

DA-53T

**Reference Manual
Operation of Version 1.5
English**

Preface

This manual describes the operation of the Delem control type DA-53T 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.
- All rights reserved. The copyright is held by Delem. No part of this publication may be copied or reproduced without written permission from Delem BV.

Version history

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

Software version	Manual version	Description
V1.5	V0817	first issue V1

This manual is valid for software version 1.5 and higher.

Table of contents

1. Operation overview and general introduction	1.1
1.1. The control unit	1.1
1.2. Front control elements	1.2
1.3. USB connectors	1.3
1.4. Operation and programming modes	1.4
1.5. Getting started	1.7
1.5.1. Introduction	1.7
1.5.2. Preparations	1.7
1.5.3. Modifying a program	1.7
1.5.4. The Auto menu and Manual menu, production modes	1.7
1.5.5. Back-up data, external storage	1.8
1.6. Programming aids	1.9
1.6.1. Listbox functionality	1.9
1.6.2. Parameter zoom functionality	1.10
1.6.3. Navigation	1.11
1.6.4. Text input and editing	1.11
1.6.5. Typing alphanumeric characters vs. special characters	1.12
1.6.6. Messages centre	1.13
1.6.7. Keylock function	1.13
1.6.8. Manual positioning	1.15
1.6.9. Software versions	1.16
2. Products, the product library	2.1
2.1. Introduction	2.1
2.1.1. The main view	2.1
2.1.2. Product selection	2.2
2.1.3. New Program, starting a numerical program	2.3
2.1.4. Edit, Copying and Deleting a product or program	2.4
2.1.5. Product Rename	2.5
3. Tool configuration	3.1
3.1. Introduction	3.1
3.2. Standard procedure	3.1
3.3. Tool selection	3.2
4. Product programming	4.1
4.1. Introduction	4.1
4.2. Program mode, parameter explanation	4.3
4.2.1. Bend parameters	4.3
4.2.2. Force	4.5
4.2.3. Speed	4.6
4.2.4. Functions	4.6
4.2.5. Product properties	4.7
4.2.6. Tools	4.7
4.2.7. Auxiliary axes	4.7
4.3. Edit / view modes	4.8
4.3.1. All Bends	4.8
4.3.2. Change tools	4.10
4.3.3. Product properties	4.11
4.3.4. Add bend	4.14
4.3.5. Bumping	4.15
4.4. Programming parameters	4.17

5. Automatic mode	5.1
5.1. Introduction	5.1
5.1.1. Auto mode, parameter explanation	5.2
Corrections	5.2
General corrections	5.3
General	5.3
5.2. View modes	5.4
5.2.1. Main	5.5
Bend selector	5.5
5.2.2. All bends	5.6
5.2.3. Macro	5.7
5.2.4. Manual positioning	5.8
5.2.5. Corrections	5.9
Calculate corrections, programming of measured angles	5.10
5.2.6. Diagnostics	5.11
5.3. Bumping correction	5.12
6. Manual mode	6.1
6.1. Introduction	6.1
6.1.1. Manual mode, parameter explanation	6.2
Bend parameters	6.2
Force	6.4
Speed	6.4
Functions	6.4
Product properties	6.4
Tools	6.5
Auxiliary axes	6.5
6.1.2. Tool setup	6.6
6.2. Programming parameters & Views	6.7
View	6.7
6.3. Macro	6.9
6.4. Manual movement of the axes	6.10
6.4.1. Movement procedure	6.10
6.4.2. Teach	6.11
6.5. Corrections	6.12
6.6. Diagnostics	6.13
6.6.1. IO status	6.14
Zoomed IO	6.15
7. Settings	7.1
7.1. Introduction	7.1
7.2. General	7.2
7.3. Materials	7.4
7.4. Backup / restore	7.7
7.4.1. Product backup	7.8
7.4.2. Product restore	7.10
7.4.3. Tool backup	7.12
7.4.4. Tool restore	7.13
7.4.5. Backup and restore for Tables and Settings	7.13
7.4.6. Directory navigation	7.14
7.5. Program settings	7.15
7.6. Default values	7.18
7.7. Computation settings	7.20
7.8. Production settings	7.22
7.9. Time settings	7.25

8. Machine	8.1
8.1. Introduction	8.1
8.2. Programming of Punches	8.2
8.2.1. Create a new punch	8.3
8.2.2. Standard punch	8.4
8.2.3. Hem bend punch	8.7
8.2.4. Air + hem bend punch	8.8
8.2.5. Big radius punch	8.10
8.3. Programming of bottom dies	8.11
8.3.1. Create a new die	8.12
8.3.2. Standard die	8.13
8.3.3. Hem bend die	8.16
8.3.4. Inside hem bend die	8.17
8.3.5. Air + hem bend U die	8.20
8.4. Backgauge	8.21
8.5. Position corrections	8.24
8.6. Maintenance	8.25
8.7. System information	8.26
A. Parameter index	A.1

1. Operation overview and general introduction

1.1. The control unit

The control looks as follows:



The precise appearance of your control may differ.

Operation of the control is mainly done over the touchscreen. A description of the functions and available touch controls is given in the next sections of this manual, aside of the description of the specific functions.

1.2. Front control elements

The Start and Stop button, integrated in the touch screen user interface:



Stop button + Start button

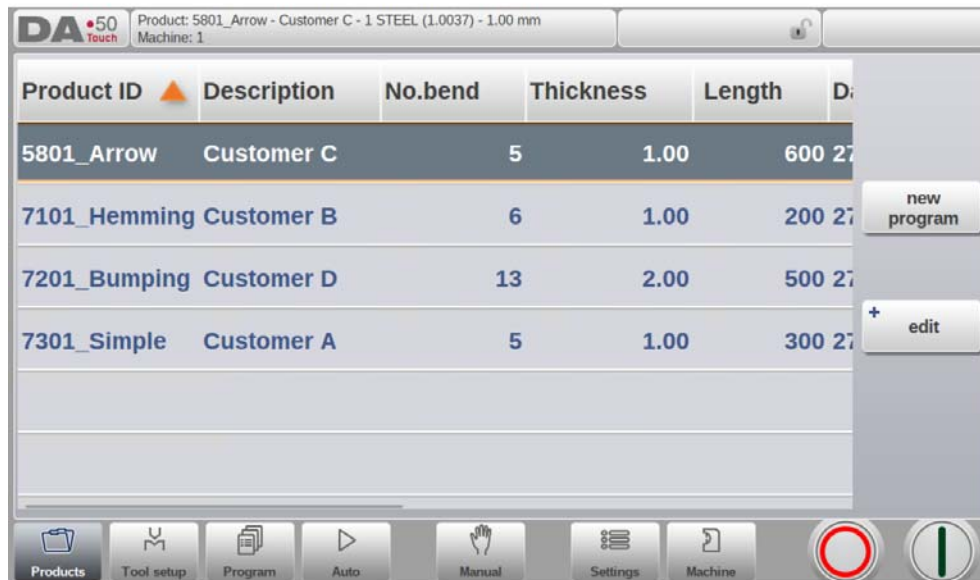
1.3. USB connectors



At the right side of the control a USB port is available for connection of external devices, such as a memory stick or an external keyboard or mouse.

1.4. Operation and programming modes

The control's main screen looks as follows:



Depending on the navigation button which is active, the screen will differ. The above main screen will appear having the Products function active.

Just by tapping the various modes, the specific mode will be selected.

The structure of the main screen is as follows:

Title panel

In the top the title panel is always shown. In this area you can find logo information, which product is loaded and (when activated) the service row. Also machine indicators can be found here.



Information panel

In the information panel all functions and visualisation related to the selected modus are displayed and can be found.

Product ID ▲	Description	No.bend	Thickness	Length	D
5801_Arrow	Customer C	5	1.00	600 27	
7101_Hemming	Customer B	6	1.00	200 27	
7201_Bumping	Customer D	13	2.00	500 27	
7301_Simple	Customer A	5	1.00	300 27	

new program
+ edit

Command panel

The command panel is part of the Information panel and is the location where the controls related to the Information panel can be found.

Navigation panel

The Navigation panel is the area where all the major modes can be found. This area is always visible. The controls, large buttons with icons, can be used to directly switch from one mode to the other.



Explanation of the main modes / navigation buttons



To make a new program and select a program out of the product library.



To setup the machine and modify existing tool setups.



To create a new CNC-program or edit an existing CNC- program numerically.



To start production of the selected program.



To program all settings for making one single bend, not related to a program.



User settings and preferences applicable to the programming of new programs. Also the required material properties can be programmed in the Material library.



User settings and preferences applicable to the machine, tooling library including the tool editor, backup/restore of data, software version info etc.

1.5. Getting started

1.5.1. Introduction

In order to obtain a bend program for a product, the control offers the possibility to create a program bend by bend and adjust specific parameters for each bend independently. This is done with the following steps:

- 1 Go to the Products mode in the navigation panel and start a new program by tapping New Program.
- 2 Enter the program's basic properties and continue by pressing Accept.
- 3 Automatically the next menu is switched to, Tool-setup, to select the desired tooling. Check the tooling, modify upon desire and one can continue to the Program menu.
- 4 In the Program menu one can start programming the bend specific parameters for the first bend. To add the next bendstep to the program the function Add bend can be used. This cycle can be repeated upon choice.
- 5 Tap Auto and press the Start button in order to produce the programmed product.

1.5.2. Preparations

Before product programming can be started, the following preparations must be made.

- The correct material properties must have been programmed in the Materials library. You can find this on the Materials page in the Settings mode.
- The correct tools must be programmed in the Tool Library. Tools are necessary to create a CNC program. You can find the libraries for the different types of tools in the Machine mode.

1.5.3. Modifying a program

The Program menu gives access to the numerical program and values of the active product.

Existing programs can be modified via this menu. The independent bendsteps can be selected and the programmed values can be monitored and modified if required.

Axes positions, if applicable, are calculated according to the machine configuration.

1.5.4. The Auto menu and Manual menu, production modes

A product program can be executed via the Auto mode. In Automatic mode, a complete program can be executed bend after bend. In the Auto mode the Step mode can be selected to have each bend 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 chapters 5 and 6.

1.5.5. Back-up data, external storage

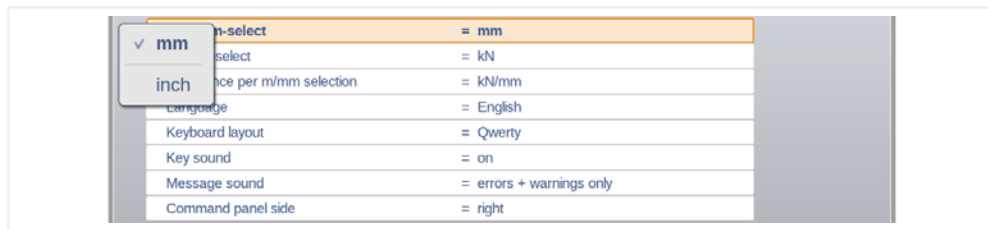
Both product and tool files can be stored externally. These files can be stored on a USB stick. This facilitates a back-up of important data and the possibility to exchange files between Delem controls.

More information about this can be found in chapter 7.

1.6. Programming aids

1.6.1. Listbox functionality

Several parameters on the control have a limited number of possible values. When selecting such a parameter, by tapping the parameter line on the screen, the list of options will open up near the position where you tapped the line, and the desired value can be selected.



To undo the selection and the opened listbox, tapping outside the box will make it close without changing the selected parameter.

1.6.2. Parameter zoom functionality

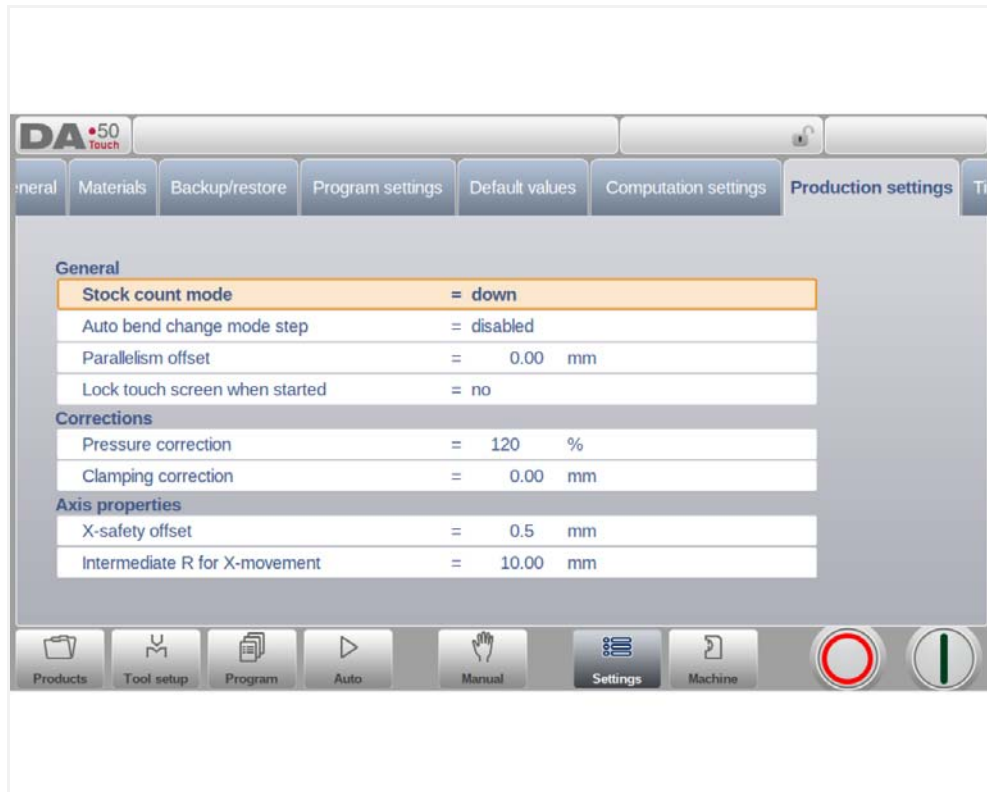
To improve the focus on parameters, and to ease the use while programming, the parameter zoom function will enlarge specific parameterlines when being programmed. When selecting eg. force in Program mode, the force lines will expand giving them better focus while fine tuning.



When selecting any other parameterline, the previous selection will be reduced and zoomed out again, as the newly selected parameterline will be zoomed in at.

1.6.3. Navigation

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



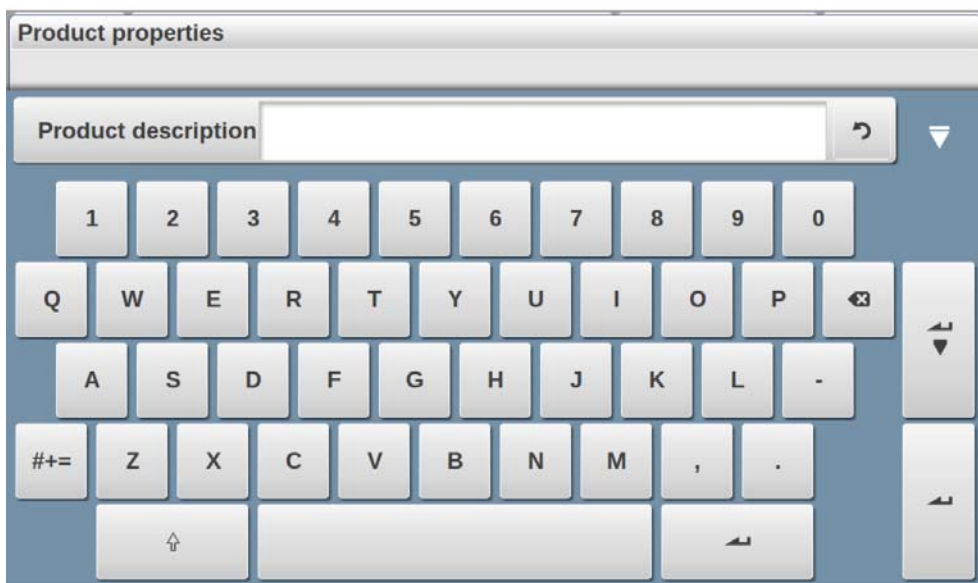
The tabs can easily be selected by just tapping them. When a tab is not completely visible or not visible at all, just by dragging the tab row horizontally, the desired tab can be "pulled" in sight and be selected.

1.6.4. Text input and editing

The cursor can be used to enter a specific value or text within an existing input. Just tap at the desired position to do so. The cursor will appear and input will be added there.

1.6.5. Typing alphanumeric characters vs. special characters

Both alphanumeric characters and special characters can be used throughout the control. A full on-screen alphanumeric keyboard will pop up when required. When editing a field which is pure numeric, the alphanumeric characters will be hidden. For fields which enable to use alphanumeric strings, the keyboard is completely available. Special characters as ? % - can be found using the special character button on the left-lower side of the keyboard.



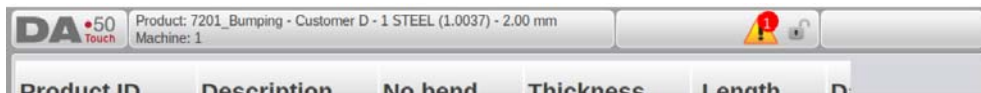
Special characters (like á, à, â, ã, ä, å, æ) are supported by the on screen keyboard by keeping a character (like 'a') pressed.



1.6.6. Messages centre

When messages are displayed coming from PLC, Safety systems or the Sequencer, these messages can be 'send' to the 'Messages centre'. When a message is displayed simultaneously the message centre symbol is shown in the top row of the page header, next to eg. the keylock symbol. When tapping this message centre symbol the messages are taken from the screen, giving way for normal programming and editing. When tapping again the actual messages are shown.

When messages are in the background, the message centre symbol has an extra indicator to show new incoming messages which are not yet shown.



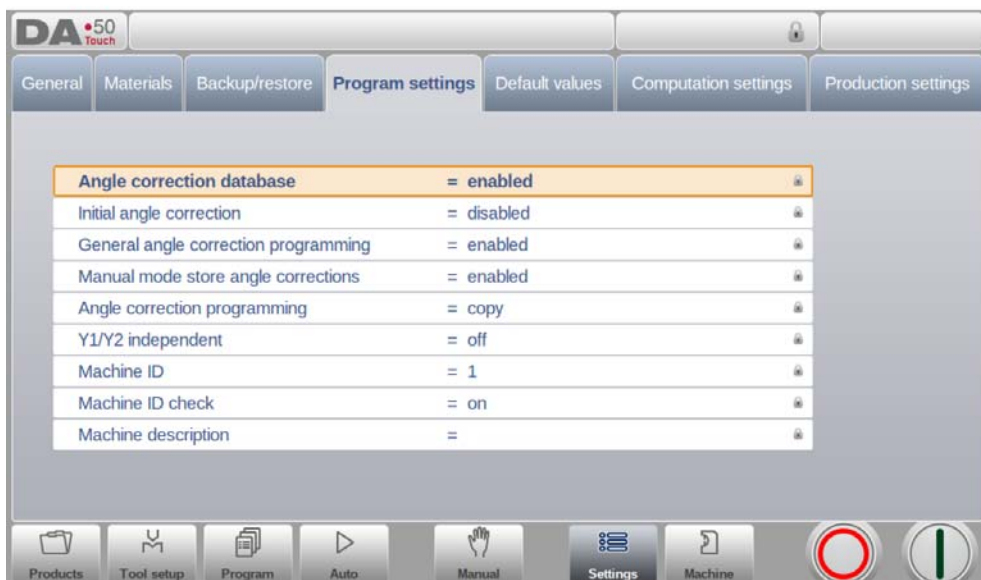
1.6.7. Keylock function

To prevent for changes to products or programs, the keylock function offers the possibility to lock the control.

There are two levels of locking the control. Program Lock and Machine Lock.

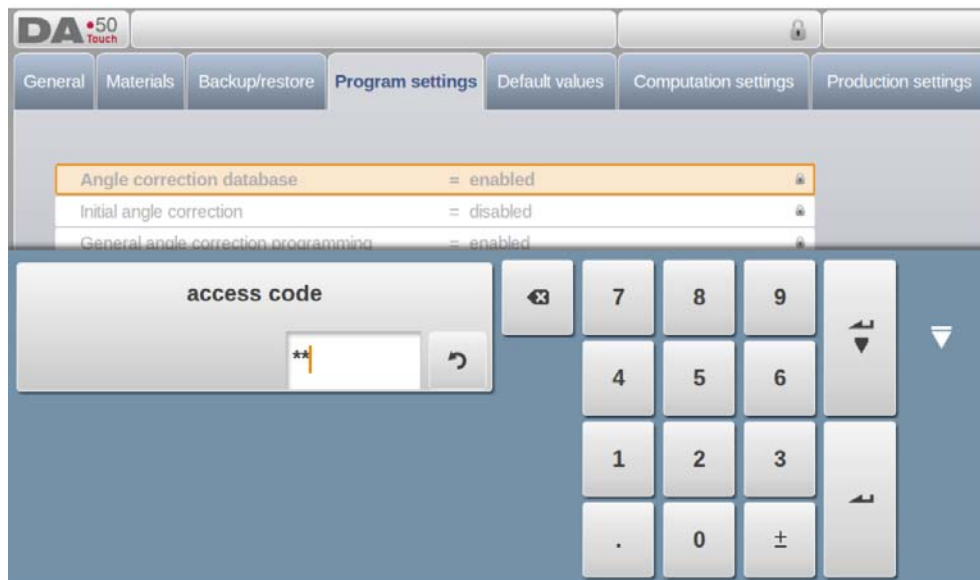
- In Program Lock, only a product can be selected and executed in Automatic mode.
- In Machine Lock, the machine is locked and the control can not be used.

To lock a control just tap the lock symbol in the top of the screen. Depending on the code which is used, the control will be in Program Lock or Machine Lock. Program Lock wil show a closed lock in grey. Machine lock will show the same lock but colored (red).



Lock symbols when Program Lock is active will also appear behind parameters to show the lock is active and modification is not possible.

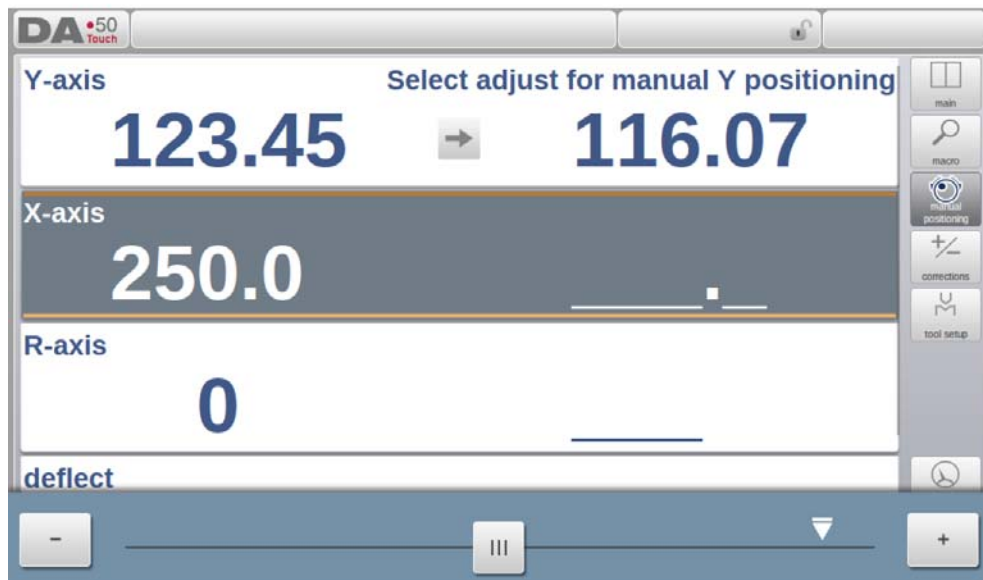
To unlock the control, tap the lock symbol and enter the appropriate code. After entering, the lock symbol will show it is unlocked and the lock symbols behind parameters will disappear.



Codes can be changed upon desire. The procedure to manages codes can be found in the Installation manual.

1.6.8. Manual positioning

On the manual positioning page in Manual mode and Automatic mode a slider at the bottom of the screen can be used to position the axis. The distance moved with the slider determines the speed of the axis. When the slider is released, the axis stops. The buttons at each end of the slider can be used to fine-tune the axis position. When "sliding" the beeper gives feedback that the axis is moving.



1.6.9. Software versions

The version of the software in your control is displayed at the System Information tab in the Machine menu.



Example of version number:

V 1.2.3

V stands for version

V 1.x.x is the major version number

V x.2.x is the minor version number

V x.x.3 is the update version number

The major version number is increased when new major features are added to the software. These software changes require additional introduction and could change the normal working order. The minor version number is increased when new features and enhancements are integrated which do not change the working order. The update version number solely is used for software changes when corrections are needed in the existing software version.

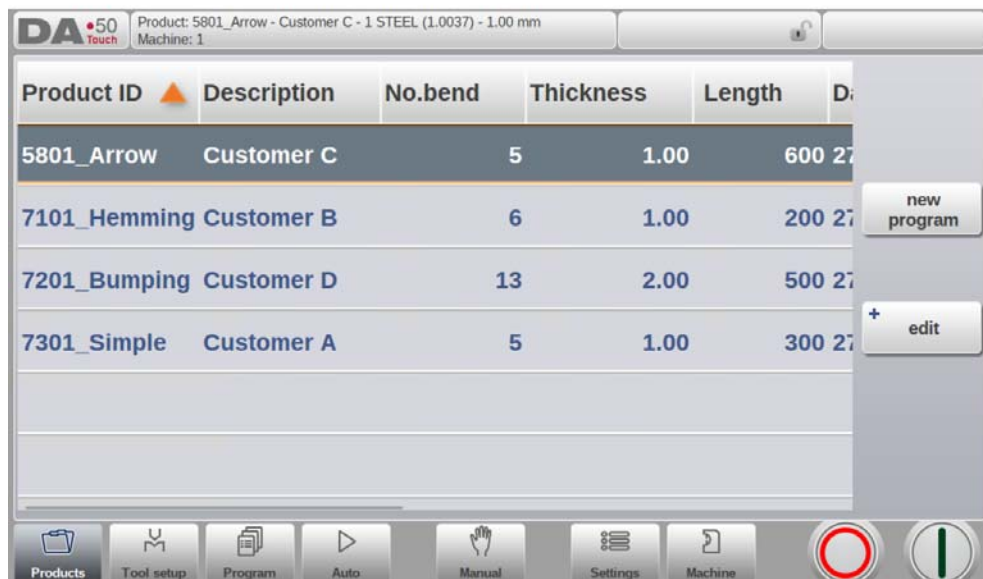
2. Products, the product library

2.1. Introduction



In the **Products mode** existing, previously produced, products can be selected to start production or for modification in order to make a similar product. To start making a new program New Program can be used from this mode.

2.1.1. The main view



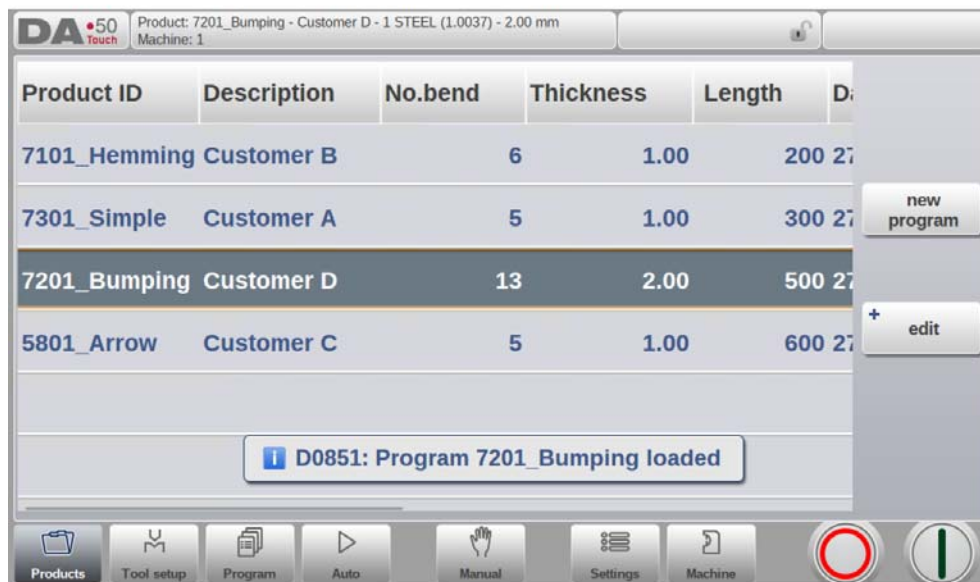
In Products mode an overview is given of the program library on the control. In this mode a product program can be selected (loaded). After that a program can be modified or executed. Each item in the list consists of its Product ID, the Product Description, the Number of bends in the product and the Date it was last used or modified.

If a product program is already active its ID is shown in the top of the screen. A program can be loaded by tapping the product ID or any other part of the product's line.

When there are more products than can be visualised in the screen simply drag the list in the upward direction until the product is visible. From then again a single tap on the product selects the product and activates it in the control.

2.1.2. Product selection

To select a product a single tap will do. The product will be selected and loaded into the memory. From here production can be started by tapping Auto. Also navigation can start through the Tool Setup and the numerical Program.



2.1.3. New Program, starting a numerical program

To start a new numerical program tap New Program.

After New Program is chosen, the programming starts with its general details like e.g. Product ID, Thickness and Material.

DA 50 Touch Product: 7201_Bumping - Customer D - 1 STEEL (1.0037) - 2.00 mm
Machine: 1

Product properties

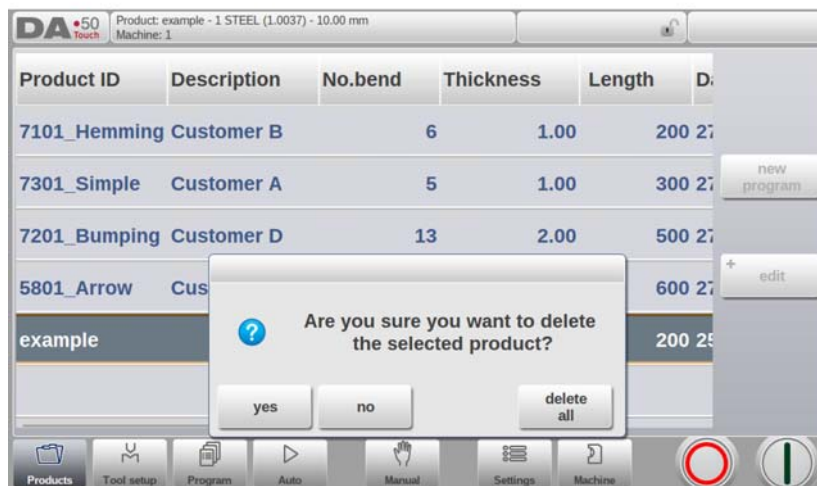
Product	
Product ID	=
Product description	=
Product properties	
Angle sel.	= α
Thickness	= 1.00 mm
Material	= 1 STEEL (1.0037)

connect **cancel**

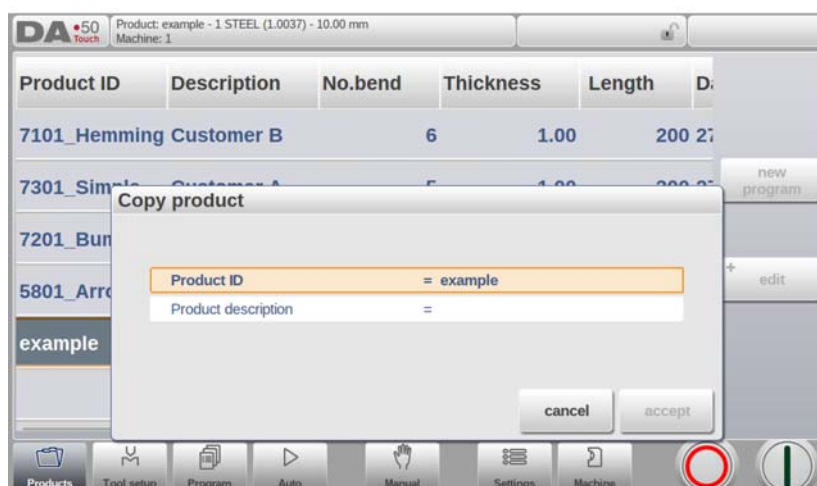
Products Tool setup Program Auto Manual Settings Machine

2.1.4. Edit, Copying and Deleting a product or program

To delete a product in the Products mode select a product by tapping it. It will be selected. After that tap Edit and use Delete. To finally delete it confirm the question. To delete all programs at once, tap Delete All.



To copy a product select a program and tap Edit and use Copy. After this the name of the product can be programmed and the copy will be done. The copied product will be an exact copy including tool setup.



2.1.5. Product Rename

Products can also be renamed. This can be done in one single step: Rename allows the user to give it a new name.

To rename a product select a program and tap Edit and choose Rename from the list. For Rename a new name can be given.

3. Tool configuration

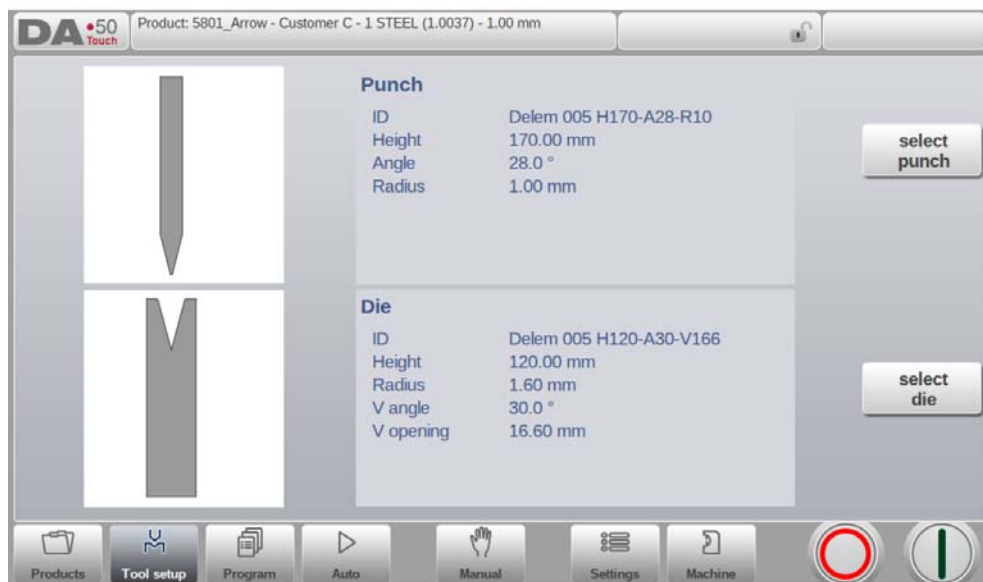
3.1. Introduction



To edit or modify a tool setup for the product, select the product from the library and use Tool Setup

3.2. Standard procedure

When the function Tool Setup has been activated, the screen shows the active machine set-up. Both punch and die can be selected from the tool library.



The Upper and Lower tool, resp. Punch and Die, in the machine are shown and can be changed.

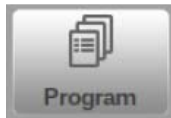
3.3. Tool selection

When selecting tools, both upper and lower tool (resp. punch and die) can be selected from the tool library.

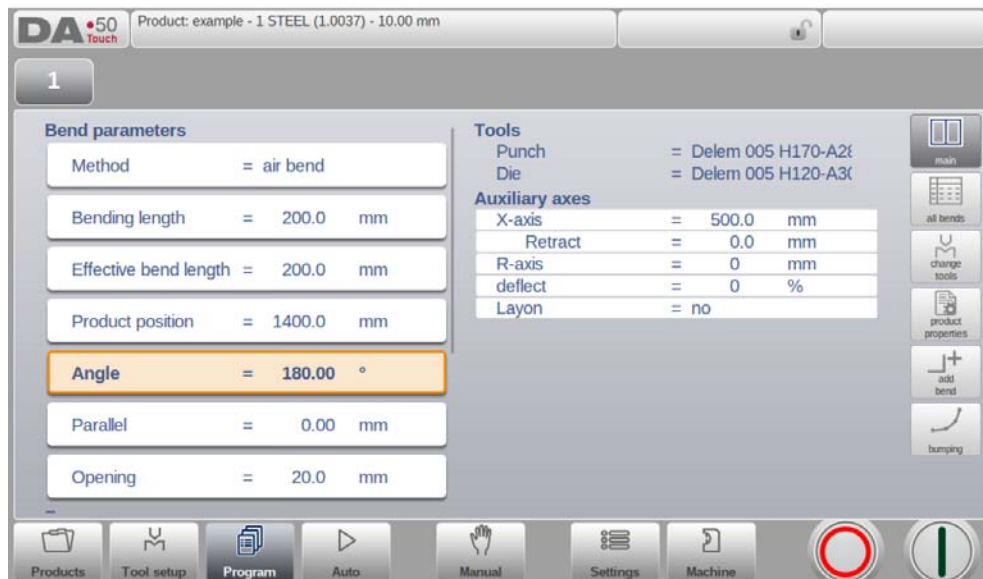
Tap Select Punch or Select Die to change tools to the configuration.

4. Product programming

4.1. Introduction



To generate or modify a numerical program, start a new program from the Products mode or use Program to enter directly



To edit an existing CNC program, select a product in the Products overview and select the navigation button Program. When starting a new program, select New Program and after giving in the main product properties and tool setup, the system will automatically switch to Program.

In both cases, a screen as shown above should appear. Programming and changing data is done in the same way in both cases.

The main screen shows the existing numerical program or, when starting a new program, the first to be programmed bend. The bend selector in the top of the screen can be used to navigate thru the bends. The indicated bends can be tapped to easily select the desired bend data.

At the side of the main screen views and functions are indicated with command buttons.

Functions

Following modes / functions are available:



Main, numerical bend data.



All Bends, shows all bend data in a table.



Change Tools, enables the change of tools for this specific bend step or all bend steps.



Product Properties, gives access to product properties which are general for the product, not bend specific.



Add Bend, adds a new bend to the existing program.

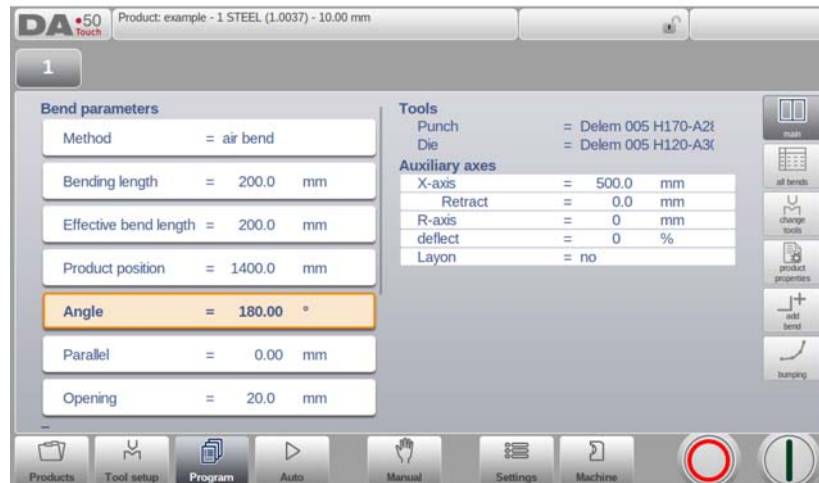


Bumping, changes a single bend step into a bumping bend.

4.2. Program mode, parameter explanation

The main screen shows the available bends and from this main screen, from every available bend, specific parameters can be viewed and edited.

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



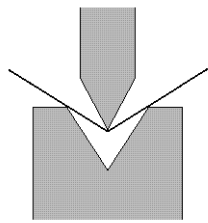
4.2.1. Bend parameters

Method

Select the required bending method. The control supports the following standard methods:

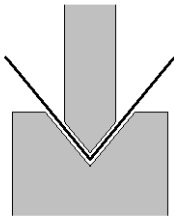
- air bend
- bottoming
- hemming
- hemming & bottoming

Bend methods



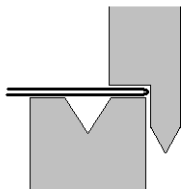
air bend

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

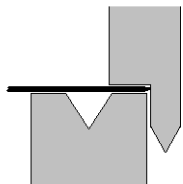
The sheet is bent by pressing the sheet between the punch and the die. The control assumes the bottom of the die as required Y-axis position.



hemming

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 'hem opening'.



hemming & bottoming

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

Note 1

The hemming bends are shown here with a special hemming 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

Length of the sheet between tools.

Effective bend length

The effective length of the sheet between tools, which is used for the calculation of the force and the crowning device (if present). This effective bend length is calculated from the product geometry; for additional bends and simultaneous bends this calculated value is the sum of the separate bend line lengths.

Product position

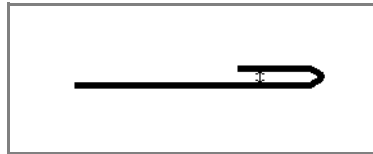
The absolute position value of the product in the Z-direction. Left machine side is reference position zero.

Angle

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.

Hem opening

The hem bend can be made with a certain opening distance between the 2 flanges. The hem opening value will be used calculating the beam position in the hemming process. By default this parameter has the value of the Settings mode parameter Default Hem Opening.



Y-axis (Bend position)

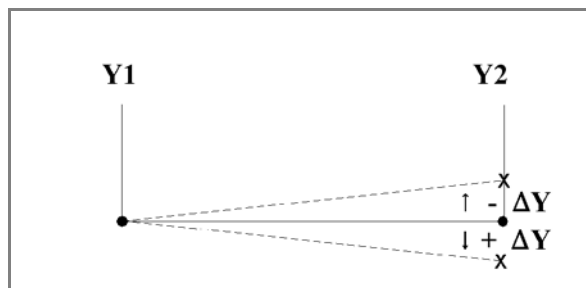
The required Y-axis position for this bend. This parameter only appears if absolute programming is selected with the parameter 'Angle sel.' This parameter also appears if the bend method is bottoming and/or hemming.

Mute

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.

Parallel

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.



Opening

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.

4.2.2. Force

Force

Maximum adjusted force during pressing (automatically computed).

Dwell time

Holding time of punch at bending point.

Decompression

Decompression distance after the bending to release the working pressure from the system.

4.2.3. Speed**Speed**

Working speed (pressing speed). Initially, the value for this parameter is copied from the parameter Default Pressing Speed in the Settings mode.

Decomp speed

The decompression speed is the programmable speed of the beam during the decompression distance.

4.2.4. Functions**Repetition**

0 = bending is skipped

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

Wait for retract

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

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

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.

Code

Programmable step change code 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

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

4.2.5. Product properties

Thickness

The thickness of the sheet.

Material

The material of the product.

4.2.6. Tools

The set tools are displayed and can be modified from the Tool Setup menu. If required the tools per bend can be specifically selected for that bend using Change Tools.

Punch

The name (ID) of the selected punch. Tap Change Tools to modify or select from the punch library.

Die

The name (ID) of the selected die. Tap Change Tools to modify or select from the die library.

Press the Change Tools button to get an overview of the available tools in the library.

4.2.7. Auxiliary axes

Auxiliary axis

The position of the selected axis.

Retract

Retract distance of the selected axis in the current bend. The "backgauge retract" is started when the beam is pinching the sheet.

Speed

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

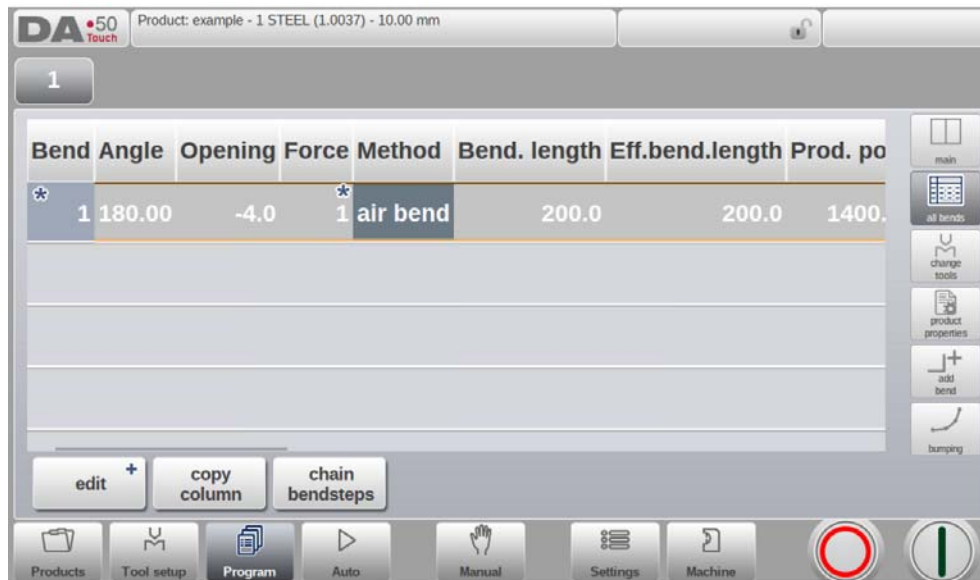
Layon

Toggle between lay surface on finger or not. This option is only selectable in case you have an R-axis in your machine.

4.3. Edit / view modes

4.3.1. All Bends

When the function All Bends has been pressed, a complete overview of the bends appears.



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. The available columns can be scrolled by finger movement / swipe.

Functions

Edit

Edit the program with one of the following commands:

- insert bend
- mark bend
- delete bend

When the function Edit has been pressed a new, temporary button bar appears with additional functions:

+Insert Bend

To insert a new bend between one of the bends. When pressed, the current bend is copied and added before the current bend.

+Mark Bend

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

+Delete Bend

To delete the bend that is currently selected.

When a bend has been marked with the function key Mark Bend several other functions become available:

+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. Select the bend that must be moved. Then press the button Mark Bend and the bend is highlighted. Now select the right place in the sequence. When the correct bend 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 Abort Mark during the procedure.

+Abort Mark

Remove the mark from the currently marked bend.

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

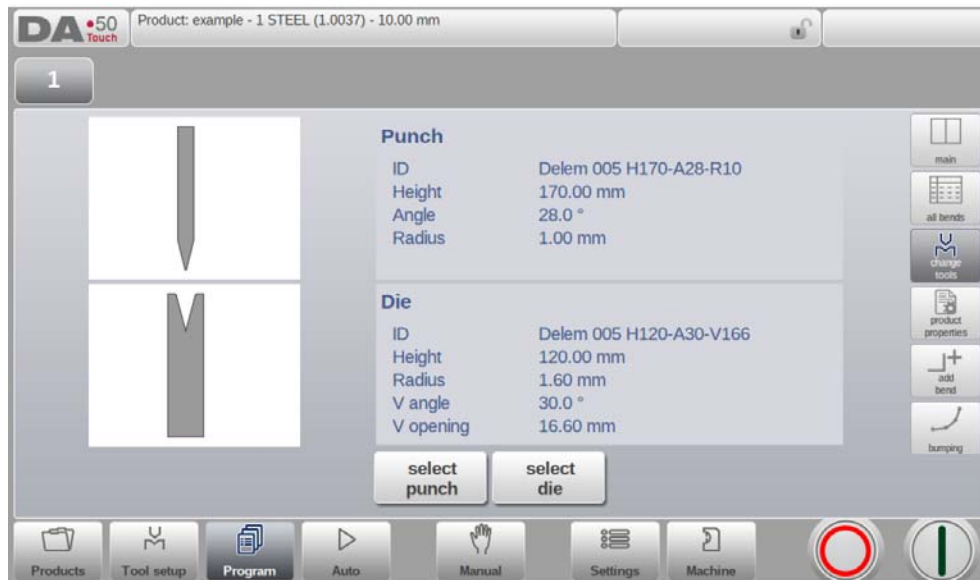
Copy Column

Copy the value of the currently selected parameter in the current bend to all other bends.

Chain Bendsteps

Chaining of bend steps is possible by just setting the number of steps in the chain (Repetition) and the backgauge position offset (X offset). The relative offset value can be positive as well as negative for flexible chaining.

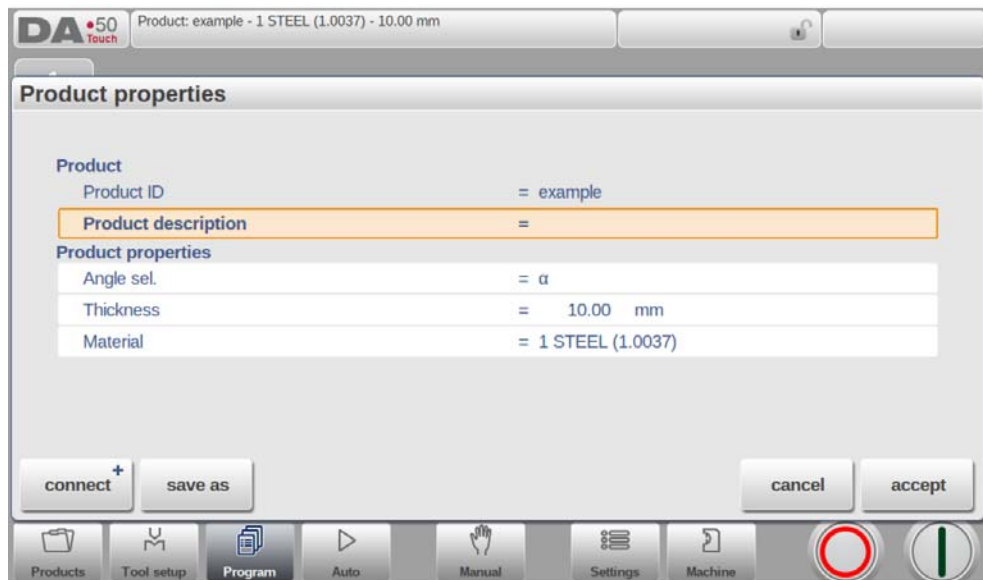
4.3.2. Change tools



To change the tools the Tool Setup menu can be used. If the tool setup needs to be changed for just one bend step, the Change Tools button can be used. The control will always ask if the changes are to be done on the whole setup or just for one bend. If the whole tool setup is required, automatically the Tool Setup menu will be switched to.

4.3.3. Product properties

To change the main product properties tap Product Properties. These parameters of the program are the same for every bend of the program (main data of program).



Parameter explanation

Product ID

A unique name to identify a product program. The maximum length is 25 characters. The product ID may contain letters and numbers.

Product description

A number or description of this program. The maximum length is 25 characters. The product description may contain letters and numbers.

Angle sel.

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.

Thickness

Thickness of the sheet.

Material

Selection of one of the programmed materials, which are used to calculate the bending depths. The control contains 6 pre-programmed materials. In total, 99 materials can be programmed on the control. The materials can be programmed on the Materials page in the Settings mode.

	Tensile strength (N/mm ²)	E-module (N/mm ²)	Strain hardening exponent
1 = Steel	470	210.000	0.23
2 = Aluminium	250	210.000	0.26
3 = Zinc	200	94.000	0.20
4 = Stainless steel	750	210.000	0.32

Functions

Connect

The function Connect is to have a possibility to connect certain programs to each other. This option can be used to produce 3-dimensional products out of two programs.

+Connect program

With the function 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:

- 1 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 the 'product library'.
- 2 Go to the Program mode and select Product properties.
- 3 In the Product properties window you select the function Connect and Connect program.
- 4 Select the product ID of the product in the other direction.
- 5 Select the second program as in step 1. Repeat steps 2 to 4. If you want to connect two programs, as in this example, you enter the product ID 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.

- 1 Select the first program
- 2 Select the Automatic mode
- 3 Program the amount of products you want to produce with the 'stock count'

- parameter.
4 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.

+Disconnect program

To stop the sequence of connected programs.

Save as

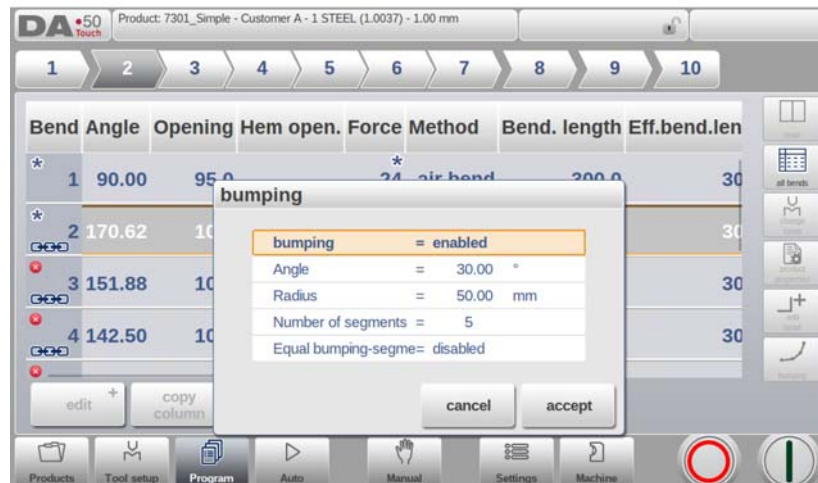
Copy the current product. When pressed, you must enter a new product ID for the copy program.

4.3.4. Add bend

To add a new bend after the last bend. When pressed, the last bend is copied and added after the last bend.

4.3.5. Bumping

From pure numerical programs a single bend step can be changed into a bumping bend.



When Bumping has been selected in the Main view, a pop-up window is shown on which the following parameters can be programmed:

Bumping

- disabled => Air bend as normal bend with a defined angle and preferred radius.
- enabled => Large radius in different steps of airbending.

Angle

The angle value to bend.

Radius

The intended radius which is programmed.

Number of segments

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.

Equal bumping-segments

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.

- Disabled (no equal sizes)

-
- Enabled (equal sizes)

When Enabled all segments will have an equal size.

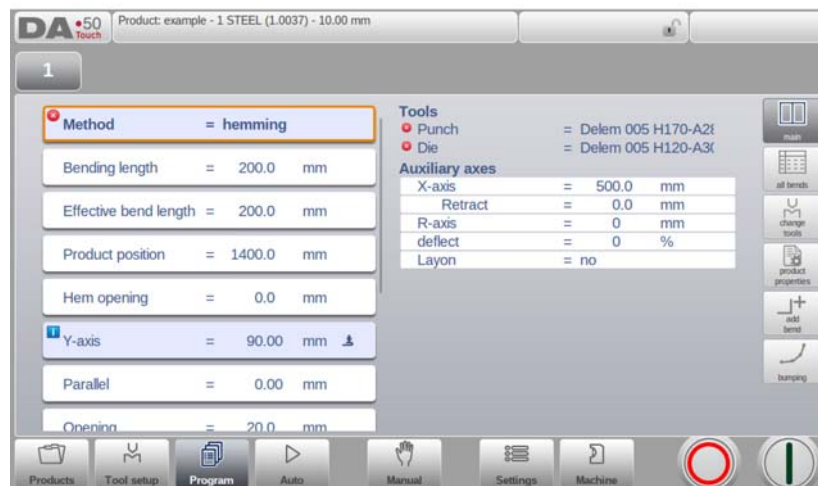
When Disabled the calculation is including half size segments. If in this case a problem with the size of the V-die opening is detected in the bend sequence determination, the user is asked whether or not to select a re-calculation with equally sized segments.

Bumping corrections can be found in Automatic mode where the user is helped in distributing the correction over the related bends.

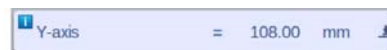
4.4. Programming parameters

Parameters in program mode can be programmed one by one. The effect of the parameter on other parameters is automatically computed.

The relation between parameters is visualized with a symbol and a background color.



When an information symbol is shown with parameters after an edited value, this parameter was changed due to the last changed input.



A star symbol is shown with parameters if the value of the parameter differs from the calculated value by the control. This can be helpful if a value is intentionally programmed different or if the value of a parameter is limited by the parameters limits.



An error symbol is shown with parameters if the value cannot be correct according to the currently programmed values. This, eg. when a hemming bend is programmed with no hemming tools programmed.



5. Automatic mode

5.1. Introduction



By tapping the navigation button Auto the control is switched to the automatic production mode.

In auto mode with the active program, production can be started. After entering Auto, the Start button can be pressed and production can begin.



The automatic mode executes the program automatically bend by bend after pushing the Start button. When selecting a different product in Products mode, which is in the library and has already been used for production, one can immediately switch to Auto and start production. Every time after a different 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 of the Auto mode screen the selected product is displayed along with the product description. At the top of the screen the bend selector shows the available bends in the program. By tapping the preferred bend the bend can be selected. The start button can be pressed to start from this bend. The details of the selected bend are shown in the available views.

The repetition of a bend and the connected programs, when applicable, are shown in the header of the screen. A connected program is also indicated in the bend selectors last position.

5.1.1. Auto mode, parameter explanation

Following is a list of the available parameters in Auto mode.

Corrections

Angle 1 / Angle 2

Corrections on angle values in this bending.

Angle corrections can be programmed for both sides of the machine, Y1 and Y2. When correction Angle 1 is entered for one side, this value is automatically copied to the correction Angle 2 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:

Programmed value of 90 degrees.

Measured value of 92 degrees.

-> Then it is required to program of correction of -2.

Programmed value of 90 degrees.

Measured value of 88 degrees.

-> Then it is required to program a correction of +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.

The angle correction database can be found on the Program Settings page in the Settings mode.

Y1 / Y2

Corrections on the Y-axis positions, in case absolute programming is used or bottoming is selected for a bend.

Auxiliary axis

Corrections on auxiliary axis positions in this bending. In case bend allowance is activated (see the Settings mode) and a program has been entered in Program mode, the X-axes correction values are the result of bend allowance calculation. The

corrections will be saved in the active bending program.

The auxiliary axis correction should be entered as following examples indicate:

Programmed value of 200 millimetres.

Measured value of 202 millimetres.

-> Then it is required to program a correction of -2.

Programmed value of 200 millimetres.

Measured value of 198 millimetres.

-> Then it is required to program a correction of +2.

deflect

Correction on the crowning device.

Only available if a crowning device is present

General corrections

Thickness

General correction on the product thickness, valid for each bend of the program.

Angle / Angle 2

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.

Depth

General correction on the Y-axis position, in case 'absolute programming' is used and 'air bend' is selected for a bend. This correction is valid for each bend of the program.

X-axes

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.

deflect

General correction on the crowning device.

Only available if a crowning device is present.

General

Stock

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

Repetition

Selection of one of the repeated steps of one bend. Useful if a bend has a repetition value larger than 1. This parameter is visible when the parameter Repetition in Program mode has been set to a value larger than 0.

Step mode

Select to use either Auto mode (no) or Step mode (yes). In Step mode you have the same possibilities as in Auto 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 the Start button on the front panel of the control.

5.2. View modes

The auto mode screen is offering a diversity of views which, depending on ones production methode, can be chosen. When selecting auto mode for the first time, the main screen will appear. On the right side of the screen the available view modes can be selected.

Following view modes are available:



Main, numerical bend data. See for details paragraph 7.2.1



All bends, shows all bend data in a table. See for details paragraph 7.2.2



Macro, large visualisation of axes values shown in a list. See for details paragraph 7.2.3



Manual positioning, large visualisation of axes values with the posibility to move axes from their position and teach the axis value in the selected bend. See for details paragraph 7.2.4



Corrections, all correction values for angle as well as axes positions. See for details paragraph 7.2.5



Diagnostics, special information on the axes positioning and I/O status of the control system. This view mode is meant for service purposes. See for details paragraph 7.2.6

The appropriate view can be switched from and to, not changing the bend data. The Start will not jump to Stop while switching view modes.

5.2.1. Main

Main view shows the numerical data of the bend along with the corrections. The corrections can be programmed here.



Both columns can be scrolled to see all data.

Bend selector

The bend selector in the top of the screen can be used to navigate thru the bends. The indicated bends can be tapped to easily select the desired bend data.



5.2.2. All bends

The all bends view mode shows a table including all bend data. The bends are shown row wise and the columns display all bend parameters.

DA 50 Touch Product: 5801_Arrow - Customer C - 1 STEEL (1.0037) - 1.00 mm

1 2 3 4 5

Y = 123.45 X = 250.0

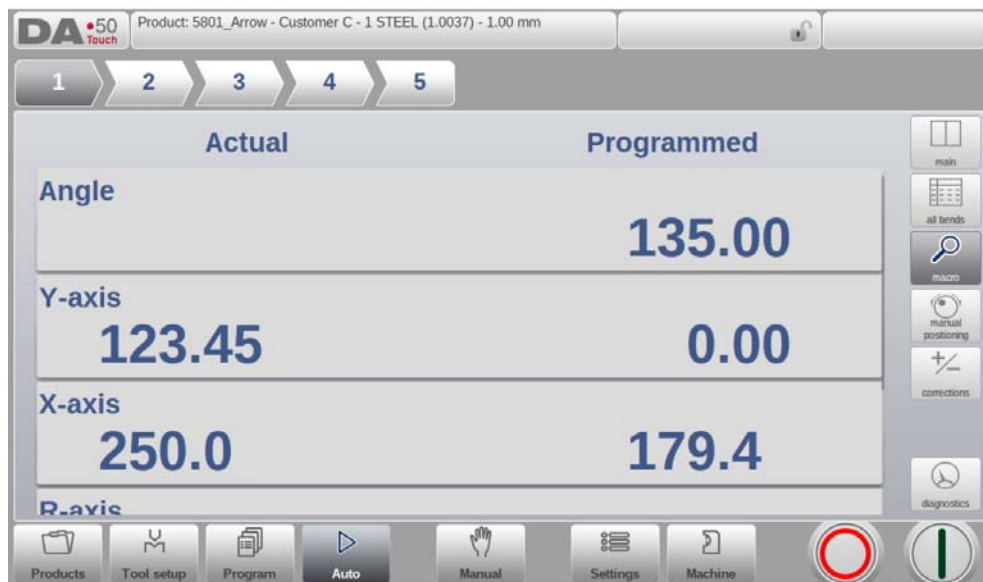
Bend	Angle	Opening	Force	Method	Bend. length	Eff.bend.length	Prod. pr
1	135.00	10.0	22	air bend	600.0	600.0	1200
2	90.00	10.0	22	air bend	600.0	600.0	1200
3	90.00	10.0	22	air bend	600.0	600.0	1200
4	90.00	58.0	22	air bend	600.0	600.0	1200
5	90.00	54.9	22	air bend	600.0	600.0	1200

main
all bends
macro
manual positioning
corrections
diagnostics

Products Tool setup Program Auto Manual Settings Machine

5.2.3. Macro

With macro view mode, the control switches to a view with only large axes values on the screen. This view can be used when working a little remote from the control, still able to read the axes values.



Next to the target position (programmed) also the actual position of all axes can be followed.

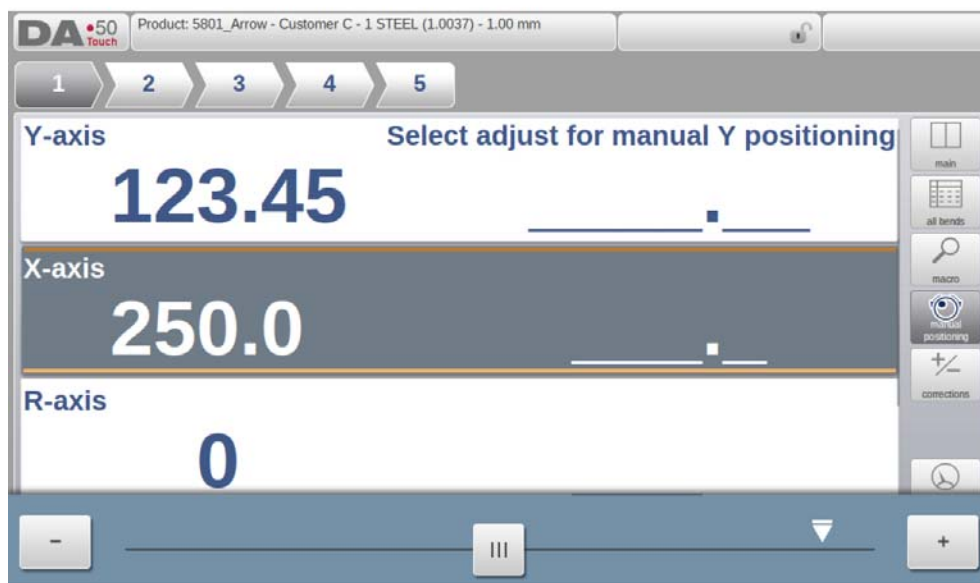
5.2.4. Manual positioning

In manual positioning view mode the axes values are shown at large. Axes can be selected and while selected the position can be controlled by moving the slider, on the bottom of the screen, out of its middle position. When releasing the slider it will return to its middle position automatically.

The teach indicator:



When the teach indicator arrow is pressed, standing in between actual value and programmed value, the value is taught to the program step.



5.2.5. Corrections

In this view mode all corrections of all bends are shown. You can browse through all corrections and change them as you see fit. If a correction for α_1 is entered then this value is, dependent on the settings parameter 'Angle correction programming', copied to the correction for α_2 , or keeping the delta between both corrections, or not influencing the correction for α_2 . Different corrections for α_2 can be entered in the field itself.

Bend	Angle	Gen. corr. α	Corr. α_1	Corr. deflect	Corr. α_2	Stored correction
1	135.00	0.00	0.00	0	0.00	
2	90.00	0.00	0.00	0	0.00	
3	90.00	0.00	0.00	0	0.00	
4	90.00	0.00	0.00	0	0.00	

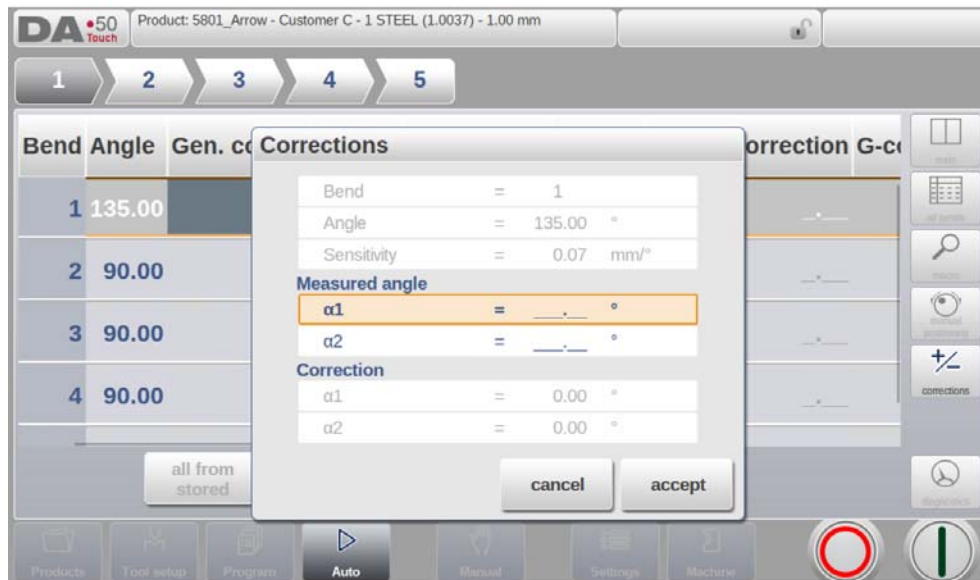
The column 'Stored correction' is only available when the Angle correction database has been activated. When activated, the column 'Stored correction' shows for each bend the correction value that is present in the database. A blank entry in this column means the database does not have a correction value 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.

All From Stored serves to copy corrections in the database to the current program: corrections in all bends are adjusted according to database values.

Calculate corrections, programming of measured angles

To calculate the corrections from measured angle values, one can use the “calculate corrections” function in the correction window. Calculate corrections will open a separate window in which, upon choice, the measured angle(s) can be programmed.



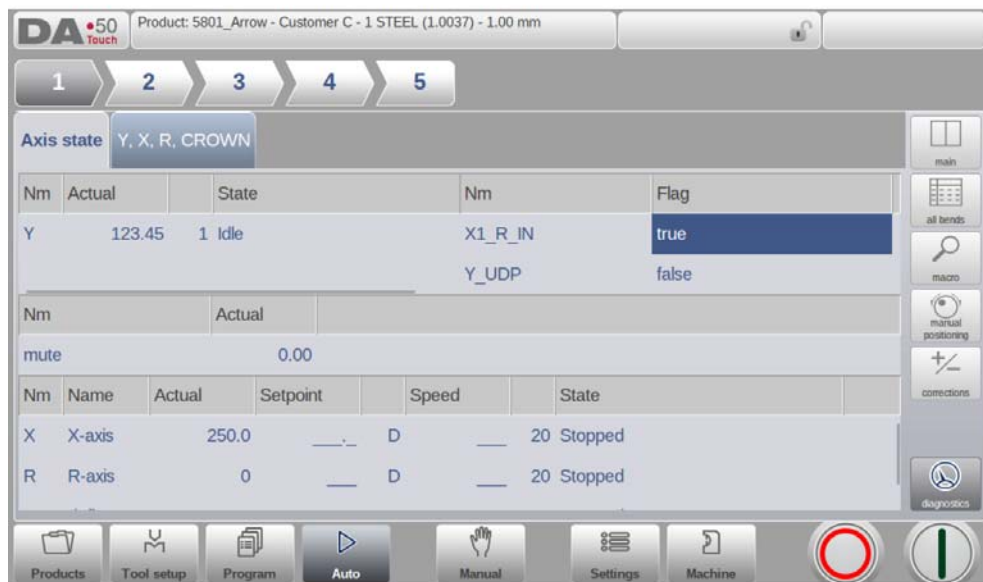
From the programmed value the control will determine a correction. The proposed result can be seen in the window itself. In the top of the window the programmed angle is shown, in the bottom of the window the resulting corrections are shown. When selecting accept, these values will be transferred to the main corrections screen.

When only one measured angle value is entered, the other values will be copied equally. If there are separate left, right or even middle values, these can be entered as well. The appropriate correction values will be determined from the entered values. The middle measured angle, if applicable, is transferred to an absolute crowning correction.

Axes corrections can also be edited in the main screen. When there are multiple axes available this special view mode can be switched to for all axes corrections.

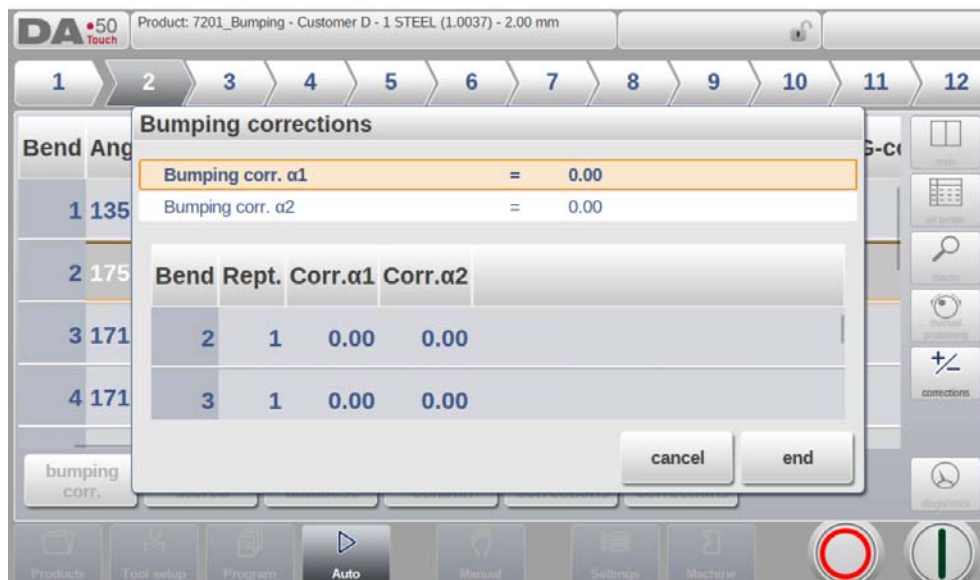
5.2.6. Diagnostics

The diagnostics view mode is meant for service purpose mostly. In diagnostics the activities of independent axes can be monitored. I/O on the control system can be followed. In rare situations this information can be helpful to diagnose operation during the bending proces.



5.3. Bumping correction

In case of a selected bumping bend a general correction for a bumping bend can be entered. This function is only available if a product is loaded that contains a bumping bend. With Bumping Corr. a new window appears in which the correction can be entered.



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, α_1 and α_2 . If a bumping correction for α_1 is entered then this value is, dependent on the settings parameter 'Angle correction programming', copied to the bumping correction for α_2 , or keeping the delta between both bumping corrections, or not influencing the bumping correction for α_2 . Subsequently all separate corrections for α_2 are recalculated. To change correction values of α_2 , use bumping correction α_2 or one of the separate corrections of α_2 .

6. Manual mode

6.1. Introduction



By tapping the navigation button Manual the control is switched to the manual production mode.

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



Manual mode is independent from Automatic mode and can be programmed independently of the programs in memory.

In the top of the Manual mode screen you can find the Y-axis and the main X-axis current position. All other axes and functions are listed one by one in the two columns below. When these Y-axis value and X-axis value are highlighted it means that the reference markers of these axes have been found and that they are positioned correctly referred to their programmed values.

6.1.1. Manual mode, parameter explanation

Following is a list of the available parameters in Manual mode.

Bend parameters

Method

Select the required bending method. The control supports the following standard methods:

- Air bend
- Bottoming
- Hemming
- Hemming & bottoming



The bend methods have been explained in more detail in the Program mode.

Bending length

Program the bending length of the sheet.

Product position

The absolute position value of the product in the Z-direction. Left machine side is reference position zero.

Angle

Angle to bend.

Corr.α 1, Corr.α 2

Correction on angle to bend.

The angle correction should be entered as following examples indicate:

Programmed value of 90 degrees.

Measured value of 92 degrees.

-> Then it is required to program Corr.α with -2.

Programmed value of 90 degrees.

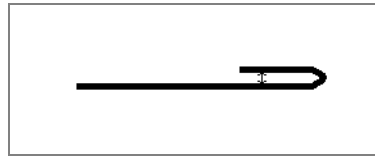
Measured value of 88 degrees.

-> Then it is required to program Corr.α with +2.

Hem opening

The hem bend can be made with a certain opening distance between the 2 flanges. The hem opening value will be used calculating the beam position in the hemming process. By default this parameter has the value of the Settings mode parameter Default Hem

Opening.



Corr.Y

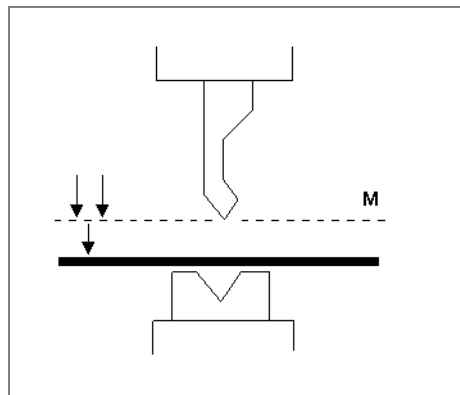
Correction on the Y-axis position, in case bottoming has been selected.

Y-axis

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

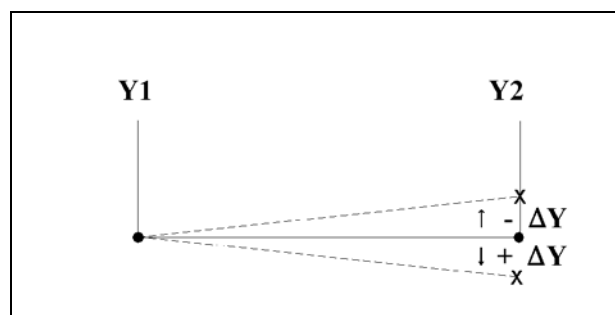
Mute

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. The programmed value is the Y-axis point above the sheet.



Parallel

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



Opening

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.

Force

Force

The programmed force applied during pressing.

Dwell time

Hold time of punch at the bending point.

Decompression

Decompression distance after the bending to release the working pressure from the system.

Speed

Speed

Pressing speed, the speed of the Y-axis during bending.

Decomp speed

The decompression speed is the programmable speed of the beam during the decompression distance.

Functions

Wait for retract

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

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

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.

Product properties

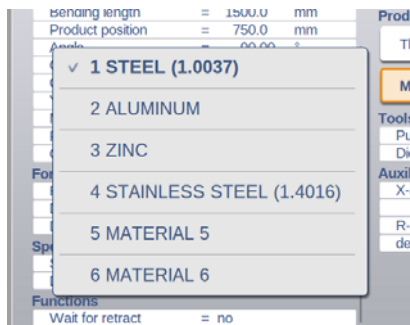
Thickness

Program the thickness of the sheet.

Material

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. The materials can be programmed on the Materials page in

the Settings mode.



Tools

Punch

The name (ID) of the selected punch. Tap to modify or select from the punch library.

Die

The name (ID) of the selected die. Tap to modify or select from the die library.

Auxiliary axes

Auxiliary axis

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

Retract

The retract distance of the axis during the bend. The "backgauge retract" is started at the pinching point.

Speed

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

The above mentioned parameters can be programmed and modified as required. After pushing the Start button the programmed parameters are active.

6.1.2. Tool setup

The programming of the tool setup in Manual mode is similar to programming the tool setup which is used in Automatic mode. Despite the fact that both modes do not share the same tool setup (enabling the usage of a complete different tool setup), the tool setup of Automatic mode could be used in Manual mode as well.

While switching from Automatic mode to Manual mode, the control offers the user to use the same tool setup in Manual mode and thus also the user is warned that in case differently programmed, one should be careful.



In the tool setup menu tools can be added or removed, similar to the main Tool Setup mode as described in chapter 3.

Adding tools (Punches / Dies)

Same as in Tool Setup, via the Add function tools can be added.

6.2. Programming parameters & Views

Parameters in manual mode can be programmed one by one. The effect of the parameter on other parameters is automatically computed.

The relation between parameters is visualized with a symbol and a background color.



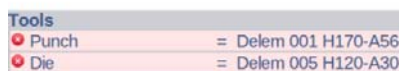
When an information symbol is shown with parameters after an edited value, this parameter was changed due to the last changed input.



A star symbol is shown with parameters if the value of the parameter differs from the calculated value by the control. This can be helpful if a value is intentionally programmed different or if the value of a parameter is limited by the parameters limits.



An error symbol is shown with parameters if the value cannot be correct according to the currently programmed values. This, eg. when a hemming bend is programmed with no hemming tools programmed.



View

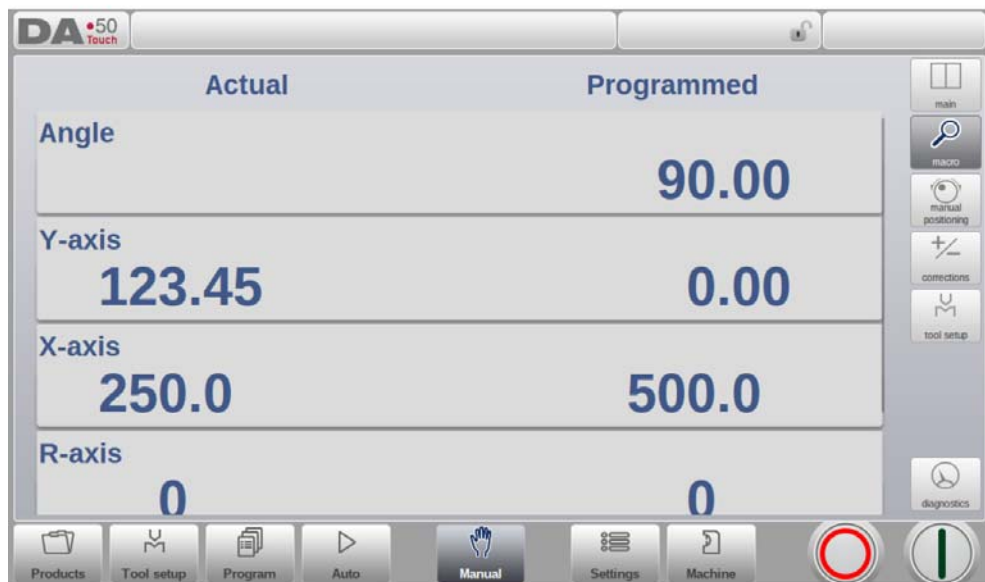
The command buttons on the right side of the screen give access to other views. Next to the

Main view, there are Macro, Manual Positioning, Corrections as also a Diagnostics view.



6.3. Macro

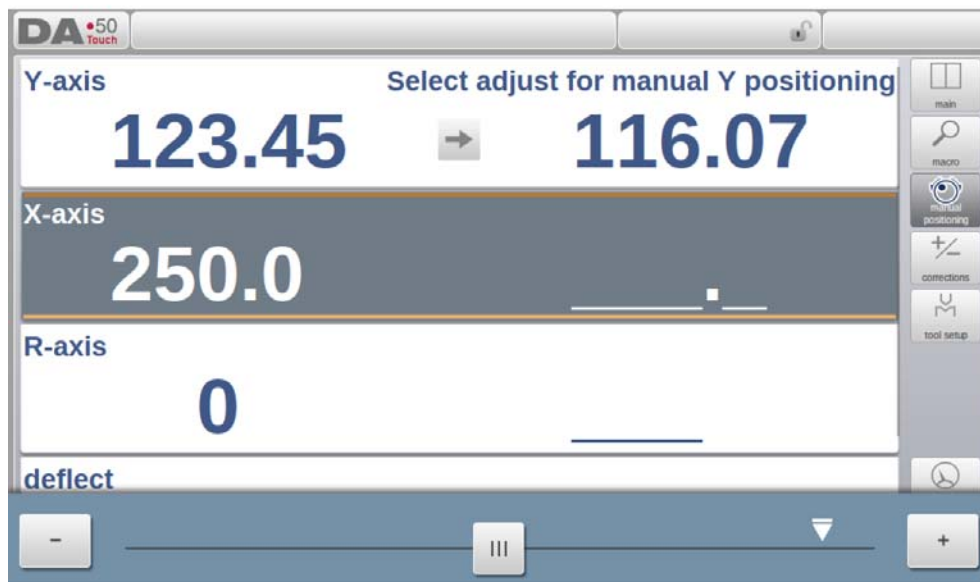
With Macro the control switches to a new view with only large axes values on the screen. This view can be used when working a little remote from the control, still able to read the axes values.



6.4. Manual movement of the axes

6.4.1. Movement procedure

To move an axis to a specific position manually, the slider at the bottom of the screen, can be used. After tapping Manual Pos in the main screen of Manual Mode, the following screen appears:



Within this mode, any of the shown axes can be moved by moving the slider out of its middle position. The procedure for moving the axis depends on the axis you wish to move. When releasing the slider it will automatically return to its middle position.

Auxiliary axes

The control must be stopped (the Stop button is on).

First select the desired back gauge axis, you will see the cursor at the required axis.

Then you can move the axis by moving the slider.

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 Start button is on).
- The 'adjust' function must be active. If this function is not active a message is shown in the upper right-hand corner.
- The Y-axis must be below mute-point.
- A pressing command must be given to the CNC.

6.4.2. Teach

To teach the control, taking over a position found by manual moving an axis, a simple procedure can be used.

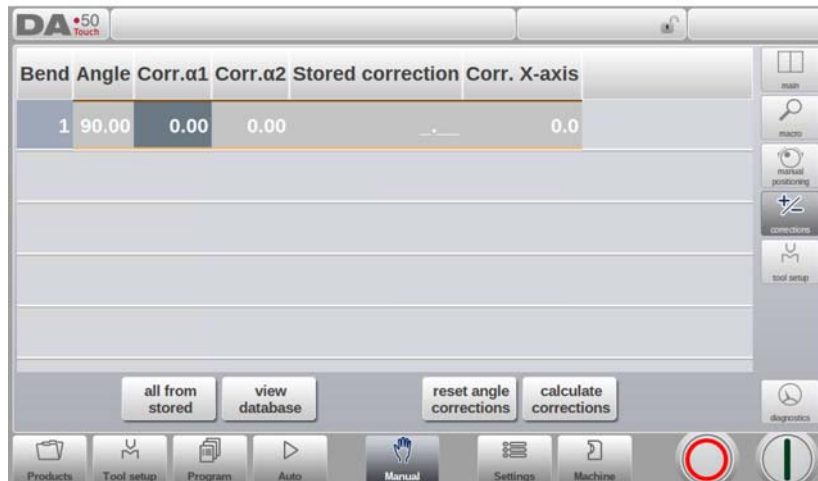
When you have moved an axis to a certain position with the slider, you may want to store this position. To do so, tap the axis name in the Programmed column. The actual axis value (left side) will appear in the programmed axis field (right side).



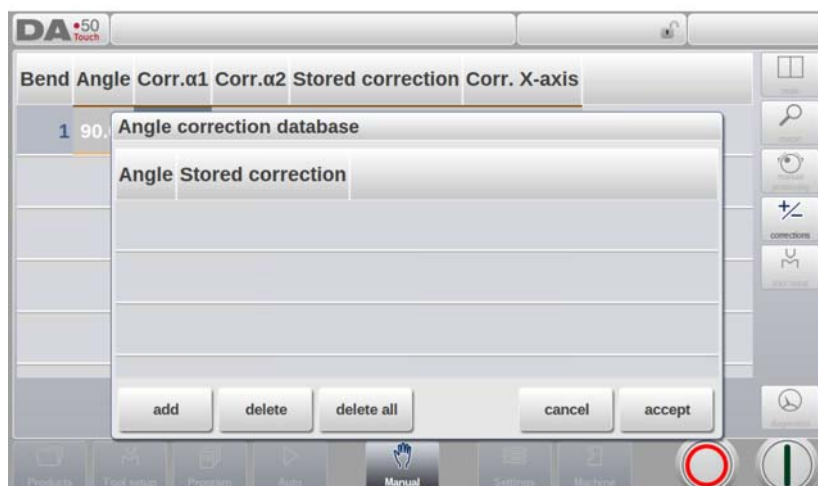
When you return to the standard screen of manual mode, the axis parameter will have the recently taught value.

6.5. Corrections

In this view mode the corrections for the bend programmed in Manual mode are shown. Since this is always a single bend, a single line will be shown.

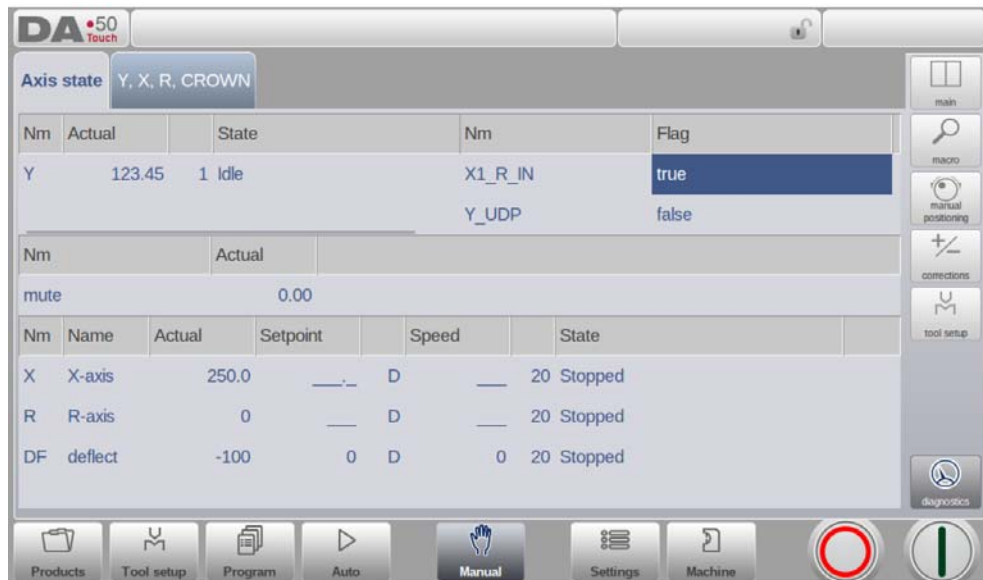


The programmed corrections can be verified here similarly to the corrections in Automode. Entries in the correction database and for initial correction can also be monitored in this screen. Since these are of significant influence on the bend result, access to the database can be used to modify. This can also be useful while finding appropriate corrections with testbending and storing the found results in the database.



6.6. Diagnostics

When tapping Diagnostics, the control switches to a view which shows 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.



6.6.1. IO status

When tapping on the I/O tab in the Diagnostics, the control switches to a 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.



Pin #	Description	Pin #	Description	Pin #	Description	Pin #	Description
1	KEY_LOCK	9	Y_UDP	17	Y_MUTE2	25	X_OK
2	NC	10	Y_ER	18	Y_CLAMP	26	X_IP
3	CR-	11	Y_MUTE	19	Y_LDP	27	R_OK
4	CR+	12	NC_START	20	C_NCRDY	28	R_IP
5	NC	13	Y_OPEN	21	Y_T	29	X_START

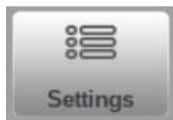
Zoomed IO

When tapping on one or more (up to 5) pins an extra page Zoomed IO is created with an enlarged view of the selected IO; selected pins will be shown in large, enabling distant monitoring.



7. Settings

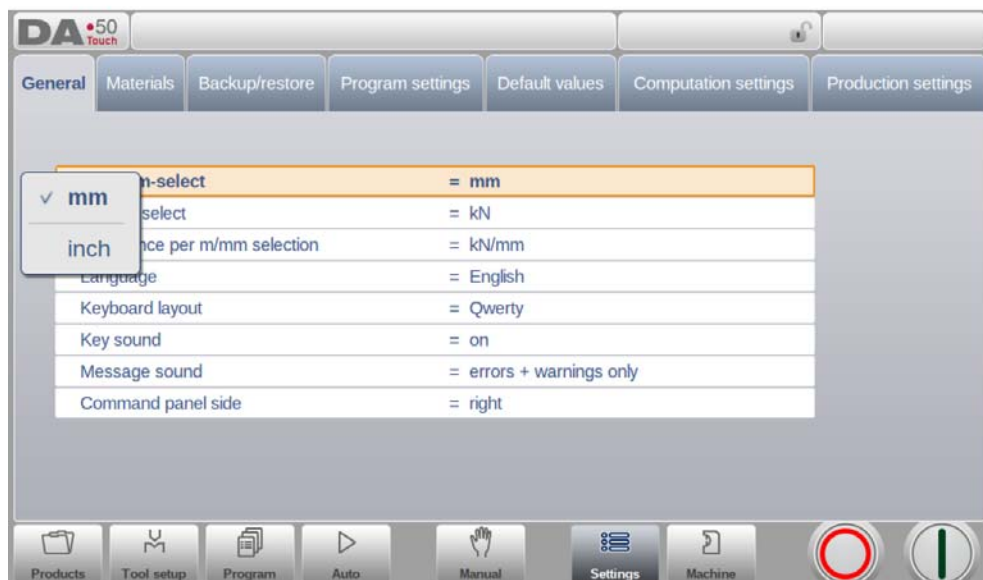
7.1. Introduction



By tapping the navigation button Settings the control is switched to Settings mode.

The Settings mode of the control, which can be found in the navigation panel, gives access to all kind of settings which influence the programming of new products and programs. Default values and specific constraints can be set.

The settings are divided across several tabs logically organizing the different subjects. In the following sections the available tabs and detailed settings are discussed.



Navigation through the tabs can be done by just tapping them and selecting the required item to adjust. Since there can be more tabs than the screen can show in one view, dragging the tabs in horizontal direction enables to view and select all available tabs.

7.2. General

Select the required tab and tap the parameter to be changed. When parameters have a numerical or alphanumerical value, the keyboard will appear to enter the desired value. When the setting or parameter can be selected from a list, the list will appear and selection can be done by tapping. Longer lists allow scrolling vertically to check the available items.

Inch/mm-select

Select to use either millimeters or inches as the unit to be used.

Ton/kN select

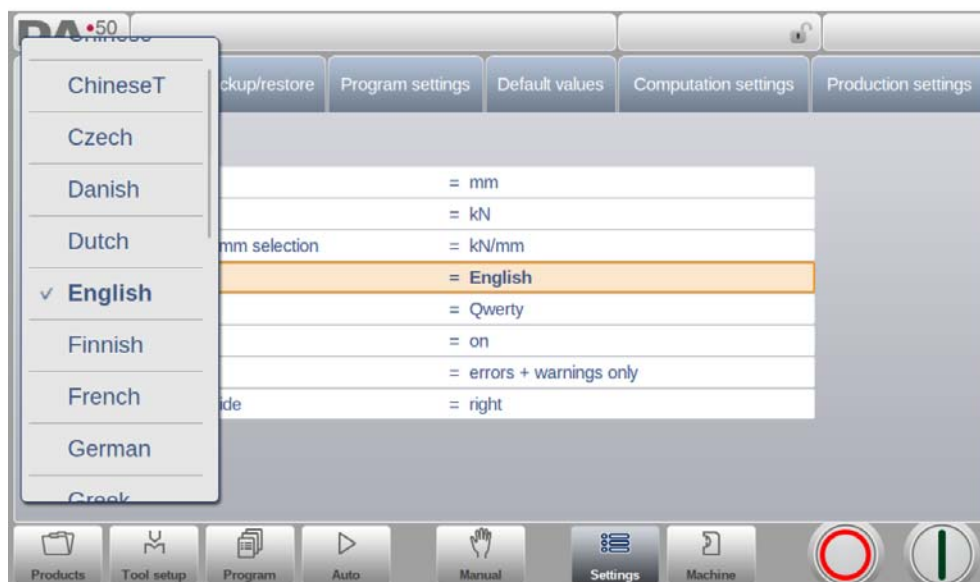
Select to use either Ton or kN as the main unit to be used for all force data.

Resistance per m/mm selection

Select to use the resistance either per meter or per millimeter.

Language

The user interface language can be selected from the list. There are more available languages than initially shown. Scroll vertically by dragging the list up and down to see all available languages. Tap to select the desired language for the user interface.
(For languages using special, non standard alphanumerical characters, the control will reboot.)



Keyboard layout

Upon choice one can select Qwerty, Qwertz or Azerty keyboard layout.

Key sound

Switch the sound function of the input panel on or off.

Message sound

Parameter to enable/disable the sound function for messages dependent on the message type.

all messages => sound on for all messages.

errors + warnings=> sound on for errors and warnings only.

errors => sound on for errors only.

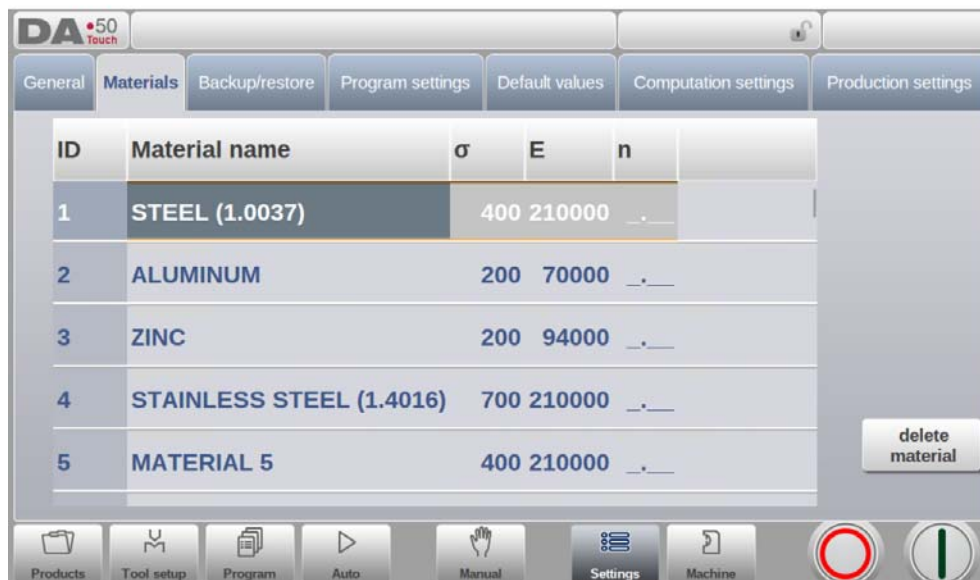
none => sound off for all messages.

Command panel side

Switch the command panel to the left side of the screen.

7.3. Materials

In this tab, materials with their properties can be programmed. Existing materials can be edited, new materials can be added or existing materials deleted. A maximum of 99 materials can be programmed on the control.



For each material 3 properties are present and can be viewed and edited.

Material name

Name of the material, as it will appear in the programming screens. The maximum allowed length of the material name is 25 characters, the name must begin with a character (not a numeral).

Tensile strength

Tensile strength of the selected material.

E module

E-module of the selected material.

Strain hardening exponent

The Material strain hardening exponent, n , is a material property that should be provided by the supplier of the material, just like the tensile strength and E-module. Entering the correct values for this parameter will give an improvement of the inner radius calculation and thus a more accurate bending depth and bend allowance calculation.

In its turn a more accurate bend allowance will result in more accurate back gauge positions.

Initially the value is set to `._.` for all the materials. This means that the parameter n is

not active. The result of all calculations is the same as with previous software versions.

The range of the parameter n is 0.01 – 1.00.

For example, a typical value for mild steel is 0.21.

When 0 is entered again, the value will be reset to $___$.

Calculate n

The Material strain hardening exponent, n , is a material property that should be provided by the supplier of the material, just like the tensile strength and E-module.

As an alternative, it can also be derived from the bend allowance. A test bend has to be made in Manual mode. When you switch to the materials table and tap the button 'calculate n ', the following window will appear on the screen:

The screenshot shows the 'Calculate n' dialog box in the DA 50 Touch software. The dialog box is overlaid on the 'Manual' screen. It contains the following fields and values:

Angle	=	90.00 °
Material	=	1
Thickness	=	1.00 mm
V opening	=	16.60 mm
Punch radius	=	1.00 mm
Die radius	=	1.60 mm
Die angle	=	30.00 °
Length	=	500.00 mm
X position	=	500.00 mm
Measured	=	mm
Bend allowance	=	mm
Calculated	=	mm
n	=	mm

At the bottom of the dialog box are 'cancel' and 'accept' buttons. The background shows a materials table with columns ID, Material, Thickness, and Length.

The parameter values are taken from the Manual mode screen. After the bend the resulting side length should be measured and entered in the window. With the difference between the programmed X-axis position and the measured side length the bend allowance and the strain hardening exponent (n) are calculated.

The accuracy of the calculation depends on the accuracy of the sheet thickness, tool parameters and side length measurement.

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 by tapping the title of the column. The materials will be sorted in ascending or descending order of that property.

To change an existing material, select the relevant line and change the values as you see fit. To delete an existing material, select the relevant line and use Delete Material. This will erase the values.

To program a new material select an empty line and start programming its values.

7.4. Backup / restore

This tab offers the possibilities to backup and restore products, tools as well as settings and tables. When products or tools originate from older control models, the DLC-file format products and tools can also be restored using this specific restore function.

For materials a specific backup and restore are available here.

Tools and products can be backed up and restored according to the following procedures. The procedures for saving or reading data are similar for all types of backup media: e.g. network or USB stick.



The actual backup directory consists of a device (USB stick, network) and a directory. The choice of devices depends on which devices have been connected to the control. If necessary, directories can be created and selected. The backup locations for storage of products and tools can be set independently.

7.4.1. Product backup

To make a backup of programs to disk, choose 'products' in the Backup section on the Backup/restore page.



When the initial backup directory has been set, the products backup screen appears.



In the backup screen the products in the selected directory are shown. Basic functions to change the view can be chosen similarly to the Products mode. This enables to easily find the required products to be backed up.

At the top of the screen, the current source location is shown as well as the backup location. To backup a product, select it by tapping, in the list. The backup marker will appear to confirm the backup action. If a product file with the same name is present on the backup location, a question is offered whether or not to replace that file. To backup all products at once, tap All.

The source where the products are located which have to be backed up can be changed with Source Directory. The directory browser appears and the desired source directory can be navigated to.

The directory where the products which need to be backed up need to go can be changed as

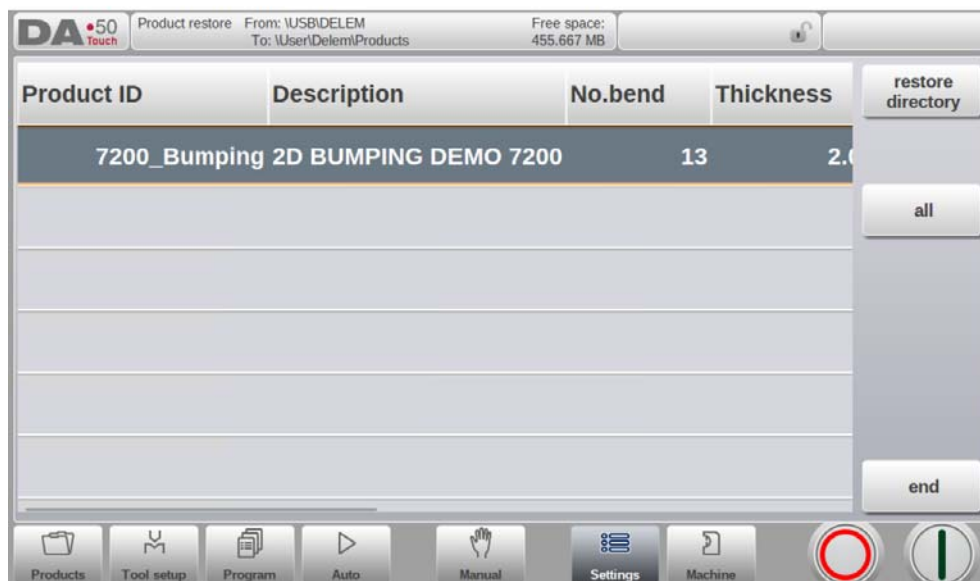
well. With Backup Directory the directory browser appears and the desired destination directory can be navigated to.

7.4.2. Product restore

To restore programs to the control, choose 'products' in the Restore section on the Backup/restore page.



When the initial restore directory has been set, the products restore screen appears.



In the restore screen the products in the selected directory are shown. Basic functions to change the view can be chosen similarly to the Products mode. This enables to easily find the required products to be restored.

At the top of the screen, the current restore source location is shown as well as the location on the control to restore to. To restore a product, select it by tapping, in the list. The restore marker will appear to confirm the restore action. If a product file with the same name is present on the control, a question is offered whether or not to replace that file.

The source location where the products to be restored are coming from can be changed with Restore Directory. The directory browser appears and the desired restore directory can be navigated to.

The directory where the products which need to be restored need to go can be changed as well. With Destination Directory the directory browser appears and the desired destination

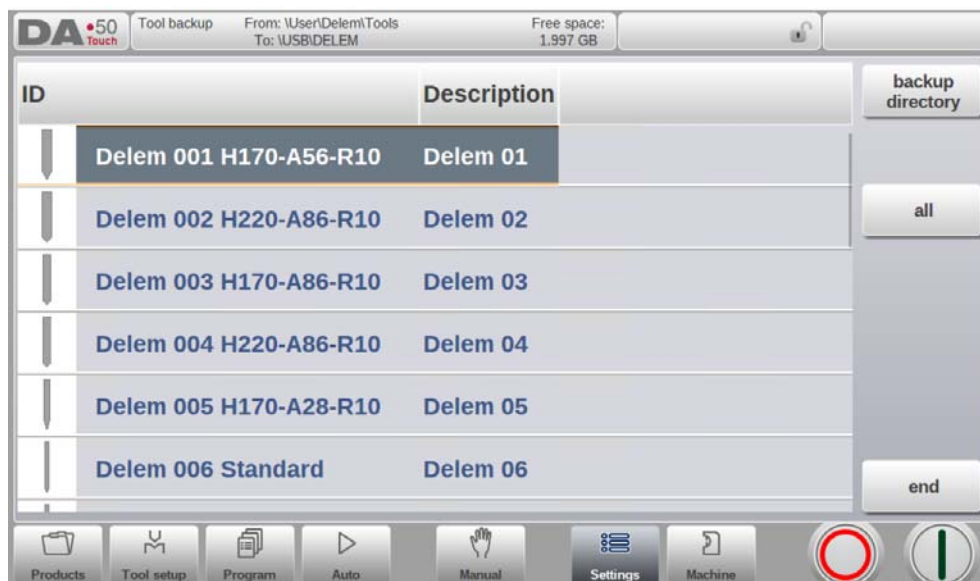
directory can be navigated to.

7.4.3. Tool backup

To make a backup of tools to disk, choose 'tools' in the Backup section on the Backup/restore page.



When the initial backup directory has been set, the tools backup screen appears.



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.

7.4.4. Tool restore

The restore procedures for tools run similar to the procedures for a product restore.

7.4.5. Backup and restore for Tables and Settings

To backup user specific settings and tables the Backup/restore tab offers specific functionality. The procedure is again similar to the backup and restore of products and tools. The special function All will automatically execute all steps sequentially for either Backup or Restore (Products + Tools + Tables + Settings).

7.4.6. Directory navigation

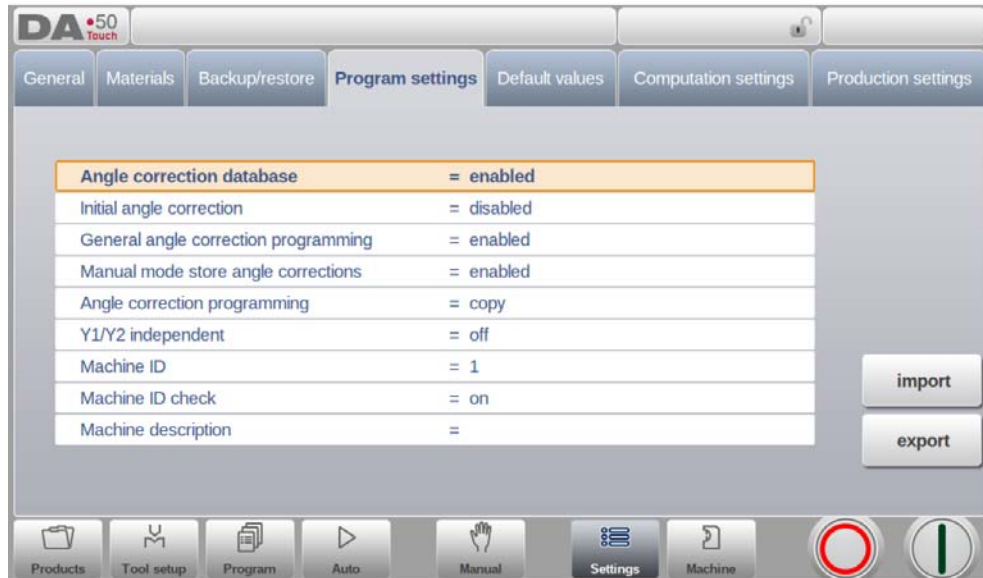
When Backup Directory is used, a new window appears with a list of available backup directories.



In this window you can browse through the directory structure of your backup device. To select the directory you are currently in, tap Select. To change from one device to another, tap the highest level, and 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 by tapping Make Subdir and Remove Subdir. If there are subdirectories present, just tap it to move to the required directory and tap Select to select it.

7.5. Program settings



Angle correction database

Parameter to enable the database with angle corrections.

Angle corrections are entered in production mode (Automatic mode). These corrections are stored in the product program.

The Angle correction database enables the possibility to store these corrections in a database. In this way corrections that have once been entered for certain bends remain available for future use in other products.

With this setting enabled, 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 difference of the corrections of the two adjacent bends is more than 5°, there will be no correction offered.

Initial angle correction

To program relative small angle corrections the initial correction database is available. This parameter is independent on the parameter "Angle correction database". The initial correction is only visible and programmable on the corrections page in Automatic mode and Manual mode. On the main page in Automatic mode and Manual mode the initial correction is not visualized. The total correction is the sum of the visualized correction and the initial correction.

Example:

- Program an angle correction of -8 degrees.
- Program an initial correction of -6 degrees. Now the total correction remains unchanged: the visualized correction is changed from -8 degrees to -2 degrees.

disabled => no initial angle corrections programmable.

enabled => initial angle corrections programmable on the corrections page

General angle correction programming

To program general angle corrections which are used in all bends of the program. These angle corrections are not related to specific bend angles and therefore not stored in the angle correction database.

disabled => no general angle corrections.

enabled => only G-corr. $\alpha 1$.

$\alpha 1$ and $\alpha 2$ => G-corr. $\alpha 1$ and G-corr. $\alpha 2$.

Manual mode store angle corrections

To enable the storage of angle corrections programmed in Manual mode. Corrections can be derived from bend results in Manual mode which later can be used during product programming.

Angle correction programming

Parameter to switch between copying or keeping the delta values or changing independently when changing angle corrections in production mode.

copy => copy $C\alpha 1$ to $C\alpha 2$ when changing $C\alpha 1$.

delta => keep delta between $C\alpha 1$ and $C\alpha 2$ when changing $C\alpha 1$.

independent => change $C\alpha 1$ and $C\alpha 2$ independently.

X correction programming

Parameter to switch between copying or keeping the delta values or changing independently when changing the X-axis corrections in production mode.

copy => copy CX1 to CX2 when changing CX1.

delta => keep delta between CX1 and CX2 when changing CX1.

independent => change CX1 and CX2 independently.

Only available when an X2-axis is present.

Y1/Y2 independent

Parameter to program the two Y-axes independently.

off => single Y-axis programming.

on => program Y1 and Y2 separately.

Machine 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.

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.

Machine ID check

When a product from a machine with a different machine ID is selected, a warning will appear on the screen. With this parameter this check can be disabled.

Machine description

The description as programmed here will only be used in the offline Profile-T to get an overview of the different machines available in the factory. With this information it will be more clear which machine is used in combination with this control.

7.6. Default values



Y opening default

Default Y-axis opening, used as initial value for the parameter 'opening' in a new program.

Default pressing speed

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

Default decompression speed

Default decompression speed, used as initial value for the parameter 'decompression speed' in a bend program.

Default wait for retract

Default value for the parameter 'wait for retract' in a bend program. This parameter determines the control behaviour in a bend program.

Default step change code

Default value for the parameter 'step change code' in a bend program. This parameter determines the moment of step change in a bend program.
The step change codes have been explained in more detail in the Program mode.

Default delay time

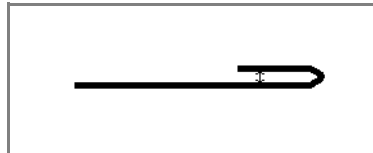
During the postprocessing, the waiting time of the X-axis at step change is set to zero. With this parameter you can preset a longer waiting time when needed for product handling.

Default dwell time

Default value for the parameter 'dwell time' in a bend program.

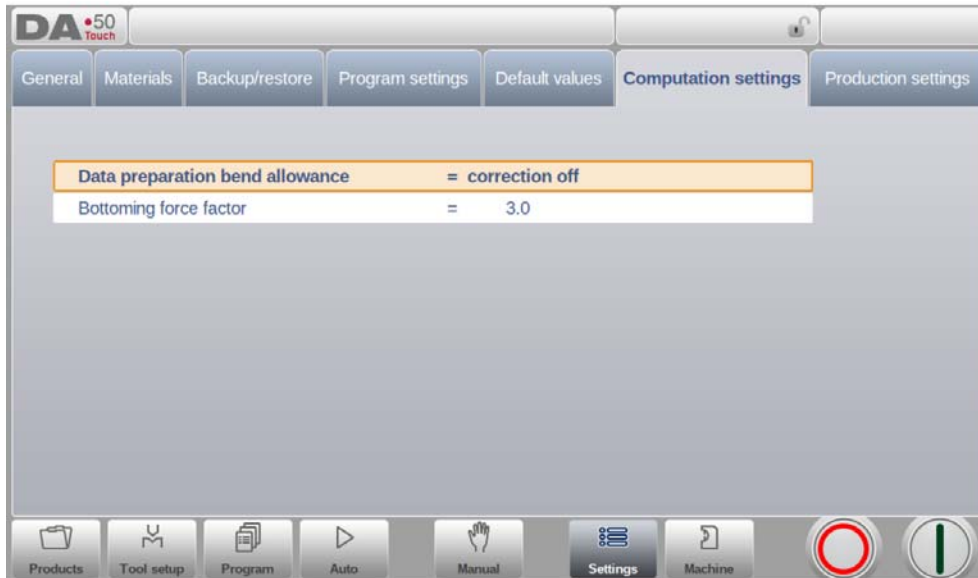
Default hem opening

The hem bend can be made with a certain opening distance between the 2 flanges. The hem opening value will be used calculating the beam position in the hemming process. This programmed default value will be used when programming a new program in the Program mode. The starting value is 0.0mm to get the two flanges of a hem bend completely upon each other without any space between the flanges.

**Default material**

Default material, used as initial selection while starting a new program.

7.7. Computation settings



Data preparation bend allowance

correction off => no bend-allowance added to numerical programming

correction on => bend-allowance correction added to numerical programming

With this parameter you can choose whether or not you wish to have programmed values corrected for bend-allowance. This setting only refers to corrections during product programming in the Program mode. 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 'Auto mode').

Bottoming force factor

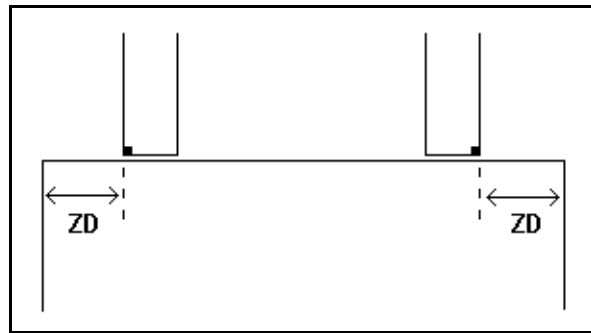
The force needed for an air bend is multiplied by this factor in order to obtain the bottoming force.

Z-distance

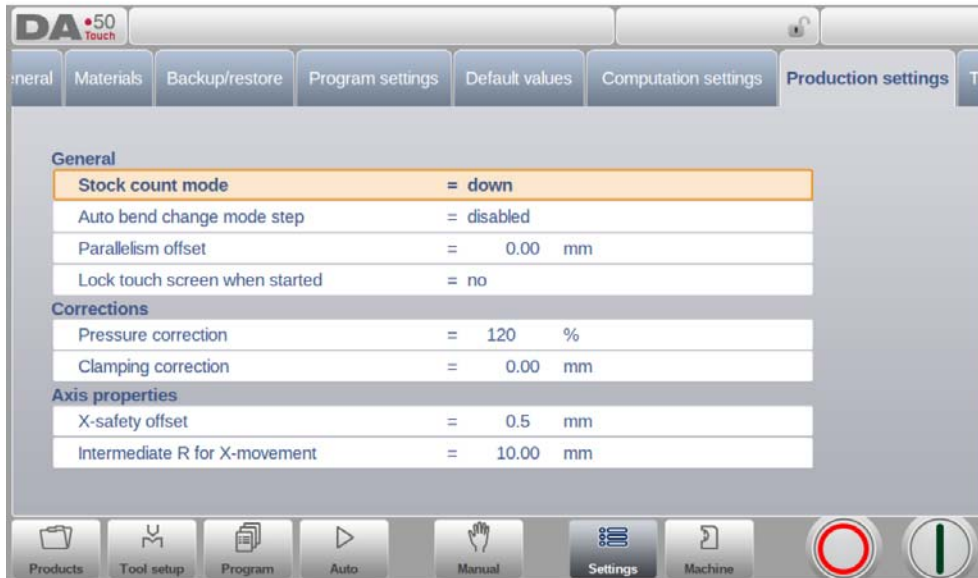
The distance from the edge of the finger to the corner of the sheet.

When an automatic Z-axis has been installed, the positions of the fingers are calculated

automatically with respect to the end of the sheet.



7.8. Production settings



Stock count mode

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 product 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 incremented after each product 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.

Auto bend change mode step

This parameter can be used to have automatic step change in the bending process with the Step mode enabled.

disabled => 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.

enabled => The next bending parameters are loaded automatically but the axes will start positioning after the start button has been pressed.

Parallelism offset

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 + Parallelism offset).

Lock touch screen when started

To enable the locking of the touch screen while the control has been started.

Pressure correction

Percentage of calculated force which actually controls the pressure valve.

Clamping correction

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.

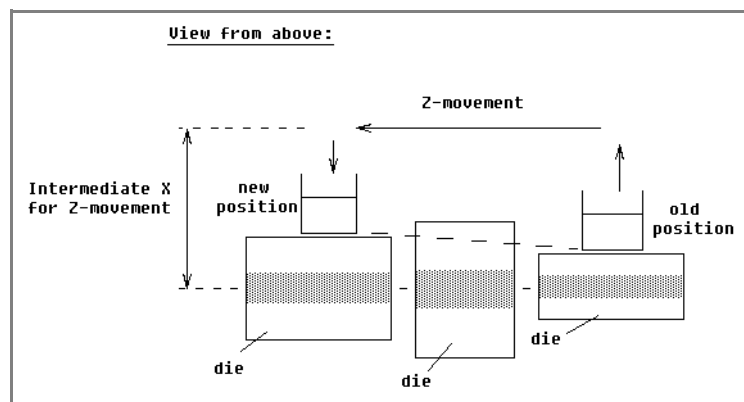
X-safety offset

Defines the safety zone (minimum X-axis value), following the contour of punch and die, which will be used in case a R-axis is mounted. This to prevent collision between finger and punch / die.

Intermediate X for Z-movement

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 offset'.

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.



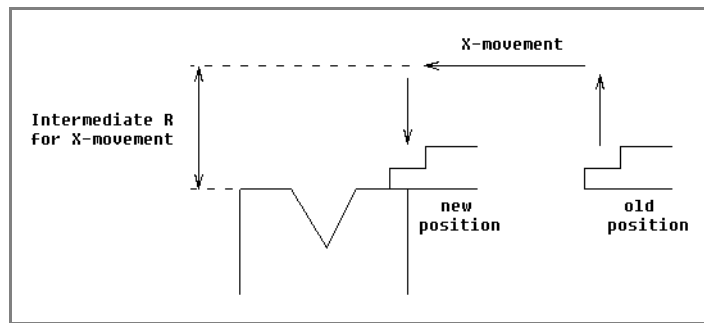
When the back gauge has to move to a different Z-position, it is checked whether the current X-position is safe. We can distinguish the following situations:

- 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.

Intermediate R for X-movement

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.



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:

$$SZ = X\text{-safe} + SD$$

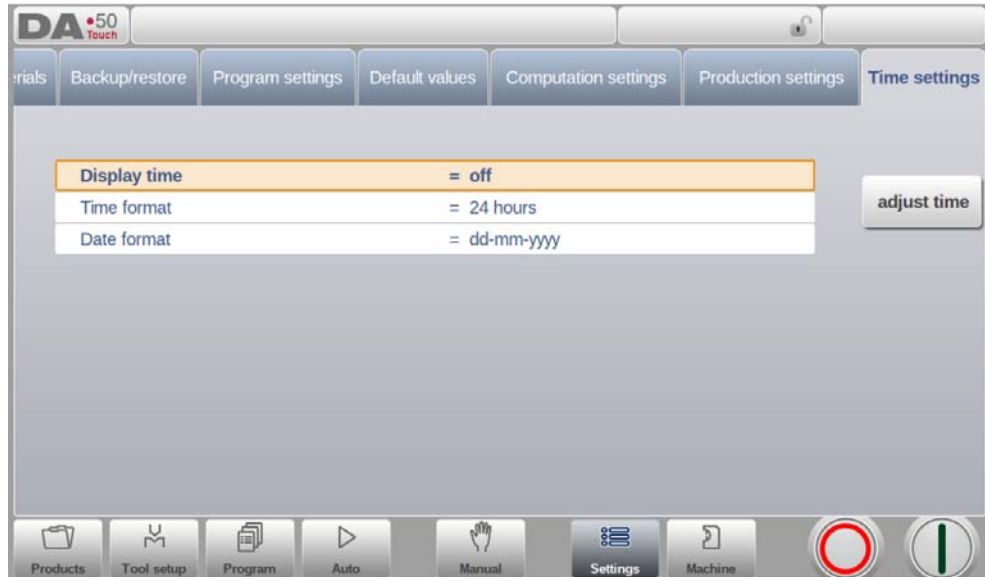
Explanation:

SZ = safety zone

X-safe = safety zone of the die

SD = safety distance, defined by machine supplier

7.9. Time settings



Display time

Display date and time on the title panel, time only or no time at all.

Time format

Display the time in 24 hours or 12 hours format.

Date format

Display the date in dd-mm-yyyy, mm-dd-yyyy or yyyy-mm-dd format.

Adjust time

To adjust the date and time. Adjusting the date and time will also adjust the date and time of the operating system.

8. Machine

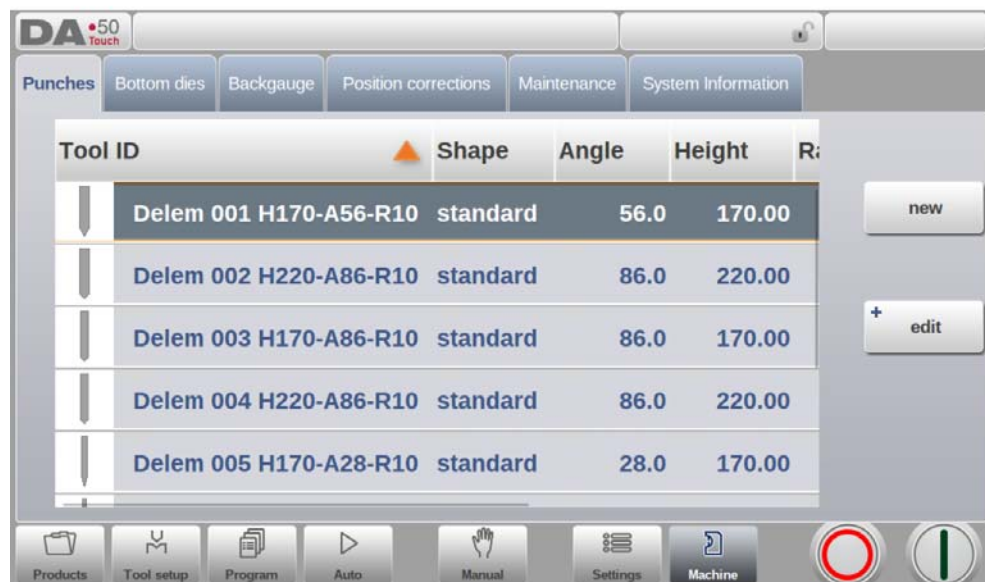
8.1. Introduction



By tapping the navigation button Machine the control is switched to Machine mode.

The Machine mode of the control, which can be found in the navigation panel, gives access to the configuration items of the machine and specific machine characteristics which influence generic calculations and machine behaviour.

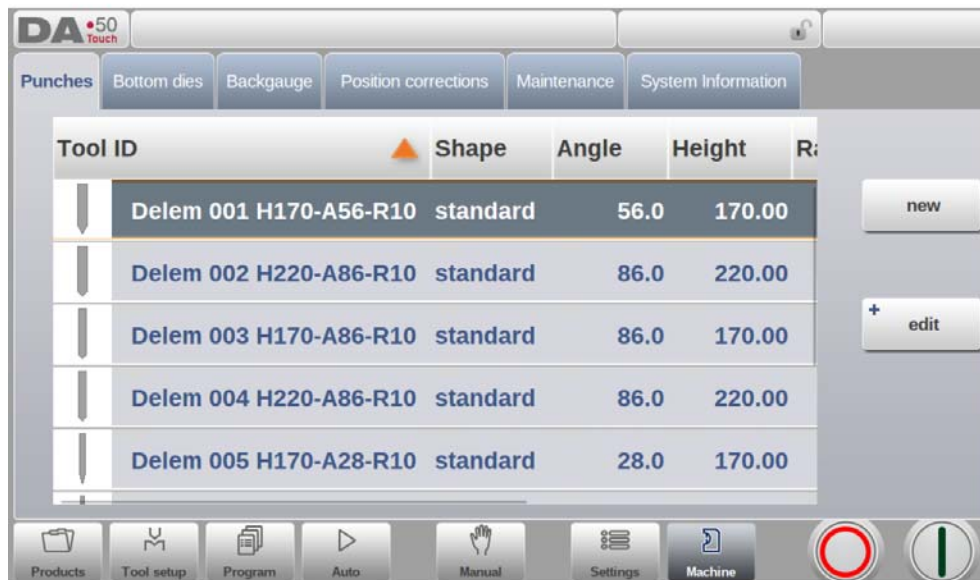
The settings are divided across several tabs logically organizing the different subjects. In the following sections the available tabs and detailed settings are discussed.



Navigation through the tabs can be done by just tapping them and selecting the required item to adjust. Since there can be more tabs than the screen can show in one view, dragging the tabs in horizontal direction enables to view and select all available tabs.

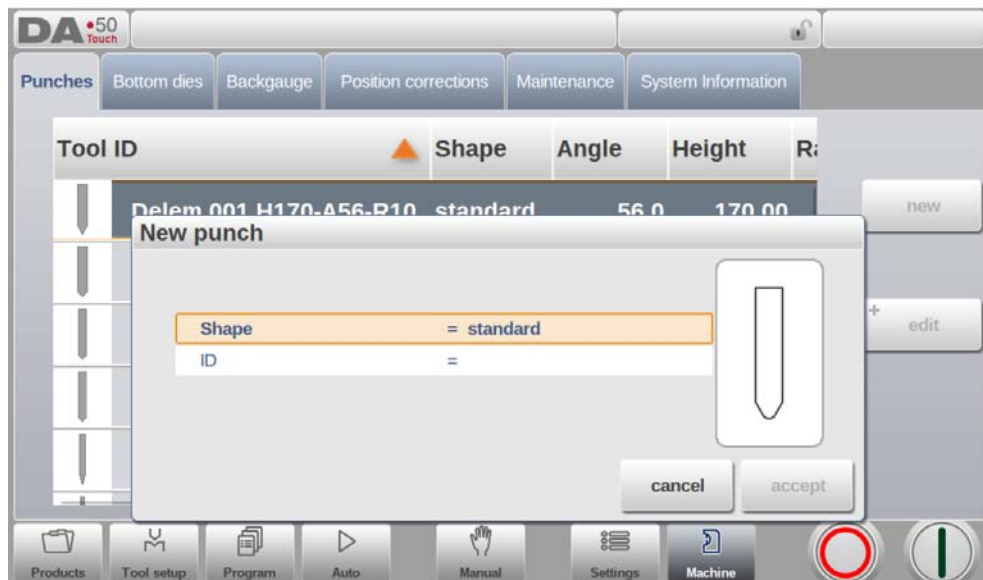
8.2. Programming of Punches

In this tab, the punches used in the machine can be programmed. New punches can be added; existing punches can be edited, copied, renamed and deleted.



8.2.1. Create a new punch

To create a new punch, tap New in the library. The punch profile can be created with help of the programming facilities of the control.



First the shape of the punch and its ID must be programmed. After that the shape details must be programmed following the wizard.

Shape

A selection must be done from the different available basic punch shapes corresponding to the required punch action. Options are:

- Standard Punch shape as normally used for air bending and basic bottoming.
- Hem Bend Punch as typical tool with flat bottom for specific hem bends.
- Air + Hem Bend Punch as tool for normal air bends and hem function.
- Big Radius Punch shape especially for big radius bends.

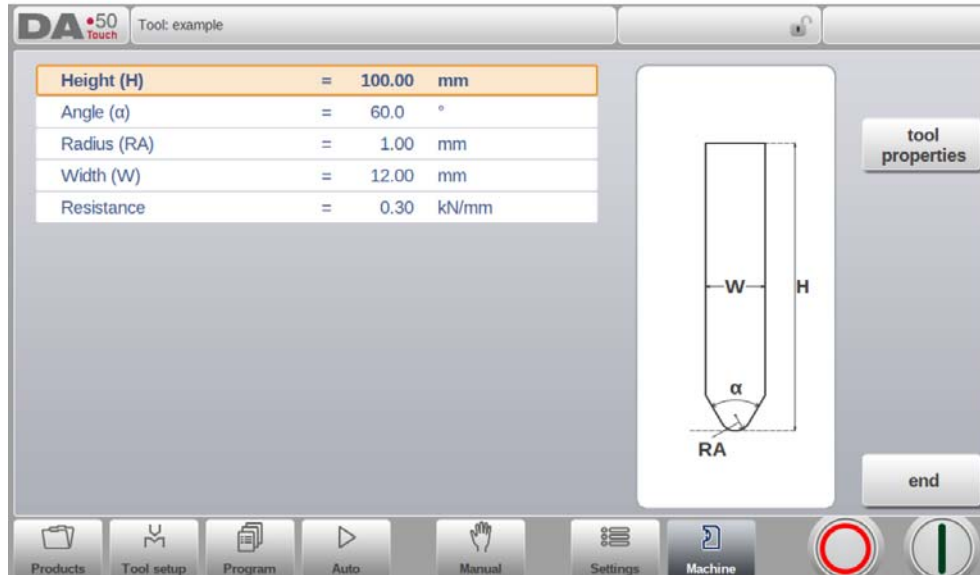
ID

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

The ID parameter may also contain alphanumeric characters. When finished, use Accept to leave this window and start with the tool data parameters. The control will prompt a window for the tool properties with the dimensions of the tool.

Depending on the chosen basic shape, the parameters will differ. In the following paragraphs the available punch shapes are described in detail.

8.2.2. Standard punch



Height

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

Angle

The angle of the punch tip.

Radius

The radius of the punch tip. This value will be used as inner radius of the bend to make when this radius value is bigger than the inner radius as will result from the bending process.

Width

The width of the tool to program.

Resistance

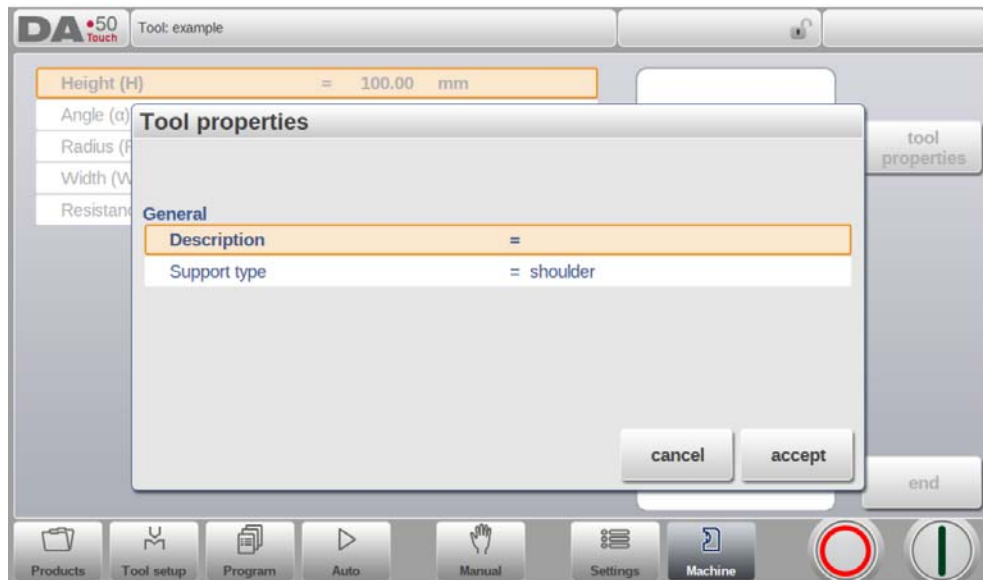
Maximum allowable force on the tool.

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.

Tool Properties

To change the generic tool data and description.

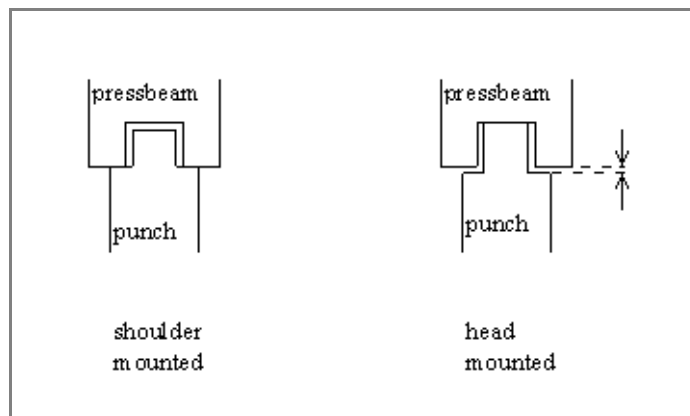


Description

A name or description of this tool. The maximum length is 25 characters. This description has already been entered in the beginning defining the tool, but can be edited in this field. The description is listed in the tool overview of the library.

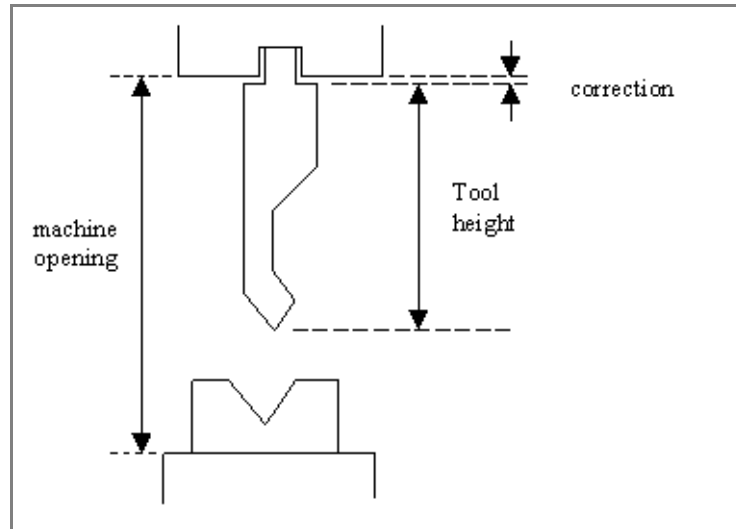
Support type

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'.



If 'shoulder mounted' is chosen, the Y-axis position is calculated from the standard tool height. This is the default setting.

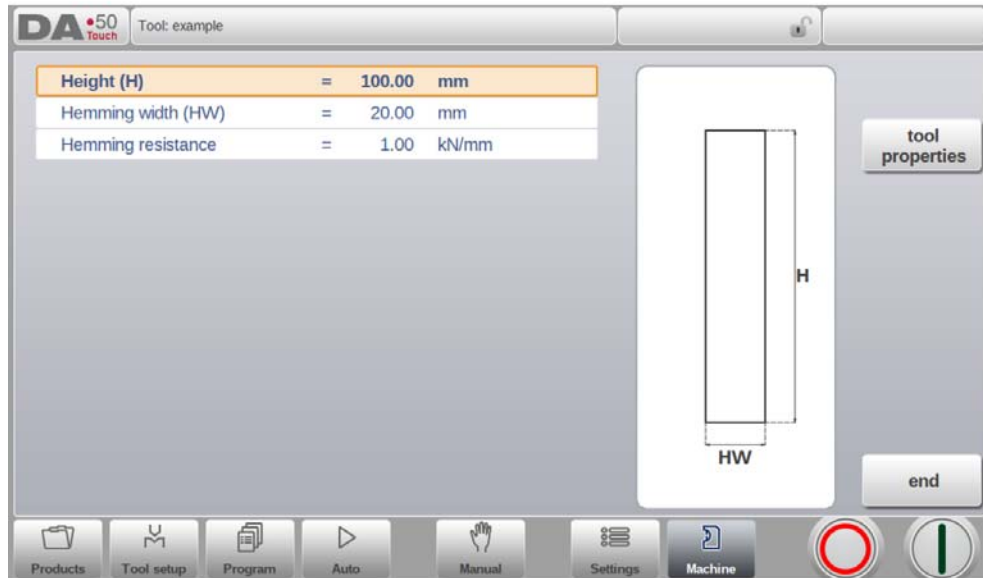
If 'head mounted' is chosen, a correction is made for Y-axis computation



Edit punch

To edit an existing tool, tap the tool in the library. The tool appears on the screen and can be edited with the programming facilities.

8.2.3. Hem bend punch



Height

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

Hemming width

The width of the tool to program.

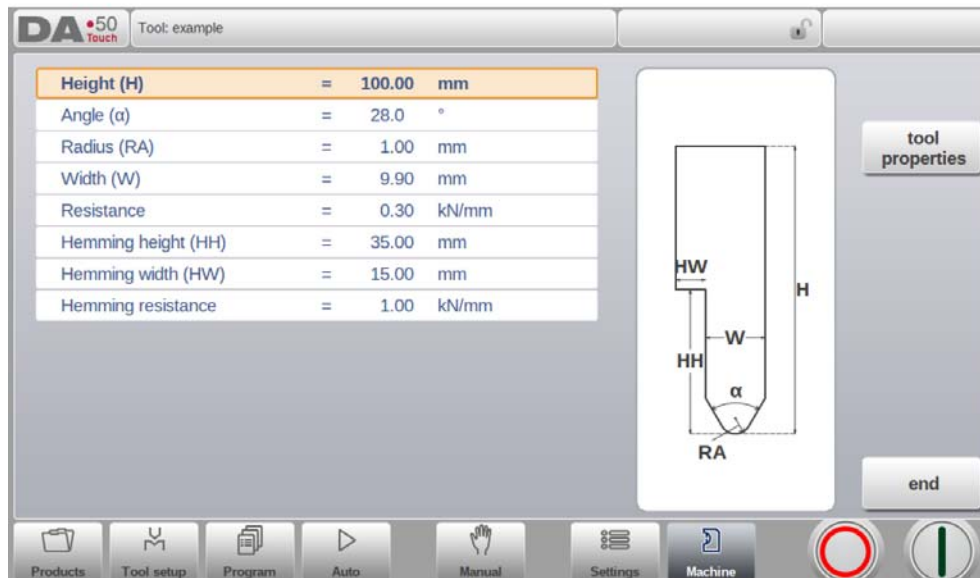
Hemming load opening

Depending on the construction of your machine you can program here an opening position for your punch at which position you can put in your product to hem the particular bend. The opening position will also take twice the sheet thickness into account.

Hemming resistance

Maximum allowable force on the tool during hemming.

8.2.4. Air + hem bend punch



Height

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

Angle

The angle of the punch tip.

Radius

The radius of the punch tip. This value will be used as inner radius of the bend to make when this radius value is bigger than the inner radius as will result from the bending process.

Width

The width of the tool to program.

Resistance

Maximum allowable force on the tool.

Hemming height

The height of the punch used to move down in case of the hemming function.

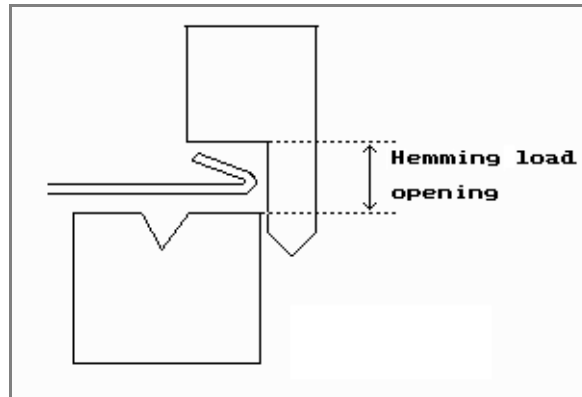
Hemming width

The width of the punch part used to lay in the product for the hemming action.

Hemming load opening

Depending on the construction of your machine you can program here an opening

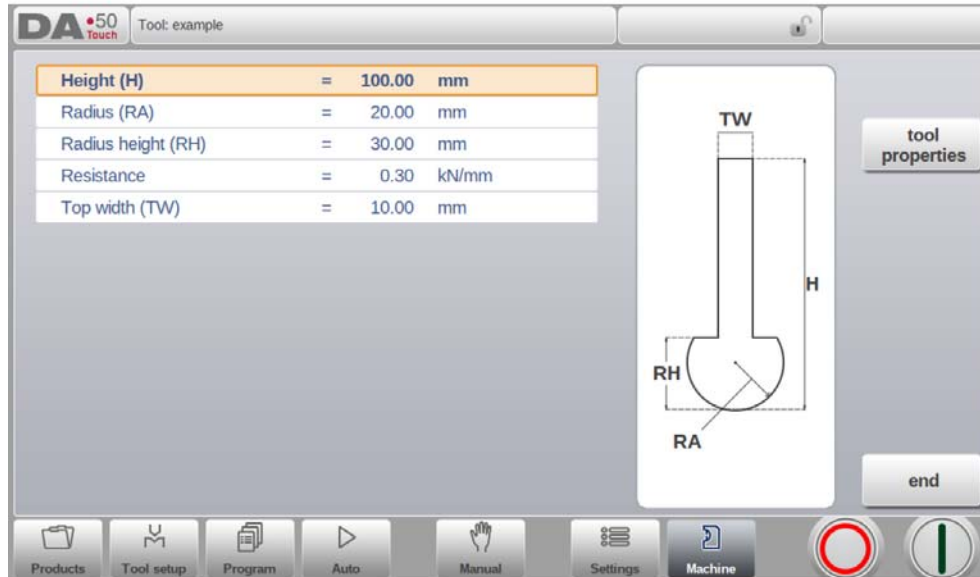
position for your punch at which position you can put in your product to hem the particular bend. The opening position will also take twice the sheet thickness into account.



Hemming resistance

Maximum allowable force on the tool during hemming.

8.2.5. Big radius punch



Height

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

Radius

The radius of the punch tip.

Radius height

The height of the big radius part of the tool.

Resistance

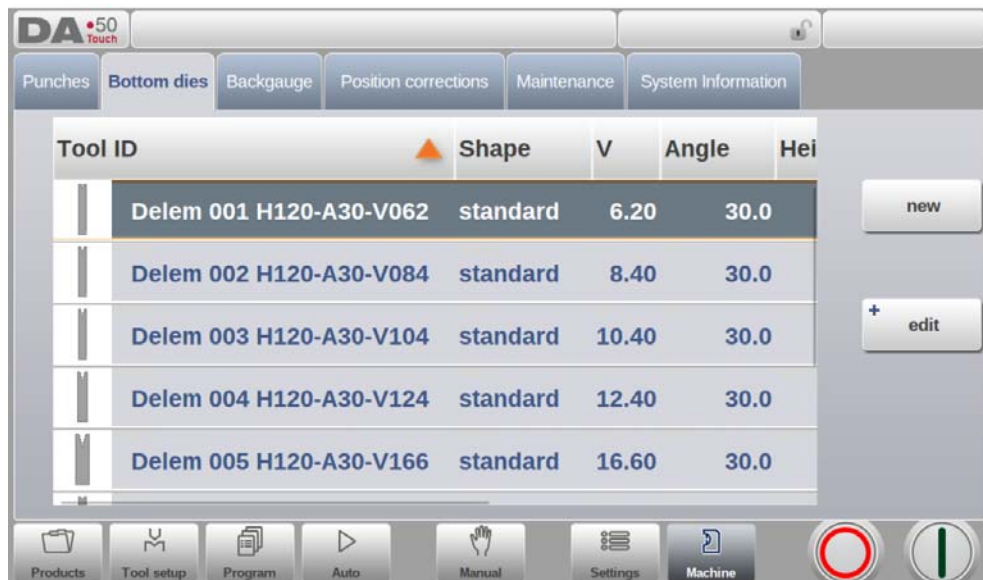
Maximum allowable force on the tool.

Top width

The width of the tool on the top side of the punch.

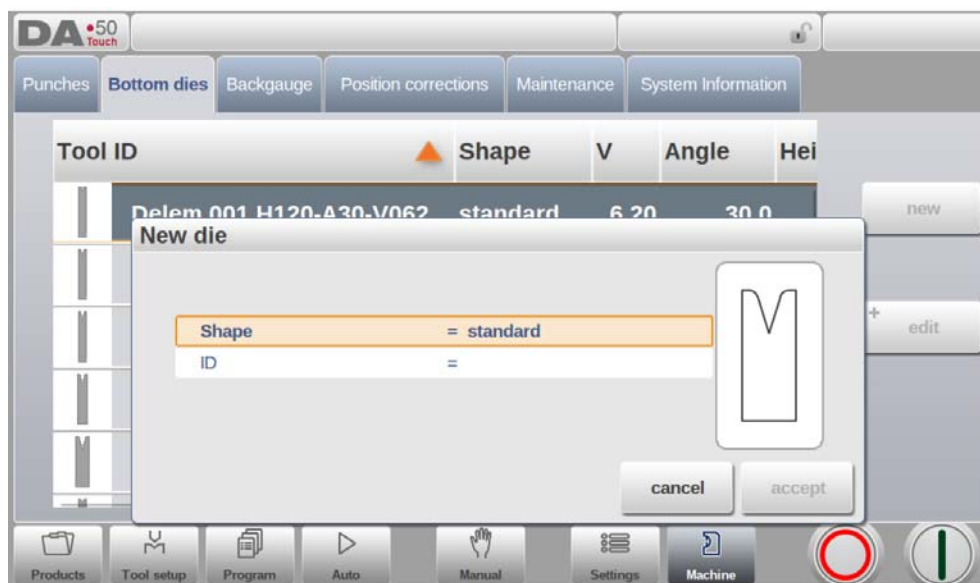
8.3. Programming of bottom dies

In this tab, the bottom dies used in the machine can be programmed. New dies can be added; existing dies can be edited, copied, renamed and deleted.



8.3.1. Create a new die

To create a new die, tap New in the library. The die profile can be created with help of the programming facilities of the control.



First the shape of the die and its ID must be programmed. After that the shape details must be programmed following the wizard.

Shape

A selection must be done from the different available basic die shapes corresponding to the required die action. Options are:

- Standard Die shape as normally used for air bending and basic bottoming.
- Hem Bend Die as typical tool with flat top for specific hem bends.
- Inside Hem Bend Die as tool for air bends and hem function.
- Air + Hem Bend U Die shape for air bends and specific hem function.

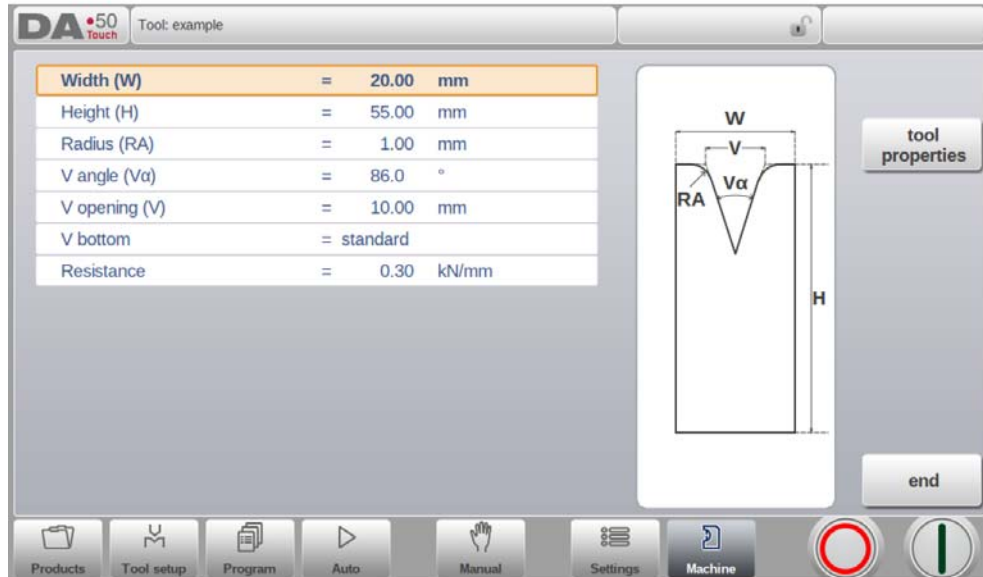
ID

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

The ID parameter may also contain alphanumeric characters. When finished, use Accept to leave this window and start with the tool data parameters. The control will prompt a window for the tool properties with the dimensions of the tool.

Depending on the chosen basic shape, the parameters will differ. In the following paragraphs the available die shapes are described in detail.

8.3.2. Standard die



Width

The width of the tool to program.

Height

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

Radius

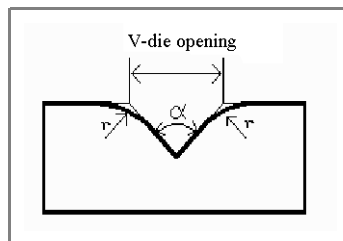
The radius of the edges of the V-opening.

V angle

The angle of the die.

V opening

The V-opening of the die.



The V-opening is the distance between the touching lines crossing.

V bottom

Herewith the different possible bottoms inside the V opening can be defined:

- Standard is a sharp angle in the bottom of the die.
- Round is a die bottom with a radius to be programmed with the parameter 'Inside radius'.
- Flat is a flat die bottom with a certain dimension to be set with the parameter 'Bottom width'.

Resistance

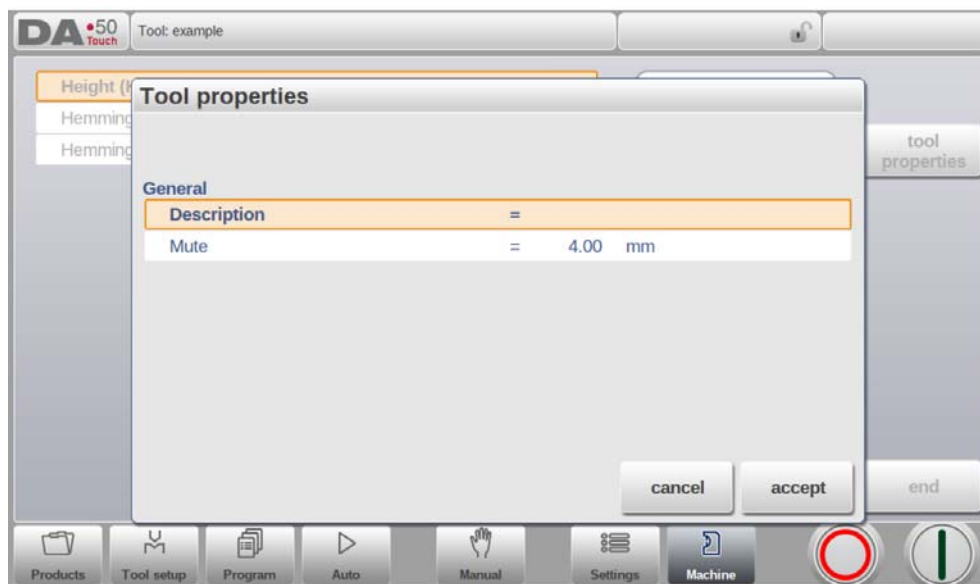
Maximum allowable force on the tool.

Orientation of the die on the screen

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

Tool Properties

To change the generic tool data and description.

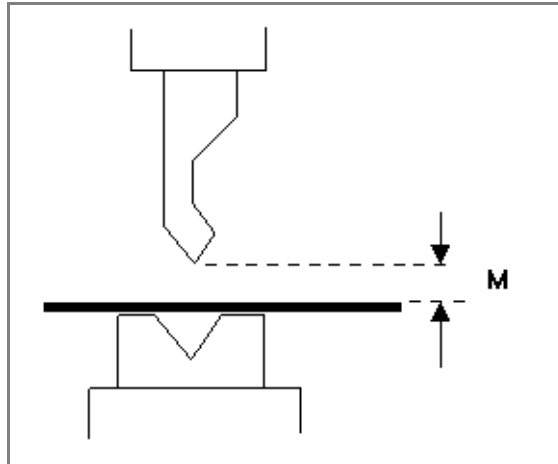


Description

A name or description of this tool. The maximum length is 25 characters. This description has already been entered in the beginning defining the tool, but can be edited in this field. The description is listed in the tool overview of the library.

Mute

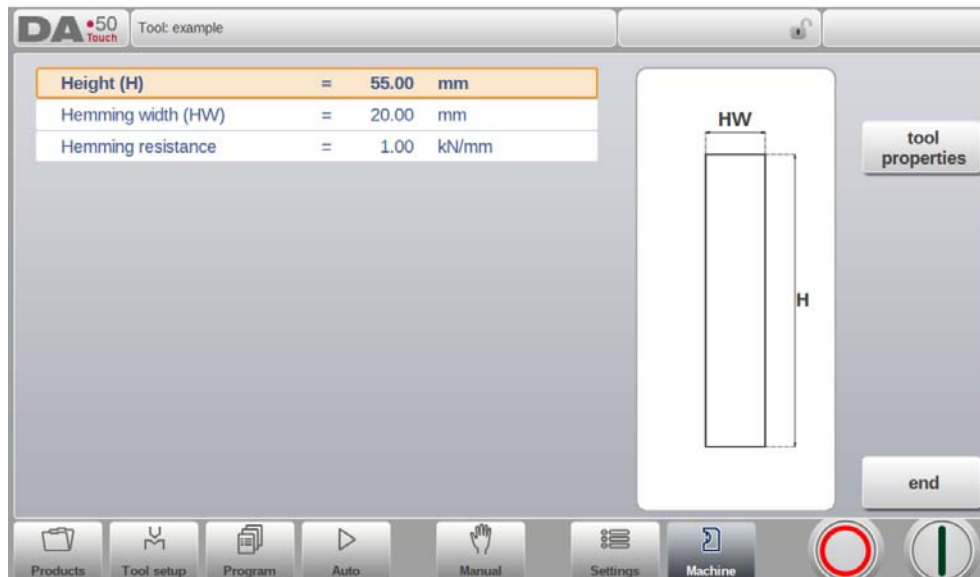
Muting distance. Distance above the sheet at which the speed change takes place.



Edit die

To edit an existing tool, tap the tool in the library. The tool appears on the screen and can be edited with the programming facilities.

8.3.3. Hem bend die



Height

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

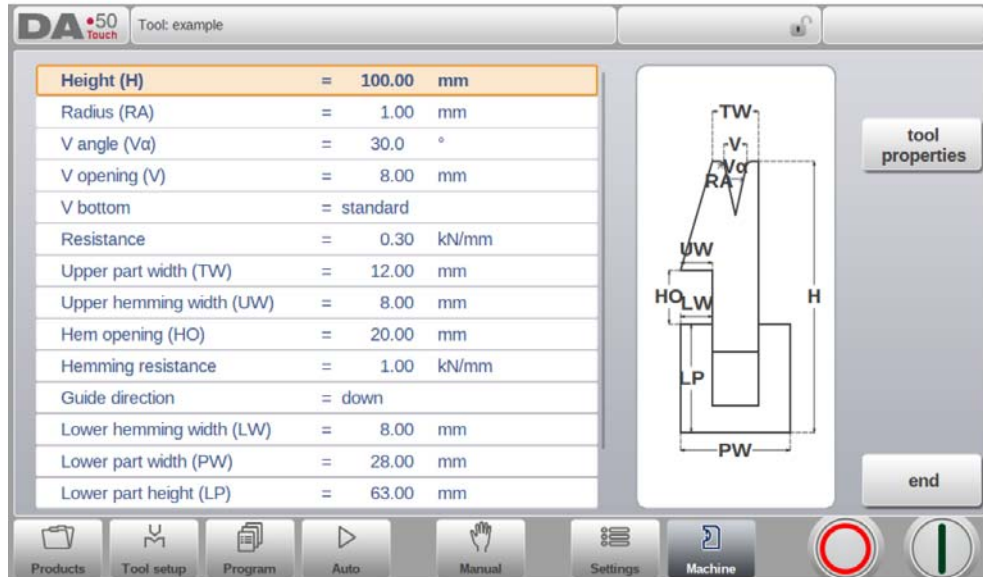
Hemming width

The width of the tool to program.

Hemming resistance

Maximum allowable force on the tool during hemming.

8.3.4. Inside hem bend die



Height

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

Radius

The radius of the edges of the V-opening.

V angle

The angle of the die.

V opening

The V-opening of the die.

V bottom

Herewith the different possible bottoms inside the V opening can be defined:

- 'Standard' is a sharp angle in the bottom of the die.
- 'Round' is a die bottom with a radius to be programmed with the parameter 'Inside radius'.
- 'Flat' is a flat die bottom with a certain dimension to be set with the parameter 'Bottom width'.

Resistance

Maximum allowable force on the tool.

Upper part width

The width of the upper part of the die.

Upper hemming width

The width of the segment in the upper part of the die used for the hemming action.

Hem opening

The opening height of the die in the opened status to place the product with the hem bend.

Hemming resistance

Maximum allowable force on the tool during hemming.

Guide direction

Defines the mechanical construction of the inside hem die.

Lower hemming width

The width of the segment in the lower part of the die used for the hemming action.

Lower part width

The width of the lower part of the die.

Lower part height

The height of the lower part of the die.

Inside hemming die type

For the different types of available inside hemming dies the specific operation mode can be selected:

- Spring Opened This die has an internal spring to push the die upwards as start position.
Pre-Bend: For the pre-bend the sheet will be placed on top of the die in this opened position and the final bend position is calculated such that the required pre-bend angle will be reached. The backgauge height (R-axis) will be calculated on the level of the opened die so that the product can be placed against the backstop.
Hem-Bend: For the hemming operation the pre-bent product will be placed in the hem opening of the die and the depth calculation of the pressing beam will be such that the die is pressed down until the hem bend is finished. The calculation will take twice the sheet thickness into account. It is possible to program an additional hem opening parameter in the specific bend parameters as offset on the final hem position.
- Open & Locked This die is standard locked in a high position for normal bending and pre-bend angles. The lock mechanism must be removed to use the hemming facility.
- Normally Closed This die is in a closed, low position for normal bending and pre-bend angles, and should be activated to get the hemming action.

Adapt decompression

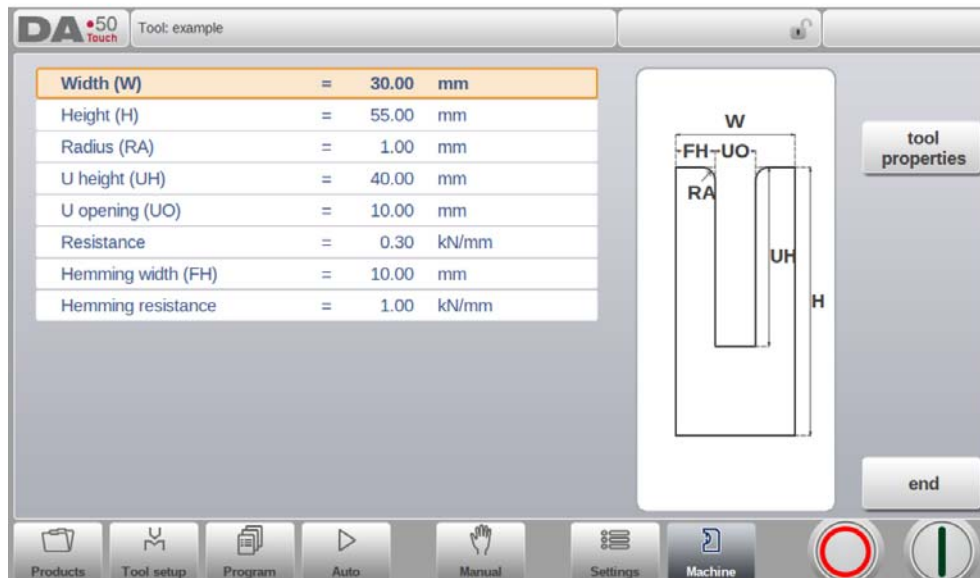
To enable the addition of the hem opening value to the decompression distance.

no => not added at all.

yes => added for both air bends and hem bends.

air bend => added for air bends only (only available for spring opened hemming dies).

8.3.5. Air + hem bend U die



Width

The width of the tool to program.

Height

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

Radius

The radius of the edges of the U-opening.

U height

The height of the U-opening of the die.

U opening

The width of the U-opening of the die.

Resistance

Maximum allowable force on the tool.

Hemming width

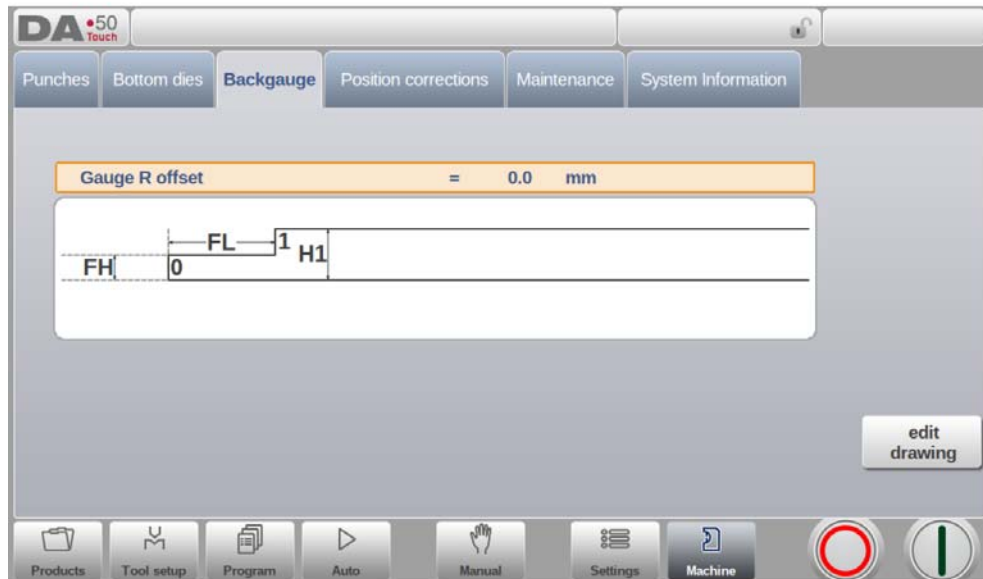
The front width of the die, meant as support for the hem bend.

Hemming resistance

Maximum allowable force on the tool during hemming.

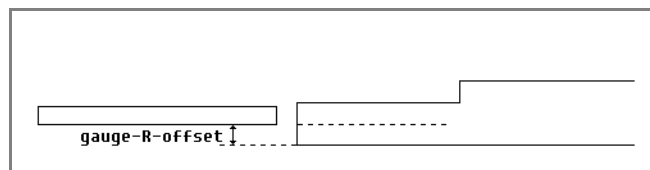
8.4. Backgauge

With the backgauge finger dimensions the R-axis movement and related X-axes movement is taken into account. Also the workpiece / backgauge collision is computed using the dimensions.



Gauge R offset

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

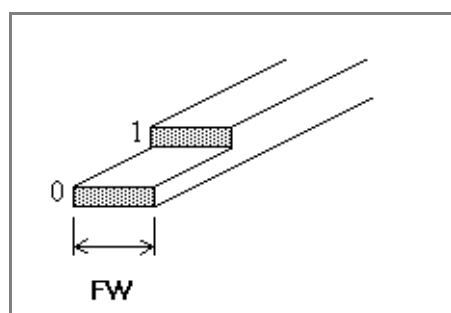


A negative value gives a lower backgauge position. This offset is only valid for gauge position 0.

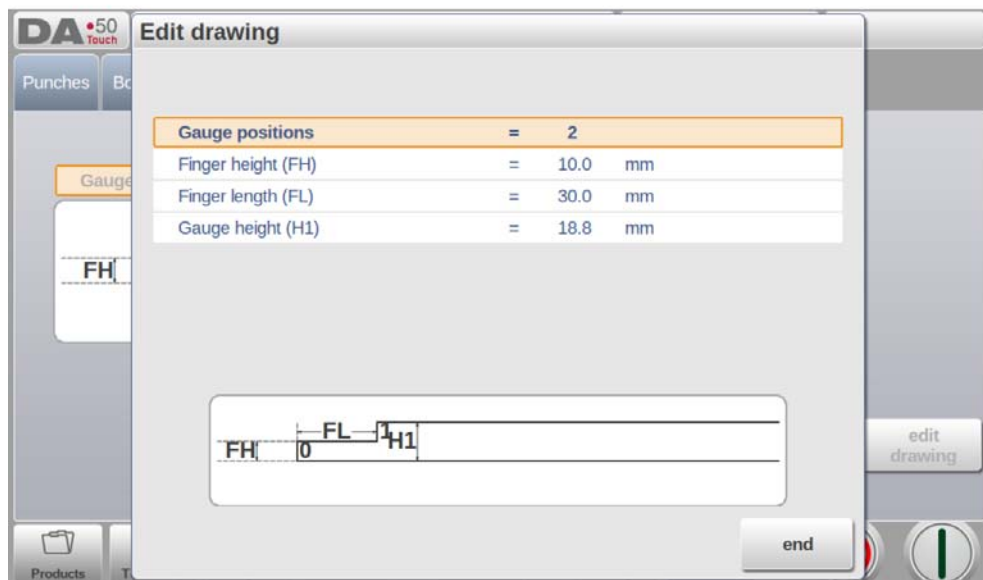
Finger width

The width of the backgauge finger. Only available when an automatic Z-axis has been

installed.



Tap Edit Drawing to make the backgauge drawing appear wherein the dimensions of the backgauge finger can be programmed.



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

Gauge positions

The number of possible gauge positions (max. 4). When this parameter is changed, a new finger geometry appears, and the finger dimensions can be programmed.

Finger height (FH)

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

Finger length (FL)

The length of the first lay-on level.

Gauge height (H1)

The height of the first lay-on level.

8.5. Position corrections



Position correction

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

Example:

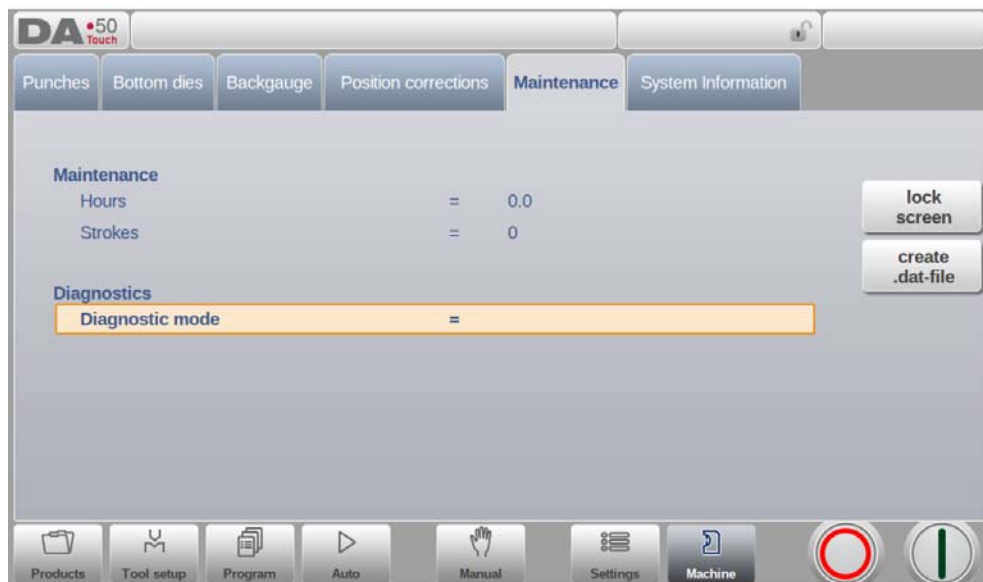
- When the programmed and displayed value = 250 and the actual, mechanical position value = 252 the correction parameter = -2.
- When the programmed and displayed value = 250 and the actual, mechanical position value = 248 the correction parameter = +2.

Position corrections are available for all auxiliary axes.

Position corrections should only be used temporarily. In case machine positions have changed after commissioning or service, positions of axes can be corrected. In normal situations these corrections should be 0.

8.6. Maintenance

On this tab maintenance related functions are located. Next to the machine hour counter and the machine stroke counter also functions to help replace modules and to store diagnostic data can be found here.



Hours

The number of hours the machine is running.

Strokes

The number of strokes the pressbeam has executed.

Diagnostic mode

Activate or de-activate a diagnostic mode for service purposes. The diagnostic mode is activated by programming a special code here. The diagnostic mode is de-activated by programming 0.

To activate the diagnostic mode, contact the machine manufacturer for more information about this mode and an access code.

Lock screen

To lock the screen, and e.g. clean the screen without changing anything, the Lock Screen function can be used. The screen is unlocked automatically after 5 seconds.

Create .dat-file

Tapping Create .dat-file will store the most important product and control data, by default on the connected USB stick. This information can be helpful for the maintenance support.

8.7. System information

On this tab system information can be found. Next to version information on the software also ID's of the installed modules and version of OEM-specific files can be read.

Next to information also software update functionality is available here.



This screen shows detailed information about the control system. This information is useful for service purposes.

Application

The version of the current application

Option ID

The unique option ID of the control

Sequencer

The version number of the running sequencer

Delem.def

The version number of the running delem.def file

Update software

With Update Software the control can install a software update set from a USB stick. The directory browser will help to select the desired update and initiate the installation process.

Backup system

The backup system function makes a complete system backup to a USB stick. A unique timestamped file is written on the USB stick. This backup holds Delem software, OEM specific data as well as the user's files.

Restore system

The restore system function can be used to restore an earlier made backup of the system. During the process selection of what will be restored can be done.

Offline software

The offline software function generates an offline software setup file on a USB stick. This setup can be used to update an existing offline software. Using the matching offline software version with the control software ensures optimal compatibility of functions.

A. Parameter index

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

Adapt decompression	8.19
Angle	4.15
Angle	4.5
Angle	6.2
Angle	8.4
Angle	8.8
Angle / Angle 2	5.3
Angle 1 / Angle 2	5.2
Angle correction database	7.15
Angle correction programming	7.16
Angle sel.	4.11
Application	8.26
Auto bend change mode step	7.22
Auxiliary axis	4.7
Auxiliary axis	5.2
Auxiliary axis	6.5
Bending length	4.4
Bending length	6.2
Bottoming force factor	7.20
Bumping	4.15
Clamping correction	7.23
Code	4.6
Command panel side	7.3
Corr.α 1, Corr.α 2	6.2
Corr.Y	6.3
Data preparation bend allowance	7.20
Date format	7.25
Decomp speed	4.6
Decomp speed	6.4
Decompression	4.6
Decompression	6.4
Default decompression speed	7.18
Default delay time	7.18
Default dwell time	7.19
Default hem opening	7.19
Default material	7.19
Default pressing speed	7.18
Default step change code	7.18
Default wait for retract	7.18
deflect	5.3
deflect	5.3
Delay time	4.6
Delem.def	8.26
Depth	5.3
Description	8.14
Description	8.5
Diagnostic mode	8.25
Die	4.7
Die	6.5
Display time	7.25
Dwell time	4.6

Dwell time	6.4
E module	7.4
Effective bend length	4.4
Equal bumping-segments	4.15
Finger height (FH)	8.23
Finger length (FL)	8.23
Finger width	8.21
Force	4.6
Force	6.4
Gauge height (H1)	8.23
Gauge positions	8.23
Gauge R offset	8.21
General angle correction programming	7.16
Guide direction	8.18
Height	8.10
Height	8.13
Height	8.16
Height	8.17
Height	8.20
Height	8.4
Height	8.7
Height	8.8
Hem opening	4.5
Hem opening	6.2
Hem opening	8.18
Hemming height	8.8
Hemming load opening	8.7
Hemming load opening	8.8
Hemming resistance	8.16
Hemming resistance	8.18
Hemming resistance	8.20
Hemming resistance	8.7
Hemming resistance	8.9
Hemming width	8.16
Hemming width	8.20
Hemming width	8.7
Hemming width	8.8
Hours	8.25
ID	8.12
ID	8.3
Inch/mm-select	7.2
Initial angle correction	7.16
Inside hemming die type	8.18
Intermediate R for X-movement	7.23
Intermediate X for Z-movement	7.23
Key sound	7.2
Keyboard layout	7.2
Language	7.2
Layon	4.7
Lock touch screen when started	7.23
Lower hemming width	8.18
Lower part height	8.18
Lower part width	8.18
Machine description	7.17
Machine ID	7.17
Machine ID check	7.17
Manual mode store angle corrections	7.16
Material	4.12

Material	4.7	Thickness	4.7
Material	6.4	Thickness	5.3
Material name	7.4	Thickness	6.4
Message sound	7.3	Time format	7.25
Method	4.3	Ton/kN select	7.2
Method	6.2	Top width	8.10
Mute	4.5	U height	8.20
Mute	6.3	U opening	8.20
Mute	8.14	Upper hemming width	8.18
Number of segments	4.15	Upper part width	8.17
Opening	4.5	V angle	8.13
Opening	6.3	V angle	8.17
Option ID	8.26	V bottom	8.14
Parallel	4.5	V bottom	8.17
Parallel	6.3	V opening	8.13
Parallelism offset	7.22	V opening	8.17
Position correction	8.24	Wait for retract	4.6
Pressure correction	7.23	Wait for retract	6.4
Product description	4.11	Width	8.13
Product ID	4.11	Width	8.20
Product position	4.5	Width	8.4
Product position	6.2	Width	8.8
Punch	4.7	X correction programming	7.16
Punch	6.5	X-axes	5.3
Radius	4.15	X-safety offset	7.23
Radius	8.10	Y opening default	7.18
Radius	8.13	Y1 / Y2	5.2
Radius	8.17	Y1/Y2 independent	7.17
Radius	8.20	Y-axis	6.3
Radius	8.4	Y-axis (Bend position)	4.5
Radius	8.8	Z-distance	7.20
Radius height	8.10		
Repetition	4.6		
Repetition	5.3		
Resistance	8.10		
Resistance	8.14		
Resistance	8.17		
Resistance	8.20		
Resistance	8.4		
Resistance	8.8		
Resistance per m/mm selection	7.2		
Retract	4.7		
Retract	6.5		
Sequencer	8.26		
Shape	8.12		
Shape	8.3		
Speed	4.6		
Speed	4.7		
Speed	6.4		
Speed	6.5		
Step mode	5.3		
Stock	5.3		
Stock count mode	7.22		
Strain hardening exponent	7.4		
Strokes	8.25		
Support type	8.5		
Tensile strength	7.4		
Thickness	4.11		
