction is remembered as a percentage of the current angle. If another bend is programmed or selected with a different angle, the correction value will be adjusted accordingly.
Only available if a part support is present.

• Angle corrections

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

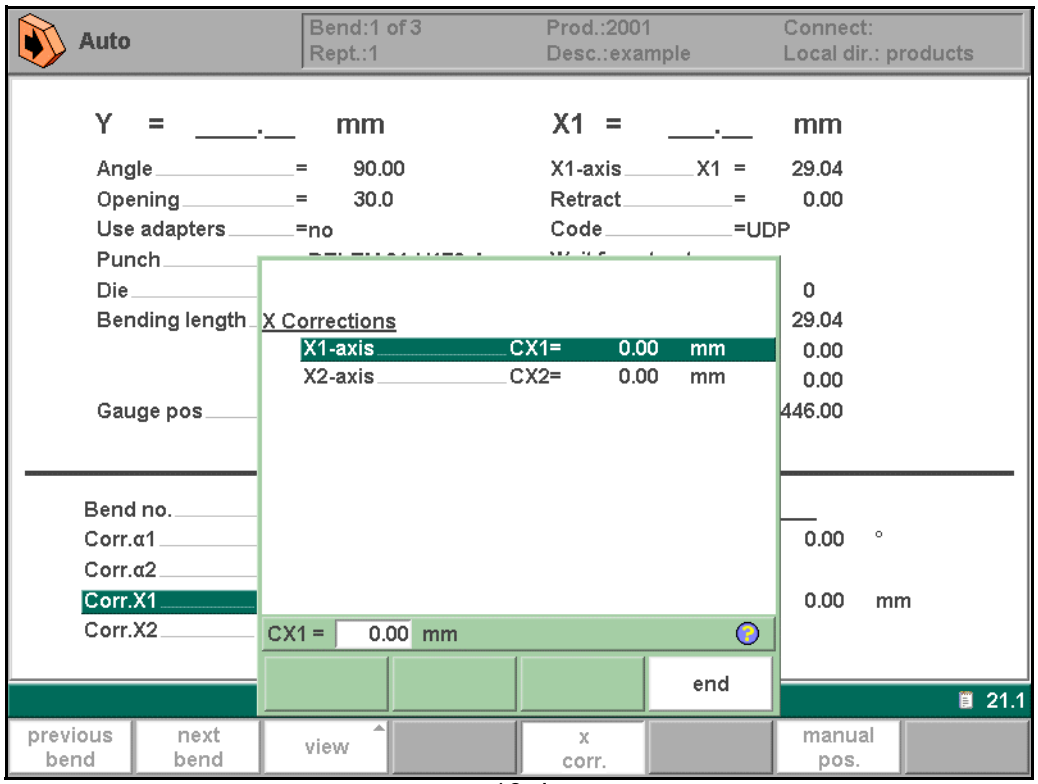


10.c

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 $\alpha 1$ is entered then this value is copied to the correction for $\alpha 2$. Different corrections for $\alpha 2$ can be entered in the field itself.
The columns 'measured $\alpha 1/2$' are present if they are enabled through the parameter 'angle correction computation' in the programming constants.
In the columns 'measured $\alpha 1/2$' 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.

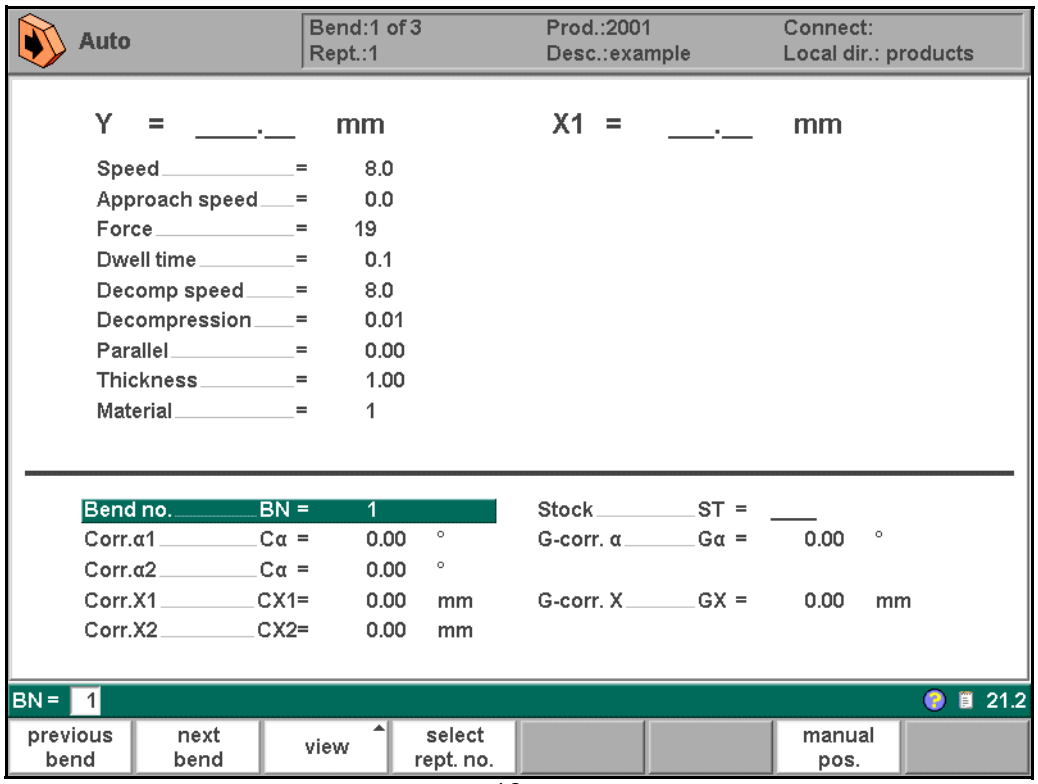
- **X corrections**

X-axis corrections can be edited in a separate window. This window can be activated by pressing the function key ‘X corr.’ in the main screen. The following window appears:



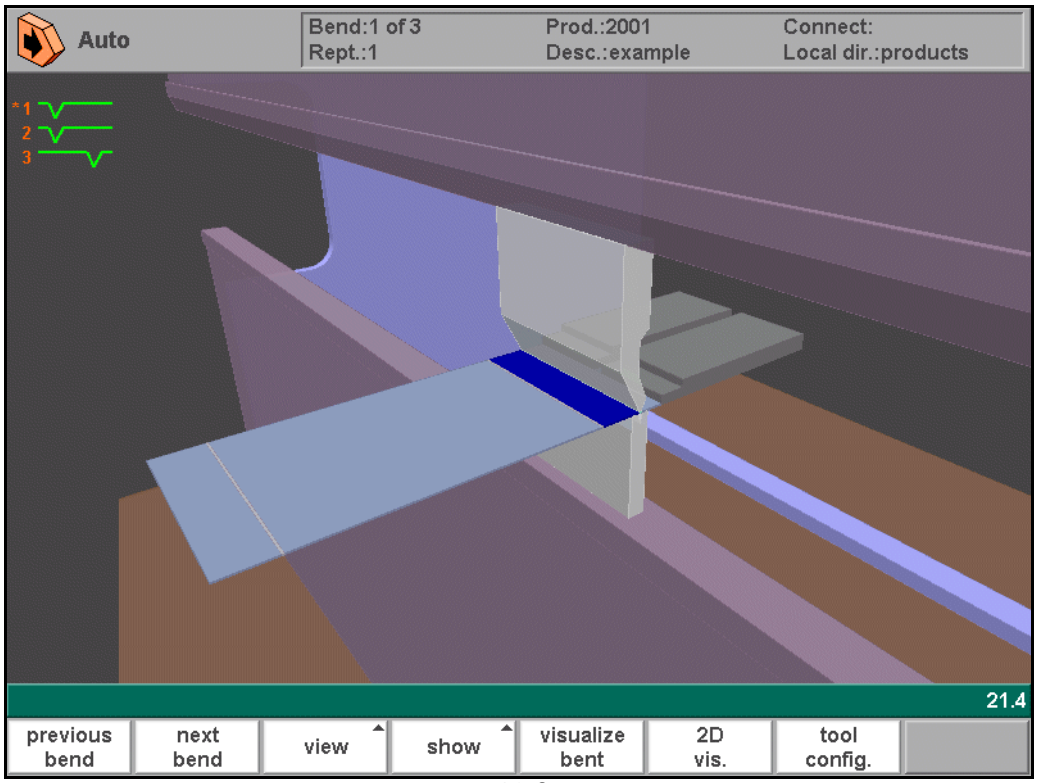
10.d

- **Functions screen**



10.e

- **Graphical visualisation**

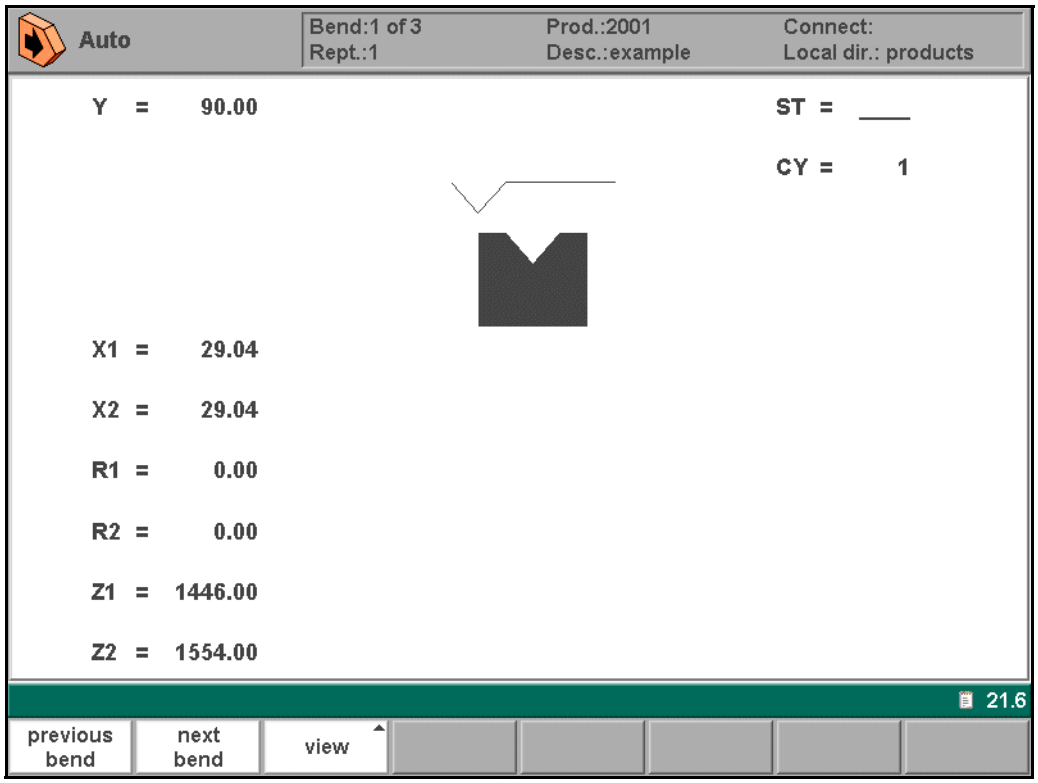


10.f

Function keys:

S4	**show**	Toggle between some possible ways of displaying the product/tool configuration: - product/tools - all
S5	**visualise bent / visualise unbent**	Visualise the product folded or unfolded.
S6	**2D vis. / 3D vis.**	Switch between 2D or 3D visualisation.
S7	**tool config.**	Open a window with the tool configuration. The configuration cannot be edited.

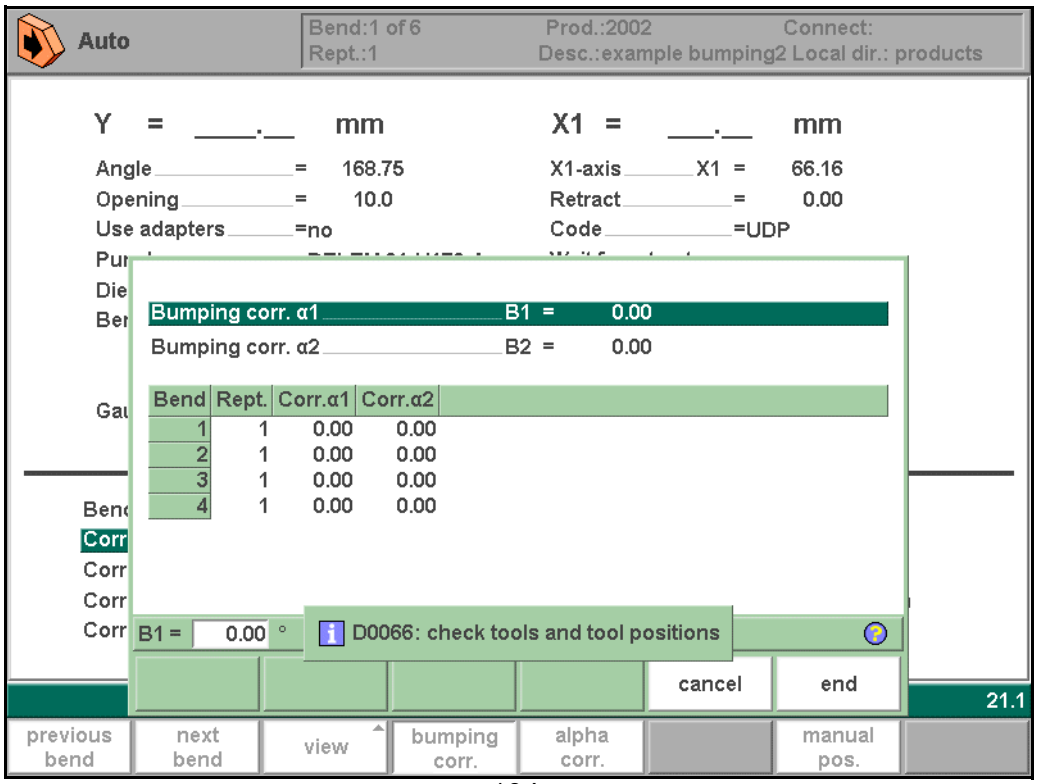
- **Zoomed values**



10.g

- **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.

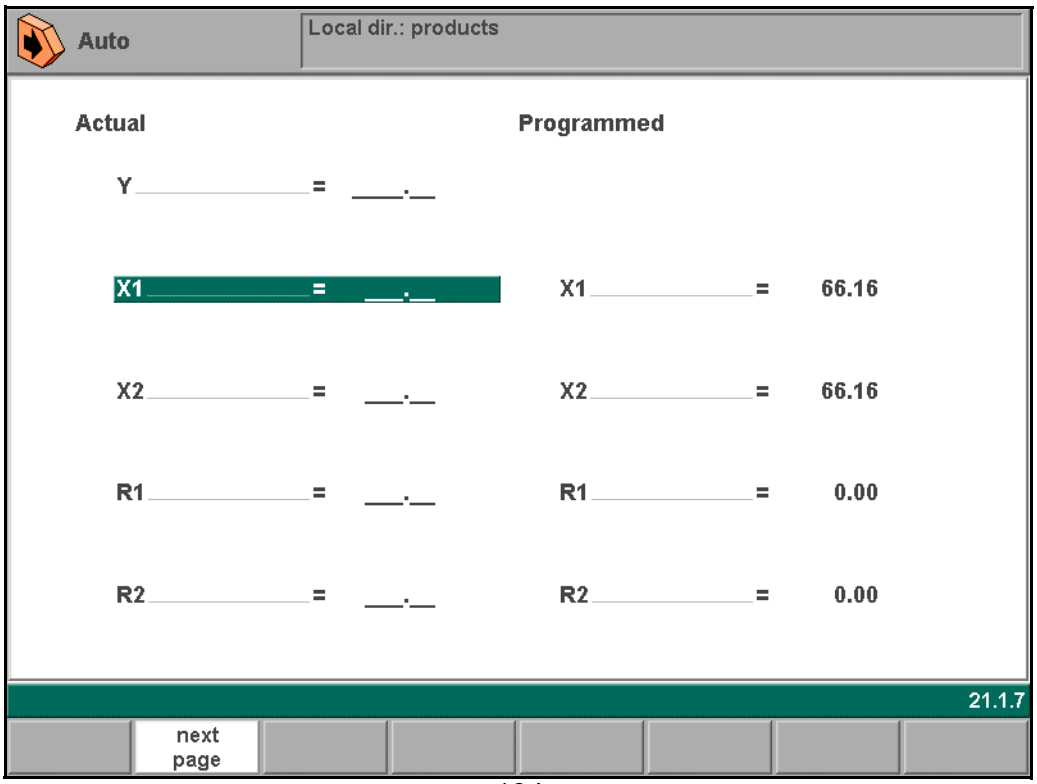


10.h

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 corrections of α2.

• Manual positioning

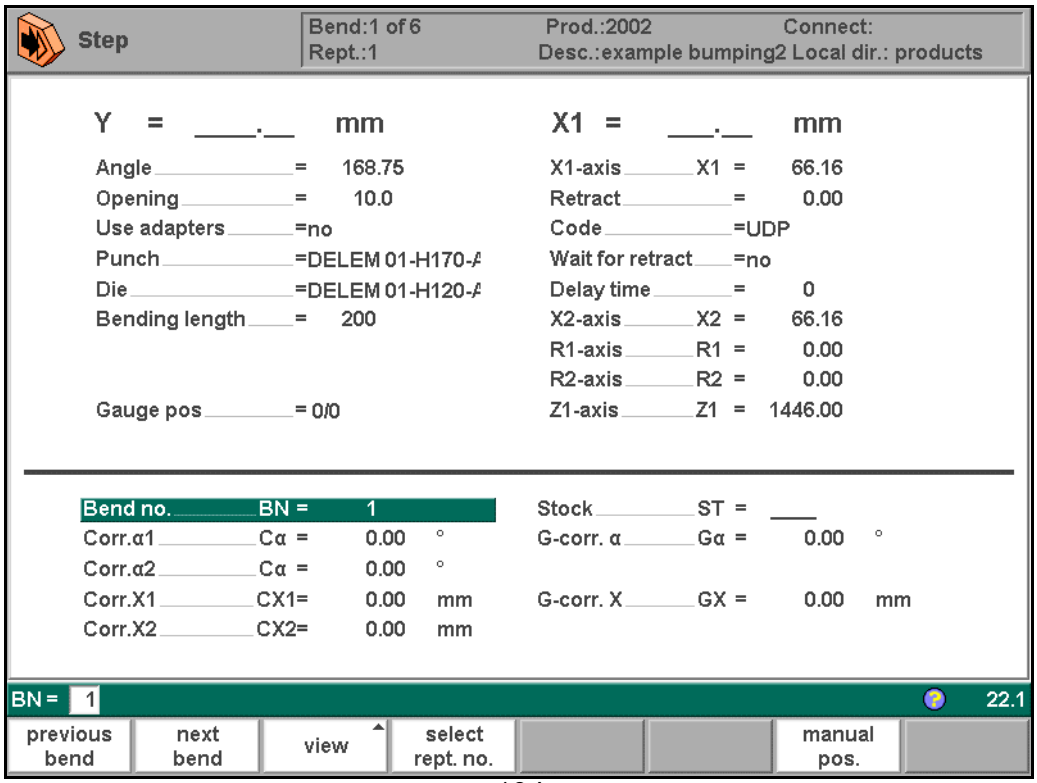
In the automatic production mode it is possible to move the selected axis manually with the hand wheel and change the axis-positions with the teach function like in manual mode.



10.i

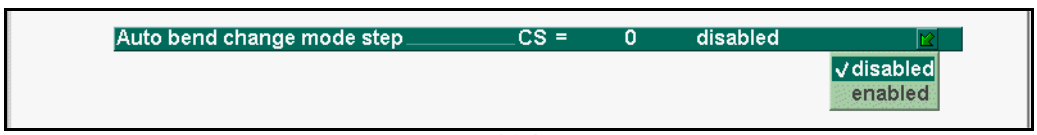
10.2. Step mode

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.



10.j

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:



10.k

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.