Baykal

Think Big, We Do.

USER GUIDES

Delem

DA-69S

DA-69S

Reference Manual

Operation of Version 4.1
English

Preface

This manual describes the operation of the Delem control type DA-69S 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
V4.1	V0623	first issue V4

This manual is valid for software version 4.1 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. Create a drawing 1.7
1.5.4. Determine bend sequence 1.9
1.5.5. Numerical program 1.9
1.5.6. The Auto menu and Manual menu, production modes 1.10
1.5.7. Back-up data, external storage 1.10
1.6. Programming aids 1.11
1.6.1. Help text 1.11
1.6.2. Listbox functionality 1.12
1.6.3. Live search 1.12
1.6.4. Auto filter 1.13
1.6.5. Navigation 1.14
1.6.6. Text input and editing 1.14
1.6.7. Typing alphanumeric characters vs. special characters 1.15
1.6.8. Calculator 1.16
1.6.9. Messages center 1.17
1.6.10. Network 1.17
1.6.11. Key-lock function 1.18
1.6.12. OEM function panel 1.19
1.6.13. Multi user system 1.20
1.6.14. Software versions 1.21

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 2.3
2.1.3. New Product, starting a new graphical product 2.4
2.1.4. New Program, starting a numerical program 2.5
2.1.5. Views 2.6
2.1.6. Product Copy and Delete 2.8
2.1.7. Product Rename and Move 2.9
2.1.8. Product Lock/Unlock 2.9
2.1.9. Selection of multiple products 2.9
2.1.10. Search function 2.10
2.1.11. Filter function 2.11
2.1.12. Change directory 2.12
2.1.13. Network product selection 2.13
2.2. Shop floor control, Job List function 2.14
2.3. The DXF import function (only available when DXF option has been installed) . . 2.16
2.3.1. Product drawing dimensions 2.19
Projection dimensions 2.19
Cutting dimensions 2.19
2.3.2. Bendlines and Layer selection with Line assignment 2.20
Bend line information 2.20
Product information 2.20
Layer selection 2.20

Layer selection switched on 2.21
Layer selection switched off 2.22
Line select 2.23
2.3.3. Conversion 2.25
Product Properties 2.26
2.3.4. Converting cutting dimensions, with bend allowance info 2.27
Adding Bend Allowance information manually 2.28
2.3.5. DXF Settings 2.29
Conversion parameters 2.29
Labels 2.30
Control codes 2.30
The DXF file 2.31
2.4. The 3D import function (Profile-T3D offline only) 2.32
2.4.1. Conversion 2.33
The 3D CAD file requirements 2.35
2.5. The DXF contour export function (only available when DXF option has been installed) 2.36

3. Product drawing 3.1

3.1. General product properties 3.1
3.1.1. Add Notes 3.5
3.2. 2D product drawing 3.6
3.2.1. Introduction 3.6
3.3. Line properties 3.8
3.3.1. Introduction 3.8
3.3.2. Projection 3.8
3.3.3. Precision selection 3.9
3.3.4. Insert / delete bend 3.10
3.4. Bend properties 3.11
3.4.1. Air bend 3.11
3.4.2. Bumping 3.12
3.4.3. Hem bends 3.14
3.4.4. Prebend, two stage bending 3.15
3.4.5. Insert temporary bend 3.15
3.5. Bend labels (DA-6xTe only) 3.17
3.6. Surface or bend line marker 3.18
3.7. Foil side indication 3.19
3.8. Measurement function in graphical views 3.20
3.8.1. Change length function 3.20
3.9. Variable product dimensions 3.22
3.9.1. Variable line properties 3.23
3.10. 3D drawing 3.24
3.10.1. Start 3D 3.24
3.10.2. 3D unfolded product 3.25
3.10.3. Edit or rotate the product 3.25
3.11. Edit a 3D drawing 3.27
3.11.1. Introduction 3.27
3.11.2. Add surface 3.27
3.11.3. Delete surface 3.27
3.11.4. Changing the length of a side 3.28
3.11.5. Changing the angle between two surfaces. 3.32

4. Tool configuration 4.1

4.1. Introduction 4.1
4.2. Standard procedure 4.1
4.3. Tool selection 4.2
Auto select 4.4
Multi-V, die feature preselect 4.4

Vario-V, selecting and setting . . . 4.4
Positioning or repositioning . . . 4.5
4.4. Tool segmentation . . . 4.7
4.5. Segmentation of individual tools . . . 4.8
4.5.1. Tool view . . . 4.8
Assignments . . . 4.9
4.5.2. Segmentation view . . . 4.9
4.5.3. Segments in the tool library . . . 4.10
4.6. Station selection and repositioning . . . 4.12

5. Bend Sequence . . . 5.1

5.1. Introduction . . . 5.1
5.1.1. View select . . . 5.4
5.1.2. Bend selector . . . 5.5
5.2. Unbend product . . . 5.6
5.3. Manual selection of bends . . . 5.7
5.3.1. Shift product . . . 5.8
5.3.2. Shift gauge . . . 5.9
5.4. Assignments . . . 5.12
5.4.1. Introduction . . . 5.12
5.4.2. Assignments - general . . . 5.13
5.4.3. Assignments - Backgauge possibilities . . . 5.16
5.5. Simultaneous bends, group selection . . . 5.19
5.6. Show bend sequence . . . 5.20

6. Product programming . . . 6.1

6.1. Introduction . . . 6.1
6.2. Program mode, parameter explanation . . . 6.3
6.2.1. Bend parameters . . . 6.4
6.2.2. Force . . . 6.6
6.2.3. Speed . . . 6.7
6.2.4. Functions . . . 6.7
6.2.5. Tools . . . 6.8
6.2.6. Gauge positions . . . 6.8
6.2.7. Auxiliary axes . . . 6.8
6.2.8. Part support . . . 6.8
6.3. Edit / view modes . . . 6.11
6.3.1. All Bends . . . 6.11
6.3.2. Change tools . . . 6.13
6.3.3. Shift product . . . 6.14
6.3.4. Shift gauge . . . 6.15
6.3.5. Product properties . . . 6.18
6.3.6. Notes . . . 6.21
6.3.7. Add bend . . . 6.23
6.3.8. Bumping . . . 6.24
6.4. Programming parameters . . . 6.26

7. Automatic mode . . . 7.1

7.1. Introduction . . . 7.1
7.1.1. Auto mode, parameter explanation . . . 7.2
Corrections . . . 7.2
General corrections . . . 7.3
General . . . 7.4
7.2. View modes . . . 7.5
7.2.1. Main . . . 7.6
Bend selector . . . 7.6
7.2.2. All bends . . . 7.7

7.2.3. Graphical . . . 7.8
7.2.4. Macro . . . 7.10
7.2.5. Manual positioning . . . 7.11
External hand terminal (optional) . . . 7.11
7.2.6. Corrections . . . 7.12
Calculate corrections, programming of measured angles . . . 7.13
7.2.7. Tool setup . . . 7.14
7.2.8. Diagnostics . . . 7.15
7.3. Notes . . . 7.16
7.4. Bumping/chaining correction . . . 7.17
7.5. Test Bend mode . . . 7.18
7.6. Shop floor control, Job List function . . . 7.19

8. Manual mode . . . 8.1

8.1. Introduction . . . 8.1
8.1.1. Manual mode, parameter explanation . . . 8.2
Bend parameters . . . 8.2
Force . . . 8.4
Speed . . . 8.4
Functions . . . 8.5
Product properties . . . 8.5
Tools . . . 8.5
Gauge positions . . . 8.6
Auxiliary axes . . . 8.6
Part support . . . 8.6
8.1.2. Tool setup . . . 8.8
8.1.3. Shift product . . . 8.9
8.2. Programming parameters & Views . . . 8.10
View . . . 8.10
8.3. Macro . . . 8.12
8.4. Manual movement of the axes . . . 8.13
8.4.1. Movement procedure . . . 8.13
8.4.2. Teach . . . 8.14
8.5. Corrections . . . 8.15
8.6. Test Bend mode . . . 8.17
8.7. Diagnostics . . . 8.18
8.7.1. IO status . . . 8.19
Zoomed IO . . . 8.20

9. Settings . . . 9.1

9.1. Introduction . . . 9.1
9.2. General . . . 9.2
9.3. Materials . . . 9.4
9.4. Backup / restore . . . 9.7
9.4.1. Product backup . . . 9.8
9.4.2. Product restore . . . 9.10
9.4.3. Tool backup . . . 9.11
9.4.4. Tool restore . . . 9.12
9.4.5. Backup and restore for Tables and Settings . . . 9.12
9.4.6. Automatic scheduled backups . . . 9.12
9.4.7. Directory navigation . . . 9.13
9.5. Program settings . . . 9.14
9.6. Default values . . . 9.17
9.7. Computation settings . . . 9.19
9.8. Production settings . . . 9.22
9.9. Production time calculation . . . 9.25
9.10. Time settings . . . 9.26

9.11. Network settings 9.27

10. Machine 10.1

10.1. Introduction 10.1
10.2. Programming of Punches 10.2
10.2.1. View 10.3
10.2.2. Create a new punch 10.5
10.2.3. Standard punch 10.6
10.2.4. Hem bend punch 10.10
10.2.5. Air + hem bend punch 10.11
10.2.6. Big radius punch 10.13
10.3. Programming of Dies 10.15
10.3.1. View 10.16
10.3.2. Create a new die 10.17
10.3.3. Standard die 10.19
10.3.4. Hem bend die 10.23
10.3.5. Inside hem bend die 10.25
10.3.6. Air + hem bend U die 10.28
10.3.7. Multi V die 10.30
10.3.8. Vario V die (only available if a Vario-V system is present) 10.33
10.3.9. Multi-V inside hem die 10.36
10.3.10. Doorframe hem die 10.39
10.3.11. Wingbend die 10.42
10.4. Frame 10.45
10.4.1. C-frame contour editor 10.47
10.5. Adapters 10.48
10.6. Backgauge 10.50
10.6.1. Backgauge finger library 10.53
10.7. Position corrections 10.54
10.8. Press beam 10.55
10.9. Table 10.58
10.10. Drawing functionality for tools, adapters and machine shapes 10.60
10.11. DXF import for tools, adapters and machine parts (only available when DXF option has been installed) 10.63
10.12. Protractor 10.65
10.13. Event logging 10.67
10.13.1. Parameters 10.67
10.13.2. Explanation 10.69
10.14. Maintenance 10.71
10.15. Scheduled maintenance (optionally available) 10.73
10.16. System information 10.74

11. Profile-60S 11.1

11.1. Introduction 11.1
11.1.1. General 11.1
11.1.2. System requirements 11.1
11.1.3. Profile-60S distribution & manuals 11.1
11.2. Profile-60S installation 11.2
11.3. Profile-60S operation 11.6
11.3.1. Profile-60S principle 11.6
11.3.2. Machine library 11.6
11.3.3. Print functionality (Profile-S2D and Profile-S3D only) 11.8
11.3.4. Transfer mode 11.9
11.3.5. Key combinations 11.10

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.

Beside the touch controls the front of the control consists of an emergency stop, the hand wheel and the start and stop buttons.

Special function keys, which can be mounted in the top panel of the control, have their specific description parallel to this user manual and will be supplied by the machine manufacturer. This user manual focuses on the control software and related machine functions.

1.2. Front control elements

The front panel, beside display, consists of the following control elements:

Emergency stop button, to be implemented by machine manufacturer.

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

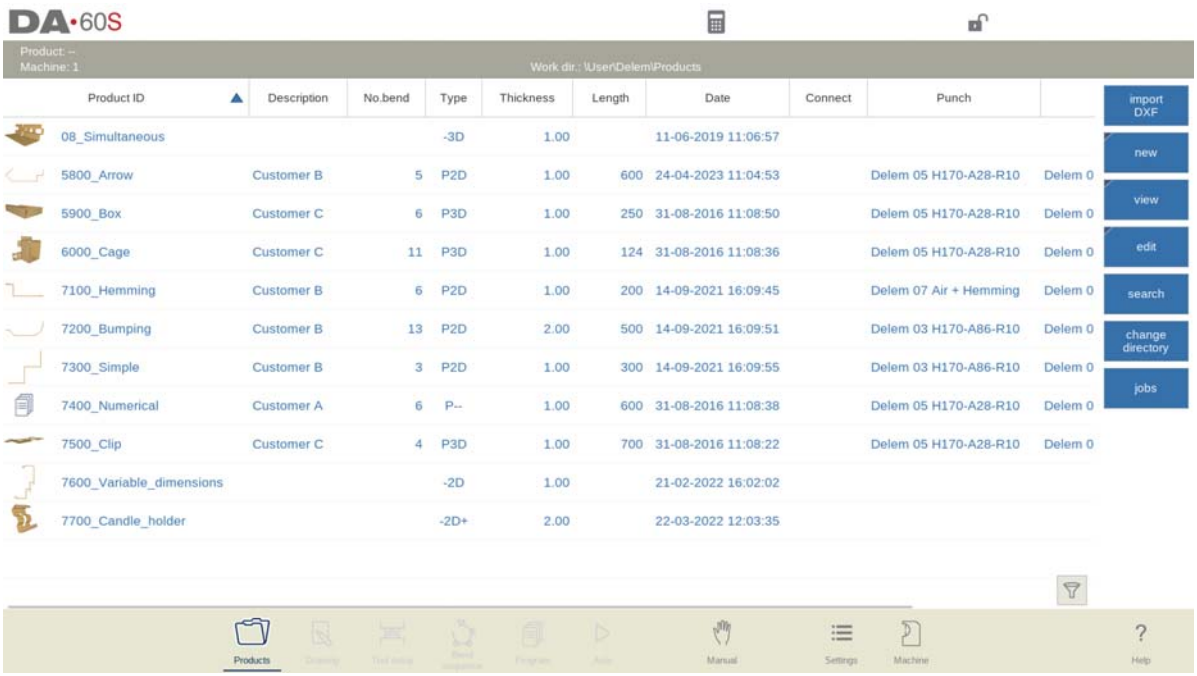
Stop button and Start button

1.3. USB connectors

At the right side of the control two USB ports are available for connection of external devices, such as memory sticks or an external keyboard / 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.

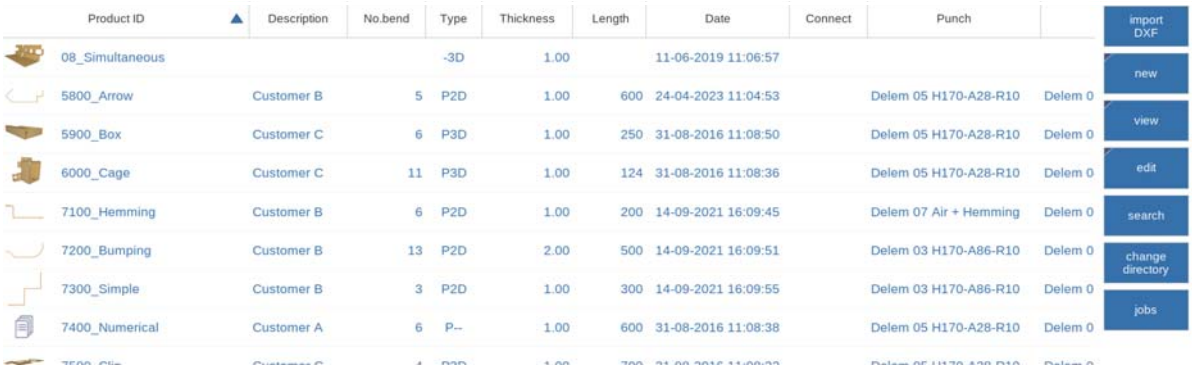
The structure of the main screen is as follows:

Title panel

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

Information panel

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



Product ID	Description	No.bend	Type	Thickness	Length	Date	Connect	Punch	
08_Simultaneous			-3D	1.00		11-06-2019 11:06:57			
5800_Arrow	Customer B	5	P2D	1.00	600	24-04-2023 11:04:53		Delem 05 H170-A28-R10	Delem 0
5900_Box	Customer C	6	P3D	1.00	250	31-08-2016 11:08:50		Delem 05 H170-A28-R10	Delem 0
6000_Cage	Customer C	11	P3D	1.00	124	31-08-2016 11:08:36		Delem 05 H170-A28-R10	Delem 0
7100_Hemming	Customer B	6	P2D	1.00	200	14-09-2021 16:09:45		Delem 07 Air + Hemming	Delem 0
7200_Bumping	Customer B	13	P2D	2.00	500	14-09-2021 16:09:51		Delem 03 H170-A86-R10	Delem 0
7300_Simple	Customer B	3	P2D	1.00	300	14-09-2021 16:09:55		Delem 03 H170-A86-R10	Delem 0
7400_Numerical	Customer A	6	P--	1.00	600	31-08-2016 11:08:38		Delem 05 H170-A28-R10	Delem 0

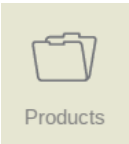
Command panel

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

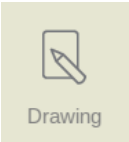
Navigation panel

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

Explanation of the main modes / navigation buttons



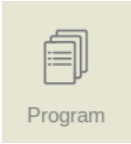
To make a new program and select a program out of the product library.



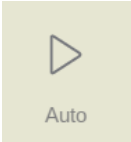
To draw/create a new product or edit an existing product (graphically).

To setup the machine and modify existing tool setups.

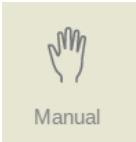
To compute and modify the bend sequence.



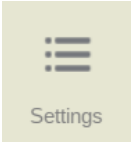
To create a new CNC-program or edit an existing CNC-program numerically.



To start production of the selected product/program.

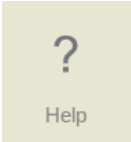


To program all settings for making one single bend, not related to a program.



User settings and preferences applicable to the programming of new products and 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.



Wherever you are in the control, via this button you can access the help system which will provide you with context sensitive information.

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 product drawing and calculate a valid bend sequence for the product. With this information, a product program is generated.
This is done with the following steps:

1. Go to the Products mode in the navigation panel and start a new product by tapping New Product.
2. Enter the product properties and start to draw a 2D product profile in the Drawing mode.
3. Check the tooling, modify or make a new set-up in the Tool Setup mode.
4. Use the Bend Sequence mode to determine the bend sequence by calculating it or manually modifying it upon your own idea's.
5. When required, modify the numerical CNC program via the Program mode.
6. 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 Material 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. Create a drawing

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

The drawing method on the Touch screen control is based on:

- Sketching
- Value setting

Sketching

For drawing of products as well as tool shapes, sketching can be done by tapping on the screen in the different directions the drawing must have. The application will follow the tapping with drawing a line between the indicated points. The last point of the design will show always a big red dot.

When the drawing dot is on the screen you can hold your finger on this position and move the finger across the screen to move the connected line in another required direction or make the line length longer. This method is the so-called 'dragging' facility. The length and angle value will be visible on the screen and can be adjusted to be exact or close to the requested value.

Value Setting

Once the product or tool is drawn by sketching, the exact line lengths and angles can be optimized by setting the desired values. Just tap 2 times on the value of the line length or angle to change and the keyboard will pop-up. The value can be entered in 2 ways of confirmation:

- Enter function
- Enter-Next function

The Enter function will close the keyboard after entering the value. The Enter-Next function will enter the value on the line or angle to change and the keyboard will remain opened for the next programming step.
In case the typed value is erroneous, the “undo” button right from the input field can be tapped to return to the original value or the backspace key on the keyboard to delete the last typed character.

Zoom Function

By pinching the screen with two fingers simultaneously one can zoom in and out on the drawing, tool or machine visualization. By spreading fingers the system will zoom-in, by bringing fingers nearer to each other the system will zoom-out.

Fit-To-Screen

In the icons on the side of the screen you will find a Fit-To-Screen function. This can be used when the drawing size is not clear in picture. Just tap once and the complete drawing will be sized to fit the drawing screen. Tap again to return to the previous zoom level.

Panning

With simultaneous touching of two fingers and dragging them over the screen (sliding in the same direction) one can pan the object in 3D view. In 2D a single finger will also enable panning.

Rotating

In 3D, rotating the product, tool or machine visualization, can be done with a single finger sliding over the screen.

More information about this can be found in chapter 3.

Features of the drawing tool

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

1.5.4. Determine bend sequence

When the product drawing is completed, the control offers Tool setup mode to program the exact tool set-up as it is organized on the machine. After this you can select the Bend Sequence mode to determine and simulate the required bend sequence.
In the Bend Sequence mode, the control shows the product, the machine and the tools. In this menu the bend sequence can be programmed and checked visually. When a bend sequence has been determined, the CNC program can be generated.
More information about this can be found in chapters 4 and 5.

Bending sequence computation

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

1.5.5. Numerical program

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

There are two possibilities to create a CNC program:
- enter a numerical program, started via Products mode, tap New Program, step by step;
- generate the program from the graphical bend simulation started via the Products mode, tap New Product, via the Drawing mode. (see: Drawing mode; product drawing).

If the program is entered by hand, there is no collision check. All program values must be entered manually. The program depends on operator experience.
If the program is generated from a graphical bend sequence, the program can be visualized during production. A generated program can be edited according to operation needs.
More information about this can be found in chapter 6.

When a drawing has been completed with a bend sequence, and the program is stored, the program is post processed and the numerical program becomes available.
The system automatically computes:
- Necessary force
- Machine adjustments such as:
    - Y-axis position
    - Decompression
    - X-axis position
    - X-axis retract
    - Y-opening
    - R-axes
    - Z-axes

Axes positions are calculated according to the machine configuration.

1.5.6. 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 behavior of the bend system.

More information about this can be found in chapters 7 and 8.

1.5.7. Back-up data, external storage

Both product and tool files can be stored externally. Depending on the configuration, these files can be stored on a network or 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 paragraph 9.4.

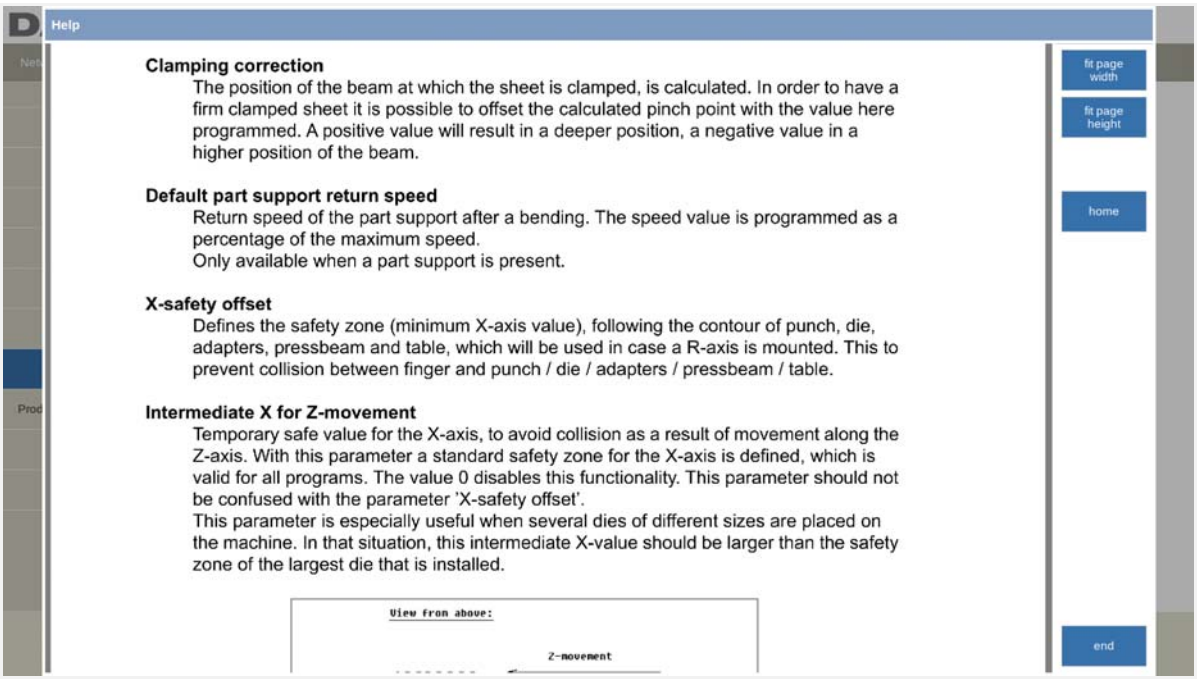
1.6. Programming aids

1.6.1. Help text

This control is equipped with an on-line Help function. When the Help-button in the navigation panel is pressed context related help will be provided.

To activate a help window for a parameter tap the Help button in the navigation panel.

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

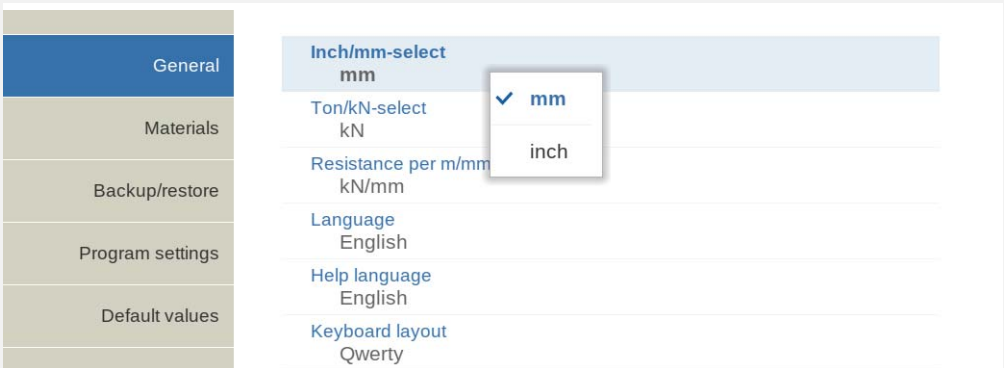


This Help window contains the same information as the Operation manual.

The help window can be used as follows:
You can scroll through the text sliding one finger in the desired direction. By tapping on the lower or upper part of the screen Previous Page / Next Page can be used to browse through the help text.
Tap Home to jump to the table of contents. Hyperlinks in the table help to directly navigate to the desired topic.
Tap End to close the Help window.

1.6.2. 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 close the opened listbox, tapping outside the box will make it close without changing the selected parameter.

1.6.3. Live search

In some modes a list of entities is offered (products, tools, materials, etc.). An example of such a menu is the Products mode (product selection). To search a particular product or tool, the search function can be used. Press the command button Search, type a part of the ID in the enter field. Automatically, the list is limited to those items that contain the typed part.
Multiple search parts can be separated by <space>.

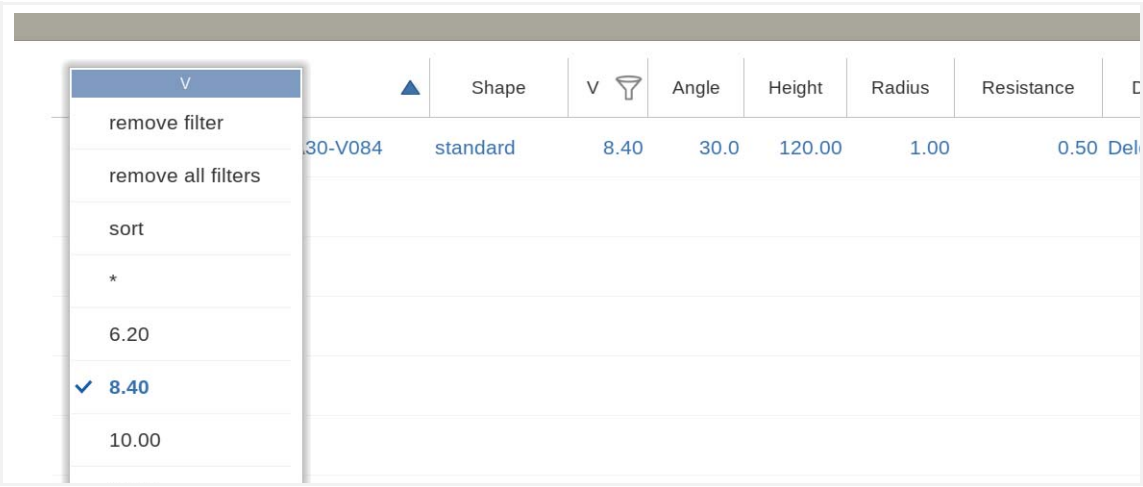
Example of a product list:	type:	shown list:
product123 product456 exampleproduct01 exampleproduct02	1	product123 exampleproduct01
	ex	exampleproduct01 exampleproduct02
	ex 1	exampleproduct01

Upon desire one can also select the search directory to find products in all directories, within the current tree or limited to the current directory.

To close an open search screen use the keyboard-close button on the right side next to the keyboard.

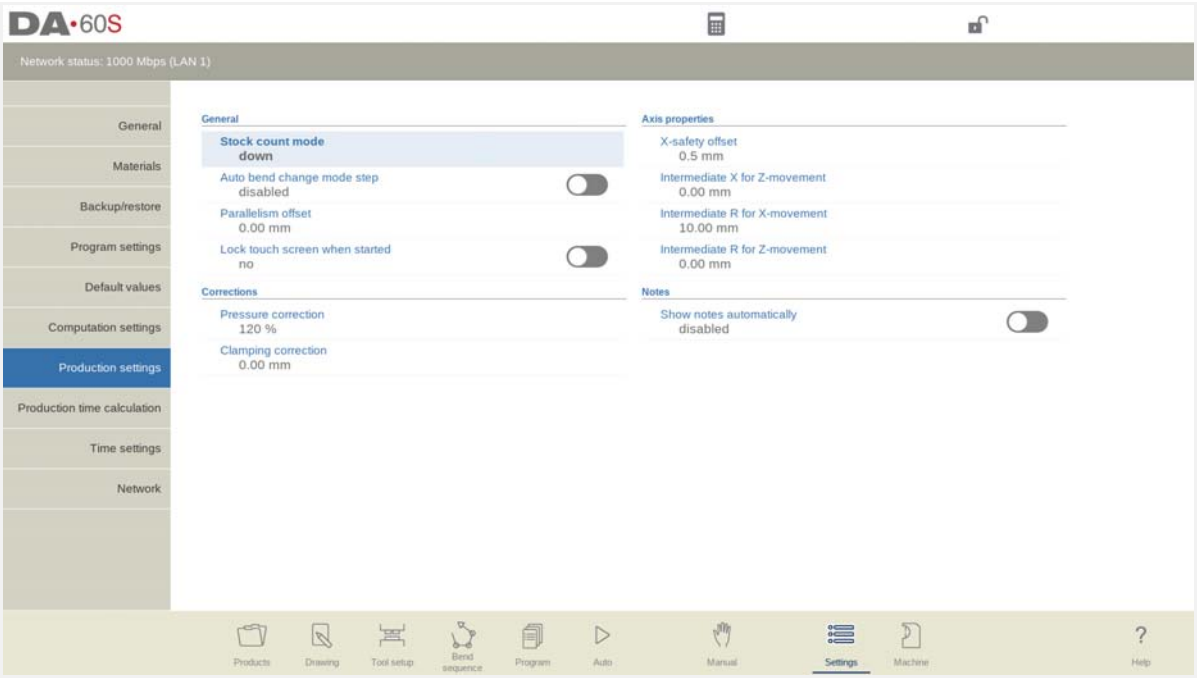
1.6.4. Auto filter

In addition to the search function the column headers of e.g. the tool selection tables and the product selection table have ‘auto-filter’ functionality.
When tapping the column header one can sort the list based on that specific column. The triangle in the header is shown as an indication of the sort order.
When tapping a header with auto-filter functionality a list box will be shown automatically offering the possible options for filtering, derived from the available entries in that column. E.g. when using this in a die selection, the V-opening can be filtered to the desired value as searched for. The list will only show dies complying the set filter.
When one ore more filters are active, this is indicated with a filter icon in the lower left corner, together with the number of filtered items and the total number of items.
Filters can be switched off in a similar way. The list also offers a remove filter function.



1.6.5. Navigation

Within some modes, the program screens are divided into tabs, vertically located at the side of the screen.



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 vertically, the desired tab can be “pulled” in sight and be selected.

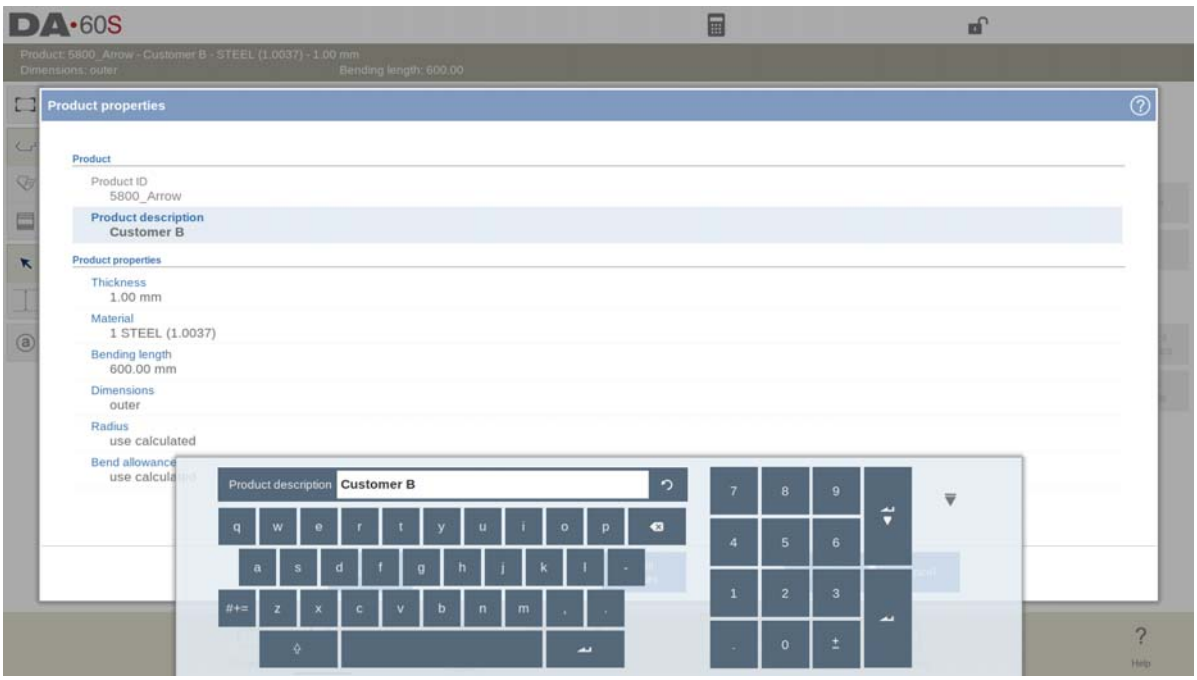
1.6.6. Text input and editing

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.
E.g. in Edit notes, where multiple lines can be entered Enter is used for linefeed. Cut, Copy and Paste are offered on the keyboard for editing convenience. Undo and Redo can also be used within this editor.

The keyboard can be shown or hidden in this multi-line editor using the keyboard key in the lower left corner of the on screen keyboard.

1.6.7. Typing alphanumeric characters vs. special characters

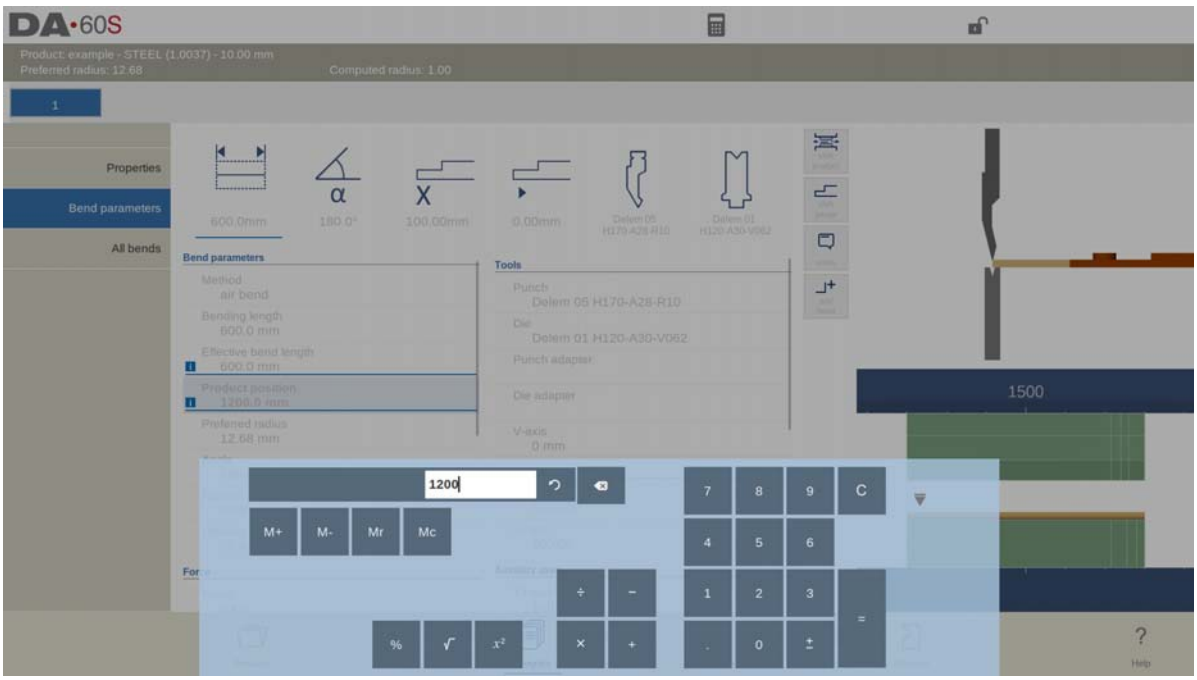
Both alphanumeric characters and special characters can be used throughout the control. A full on-screen alphanumerical keyboard will pop up when required.
When editing a field which is pure numeric, the alphanumeric characters will be “greyed-out” and only the numerical keypad can be used. For fields which enable to use alphanumerical 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.

1.6.8. Calculator

The CNC control is providing a “desktop calculator” available for the operator.
At the top of the screen the calculator icon can be used to switch to calculator functionality.
The keyboard area provides calculator functions which can be used autonomously. Standard functions (add, subtract, multiply and divide) including percentage, square root, square and memory functions are available.



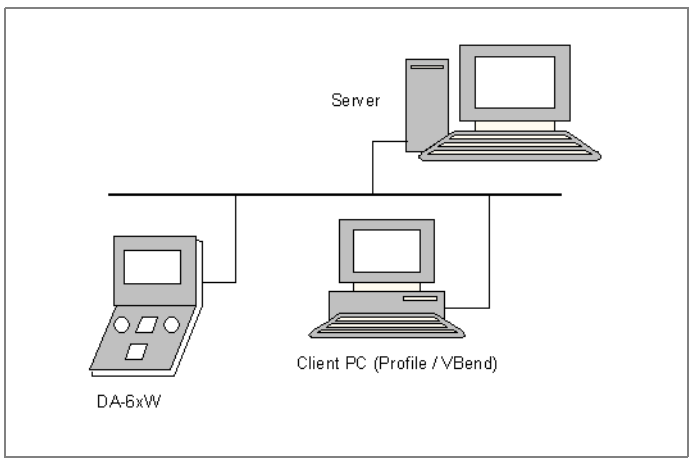
In case calculations should use parameter values as an input, and results as an output, one can switch to the calculator from the parameter entry. This will bring the parameter value to the calculator, and the result back to the input line.
No cutting or pasting is necessary. Only when entering and confirming the calculated value in the parameter entry line, it will be used forward.

1.6.9. Messages center

When messages are displayed coming from PLC, Safety systems, LUAPs or the Sequencer, these messages can be ‘send’ to the 'Messages center'. When a message is displayed simultaneously the message center symbol is shown in the top row of the page header, next to e.g. the calculator and key-lock symbol. When tapping this message center 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 center symbol has an extra indicator to show new incoming messages which are not yet shown.

1.6.10. Network

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



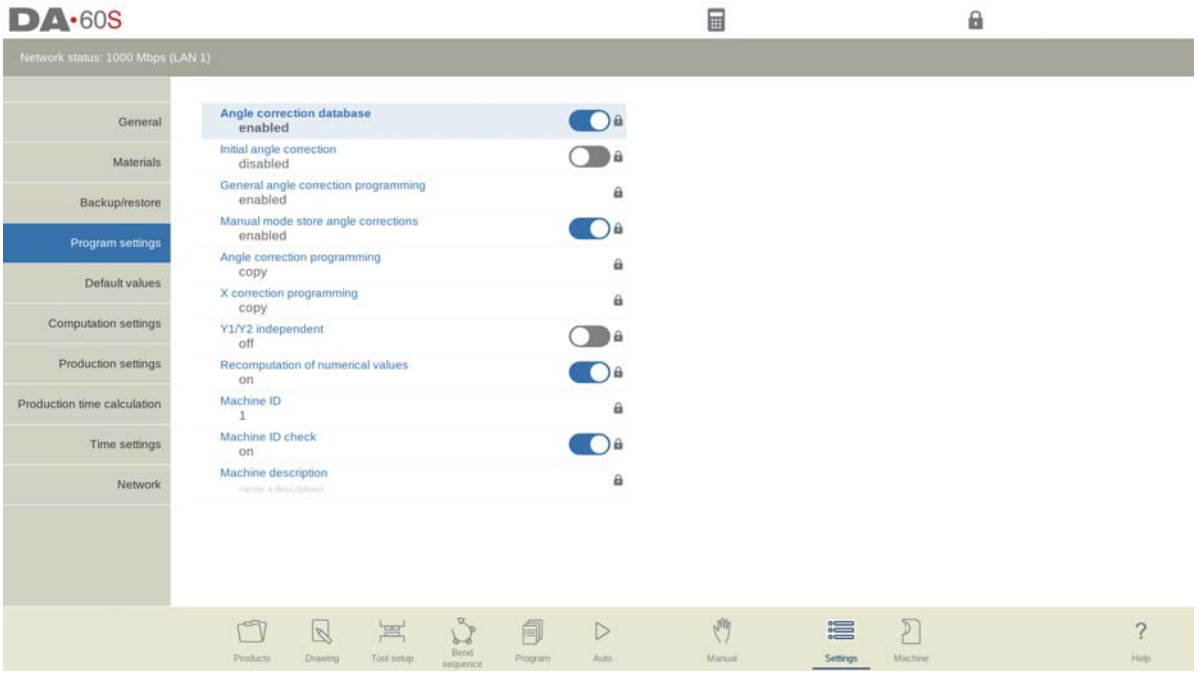
1.6.11. Key-lock function

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

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

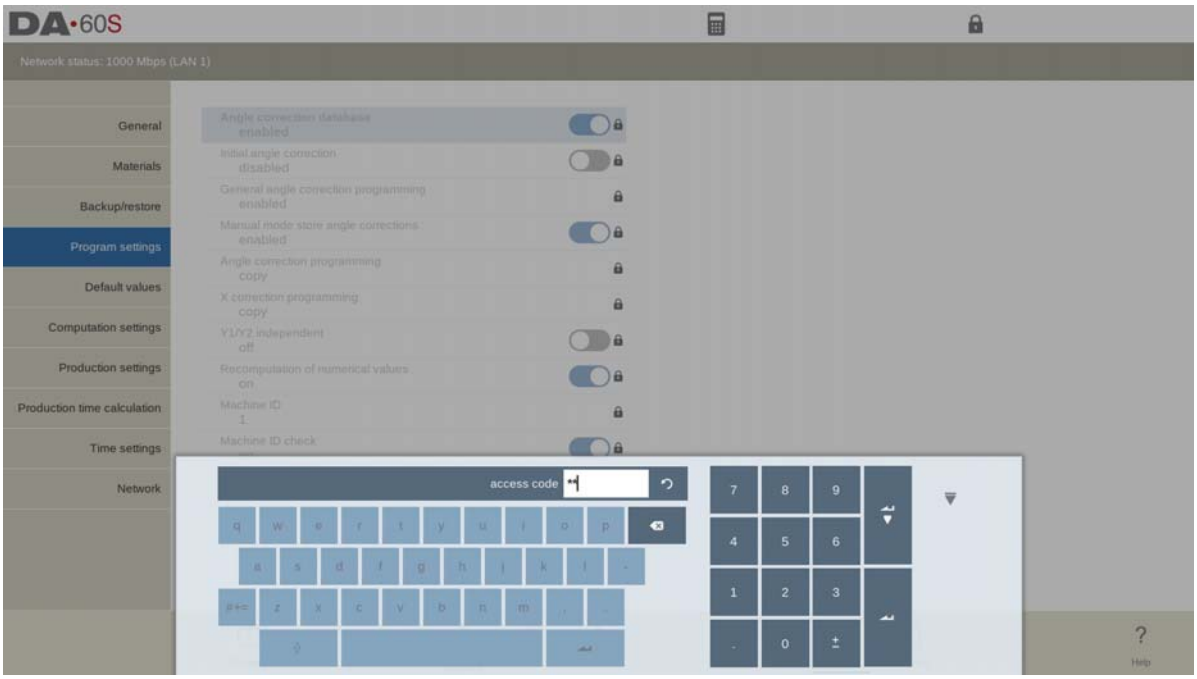
- In Program Lock, only a product can be selected and executed in Automatic mode.
- In Machine Lock, the machine is locked and the control can not be used.

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 will show a closed lock in gray. 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.

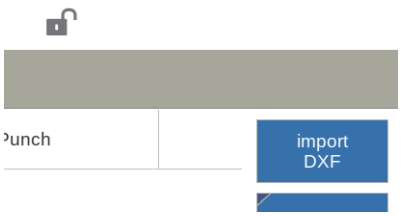
To unlock the control, tap the lock symbol and enter the appropriate code. After entering, the lock symbol will show it is unlocked and the lock symbols behind parameters will disappear.



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

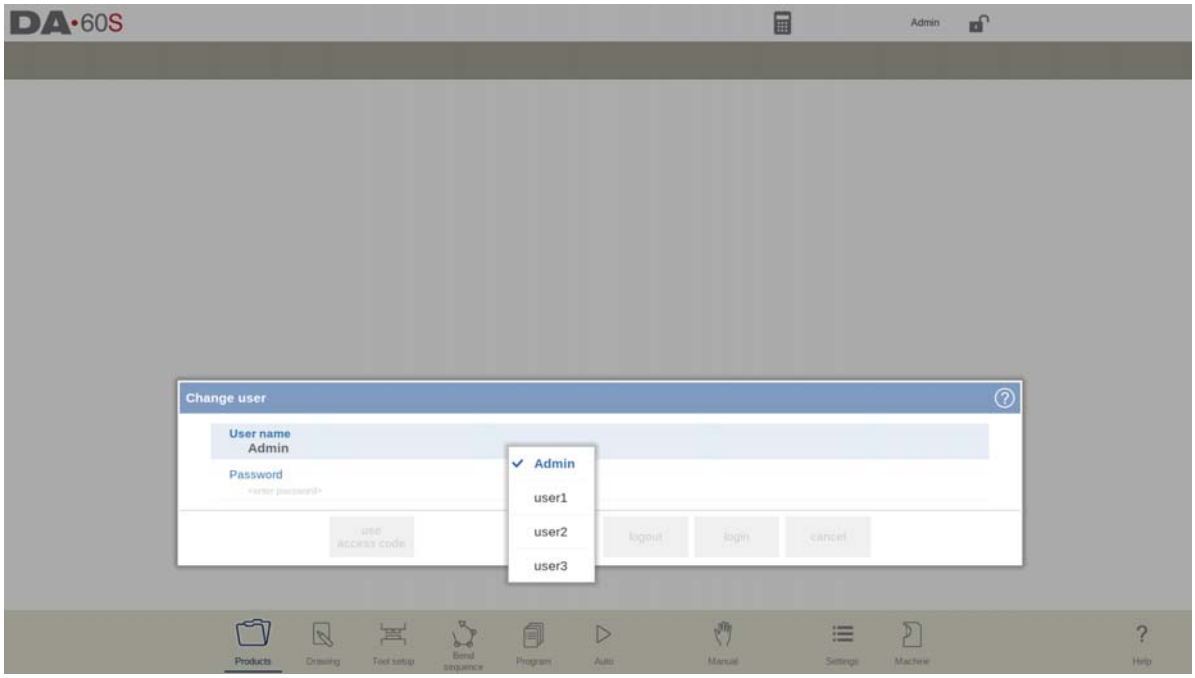
1.6.12. OEM function panel

Depending on machine manufacturers implementation, the upper right corner of the screen can be used for special indicators.
To access functions related to those indicators, the OEM function panel can be opened by tapping this corner of the screen.



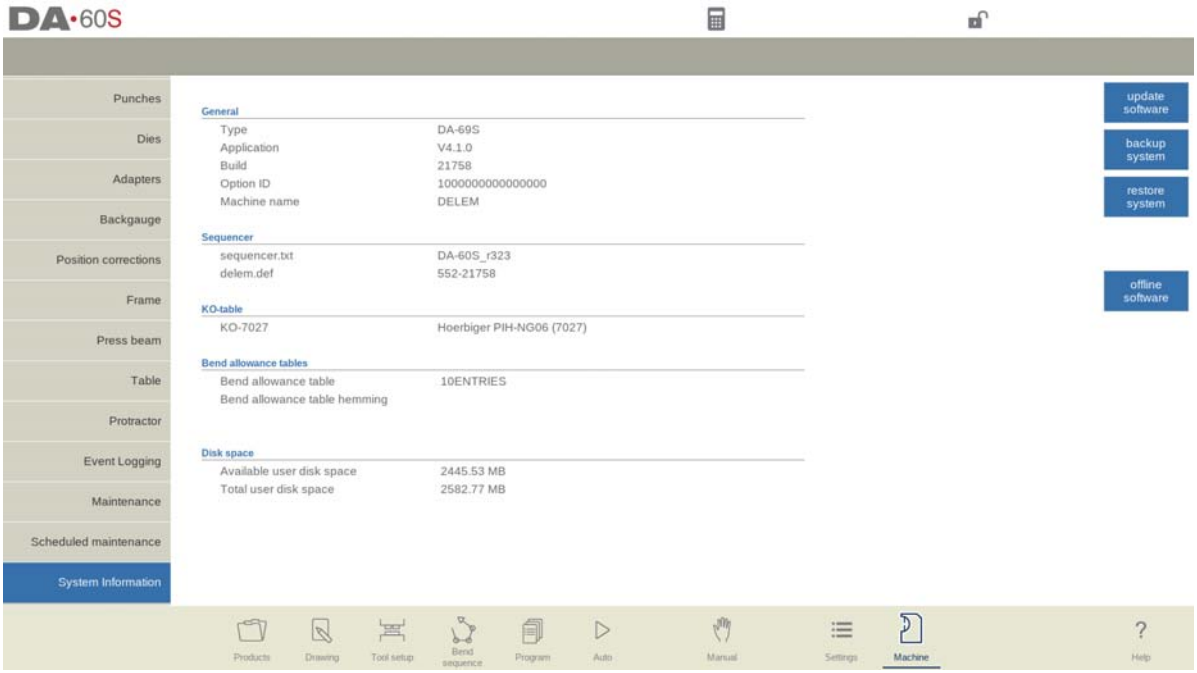
1.6.13. Multi user system

Upon configuration the multi user system enables the use of login / logout functionality for users. In the Products mode the ‘change user’ function enables the change of a user as well as the logout of the current user. When this system is active one needs to login at every start-up.



1.6.14. Software versions

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



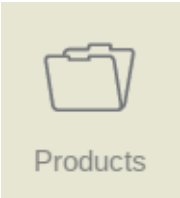
Example of version number:

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.

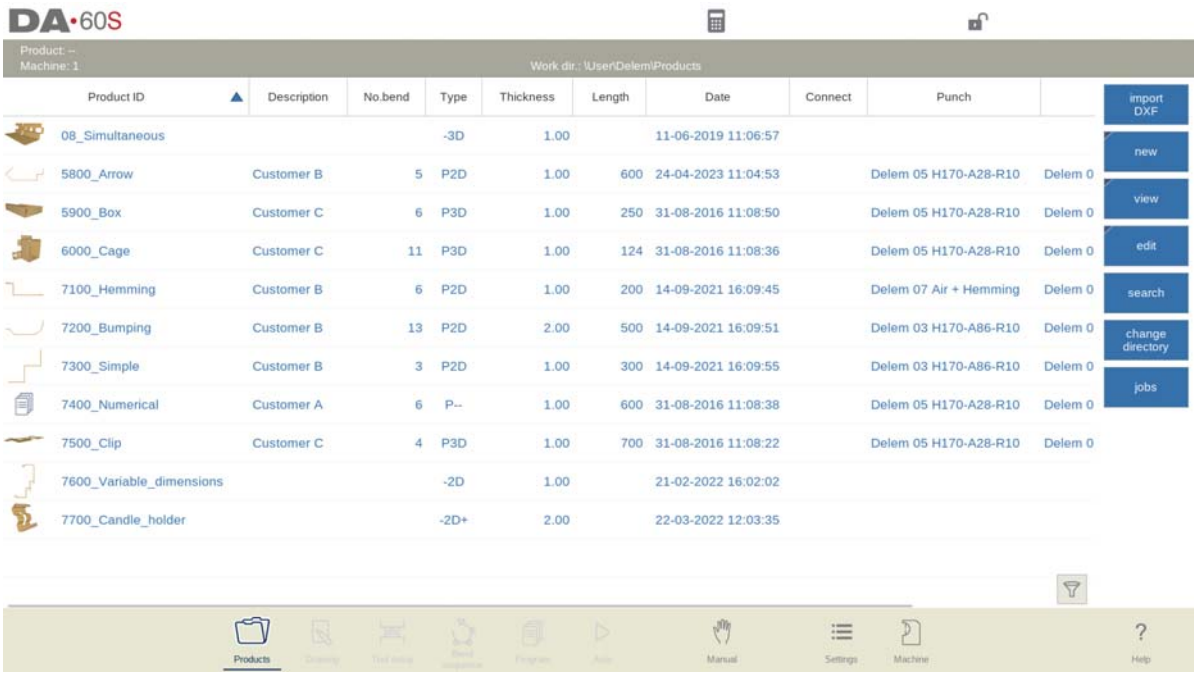
2. Products, the product library

2.1. Introduction



In the **Products mode** existing, previously produced, products can be selected to start production or modification in order to make a similar product. To start making a new product or program New Product or 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 a thumbnail of the graphical product (for numerical programs a symbol is shown), its Product ID, the Product Description, the Number of bends in the product, what kind of product it is (Type) and the Date it was last used or modified.
The Type indication of the product shows following types of products:

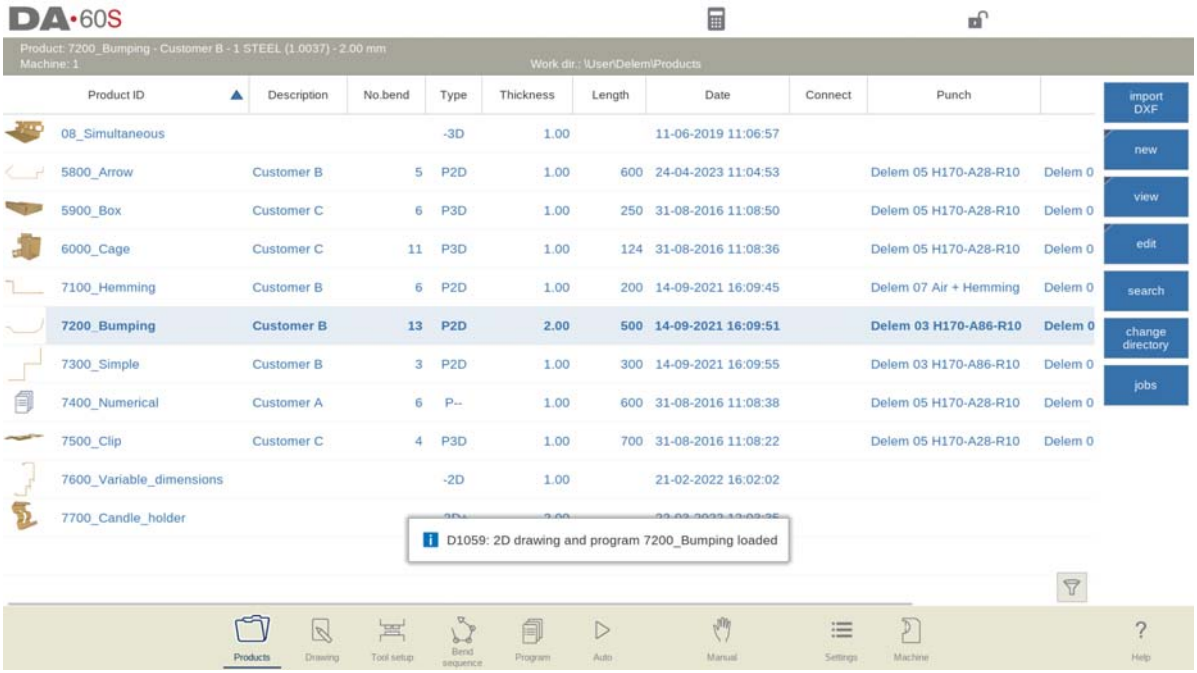
P--	the product has a CNC program, there is no drawing
-2D	the product consists of a 2D drawing, there is no CNC program
P2D	the product has a 2D drawing and a CNC program
-3D	the product consists of a 3D drawing, there is no CNC program
P3D	the product has a 3D drawing and a CNC program

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

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

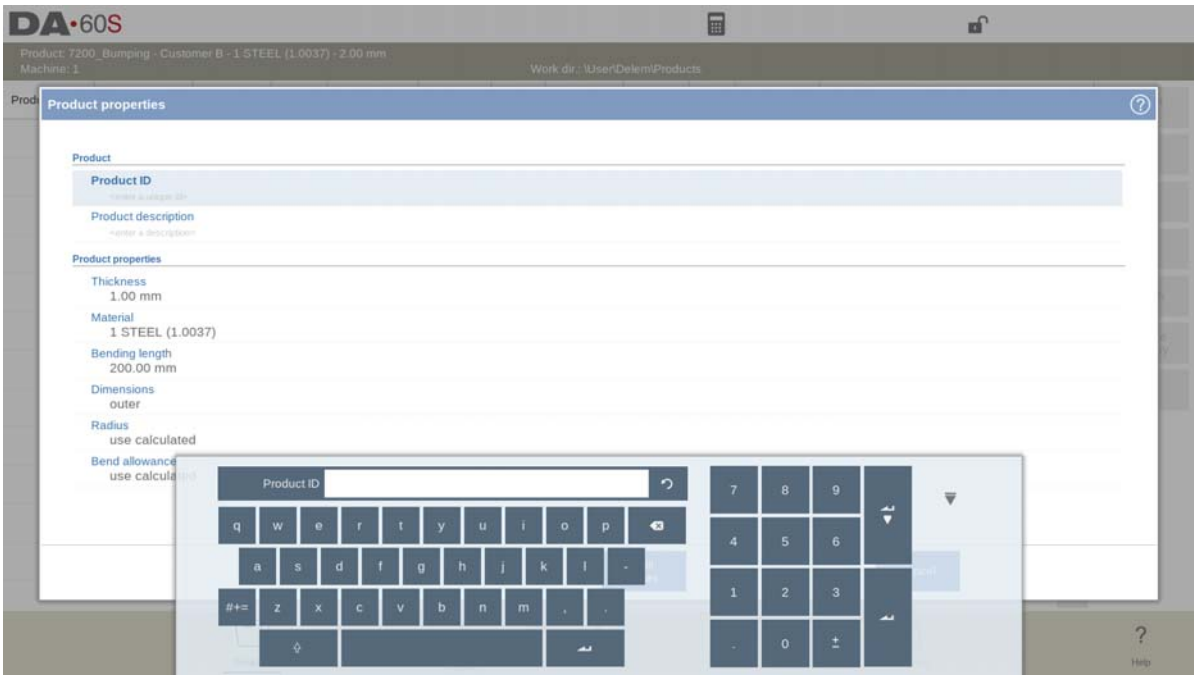
2.1.2. Product selection

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 Products Drawing (if existing), its Tool Setup, the Bend Sequence and the numerical Program of the product.



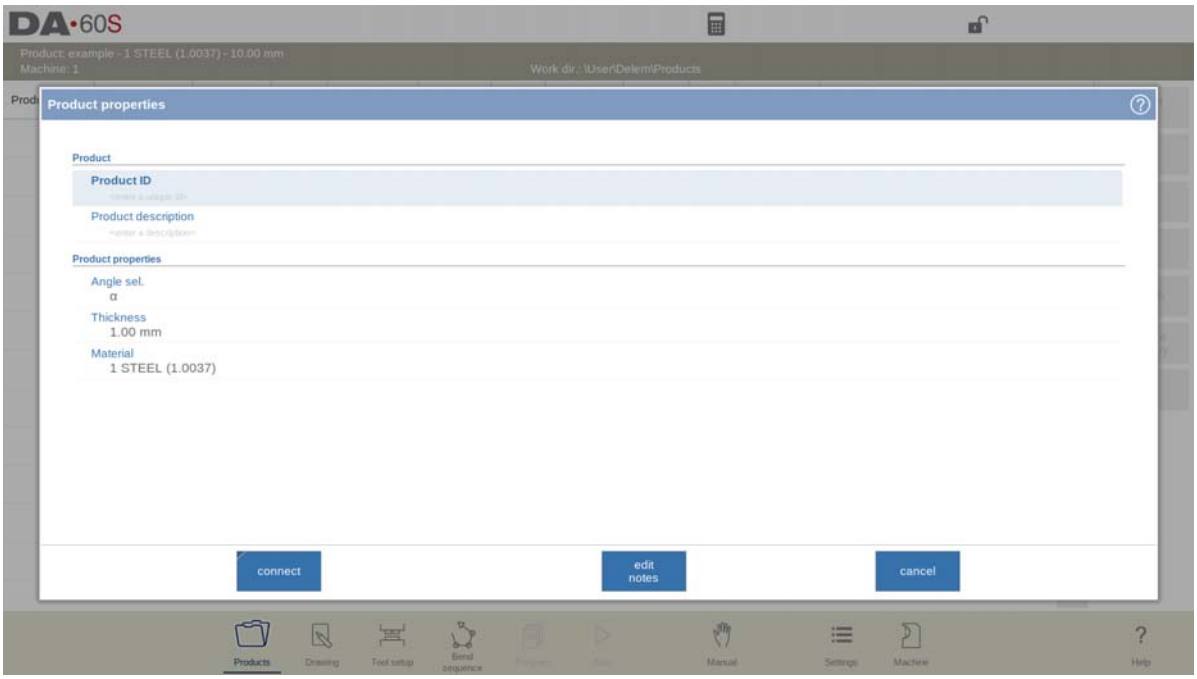
2.1.3. New Product, starting a new graphical product

To start a new graphical product tap New and Product.
After New and Product are chosen, the programming of a new product starts with its general details like Product ID, Thickness and Material.



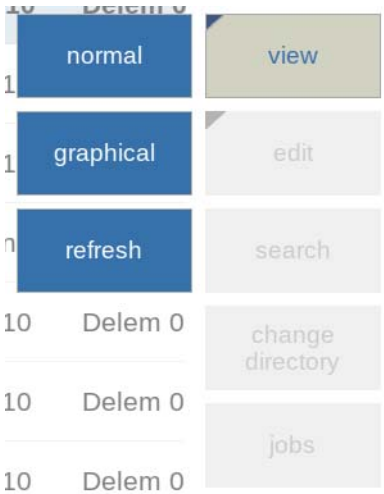
2.1.4. New Program, starting a numerical program

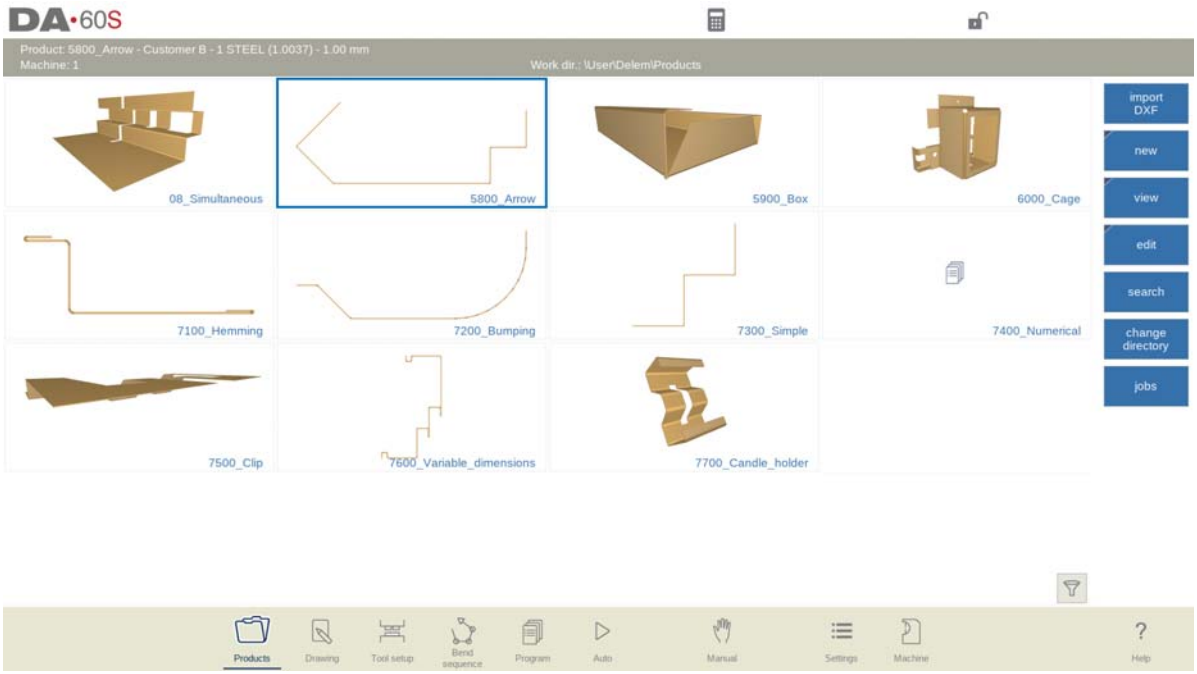
To start a new numerical program tap New and Program.
After New and Program are chosen, the programming starts with its general details like e.g. Product ID, Thickness and Material.

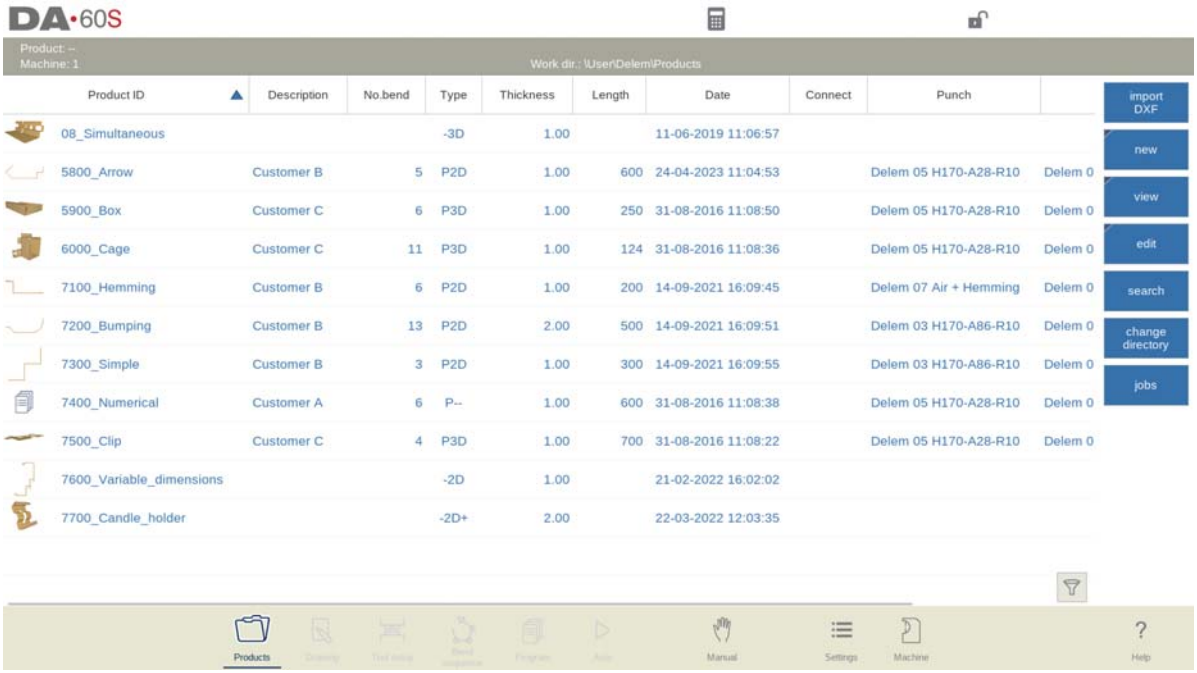


2.1.5. Views

To view the products as a simple list, or completely graphical, the View function can be used. By tapping View one of the two view modes can be selected.

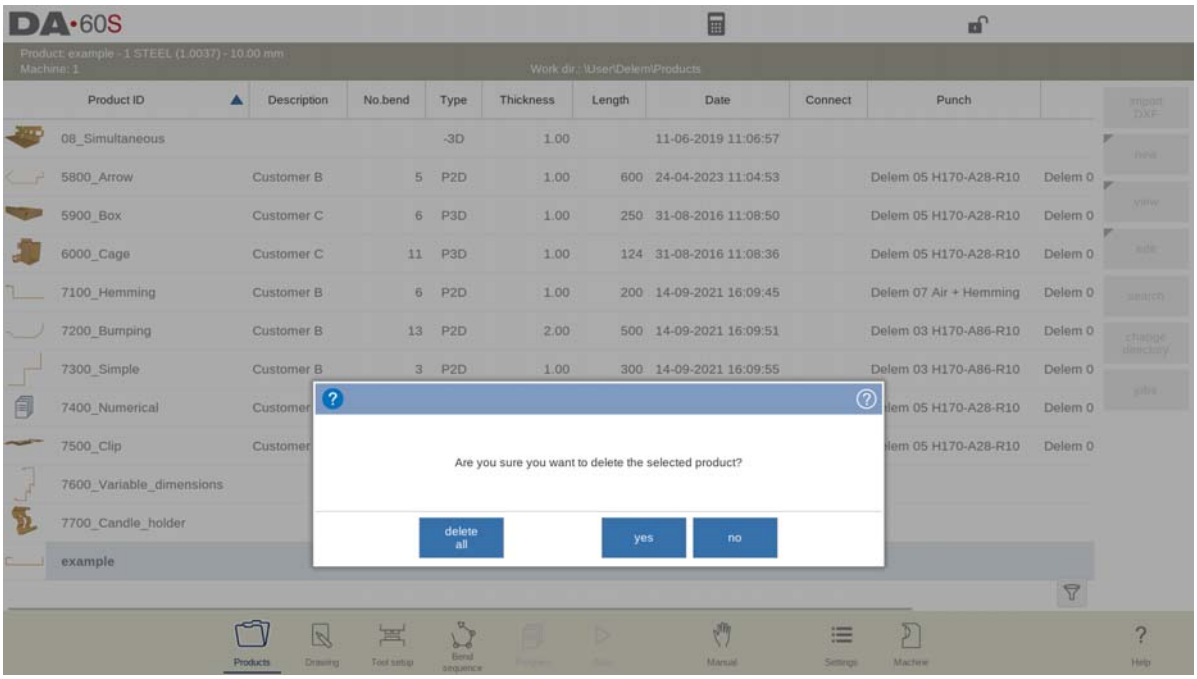




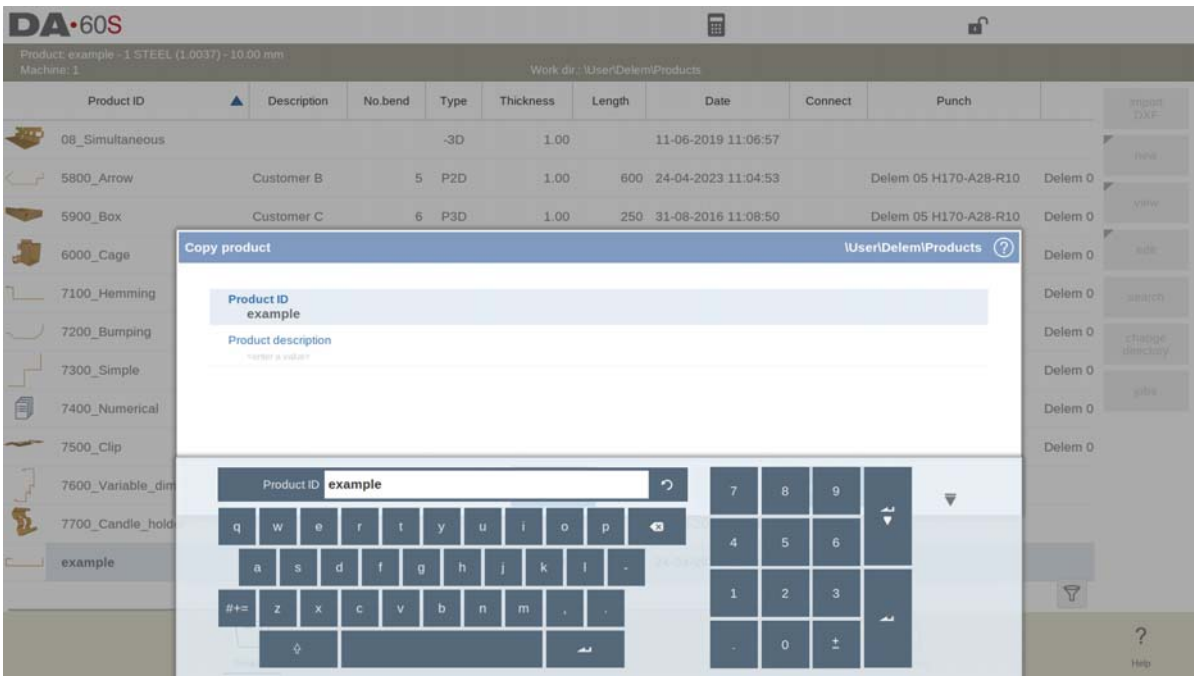

DA•60S
Product: --
Machine: 1
Work dir.: \User\Delem\Products
Product ID
Description
No.bend
Type
Thickness
Length
Date
Connect
Punch
08_Simultaneous -3D 1.00 11-06-2019 11:06:57
5800_Arrow Customer B 5 P2D 1.00 600 24-04-2023 11:04:53 Delem 05 H170-A28-R10 Delem 0
5900_Box Customer C 6 P3D 1.00 250 31-08-2016 11:08:50 Delem 05 H170-A28-R10 Delem 0
6000_Cage Customer C 11 P3D 1.00 124 31-08-2016 11:08:36 Delem 05 H170-A28-R10 Delem 0
7100_Hemming Customer B 6 P2D 1.00 200 14-09-2021 16:09:45 Delem 07 Air + Hemming Delem 0
7200_Bumping Customer B 13 P2D 2.00 500 14-09-2021 16:09:51 Delem 03 H170-A86-R10 Delem 0
7300_Simple Customer B 3 P2D 1.00 300 14-09-2021 16:09:55 Delem 03 H170-A86-R10 Delem 0
7400_Numerical Customer A 6 P-- 1.00 600 31-08-2016 11:08:38 Delem 05 H170-A28-R10 Delem 0
7500_Clip Customer C 4 P3D 1.00 700 31-08-2016 11:08:22 Delem 05 H170-A28-R10 Delem 0
7600_Variable_dimensions -2D 1.00 21-02-2022 16:02:02
7700_Candle_holder -2D+ 2.00 22-03-2022 12:03:35
import DXF
new
view
edit
search
change directory
jobs
Products
Manual
Settings
Machine
Help

2.1.6. Product Copy and Delete

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 products and programs at once, tap Delete All.



To copy a product select a product or program and tap Edit and use Copy. After this the name of the product can be programmed and the copy will be done. The product will appear in the same directory. The copied product will be an exact copy including tool setup and bend sequence if available.

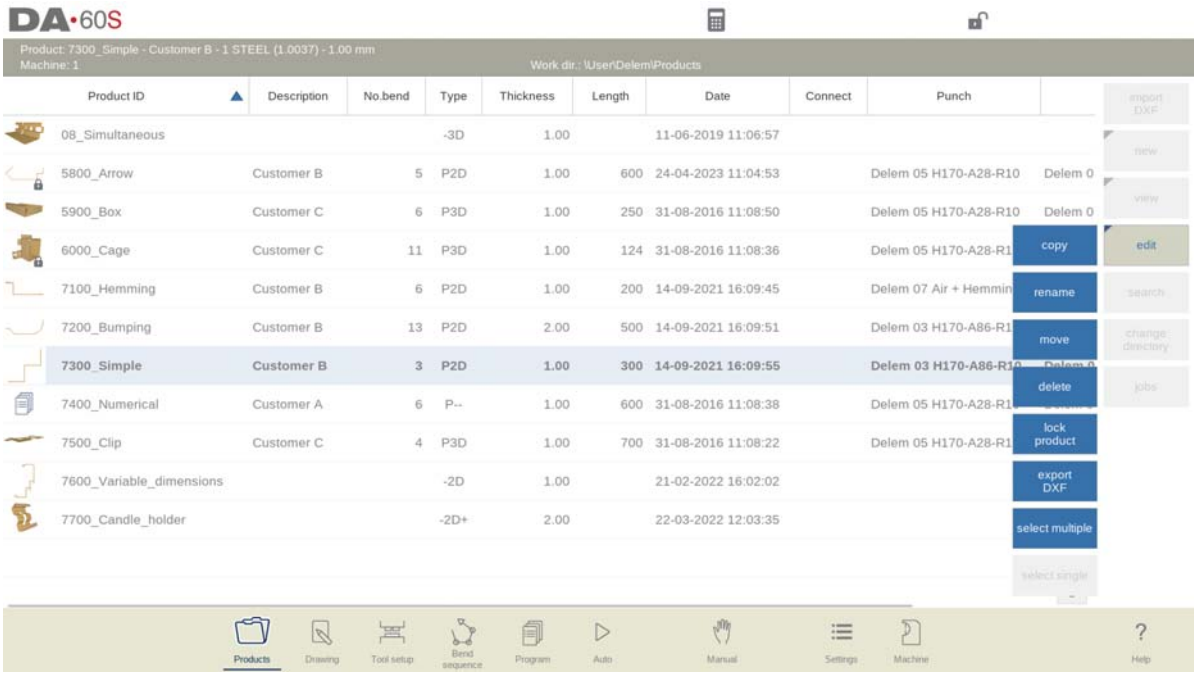


2.1.7. Product Rename and Move

Products can also be moved and renamed.This can be done in one single step: Move moves a product to a new directory, Rename allows the user to give it a new name within the same directory.
To move or rename a product select a product or program and tap Edit and choose Move or Rename from the list. For Rename a new name can be given. The product will appear in the same directory. For move a new location can be selected.

2.1.8. Product Lock/Unlock

The product Lock/Unlock function provides a simple method to prevent accidental changes to finished programs or products. In this way products which have been tuned and found to be good cannot be changed unless the product is unlocked.
When tapping on Edit the Lock Product / Unlock Product feature can be switched for each product or program.



2.1.9. Selection of multiple products

To select multiple products tap Edit and choose Select multiple. Now multiple products can be selected. This multiple selection can be copied, moved, deleted, locked or unlocked.

To select a single product again tap Edit and choose Select single.

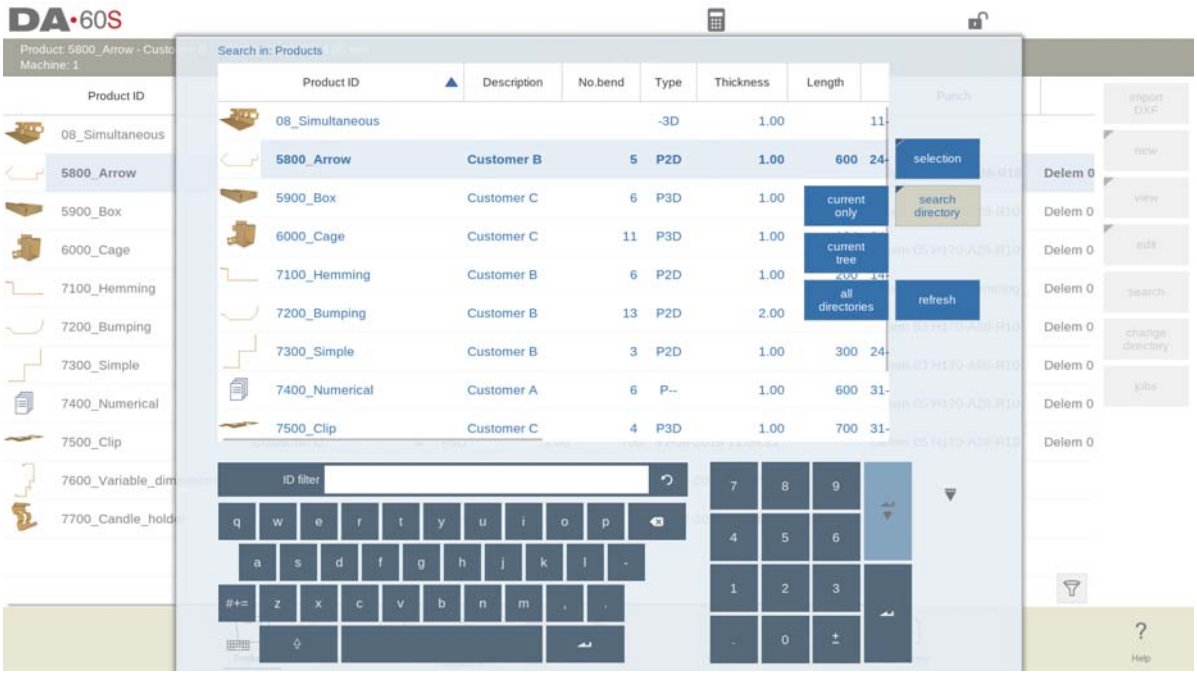
2.1.10. Search function

To make finding products easier the search function enables live searches through-out the Products mode.
When tapping Search, the search screen will show. By typing the desired filter string, optionally divided by spaces, the live search will start.

Upon desire one can also select the search directory to find products in all directories, within the current tree or limited to the current directory. Optionally, using Selection, the search can be done on Product ID or on Product Description.

You can either enter a complete name or number or only a part of it. If you enter part of a name and this part occurs in several product names, the control will show all product names that contain that part. It is also possible to enter a combination of name and number.

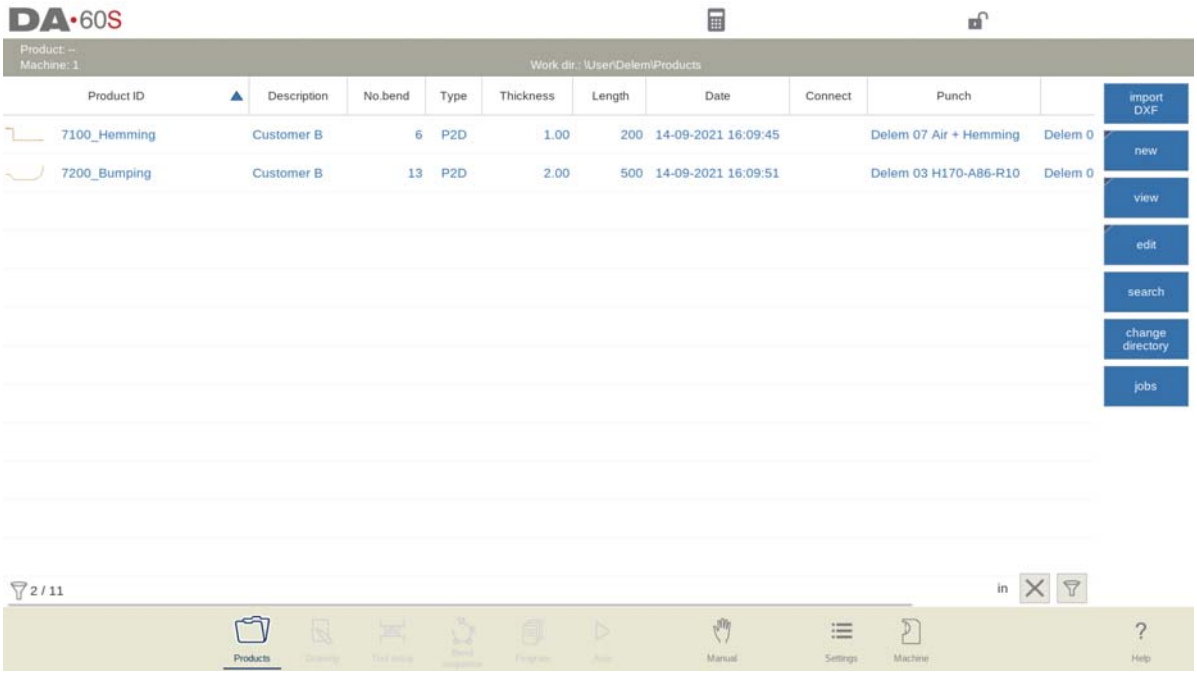
See also section 1.6.3 about 'Live search'.



2.1.11. Filter function

In addition to the search function, a filter function is available to reduce the number of visible products in the overview of Product mode.

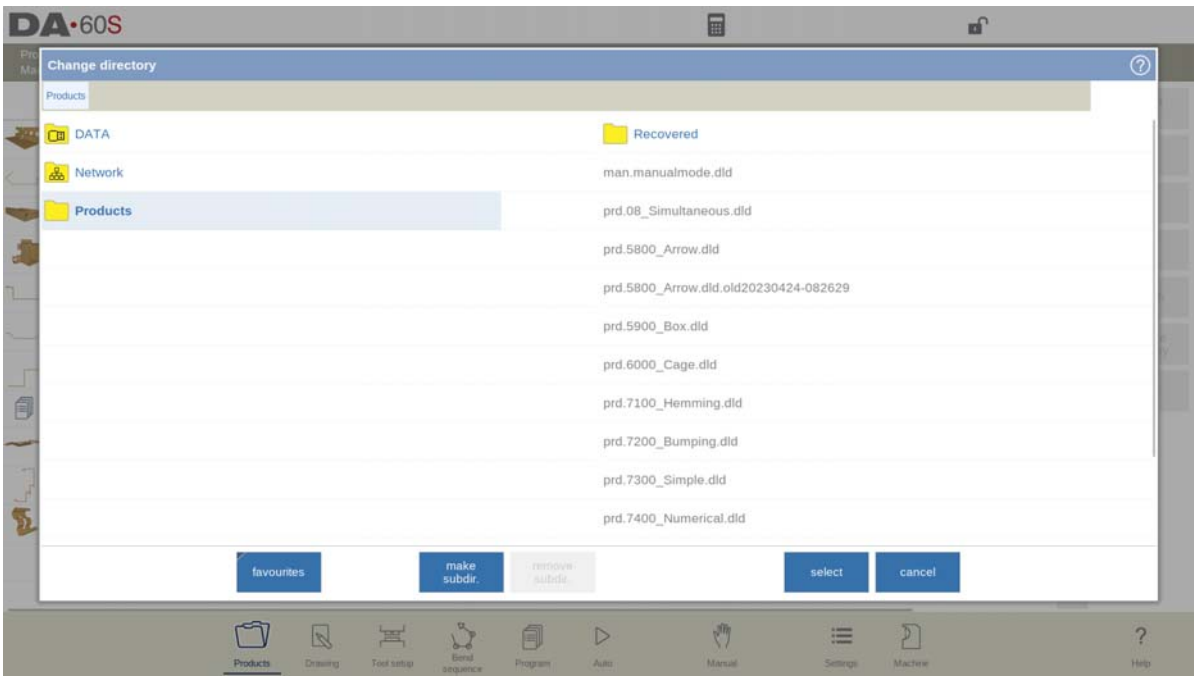
When tapping the filter icon in the lower right corner, the keyboard shows the text which is used to filter the list of products.
When this filter is active, this is indicated with a filter icon in the lower left corner, together with the number of filtered items and the total number of items.



This filter can be switched off by tapping on the cross icon in the lower right corner next to the filter icon.

2.1.12. Change directory

To change to a different product directory, or to add a new product directory, tap Change Directory. When an obsolete directory must be removed, select the directory and tap Remove Directory. When a desired directory is reached, tap Select to jump back to the Products screen which will show all products in the directory. The active local directory name is displayed in the header.



make subdir.	Add a new subdirectory. The name of the subdirectory can be any alphanumeric string of maximal 24 characters. (Avoid the use of slashes '/').
remove subdir.	Delete a subdirectory. If the directory contains files or subdirectories, the control will return questions to confirm the deletion for both. The default directory “PRODUCTS” cannot be deleted.

In this menu it is also possible to remove existing subdirectories (only an empty one) and to make new subdirectories. Tap Make Subdir and enter the new name. Subdirectories are called subdirectories because these directories reside under the local directory 'DELEM\PRODUCTS'. The name of the subdirectory cannot be changed.

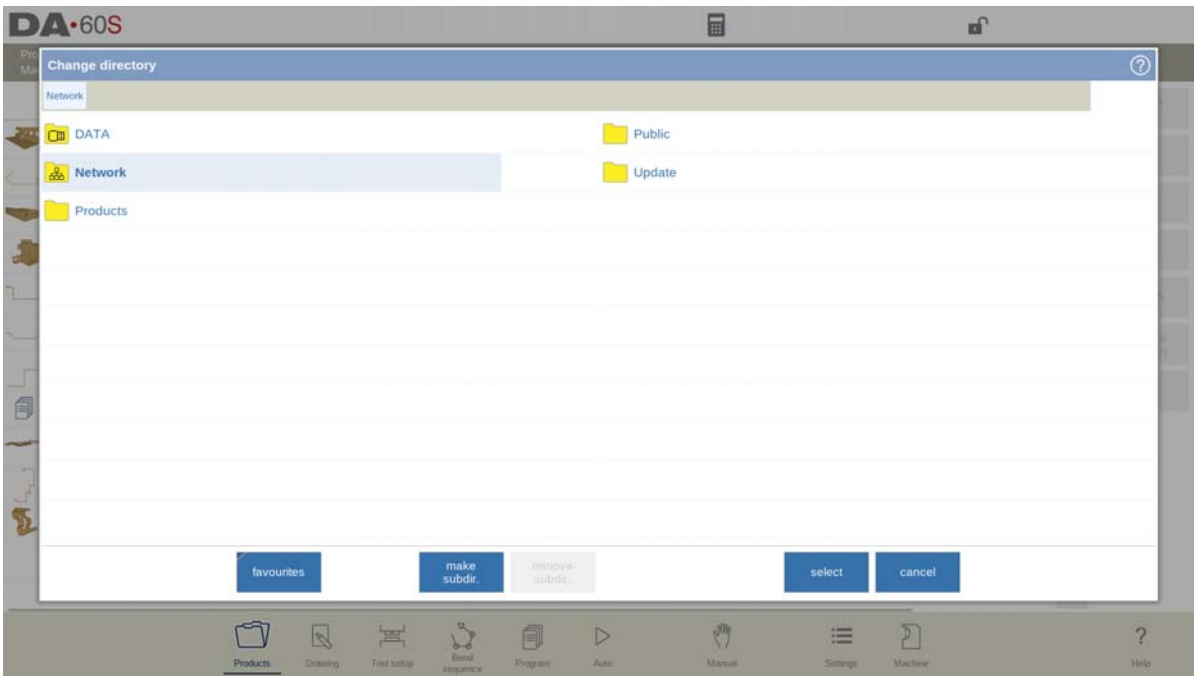
In this menu it is not possible to copy products from one subdirectory to another subdirectory; this can be done in Drawing mode or Program mode.

When you leave the product selection menu the control remembers the active subdirectory and the active product (if a product was selected) until another directory or product is selected.

2.1.13. Network product selection

When a network directory has been mounted in the control this mounted directory can be found under Network. Network is available next to the Product directory when using change directory. The name of the mounted drive indicates availability for product selection and storage.

The network directories can be navigated thru in the directory browser. Directories can be selected, added and removed and products can be selected. When a desired directory is reached, tap Select to jump back to the Products screen which will show all products in the directory. The network directory is now the active local directory. Its name is displayed in the header of the screen.



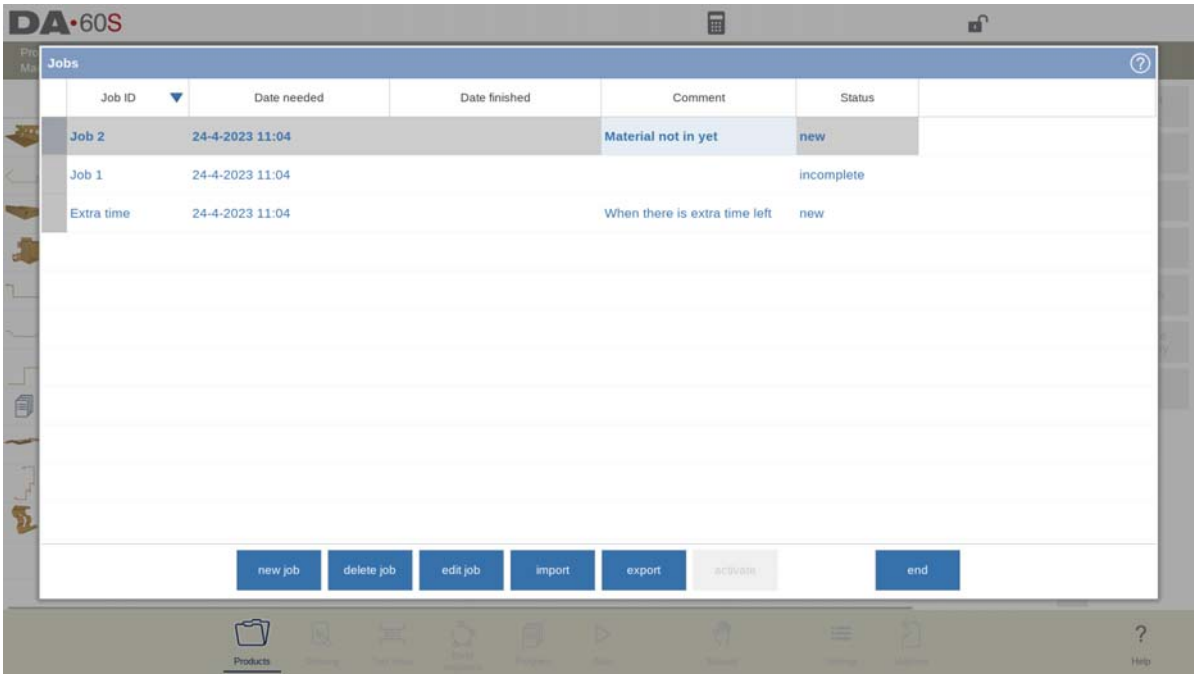
When you leave the product selection menu the control remembers the active subdirectory and the active product (if a product was selected) until another directory or product is selected.

While working with a “read-only” network, or when the network connection is interupted, the product will be saved in subdirectory “Recovered”. This can be found as a subdirectory under Products.

By tapping the Refresh button (on View) in Products mode the product library that is displayed on the screen is refreshed, which can be useful when working from a network location.

2.2. Shop floor control, Job List function

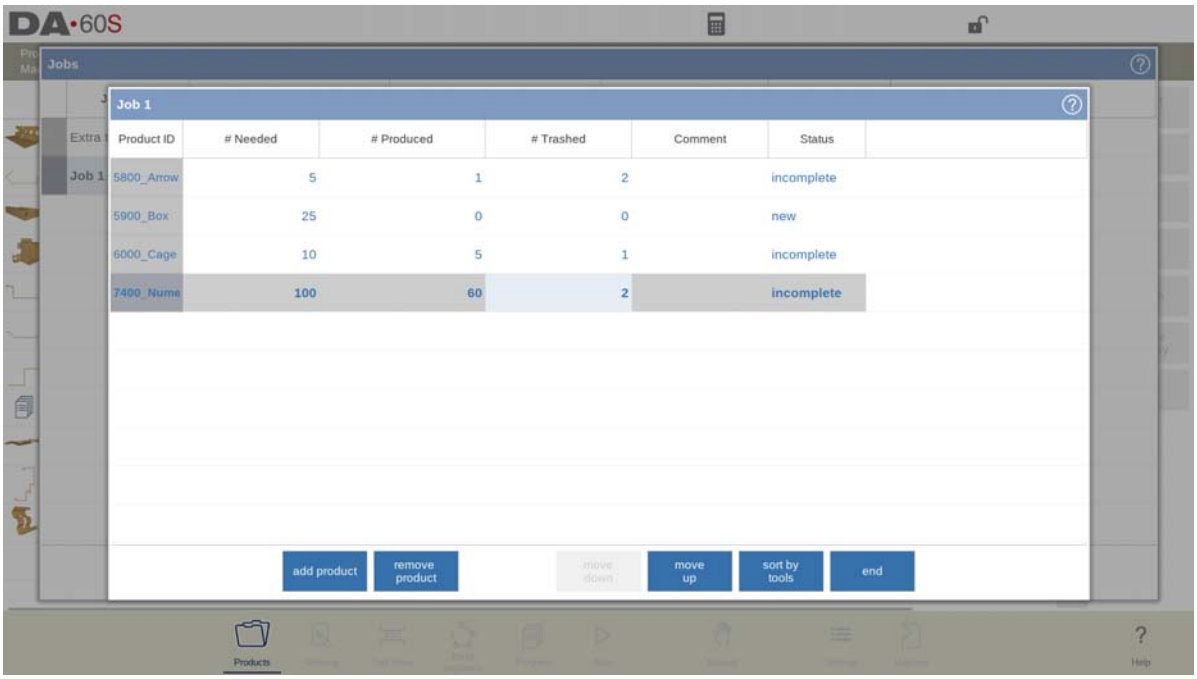
To enable the functionality to control the orders being processed at the machine, the Job List feature is available. Over this functionality job orders can be generated and sent to the machine. The operator can open the available job orders and can produce according to these Job Lists.
The Job List functionality is started with the command button Jobs. When starting this function, the Jobs screen will open and show the jobs which have been prepared already. All jobs have a Job ID, a Date needed and a Date finished. One can also add Comment to a job. The Status of the job can also be seen on the Jobs screen.



Job List status indication:
- New: no products have been produced yet;
- Incomplete: the desired number of products has not been produced yet;
- In progress: this job is currently loaded on the machine and in production,
- Finished: all batches of products have been confirmed as finished.

Next to adding new jobs, with the function key New Job, it is also possible to import and export job files. Jobs can be exported and imported, upon desire including the products, to make transfer from Profile-T offline software to the machine as easy as possible.

To edit a job, the function key Edit Job will open the job and Products can be added. As an example the first job Is opened.



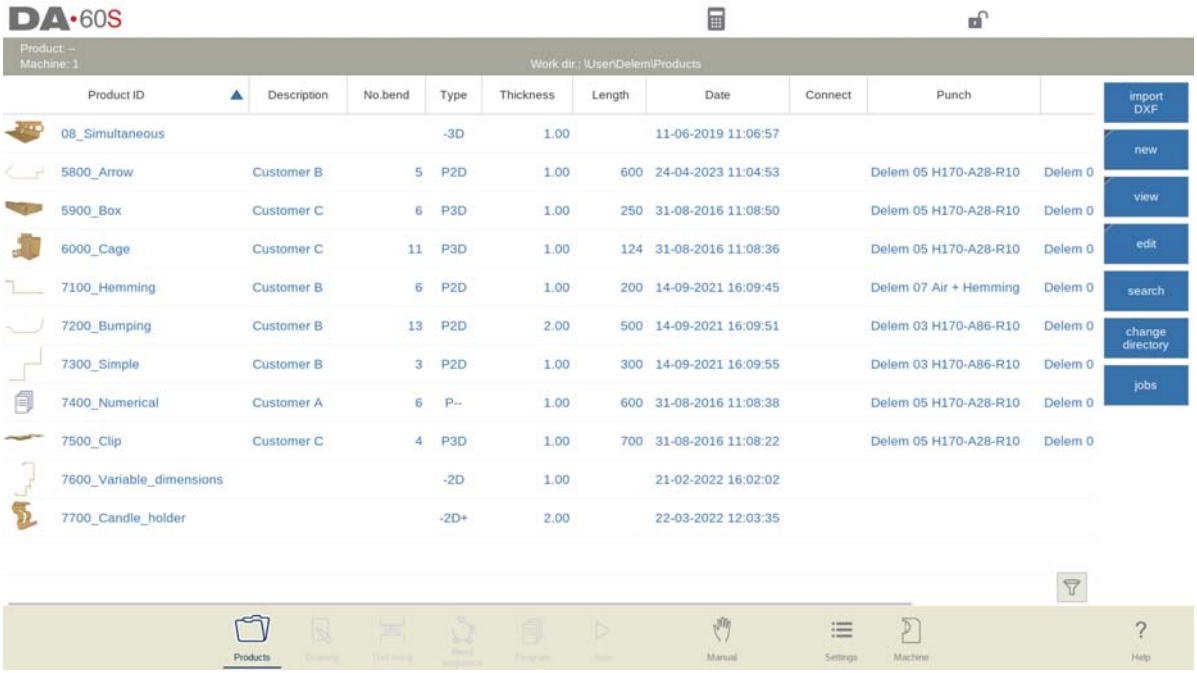
For every product in the job, the Needed number of products, the Produced number of products and the Trashed number of products are shown. The last two, of course, are coming as a result of executing the job in production. So, via this screen also the progress can be monitored.

On the Jobs screen, the function key Activate can used to start the production of the selected job. The control will switch to Automatic mode automatically, and the first product to be produced from the job will be loaded. The Start button can be used to really start bending.

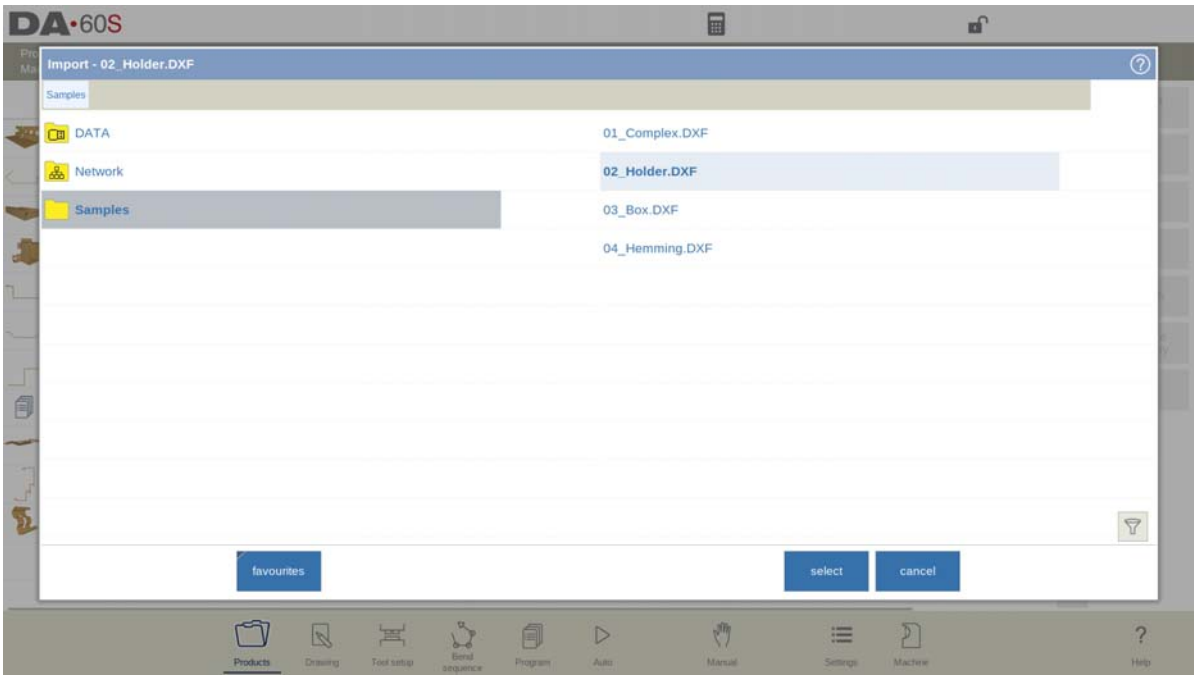
2.3. The DXF import function (only available when DXF option has been installed)

An alternative for drawing the desired product in the control, the control can also import an externally generated CAD-system output file. This paragraph will explain the use of the DXF converter to import DXF files and its functionality.

The DXF import option is started with the command button Import DXF, which opens a file selection browser to select the DXF file.

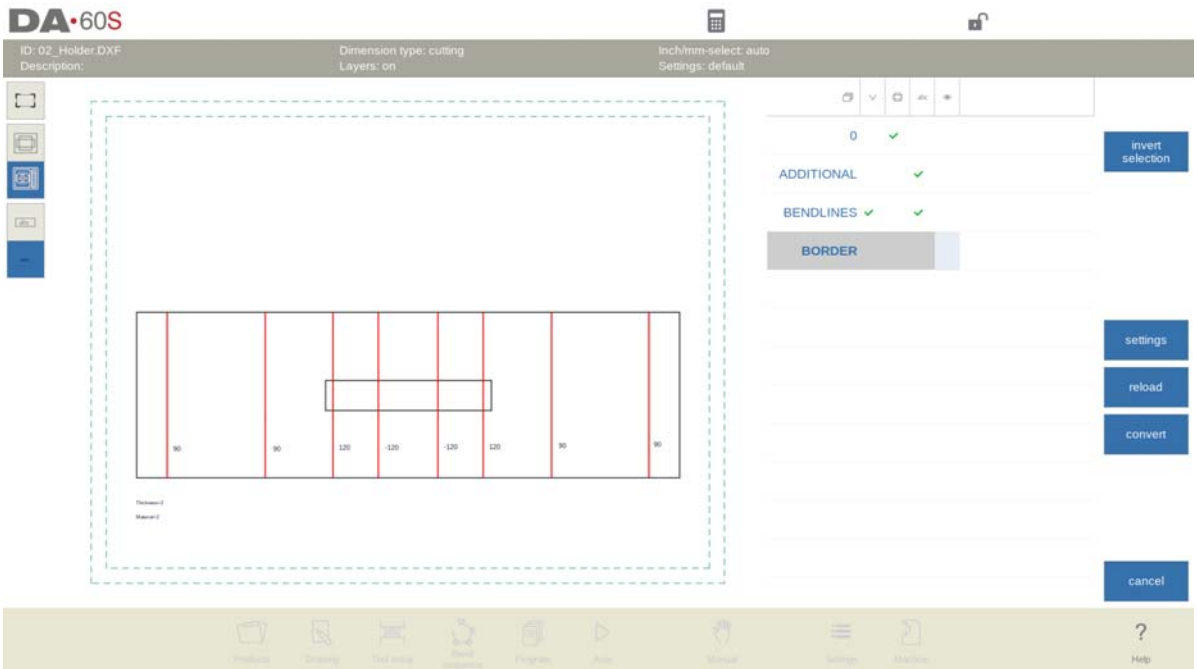


Files can be located on eg. a USB stick or on the network directory. One can browse to the location and select the file.

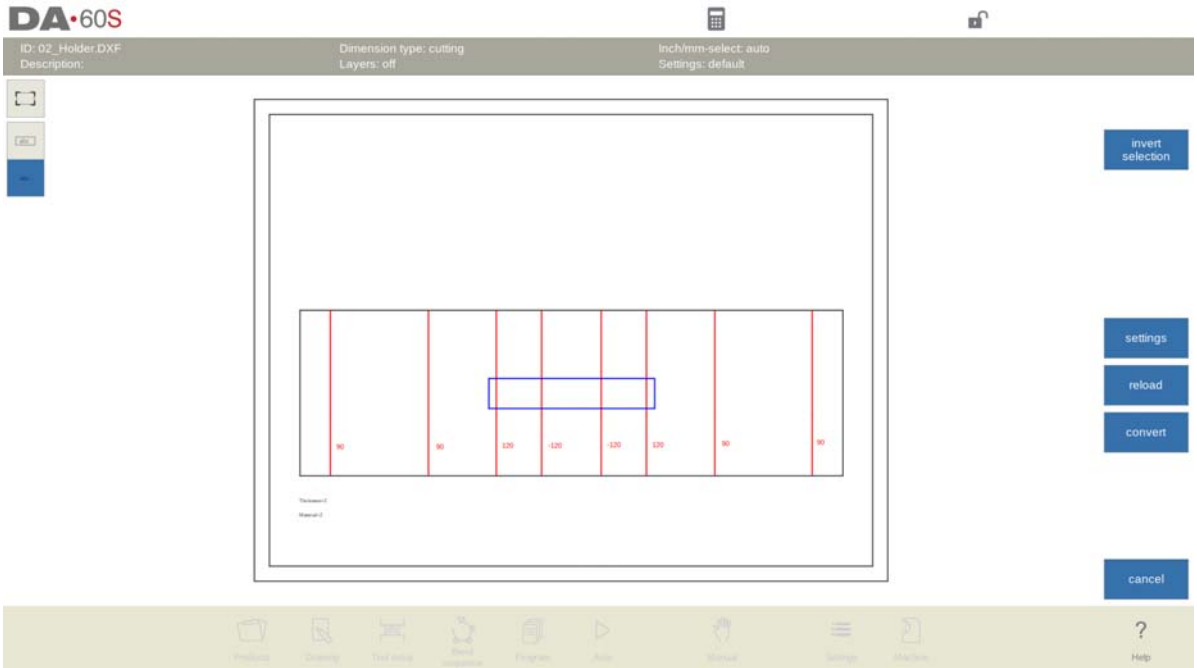


For the DXF file to be imported it is advised to create the original drawing as accurate as possible. Bend lines should be connected to contour lines to obtain an accurate product drawing. If this is not the case, the DXF Converter can correct small errors.

After selecting the DXF file the DXF import function window will open showing the DXF file. If this is with layer selection on, the drawing will be dashed, since no line is yet assigned to what it means.



If layer selection is off, the DXF file will be shown with the original line colors as been found in the file.


DA·60S
ID: 02_Holder.DXF
Description:
Dimension type: cutting
Layers: off
Inch/mm-select: auto
Settings: default
invert selection
settings
reload
convert
cancel
90
90
120
-120
-120
120
90
90
Help

2.3.1. Product drawing dimensions

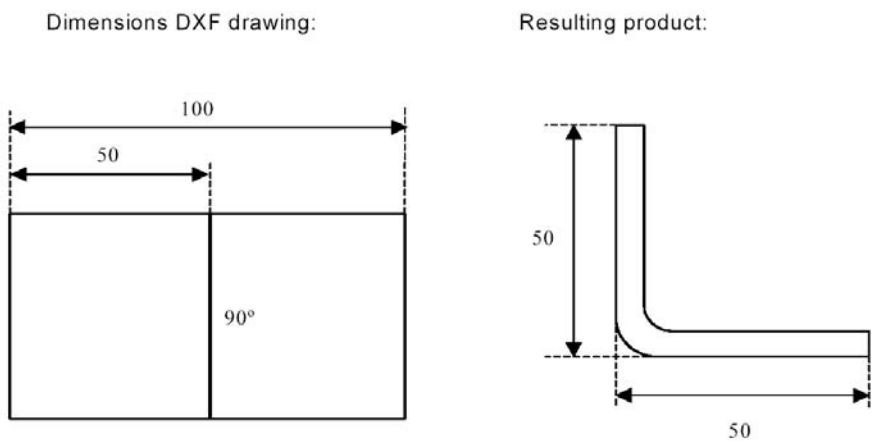
The drawing file can be organised in two ways:

- projection dimensions
- cutting dimensions (default)

These ways are described in the following subparagraphs.
During operation in the DXF converter, it is possible to switch between cutting dimensions and projection dimensions. This can be done in the DXF conversion settings.

Projection dimensions

In this situation all product sides and bend lines have the length of the resulting product. The drawing does not represent the real size of the sheet that must be bent, but is merely a representation of how the sheet is to be organised into bends and surfaces. When such a drawing is loaded and converted, the converter will construct a product drawing with exactly the same sizes as are present in the original drawing. Later on additional information is added about material, sheet thickness and product dimensions. It is left to the control to create a CNC program with proper axes positions that will result in a product as intended.



In this example, the length in the original DXF drawing may simply be 100, separated by a bendline. The resulting product will consist of two sides, each with a length of 50.
The dimensions are defined as outside dimensions.

Cutting dimensions

In this case the DXF drawing represents the exact sheet, as it is cut and will be used to bend a product of. When the DXF converter is set to convert cutting dimensions, based on material and thickness of the sheet, bend allowance information is required during the conversion process. If this information is not present the converter will ask for it.
The converter will use these dimensions and a bend-allowance table to construct a 3D product drawing with projection dimensions. When the product is used in the control, the same bend allowance table should be used to create a CNC program that can process a sheet as shown in the original DXF drawing.

2.3.2. Bendlines and Layer selection with Line assignment

For a proper conversion, the assignment of specific product properties to lines in the DXF is important.
Depending on the content of the DXF, the bendlines, contour and additional text information can be assigned layer by layer. In case layer selection is set off, the bendlines are being searched for automatically.

Bend line information

With the bendlines the angle information can be defined via text near the line. The text labels, to be configured in Settings::Labels, can be used as follows:

- Default: normal air bending with a positive or negative angle value.
- Hem: an H followed by a positive or negative value of the pre-bend angle.
- Radius: an R followed by the value of the radius.
- Bend allowance: an B followed by the value of the bend allowance.

Definition:

- positive value: the flange bends upwards,
- negative value: the flange bends downwards.

Product information

Beside the actual product drawing a DXF drawing can contain other information, such as manufacturer name, dimension lines, product description etc. If this information is organised in other layers than the product drawing then this information can be filtered out by selecting only certain layers for conversion. Otherwise, it possible to remove unnecessary information in the converter program before conversion of the drawing is started.

Layer selection

Depending on the DXF import settings, which can be entered from the main screen, layer selection can be set on or off.
In case of layer selection on, the layer property list visualization can be switched. The buttons in the left top corner enable this choice. Following paragraphs describe the difference between Layer selection switched on and Layer selection switched off.

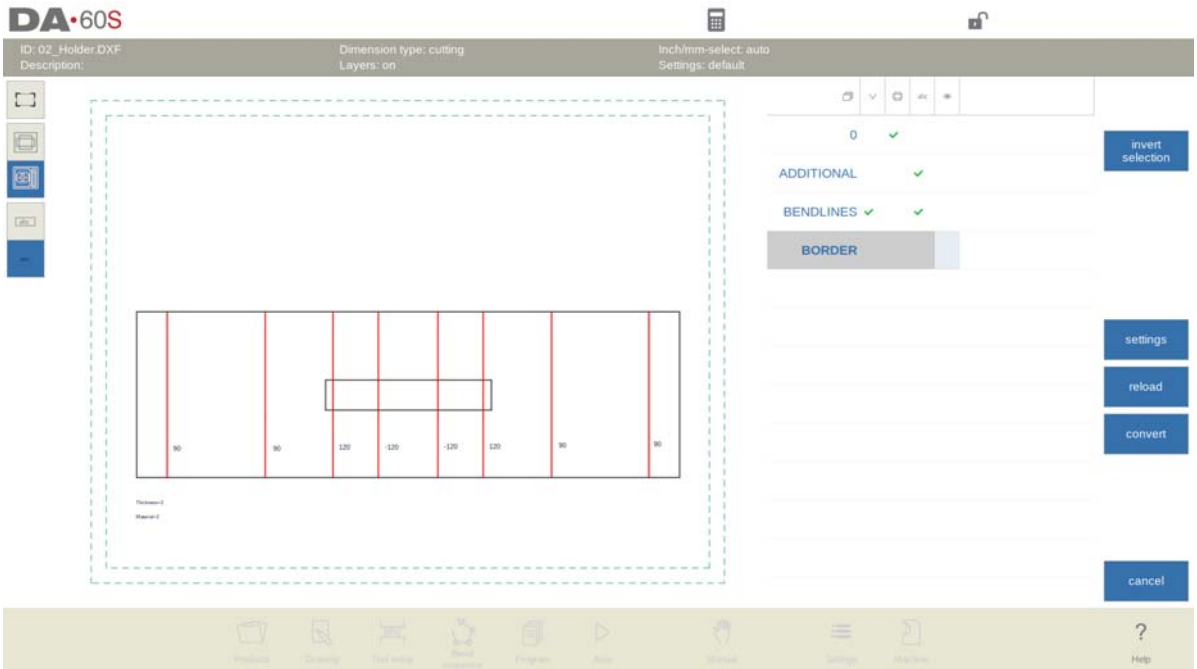
Layer selection switched on

With layer selection on, the available layers in the DXF are shown in the list.
In the header the symbols of angle, contour, text and visualization are shown and indicate the layer assignment. Each layer can be assigned to the specific property column by checking a check box. The layer can be assigned as bendlines or contour lines. The texts can be assigned as special information. This as configured in Settings::Labels.
The visualization switches a complete layer off or on. This does not exclude it from the later conversion.
If no check markers are set a layer will be skipped during conversion.

When tapping the layer name, the lines from that layer will be indicated.

The drawing will be visualized in dashed lines before any line is assigned.
When lines have been assigned following colors will indicate the assignment:

- Blue: Contour line, this line is a part of the outer contour of the product.
- Red: Bend line, this line is a bending.
- Black: Assigned texts will be shown in black.

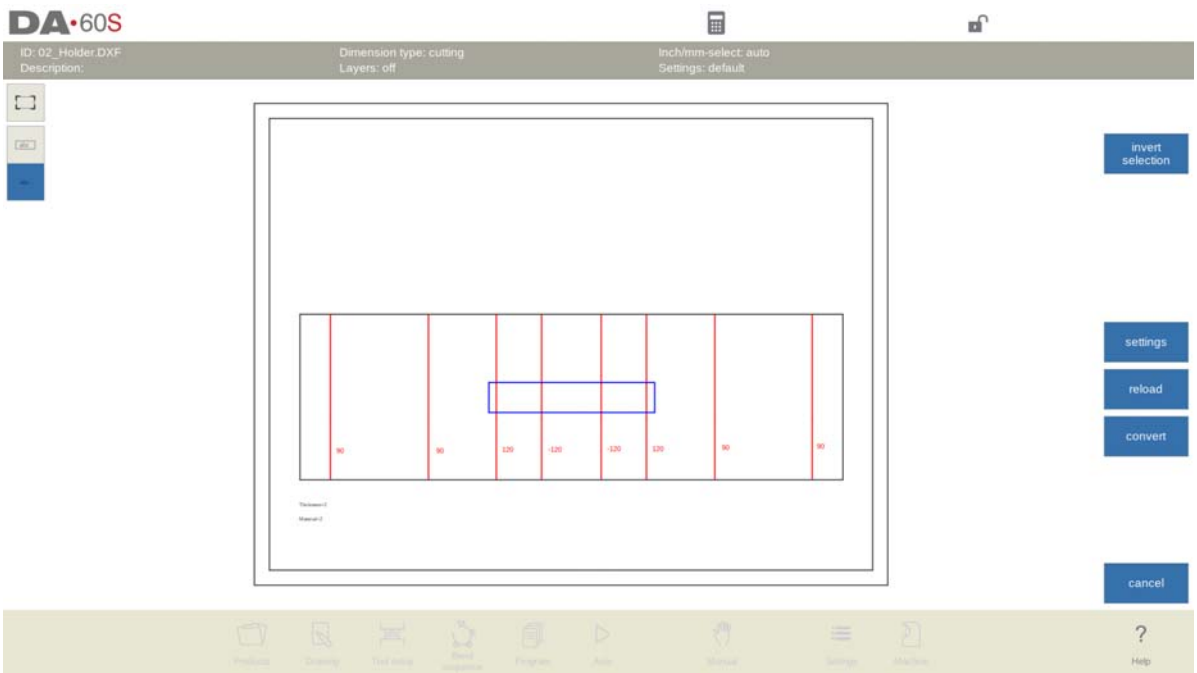


For bendlines, lines as well as texts can be checked. The text near the line indicates its angle value or radius or if it is a hemming. When assigning text to bendlines, the import will use this. For contour lines text information is not applicable.
The product properties, eg. sheet thickness and material, can be imported as text, based on the labels set in the DXF conversion settings.

Layer selection switched off

If layer selection is off, the control will automatically assign line properties. Line properties can only be viewed (length) not changed. After selecting a line, the property command button will appear.

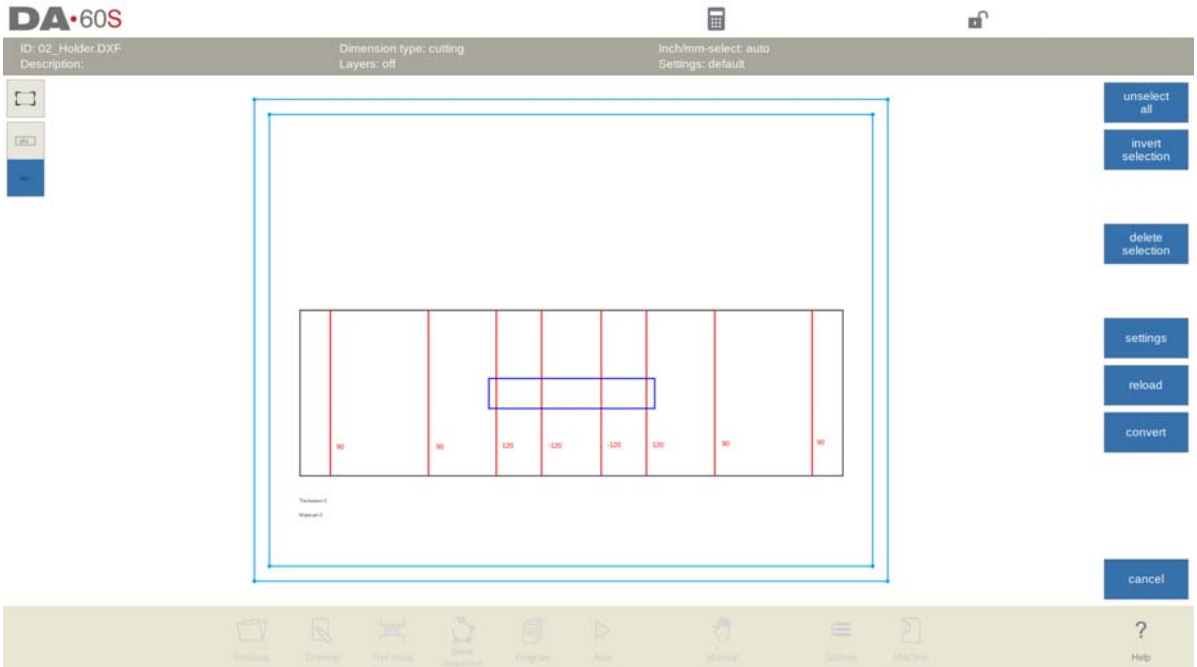
Colors shown are the colors from the original DXF.



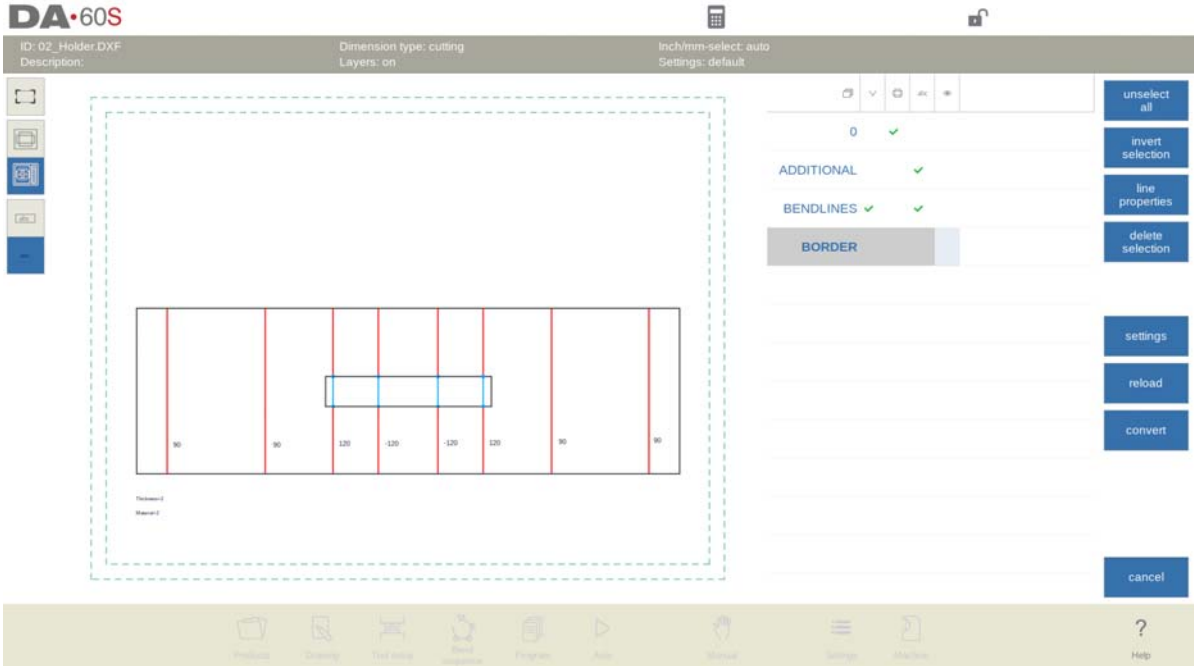
Line select

In the window of the DXF converter lines can be selected, by tapping on these lines, and selections can be inverted. When dragging a box over multiple lines, multiple lines can be selected at ones. Dragging from upper left to lower right will only select lines completely covered by the selection box. Coming from lower right up will select all lines covered by the box.
After selecting, lines can be deleted or, if layer selection is on, their joint properties can be changed. Eg. contour lines can be assigned as contour line by selecting them all and changing their properties to Contour. Polylines can be split into separate lines with the Split function.

Line selection with layer selection off:



Line selection with layer selection on:

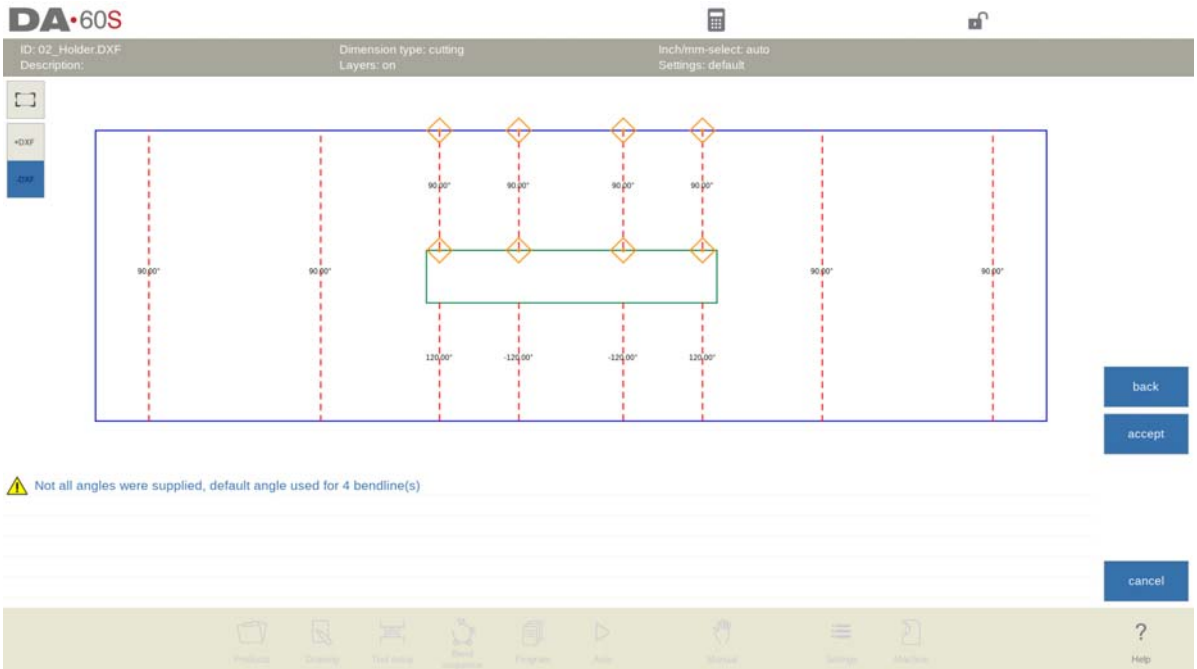


2.3.3. Conversion

When the assignments have been set properly, conversion can be executed by tapping the Convert button.

The conversion preview will be shown when there are warnings or errors.
During conversion the DXF drawing is represented by lines like contour line, bend line and inside contour lines. Colors indicate the property for the line conversion.The lines of the product drawing will have different colors after the conversion. Every colour has its own meaning:

- Blue: Contour line, this line is a part of the outer contour of the product.
- Red: Bend line, this line is a bending.
- Green: Inside contour, this line is a part of the inner contour of the product.
- Black: Assigned texts will be shown in black.

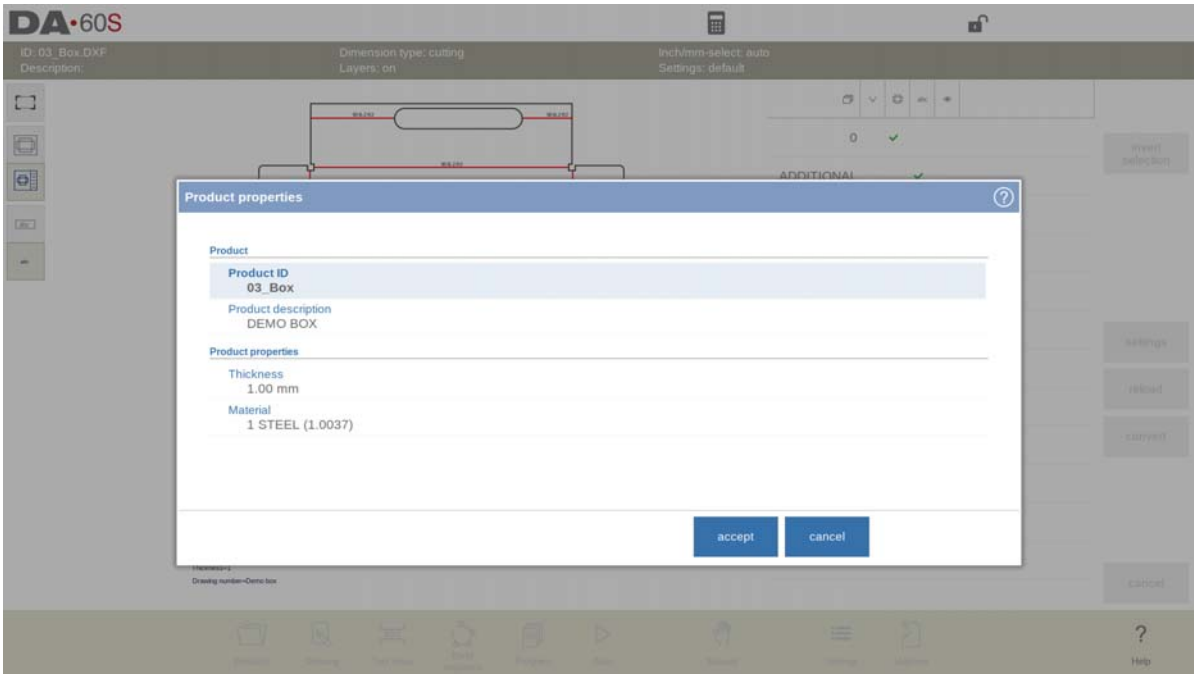


At the bottom of the screen warnings and infos will be shown if applicable. In the preview itself the warnings or errors will be indicated with red squares.

With tapping Accept the actual conversion will be done. If anything needs to be changed, the Back command button will step back to the previous screen.

Product Properties

After accepting the preview, converted Product Properties will be shown and can be modified.

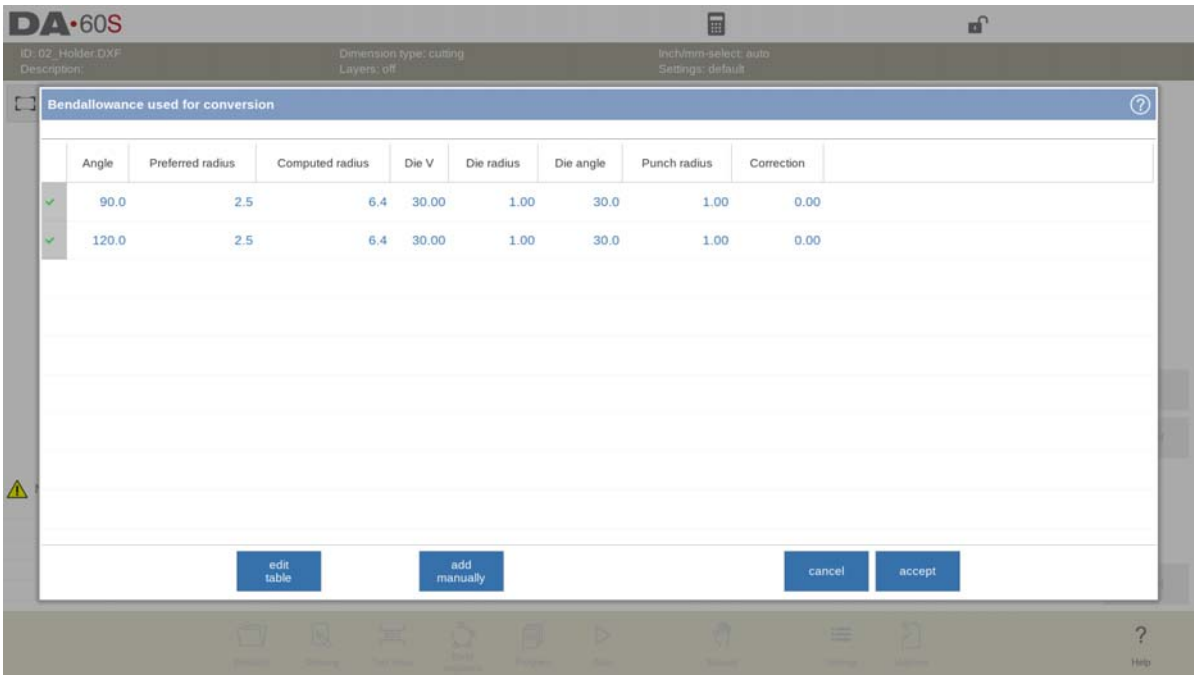


By accepting and closing this window the actual conversion will take place. The result of the conversion is shown in Drawing.
One can continue with the tool set-up and bend sequence determination as with a product drawn by hand.

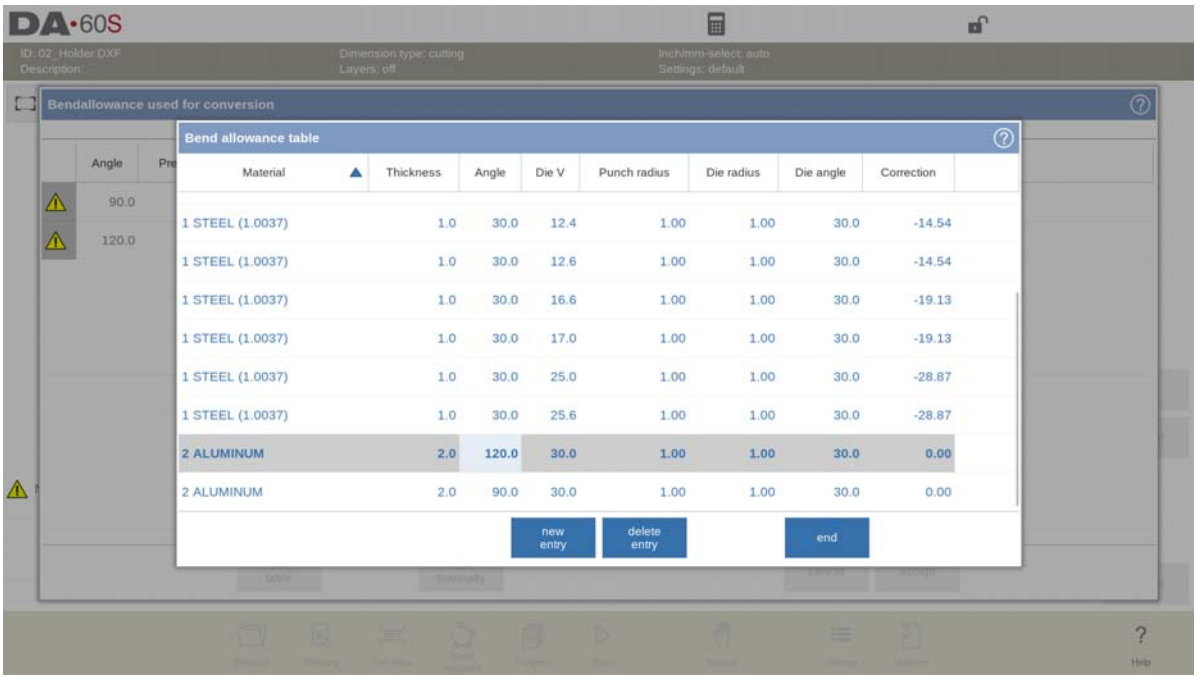
2.3.4. Converting cutting dimensions, with bend allowance info

At the last stage of converting a DXF with cutting dimensions, the bend allowance which has been used during unfolding, needs to be reused in the conversion.
Therefore the conversion of cutting dimensions will always use the bend allowance table of the control and it will check if for all bends bend allowance information is available.

In case there is just one set of bend allowance parameters available for each bend, this will be used. The bend allowance pop-up will show the angles from the product with the found bend allowance. If more entries in the table can be valid, one needs to select the appropriate bend allowance line. The preferred and calculated radius can be helpful in this selection.



If no bend allowance entry is available, the bend allowance table can be edited directly from this pop-up by tapping the edit table button. The following window will appear and the bend allowance information can be added to the table.



Adding Bend Allowance information manually

In case the bend allowance information is given with the DXF cutting dimension drawing, included in the DXF or as separate info, there is the possibility to manually enter this information.

When a DXF is imported and the bend allowance value per bend is included (based on the bendline info) these values are imported and used with the desired bend. The control will use this as an input which overrules the bend allowance calculation or table search of the control. If this information is entered during the DXF conversion, this will work equivalently, bypassing the controls calculation or table search.

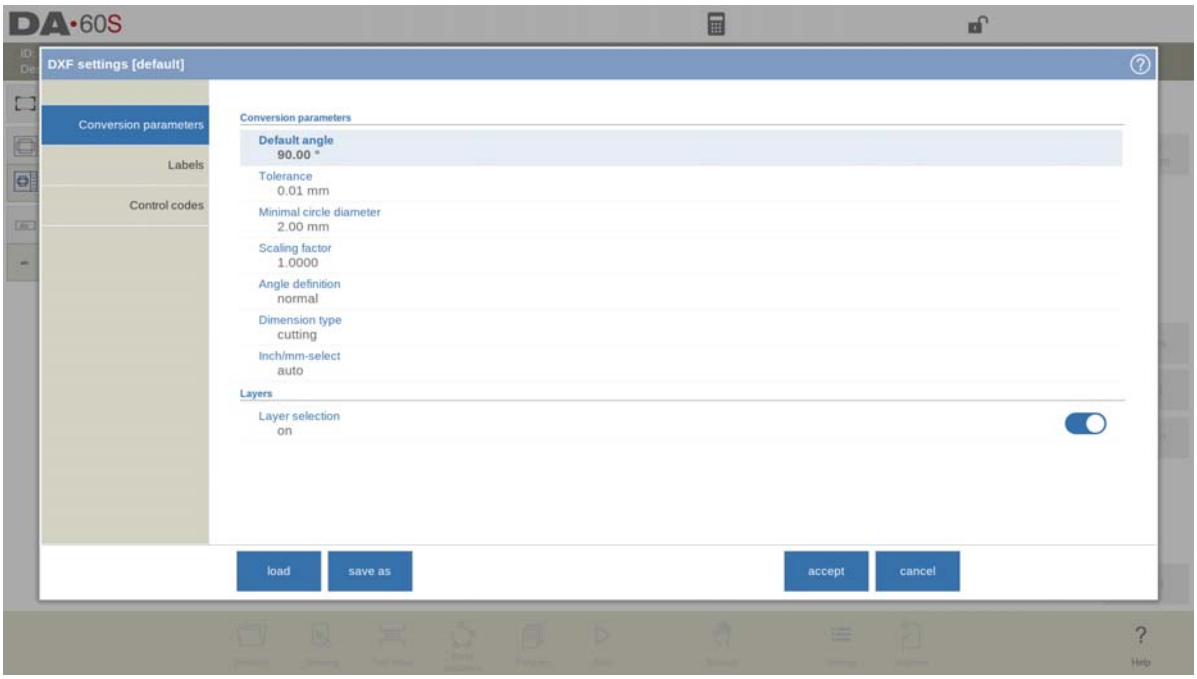
If the desired bend allowance information is not present from the table, one can also manually add the information just before actual converting. In case the bend allowance is not given before or during the DXF conversion this will be automatically prompted. One can select either to select from existing entries from the bend allowance table, or enter bend allowance just for this conversion.

To enable the function of programming bend allowance information in the Drawing function of the control (bend property) this needs to be switched on in the product properties. This is automatically done in case of importing.

2.3.5. DXF Settings

In the DXF converter settings the conversion parameters can be configured. It is possible to store multiple settings files for specific drawing types. Save as and load functions are available.

Conversion parameters



Default angle

Bendlines not being indicated with a bending angle value will be converted with this default angle.

Tolerance

The tolerance of non connected lines towards boundaries can be set. Lines within the tolerance will be connected to prevent failure during converting.

Minimal circle diameter

The minimal circle diameter of a circle to be converted, otherwise skipped.

Scaling factor

If the DXF is scaled, this factor can be set to unscaled to actual dimensions.

Angle definition

Normal angle definition or complement angles (180-alpha) can be chosen.

Dimension type

Either projection or cutting dimensions can be chosen.

Inch/mm-select

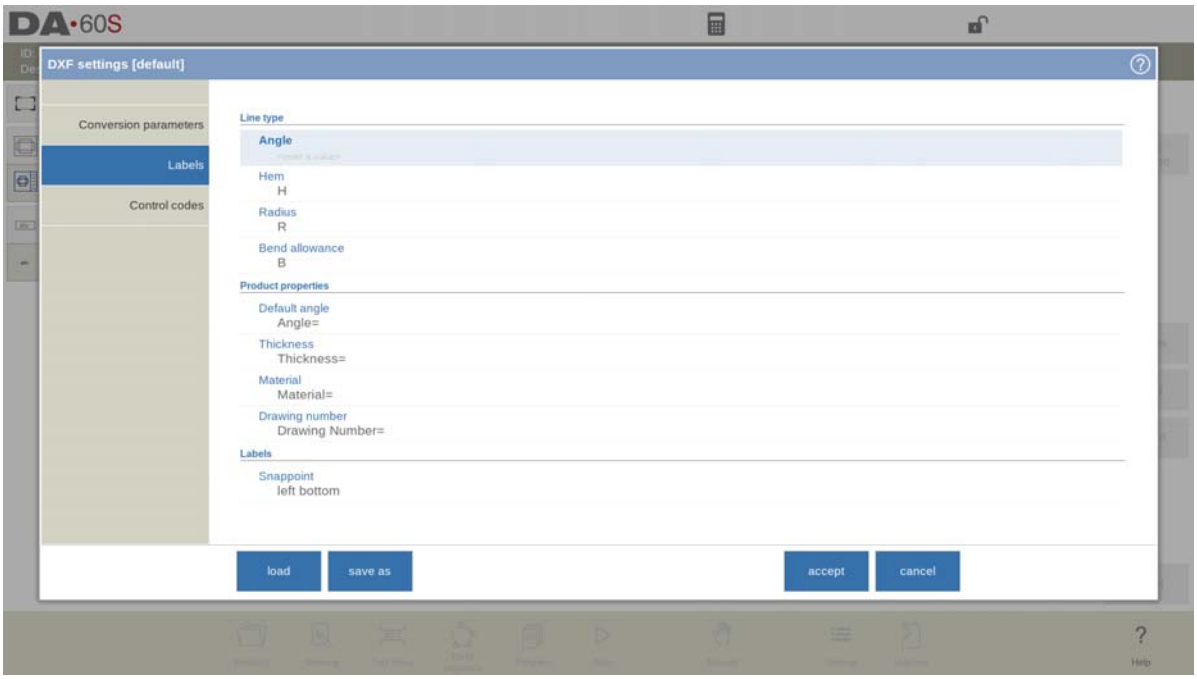
The conversion of the DXF will use mm or inches as a default, when set to auto the setting of the control will be used.

Layer selection

Layer selection on / off to enable the assignment of conversion properties by layer.

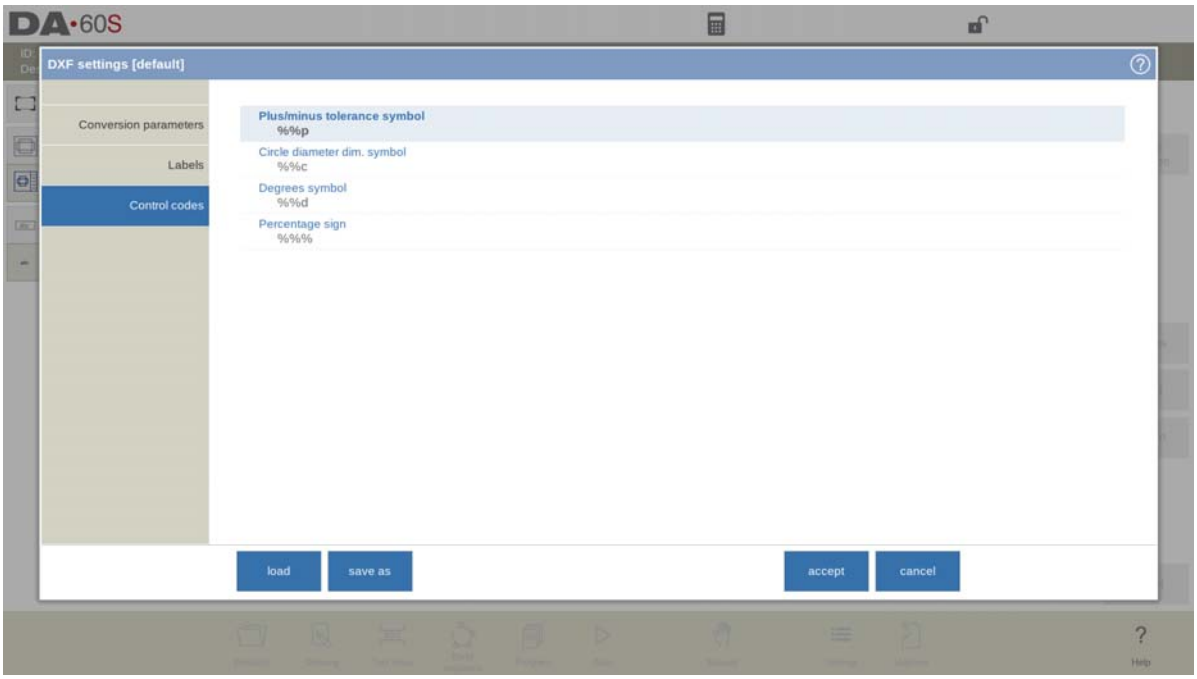
Labels

Line types and product properties can be labeled within the DXF in order to automatically assign them. With these settings the labels can be configured.



Control codes

If specific control codes are used in the DXF text, in this settings window these codes can be configured.



The DXF file

The DXF converter reads the entities it knows from the DXF file. The entities are: Text, Mtext, Line, SPLine, 3D, Line, Polyline, LWPolyline, Arc, Circle, 3D face, Solid, Trace and Insert. Furthermore:

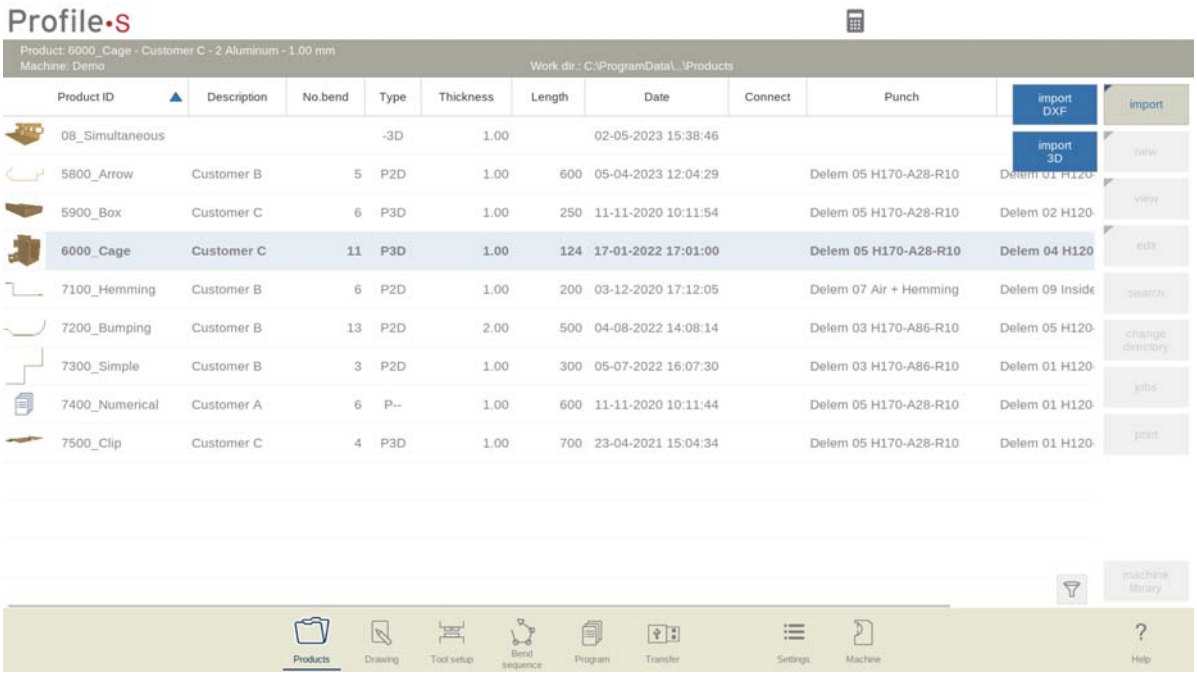
- The converter can handle ECS co-ordinates. Note: ECS is used in 3D drawings, so the converter interprets the values of the ECS for 2D.
- The drawing has to be in the X-Y plane.

2.4. The 3D import function (Profile-T3D offline only)

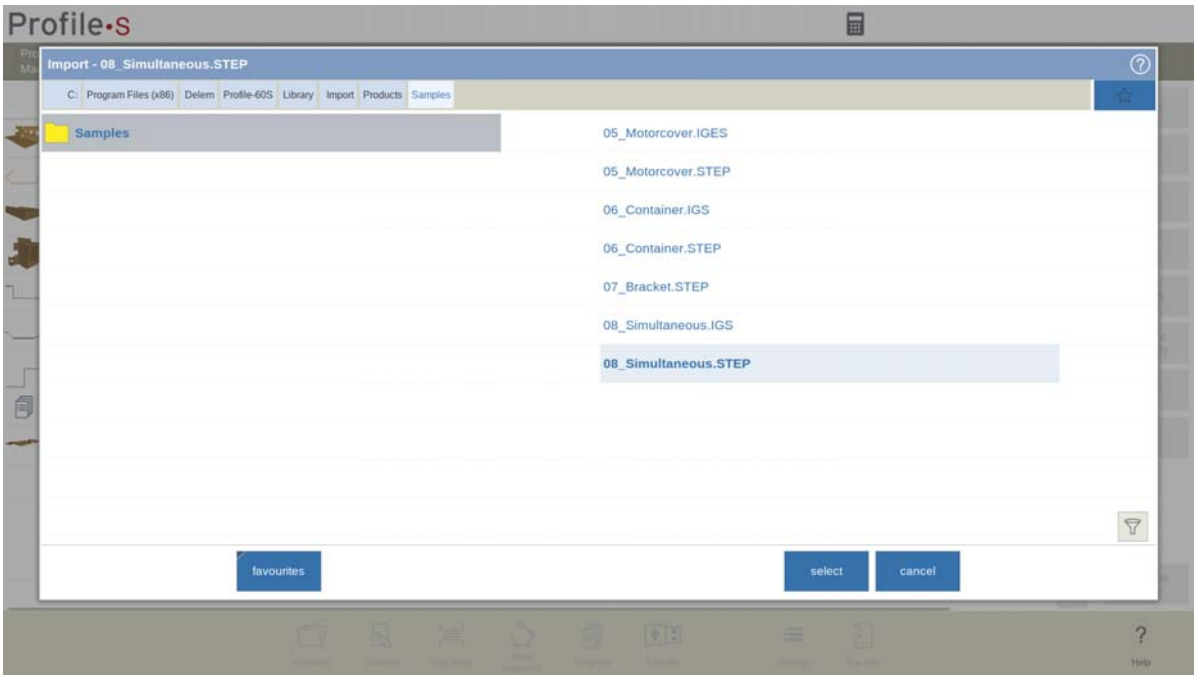
An alternative for drawing the desired product in the application is to import an externally generated CAD-system file. With the 3D import function Profile-T3D can import the generic .IGES and .STEP files.

This paragraph will explain the use of the 3D import functionality for .IGES and .STEP files. (This functionality is solely available in Profile-T3D)

The 3D import function is started with the Import command button Import 3D, which opens a file selection browser to select an .IGES or .STEP file.

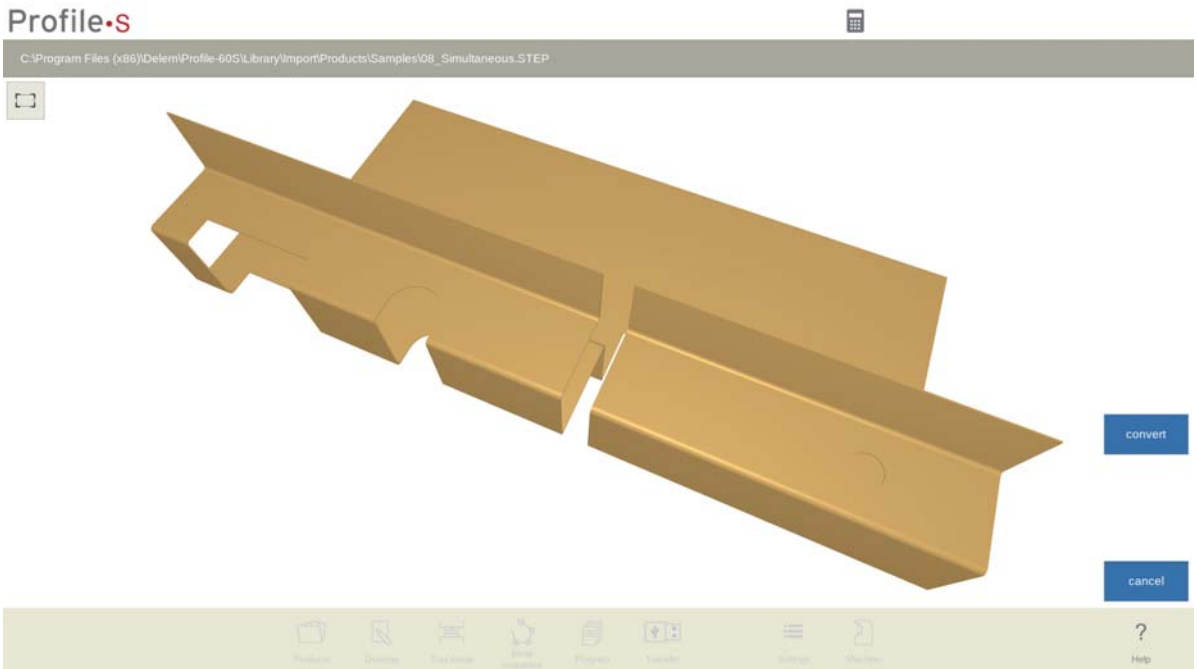


Files can be located on e.g. a USB stick or on the network directory. One can browse to the location and select the file.



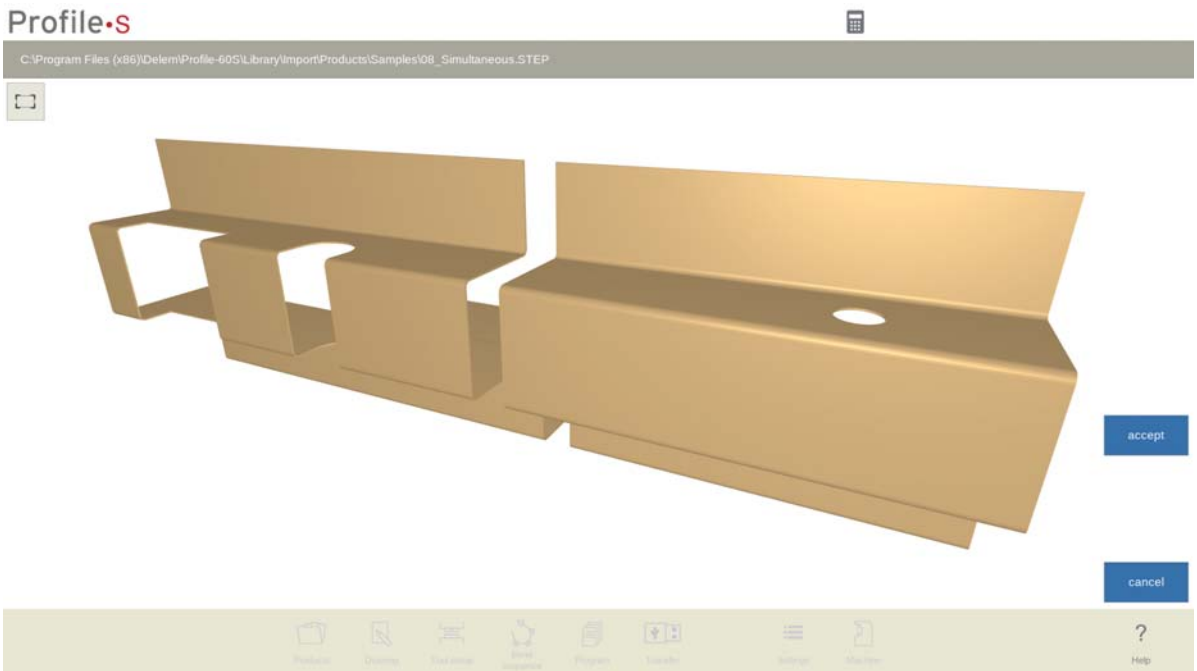
After selecting the desired file, the application will show the 3D file. The conversion can be started.

2.4.1. Conversion



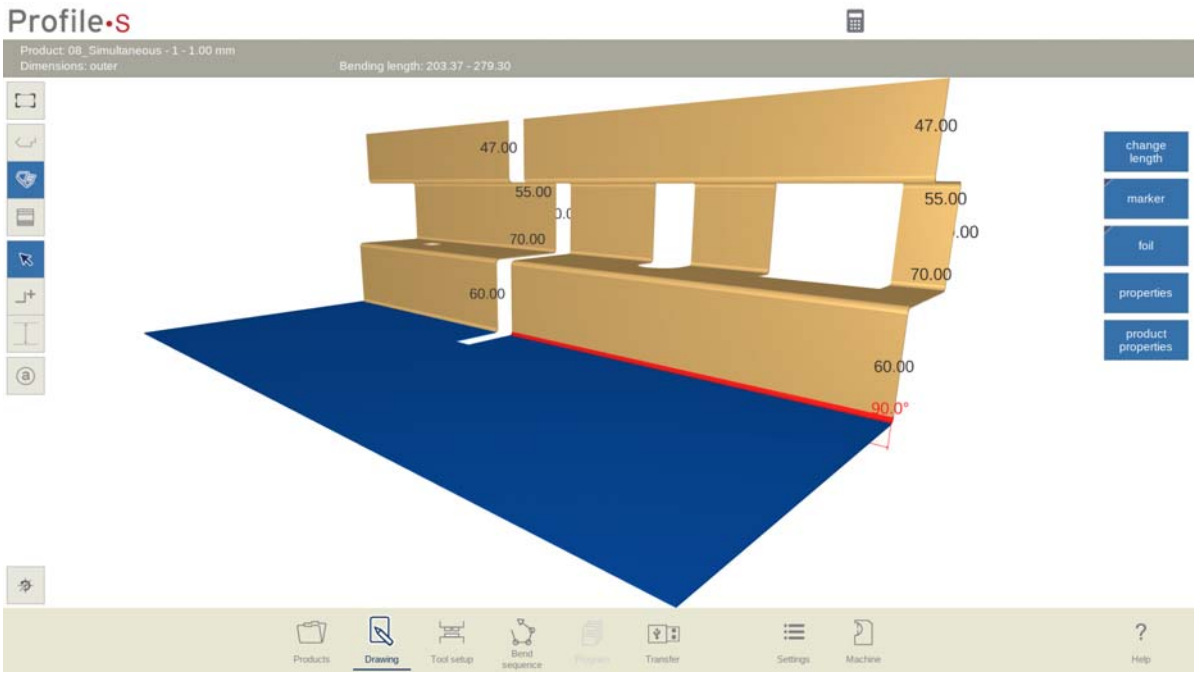
The conversion can be started by tapping Convert. This will start the interpretation of the 3D design into a product with bendlines and sheetmetal specific characteristics.

The application will show the product with the designated bendlines.



In case the converter does not find any imperfections the conversion is previewed and one can finalize the conversion with tapping Accept.

From here the product is converted and shown in Drawing mode.
One can continue with tool selection and bend sequence programming.

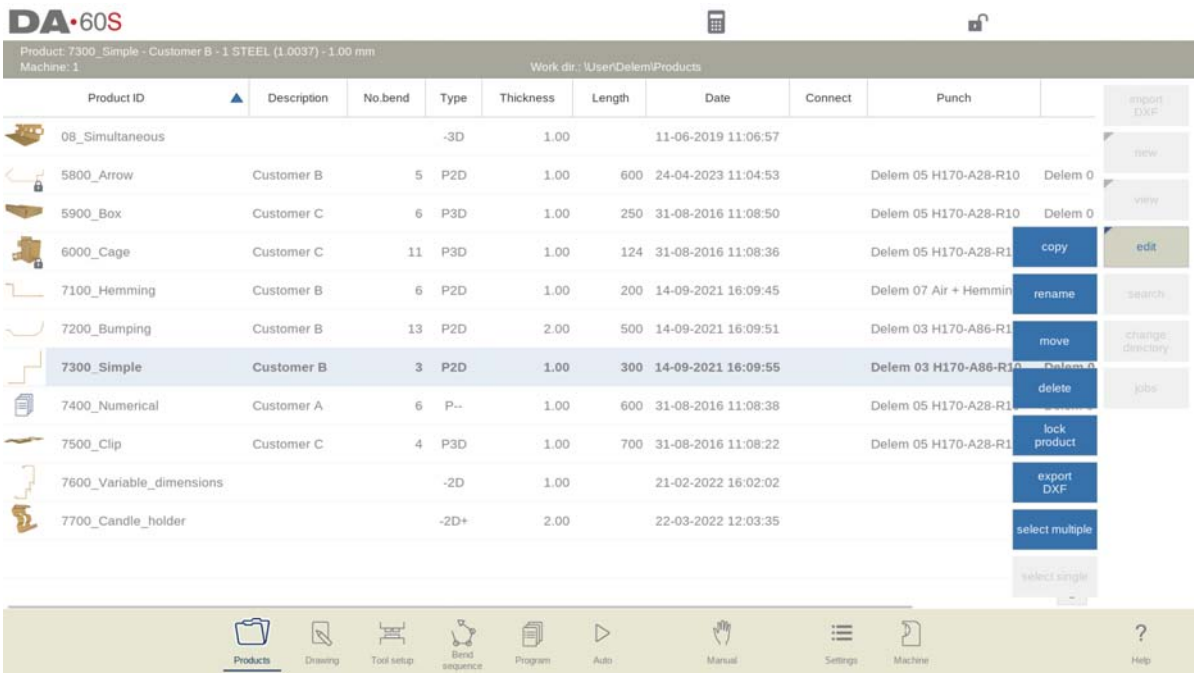


The 3D CAD file requirements

For the .IGES and .STEP files to be imported it is important to meet the constraints for sheet metal product designs. The constraints are available upon request at Delem.
The 3D file to be imported must of course be generated with the objective to design a sheet metal part which can be processed on a press brake.

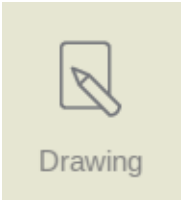
2.5. The DXF contour export function (only available when DXF option has been installed)

As part of the DXF-option the Export DXF function, on Edit in Products mode and on Transfer (Profile-T), enables the export of any product including the bend deductions as a contour. This contour is stored as a DXF and holds the cutting dimensions.



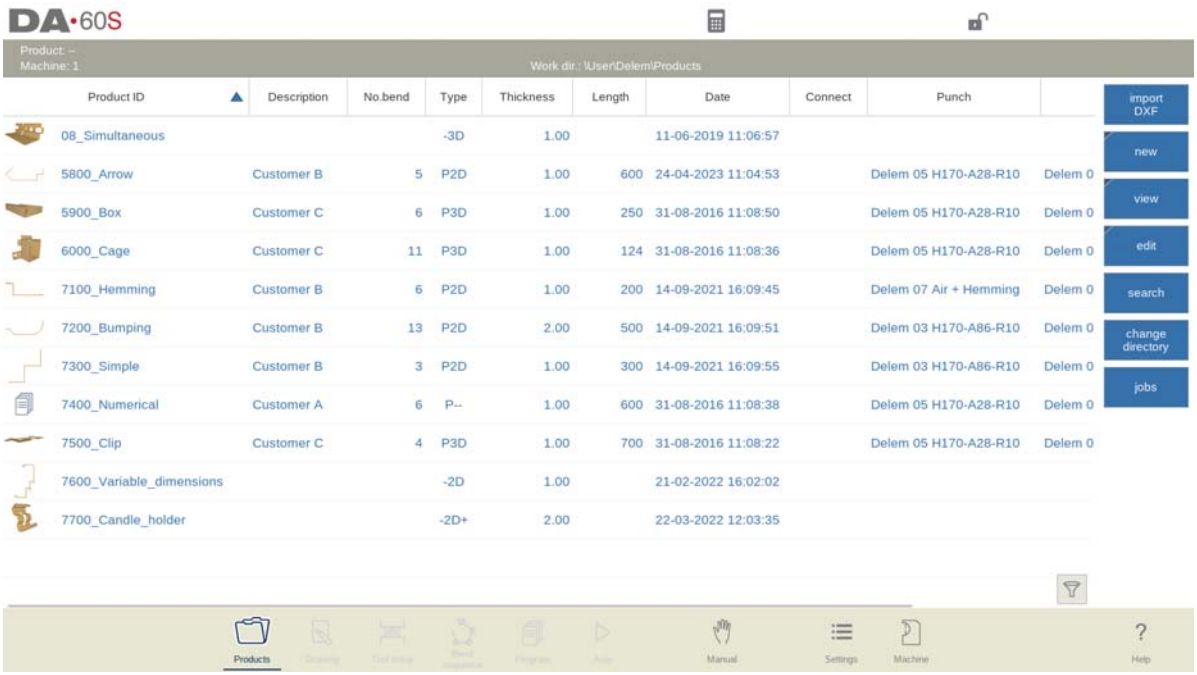
3. Product drawing

3.1. General product properties

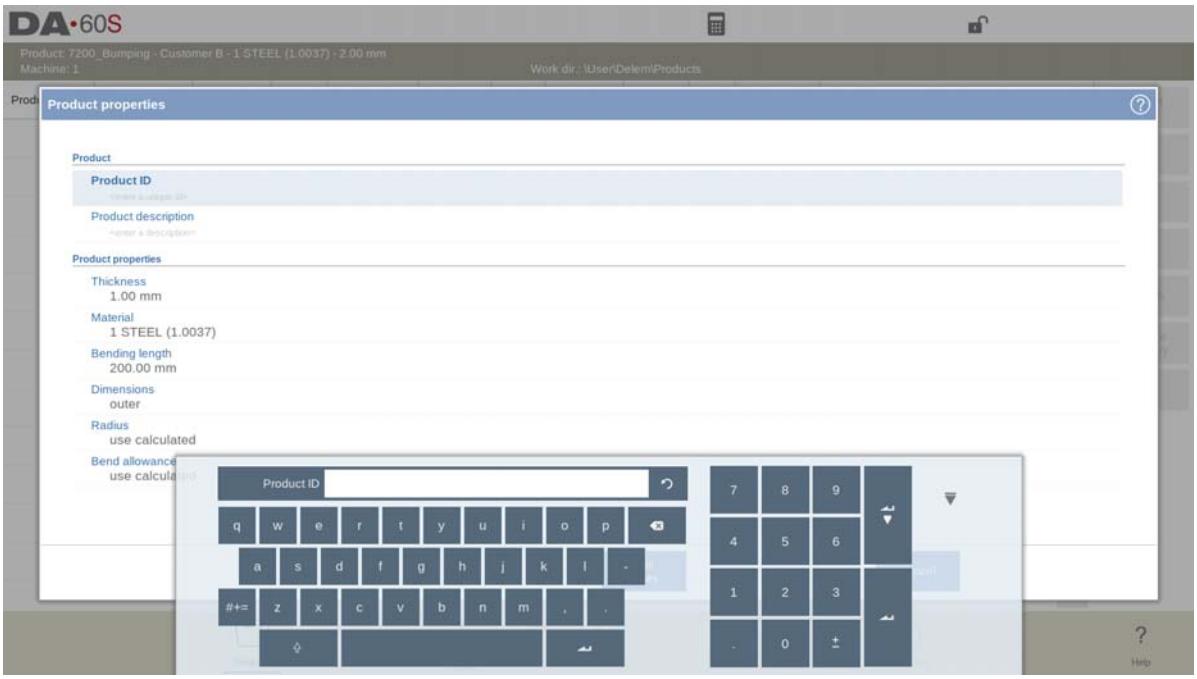


To edit an existing product drawing, choose the specific product from the Product library and select Drawing.

To start a new product drawing, choose New Product in the product library



When a new product drawing is started a screen with general product properties appears. First these properties, general data, should be set before starting with the product drawing.



Product ID

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

Product description

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

If an existing product ID is entered, a warning appears that this product already exists. You are asked whether to overwrite that existing product with the new product or not. If you choose 'yes', the existing product is erased. If you choose 'no', you must enter a new ID.

Thickness

Thickness of the sheet.

Material

Selection of the material type of the sheet. The control contains 4 pre-programmed materials. See Settings to program materials. Press the required material to select it as active.

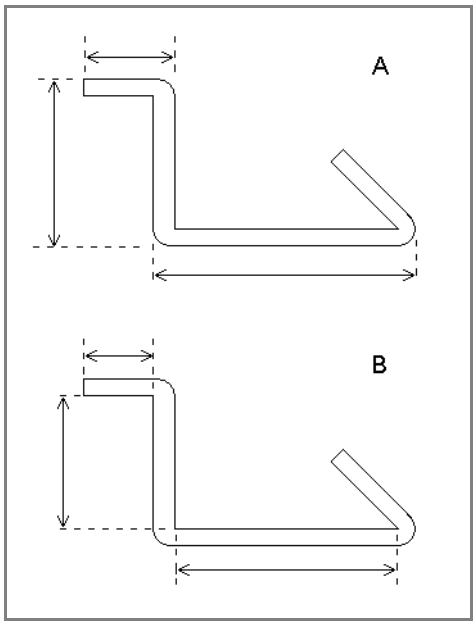
Bending length

The bend length in Z-direction of the sheet.

Dimensions

Determine the use of the outside (A) or the inside (B) dimensions in the product drawing when new sides or surfaces are added. The figure below shows the definition of both the dimensions. By default this parameter has the value of the Settings mode parameter

Default Dimensions.



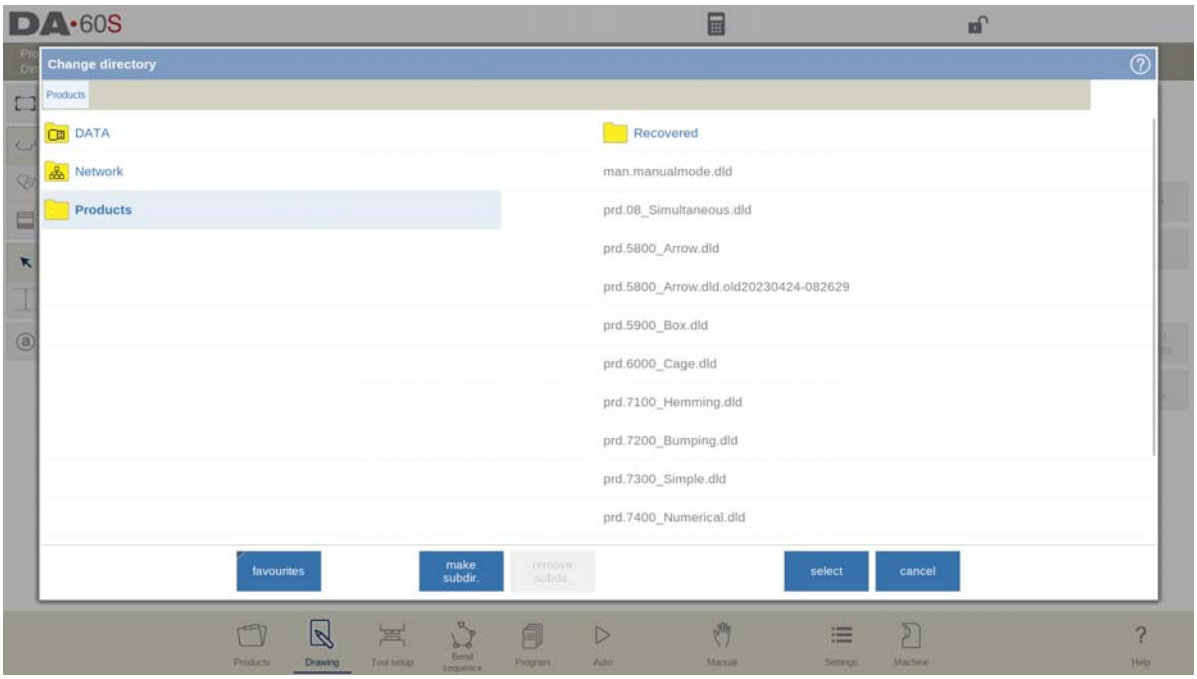
Radius

The product radius is recalculated after selecting the tools in the tool setup, because the chosen tooling influences the resulting radius. By default this setting is ‘use calculated’. With this parameter one can overrule and select ‘use programmed’; in that case the product radius will not be recalculated and used as programmed.

Bend allowance

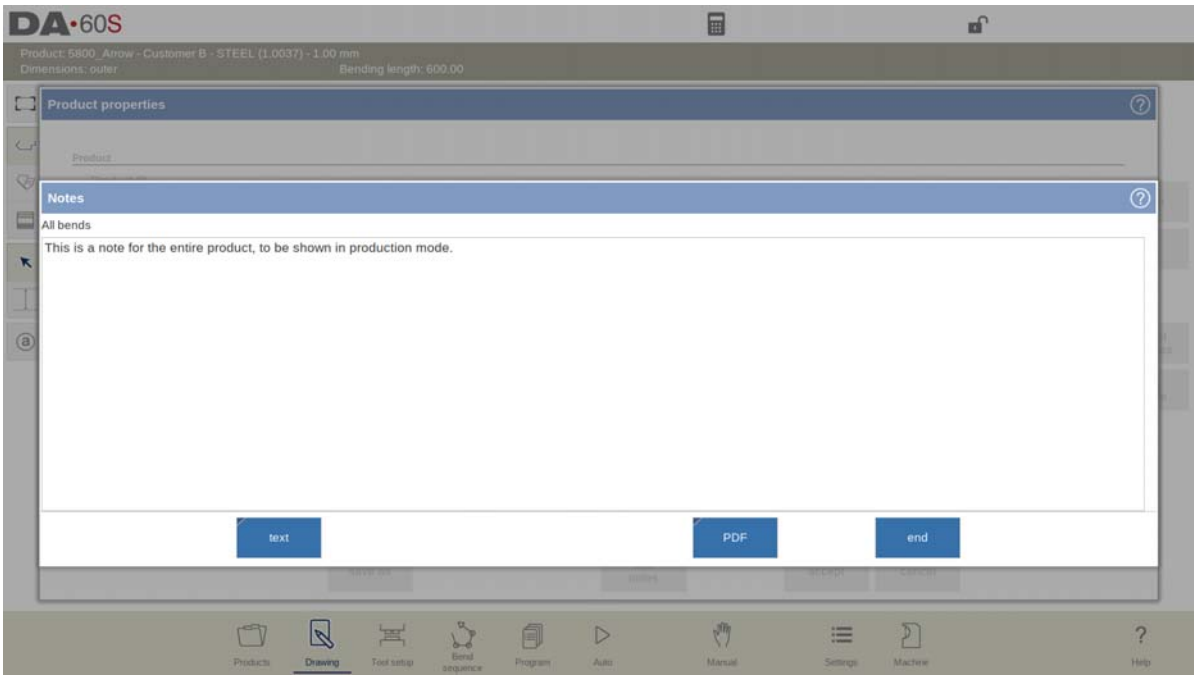
For the bend allowance of this product you could select ‘use calculated’ or ‘use programmed’. With ‘use calculated’ the bend allowance will be calculated with the Delem formula or the bend allowance table.

To change the active directory select ‘Save as’ and ‘Change directory’. The current product is automatically copied to the new directory.

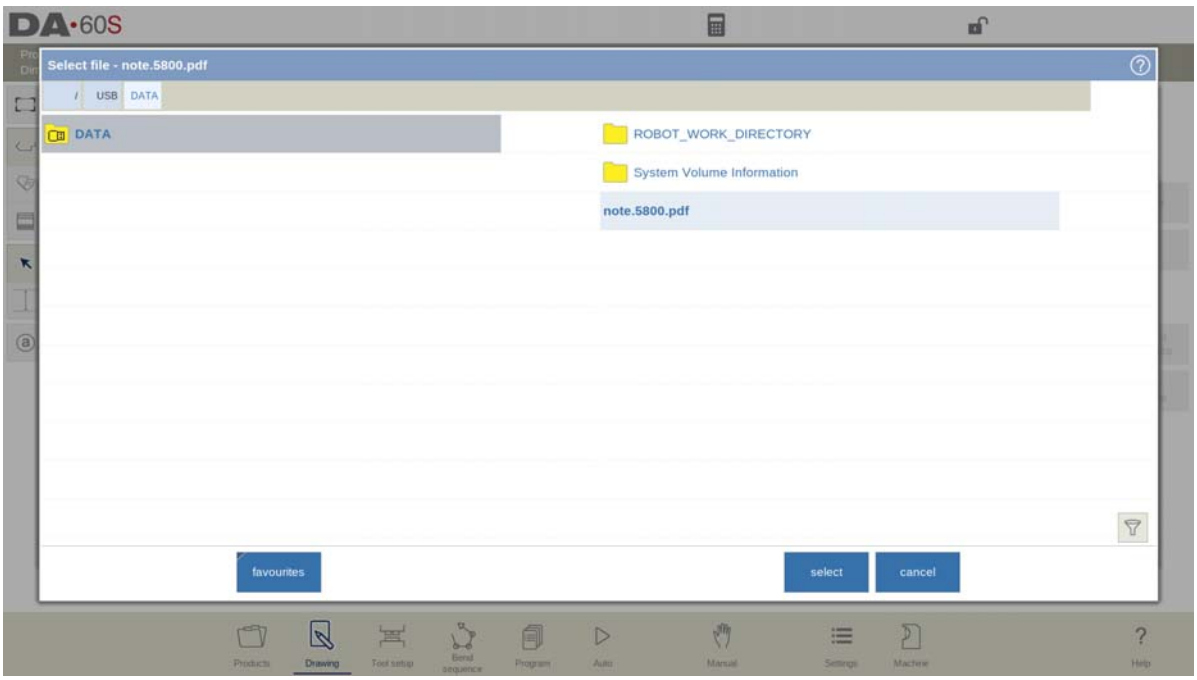

DA·60S
Change directory
Products
DATA
Network
Products
Recovered
man.manualmode.dld
prd.08_Simultaneous.dld
prd.5800_Arrow.dld
prd.5800_Arrow.dld.old20230424-082629
prd.5900_Box.dld
prd.6000_Cage.dld
prd.7100_Hemming.dld
prd.7200_Bumping.dld
prd.7300_Simple.dld
prd.7400_Numerical.dld
favourites
make subdir.
select
cancel
Products
Drawing
Tool setup
Program
Auto
Manual
Settings
Machine
Help

3.1.1. Add Notes

When Edit Notes has been pressed, a new window appears in which you can edit the text about the current product. The possible characters are displayed on the keyboard.



To attach a PDF file to the note tap Attach PDF. Via the directory browser a PDF file can be selected and will be incorporated in the product file.

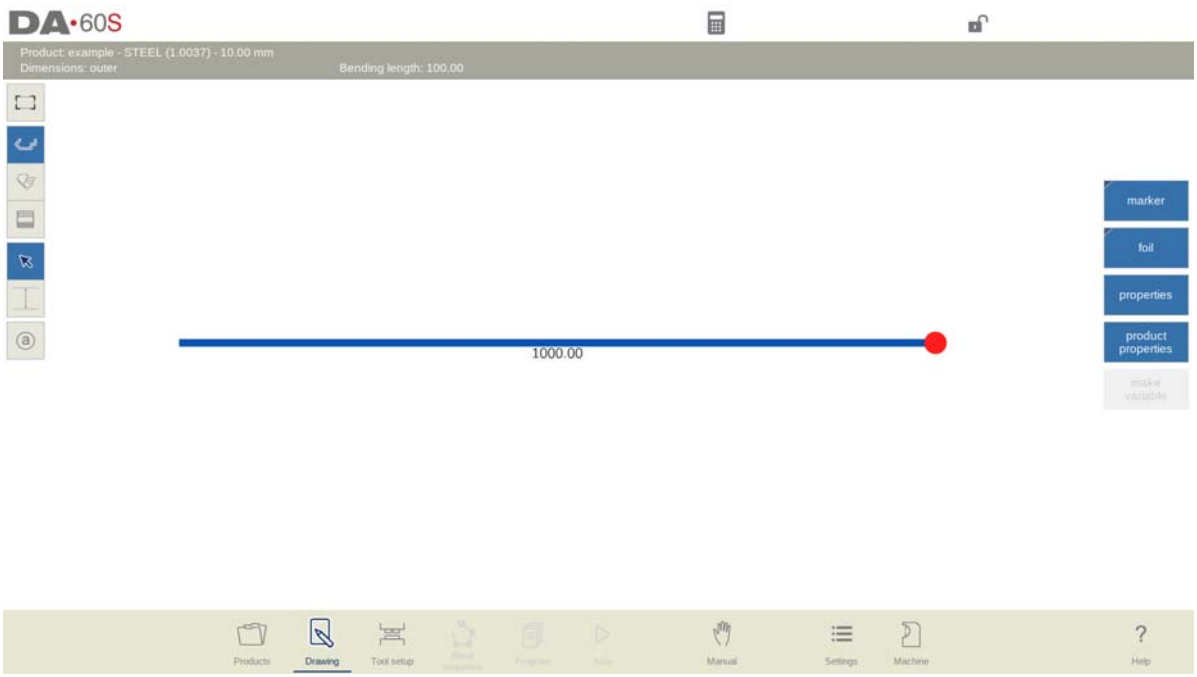


When solely a PDF file is attached without a textual note, the PDF will be shown instantaneously when the user presses the Notes indicator in Auto mode.

3.2. 2D product drawing

3.2.1. Introduction

After entering the general product data the drawing screen appears.

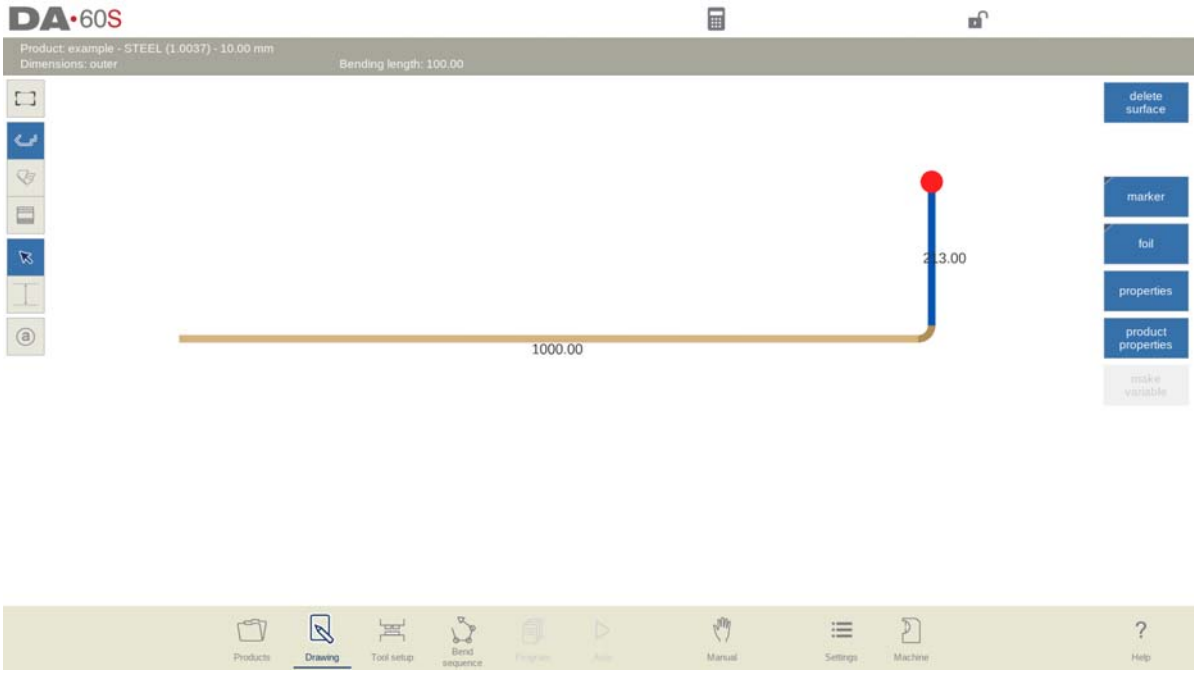


In the upper information row you will find the information about product ID, product description and inside/outside dimensions selection.

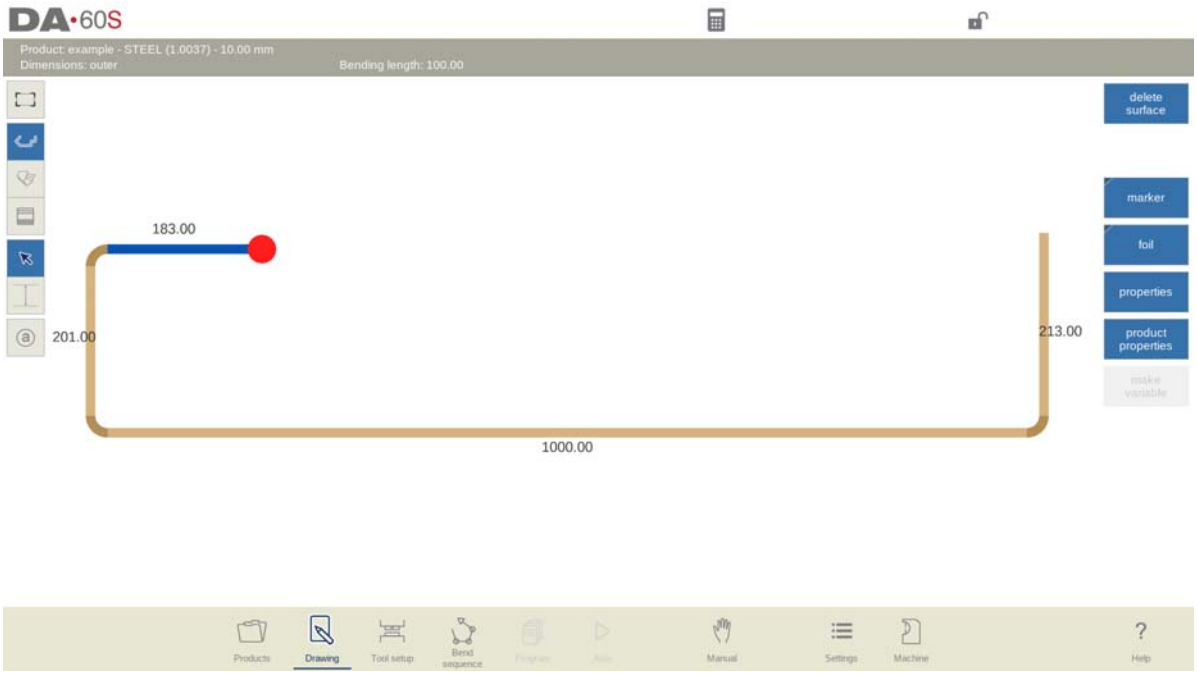
Now you can create the profile of the product. It is possible by using your fingers to tap and create quickly the product in 'sketch' mode. After that the real product dimensions and corresponding values can be entered by using the keyboard.

It is also possible to directly enter the angle of the bend followed by the length of that side by using the keyboard and the Enter button. The properties are prompted in the input bar of the screen on the keyboard panel. This procedure continues until the product has the desired profile.

The product data can be changed by selecting Product Properties. The properties of the product angles and lines can be changed by selecting Properties.



The currently active element (line or angle) is highlighted. In a product drawing you can program up to a maximum of 99 bends per product (graphical programming).

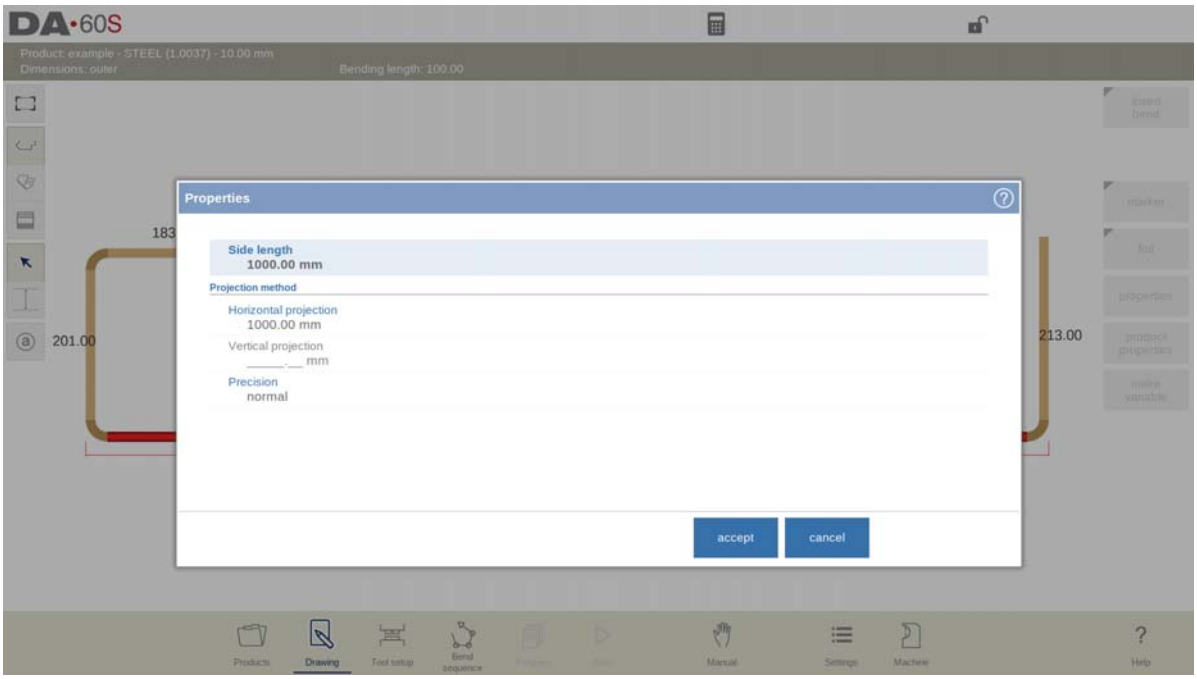


When the product drawing is finished it is possible to navigate to the next step in the programming process; to determine first the Tool Setup and after the Bend Sequence.

3.3. Line properties

3.3.1. Introduction

When the cursor is on one of the product lines it is possible to change the properties of that line by selecting Properties.



3.3.2. Projection

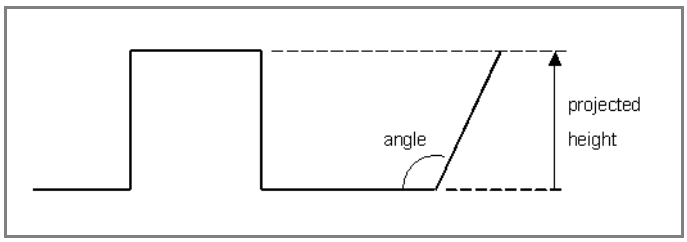
Inside the window with line properties, the following projection properties can be programmed:

Horizontal projection

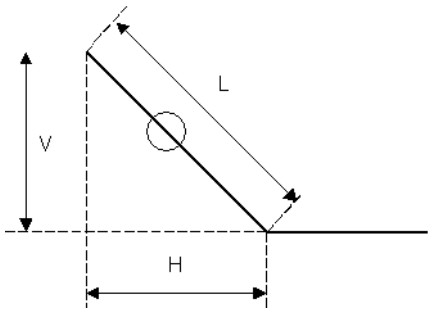
The horizontal distance a line must measure, regardless of its angle value.

Vertical projection

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



The projection function is a useful aid for drawing diagonal lines between points without having to compute the precise side length. When a line is selected, simply enter a horizontal or vertical projection distance and press Enter. The required line length is computed and applied to the selected segment.



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

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

3.3.3. Precision selection

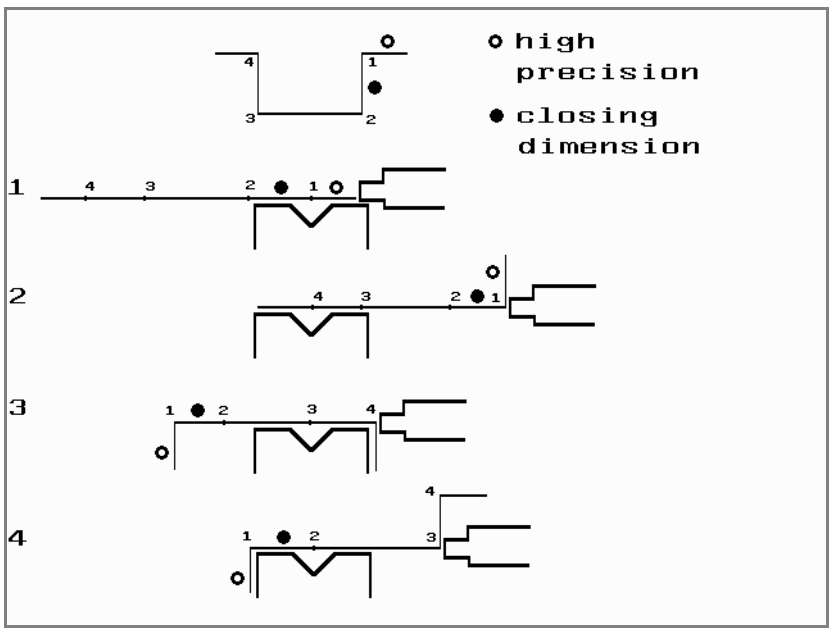
When the drawing cursor is on a line segment, you can select the level of precision for this line. Enter the properties and go to the parameter Precision.

Precision

Select the level of precision for a line:

Normal: Achieve normal accuracy for this segment.
High: At bend sequence computation the backgauge stop position will be chosen to get the highest possible precision for this line interval.
Closing dimension: At bend sequence computation the backgauge stop position will be chosen to get the resulting tolerances in this line interval.

Example:



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

Notes

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

3.3.4. Insert / delete bend

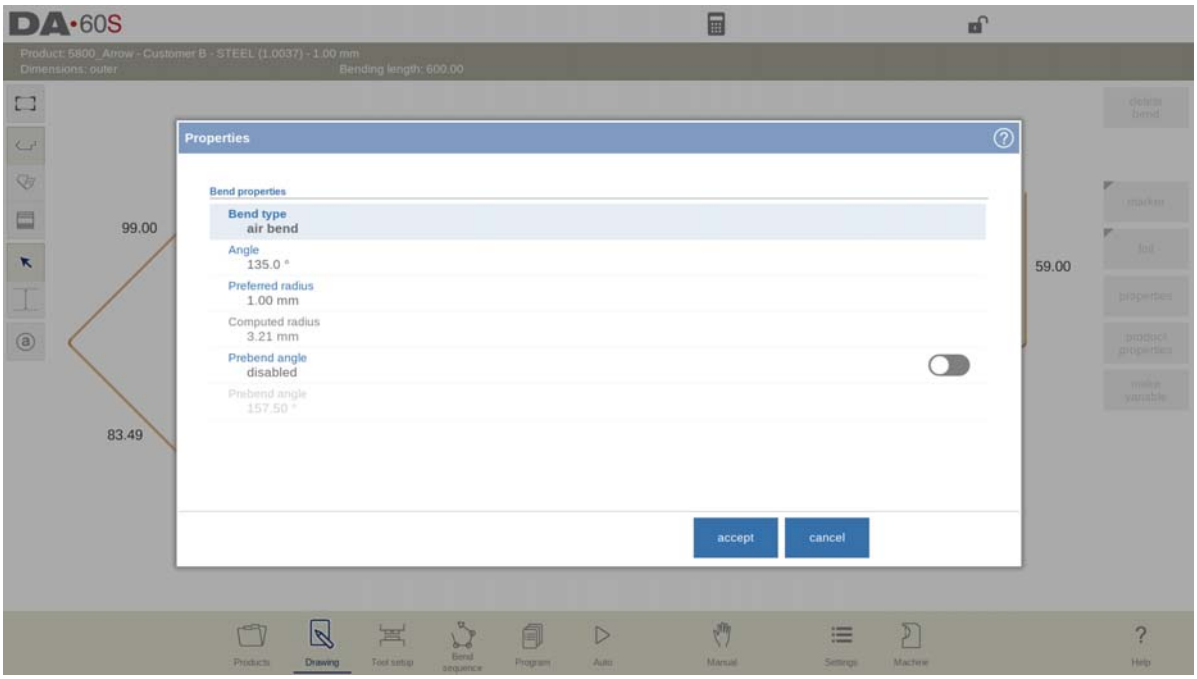
Dependent on the cursor selection it is possible to insert or delete a bend.

- If the cursor is on a line segment (except for the end line of the product), it is possible to insert a new bend. The side length will be replicated to the new bend.
- If the cursor is on a bend (angle), it is possible to delete that bend.
- If the cursor is at an end line of the product, the line can be deleted.

3.4. Bend properties

3.4.1. Air bend

Drawing a product graphically is simply programming the line length, angle value, next line length, etc. till the product has its required shape. The bends in the product have their standard or specific properties. The bend properties can be set by selecting the bend and selecting Properties.



Bend type

With the bend type it can be determined what kind of bending process is required. Options are:

- Air bend as normal bend with a defined angle and preferred radius.
- Bumping as bend process to realise a large radius in different steps of air bending.
- Hem bend as bend process including the pre-bend (a standard air bend with a sharp angle) and hemming action with help of a special hemming tool.

Angle

The angle value to bend.

Preferred Radius

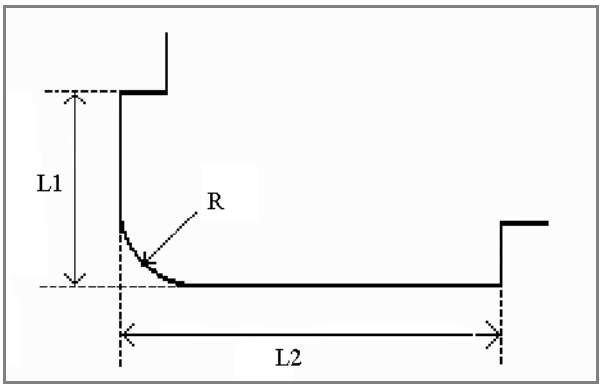
The intended radius which is programmed. This radius can also be a large radius requiring large radius tools.

Computed radius

The resulting radius as computed from the control settings. Amongst others the resulting radius is depending on the used tools in the bending process.

A large radius is meant to be bent with a special radius punch with large radius. If such a punch is not available, the bumping method can be selected. The radius value may not exceed the length of any adjacent sides. For the definition of the line lengths to be

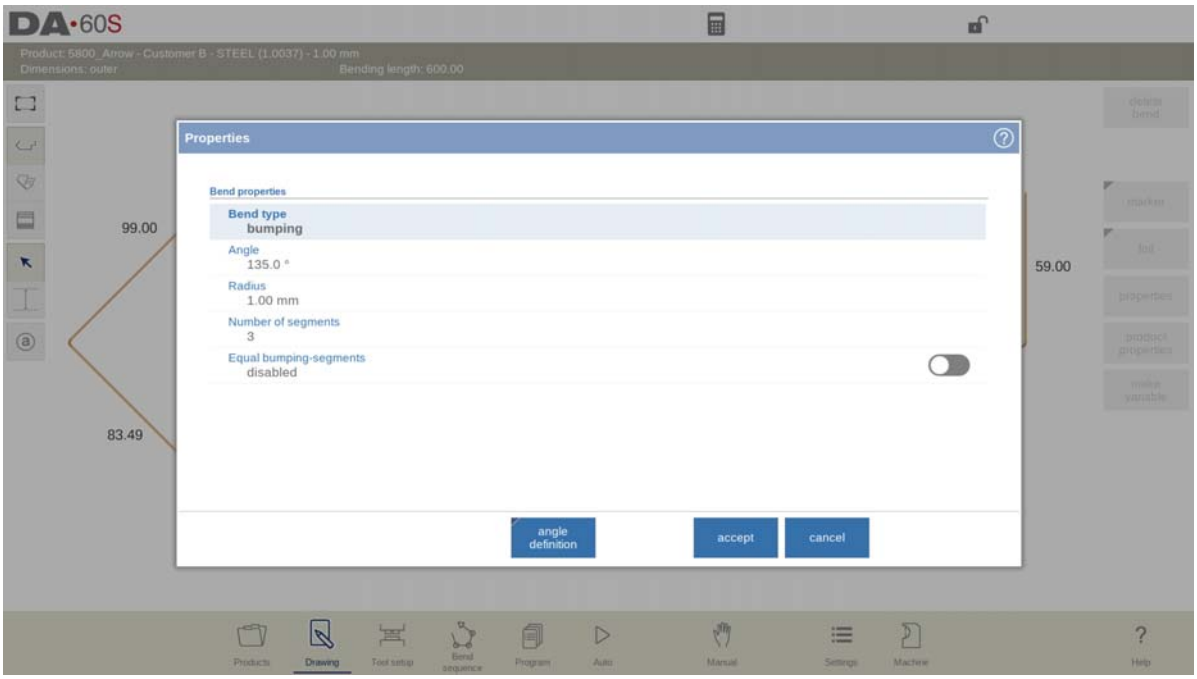
programmed in the part connected to a radius bend, see figure below.



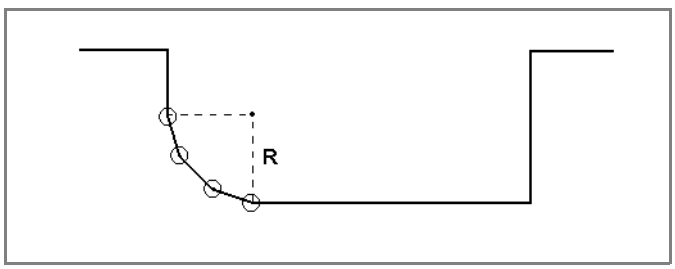
Lengths L1 and L2 must be equal or bigger than the radius R.

It is also possible to create a bend by placing the cursor on the flange end where the bend is required and select Properties. In this way the pop-up window will appear with an extra parameter to fill-out.

3.4.2. Bumping

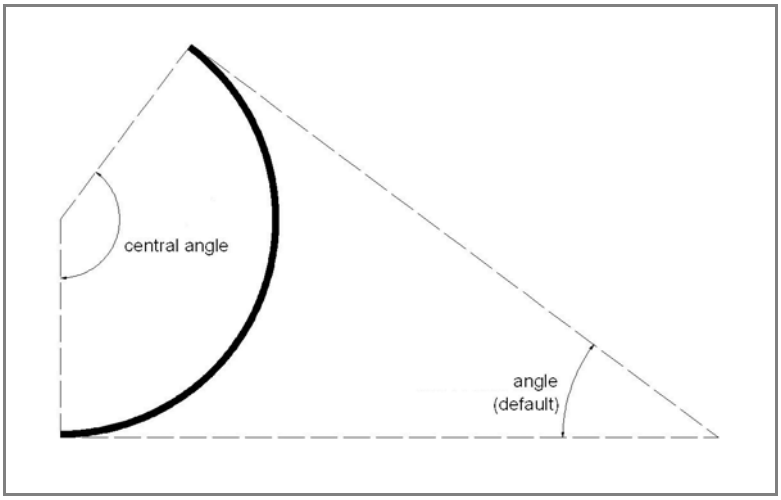


If a tool with large radius is not available, the bumping method can be chosen. With this method, a large radius in a product is obtained by a series of slight bends in succession.



First you can choose the Angle Definition. The available definitions are:

- The default angle is the angle which could be programmed as standard.
- The central angle is the supplement of the default angle (i.e. 180 degrees - default angle).



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

Central angle

The supplement of 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)

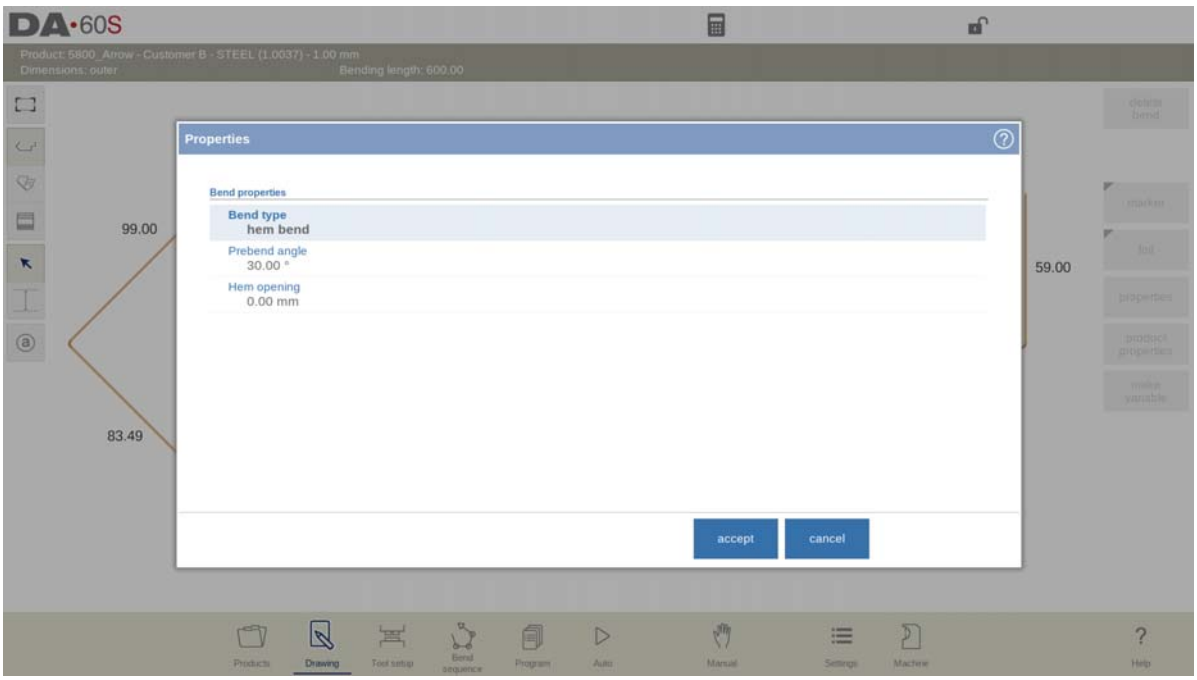
When Enabled all segments will have an equal size.
When Disabled the calculation is including half size segments. If in this case a problem with the size of the V die is detected in the bend sequence determination, the user is asked whether or not to select a re-calculation with equally sized segments.

When you must program such a bend, first program a standard angle with adjacent sides. Then put the cursor back on the angle, tap the function Properties and select Bumping as Bend Type. You will be prompted to program the radius, the number of segments and if you like to have equal bumping segments or not. After programming these parameters the radius

is drawn in the product and a message will appear on the screen with information about the smallest calculated segment length. The smallest segment length has consequences for the selection of the die.

3.4.3. Hem bends

When creating the required profile of the product with a hem bend it is possible to first prepare a flange with a prebend angle, place the cursor on the bend and select Properties. The bend properties can be programmed in the pop-up window.



It is also possible to create a hem bend by placing the cursor on the flange end where the hem bend is required and select properties. In this way the pop-up window will appear with an extra parameter to fill-out.

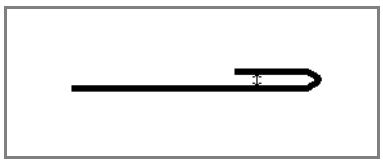
Bend properties

Prebend angle

The required angle of the prebend, normally a sharp angle. The angle can be set upon the best practical value, the default value is 30 degrees.

Hem opening

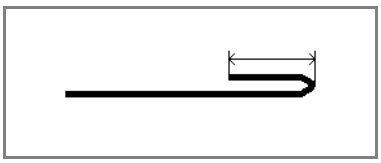
The hem bend can be made with a certain opening distance between the 2 flanges. The hem opening value will be used calculating the beam position in the hemming process. By default this parameter has the value of the Settings mode parameter Default Hem Opening.



Side properties

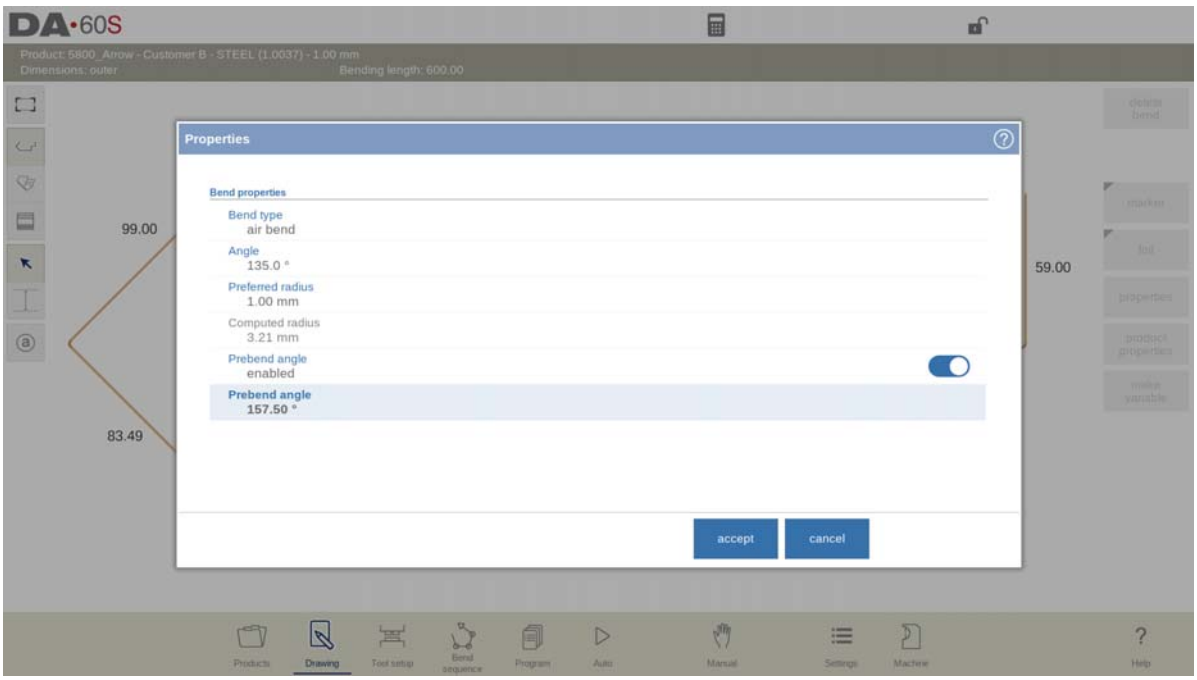
Side length

The length of the flange to hem.



3.4.4. Prebend, two stage bending

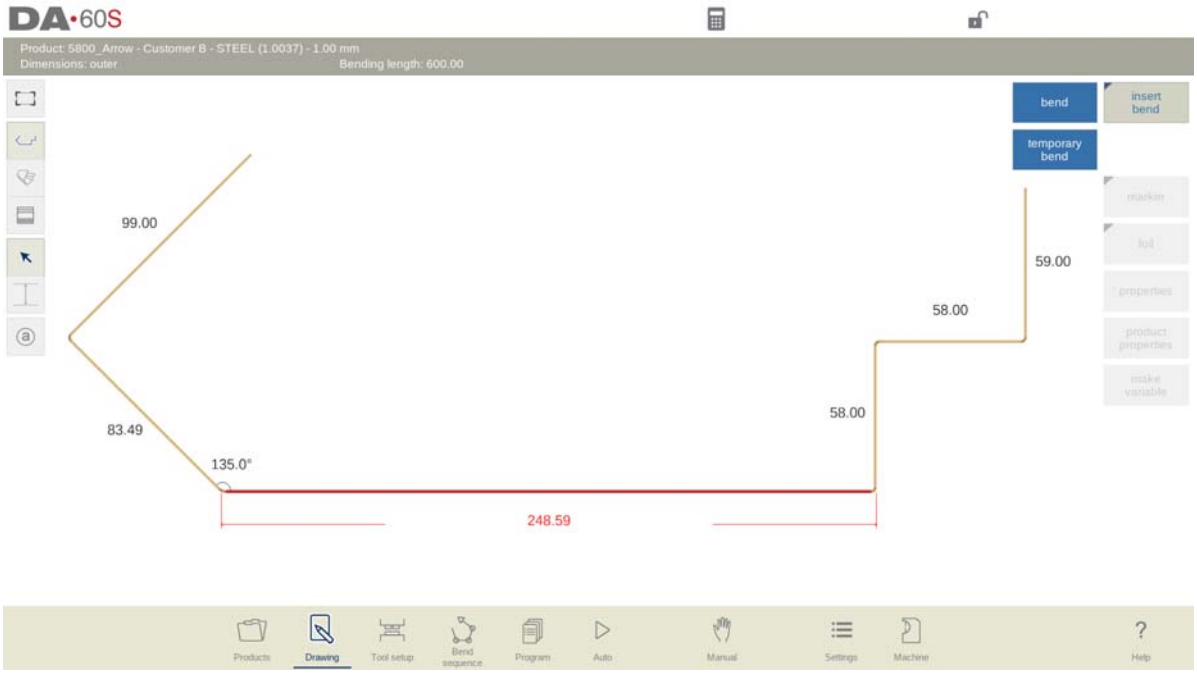
It can be helpful to make a normal bend in two stages, enabling a specific bend sequence, one can opt to make the bend with a prebend. The property to add a prebend to an angle in a product can be found in the properties of any bend. When programming it as such, the control will divide the bending of that particular bend in a prebend and the final bend, as two separate bend steps.



3.4.5. Insert temporary bend

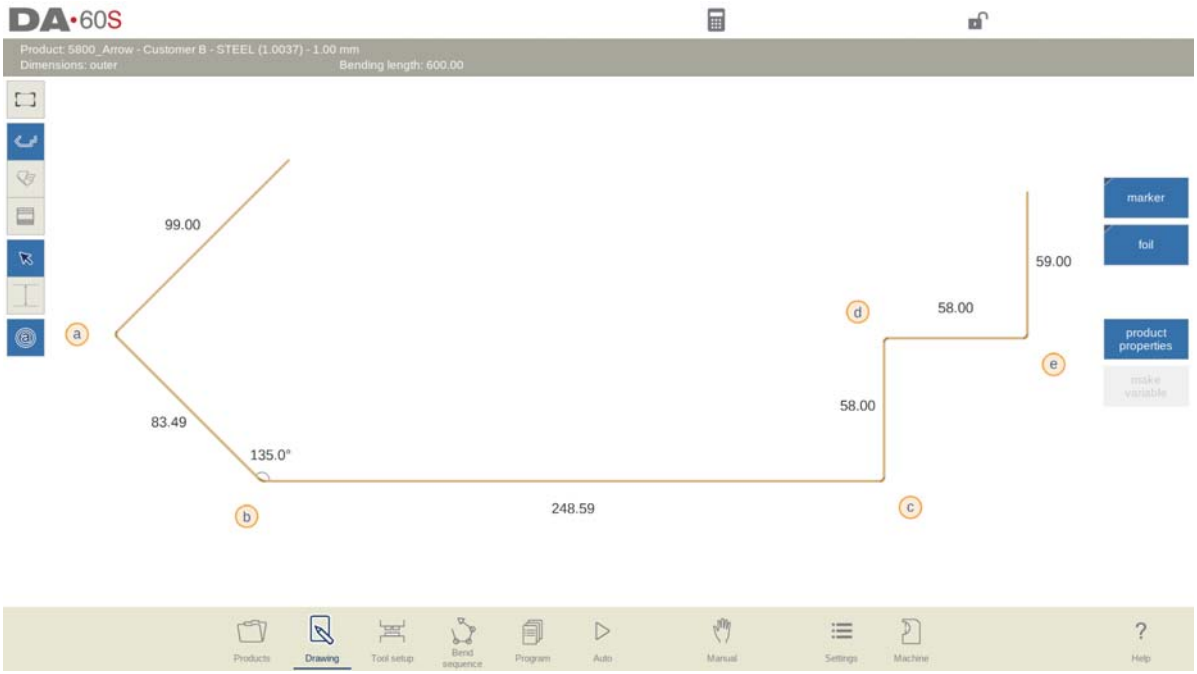
It can be helpful or even needed for the bend process, to insert a bend which finally again is taken out, enabling a certain bend sequence without collision. With the temporary bend feature a bend can be inserted which in a later bend step is unbent again to a flat side. The property to add a prebend to an angle in a product can be used for this functionality but more specific this function is also added to the insert bend function.

When tapping insert bend, the choice can be made to insert a temporary bend. This bend, including a prebend will then be processed and taken into account for the bend sequence calculation.


DA·60S
Product: 5800_Arrow - Customer B - STEEL (1.0037) - 1.00 mm
Dimensions: outer
Bending length: 600.00
bend
insert bend
temporary bend
99.00
59.00
58.00
58.00
83.49
135.0°
248.59
Products
Drawing
Tool setup
Bend sequence
Program
Auto
Manual
Settings
Machine
Help

3.5. Bend labels (DA-6xTe only)

In addition to the view modes of the control, in Bend Sequence, but also in Drawing and Auto-mode's graphical visualization, the bend label view is available.
Bend labels automatically codes al bendlines with a unique identifier in alphabetic order which helps the operator to identify the bends while e.g. manipulating the bend sequence or while drawing or producing keeping, a hold on the product and bendlines.
To switch on the bend labels simply tap the bend label view button. You will see small labels pop-up at every bendline.



3.6. Surface or bend line marker

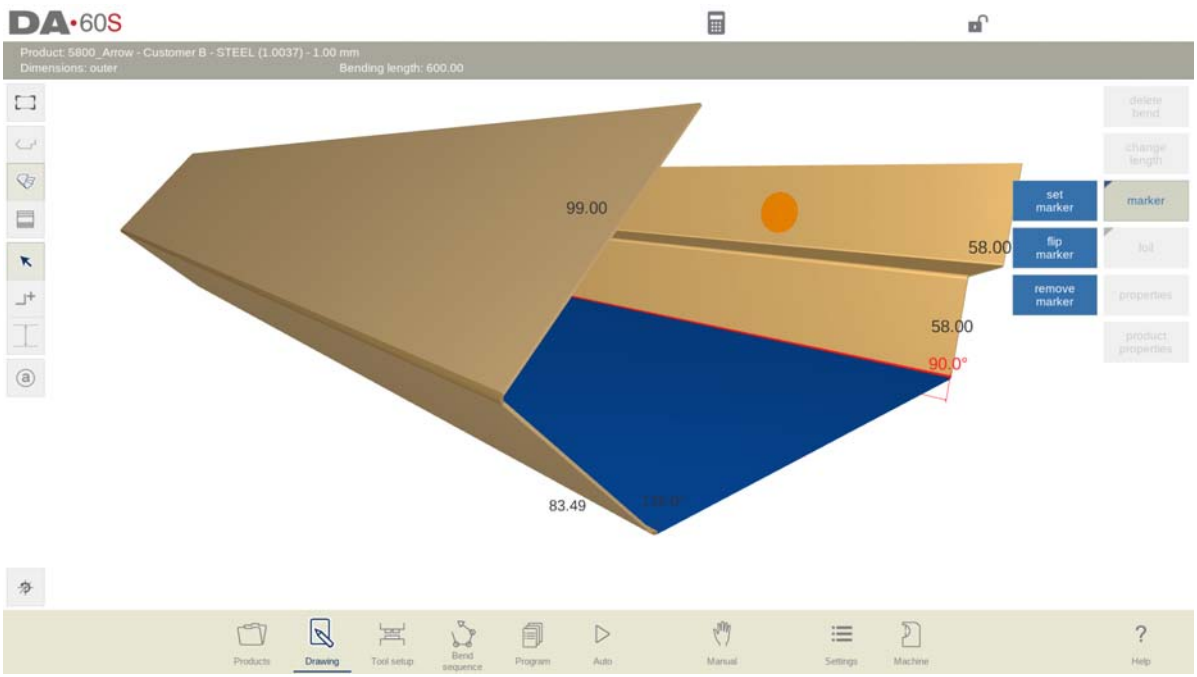
With the marker function a specific surface or bendline can be marked improving the recognition specific of sides and bendlines.

Functions

Set marker
One can select and point at a specific point in the product to place the marker. A cylindrical “label” will be placed at the point of insertion. The marker can be moved to a new position by setting the marker on a new position.
This marker will be shown in Automatic mode.

Flip marker
The top/bottom colors can be swapped if this would suit the situation better.

Remove marker
Remove the marker from the product.



3.7. Foil side indication

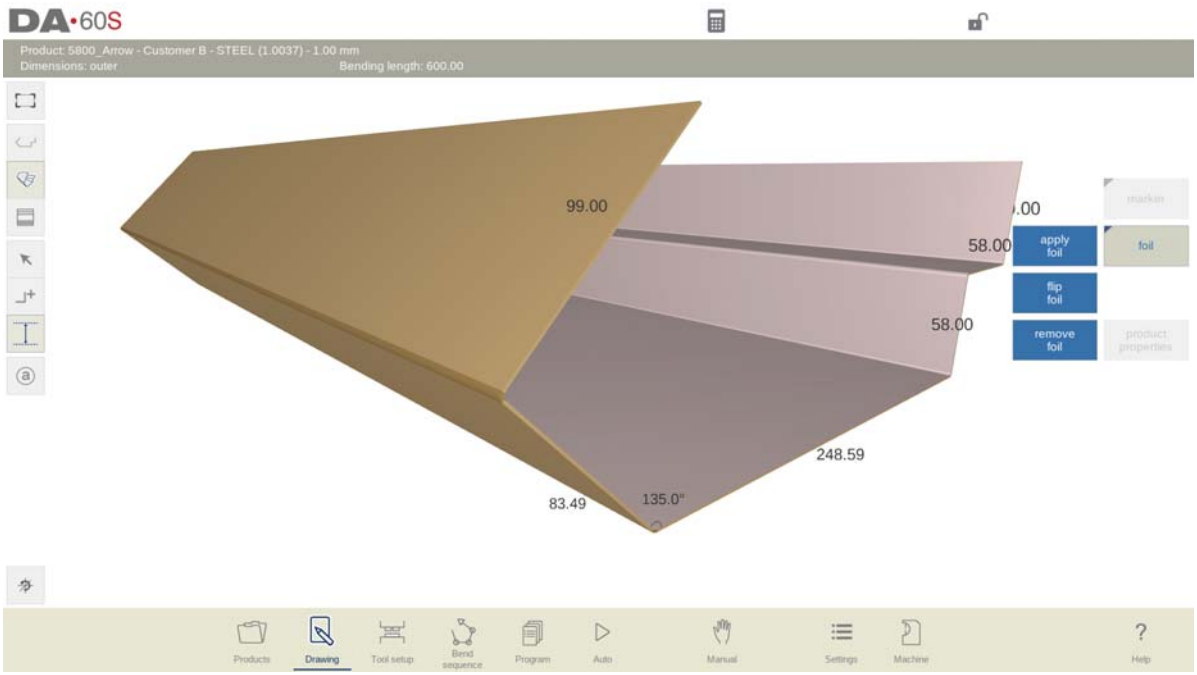
To increase the recognition and to prevent erroneous positioning of the unfolded product one can add a foil side to the product visualization. With functions ‘apply foil’ and ‘flip foil’ the desired side can be selected.

Functions

Apply foil
One can apply a foil indication to the product. One side of the products color will be clearly changed indicating the foil side.

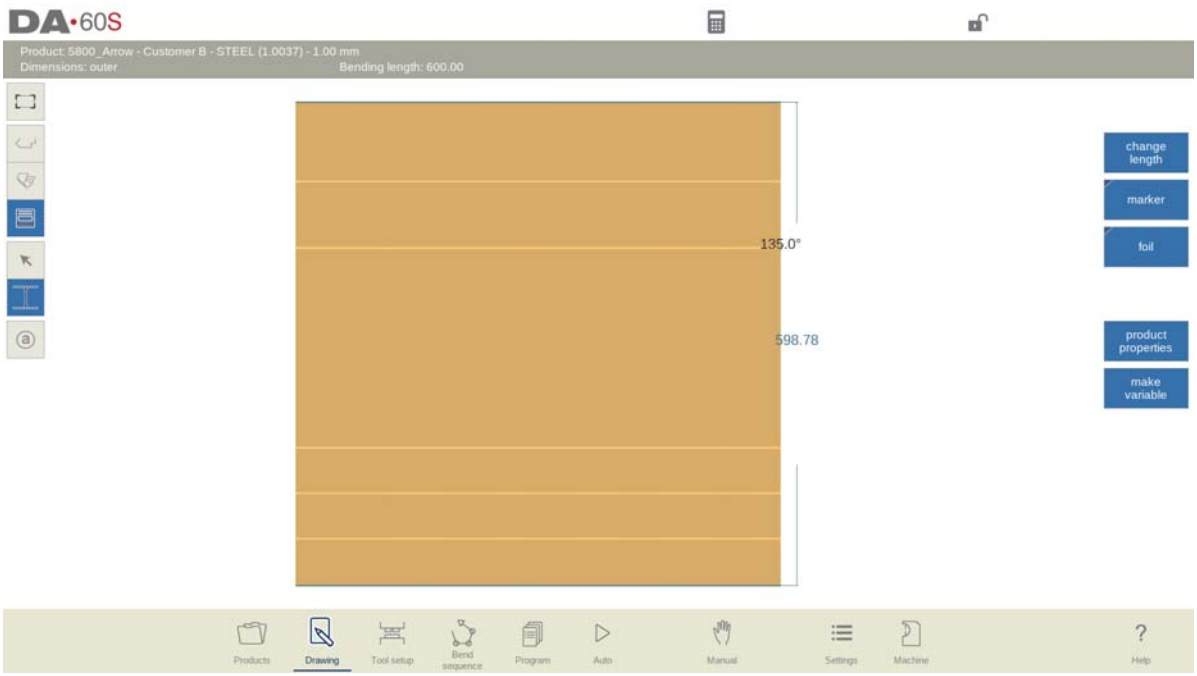
Flip foil
The top/bottom of the applied foil side can be swapped if this would suit better.

Remove foil
Remove the foil from the product.



3.8. Measurement function in graphical views

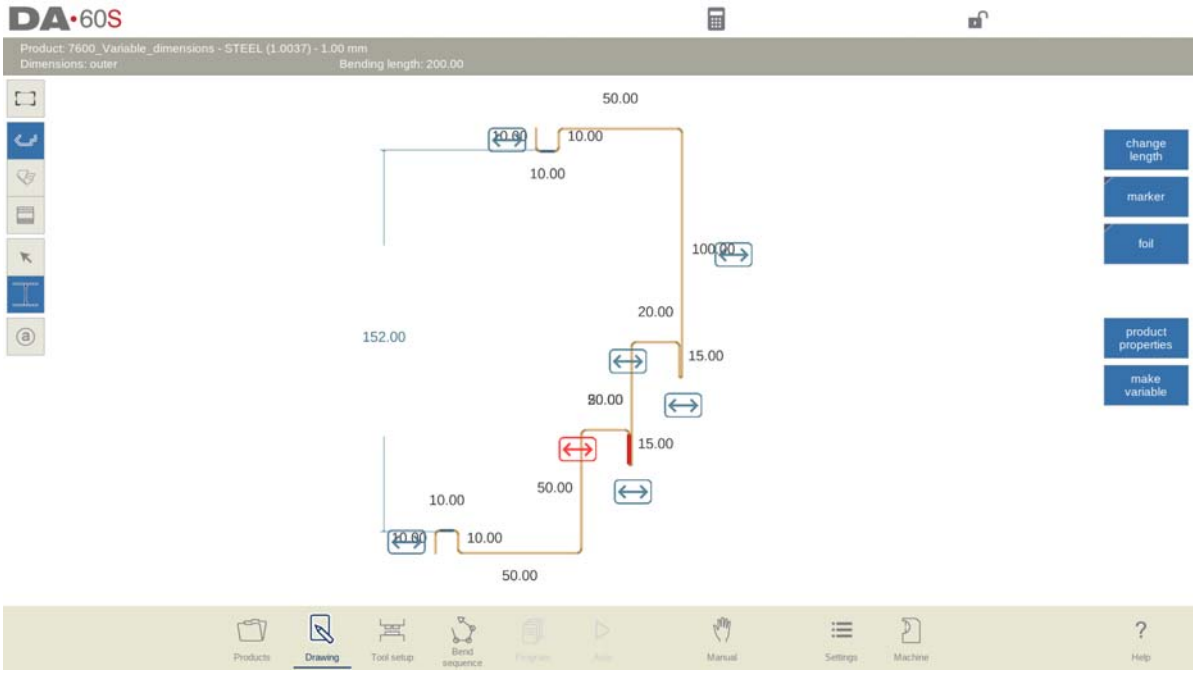
Within the graphical drawing screens the measurement function enables the measurement of developed length in various directions. By selecting opposing lines in the similar direction the resulting dimension line will be shown within the drawing.



The incorporation of the calculated bend allowance is only available after tool selection and a complete bend sequence has been carried out.

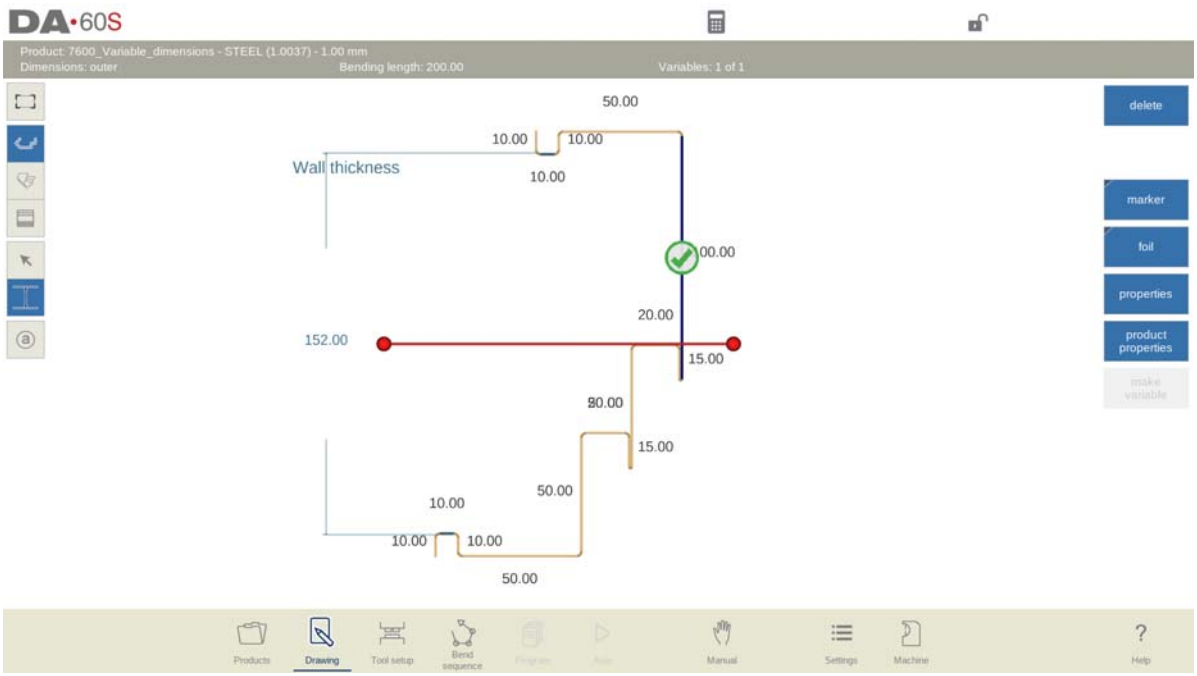
3.8.1. Change length function

The change length function in measurement mode enables the change of length for a horizontal or vertical cumulative length. When selecting this function the control will show the segment selection buttons next to all segments at what point the desired change length will be implemented.
To select either horizontal or vertical projection, the dimension can be tapped as it becomes active.


DA•60S
Product: 7600_Variable_dimensions - STEEL (1.0037) - 1.00 mm
Dimensions: outer
Bending length: 200.00
50.00
10.00
10.00
100.00
20.00
152.00
15.00
80.00
15.00
50.00
10.00
10.00
50.00
change length
marker
foil
product properties
make variable
Products
Drawing
Tool setup
Bend sequence
Manual
Settings
Machine
Help

3.9. Variable product dimensions

Programming variable product dimensions, meant for modification of existing products making a variant, is supported with the variable line functionality.

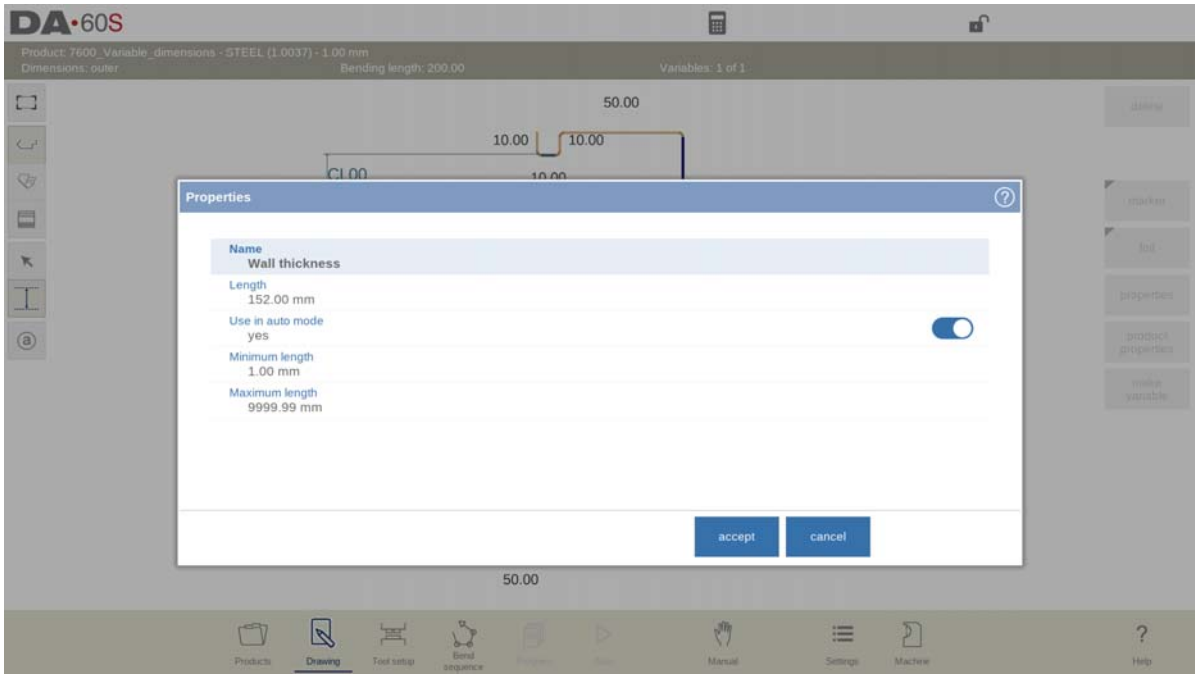


Based on a variable line in the product, one can stretch or shrink a product at the line position. The properties of the variable line allow the change of the reference side length. The offset generated from this change will effect the intersected segments from the same variable line. An advanced function of the variable line allows the change of length to be entered at production time in Automatic mode. The variable(s), based on the name of the variable line(s), can be found in Automatic mode and will implement the change of length in all sides intersected by the variable line(s).
After closing the Variables dialog in Automatic mode, the product will be verified, possible collisions will be shown and the program is ready to be started.

Variable lines can be inserted by tapping Make variable, based on a selected side length, or based on a measured product dimension. The inserted variable line can be moved along the segment line, pointing the position at which the strech or shrink is applied. The length of the variable line can be dragged to include or exclude line segments. Colors, symbols and snap lines will be shown while dragging indicating whether lines are included, or whether lines are able to be stretched.
When deleting the reference side, the variable line will become inactive. This is indicated in Automatic mode by a red color of the inactive variable.

Variable lines have to intersect segments perpendicularly and their reference side needs to remain in the product. For 2D products, variable lines can be added in 2D view and Unfolded view. For 3D products, variable lines can be added in Unfolded view.

3.9.1. Variable line properties



After placing a variable line in the product, select the variable line and tap Properties to edit the variable line properties.

Name
The name of the variable line.

Length
The length of the reference side.

Use in auto mode
Available in Automatic mode: Yes or No.

Minimum length
The minimum length value, for input in Automatic mode.

Maximum length
The maximum length value, for input in Automatic mode.

3.10. 3D drawing

3.10.1. Start 3D

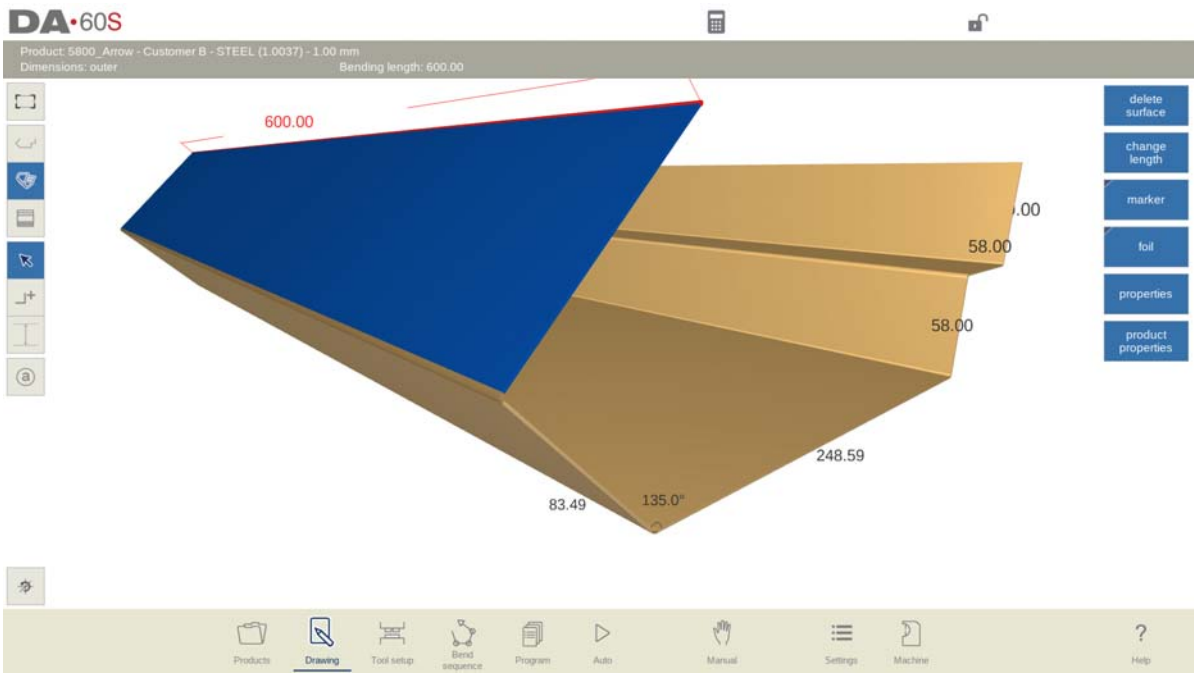
When you would like to create a 3D drawing, the 2D profile view can be switched to a 3D drawing. The command icons on the opposite side of the command buttons can be used to switch views to 2D, 3D and 3D unfolded.

2D view mode, showing the product as a 2D profile

3D view mode, showing the product in 3D

3D unfolded mode, showing the unfolded product as a flat sheet

The product, started as a 2D profile, will appear in its 3D view, the surface depth is derived from the initially programmed length.



The active surface is highlighted in a different color, the cursor marking the selected bend, is also highlighted in red.
From this point forward, the drawing can be edited as necessary. Surfaces can be added and deleted, side and angle properties can be changes and for specific surface properties extra functions are shown in the drawing screen.
The main drawing navigation is divided in a selection mode and a mode in which surfaces can be added. In the Selection mode, to be selected tapping the arrow button within the drawing screen, the bend lines and surfaces can be selected. Specific functions and properties can be

changed. In the Add Surface mode, to be selected tapping the button with the product symbol and + sign, the product can be given extra surfaces on open sides.

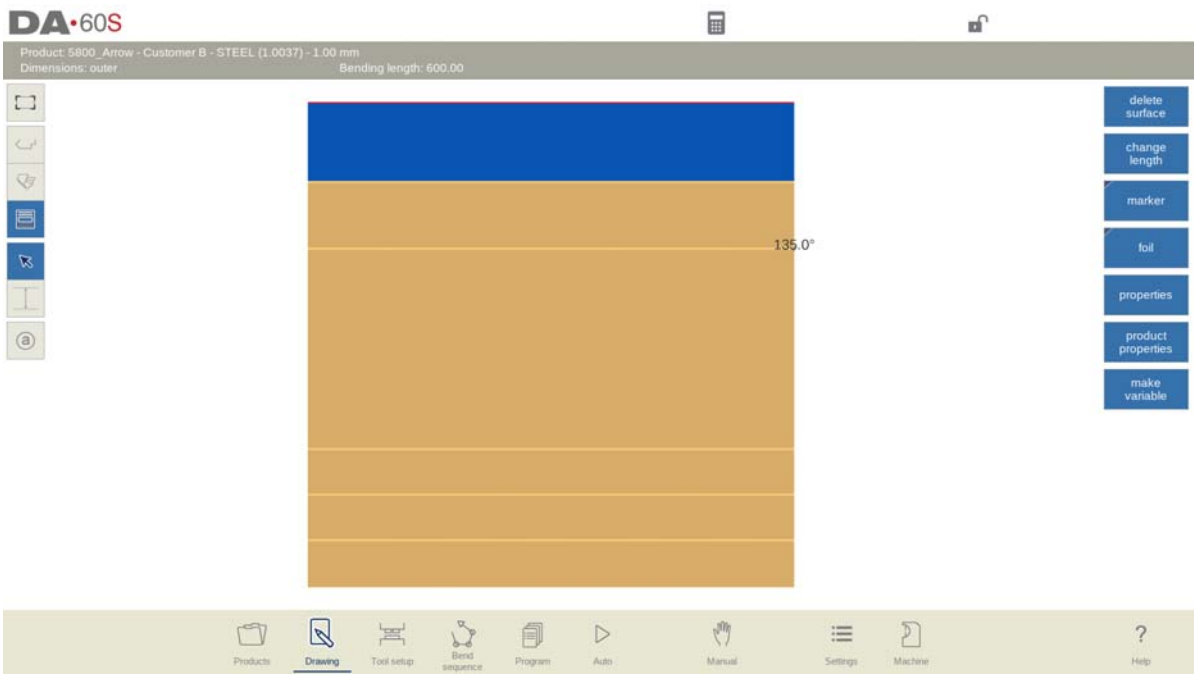
Selection mode, bendlines and sides can be selected and modified.

Add Surface mode, extra surfaces can be added on open sides.

Before adding a new surface the cursor must be put on a free side of an existing surface.
A new surface is added by tapping on the desired position next to the bend line cursor. The new surface will be added and stretched to that point. Also entering the angle with the existing surface and the height of the new surface can be used as an alternative way to program.
A new surface always has a rectangular shape. It can be changed afterwards as necessary.
While tapping to sketch the extension if the product, the angle of the surface with the current surface is snapped to 45 degrees or a multiple of 45. When the cursor is on the end of the product (a free line) also the cursor can be used to manipulate the side length and angle. This is snapped to multiples of 5 degrees. Exact angles and side lengths can of course be given in by typing.

3.10.2. 3D unfolded product

When the view '3D unfolded' has been chosen, the product is shown as a flat sheet, prior to bending.

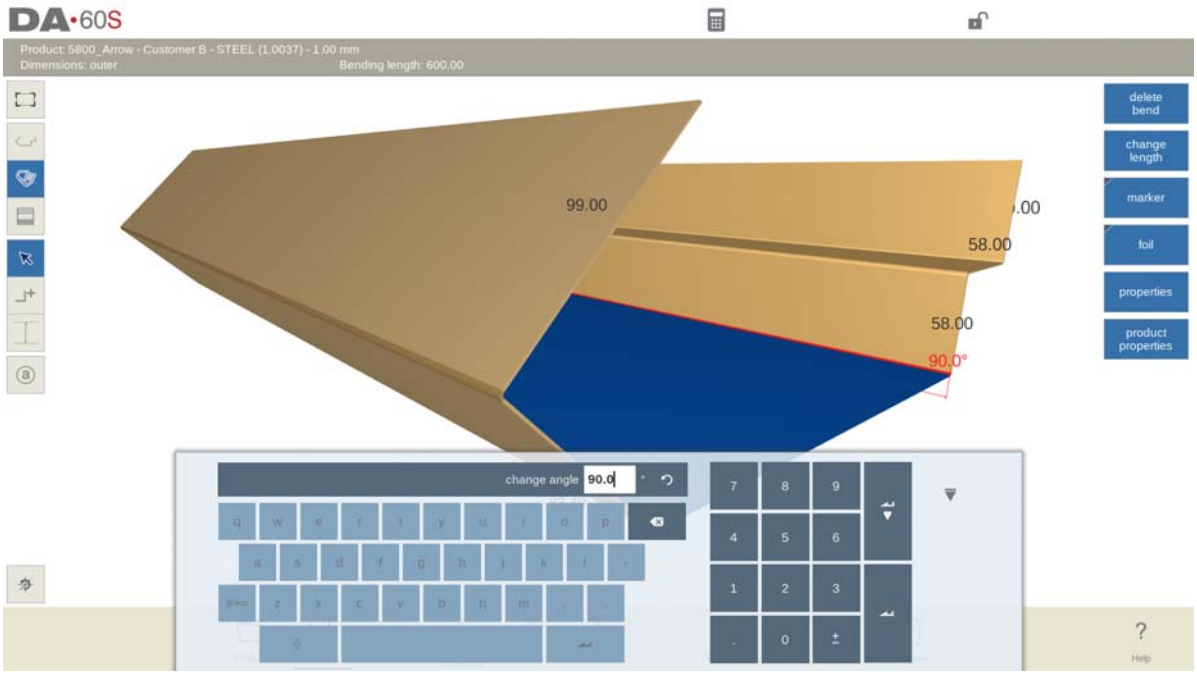


Tap View to return to select another view mode.

3.10.3. Edit or rotate the product

When the cursor is placed on a free edge of the product, it is possible to add a surface to this edge.

To do so, select a free edge by tapping, tap the Add Surface mode and point where the next surface has to end. It is also possible, with an opened keyboard, to add the desired angle and give in the desired surface length.
When the cursor is placed between surfaces, it is not possible to add surfaces. The Add Surface mode will not be selected after tapping



Navigation functions

Fit to view function, to scale the product to fit the view.

Reset view function, to reset the view (i.e. rotation and zoom) to default.

These functions can be used during all operations.

3.11. Edit a 3D drawing

3.11.1. Introduction

A product consists of multiple surfaces, starting with the base surface.
The 3D drawing function offers several functions to draw surfaces and change their shapes. Shapes with four corners such as a square, a rectangle, a trapezium, a parallelogram or any random quadrangle are possible.
With the drawing tool you can execute the following manipulations:

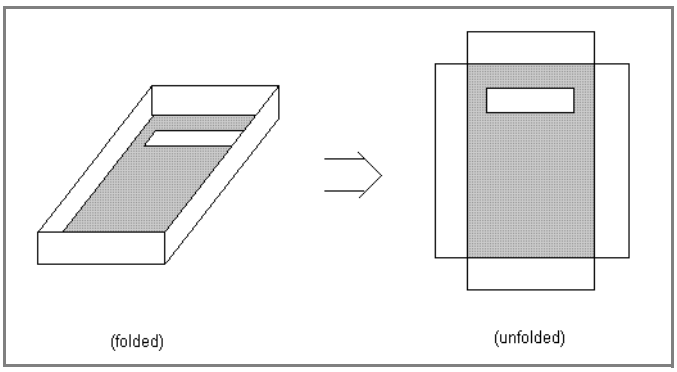
1. Add surface
2. Delete surface
3. Change side length
4. Change angle between two connected surfaces

3.11.2. Add surface

Adding a new surface is only possible at a free side of an existing surface. At each side one other surface can be connected. Adding a surface is done by selecting Add Surface mode and tapping the desired surface endpoint or entering an angle and a length in this order. The result will be a new, rectangular surface. The value of the angle entered, or when afterwards correcting a exact angle value, must be between plus or minus 180 degrees.

Beside the outline of a surface also the contour of a hole is considered as an outline with the same possibilities. A hole must be entirely in one surface and can have any shape.

A surface or total product can contain more than one hole. Holes cannot be added with the drawing tool. The product drawing of the following figure has 16 outline sides (12 surface outline sides and 4 hole outline sides) which can be selected.

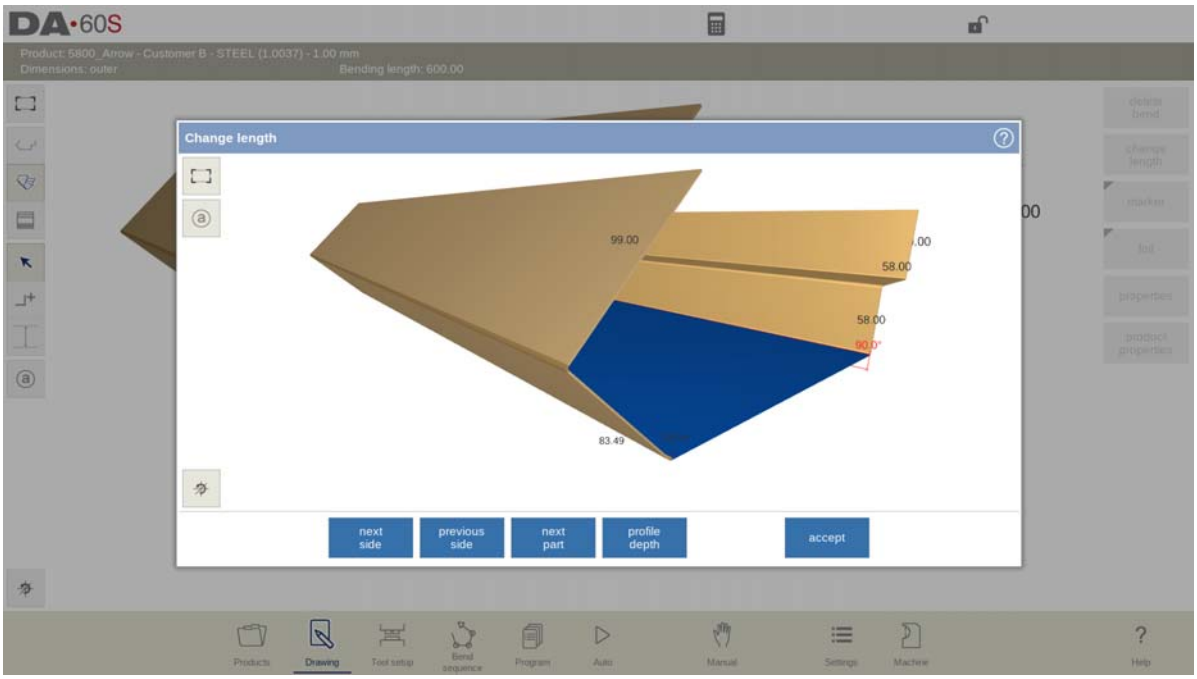


3.11.3. Delete surface

Deleting a surface is only possible when the surface has only one other connecting surface and the surface is not the base surface.

3.11.4. Changing the length of a side

Before changing the length of a side you have to select the side to be changed. Just tap the desired side to select it. The actual length value is displayed near to the product side and also in the keyboard input row.

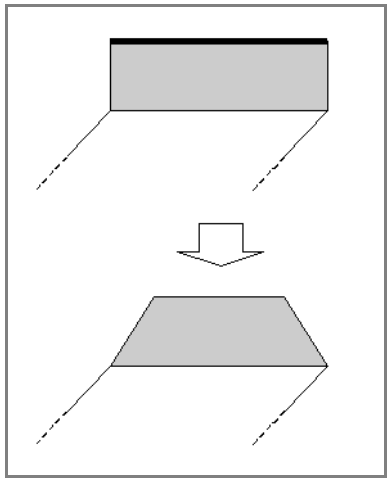


Then you can use Change Length to give in the new length. A pop-up window appears in which the new length can be programmed. In this window, the function Next Part can be used to define how to change the length of the side: from both the ends or from only one of the ends. Enter a negative value to shorten the length with the programmed value or a positive value as the new length.

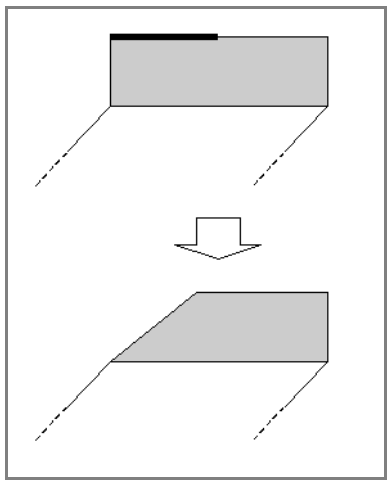
Example:

Existing length is 100

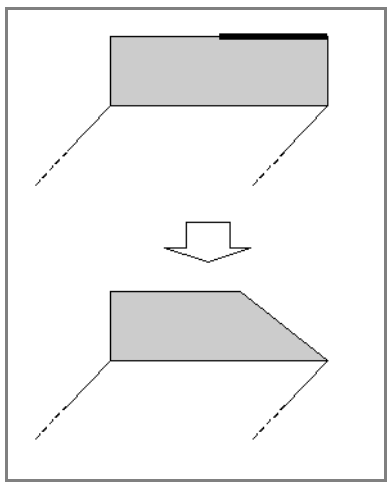
1) The side must be changed evenly: make sure the entire side is highlighted.
Enter value 60.
Result: the side is shrunk on both sides.



2) The side must be changed on the left part. Tap Next Part until the side is highlighted as shown.
Enter value 60.
Result: the side is shrinkage on the left side.



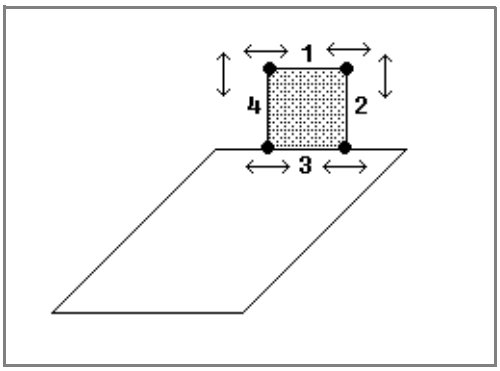
3) The side must be changed on the right part. Tap Next Part until the side is highlighted as shown.
Enter value 60.
Result: the side is shrunk on the right side.



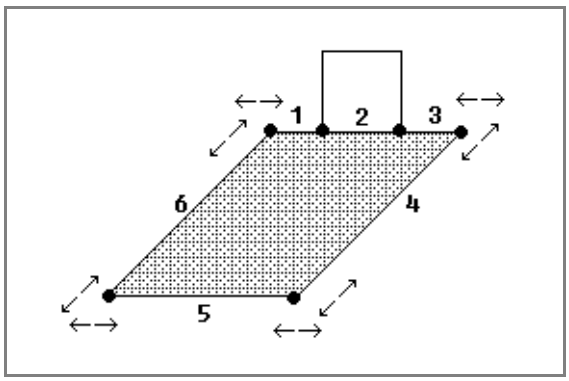
By entering a negative value of -40 you get the same results.
The new side length is the same in each case. However, the resulting shapes are different because the extension or shortening starts at different ends.

Changing the length of a side is allowed in case this change has only effect on the shape of the active surface.
It is allowed to change the length of a bend line, as long as other surfaces and bends are not affected by this change.

Changing the length of a side is not allowed in case both ends belong to the bend line and both ends have a side that is in line with the direction of the bend line. That is why in the below figure both ends of side number 3 can be moved while this is not possible for side number 2 of figure further below.

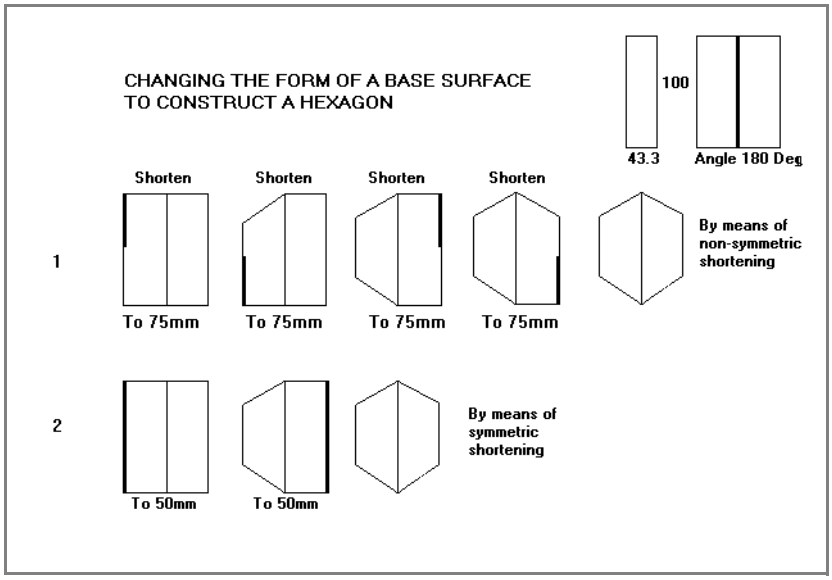


Side 1: this side can be extended or shrunk in both directions (both ends are free).
Side 2: only the free end that is not connected to a bend line can be changed.
Side 3: this side can be extended in both directions.
Side 4: only the free end that is not connected to a bend line can be changed.

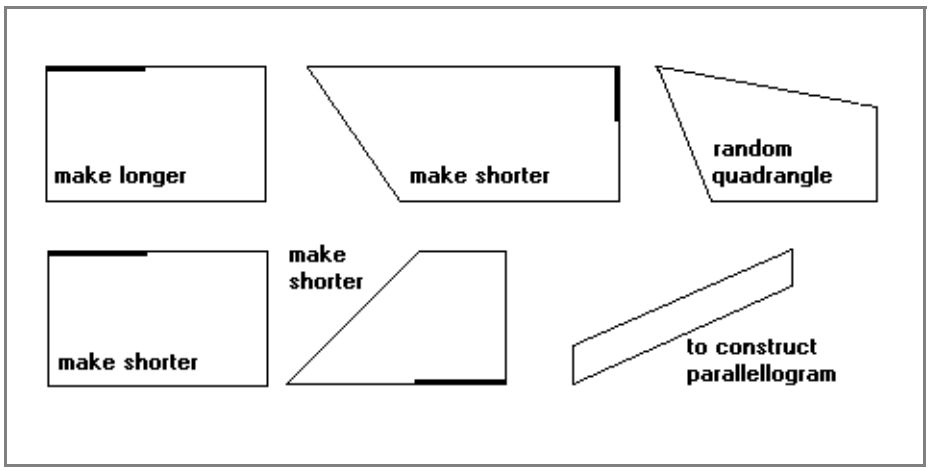


Side 1: only the free end that is not connected to a bend line can be changed.
Side 2: this side cannot be changed (there are sides connected to both ends which are in line with the side itself).
Side 3: only the free end that is not connected to a bendline can be changed.
Side 4: this side can be extended or shrunk in both directions (both ends are free).
Side 5: this side can be extended or shrunk in both directions (both ends are free).
Side 6: this side can be extended or shrunk in both directions (both ends are free).

The next picture explains two ways to create a hexagon. Both options start with a base of two rectangles with an angle of 180 degrees.

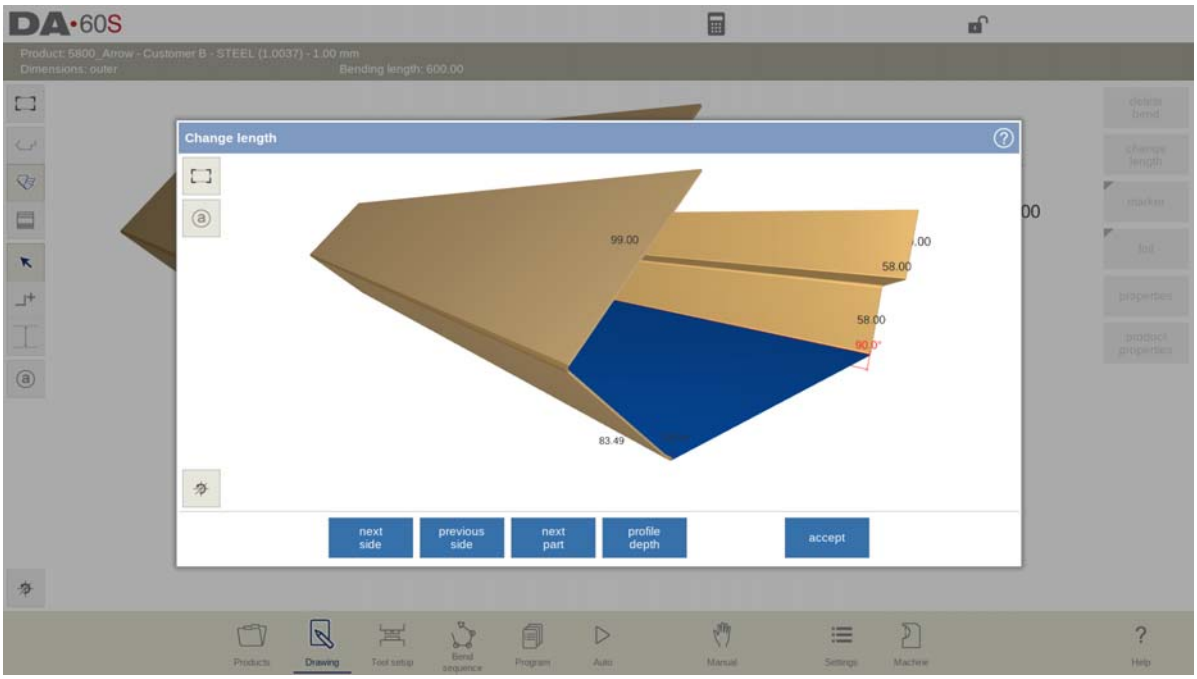


Some other examples of special shapes by making the sides shorter or longer.



Make sure the cursor points to the corner for the new length.

When a 3D product still has bends in one direction, the basic size of the product can still be changed easily. This is typically the case shortly after the drawing is switched from a 2D profile to a 3D drawing. Within the dialog ’change length’ the function keys ’profile depth’ and ’surface length’ can be used to change the product size.



3.11.5. Changing the angle between two surfaces.

You can change the angle between surfaces into any other angle between -180° and 180° without constraints. The drawing tool always displays the product at relative scale. Collision of surfaces due to another angle is also shown at relative scale.

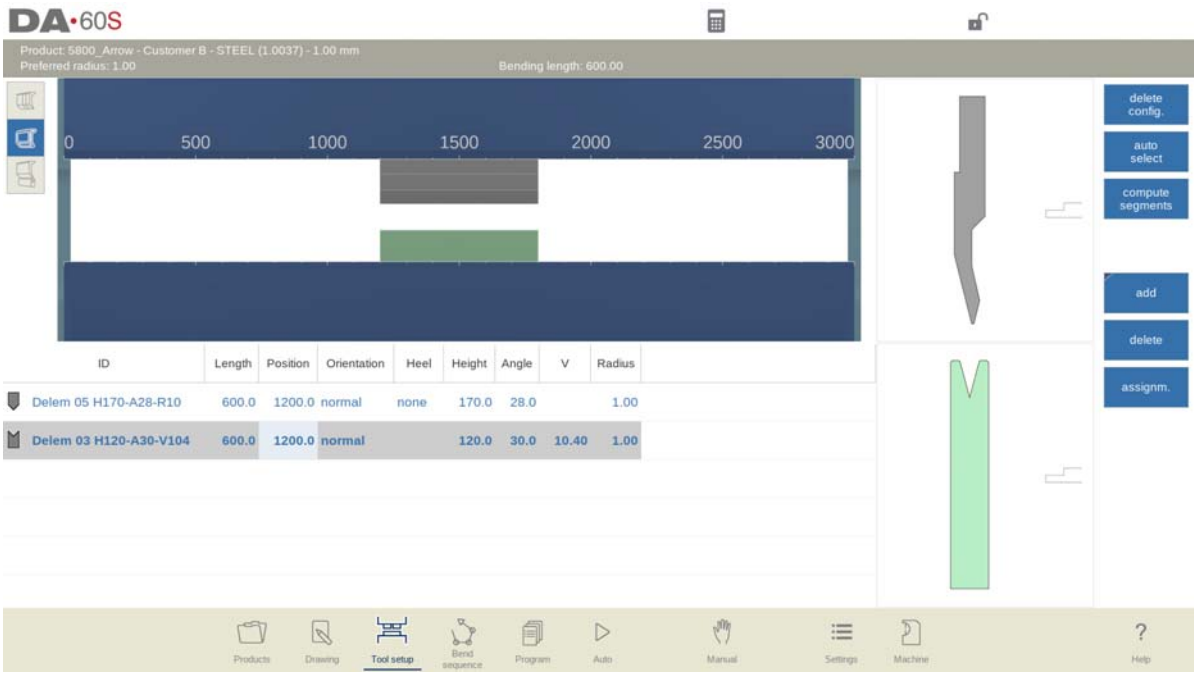
4. Tool configuration

4.1. Introduction

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

4.2. Standard procedure

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



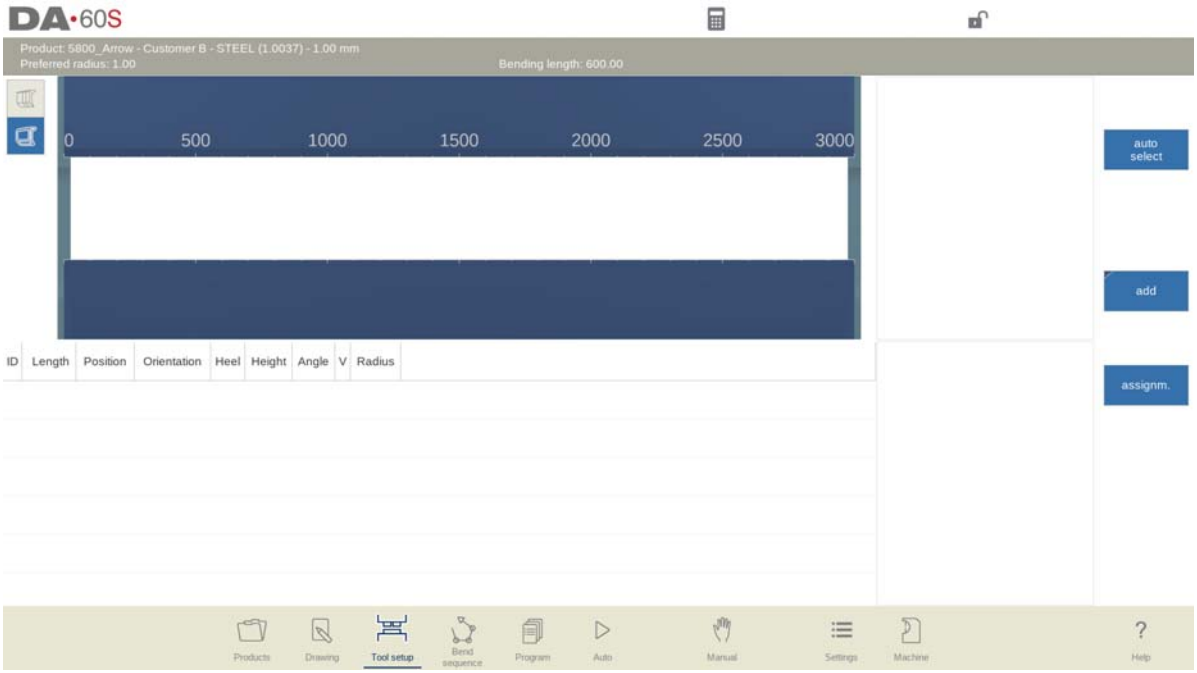
In the front view, the following machine elements are shown, from top to bottom:

- Press beam
- Adapter for punch (if a punch adapter has been programmed)
- Punch
- Die
- Adapter for die (if a die adapter has been programmed)
- Table

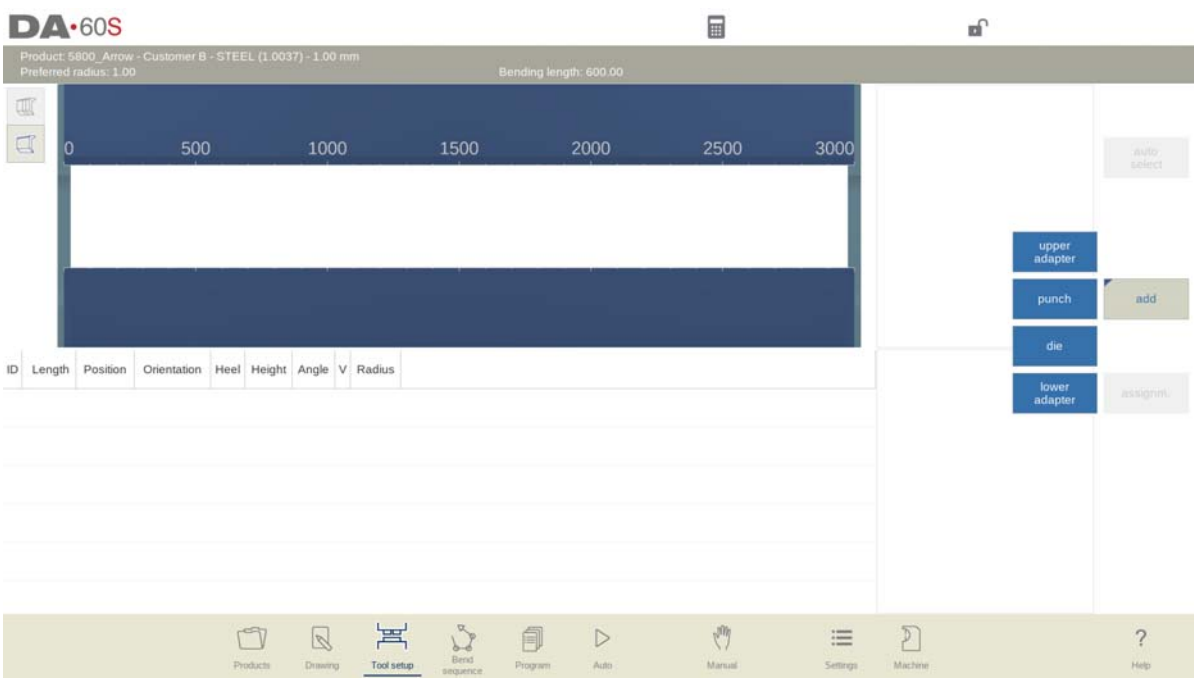
The machine parts have already been preselected in the Machine mode. Normally these parts will not change. Whether an adapter can be programmed depends on the adapter parameters in the same Machine mode.

4.3. Tool selection

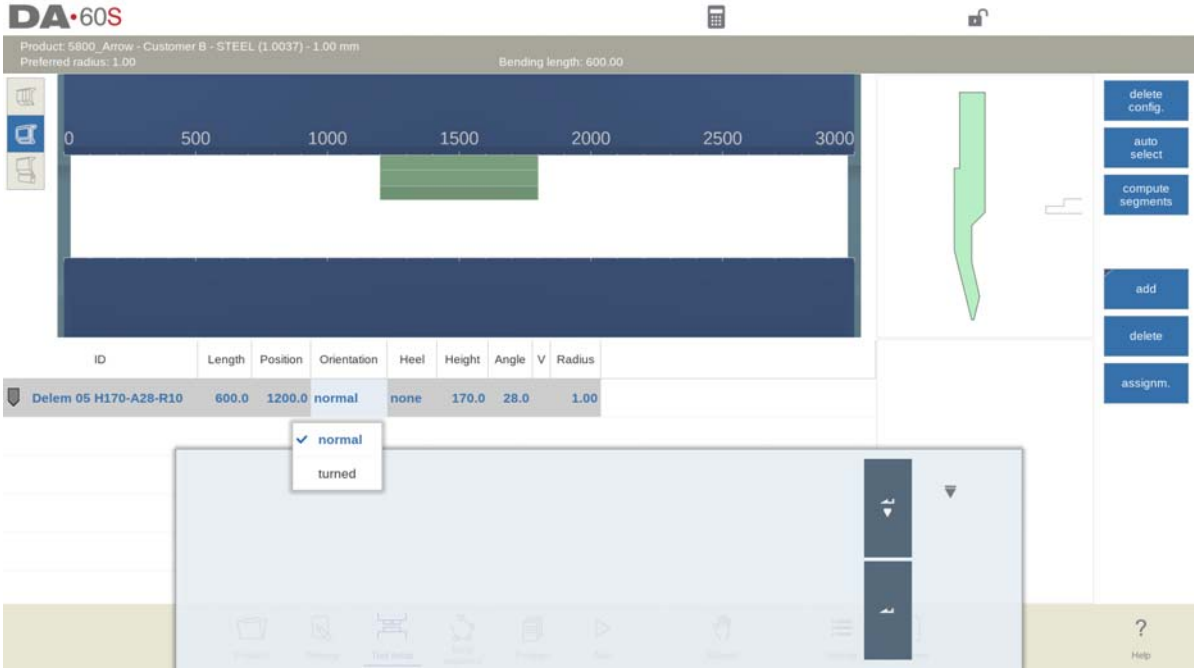
When starting a new tool configuration, the machine opening is empty.



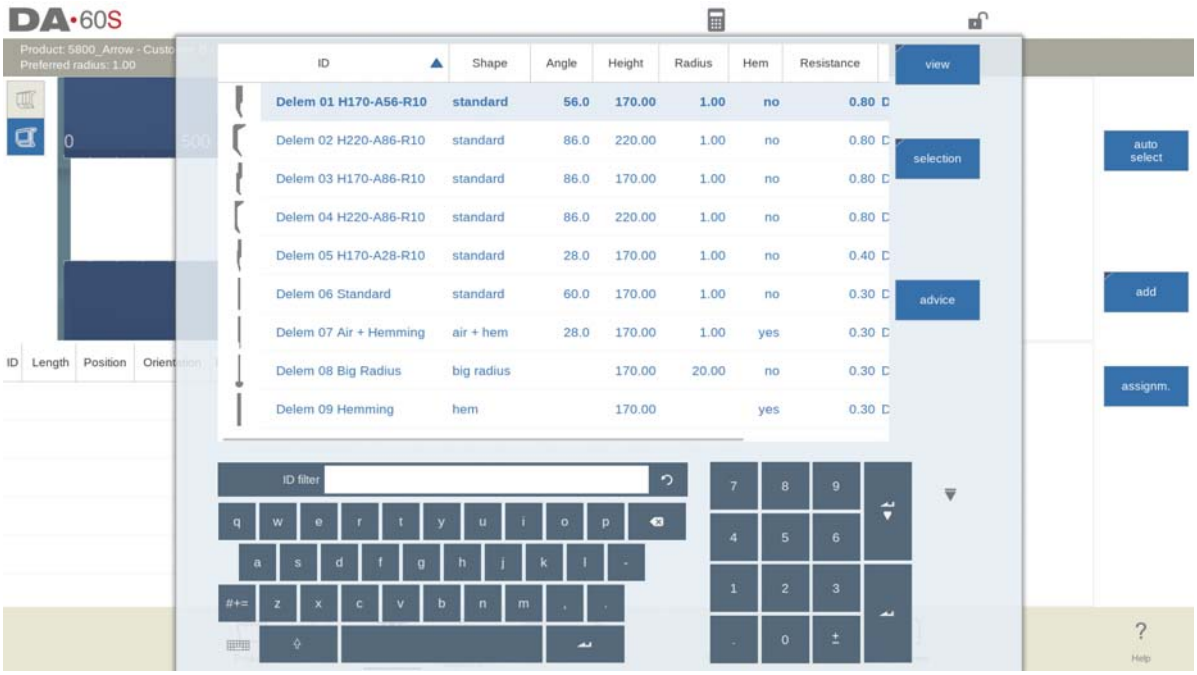
Select Add to add a tool to the configuration; punch, die or adapter (if enabled).



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



After a tool is placed, the tool ID can be changed by selecting the Punch ID in the screen and tapping on the List view.



If only a part of the tool ID is typed, the control automatically offers a list of tools with the typed characters. This is a filtering function.

By tapping Advice from this screen a limited selection from the available tools will be made and displayed. The criteria for the selection are:

- Product radius; the selected tool must result in a product radius close to the preferred radius. The resulting radius must be in the range of the preferred radius +/- 50%.

- The necessary bending force should not exceed the resistance of the tools.
- The angle of the tools should be equal to or smaller than the required product angle.
- The bend method should match. For example, hemming tools are selected when the product contains hem bends.

The full list can be displayed by tapping Show All.

Auto select

From the Tool Configuration screen the function Auto select is offered. This function helps the user to select a set of tools (punch and die) best fitting to the programmed bend given the maximum bendline size for this product. The automatic selection from the tool library will be done finding the best possible fitting tools for one bend (resulting in a most optimal product radius). It will not generate a full tool setup since therefore the bend sequence is a most important constraint. The Auto select function will replace the existing tool setup.

Multi-V, die feature preselect

In case of a Multi-V die, i.e. a die with more than one V- or U-opening, a specific V- or U-opening of the Multi-V die can be selected or the Multi-V die (with V = *) can be selected. If a specific V- or U-opening is selected, this opening is used by the bend sequence calculation; if the Multi-V die (with V = *) is selected, the control will use the opening which will result in a product radius as close as possible to the programmed product radius.

ID	Shape	V	Angle	Heigh
Multi-V	multi V	*	*	55.
Multi-V	standard	10.00	86.0	55.
Multi-V	standard	12.00	60.0	55.

If during the bend sequence calculation another V- or U-opening is selected, this new opening is used by the control for the remaining bends (which are not in the bend sequence yet).

Vario-V, selecting and setting

In case of a Vario-V die, i.e. a die with a variable V- or U-opening (only available if a Vario-V system is present), the selection of the die is equal to any other die. When the Vario-V is selected the V-value is not programmed yet. From here one can choose to continue and make a bend sequence. The control will use the most suitable V-value, taking the available Vario-V positions into account.

ID	Shape	V	Angle	Heigh
Vario-V	vario V	___.__	86.0	55.

In case a desired V-value is programmed, the control will use this programmed value in the bend sequence calculation.

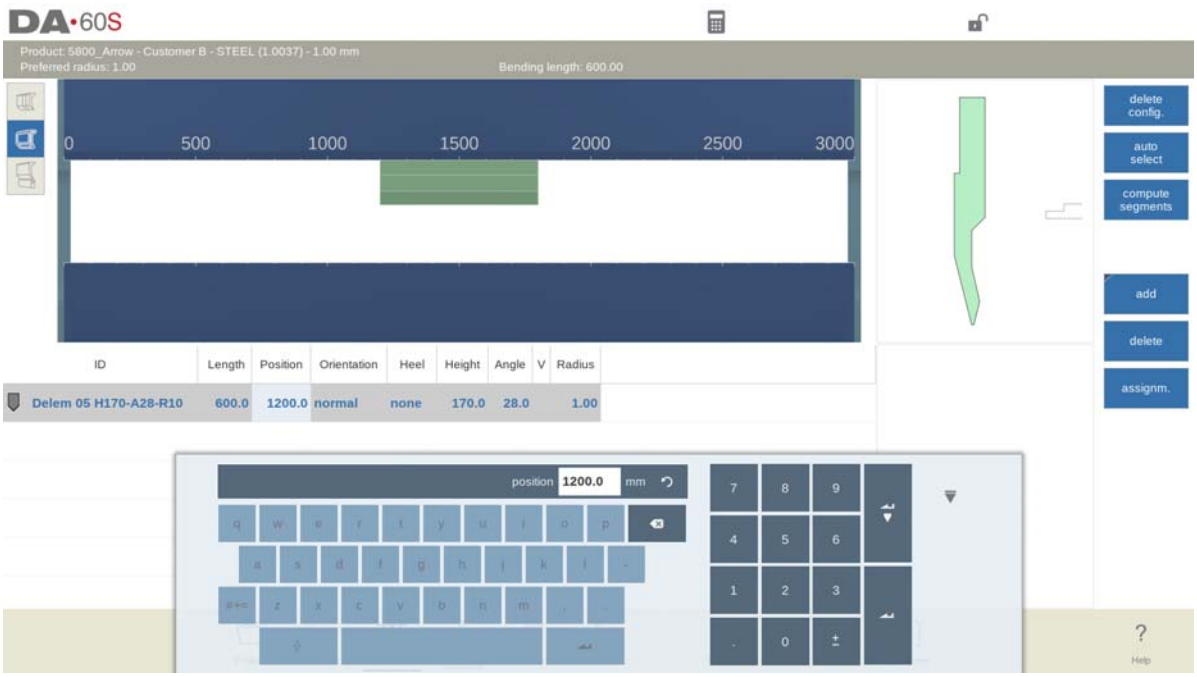
When the Vario-V die system is using discrete positions, only values from the available positions will be possible. When programming differently, the nearest available value will be used.

In the Bend sequence mode the selected V-opening (Multi-V) or V-value (Vario-V) can be modified. Using Modify, Shift Die, the selected V-opening or V-value is shown and can be modified by tapping on it. In Program mode the Die positioning function offers similar functionality.

Positioning or repositioning

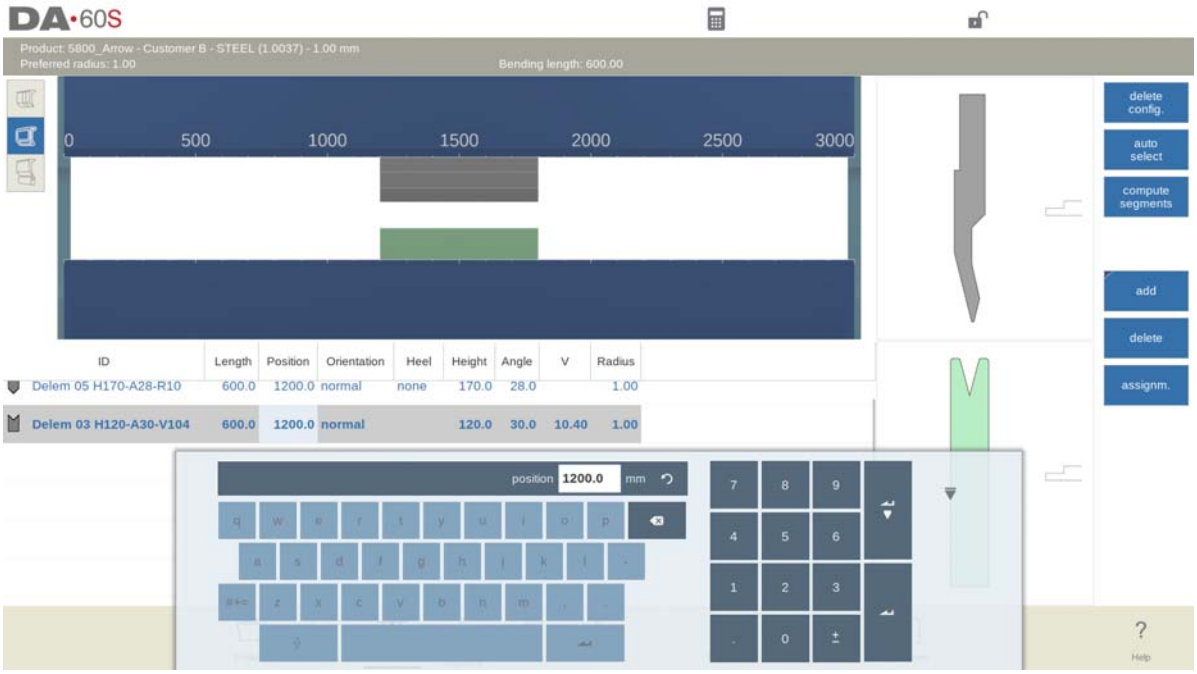
To change length and position of a tool, move the cursor to the appropriate field, type a new value and press ENTER. When the length and position are changed, the tool is ready.
Tools can be selected by tapping them, their position can also be changed by dragging them to the desired position.

The repositioning of tools by dragging is enhanced with a feature to enable accurate positioning. When, during dragging, the finger is moved downward, the positioning speed is decreased. The further the finger is moved downward, the slower and more accurate the positioning can be done.



When dragging tools, or tool stations, the snapping function will assist in aligning the upper tool with the lower tool at left or right, or the tools or tool stations with the middle of the machine. At the time tools are within the snapping zone, a red line will appear indicating that release will help positioning towards the indicated edge or position.

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



The tool set-up can be modified with the available functions or by changing a value in the list view. Punches and dies can be added or deleted, the existing tools can be moved to a different position, their length can be modified, the orientation can be changed (i.e. the tool or adapter can be turned around) and heel types can be changed.

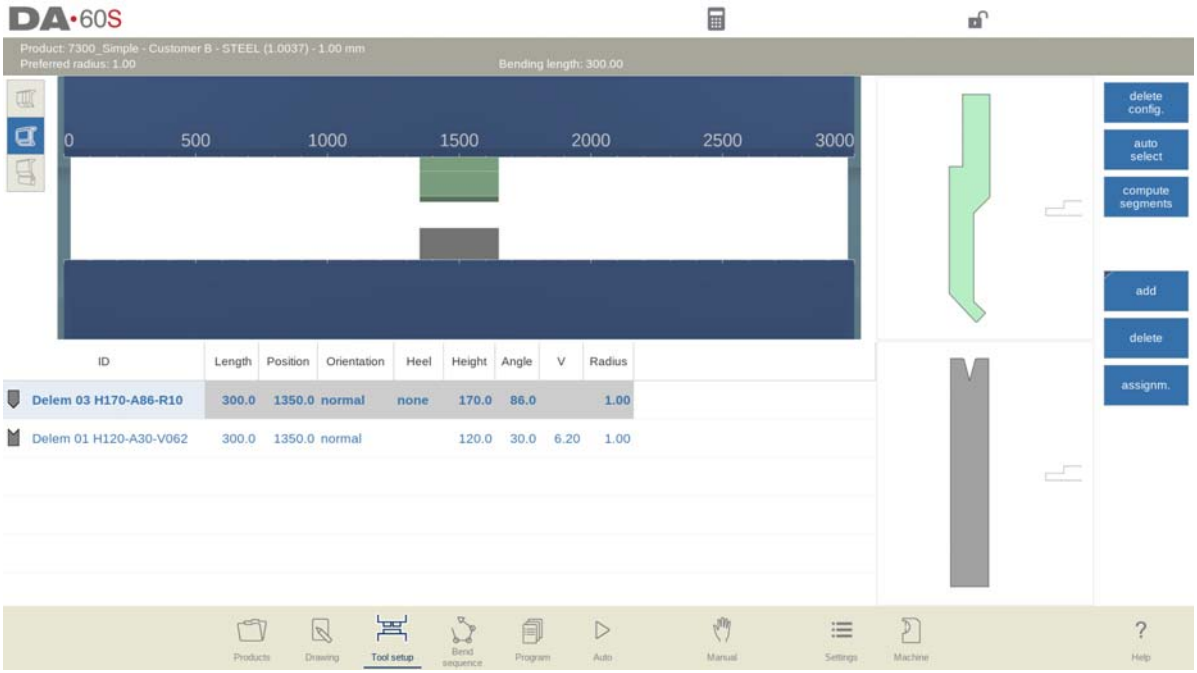
Delete config.
The existing configuration is deleted and a new tool configuration can be started.

Add
Add a new tool to the tool configuration. When pressed, some new functions are offered to choose the tool: upper adapter (if enabled), punch, die or lower adapter (if enabled).

Delete
Delete the currently selected tool.

4.4. Tool segmentation

When using segmented tools, from which desired sized tools can be composed, the control can support this and can help to generate the appropriate segmentation.



In below paragraph the functionality for segmentation is explained, including the use of the three views on the tool setup. Next to the Tool Setup screen the ability to have segmentation features available is depending on the programmed segments for each tool. This programming can be done in Machine mode under the Punches and Dies libraries. More on programming of segments in the Tool library can be found at the end of this paragraph.

From within the Tool Setup screen there are three available view modes. With the selection buttons on the left side of the machine front view following views can be chosen:

Segmentation view: showing segmentation details of tools and enabling handling of individual segments

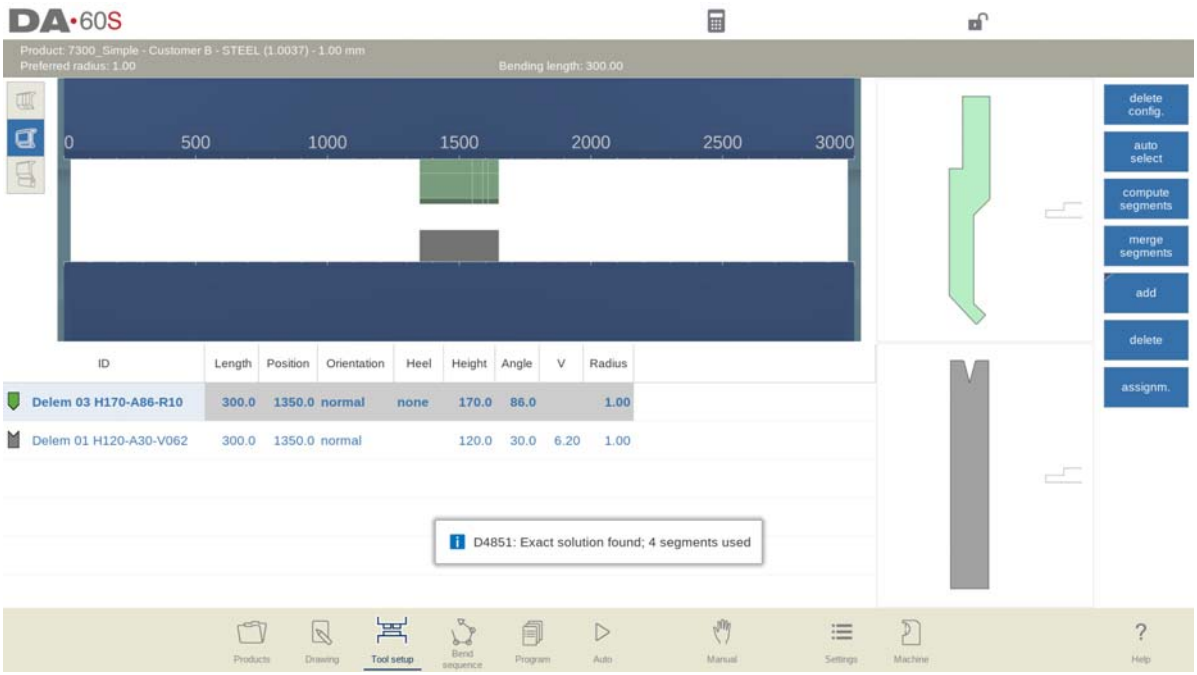
Tool setup view: showing the tools as required for production and enabling setup of desired tools

Tool station view: enabling view, selection and handling of tool stations

4.5. Segmentation of individual tools

After the process of setting up the desired tools for the products to be made, the Bend Sequence mode can calculate the most efficient bend sequence.
Upon desire the tools can be segmented, helping in the selection of the segments creating the correct tool length.

The tool segmentation function automatically calculates the required segmentation and uses the assignments “maximum inter tool distance” and upon choice the “tool length tolerance” for finding the best solution.



4.5.1. Tool view

For the segmentation of tools one can press the tool segmentation function in Tool view.
The system will, based on the programmed segment lengths and number of available segments, calculate the segmentation for that specific tool. This, taking all stations using the same tool into account (tool segments must have been programmed in the specific tools).

After starting the function, the found results are shown and the optimization process results can be followed. When an exact match has been found the indicator will turn green (later the color of the tool symbol is colored similarly).
When a non-exact but valid length is found, taking the assignments into account, the indicator will turn yellow. This can mean that either the inter tool distance or the tool length tolerance has been used. Also it can mean that, given the fact that it is a 2D product, the tool is longer than desired. The results are shown in an information message shown.
When the result of the automatic calculation is that there is no valid segmentation possible, the indicator will turn red. No segmentation will be applied.

The calculation proces can be interrupted with Cancel, or Stop accepting the current achieved optimization.

To merge a segmented tool to a non-segmented tool, the Merge segments button can be used. When changing the properties (e.g. length) of a segmented tool, it will automatically merge into a non-segmented tool.

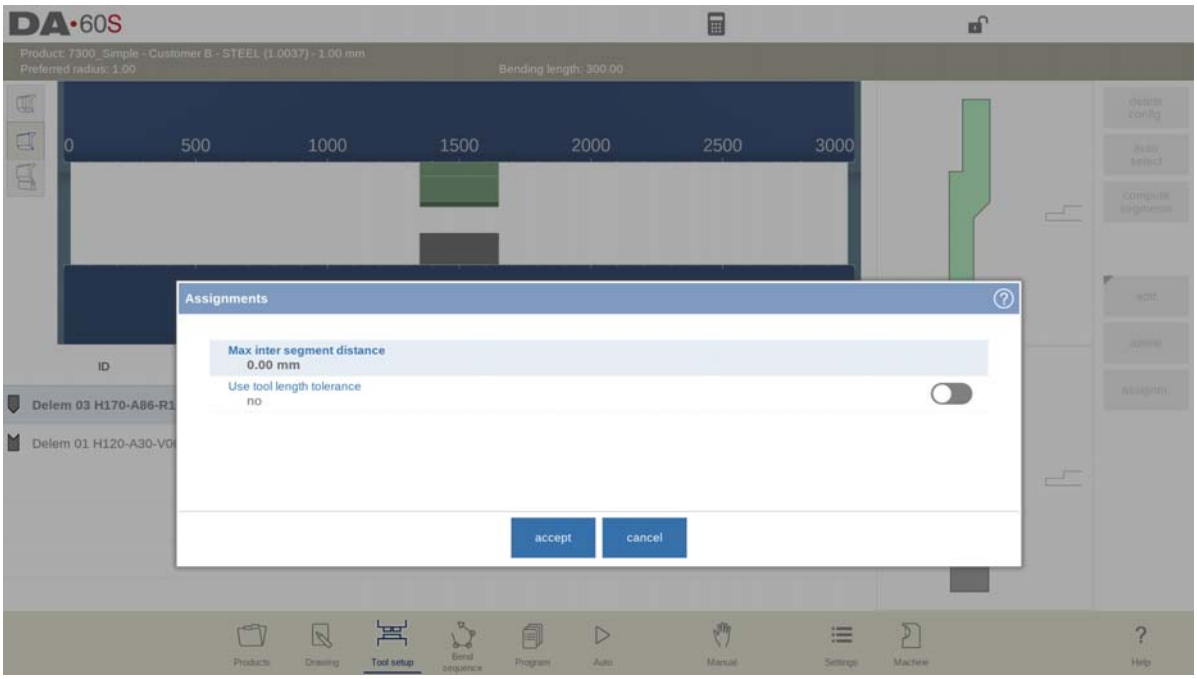
Assignments

The assignments used during the segmentation calculation can be found by tapping the Assignments button.

The following assignments are available:

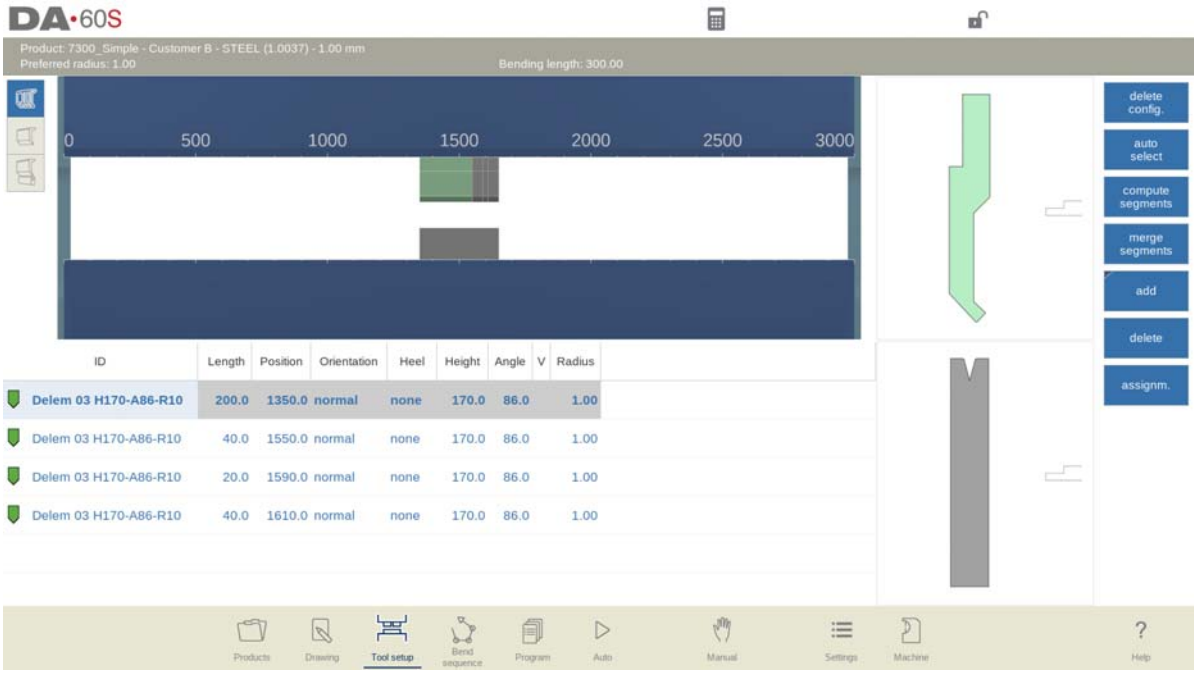
Max inter segment distance
The distance allowed in between segments.

Use tool length tolerance
To enable this tolerance for the allowed segmentation deviation.



4.5.2. Segmentation view

When switching to Segmentation view, the segments of the tools will be shown, both in the graphical visualization and in the list below.
Only the segments from the selected tool are shown. The individual segments can be moved and changed, and the list shows the division of segments of which the tool can be build from.

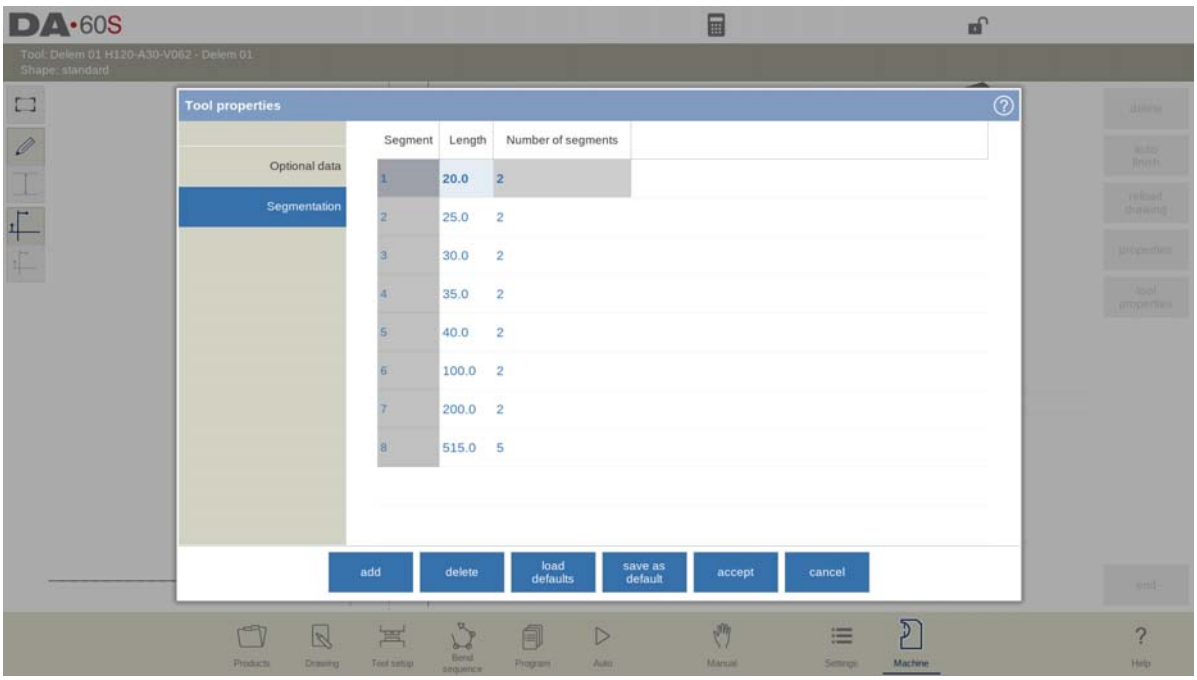


Segments can be changed within the segmentation view. The available segments in stock are not taken into account at that time. With a renewed segmentation this can be verified.

When changing tool length or type, segmentation will be lost, and need to be generated again.

4.5.3. Segments in the tool library

To enable the use of segments, and the calculation of segmentation based on the available segments, one needs to fill out the library. This can be done in the tool programming, to be found in Machine mode, under Punches or Dies in the tool properties.

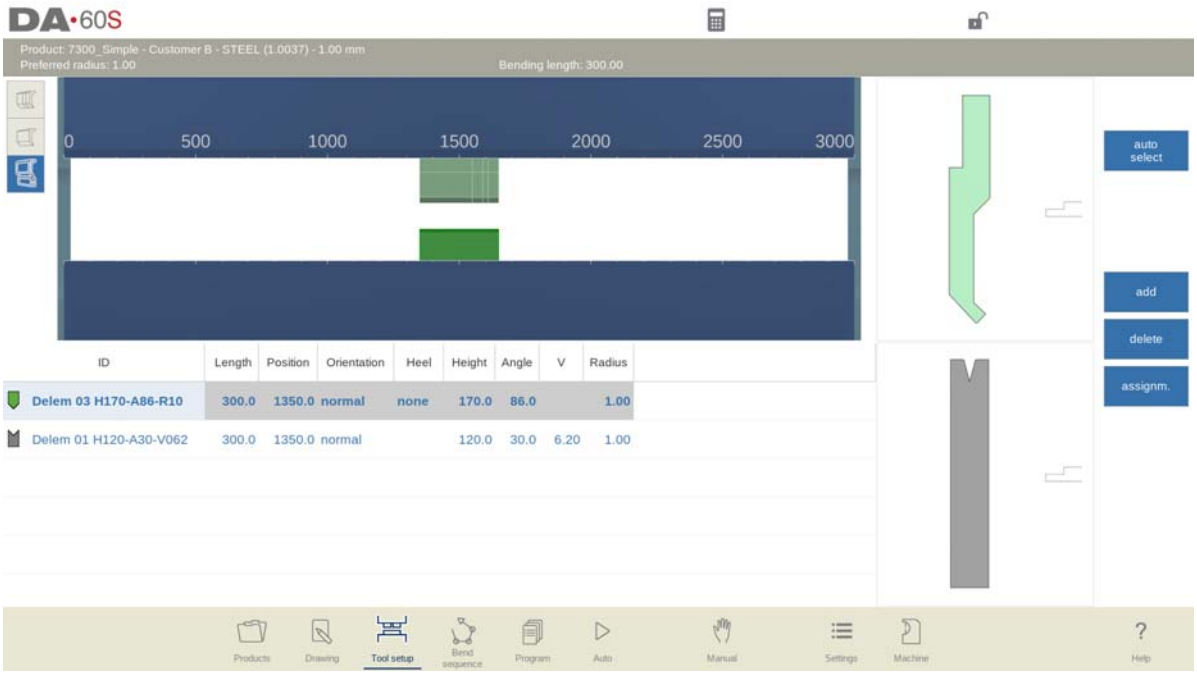


In every tool, the segment's length, the optional heel shape and the available amount of segments can be programmed on the Segmentation tab.

4.6. Station selection and repositioning

The third Tool setup view is the Station view. In Station view the complete tool stations are highlighted when selected, and can be repositioned by programming an alternate position or dragging to the desired new position in the machine.

A tool station is automatically defined when there is an overlap from punches with dies. This, meaning that a tool station is considered a station when e.g. there is an exact position of punch and die opposite of each other. When there is a shifted position but still overlap in between punch and die, this is still considered a tool station. Even when two punches are opposite of a single die, which can be useful bending constrained bends, this is considered a tool station. These stations can be repositioned without losing their relative positioning.

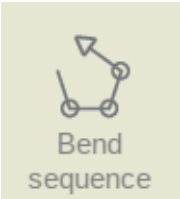


In station view an existing tool station can be copied with the ‘add’ button, and deleted with the ‘delete’ button.

Station view is not changing anything to the tool details.

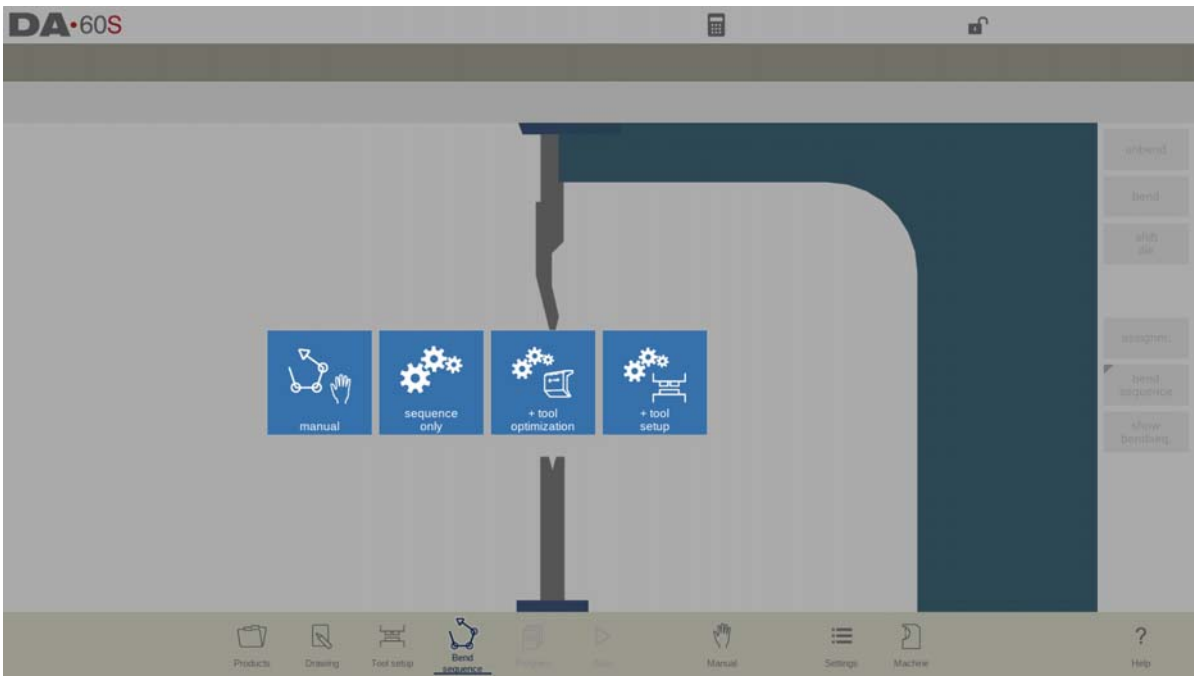
5. Bend Sequence

5.1. Introduction



To generate or modify a bend sequence for the product drawing, select the product from the library and use Bend Sequence

When a tool configuration is available, the bend simulation can be started to determine a bend sequence for the active product. The bend sequence determination is started by tapping the navigation button Bend Sequence.



The bend sequence determination can be realized by automatic calculation starting with the bent product. It is also possible determining the sequence manually starting with the flat product, not using the automatic calculation.

In the bend sequence screen a new product will start with the selection of the bend sequence method. One can select Manual bend sequence programming, Automatic bend sequence computation, Automatic bend sequence computation with tool optimization, or Automatic bend sequence computation with renewing the tool setup. When starting the computation, the product is shown in its final state. In order to obtain a bend sequence, the product will be unfolded from the last bend to the first. This is either done automatically, or can be done with the available function keys.

When it is preferred to program the bend sequence manually, it will start with the unfolded product. Manual bend sequence determination is a main selection in the bend sequence

screen.

Main bend sequence selection options

Manual
With this determination method the bend sequence has to be programmed manually.

Sequence only
This computation method will calculate a bend sequence based on the existing tool setup in the machine.

Sequence + tool optimization
This computation method will use the existing tools in the machine to optimize tool positions and tool stations in order to have an optimized bend sequence on the adjusted tool length and positions.

Sequence + tool setup
This computation method will remove the existing tool setup and calculate an optimized bend sequence based on the best fit tools for every bend from the tool library (resulting in a most optimal product radius).

During the computation the bend sequence computation can be canceled or stopped;
- Cancel is shown when no valid bend sequence has been computed yet
- Stop is shown when a valid bend sequence has been computed already. This could be the not optimum in production time.

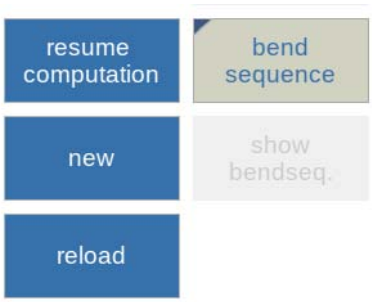
Functions

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

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

Assignments
Open a screen with assignments, parameters for the bend sequence computation.

Bend sequence
To resume the computation of the current bend sequence, to start a new bend sequence, to reload the original bend sequence or to save a complete bend sequence. These functions can be found under the bend sequence command button.

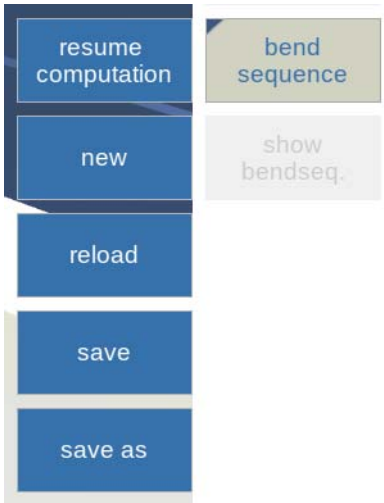


+Resume computation
Compute a valid bend sequence for the remaining bends starting with the selected bend.

+New
Start a new bend sequence, any existing sequence is removed.

+Reload
Restore an existing bend sequence from disk, but disregard any related CNC program. This is also including corrections.

After a bend sequence has been calculated or determined, the Save and Save as can also be found in this menu.



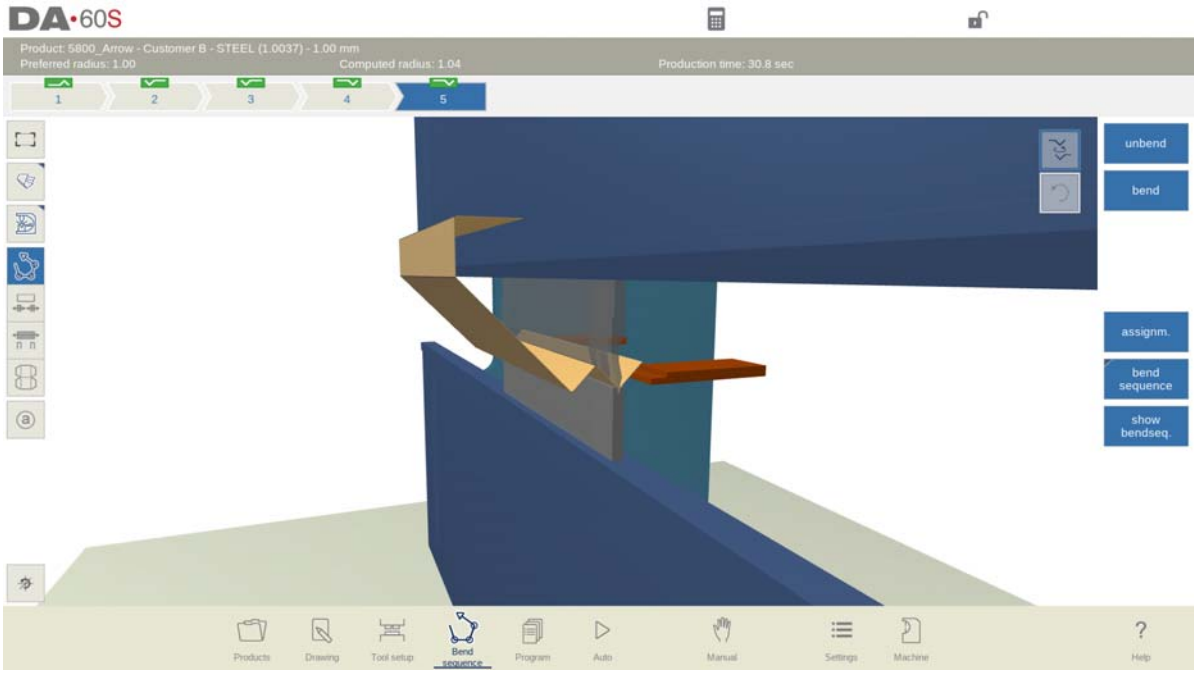
+Save
Save a bend sequence will save the resulting CNC program on disk. The resulting CNC program contains all necessary axis positions and tool IDs. This command can only be executed when a complete bend sequence has been determined.

+Save as
Save as can be used to save the current product and its bend sequence under an alternative name. This enables an easy way to make product differentiations (variants).

When the bend sequence screen is left, the product will automatically be saved. If there are changes made, and save is not used, this will be prompted on the screen.

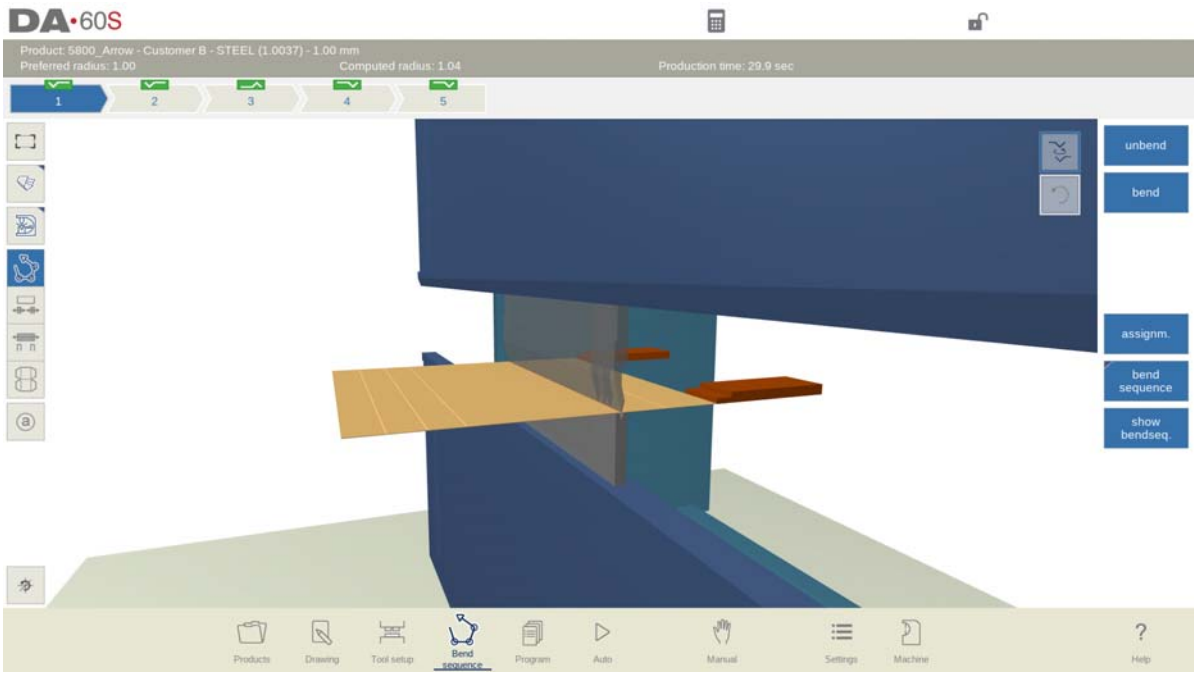
Show bend sequence
To show a mosaic screen with a step-by-step graphical overview of the bend sequence.

The bend sequence can also be viewed and determined in a 3D view. This can be achieved via the view mode selection.



5.1.1. View select

Within the bend sequence screen views can be switched upon required choice.
The view functions are located opposite of the command buttons in the main screen.



View functions

The upper command icons give direct access to 2D, 3D view selection.

The visualization of machine, tools and product can be selected with the second set of command icons in the screen.
Toggle between some possible ways of displaying the product/tool configuration:

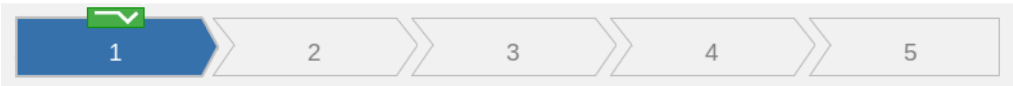
- product
- product/tools
- all

Reset view function, to reset the view (i.e. rotation and zoom) to default. When pressed again, the question to reset the view for all bends is shown.

Fit to view function, to scale the product/tools/machine to fit the view.

5.1.2. Bend selector

Within the bend sequence screen bends can be selected and navigated thru with the bend selector. In the top of the screen the number of bends is indicated with preliminary bend selectors. After completing the bend sequence, these are all colored, active and show a turn indicator.



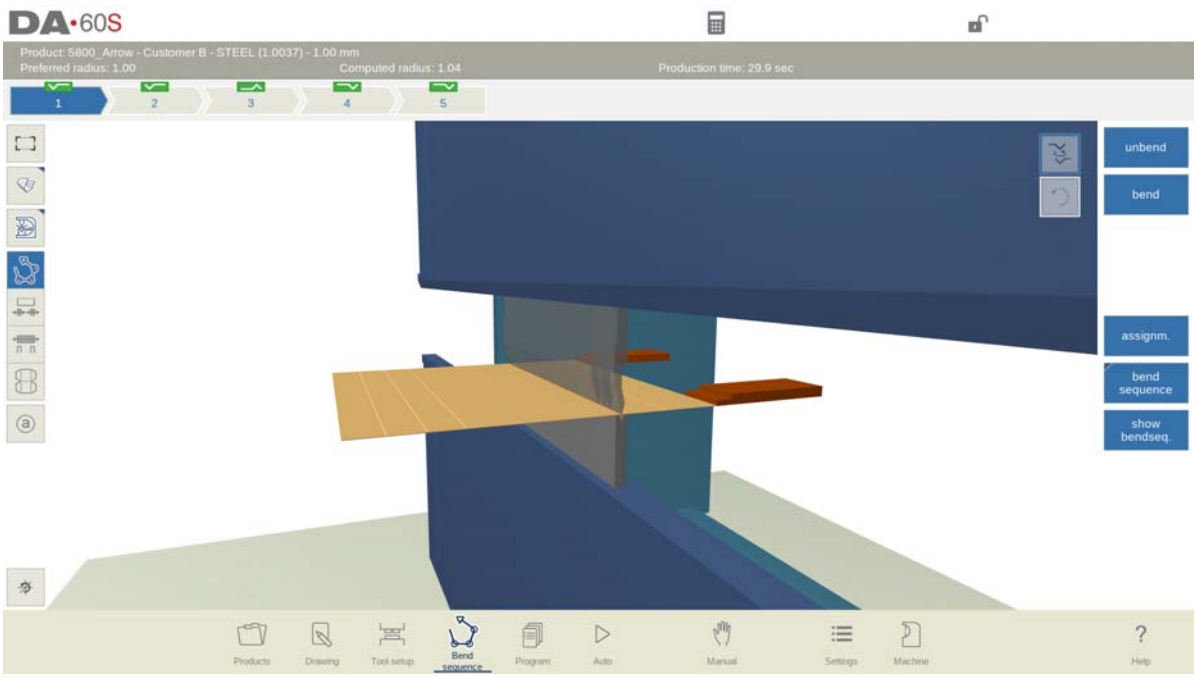
From then bends can be tapped to easily to select the desired bend data. In the bend selector the turn indicator will be displayed showing green, yellow or red colors to indicate the level of complying the assignments of the bend sequence.

5.2. Unbend product

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

- Press the function key Compute. The control will automatically compute the quickest possible bend sequence for this product.
- Press the function key Unbend repeatedly, until the product is completely unbent.

When the product is completely unbent, press the function keys Bend sequence and Save to generate and save the CNC-program.

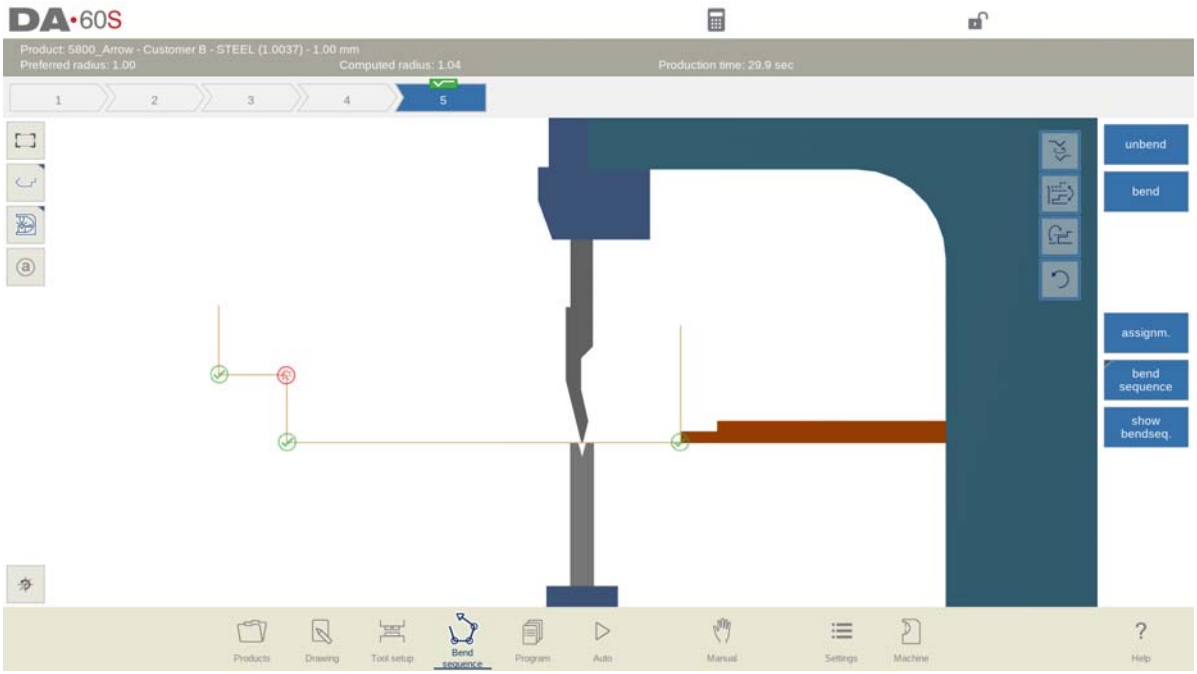


It is possible that no bend sequence has been found for various reasons:

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

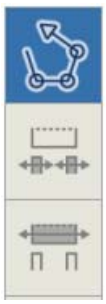
5.3. Manual selection of bends

Normally the control proposes the next (un)bend in a sequence. This is computed by the control depending on the programmed assignments and of course the product shape and applied tools. For various reasons it can be necessary to choose another bend for the bend sequence. The bend sequence can be determined/changed through the manual selection features in the bend simulation menu.



One can either start from a bent product pointing out the desired unbend sequence, finding the desired bend sequence or start with a flat sheet pointing out the bend sequence. This selection is done tapping the bend sequence button. One can select ‘new’ for starting an unbend sequence, or ‘new flat’ for selecting a bend sequence.

The ‘manual bend sequence mode’, which is active in 2D view by default and which must be selected in 3D view on the left side of the screen, must have been selected. Other modes, which are available in 3D view only and which disable the manual bend sequence mode, are ‘shift gauge mode’ and ‘shift product mode’.



Starting from a bent product, one can tap any bend adding it to the (un)bend sequence. The bend selector will indicate it has been added at the next free position. While unbending starts from the highest bend number towards bend 1.
Starting from a flat unbent product, one can tap any bend adding it to the bend sequence. In

this case it is being bent and the bend sequence will be filled from bend 1 towards the highest bend number.
The bends which are possible without collision are shown with a green circle; the bends resulting in a collision will be shown with a red circle. Yellow circled bends need e.g. a swap to fit without collision.
When tapping a bend already in a bend (or unbend) sequence, the program will select that bend step to continue the bend sequence from there.

Functions

The following functions, available for modification of the bend position, are shown in the screen when active.

Swap
Turn the product between the tools (back to front).

Change side
Alters through the available backgauge positions.

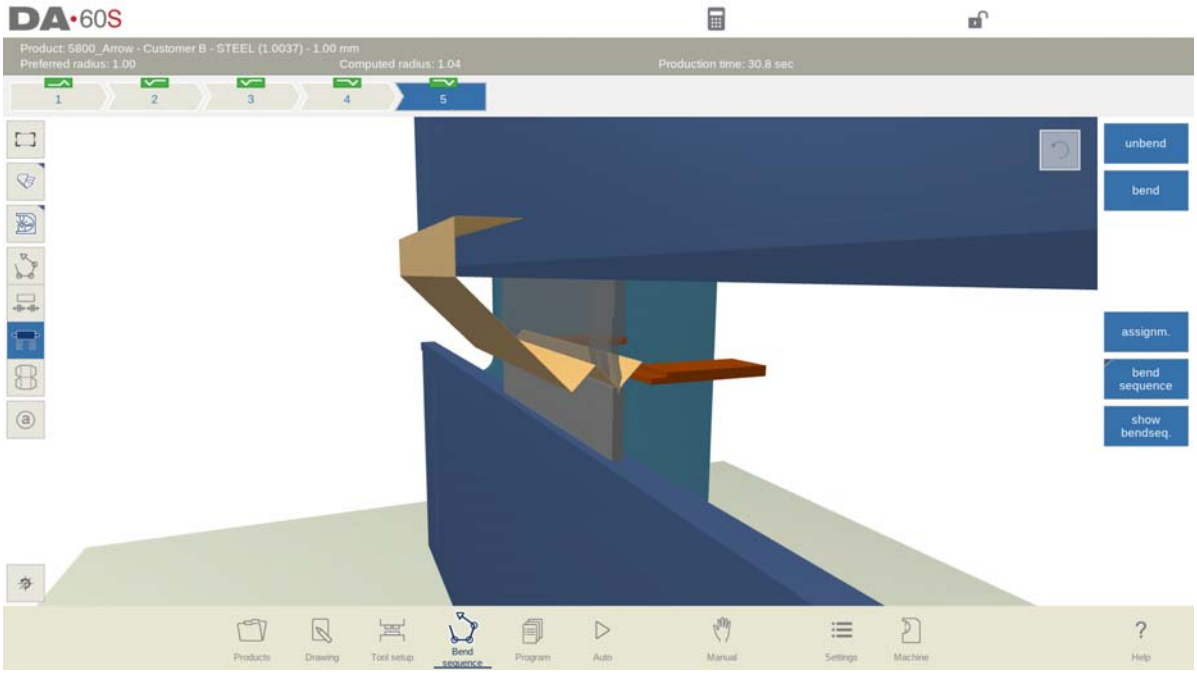
Lay on finger
Alters through the available product lay on positions. This option is only selectable in case you have an R-axis in your machine.

Undo
Undo the last modification to the bend sequence.

5.3.1. Shift product

In the bend simulation menu, the control computes the next possible bend to unbend. The product is placed between the tools, where there is no collision with the tools or the machine. In case you want to shift the product under the tool set (which is mounted), you can move the

product by selecting the function ‘Shift product’.



Functions

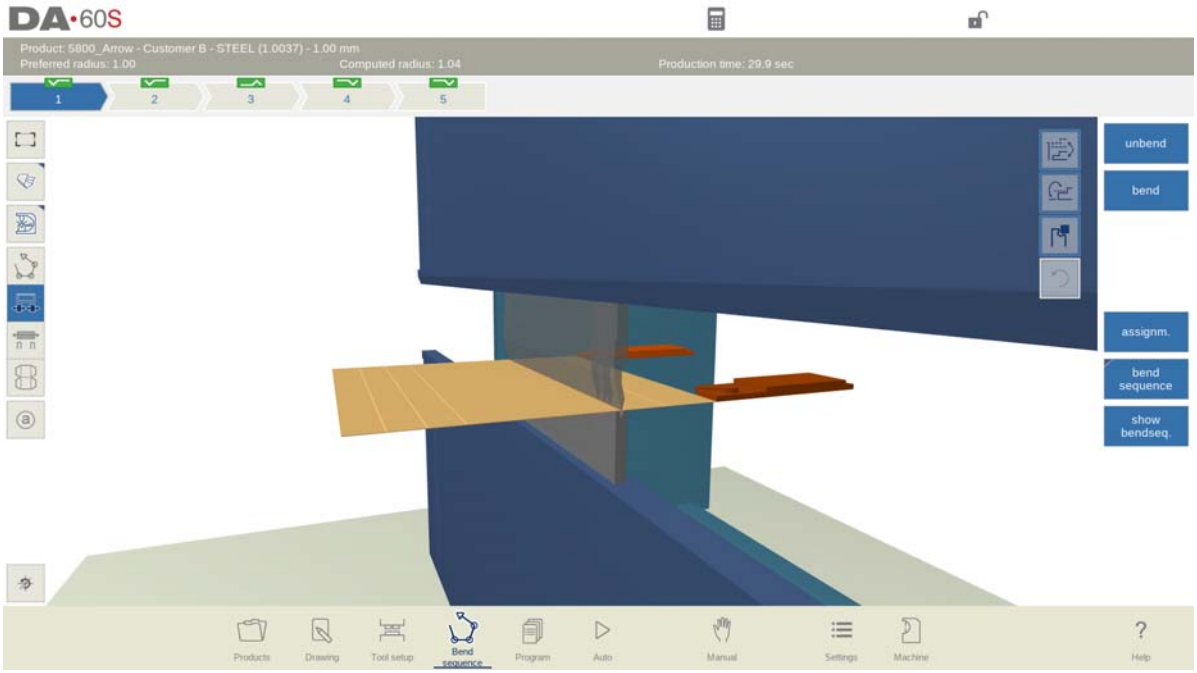
Shift product
Move the product by dragging it to the desired position, upon choice under a different tool station.

Undo
Undo the last change to the product position.

5.3.2. Shift gauge

The control automatically computes at each bend the X axes, R axes and Z axes positions. It takes into account the values of the option assignments and searches for a solution without collision of the fingers with the product. In order to be able to choose alternative positions, you

can move the fingers manually by selecting the function ‘Shift Gauge’.



The fingers can be moved by means of the different available functions.

Functions

Select finger
Select the finger to move.

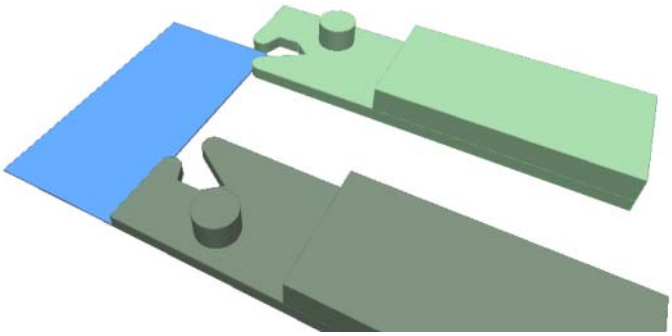
Shift finger
Move the selected finger by dragging it to the desired position.

Change side
Alters through the available backgauge positions.

Lay on finger

Alters through the available product lay on positions. This option is only selectable in case you have an R-axis in your machine.

Pinch corner
Position the finger around the corner of the contact surface to fixate the product both in X-direction and Z-direction; this is only possible on a machine with special fingers that have the pinching possibility.



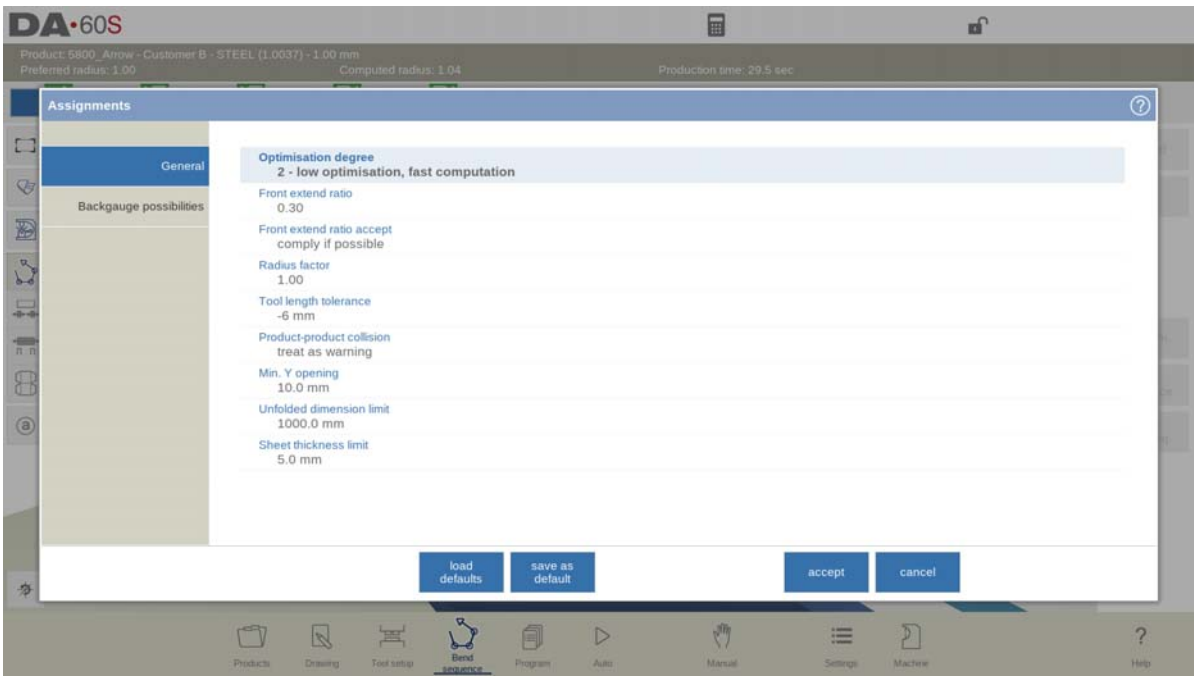
Undo
Undo the last change to the finger position.

5.4. Assignments

5.4.1. Introduction

The Assignments are parameters with which the bend sequence computation is controlled. The assignments screen is opened from the bend sequence screen with the function key Assignm.

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



Functions

Load defaults
Load default assignment settings. It is possible to determine a set of assignments which have the most optimal values for your situation. This set can be stored by tapping Save As Default. While programming another product you can recall this previous fixed set by loading the values via Load Defaults.

Save as default
Save the current settings as default assignments settings.

Cancel
Leave the current screen without saving changes.

Accept
Save the changes and leave the current screen.

5.4.2. Assignments - general

Optimization degree

Range 1-5.
The number of alternatives to be computed for each bend must be entered here.
The higher this number the more alternatives are to be examined by the control, so the longer the computing time will be:
1 - lowest optimization, fastest computation
2 - low optimization, fast computation
3 - medium optimization, medium computation
4 - high optimization, slow computation
5 - highest optimization, slowest computation

Front extend ratio

Range 0.01 - 1.0.
This is the ratio of the minimum allowable length of your product which extends in front of the press to the total blank length of the product. You must have a minimum length of your product in front of the press to be able to handle the product.

Front extend ratio accept

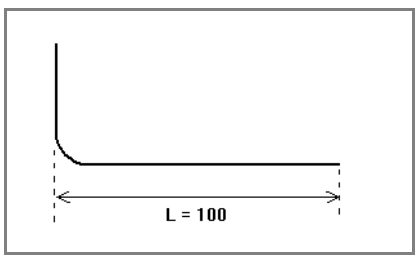
Comply If Possible:
This means that when possible the computer tries to comply to the front extend ratio and only when this will result in no solutions to be found it will accept that the length in front is smaller than the specified ratio.

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

Radius factor

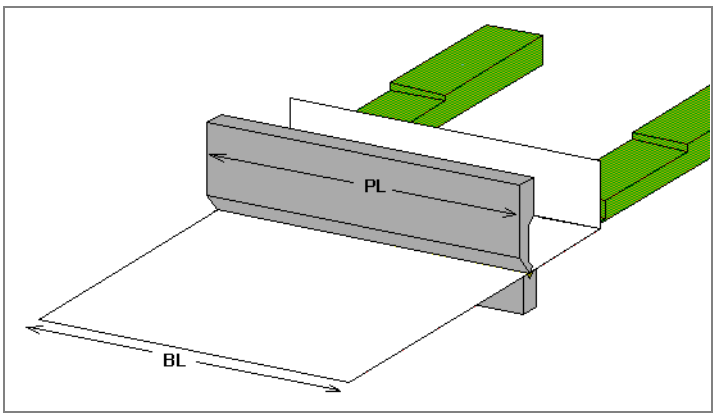
Range 0.1 - 2.0.
The computed radius of a bend is multiplied by this factor for correction purposes. This correction affects the X-axis position and the computed blank length. The initial value is 1.

The control computes the X-axis position necessary to obtain L=100 as shown in the figure below. The accuracy of the length L is dependent on the material parameters like thickness, strength and kind of material. In order to have a correction possibility with the radius factor you can optimize this computation.



Tool length tolerance

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



Product-product collision

Specify if product-product collision must be detected:

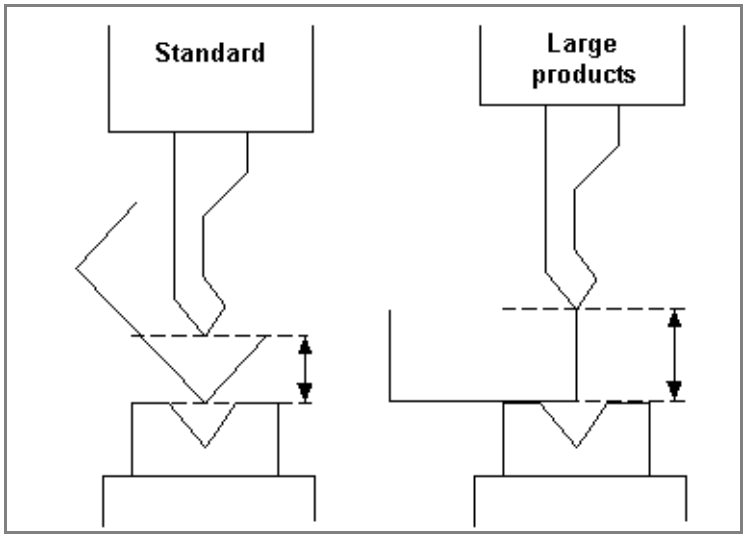
- disabled: product-product collision detection is disabled
- treat as warning: product-product collision detection is enabled, and a product-product collision is treated as a warning
- treat as error: product-product collision detection is enabled, and a product-product collision is treated as an error

Minimum Y opening

During post processing of the programmed product, the control always computes an optimal opening of the press beam to handle your product. Here you can program a minimum required opening. The programmed value is the distance above the speed change point (Mute).

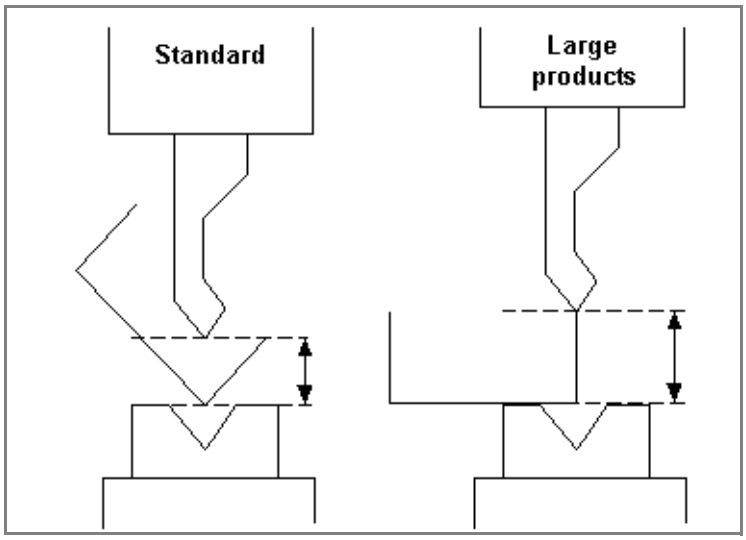
Unfolded dimensions limit

The Y opening is calculated related to the boundaries of the product. Upon the setting of this parameter, beyond the unfolded dimension limit, the opening (for large products) will be based on the side length, preventing necessary tilting to move the product out.

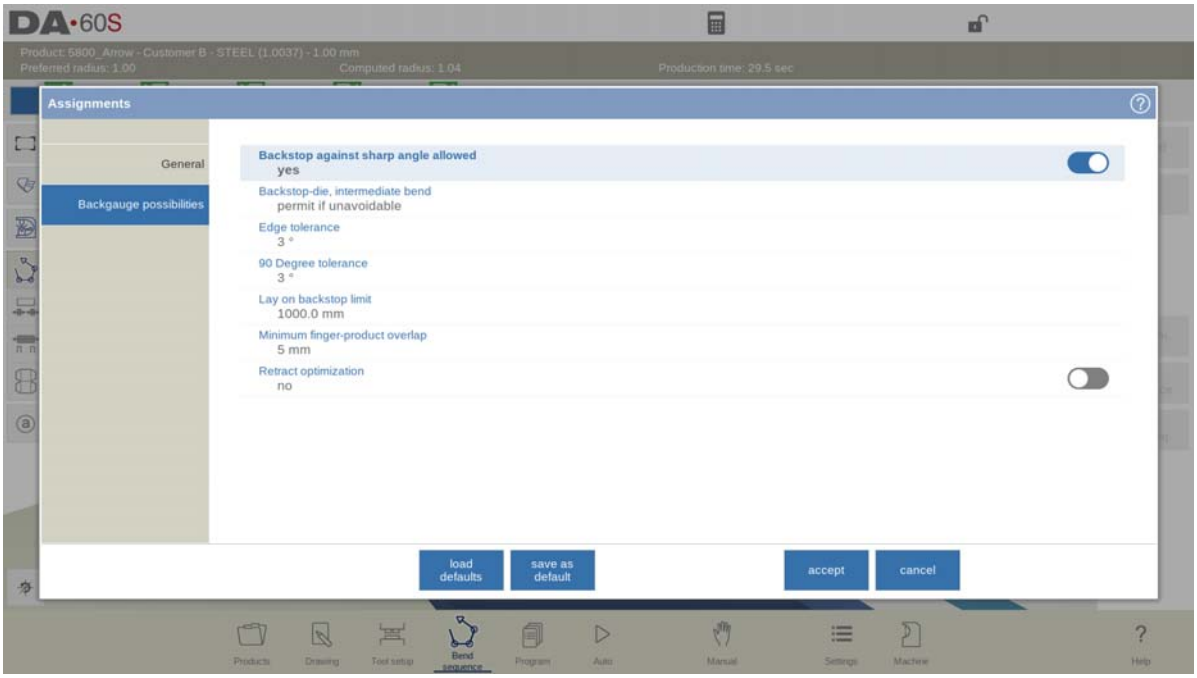


Sheet thickness limit

The Y opening is calculated related to the boundaries of the product. Upon the setting of this parameter, beyond the sheet thickness limit, the opening (for large products) will be based on the side length, preventing necessary tilting to move the product out.



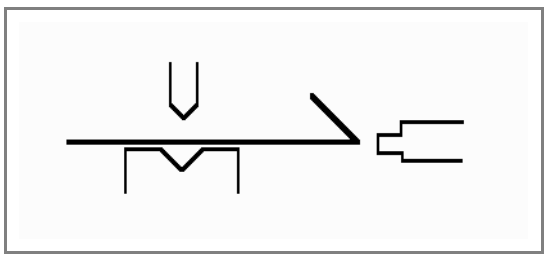
5.4.3. Assignments - Backgauge possibilities



Backstop against sharp angle allowed

Specify if backstop may be placed against an angle smaller than 90°.

- No = not allowed
- Yes = allowed

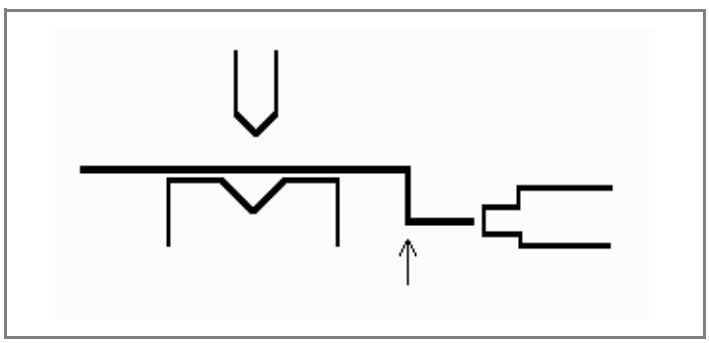


Backstop-die, intermediate bend

Set to allow if there may be a bend between the die and backstop.

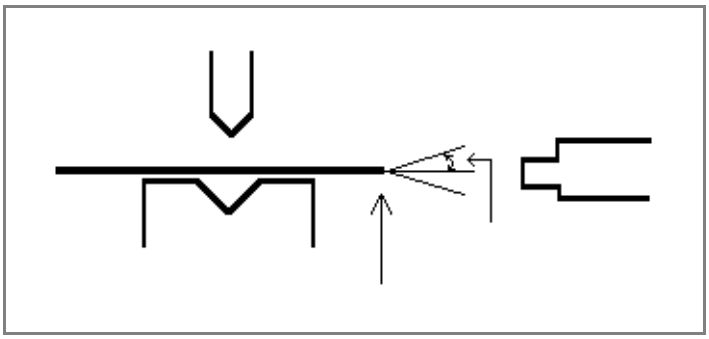
Selection possibilities:

- Permitted
- Permit if unavoidable: if it results that no solutions are to be found, than it is permitted
- Prohibited: never allowed



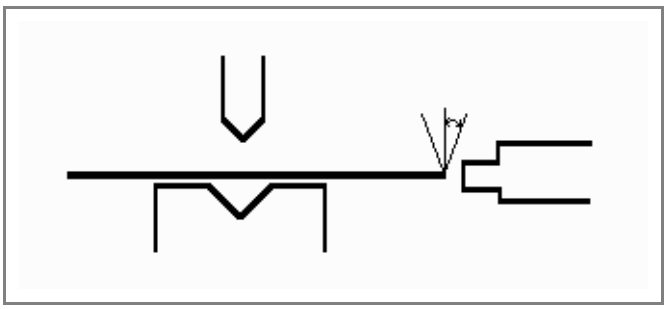
Edge tolerance

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



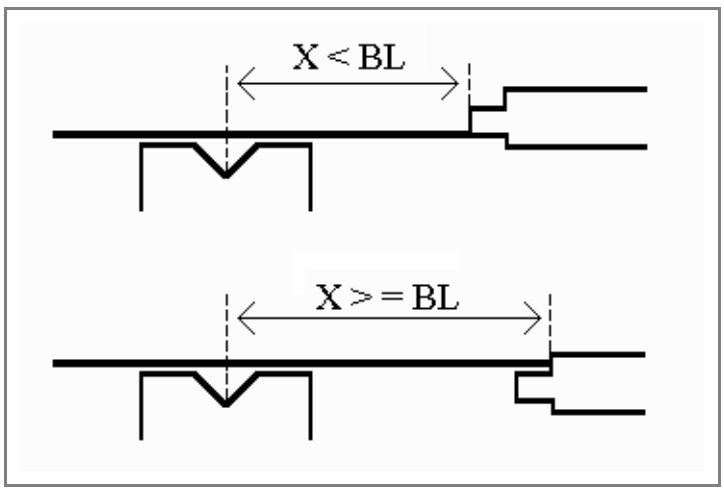
90 Degree tolerance

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



Lay-on backstop limit

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



Minimum finger-product overlap

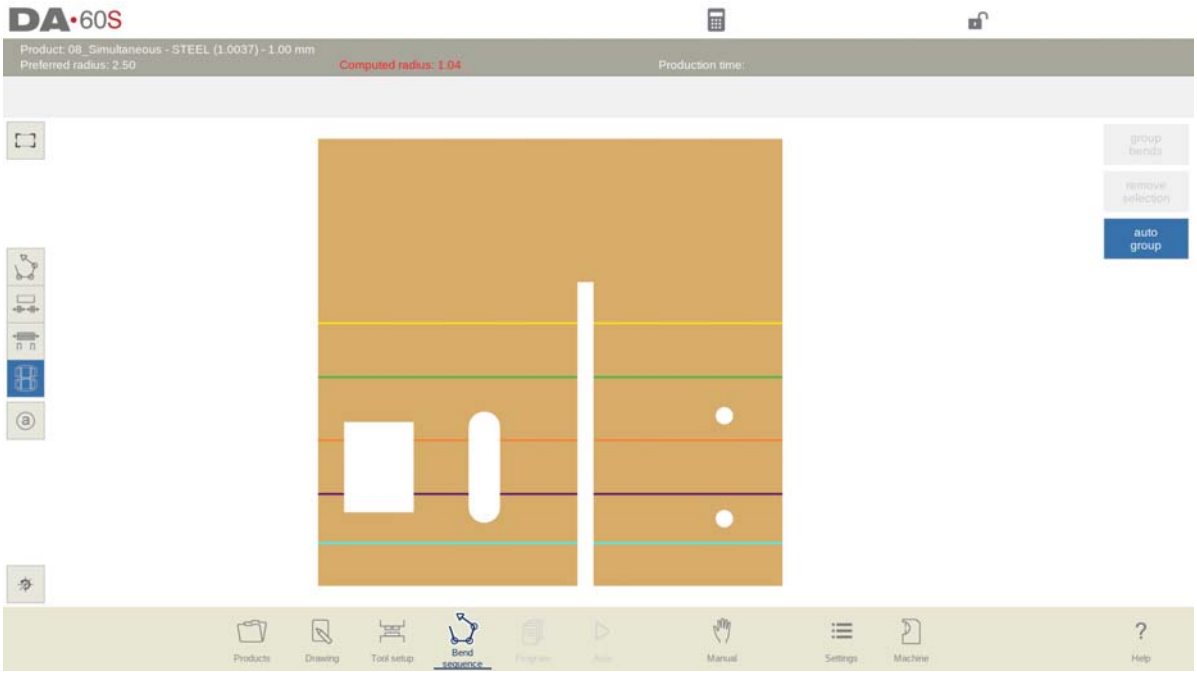
The minimum overlap between backgauge finger and product, to avoid unnecessary/ unwanted parking of a backgauge finger in the bend sequence calculation.

Retract optimization

Retract optimization for graphical bend steps. When enabled, an optimization is carried out during the post processing and storage of the product. When retracting in the direction of the next bend's backgauge position, that new backgauge position will be used as retract position. So, the backgauge will directly be ready for the next bend step. By default this optimization is off.

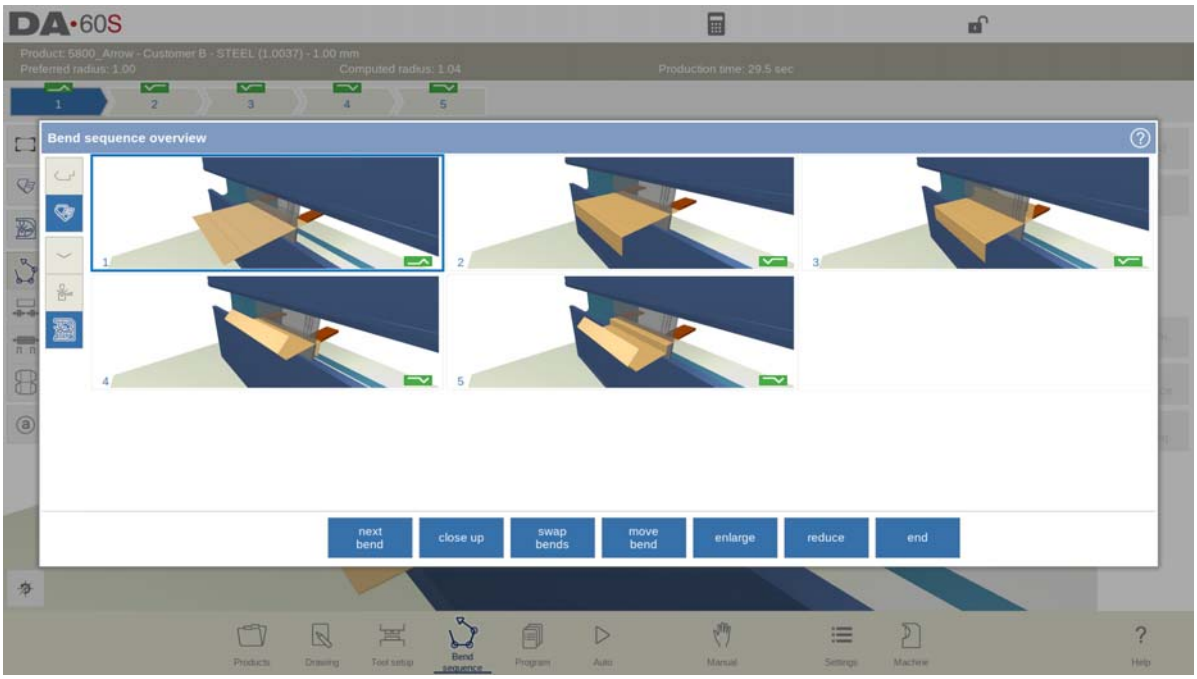
5.5. Simultaneous bends, group selection

Additionally to the automatic selection of simultaneous bends and constrained bends a group function is available in the bend simulation menu. By selecting the unfolded view in the bend simulation menu one can select, deselect and group bend lines. This can be very useful in case of bend lines which are not seen to be in the same line, able to be bent simultaneously. In case the control has grouped bends automatically one can ungroup and bend bend lines separately.



5.6. Show bend sequence

When the function Show Bend Sequence has been pressed, a graphical overview of the bend sequence is shown.

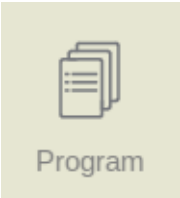


This option can be called at any time after the first unbend has been made. The graphical overview displays the determined bends as well as the not yet determined bends (question mark sign).

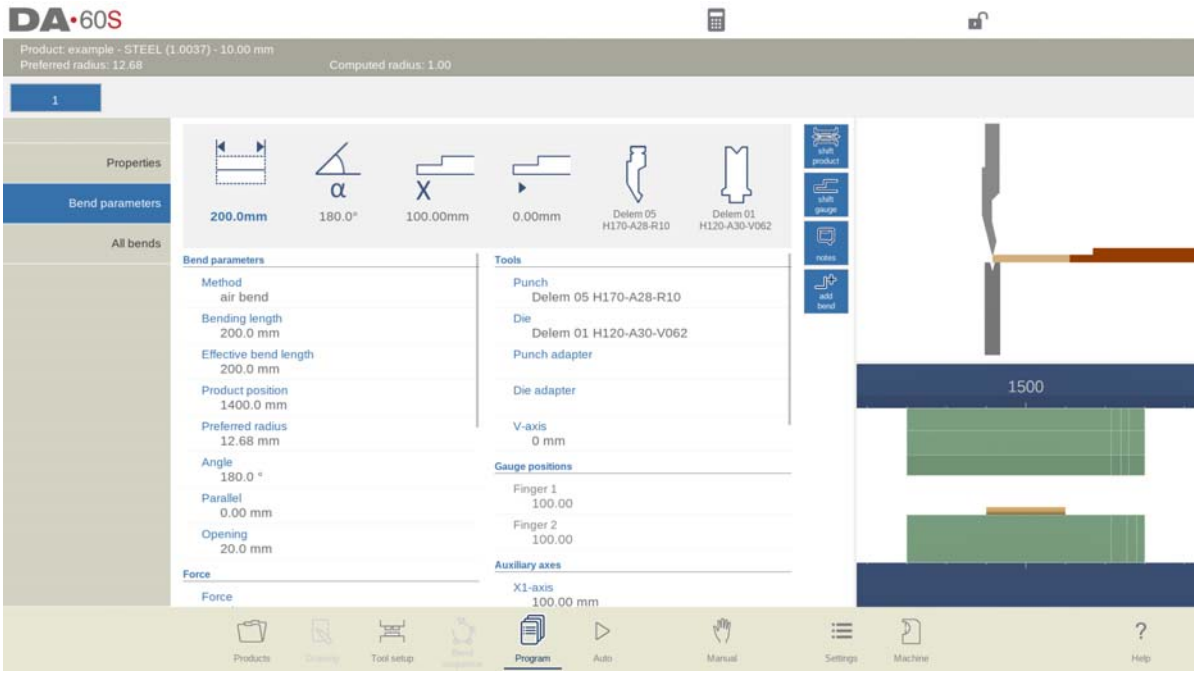
Each image in the overview can separately be enlarged or reduced with the available functions. The images can also be rotated by finger movement.

6. Product programming

6.1. 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. At the side of the screen a graphical representation is shown and the position of the product in the tool station. The bend selector in the top of the screen can be used to navigate through the bends. The indicated bends can be tapped to easily select the desired bend data.

Above the numerical information, large icons (tiles) with related values are showing the most used parameters. These tiles can be selected, giving direct access for modification. The values can also be found amongst all other numerical program parameters.

At the side of the numerical information views and functions are indicated with command buttons.

Functions

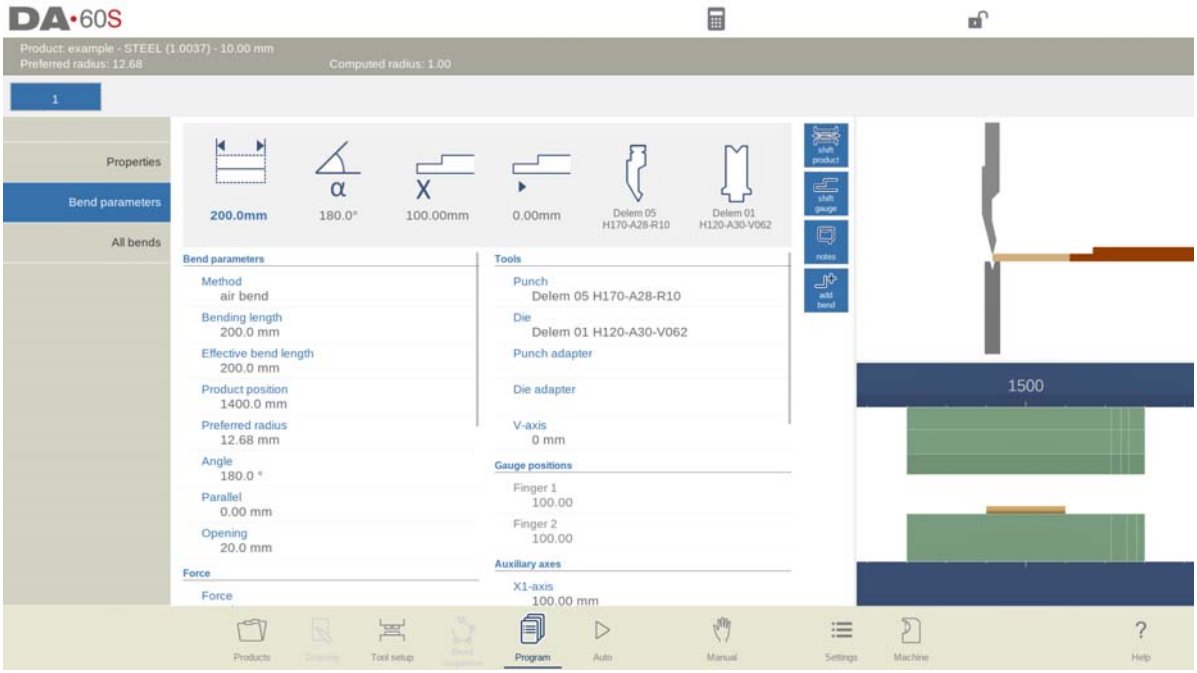
Following modes / functions are available:

Bend parameters	Main, numerical bend data which can be combined upon choice with graphical information / simultaneously shown (if available).
All bends	All Bends, shows all bend data in a table upon choice simultaneously with graphical information of the selected bend (if available).
Properties	Product Properties, gives access to product properties which are general for the product, not bend specific.
	Shift Product, enables the positioning of the product in respect to the tool positions.
	Shift Gauge, enables the repositioning of the backgauge fingers. Depending on the product type and option software for the gauge wizard, it can offer the graphical gauge wizard for finding alternative gauge positions.
	Notes, opens a window enabling viewing and editing of notes about the current product or specific bend. These notes are available during production.
	Add Bend, adds a new bend to the existing program.

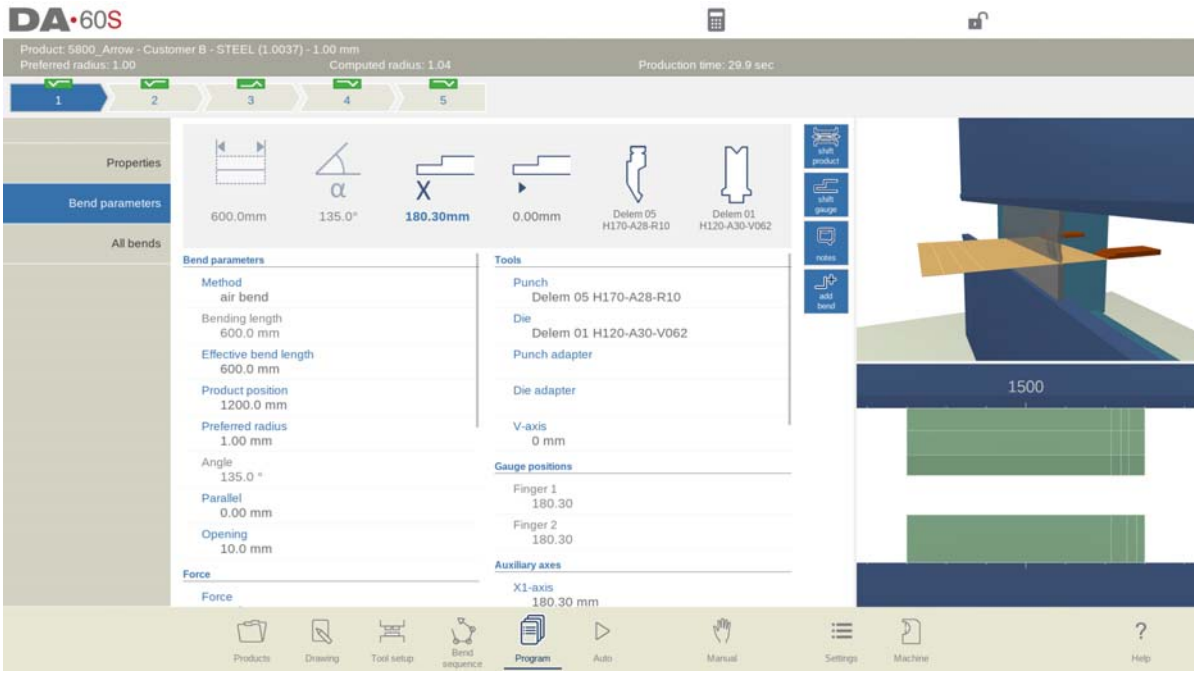
6.2. Program mode, parameter explanation

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

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



In case of a graphical product graphical information can be shown as well.



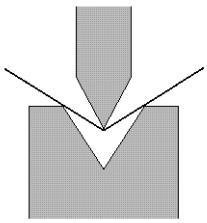
6.2.1. Bend parameters

Method

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

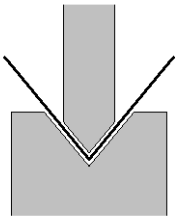
- air bend
- bottoming
- hemming
- hemming & bottoming
- handling

Bend methods



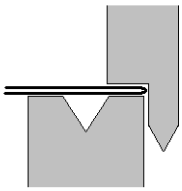
air bend

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.



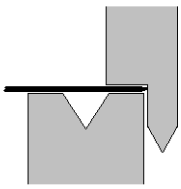
bottoming

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.

handling

No movement of the Y-axis, so the current program step becomes a non-bending step. Only the backgauge axes will position according to the programmed values, e.g. in order to enable handling of the product. An external step change (C-input) is required.

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.

Deflect method

dynamic =>The crowning will be controlled automatically in real-time during the bending, applying the proper corrections when needed.
non-dynamic =>The crowning will behave as a standard hydraulic crowning; there won't be real-time corrections.

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.

Preferred radius

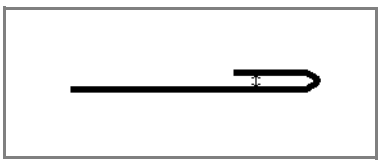
The intended radius which is programmed. As initial value the programmed product thickness is used for default radius calculation. This radius can also be a larger radius requiring large radius tools.

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.



Fast hemming

When fast hemming has been enabled, the Y-axis will move at high speed down as soon as it is below the top of the die, until the hemming load opening has been reached. This will reduce the cycle time, especially in case of hemming with a U-die.
Only available when enabled by the machine manufacturer.

Y-axis (Bend position)

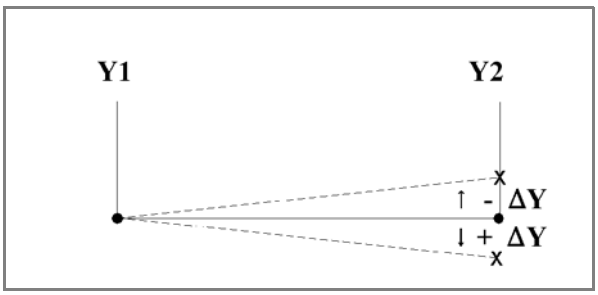
The required Y-axis position for this bend. This parameter only appears if absolute programming is selected with the parameter 'Angle sel.' This parameter also appears if the bend method is bottoming and/or hemming.

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.

6.2.2. 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.
The decompression distance can optionally be set to a specific point in the cycle. Standard the decompression distance is computed, the value which is minimal required. Optionally one can chose for Mute, the calculated mute point, or Clamp, the calculated

clamping point. Both options have a longer distance than the calculated one.

6.2.3. 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.

6.2.4. 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:

- ER: Bending number change (step change) at end of decompression (next bend parameters active).
- MUTE: Step change at muting position when the beam moves in opening direction.
- UDP: Step change at upper dead point.
- UDP STOP: Step change at upper dead point without movement of any axis and the control goes to “stop”.
- 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 backgauge performed. See also code 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 backgauge will be performed.
- UDP PST: Step change at upper dead point when the part supports have reached their zero position. Only available if a part support is present.

Delay time

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

6.2.5. Tools

The set of 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.

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.

Punch adapter
The name (ID) of the selected punch adapter. Tap to modify or select from the punch adapter library. Whether an adapter can be programmed depends on the parameter Use punch adapter in the Machine mode.

Die adapter
The name (ID) of the selected die adapter. Tap to modify or select from the die adapter library. Whether an adapter can be programmed depends on the parameter Use die adapter in the Machine mode.

Press the Change Tools button to get an overview of the available tools in the library. Press the function Orientation in the tool selection overview to change the orientation of the tool (i.e. to turn the tool around).

6.2.6. Gauge positions

Finger (1/2/3/4)
The finger (contact) position, corresponding to the X-axis position and lay on position.

6.2.7. 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.

6.2.8. Part support

PST-axis
With this parameter the part support can be switched on or off. When it is switched off, the part support will stay at its zero position during the bend.

R-position

The height of the part support before and after the bend. By default, the height will be set at the top of the die (R-position = 0.00 mm).
Only available when enabled by the machine manufacturer.

Method

With this parameter one can define how and when the part support should move back to its zero position after the bend has been completed. Upon choice one can select the part support to return when the Y-axis is at UDP (‘Return at UDP’) or to return already when the Y-axis is opening (‘Return when opening’). Other choices are ‘Product tilt‘, which will enable the use of an extra tilt to avoid collision between the product and the punch during the Y-axis opening movement, and ‘Static angle support’, which enables the user to bring the part support in a fixed angle; in that case the part support will not follow the product but act as a support table under an angle.

Tilt angle

The additional angle that the part support will move.
A part support at the front side of the machine will move to a higher position (bigger angle). A part support on the back side will move to a lower position (smaller angle).
Only available when method ‘Product tilt” has been selected.

Tilt speed

The speed to move to the tilt angle position; this is a percentage of the maximum operation speed.
Only available when method ‘Product tilt” has been selected.

Tilt clearance

In case the calculated (or programmed) decompression distance is not sufficient for the tilt movement, an additional distance (‘Tilt clearance’) can be programmed; this value is added to the programmed decompression distance. The part support will only follow during the movement of the programmed decompression distance, and will not follow anymore during the movement of the tilt clearance.
Only available when the method ‘Product tilt” has been selected.

Angle

The static angle to which the part support will move before the bend.
Only available when method ‘Static angle support” has been selected.

Speed

The speed, as a percentage of the maximum speed, with which the part support moves to the static angle.
Only available when method ‘Static angle support” has been selected.

Return speed

Return speed of the part support after a bend. The speed value is programmed as a percentage of the maximum speed.

Safety stop angle

The angle at which the support will stop during the return movement; the control will switch to stop. An extra start action at the control makes the part support return to its zero position.

Initial angle

The initial angle (typically the pre-bend angle of a two-stage bend) to which the part support will be moved at step change. The part support will follow as soon as the angle of the product passes the initial angle of the part support. Only valid for part support axes in front of the machine.

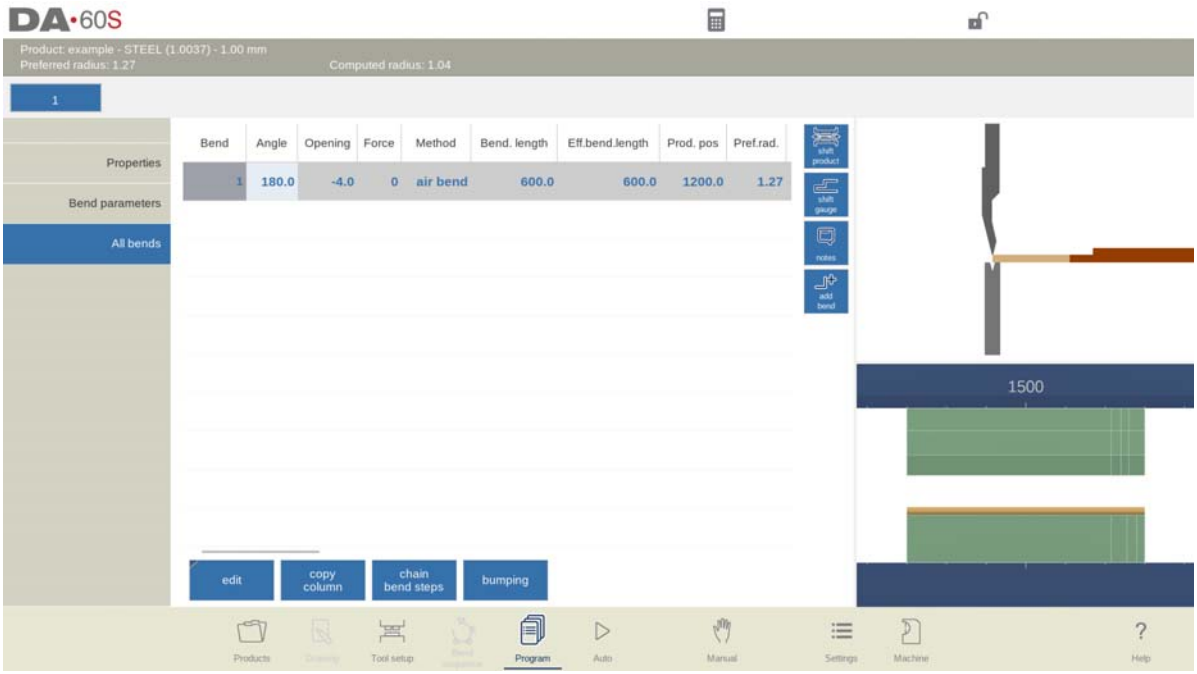
Initial angle speed

The speed to move the part support to the initial angle. Only valid for part support axes in front of the machine.

6.3. Edit / view modes

6.3.1. 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.

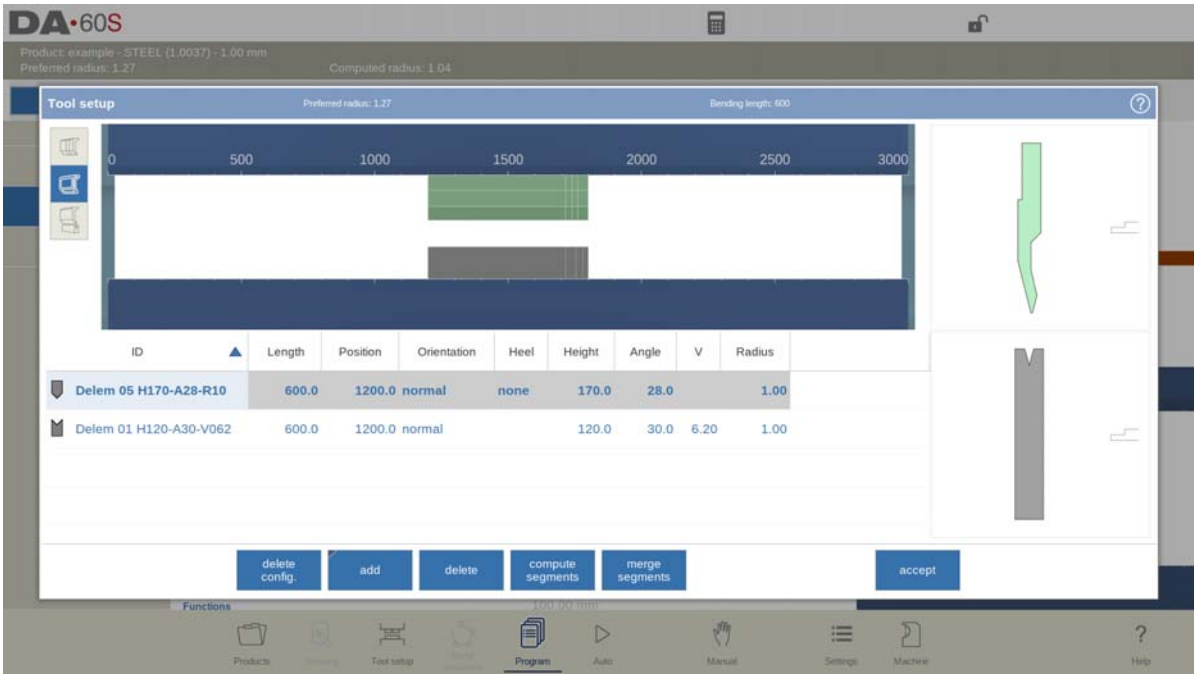
+Abort Mark
Remove the mark from the currently marked bend.

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.

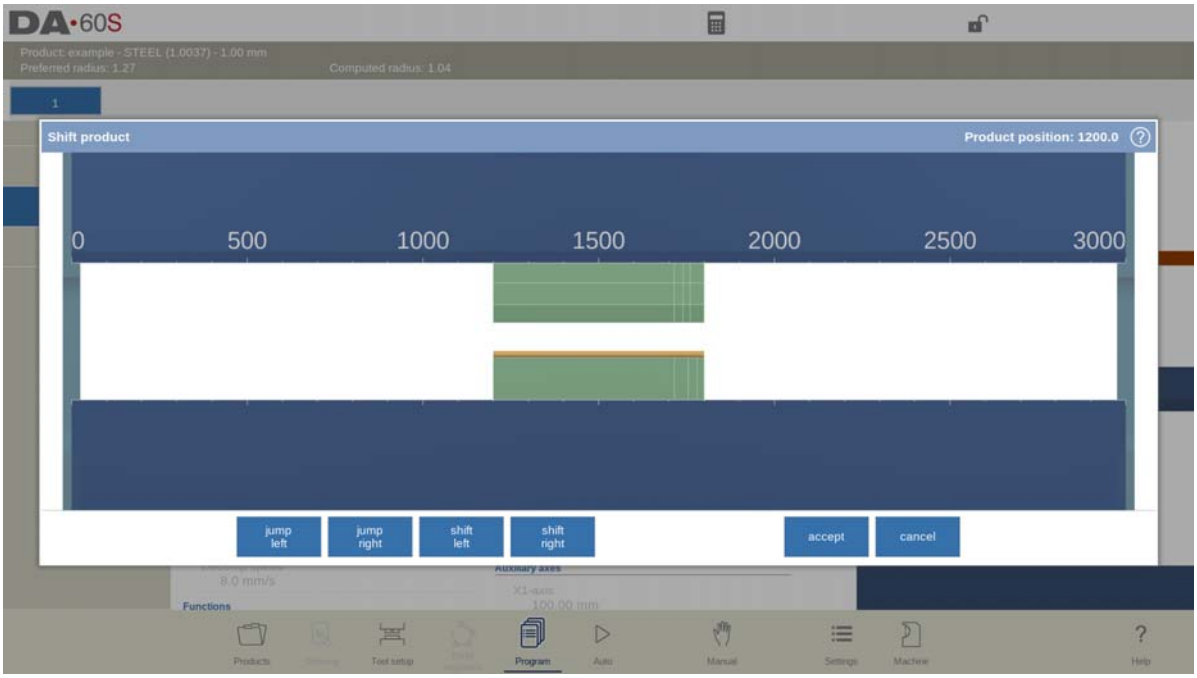
Chain Bend steps
Chaining of bend steps is possible by just setting the number of steps in the chain (Repetition) and the backgauge position offset (X offset). The relative offset value can be positive as well as negative for flexible chaining.

6.3.2. Change tools



To change the tools the Tool Setup menu can be used. When using the Program mode for numerical programming, the Tool Setup will be used as standard. 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.

6.3.3. Shift product



To shift the product's position the Shift Product function opens a dialog enabling the product to be shifted on and in between the tools. The product can even be shifted over tool stations. While shifting the user is helped with snapping points. The product's position can easily be moved to the side of tools or in the middle of the station.

Functions

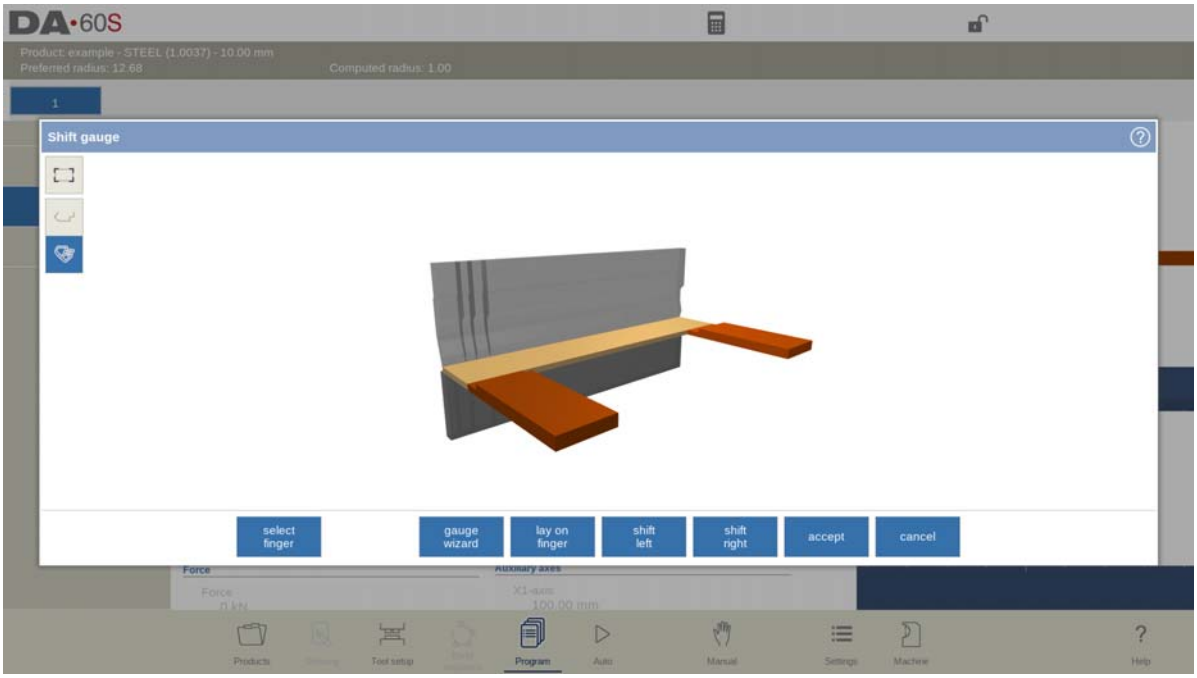
Jump Left
Move the product to another tool set combination in the left direction.

Jump Right
Move the product to another tool set combination in the right direction.

Shift Left
Shift the product 1 millimeter to the left within the same tool set.

Shift Right
Shift the product 1 millimeter to the right within the same tool set.

6.3.4. Shift gauge

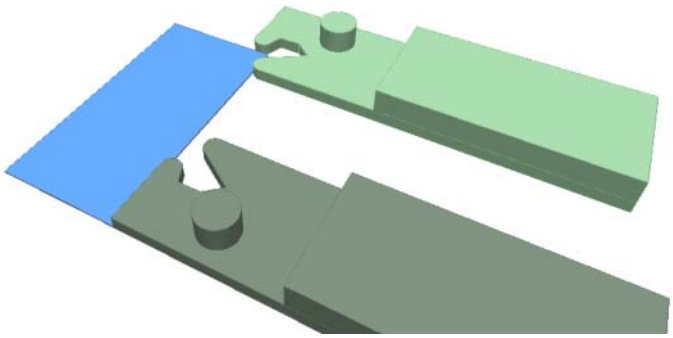


To change the positions where the backgauges are touching the product the Shift Gauge function opens a dialog enabling the user to modify the backgauge positions. In Shift Gauge the product's side is shown and the backgauge fingers can be dragged to a alternative position. The backgauge will automatically “find” the product side as exact positions will be programmed in this way.
Next to changing the position also lay-on can be selected within Shift Gauge. This function will automatically bring the backgauge finger to an available different height starting the use of the backgauge finger position for this height level.

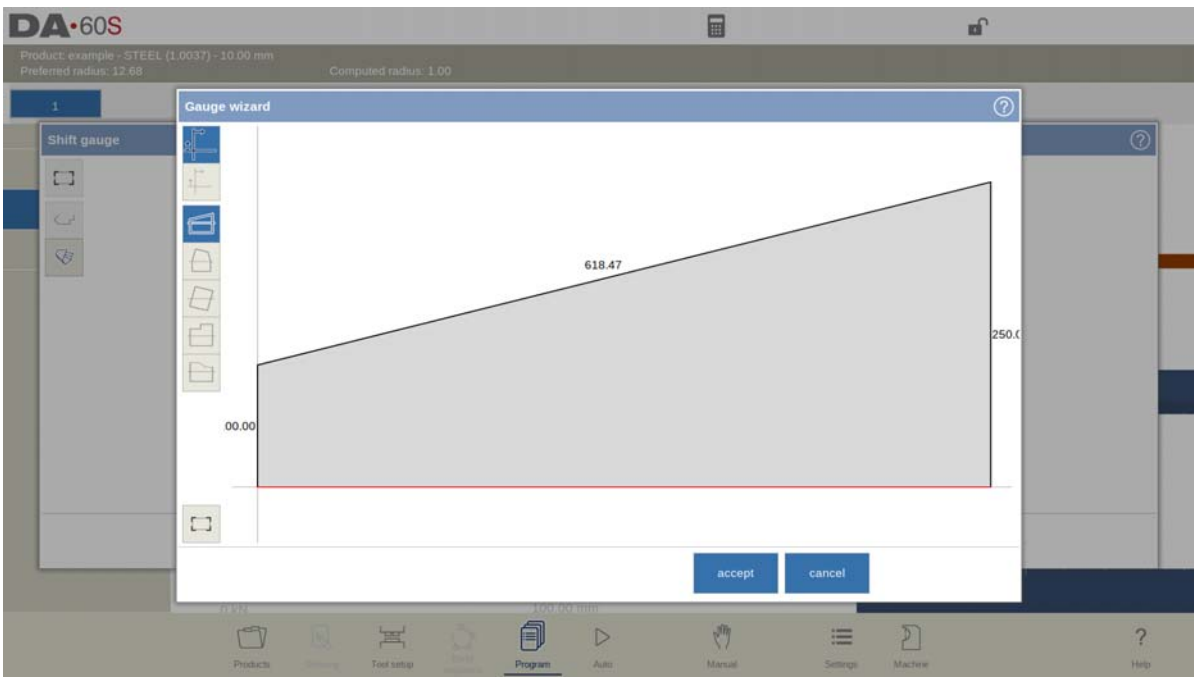
Functions

Select Finger
Select the finger to move.

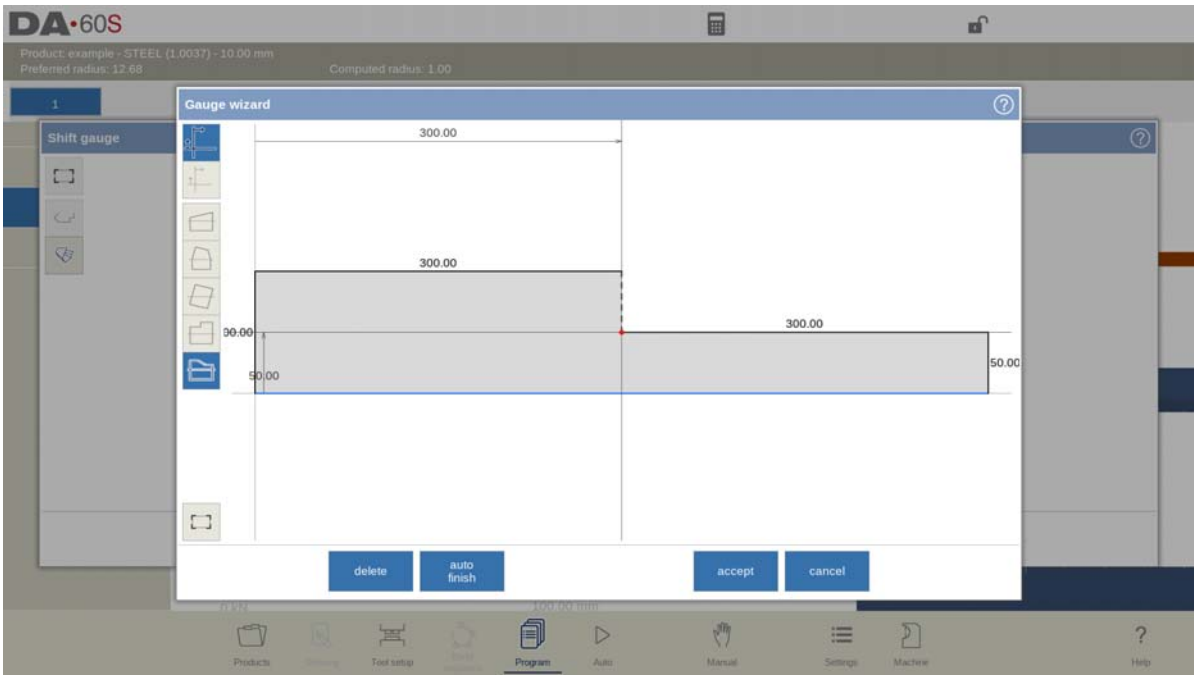
Pinch corner
Position the finger around the corner of the contact surface to fixate the product both in X-direction and Z-direction; this is only possible on a machine with special fingers that have the pinching possibility. Also the control should be equipped with this option.



Gauge wizard
For numerical (non-graphical) programs the (optional) Gauge Wizard enables the user to set the desired product side shape using this as a help for repositioning of the fingers; this is only possible in case the ‘X angle programming’ option has been activated. In this way one can program different X-axes positions following a stepped shape, or when setting a desired diagonal shape of the flange one can program X-axes positions according to this line.



The gauge wizard offers 4 different standard shape templates which can be altered upon choice, to meet the actual shape of the numerical programmed product.



The 5th choice is not a template, but enables free drawing of the product side to use this for finding the best gauge positions.

Change Side
Alters through the available backgauge positions.

Lay On Finger
Alters through the available product lay on positions. This option is only selectable in case you have an R-axis in your machine.

Shift Left
Shift the selected finger 1 millimeter to the left.

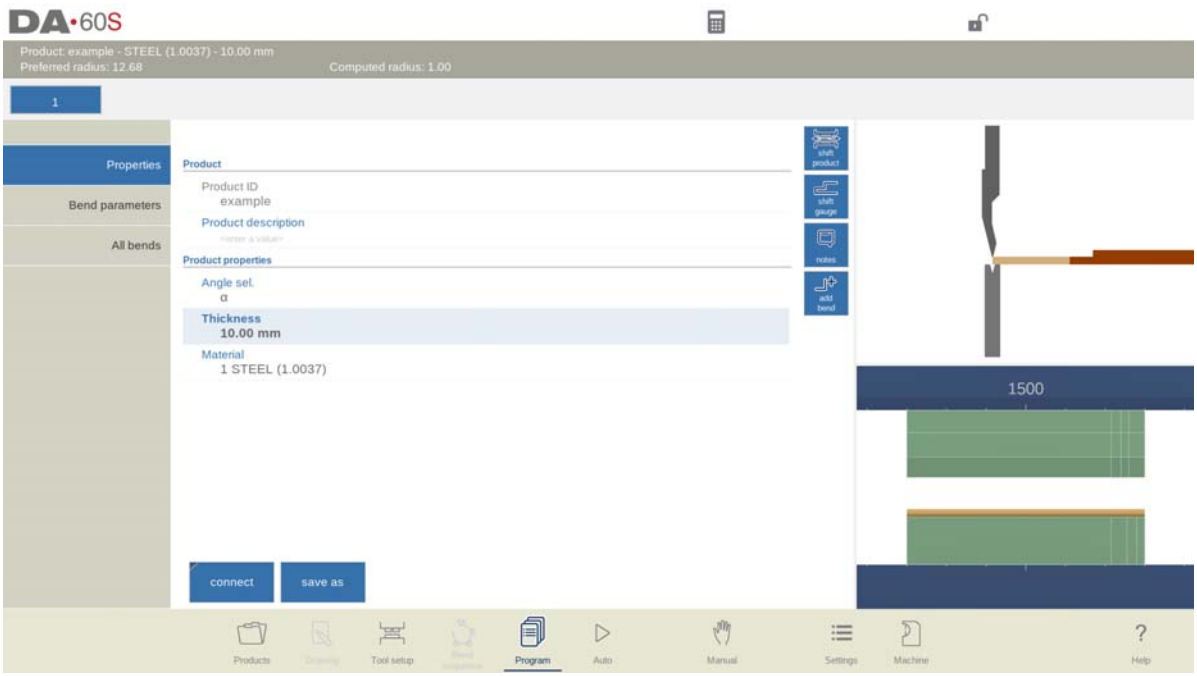
Shift Right
Shift the selected finger 1 millimeter to the right.

Cancel
Leave the current screen without saving changes.

Accept
Save the changes and leave the current screen.

6.3.5. Product properties

To change the main product properties tap 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 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.

	Tensile strength (N/mm²)	E-module (N/mm²)	Strain hardening exponent
1 = Steel 2 = Aluminium 3 = Zinc 4 = Stainless steel	470 250 200 750	210.000 210.000 94.000 210.000	0.23 0.26 0.20 0.32

Blank length

The required length of the original sheet from which the product is bent. This value is computed from a 2D drawing.

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.

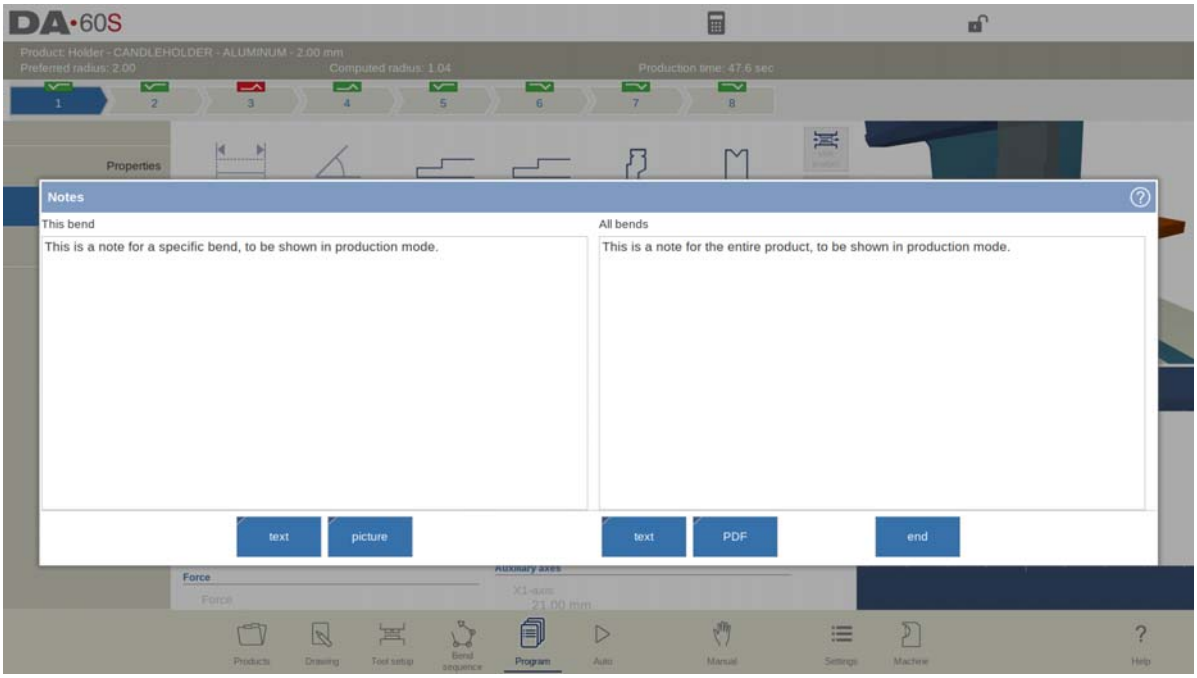
Save as
Copy the current product. When pressed, you must enter a new product ID for the copy program; you can also choose another location (directory) on the local disk to store the current product.

Remove drawing
Remove the graphical drawing from the product.

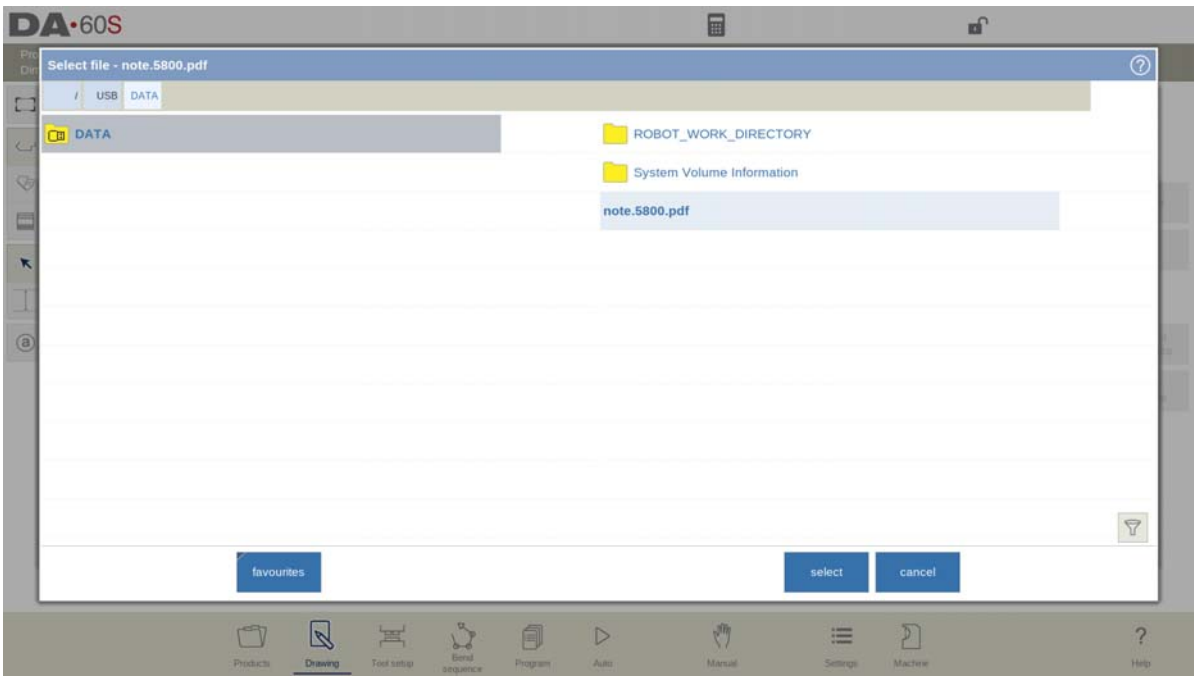
Edit notes
Opens a window enabling viewing and editing of notes about the current product. See more on Edit Notes further in this paragraph.

6.3.6. Notes

It is possible to add a note to your product, in order to store comment or background information about the current product. It is also possible to add a note to each bend. The note is a simple text field, it has no influence on product values or bend sequence calculations.



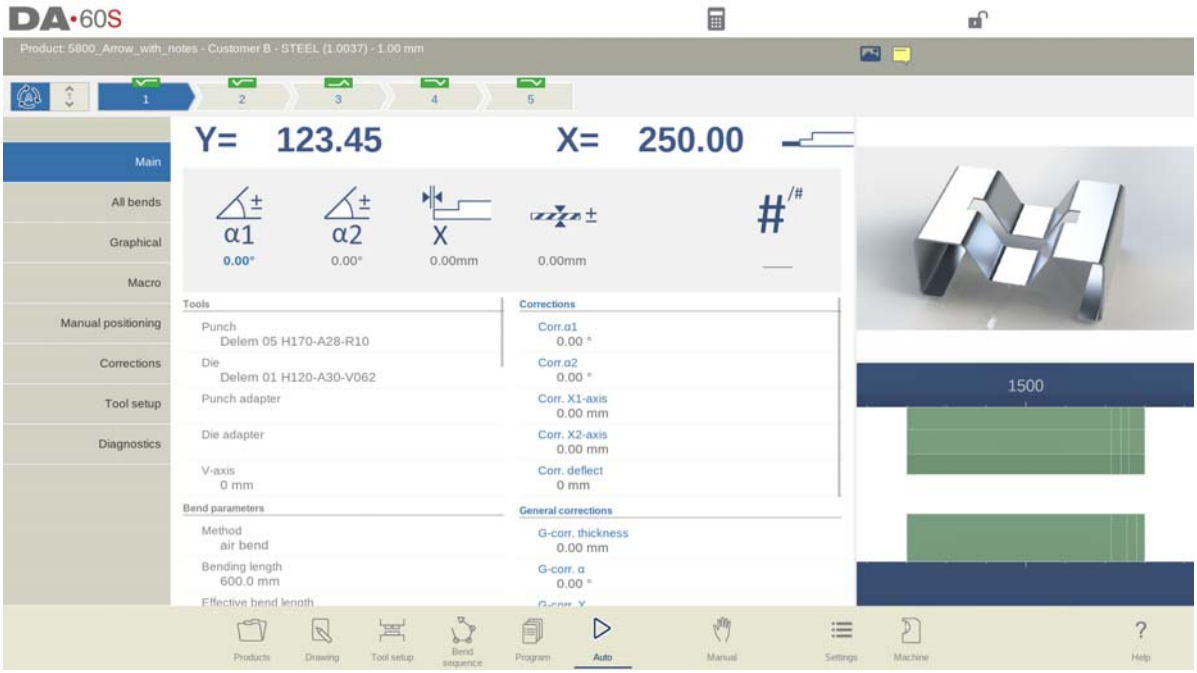
PDF Note
To attach a PDF file to the note tap Attach PDF. Via the directory browser a PDF file can be selected and will be incorporated in the product file.



When solely a PDF file is attached without a textual note, the PDF will be shown instantaneously when the user presses the Notes indicator in Automatic mode.

Photo Notes

To attach a photo to a specific bend tap Add Picture. Via the directory browser a PNG file or JPG file can be selected and will be incorporated in the product file.
When a photo is attached to a specific bend, it can be shown while processing this bend.



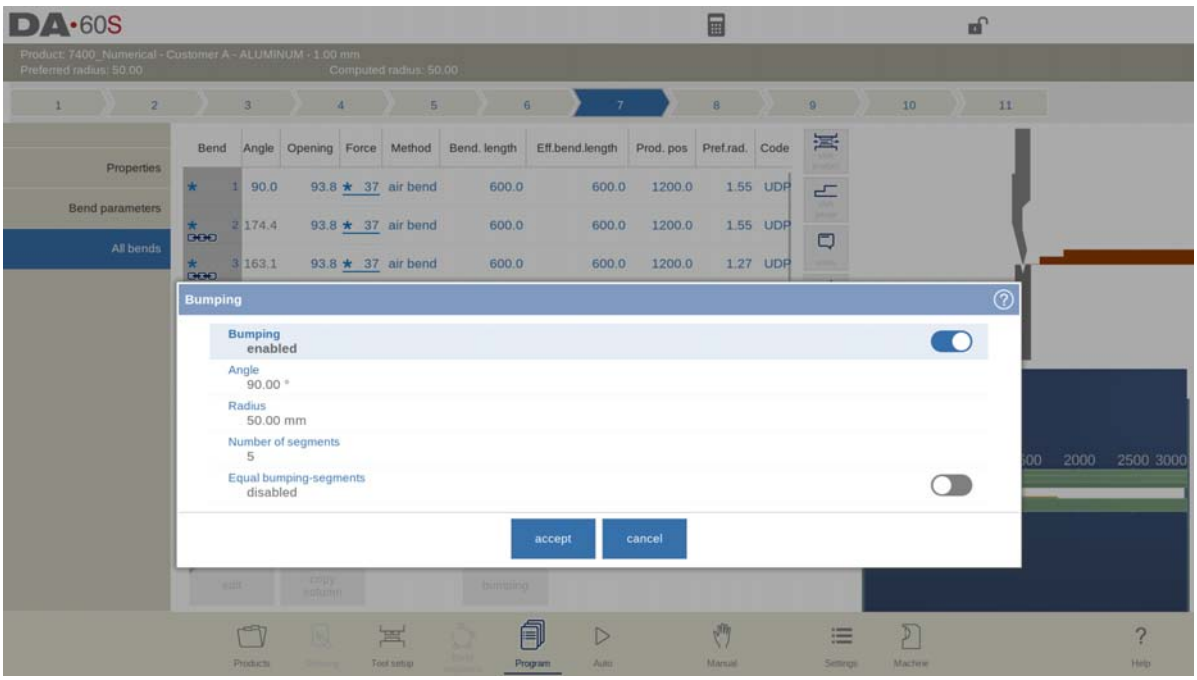
An additional graphical visualization mode enables the full screen visualization of the picture in Automatic mode. A specific photo can help the operator to understand specific product properties or remind to pay attention to something in particular.

6.3.7. Add bend

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

6.3.8. Bumping

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



When Bumping has been selected in the All Bends 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 air bending.

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)

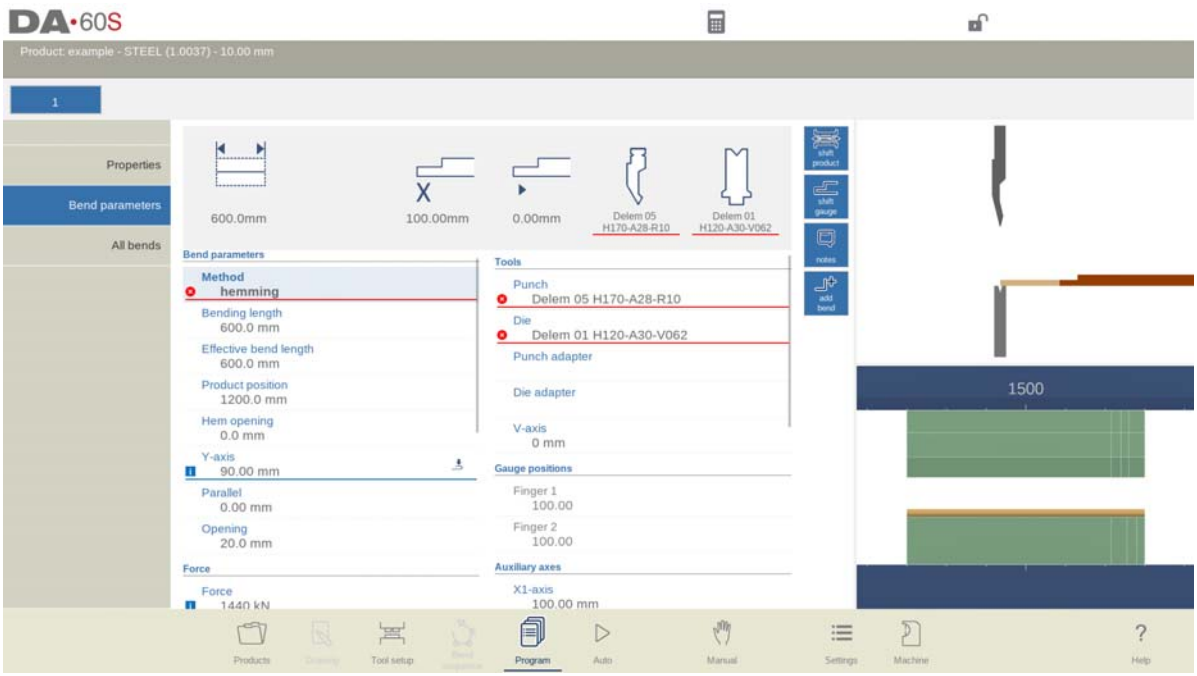
- Enabled (equal sizes)

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.

6.4. Programming parameters

Parameters in program mode can be programmed one by one. The relation between parameters is visualized with a symbol and a background color.



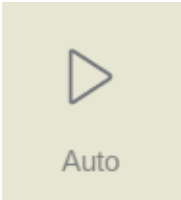
When an information symbol is shown with parameters after an edited value, this parameter was changed due to the last changed input.

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.

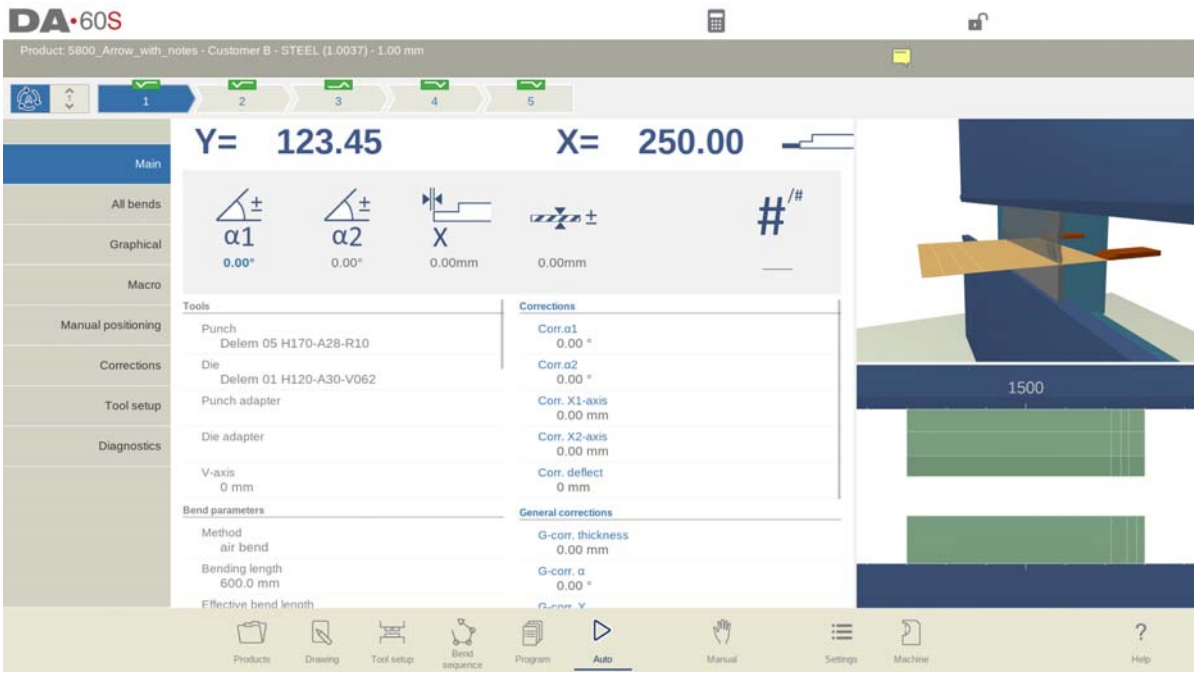
7. Automatic mode

7.1. 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.

Above the bend parameters, large icons (tiles) with related values are showing the most used ones. These tiles can be selected, giving direct access for modification. The values can also be found amongst all other automatic mode parameters.

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.

7.1.1. 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 millimeters.
Measured value of 202 millimeters.
-> Then it is required to program a correction of -2.
Programmed value of 200 millimeters.
Measured value of 198 millimeters.
-> Then it is required to program a correction of +2.

deflect

Correction on the crowning device.
Only available if a crowning device is present.

PST

Correction on the part support axes. The correction will be gradually introduced during the bend, starting from the pinching point. At the final bend position, the added correction will be equal to the programmed value.
Only available if a part support 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, valid for each bend of the program.
Only available if a crowning device is present.

PST

General correction on the part support axes, valid for each bend of the program. The value should be programmed in the same manner as for the correction per bend.
Only available if a part support 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.

7.2. View modes

The auto mode screen is offering a diversity of views which, depending on ones production method, 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.

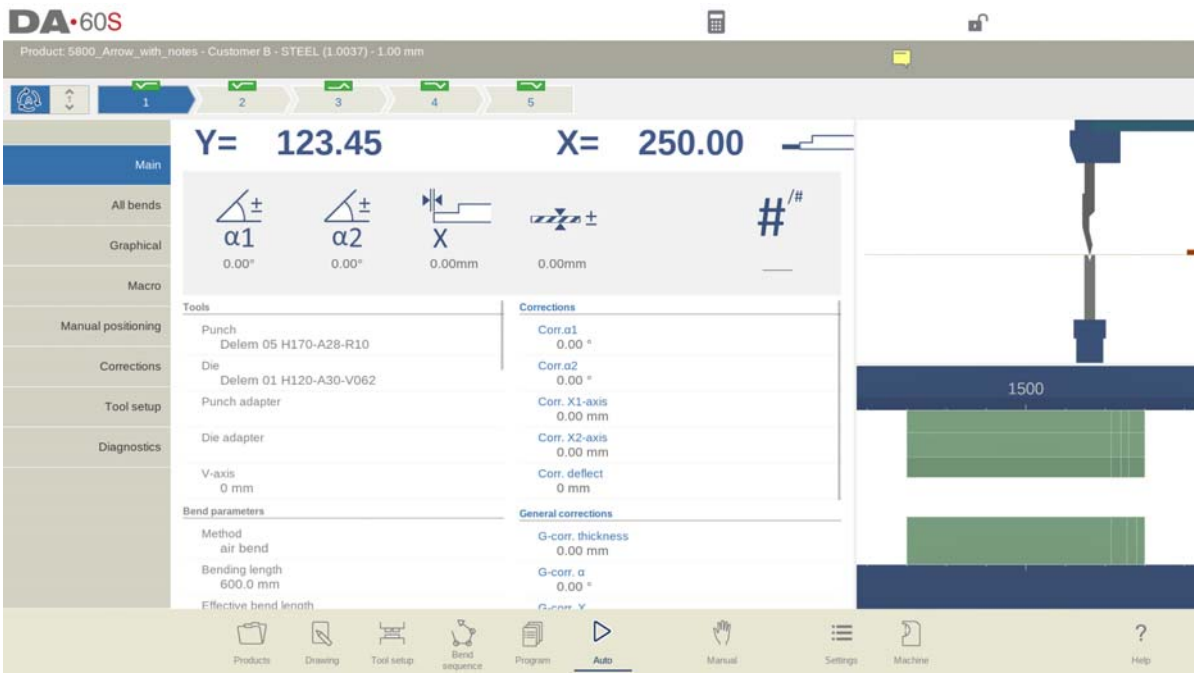
Following modes are available:

Main	Main, numerical bend data as well as graphical information simultaneously shown (if available). See for details paragraph 7.2.1
All bends	All bends, shows all bend data in a table upon choice simultaneously with graphical information of the selected bend (if available). See for details paragraph 7.2.2
Graphical	Graphical, full screen graphical visualization of the selected bend. See for details paragraph 7.2.3
Macro	Macro, large visualization of axes values shown in a list upon choice simultaneously shown with graphical information of the selected bend (if available). See for details paragraph 7.2.4
Manual positioning	Manual positioning, large visualization of axes values with the possibility to move axes from their position and teach the axis value in the selected bend. See for details paragraph 7.2.5
Corrections	Corrections, all correction values for angle as well as axes positions. See for details paragraph 7.2.6
Tool setup	Tool setup, visualization of the tool setup required for the selected program. See for details paragraph 7.2.7
Diagnostics	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 7.2.8

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.

7.2.1. Main

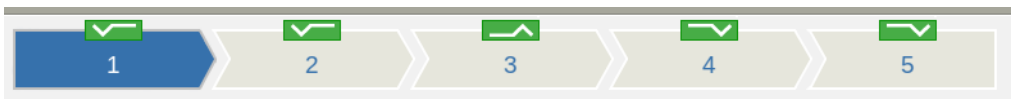
Main view shows the numerical data of the bend along with the corrections. The corrections can be programmed here. The splitter control divides the screen in graphical visualization and the numerical data. This can be closed if only numerical data is required.



Both columns can be scrolled to see all data. In the graphical view quadrant zoom, pan and rotate can be used to view the bend position in the machine.
The tool view quadrant zooms automatically in on the used tools. During the bending proces the tool station to be used will be indicated to help the user to select the correct station.

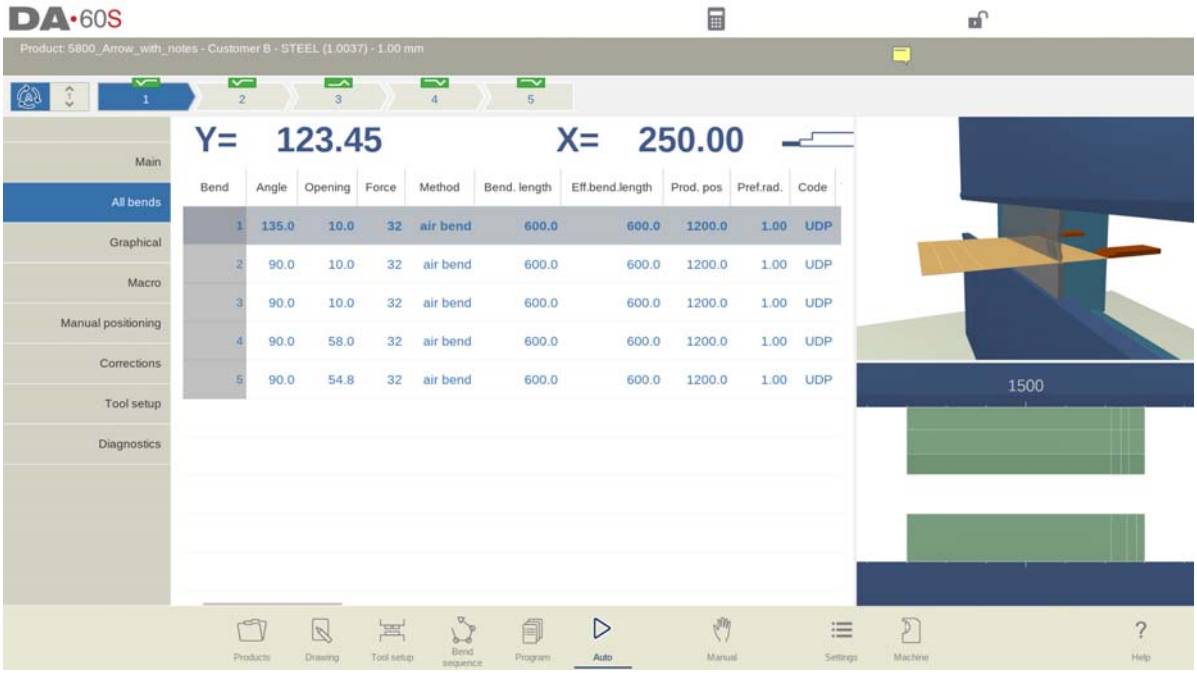
Bend selector

The bend selector in the top of the screen can be used to navigate through the bends. The indicated bends can be tapped to easily select the desired bend data. In the bend selector the turn indicator will be displayed showing green, yellow or red colors to indicate the level of complying the assignments of the bend sequence.



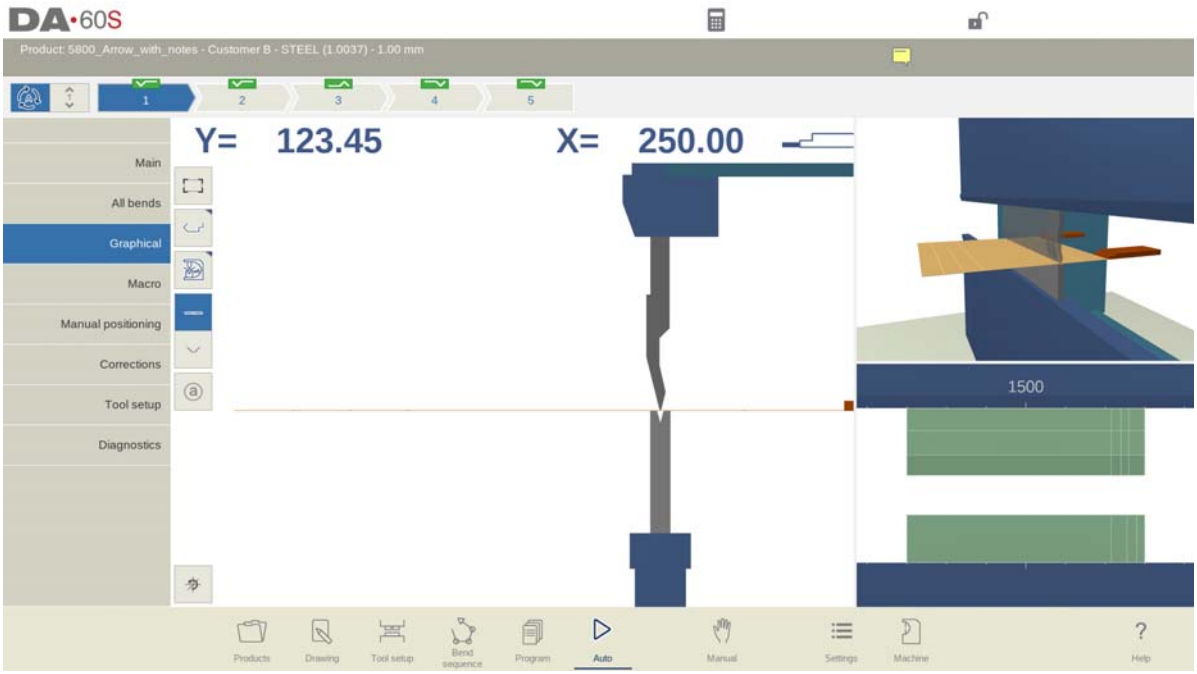
7.2.2. All bends

The all bends view mode shows, with or without the opened graphical pane, a table including all bend data. The bends are shown row by row, and the columns display all bend parameters.

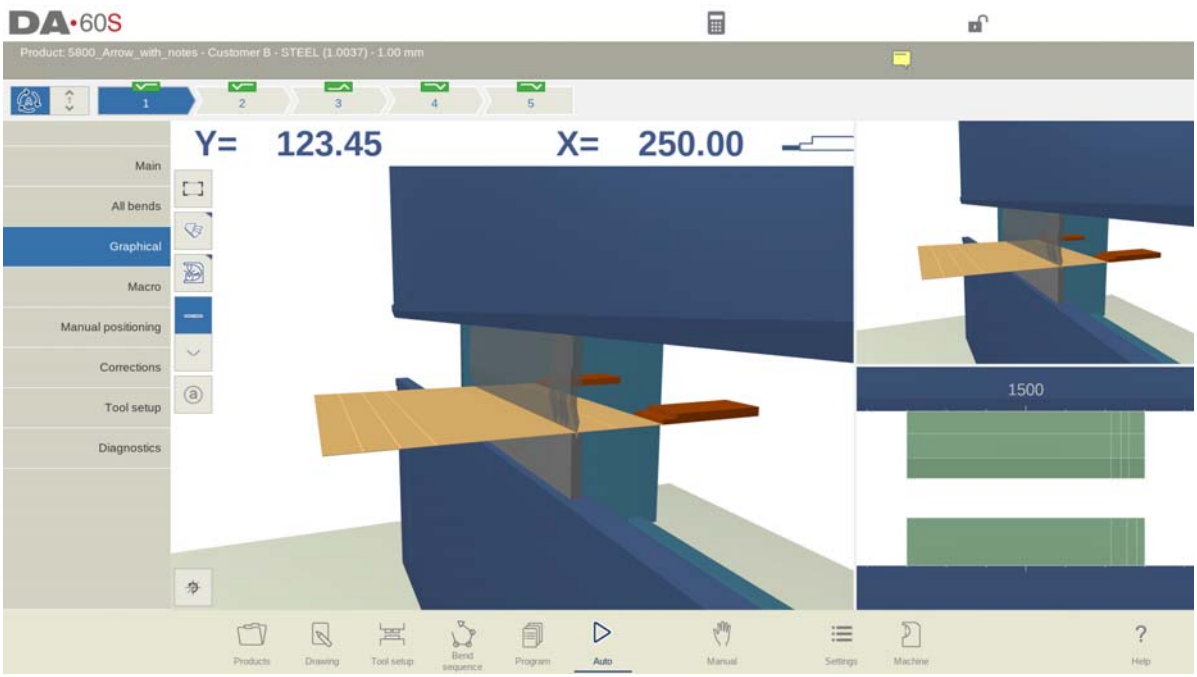


7.2.3. Graphical

In graphical view mode a full screen graphical view of the bend process is given.

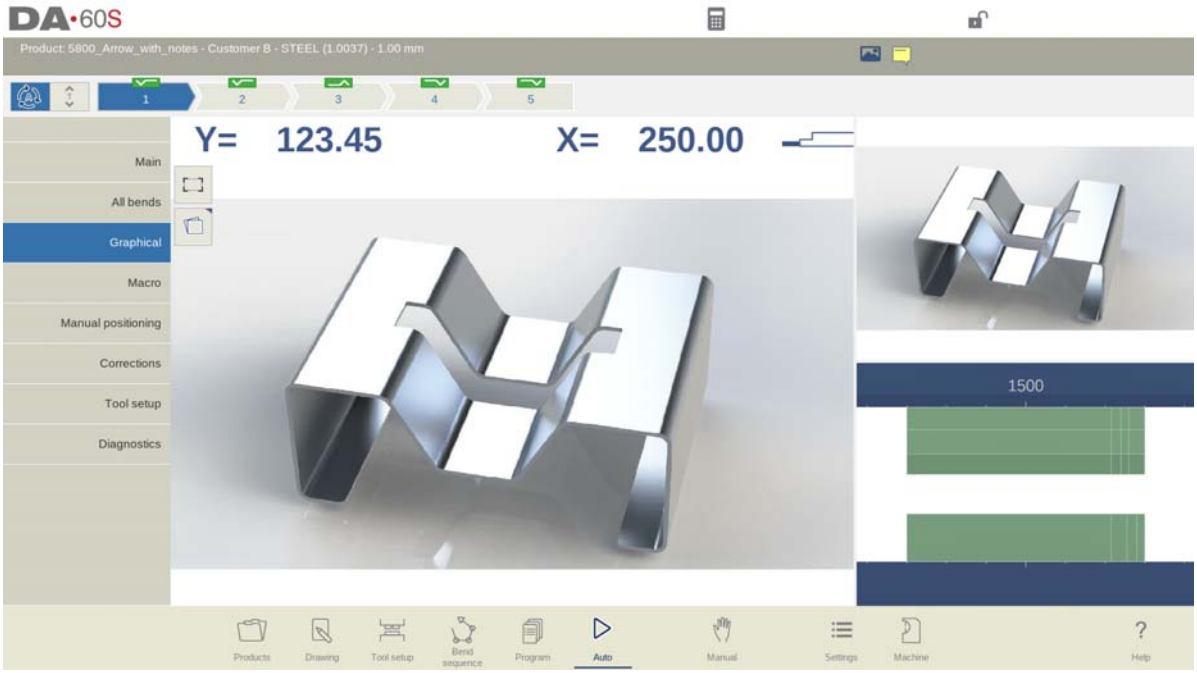


Optionally the view can be switched to 3D to have the product and machine shown in 3D. This can be toggled.



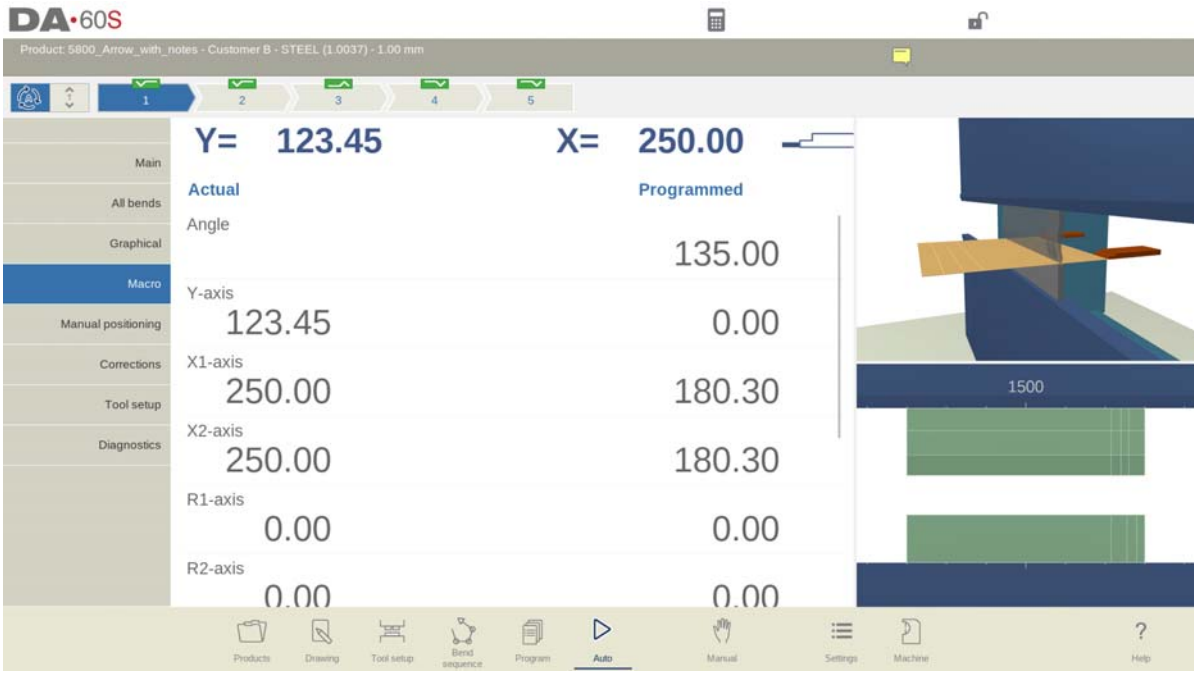
Changes to the view angle and zoom level will also be used in the smaller graphical visualizations in other view modes.

Optionally the view can be switched to photo view to have an included photo visualised at a specific bend step.



7.2.4. 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.



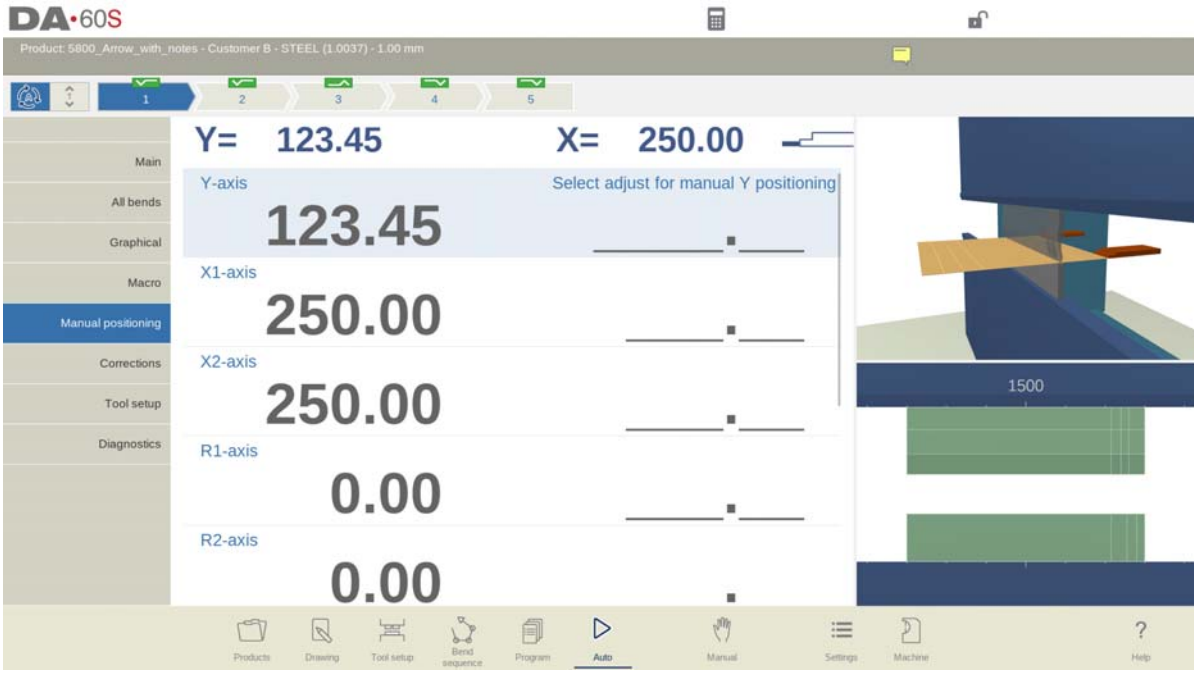
Next to the target position (programmed) also the actual position of all axes can be followed.

7.2.5. 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 turning the hand wheel.

The teach indicator:

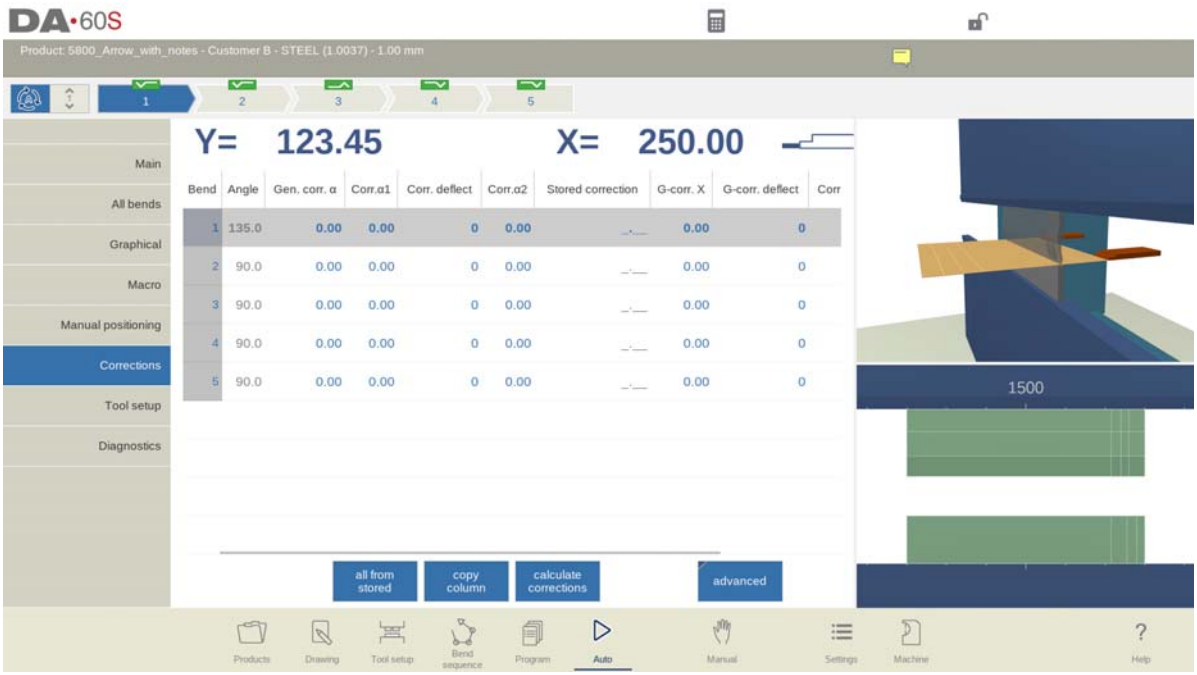
When the teach indicator arrow is pressed, standing in between actual value and programmed value, the value is taught to the program step.



External hand terminal (optional)
The manual positioning functionality can be expanded by using an external hand terminal. This hand terminal allows the operator to remotely control the hand wheel functionality, including axes selection and bend selection, and teach found axes positions to the program.

7.2.6. 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 $\alpha 1$ is entered then this value is, dependent on the settings parameter 'Angle correction programming', copied to the correction for $\alpha 2$, or keeping the delta between both corrections, or not influencing the correction for $\alpha 2$. Different corrections for $\alpha 2$ can be entered in the field itself.

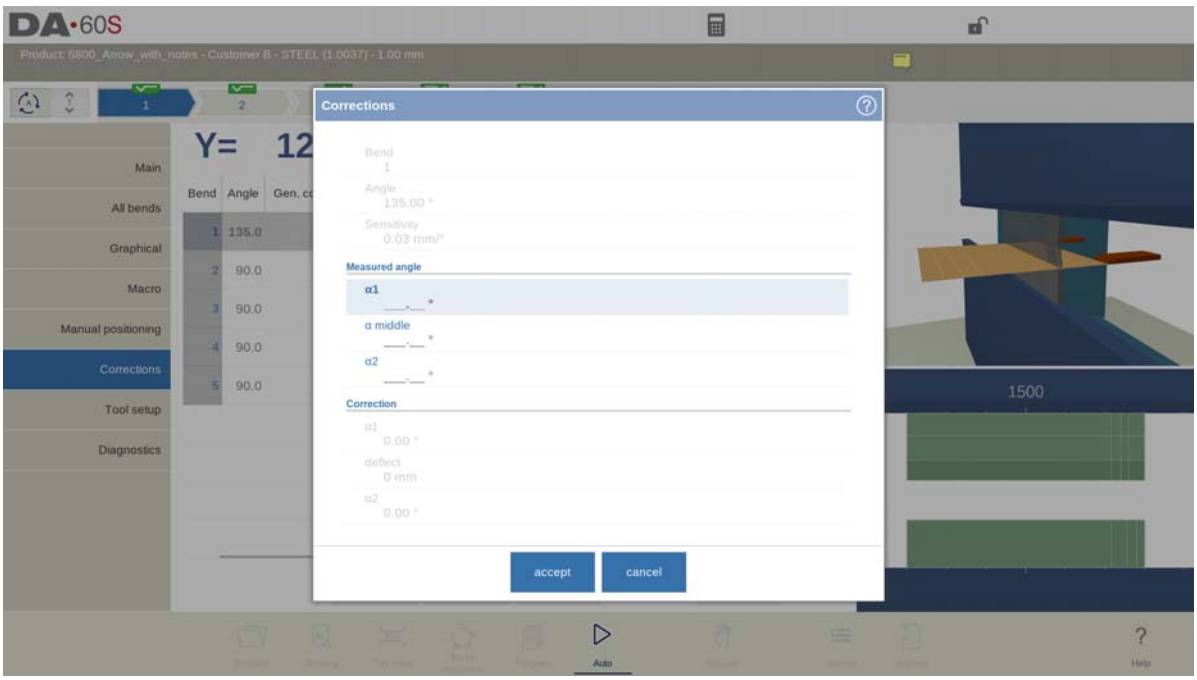


The column 'Stored correction' is only available when the Angle correction database has been activated. When activated, the column 'Stored correction' shows for each bend the correction value that is present in the database. A blank entry in this column means the database does not have a correction value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

Bend Allowance
The function Bend Allowance helps the user to add entries to the bend allowance table in case this is active. Based on the active bend parameters only the bend allowance corrections need to be entered before adding. When the measured value is given, this will result in a calculated bend allowance, derived from the difference between the programmed value and the measured value. To enable the bend allowance table go to the Settings mode.

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.



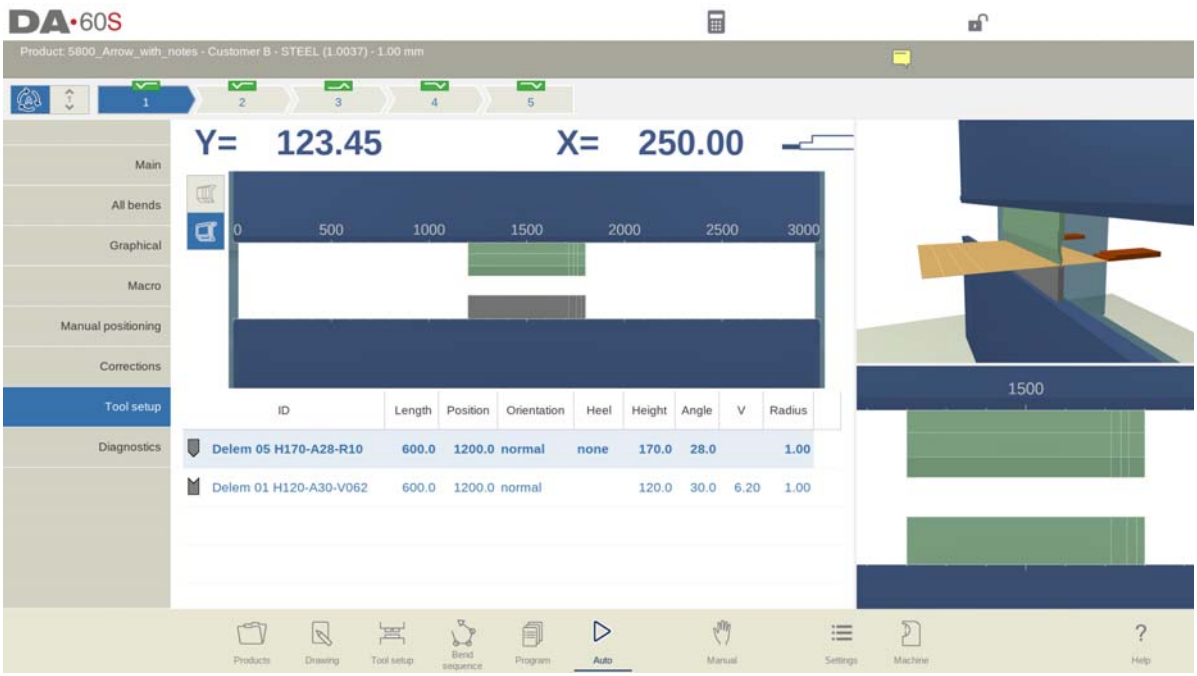
From the programmed value the control will determine a correction. The proposed result can be seen in the window itself. In the top of the window the programmed angle is shown, in the bottom of the window the resulting corrections are shown. When selecting accept, these values will be transferred to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

When using the optional protractor device, measured values can be taken directly into the correction field or can also been obtained within the “calculate corrections” window. After a satisfactory measurement, corrections are transferred to the main corrections window. If values are entered directly in the correction field, new corrections are only accumulated after a new bend cycle.

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.

7.2.7. Tool setup

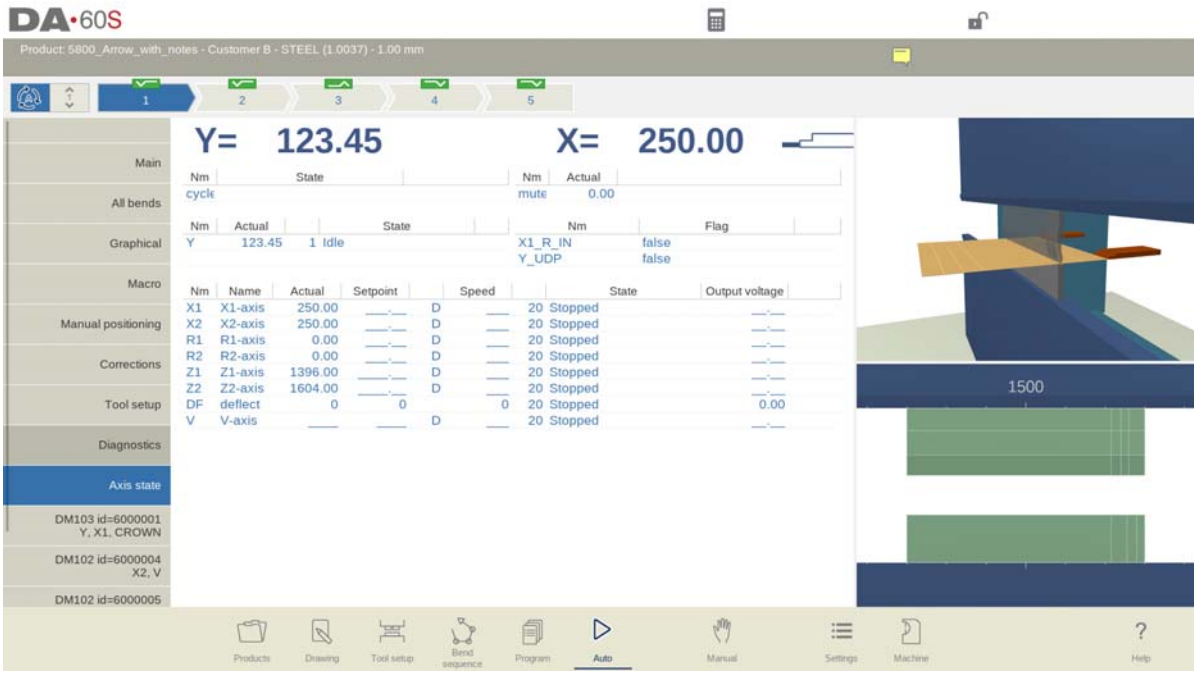
Tool setup shows the tool configuration which is required for the selected product. In the tool setup view mode also tool properties, next to their required position, can be checked.



In this tool setup view mode no changes can be made to the actual setup. If this is required one has to switch to the Tool Setup menu, outside Auto mode.

7.2.8. Diagnostics

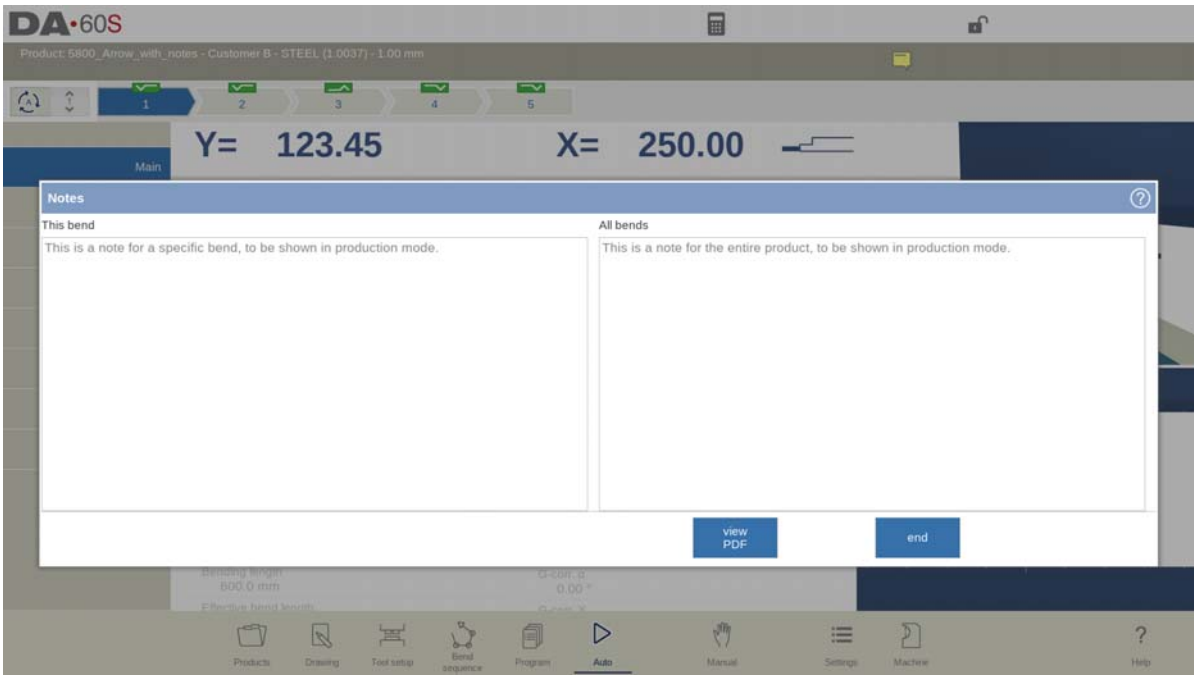
The diagnostics view mode is meant for service purpose mostly. In diagnostics the activities of independent axes can be monitored. I/O on the control system can be followed. In rare situations this information can be helpful to diagnose operation during the bending process.

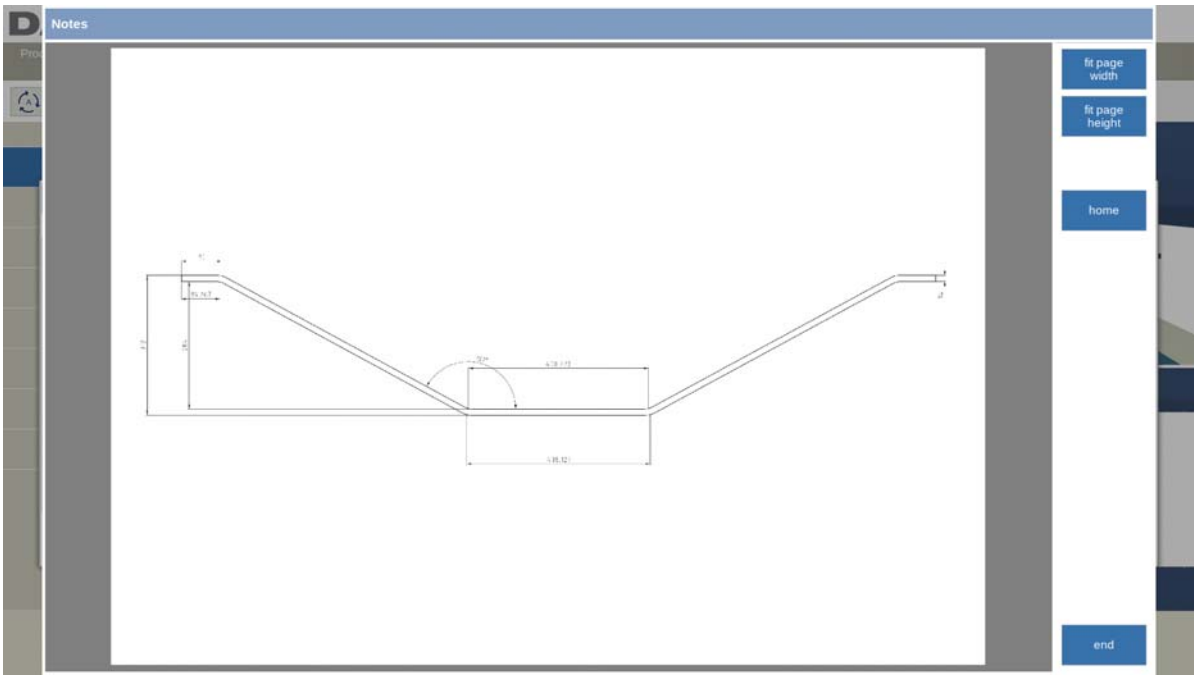


7.3. Notes

The notes which can be added to a product or program can be viewed in Automatic mode. With the presence of the notes indicator it's indicated that notes have been added to this product, and by tapping the indicator these will be shown.

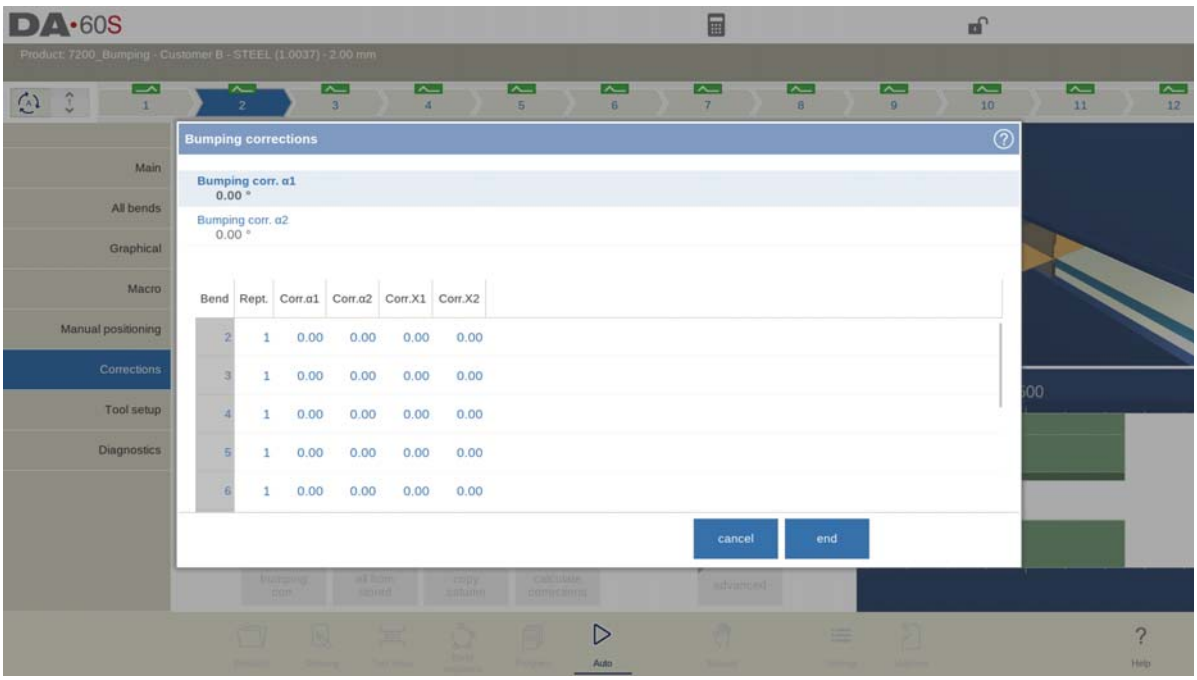
Notes can be added generally to a product or program but also to specific bends. Within the notes also PDF documents can have been included. The PDF button will open the document.





7.4. Bumping/chaining correction

In case of a selected bumping/chaining bend a general correction for a bumping/chaining bend can be entered. This function is only available if a product is loaded that contains a bumping/chaining bend.
With Bumping Corr. or Chaining 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/chaining corrections can be programmed independently for both sides, $\alpha 1$ and $\alpha 2$.
If a bumping/chaining correction for $\alpha 1$ is entered then this value is, dependent on the settings parameter 'Angle correction programming', copied to the bumping/chaining correction for $\alpha 2$, or keeping the delta between both bumping/chaining corrections, or not influencing the bumping/chaining correction for $\alpha 2$. Subsequently all separate corrections for $\alpha 2$ are recalculated. To change correction values of $\alpha 2$, use bumping/chaining correction $\alpha 2$ or one of the separate corrections of $\alpha 2$.

7.5. Test Bend mode

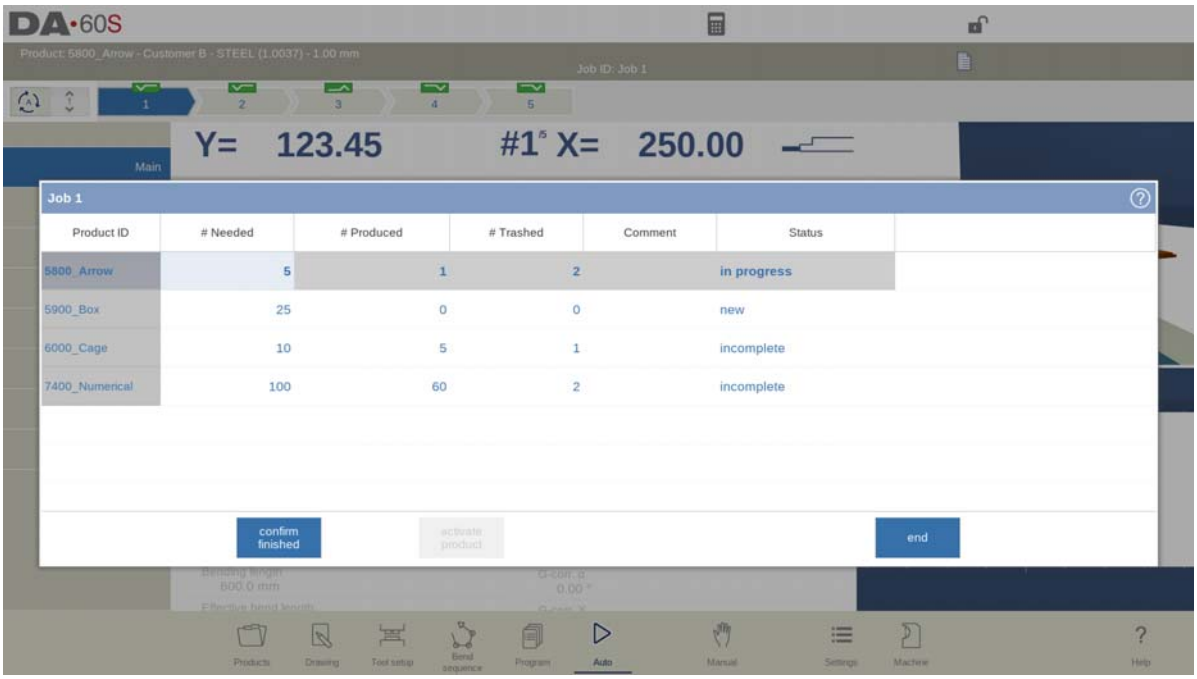
Test Bend mode enables the operator to tune the current bend.

When switched on, the axes will stay at their retract position after the first bend cycle, the part support (if present, and enabled) will remain at its angle at ER, and there will be no step change. After the Test Bend is finished the Y-axis will be at UDP, and the control is stopped. After measuring the angle a correction can be programmed and the current bend can be re-bent. The part support (if present, and enabled) will start following again as soon as the Y-axis passes the (original) position, at which the part support should start following again.

7.6. Shop floor control, Job List function

When the control is in a Job production process, this is indicated with the Job indicator.

Jobs can be viewed and managed by tapping on this Job indicator.



When a different batch of products needs to be produced from the active job list, the function key Activate product will make the job system switch to the selected product from the active job list.

When producing parts, the operator can update the number of trashed products in order to report the required amount of unfolded products. The number produced will be adjusted automatically but can be overwritten upon desire. In the comment field both the operator and the job manager can add comment.

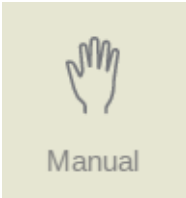
Job status indication:

- New: no products of this batch have been produced yet;
- Incomplete: the desired number of products of this batch has not been produced yet;
- In progress: this batch of products is currently loaded on the machine and in production,
- Finished: this batch of products has been confirmed as finished.

The job will remain active until another product or job is selected in Products mode. When a new product is loaded, the job will be inactivated and needs to be selected and started again if required.

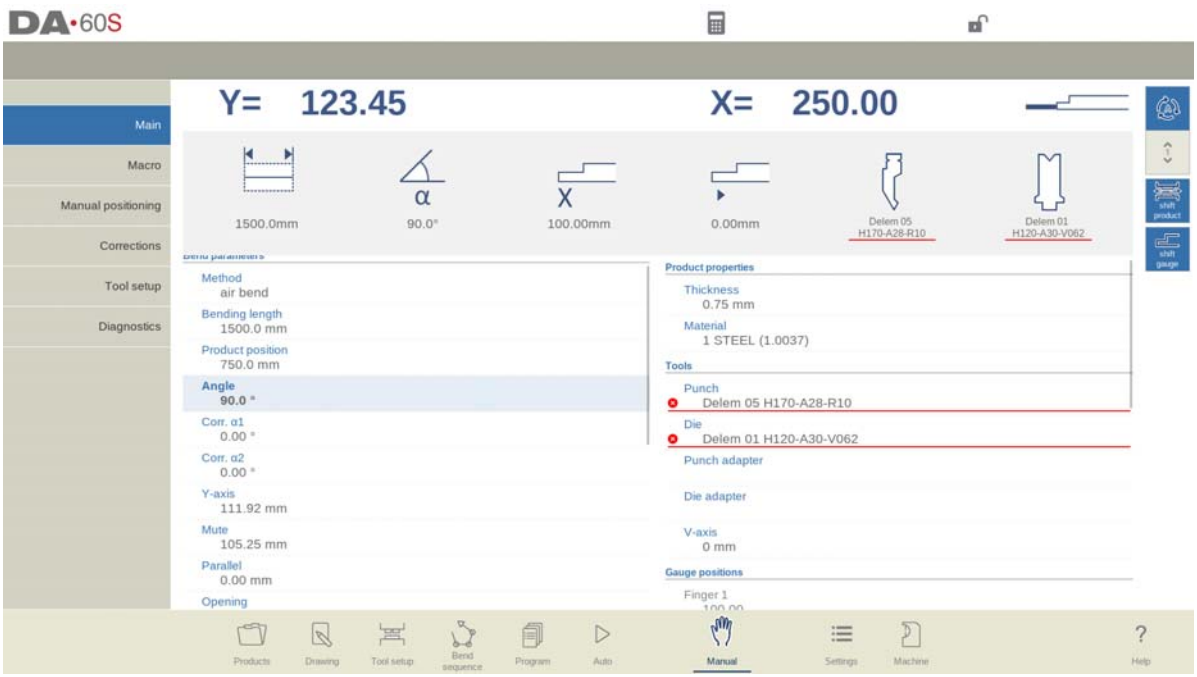
8. Manual mode

8.1. Introduction



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

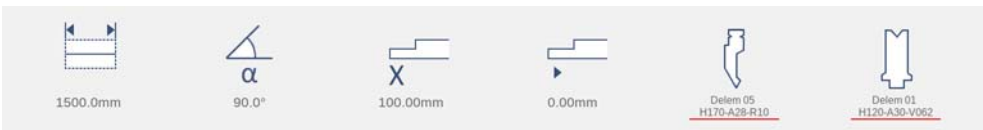
In manual mode you program the parameters for one bending. This mode is useful for testing, for calibration and for single bends.



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.

Above the axes and functions, large icons (tiles) with related values are showing the most used ones. These tiles can be selected and the values can directly be modified.



8.1.1. Manual mode, parameter explanation

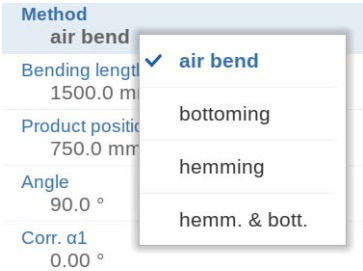
Following is a list of the available parameters in Manual mode.

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.

Deflect method

dynamic =>The crowning will be controlled automatically in real-time during the bending, applying the proper corrections when needed.
non-dynamic =>The crowning will behave as a standard hydraulic crowning; there won't be real-time corrections.

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.

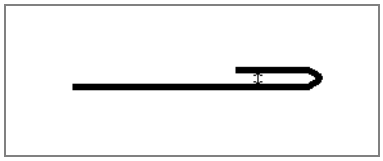
Corr.α 1, Corr.α 2

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.



Fast hemming

When fast hemming has been enabled, the Y-axis will move at high speed down as soon as it is below the top of the die, until the hemming load opening has been reached. This will reduce the cycle time, especially in case of hemming with a U-die.
Only available when enabled by the machine manufacturer.

Corr.Y

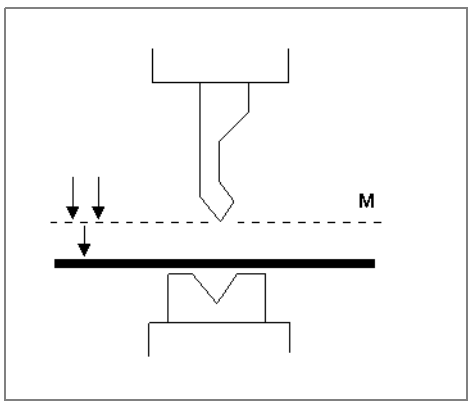
Correction on the Y-axis position, in case bottoming has been selected.

Y-axis

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

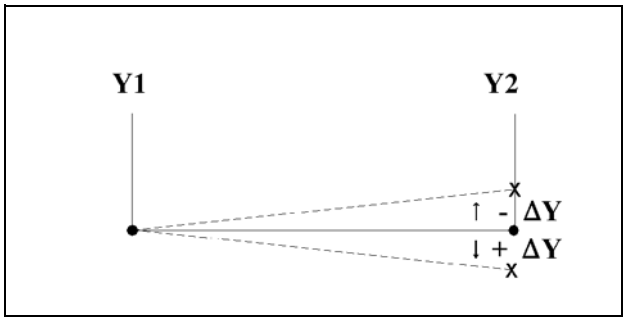
Mute

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.
The decompression distance can optionally be set to a specific point in the cycle. Standard the decompression distance is computed, the value which is minimal required. Optionally one can chose for Mute, the calculated mute point, or Clamp, the calculated clamping point. Both options have a longer distance than the calculated one.

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.

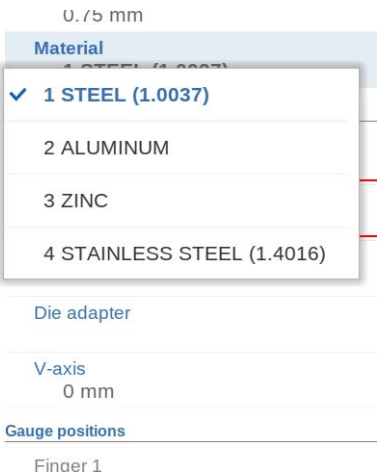
Product properties

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.

Punch adapter

The name (ID) of the selected punch adapter. Tap to modify or select from the punch adapter library. Whether an adapter can be programmed depends on the parameter Use punch adapter in the Machine mode.

Die adapter

The name (ID) of the selected die adapter. Tap to modify or select from the die adapter library. Whether an adapter can be programmed depends on the parameter Use die

adapter in the Machine mode.

Gauge positions

Finger (1/2/3/4)

The finger (contact) position, corresponding to the X-axis position and lay on position.

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. When you have a R1-axis and a R2- axis the programmed R1-axis value is automatically copied to the R2-axis value. The R2-axis value can, if necessary, be changed afterwards.

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.

Part support

PST-axis

With this parameter the part support can be switched on or off. When it is switched off, the part support will stay at its zero position during the bend.

R-position

The height of the part support before and after the bend. By default, the height will be set at the top of the die (R-position = 0.00 mm).
Only available when enabled by the machine manufacturer.

Method

With this parameter one can define how and when the part support should move back to its zero position after the bend has been completed. Upon choice one can select the part support to return when the Y-axis is at UDP (‘Return at UDP’) or to return already when the Y-axis is opening (‘Return when opening’). Other choices are ‘Product tilt‘, which will enable the use of an extra tilt to avoid collision between the product and the punch during the Y-axis opening movement, and ‘Static angle support’, which enables the user to bring the part support in a fixed angle; in that case the part support will not follow the product but act as a support table under an angle.

Tilt angle

The additional angle that the part support will move.
A part support at the front side of the machine will move to a higher position (bigger angle). A part support on the back side will move to a lower position (smaller angle).
Only available when method ‘Product tilt” has been selected.

Tilt speed

The speed to move to the tilt angle position; this is a percentage of the maximum operation speed.
Only available when method ‘Product tilt” has been selected.

Tilt clearance

In case the calculated (or programmed) decompression distance is not sufficient for the tilt movement, an additional distance (‘Tilt clearance’) can be programmed; this value is added to the programmed decompression distance. The part support will only follow during the movement of the programmed decompression distance, and will not follow anymore during the movement of the tilt clearance.
Only available when the method ‘Product tilt” has been selected.

Angle

The static angle to which the part support will move before the bend.
Only available when method ‘Static angle support” has been selected.

Speed

The speed, as a percentage of the maximum speed, with which the part support moves to the static angle.
Only available when method ‘Static angle support” has been selected.

Return speed

Return speed of the part support after a bend. The speed value is programmed as a percentage of the maximum speed.

Safety stop angle

The angle at which the support will stop during the return movement; the control will switch to stop. An extra start action at the control makes the part support return to its zero position.

Initial angle

The initial angle (typically the pre-bend angle of a two-stage bend) to which the part support will be moved at step change. The part support will follow as soon as the angle of the product passes the initial angle of the part support. Only valid for part support axes in front of the machine.

Initial angle speed

The speed to move the part support to the initial angle. Only valid for part support axes in front of the machine.

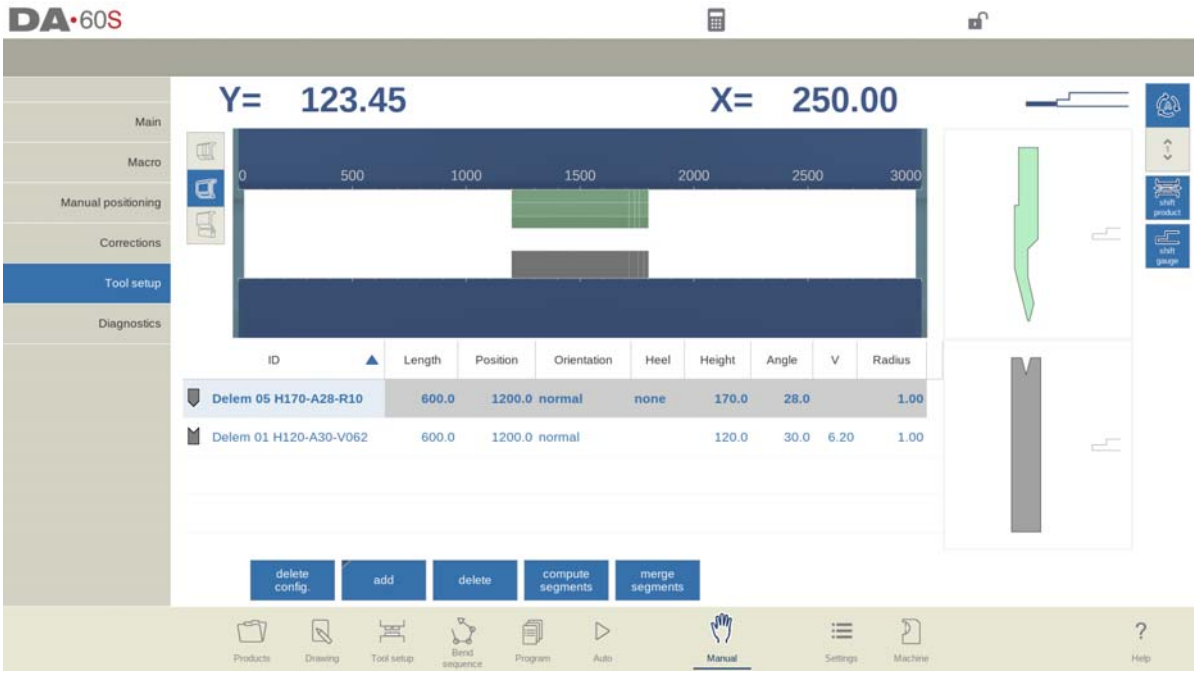
Correction

Correction on the part support axes. The correction will be gradually introduced during the bend, starting from the pinching point. At the final bend position, the added correction will be equal to the programmed value.

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

8.1.2. 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.

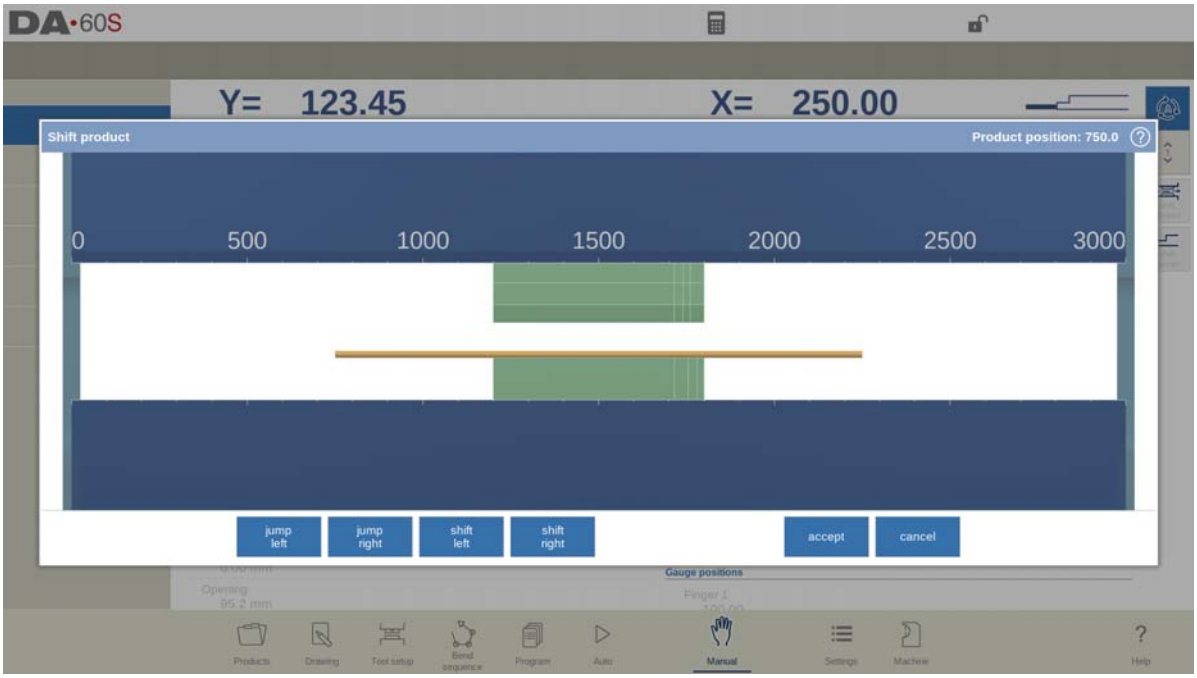


In the tool setup menu tools can be added, removed or repositioned, similar to the main Tool Setup mode as described in chapter 4.

Adding tools (Punches / Dies / Punch adapters / Die adapters)

Same as in Tool Setup, via the Add function tools and adapters can be added. If available, depending on the selected tool, also segmentation is possible in Manual mode.

8.1.3. Shift product



To shift the product's position the Shift Product function opens a dialog enabling the product to be shifted on and in between the tools. The product can even be shifted over tool stations. While shifting the user is helped with snapping points. The product's position can easily be moved to the side of tools or in the middle of the station.

Functions

Jump Left
Move the product to another tool set combination in the left direction.

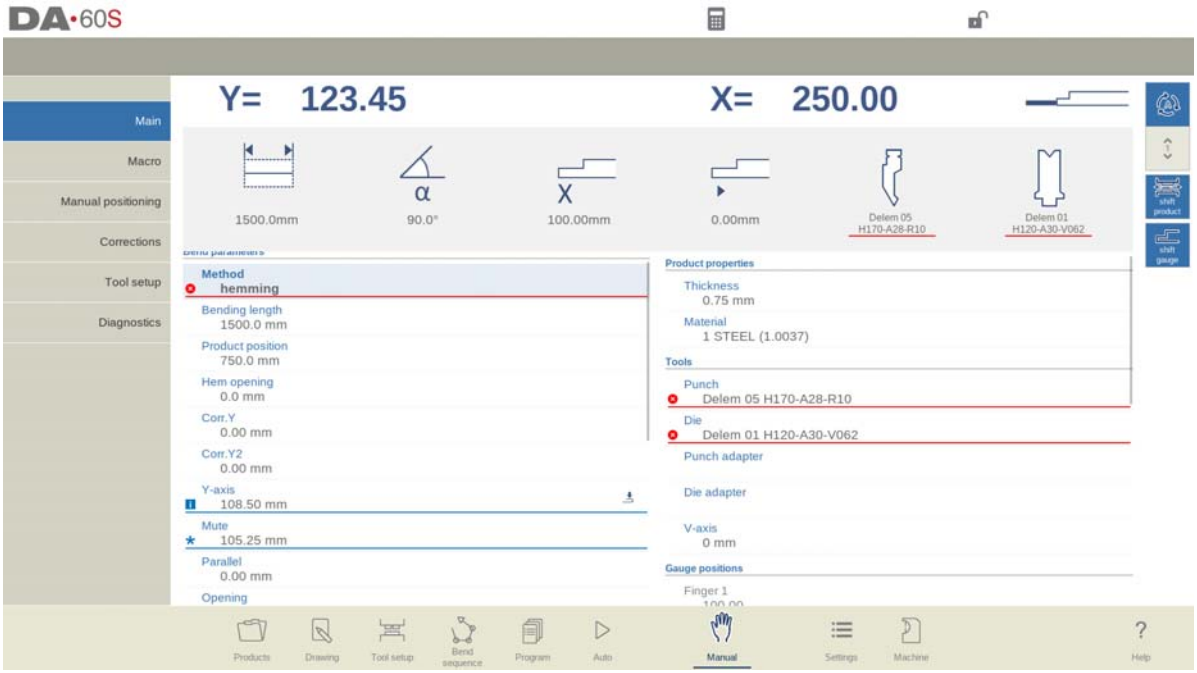
Jump Right
Move the product to another tool set combination in the right direction.

Shift Left
Shift the product 1 millimeter to the left within the same tool set.

Shift Right
Shift the product 1 millimeter to the right within the same tool set.

8.2. Programming parameters & Views

Parameters in manual mode can be programmed one by one. 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.

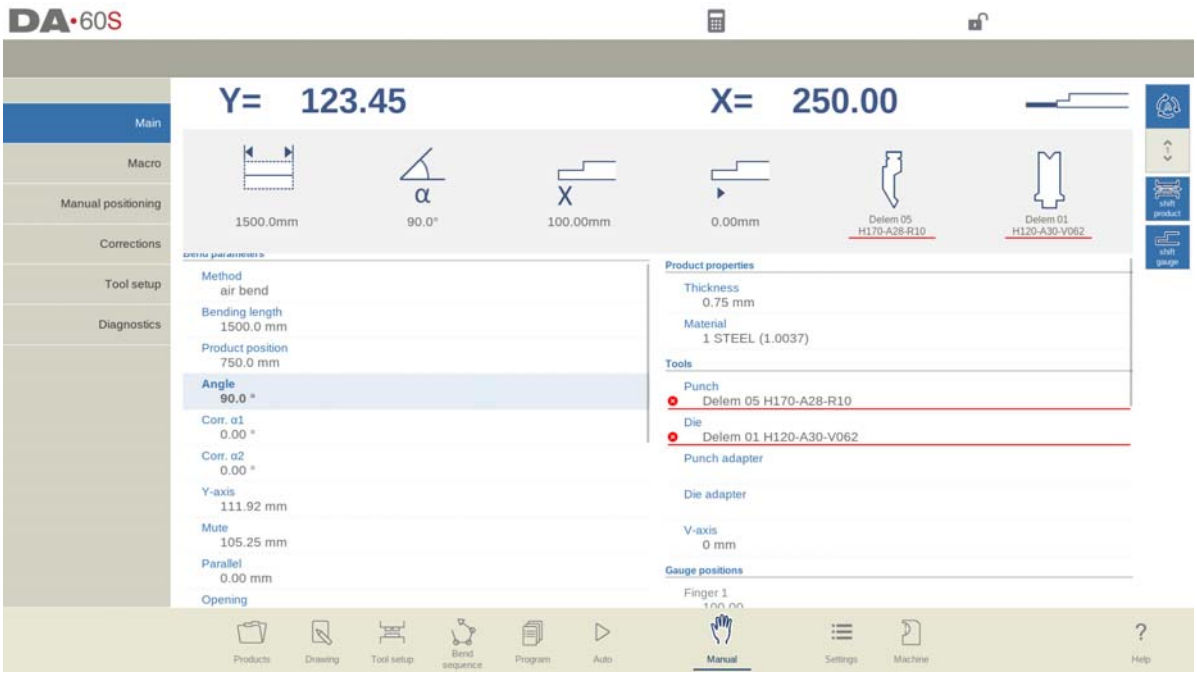
A star symbol is shown with parameters if the value of the parameter differs from the calculated value by the control. This can be helpful if a value is intentionally programmed different or if the value of a parameter is limited by the parameters limits.

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

View

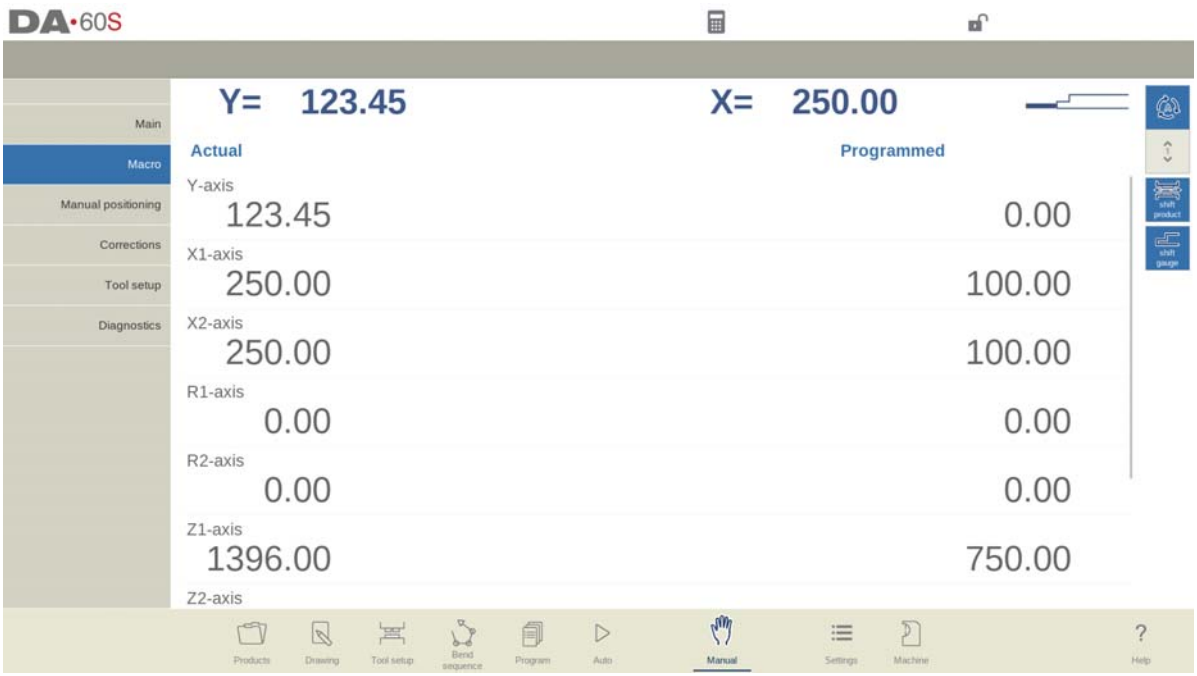
The command buttons on the right side of the screen give access to other views. Next to the

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



8.3. Macro

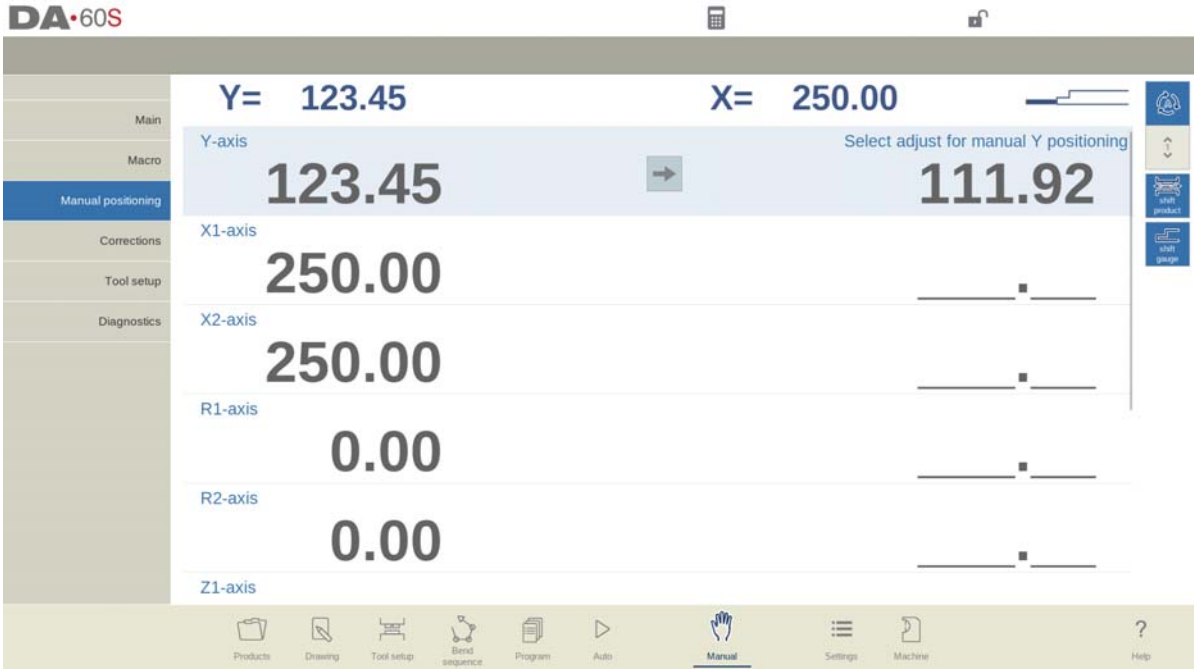
With Macro the control switches to a new view with only large axes values on the screen. This view can be used when working a little remote from the control, still able to read the axes values.



8.4. Manual movement of the axes

8.4.1. Movement procedure

To move an axis to a specific position manually, the hand wheel on the front panel of the control can be used. After tapping Manual positioning in the main screen of Manual Mode, the following screen appears:



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

Auxiliary axes

The control must be stopped (the LED on the Stop button is on).
First select the desired backgauge axis, you will see the cursor at the required axis.
Then you can move the axis by turning the wheel.

Y-axis

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

- The control must be started (the LED on the Start button is on).
- The 'adjust' function must be active. 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.

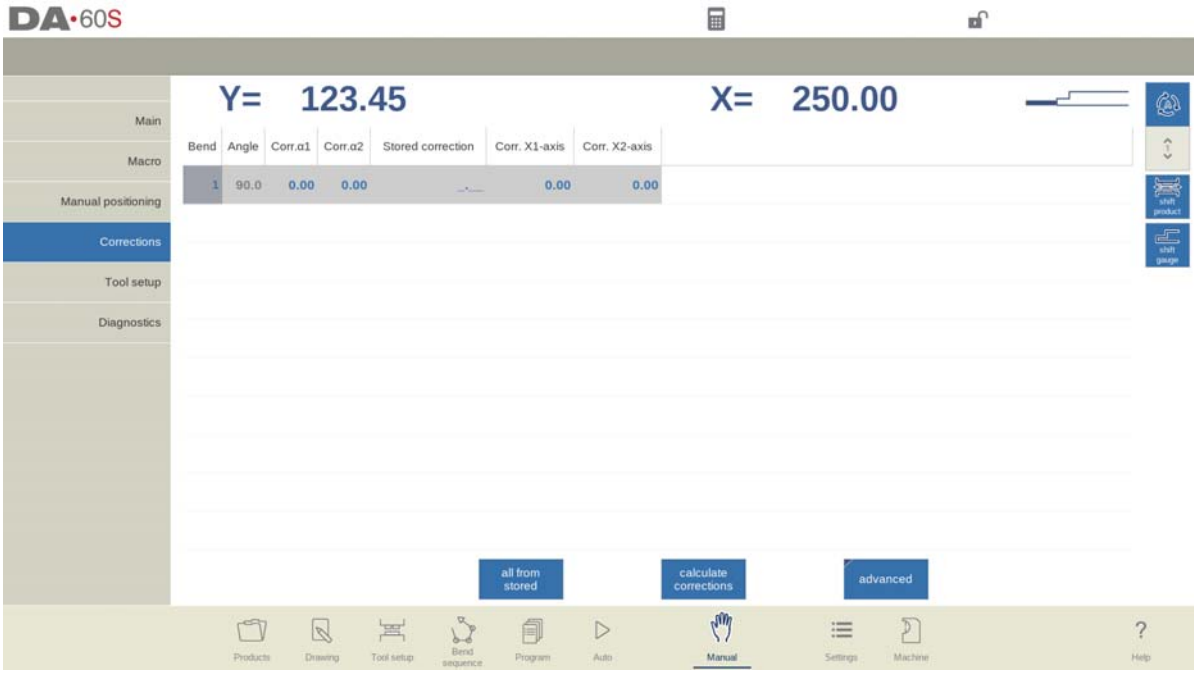
8.4.2. Teach

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 hand wheel, 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).

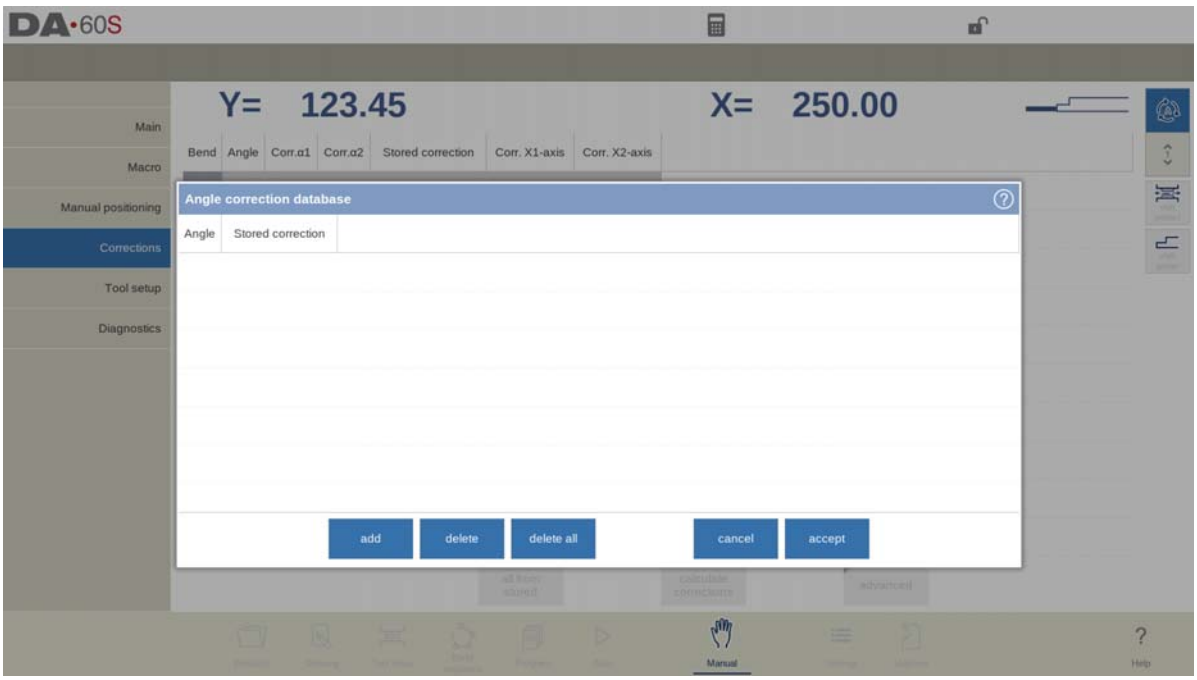
When you return to the standard screen of manual mode, the axis parameter will have the recently taught value.

8.5. 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 Automatic mode. 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 test bending and storing the found results in the database.



Bend Allowance

The function Bend Allowance helps the user to add entries to the bend allowance table in case this is active. Based on the active bend parameters only the bend allowance corrections need to be entered before adding. When the measured value is given, this will result in a calculated bend allowance, derived from the difference between the programmed value and the measured value. To enable the bend allowance table go to the Settings mode.

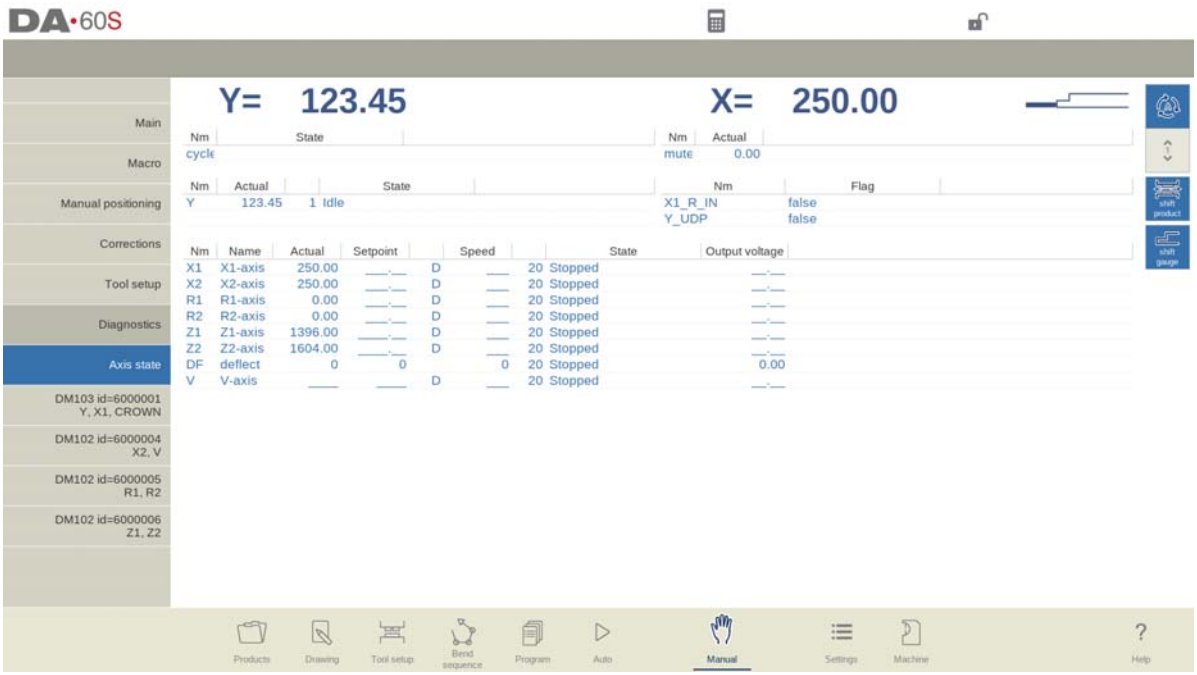
8.6. Test Bend mode

Test Bend mode enables the operator to tune the current bend.

When switched on, the axes will stay at their retract position after the first bend cycle, the part support (if present, and enabled) will remain at its angle at ER, and there will be no step change. After the Test Bend is finished the Y-axis will be at UDP, and the control is stopped. After measuring the angle a correction can be programmed and the current bend can be re-bent. The part support (if present, and enabled) will start following again as soon as the Y-axis passes the (original) position, at which the part support should start following again.

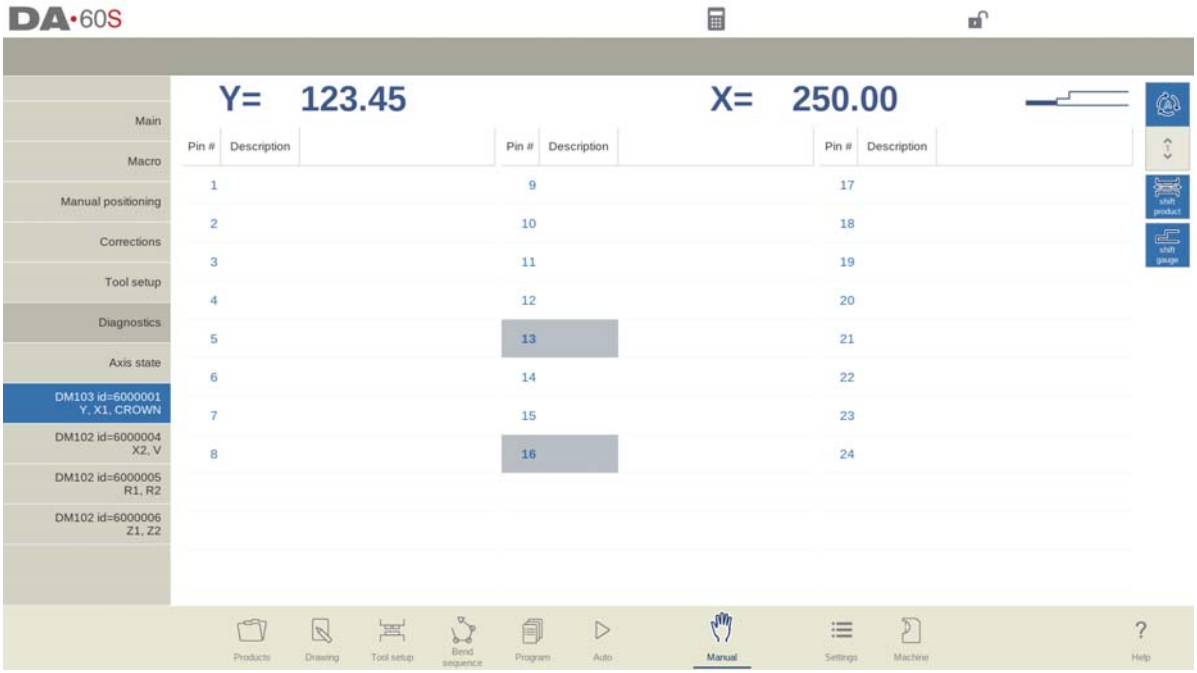
8.7. Diagnostics

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 behavior during a bend cycle.



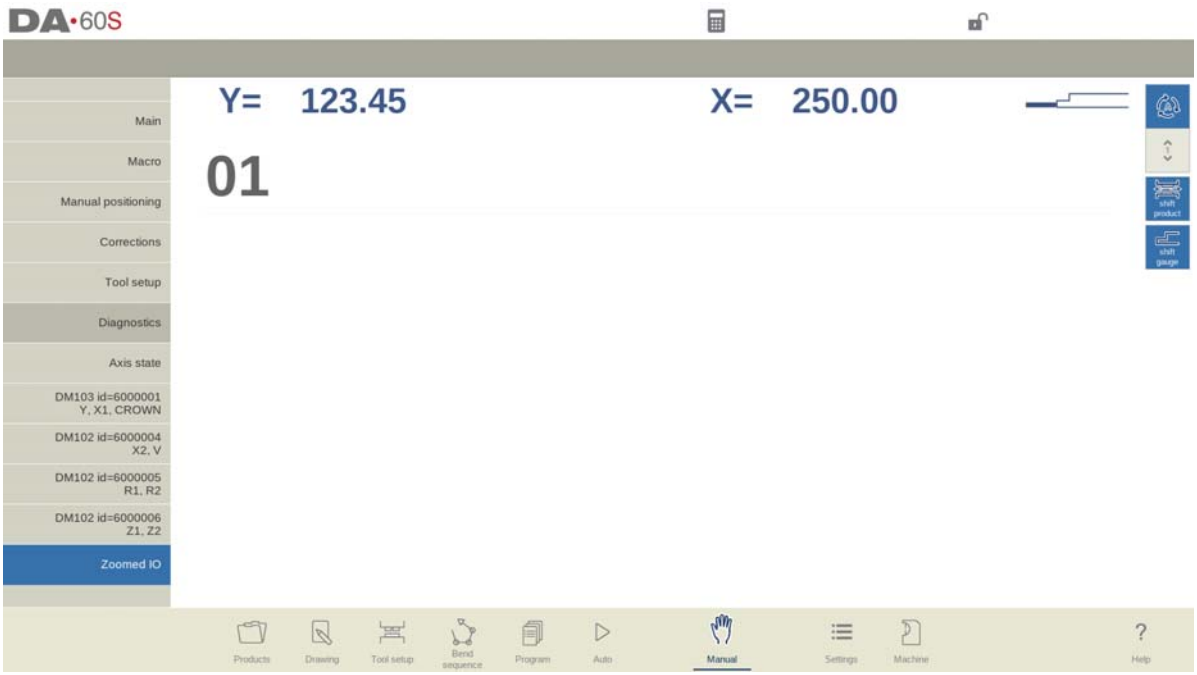
8.7.1. IO status

When tapping on one of the modules 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 behavior during a bend cycle.



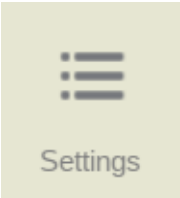
Zoomed IO

When tapping on one or more (up to 8) 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.



9. Settings

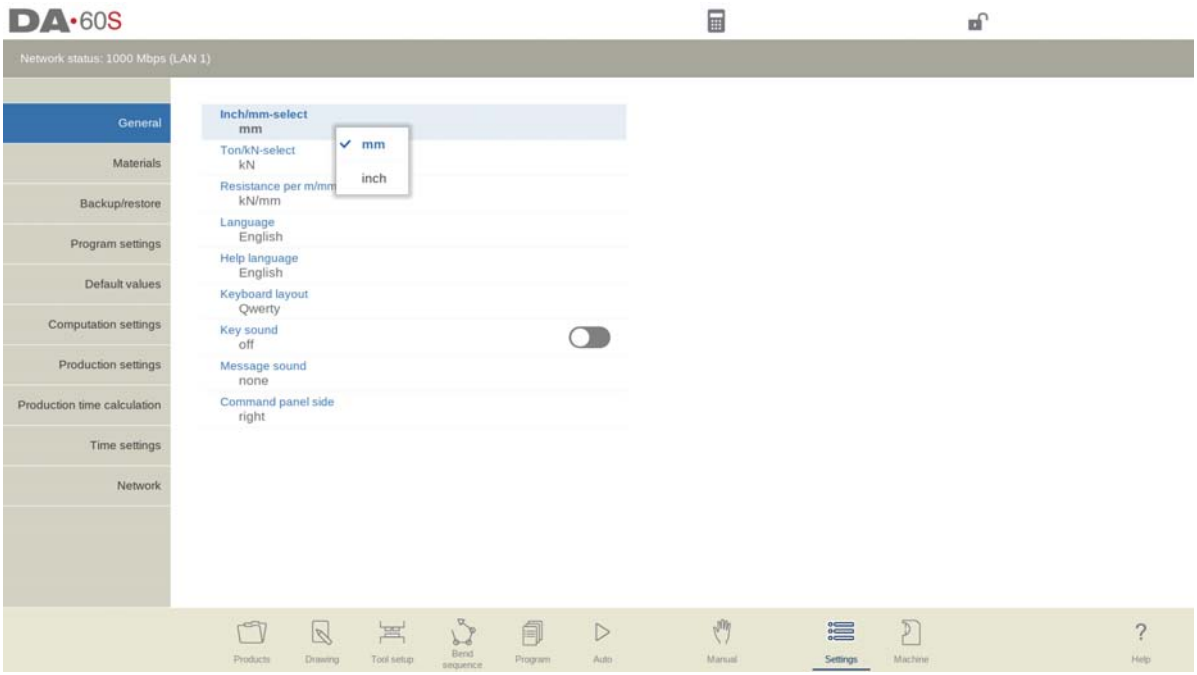
9.1. Introduction



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 a diversity 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, which can be selected at the side of the screen, logically organizing the different subjects. In the following sections the available tabs and settings details 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 vertical direction enables to view and select all available tabs.

9.2. 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

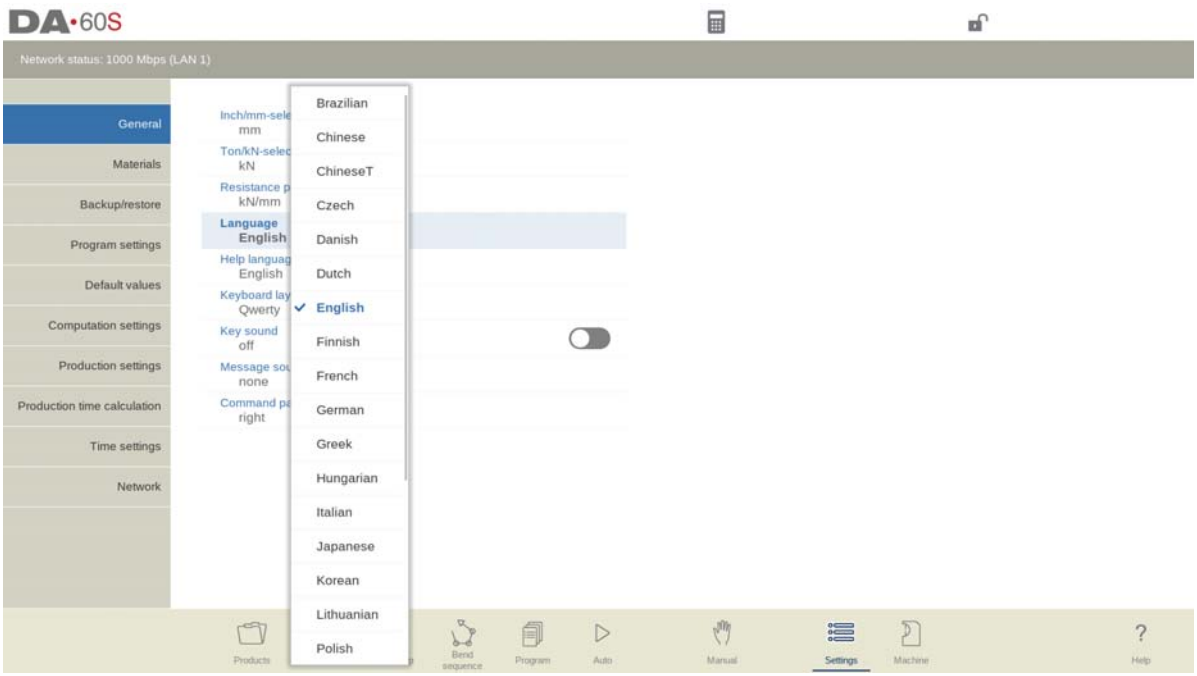
Select to use either Ton or kN as the main unit to be used for all force data.

Resistance per m/mm selection

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

Language

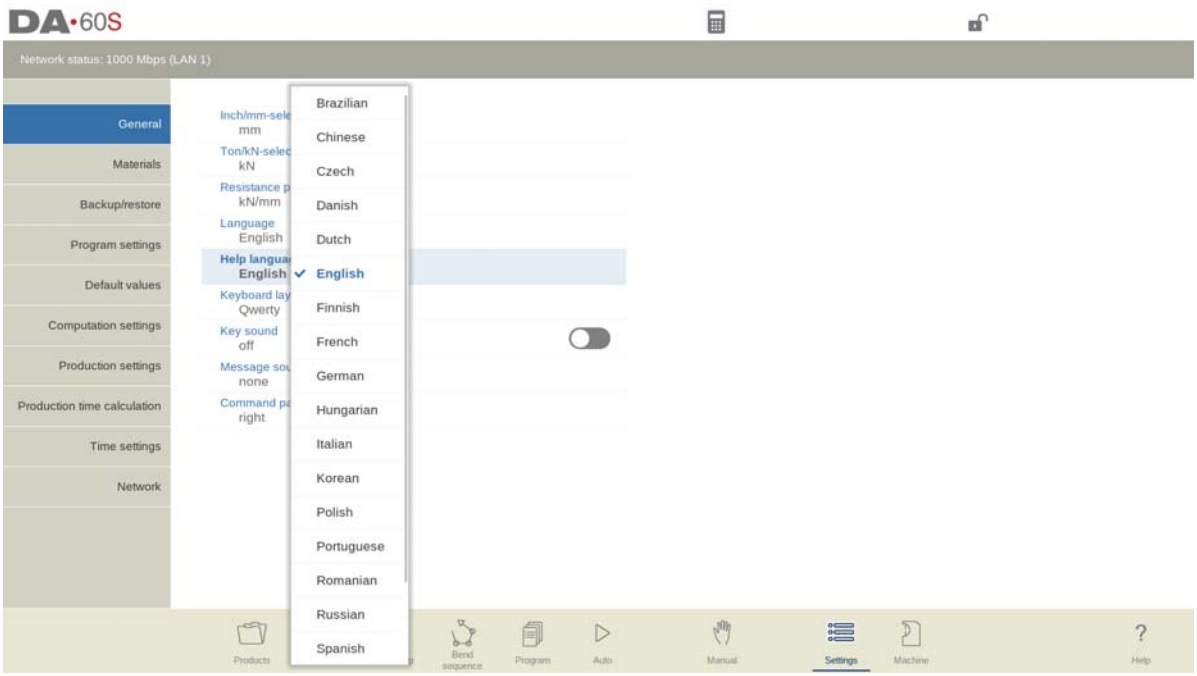
The user interface language can be selected from the list. There are more available languages than initially shown. Scroll vertically by dragging the list up en down to see all available languages. Tap to select the desired language for the user interface.
(For languages using special, non standard alphanumerical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



Help language

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

When Help Language is selected, the additional function Add Help Language appears. With this function you can install a new help language on the control. Make sure the required help file is present on the control disk or another accessible location (network, USB stick). It will automatically be selected and installed.



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.

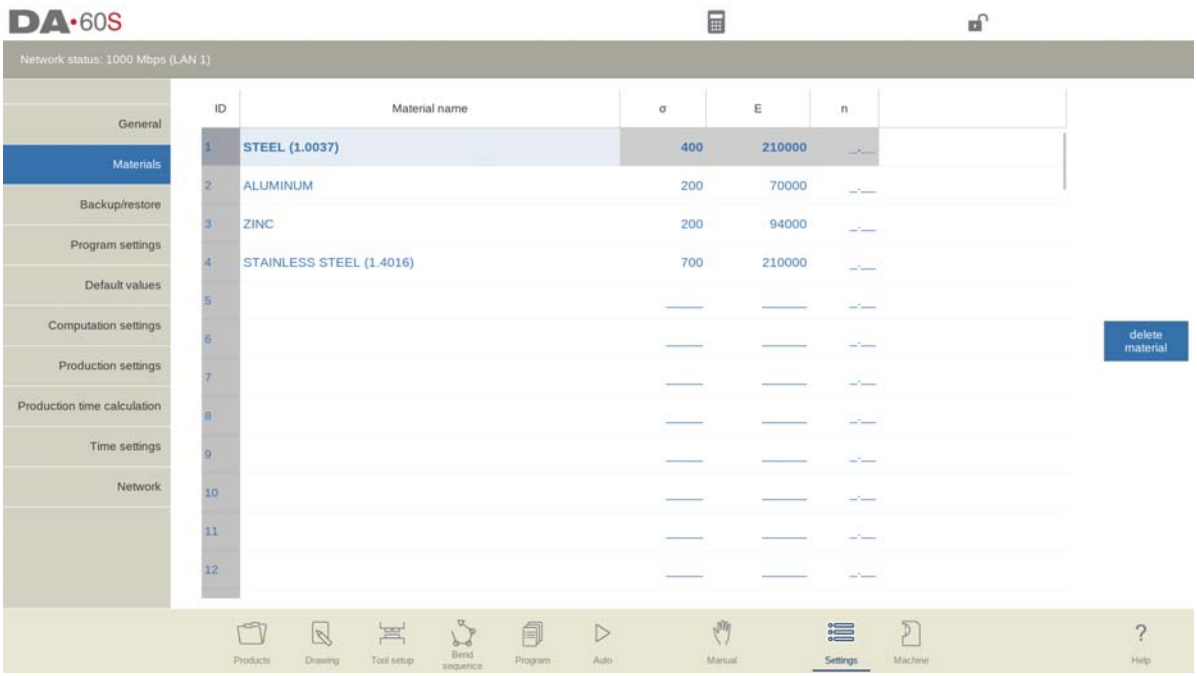
all messages => sound on for all messages.
errors + warnings=> sound on for errors and warnings only.
errors => sound on for errors only.
none => sound off for all messages.

Command panel side

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

9.3. 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

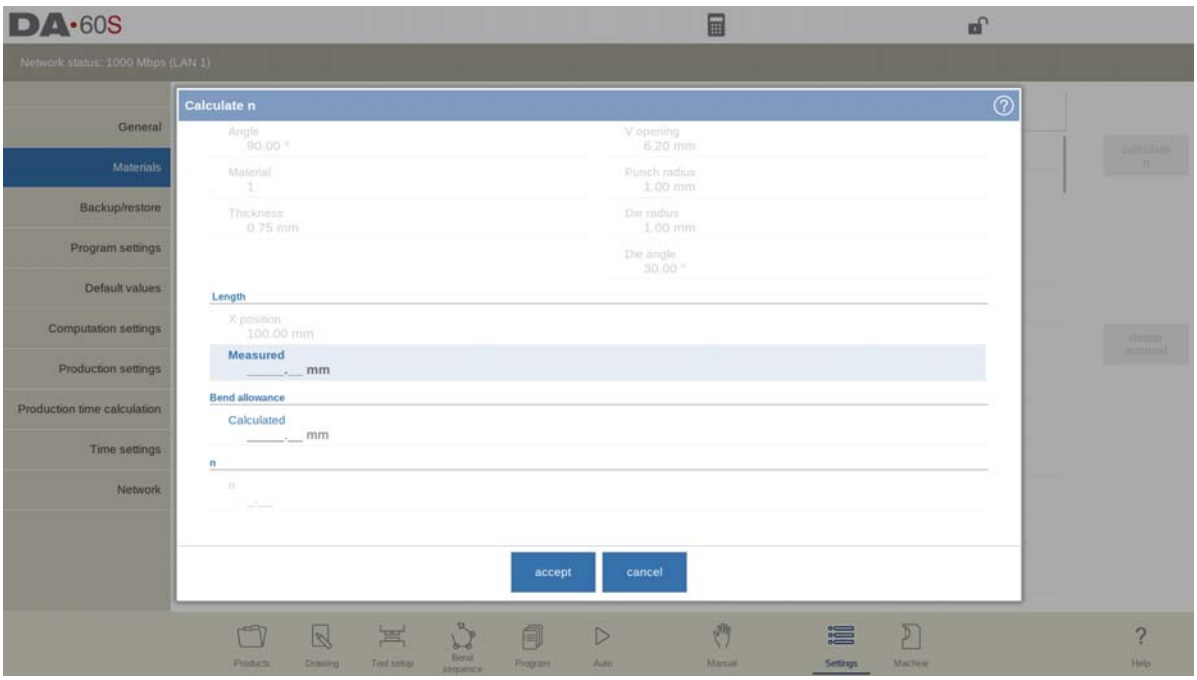
The Material strain hardening exponent, n, is a material property that should be provided by the supplier of the material, just like the tensile strength and E-module. Entering the correct values for this parameter will give an improvement of the inner radius calculation and thus a more accurate bending depth and bend allowance calculation.
In its turn a more accurate bend allowance will result in more accurate backgauge positions and a more accurate developed length (blank length).
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 _.__

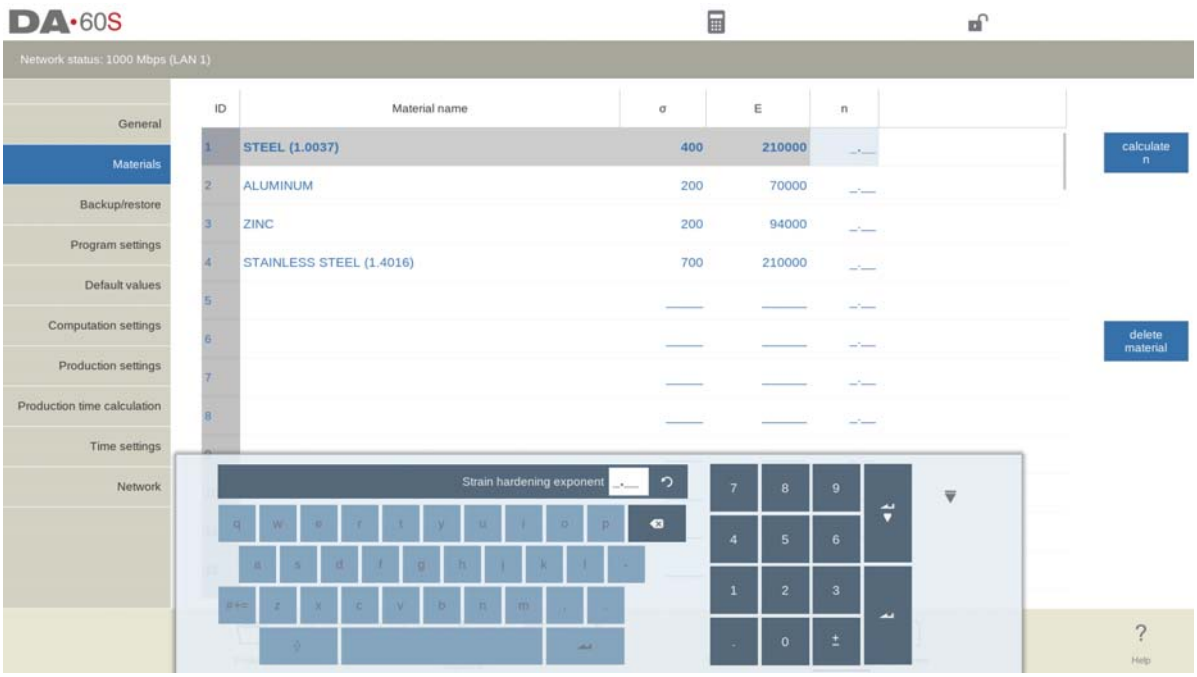
Calculate n
The Material strain hardening exponent, n, is a material property that should be provided by the supplier of the material, just like the tensile strength and E-module.
As an alternative, it can also be derived from the bend allowance. We can distinguish two situations, with or without bend allowance table.

No bend allowance table
When no bend allowance table is active, 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 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.

With bend allowance table
When a bend allowance table is active, the strain hardening exponent can be derived from this table.
Select the parameter n for a certain material and tap on the button 'calculate n'. The calculated strain hardening exponent will be prompted in the input line.



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.

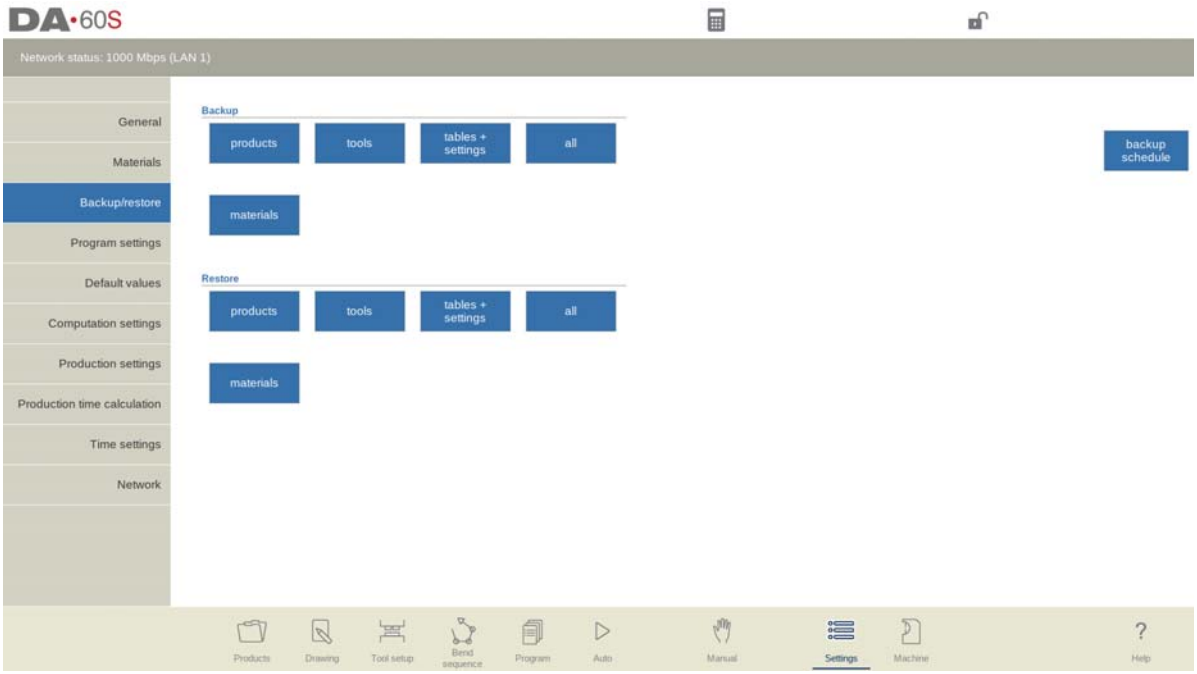
To change an existing material, select the relevant line and change the values as you see fit. To delete an existing material, select the relevant line and use Delete Material. This will erase the values.

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

9.4. Backup / restore

This tab offers the possibilities to backup and restore products, tools as well as settings and tables. 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.

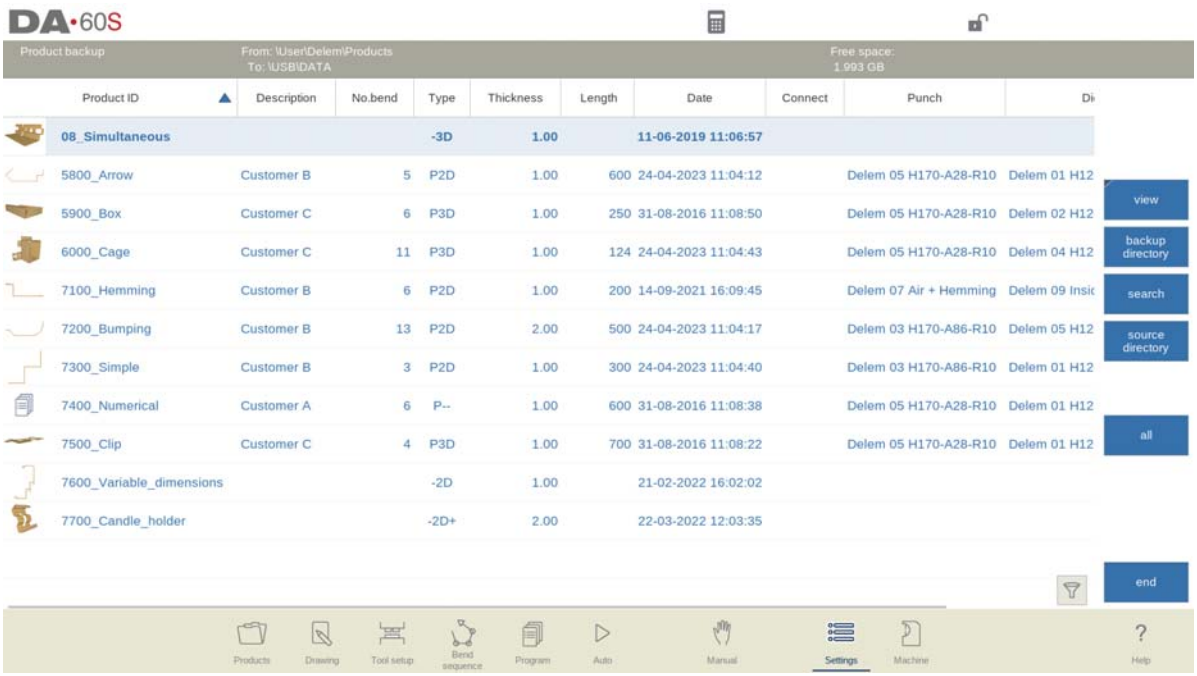


The actual backup directory consists of a device (USB stick, network) and a directory. The choice of devices depends on which devices have been connected to the control. If necessary, directories can be created and selected. The backup locations for storage of products and tools can be set independently.

9.4.1. 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.



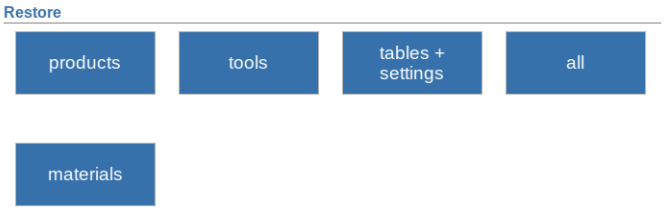
In the backup screen the products in the selected directory are shown.
Basic functions to change the view can be chosen similarly to the Products mode. This enables to easily find the required products to be backed up.

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 ones, tap All.

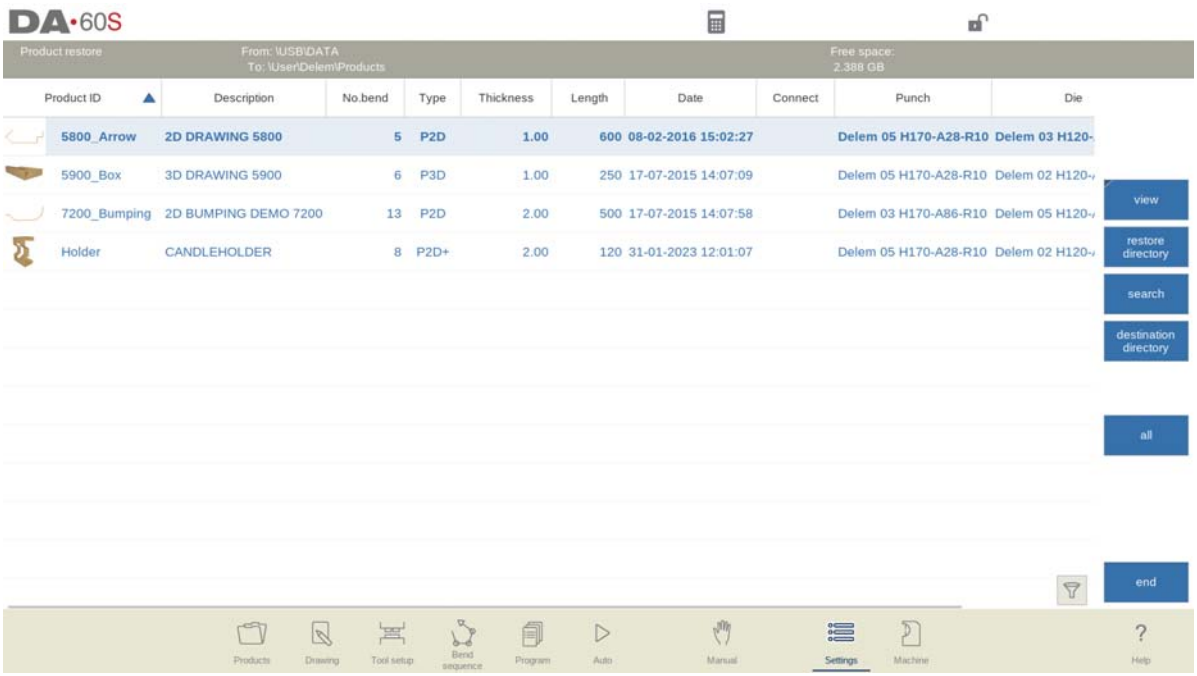
The source where the products are located which have to be backed up can be changed with Source Directory. The directory browser appears and the desired source directory can be navigated to.
The directory where the products which need to be backed up need to go can be changed as well. With Backup Directory the directory browser appears and the desired destination directory can be navigated to.

9.4.2. 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.

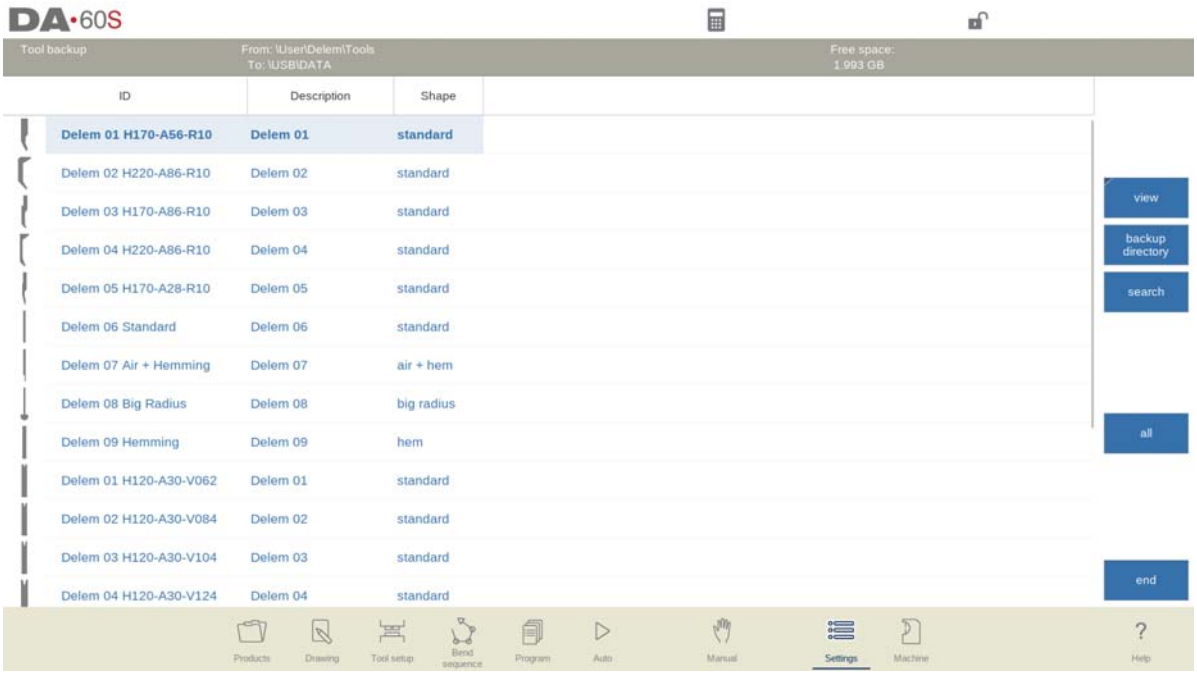
At the top of the screen, the current restore source location is shown as well as the location on the control to restore to. To restore a product, select it by tapping, in the list. The restore marker will appear to confirm the restore action. If a product file with the same name is present on the control, 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.

9.4.3. 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.

9.4.4. Tool restore

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

9.4.5. Backup and restore for Tables and Settings

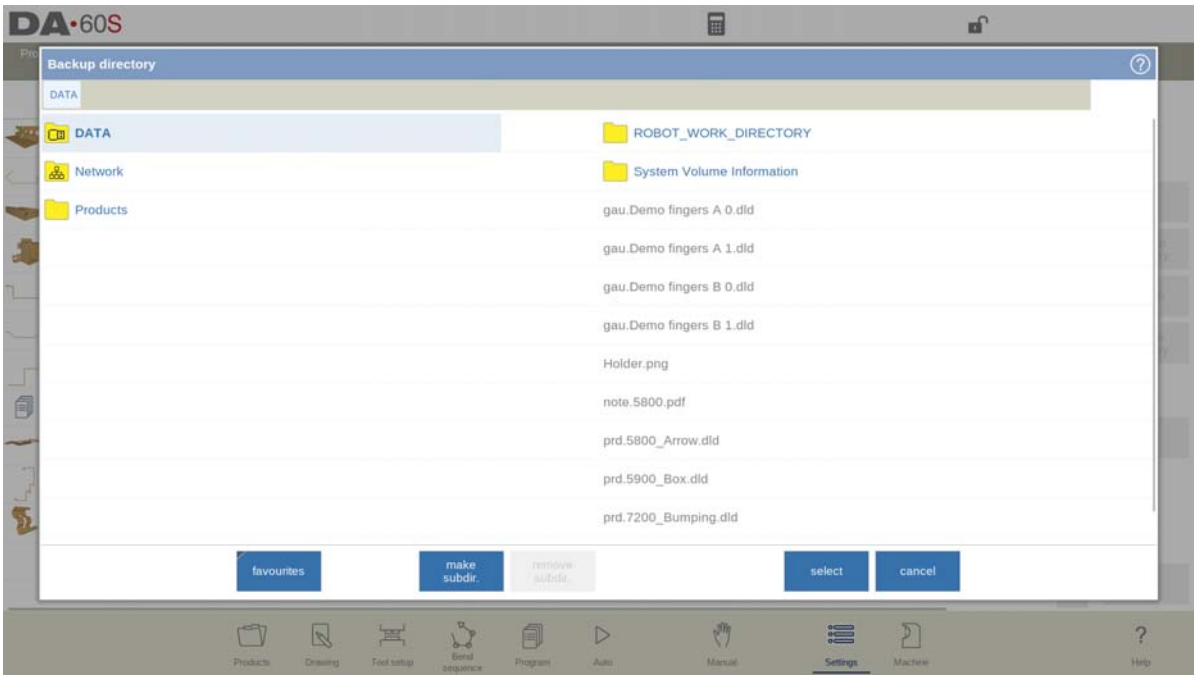
To backup user specific settings and tables the Backup/restore tab offers specific functionality. The procedure is again similar to the backup and restore of products and tools. The special function All will automatically execute all steps sequentially for either Backup or Restore (Products + Tools +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

9.4.6. Automatic scheduled backups

To automatically perform a 'backup all' action in a specific time interval the scheduled backup settings can be set by choosing ‘backup schedule’ on the backup/restore page. The interval can be set from 1 to 31 days.
If the control is ready (i.e. not started) and the time is due, a message is shown asking the user to backup now or to be reminded later. The reminder (by default 1 hour) can be set from 1 to 24 hours. The desired location of the backup can also be set.

9.4.7. Directory navigation

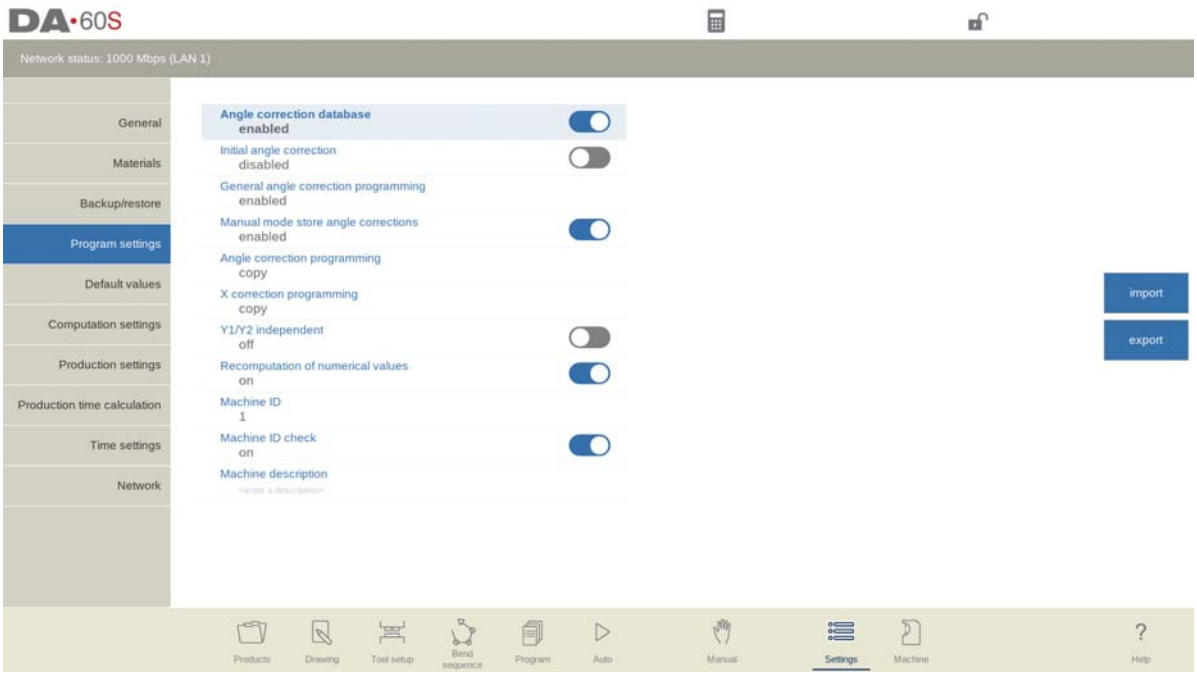
When Backup directory is used, a new window appears with a list of available backup directories.



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.

9.5. 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. α1.
α1 and α2 => G-corr. α1 and G-corr. α2.

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 Cα1 to Cα2 when changing Cα1.
delta => keep delta between Cα1 and Cα2 when changing Cα1.
independent => change Cα1 and Cα2 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.

Recomputation of numerical values

Parameters in Manual mode and Program mode can be programmed one by one. This

parameter determines the influence of a parameter change on other parameters.
disabled => the influenced parameters are only recomputed when over-entering, accepting the proposed changed values.
enabled => the influenced parameters are recomputed automatically.

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.

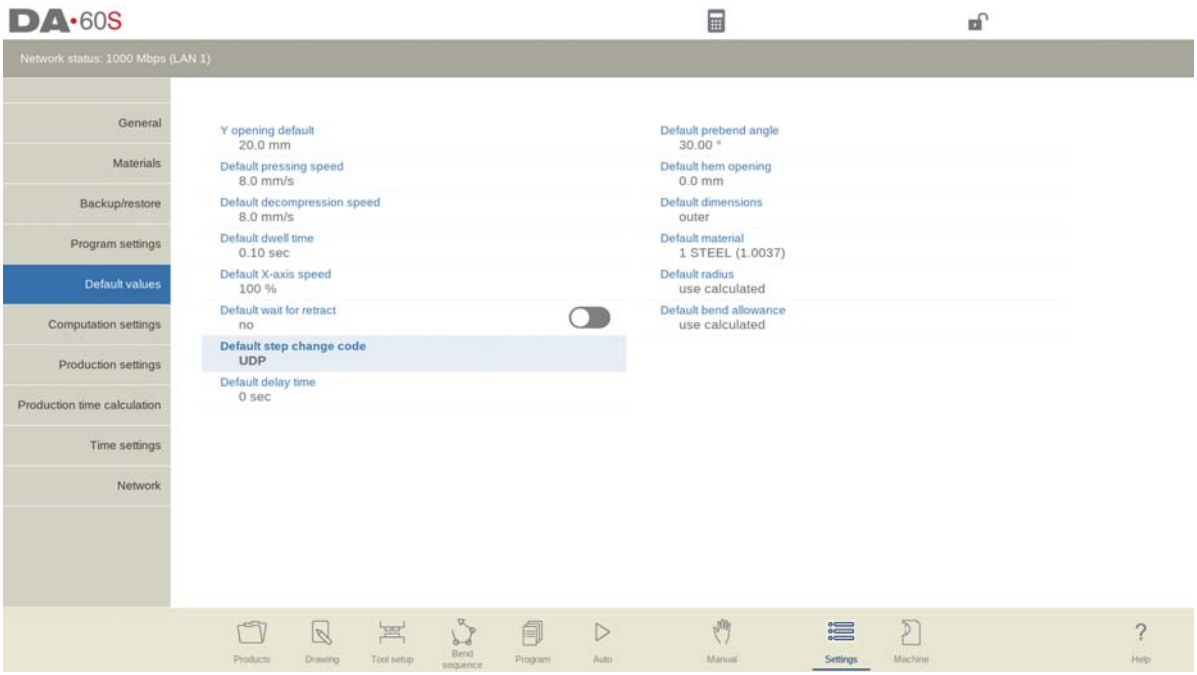
Machine ID check

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.

9.6. 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 dwell time

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

Default X-axis speed

Default X-axis speed, used as initial value for the parameter '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 behavior in a bend program.

Default step change code

Default value for the parameter '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 post processing, the waiting time of the X-axis at step change is set to zero.

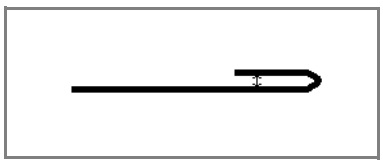
With this parameter you can preset a longer waiting time when needed for product handling.

Default prebend angle

Default value for the parameter 'prebend angle' in a graphical product.

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 the product with hem bend graphically in the Drawing mode or by 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 dimensions

A graphical product drawing can be made in outside or inside dimensions. This parameter defines the default value of the parameter “dimensions (D1)” when creating a graphical drawing.

Default material

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

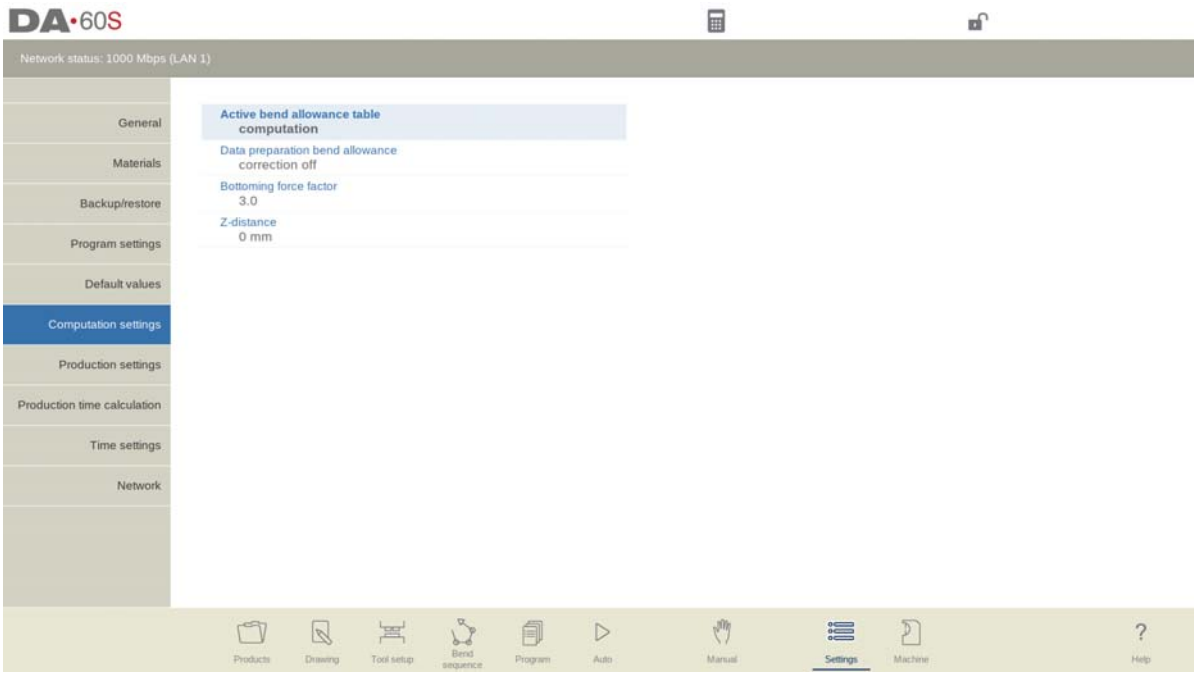
Default radius

Default value for the parameter 'radius' on the product properties dialog of the drawing.

Default bend allowance

Default value for the parameter 'bend allowance' on the product properties dialog of the drawing.

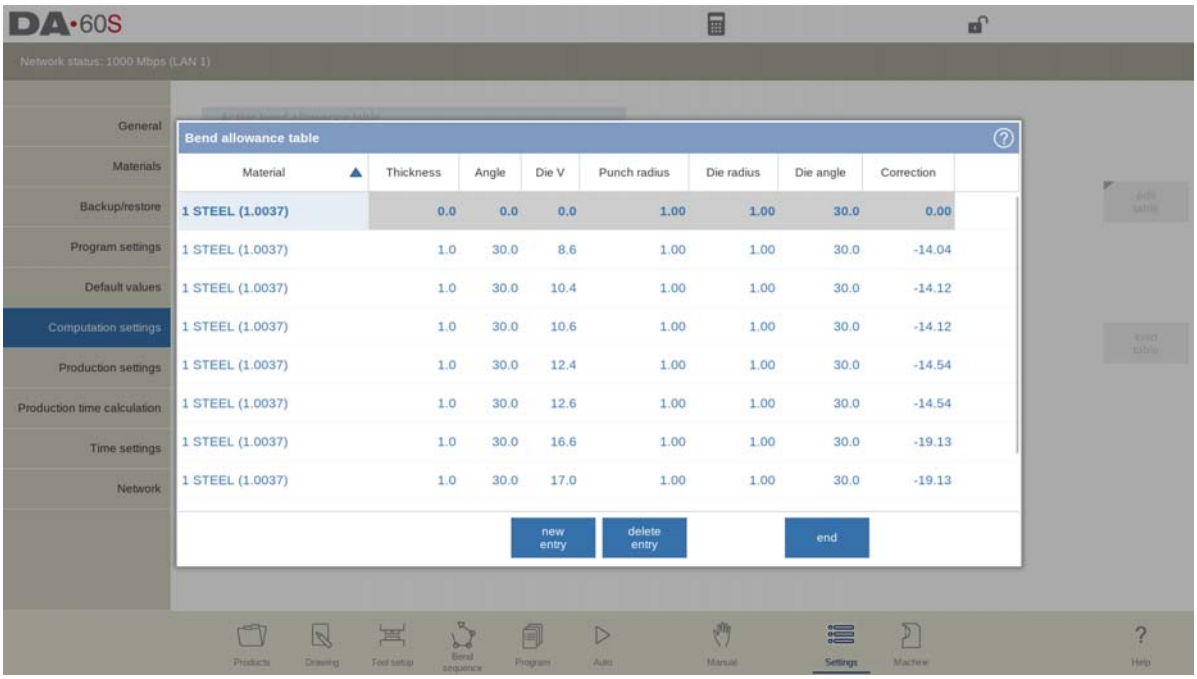
9.7. Computation settings



Active bend allowance table

computation => the control will calculate the bend alowance
table => the bend allowance table will be used
Bend-allowance is the correction of the X-axis due to sheet shortening after bending. With this parameter the method for bend-allowance calculation is chosen. 'Computation' means the standard formula of the control is used to calculate the bend-allowance. 'Table' means a bend-allowance table with correction values can be used.

When the cursor is on the parameter ‘Active bend allowance table’ and the bend-allowance table has been selected then an extra function is available: Edit Table. With Edit Table the table can be modified to meet user requirements. The table appears in a new window, with its own controls.



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

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

It is also possible to load a existing bend allowance table.
Tab Load Table to navigate to the desired file location.

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').

This choice has no effect on the post process function in the Drawing mode. When the control post processes to calculate a CNC program from a drawing with bend sequence, the control will always take bend-allowance into account.

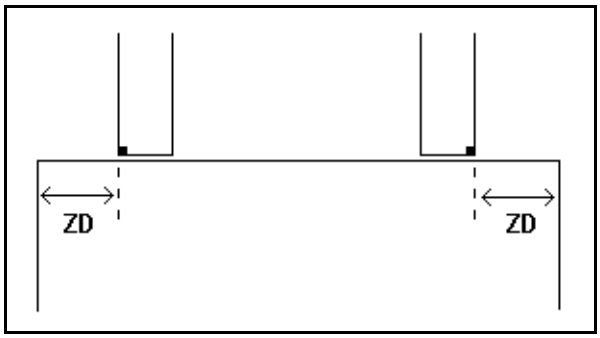
Bottoming force factor

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

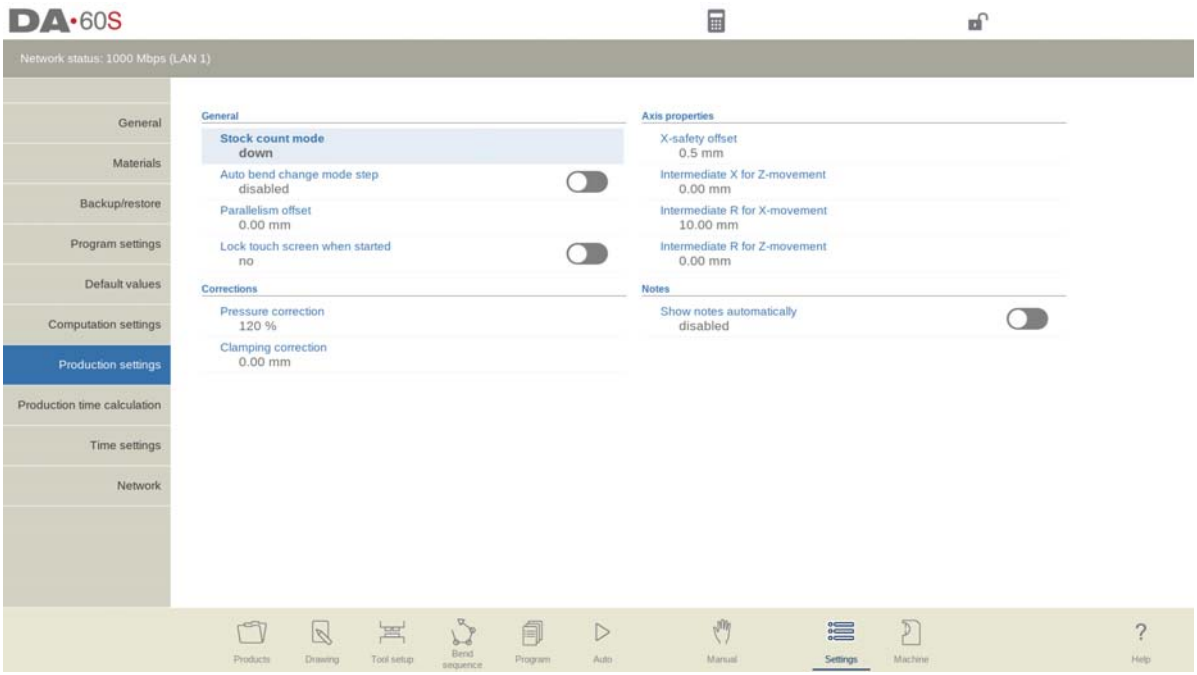
Z-distance

The distance from the edge of the finger to the corner of the sheet.
When automatic Z-axes have been installed, the positions of the fingers are calculated

automatically with respect to the end of the sheet.



9.8. 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.

Default part support return speed

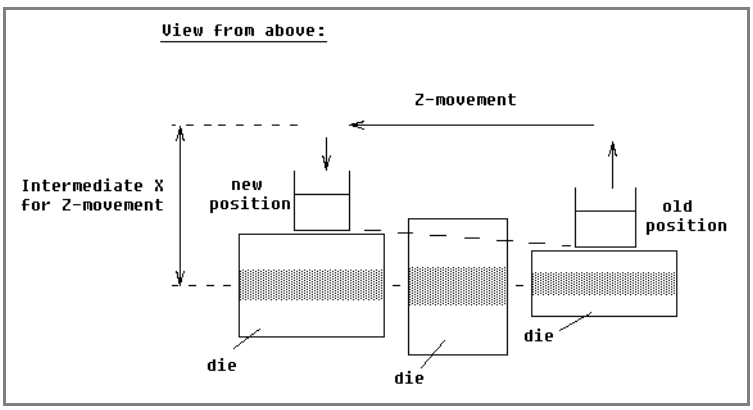
Return speed of the part support after a bending. The speed value is programmed as a percentage of the maximum speed.
Only available when a part support is present.

X-safety offset

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

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.

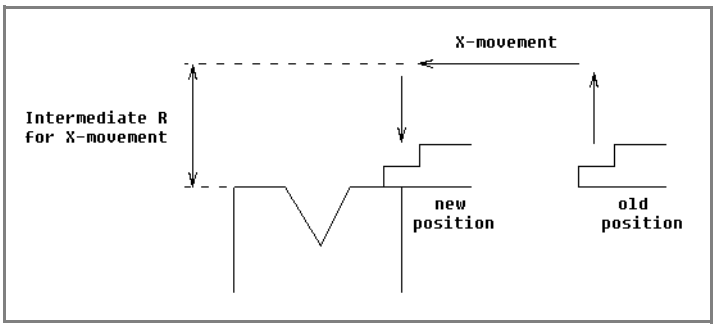


When the backgauge has to move to a different Z-position, it is checked whether the current X-position is safe. We can distinguish the following situations:

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

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:

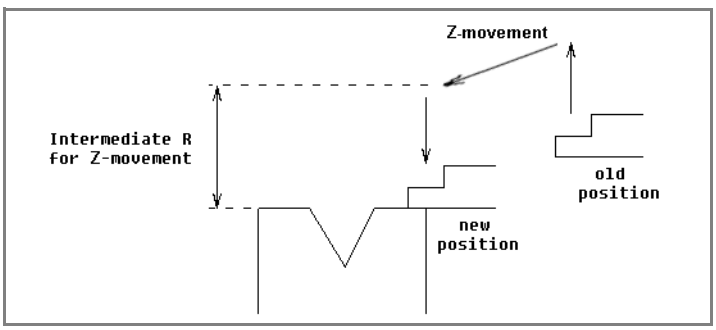
SZ = X-safety offset + SD

Explanation:

SZ = safety zone
X-safety offset = safety zone of the die
SD = safety distance, defined by machine supplier

Intermediate R for Z-movement

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



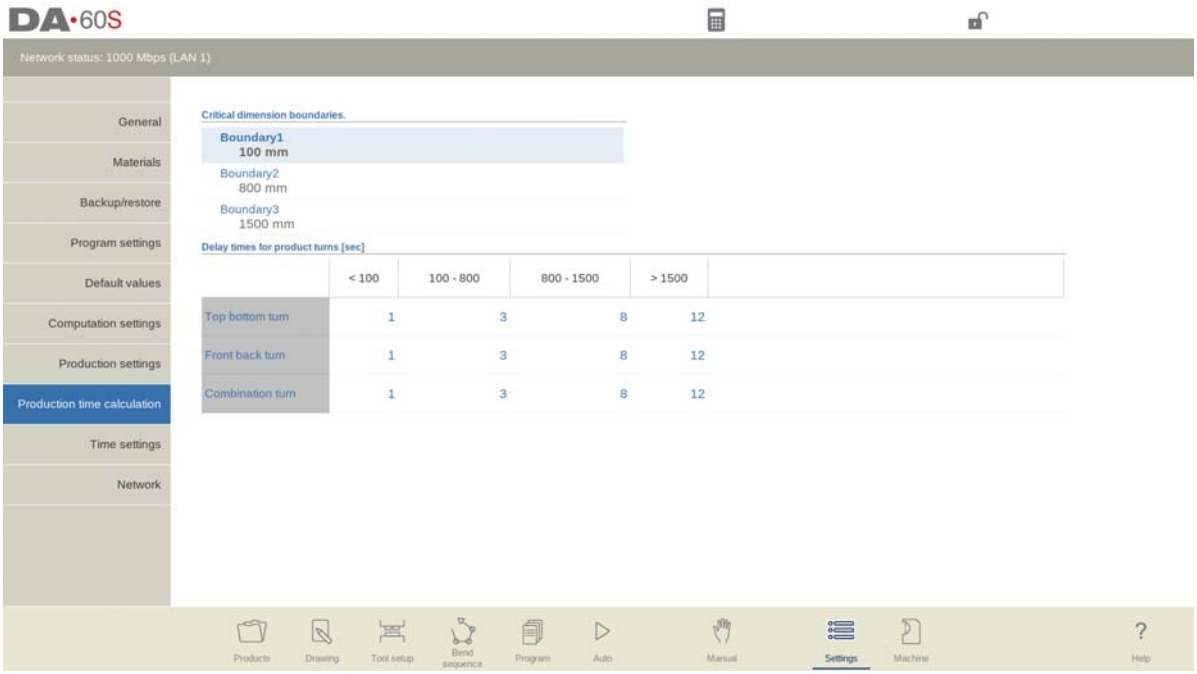
The sequence will be as follows:

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

Show notes automatically

In some situations it is required to show the bend step notes automatically when automatic mode is started and when a new bend is selected. When this parameter has been programmed as “enabled” the notes are shown automatically in automatic mode.

9.9. Production time calculation



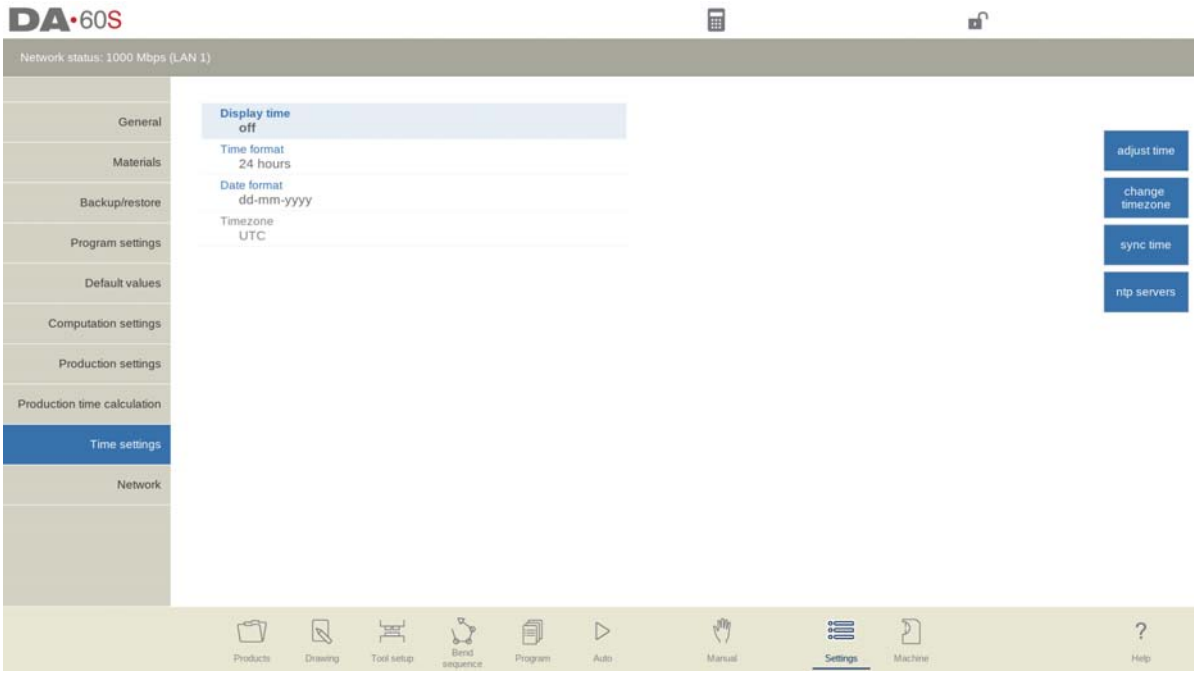
The parameters on this page are used to calculate the production time for a product in the bend sequence computation process. This production time depends on the positioning speed of the axes and the product handling times. The positioning speed is depending on machine settings.

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

Boundary 1/2/3

Threshold values for product size. For values between these boundary values different product turn times can be programmed.

9.10. 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.

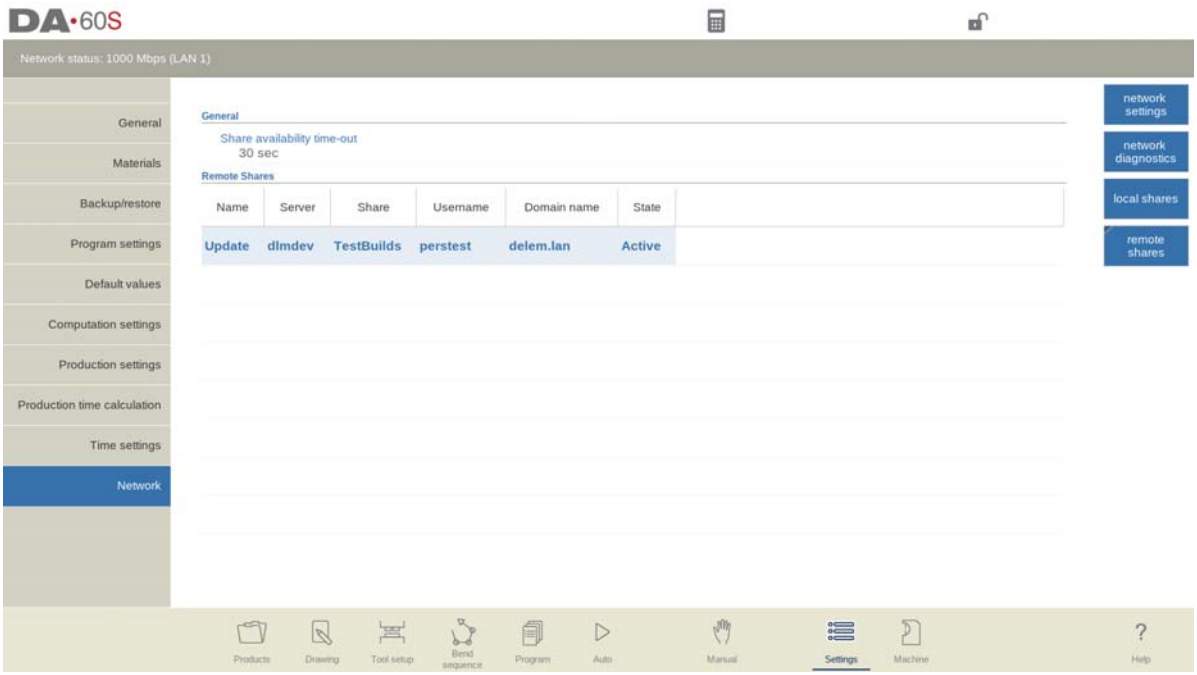
Date format

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

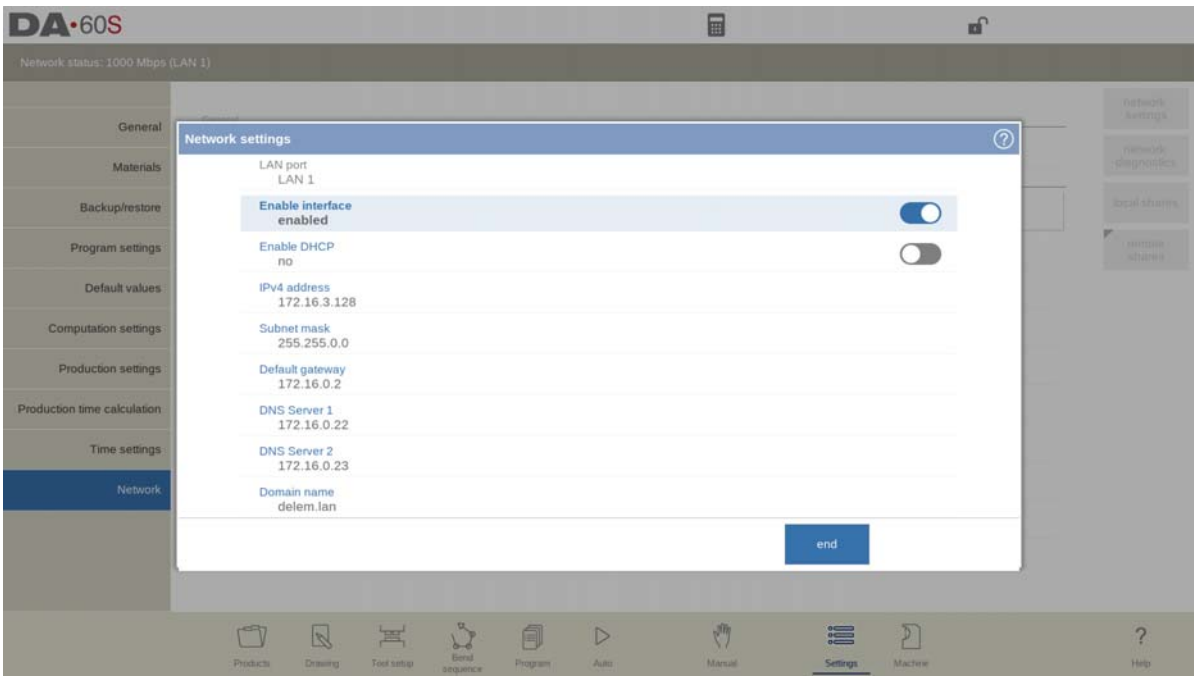
Adjust time

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

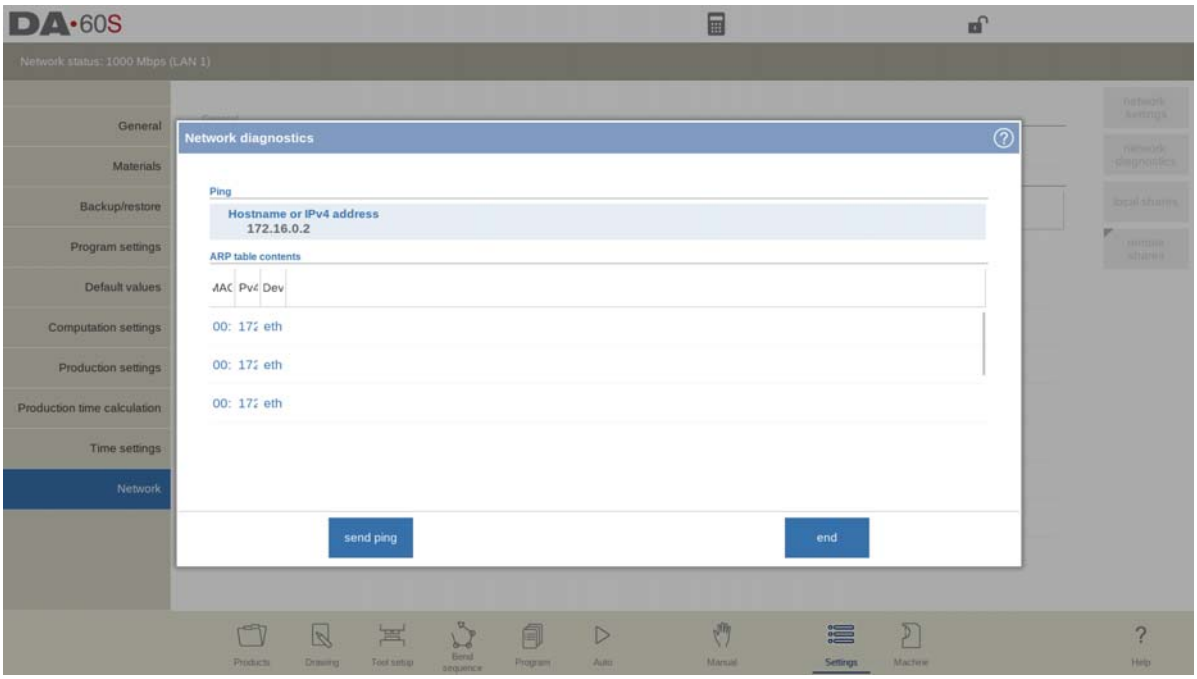
9.11. Network settings



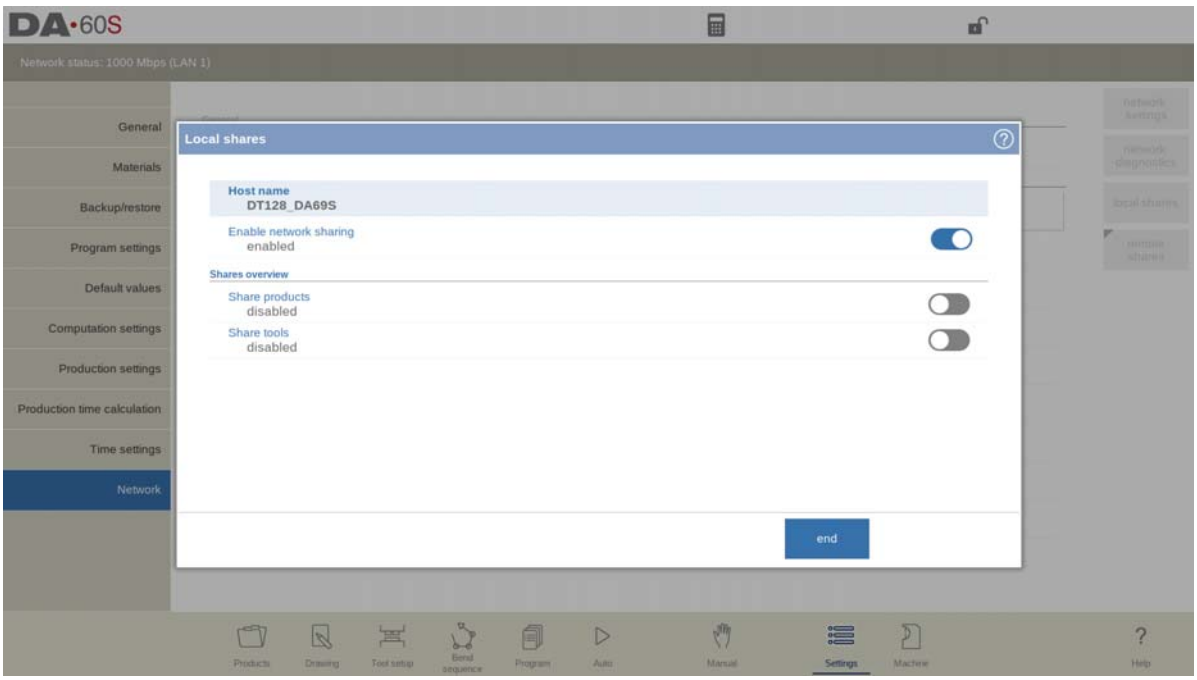
When the Network option has been installed in your CNC, you will have additional settings parameters within the Settings mode of the control. Within the Settings mode you will find the tab Network offering pages and parameters for Network settings, Network diagnostics, Local shares and Remote shares.



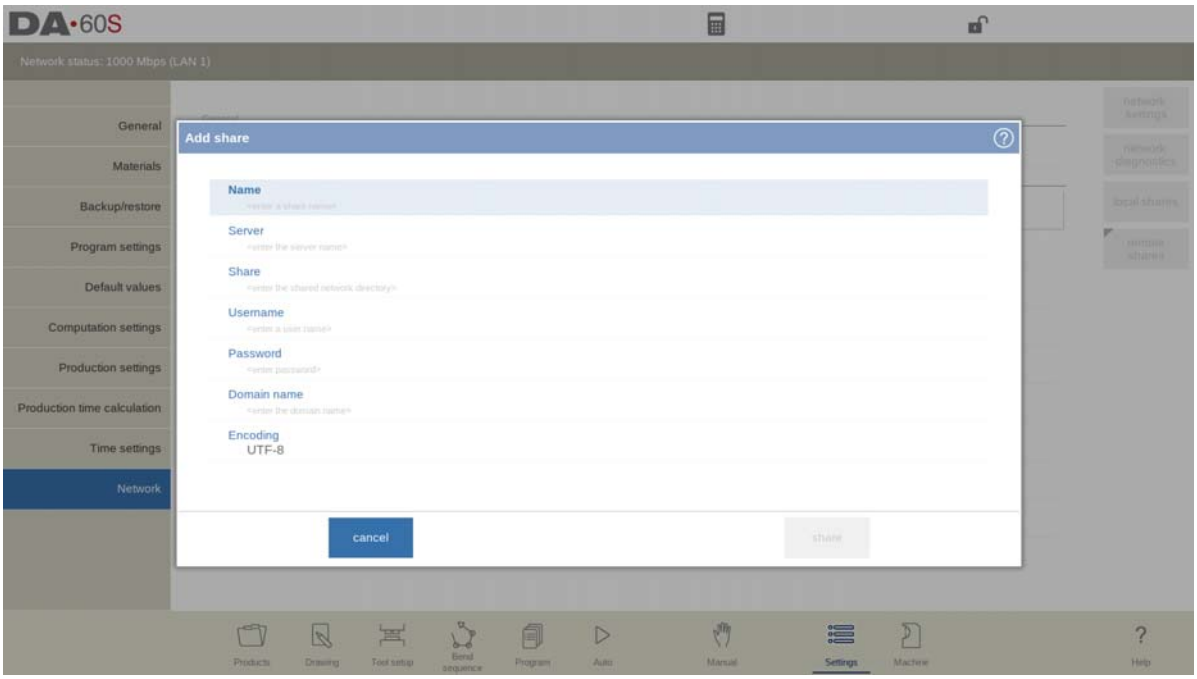
For setting the network connection several standard network parameters are available that must be set with help of your system administrator.



To check the basic functionality of data transfer on the network some diagnostic tools are available.



To share local directories of your CNC to have them accessible on the network it is possible to enable the required information.



In the other direction it is also possible to share remote directories in the network so that you could access those locations from the CNC. For this action you can add remote shares or remove them when required.

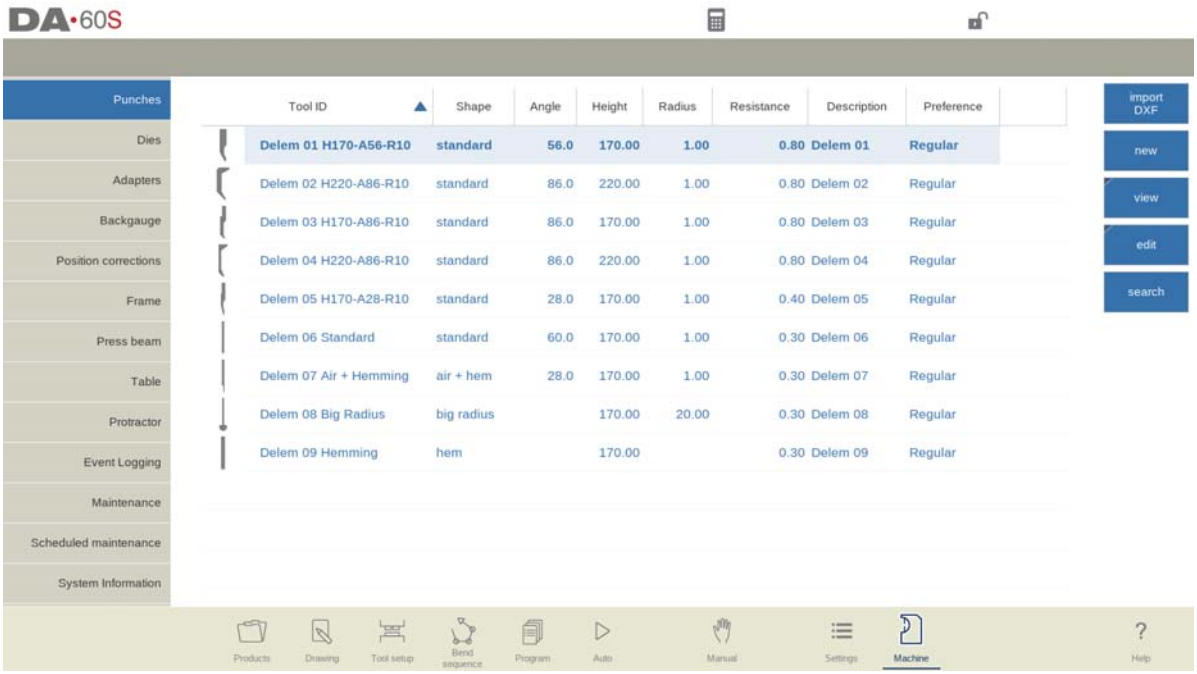
10. Machine

10.1. 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 behavior.

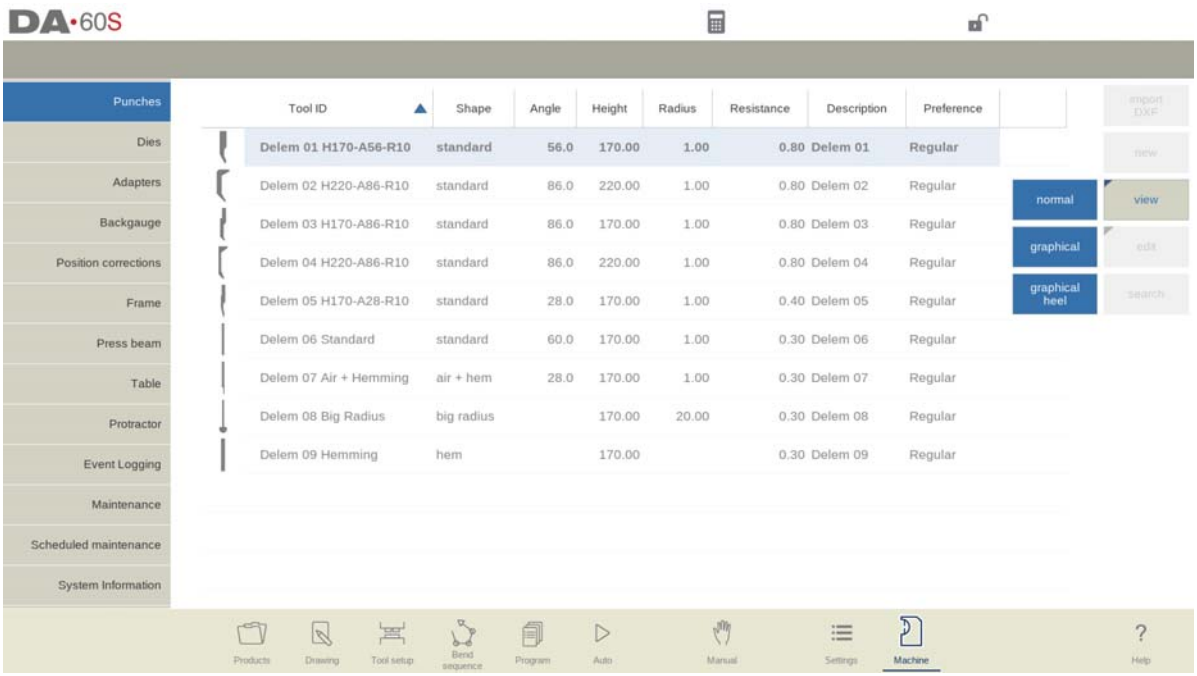
The settings are divided across several tabs, located a the side of the screen, logically organizing the different subjects. In the following sections the available tabs and settings details are discussed.



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 vertical direction enables to view and select all available tabs.

10.2. Programming of Punches

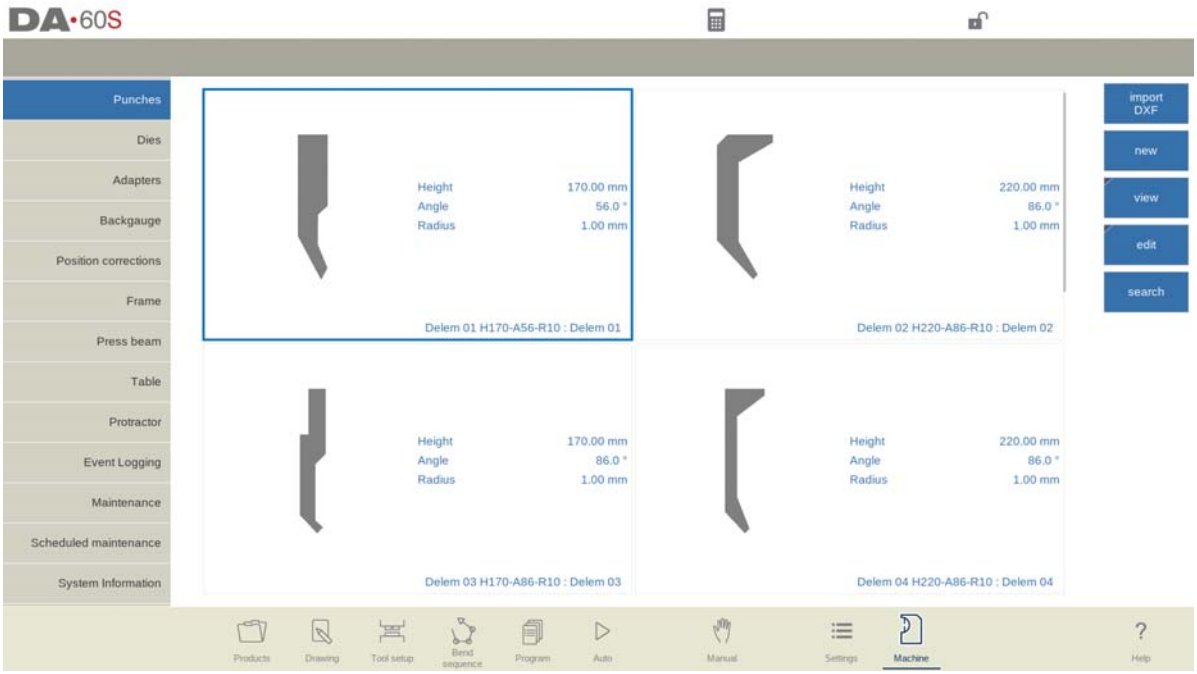
In this tab, the punches used in the machine can be programmed. New punches can be added; existing punches can be edited, copied, renamed and deleted.



10.2.1. View

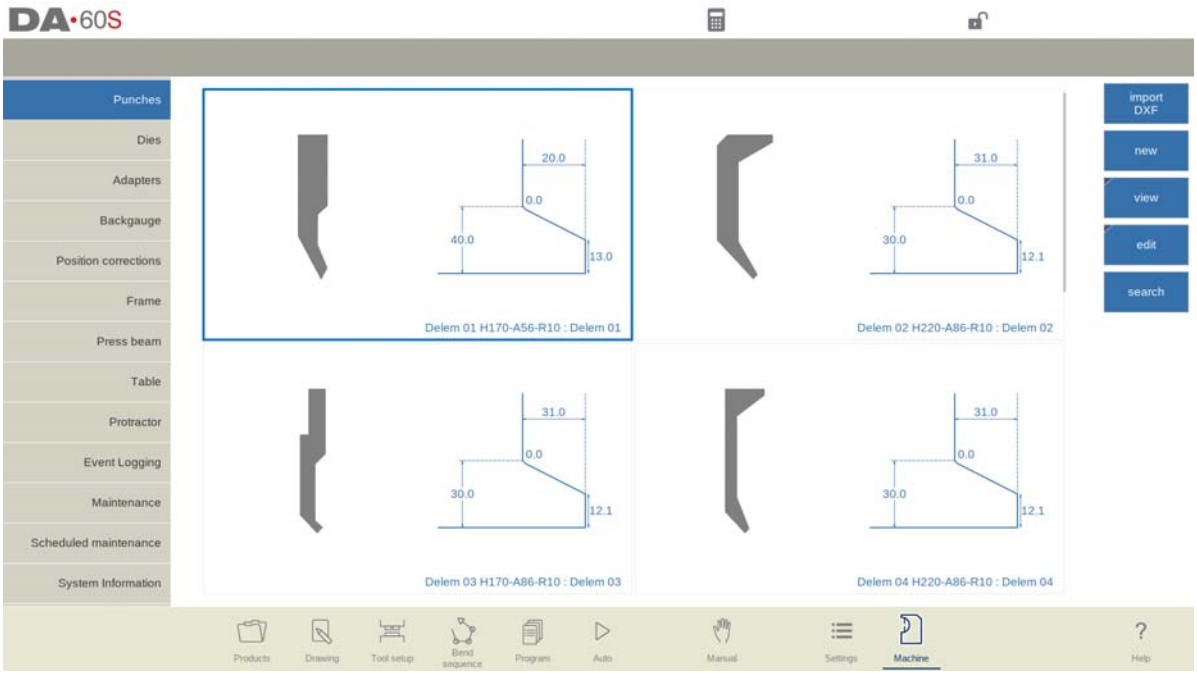
In the main page, a list of available punches is shown. By using the View function, similar to Products mode, different views can be selected. Next to the default view Normal also Graphical and Graphical Heel are available.

Graphical directory



In Graphical the geometry of the tools is shown as well as the main properties.

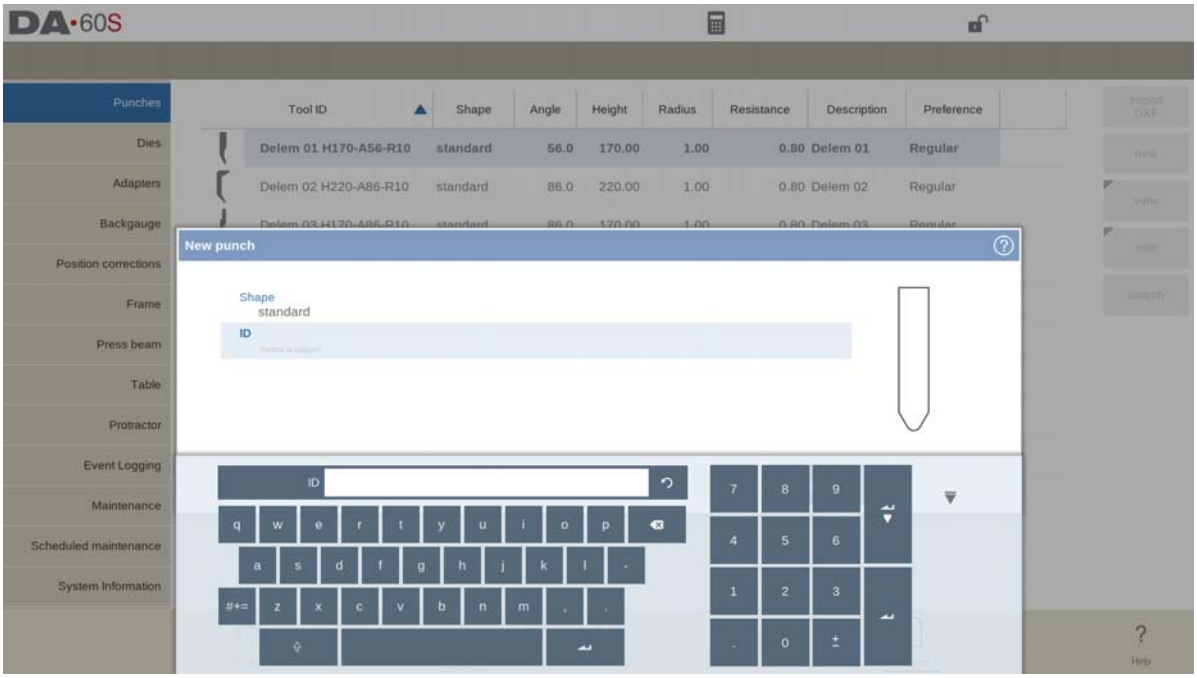
Graphical directory punches with heels



In Graphical heel the geometry of the tools is shown as well as the heel properties.

10.2.2. 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 and drawing 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.

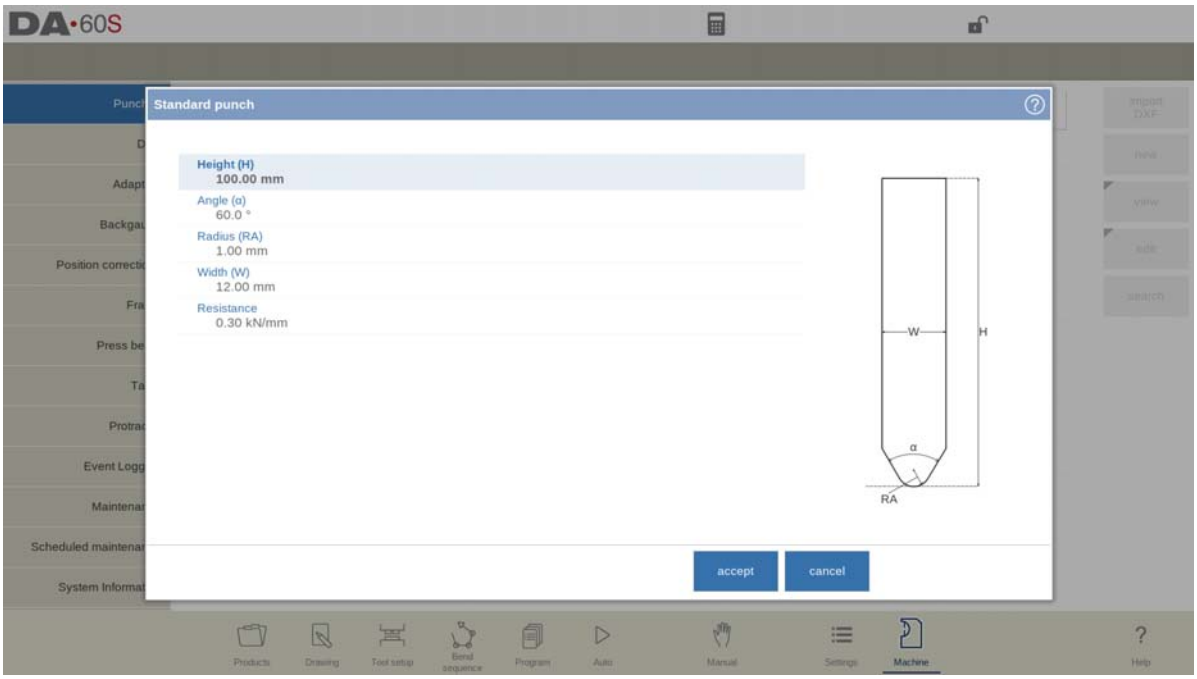
The ID parameter may also contain alphanumeric characters. When finished, use Accept to leave this window and start with the tool data parameters. The control will prompt a window for the tool properties starting with the initial dimensions of the tool.

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

DXF import (only available when DXF option has been installed)

The shape of a punch can also be loaded via the optional DXF import function. For details see paragraph 10.11.

10.2.3. 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

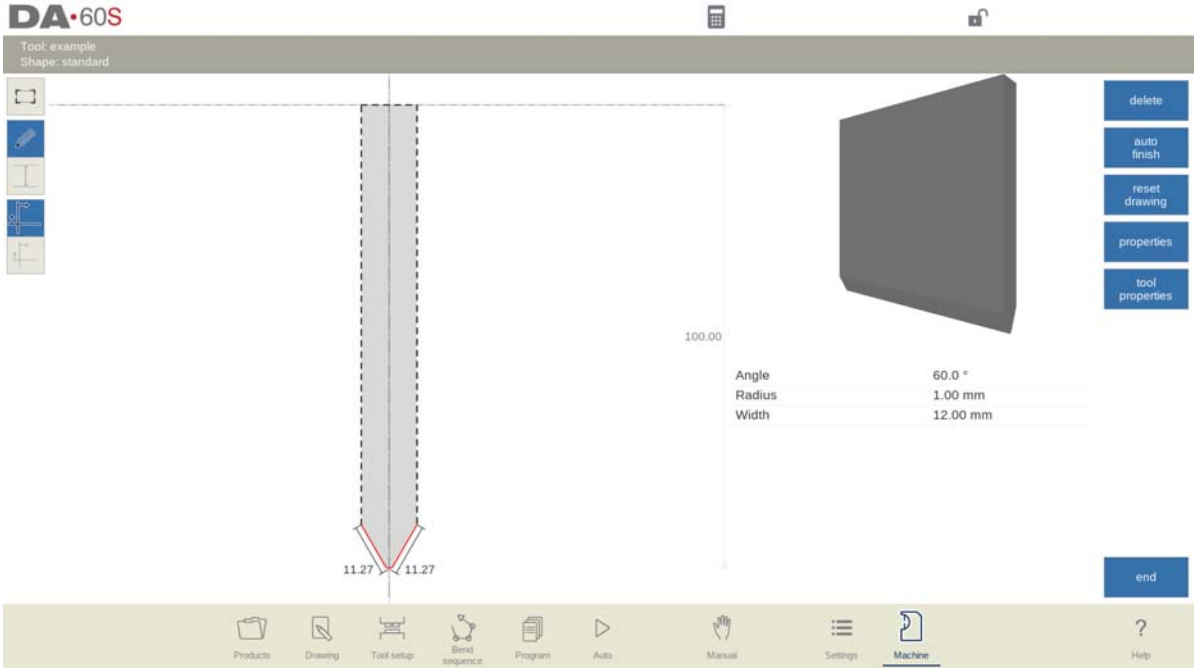
Maximum allowable force on the tool.

Orientation of the punch on the screen

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

Drawing

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.



Mount point (optionally programmable)

Within the drawing one also finds the mount point of the punch. This point, indicated with the triangle arrow, shows at what point the punch will be connected and placed on the press beam or in an adapter. For both normal and turned a separate mount point is available. Mount points can also be found in adapters and in the press beam. When this function is not enabled, the indicators are not shown.

Following functions are available while drawing

Delete Line

To delete a line segment.

Auto Finish

Finishing the tool outline to the top of the tool automatically.

Reset Drawing

To reset the programmed drawing of the tool till the basic, initial shape, when creating a new punch.

Reload Drawing

To reload the programmed drawing of the tool till the basic, initial shape, when changing an existing punch.

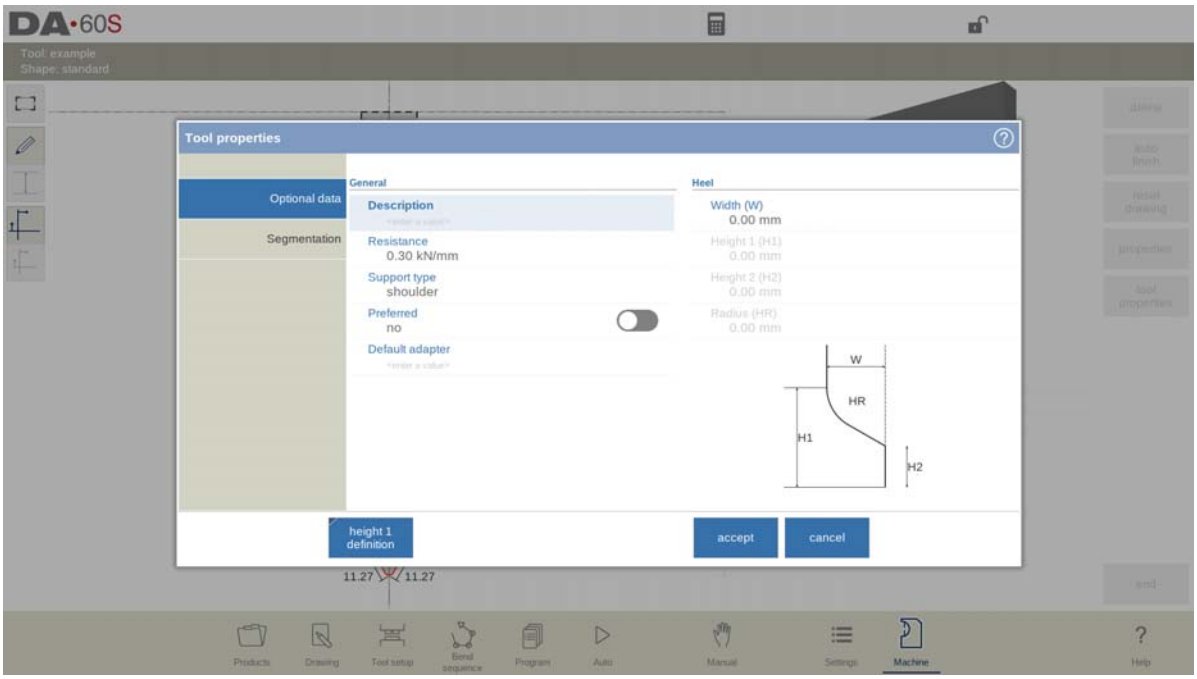
Properties

To change the specific properties of the line or angle, add or remove a radius, change the length, etc. It is e.g. possible to add a radius in the outline of the tool.

Hemming can be part of the properties of lines. Within the shape of a tool a surface can be appointed as a hemming surface. This will enable the tool for hemming operation.

Tool Properties

To change the generic tool data and description.



Description

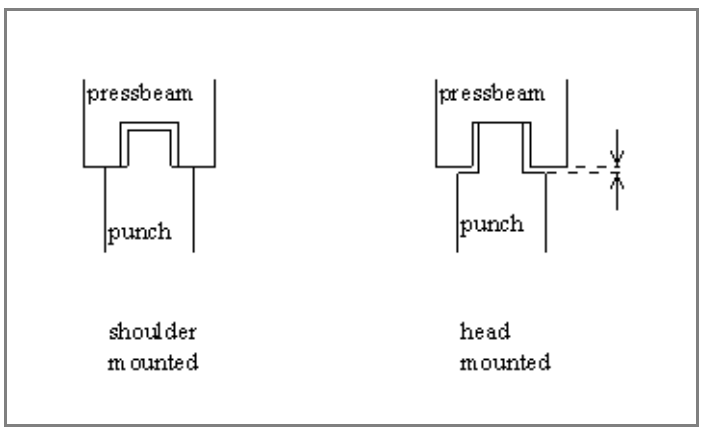
A name or description of this tool. The maximum length is 25 characters. This description has already been entered in the beginning defining the tool, but can be edited in this field. The description is listed in the tool overview of the library.

Resistance

Maximum allowable force on the tool.

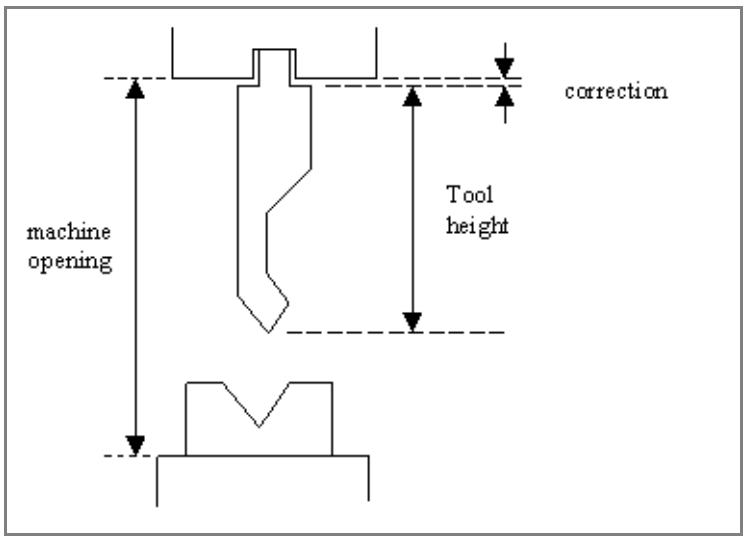
Support type

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



Preferred

For automatic tool selection, based on equivalent fitting of a tool for a specific bend, this parameter can sign a tool to be preferred above others. The automatic tool selection will prevail the tools being signed as preferred.

Default adapter

When a tool is used with a specific adapter and this combination is used commonly, the adapter can be set as default in the properties of the tool. Every time the tool is selected, automatically also the adapter will be loaded in the tool setup.

Heel dimensions

Width

Width of the heel.

Height 1

Height1 of the heel.

Height 2

Height2 of the heel.

Radius

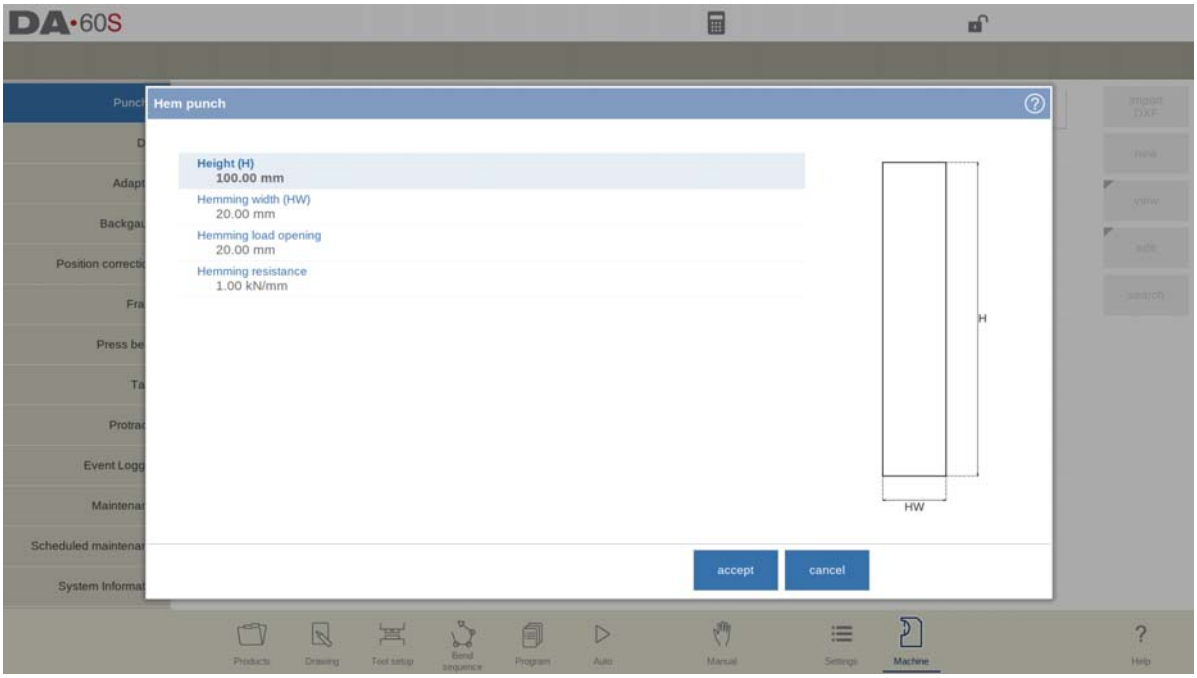
The radius in the heel segment.

Height 1 and height 2 do not influence the tool height you have programmed before.

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 drawing facilities.

10.2.4. 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

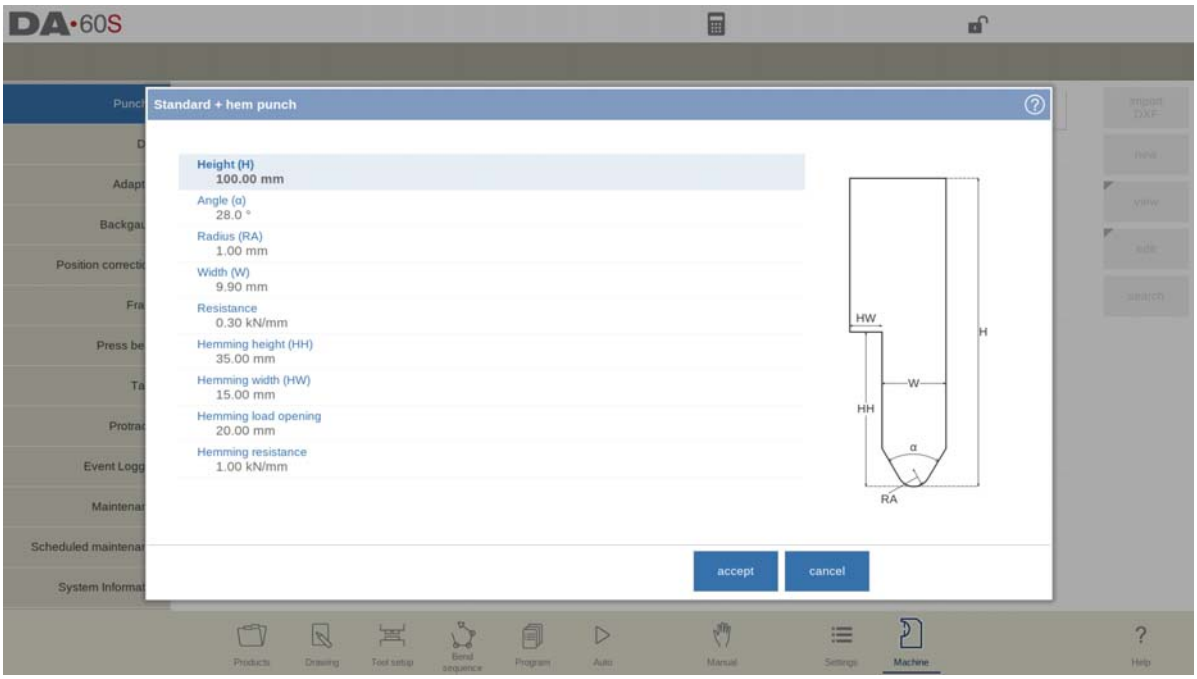
Depending on the construction of your machine you can program here an opening position for your punch at which position you can put in your product to hem the particular bend. The opening position will also take twice the sheet thickness into account.

Hemming resistance

Maximum allowable force on the tool during hemming.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.

10.2.5. 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.

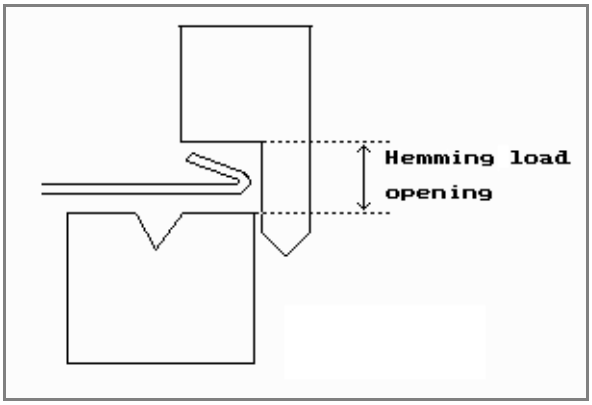
Hemming width

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

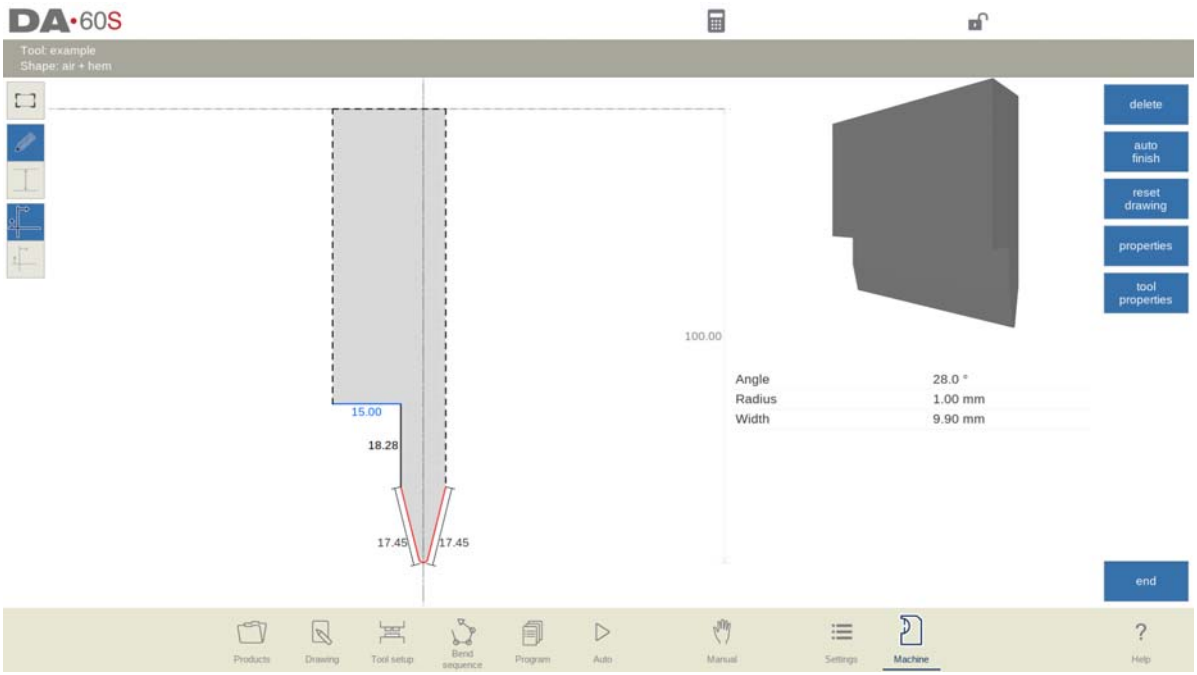
account.



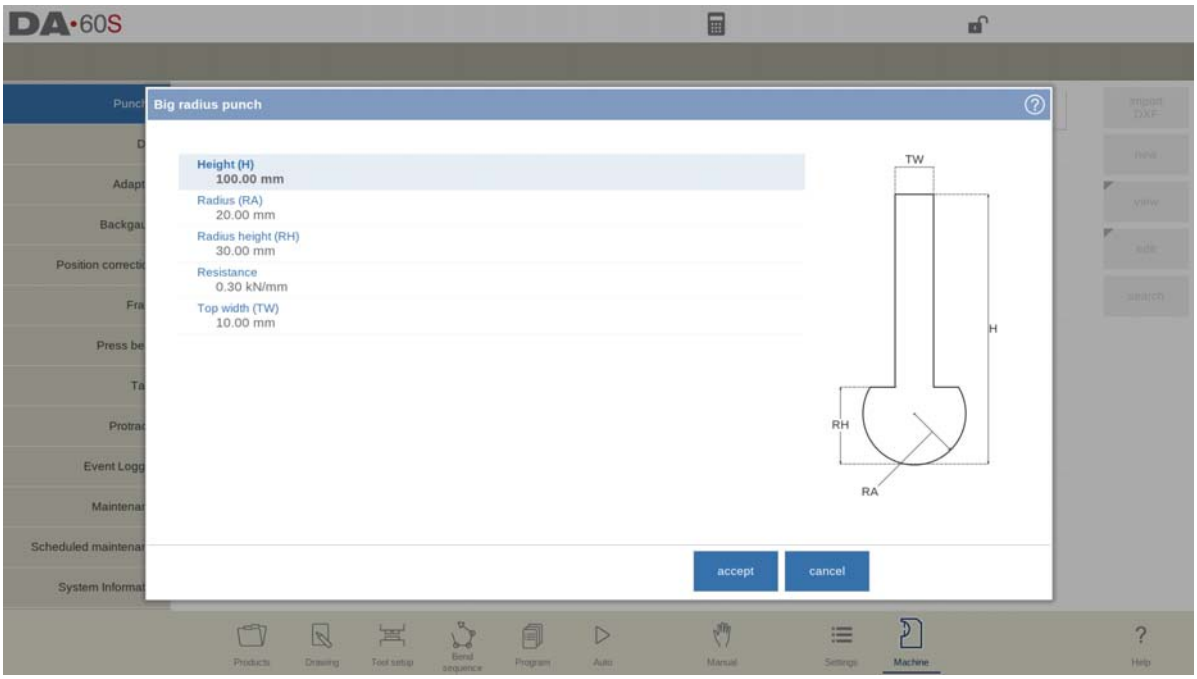
Hemming resistance

Maximum allowable force on the tool during hemming.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.



10.2.6. Big radius punch



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

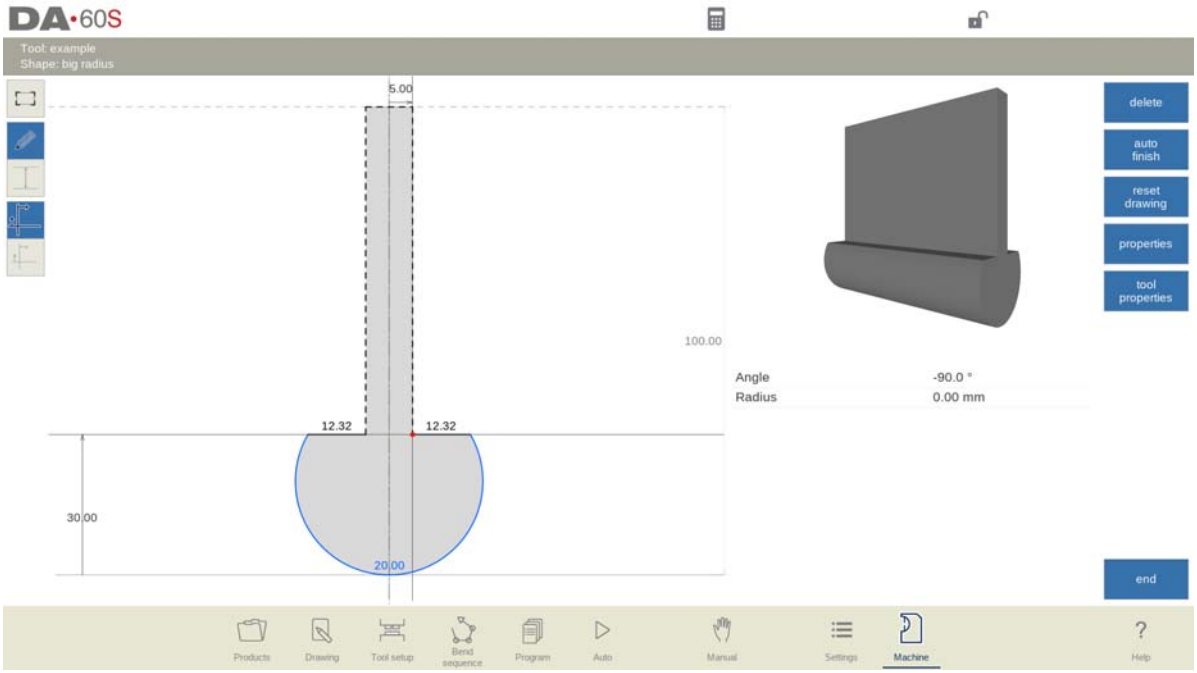
Radius
The radius of the punch tip.

Radius height
The height of the big radius part of the tool.

Resistance
Maximum allowable force on the tool.

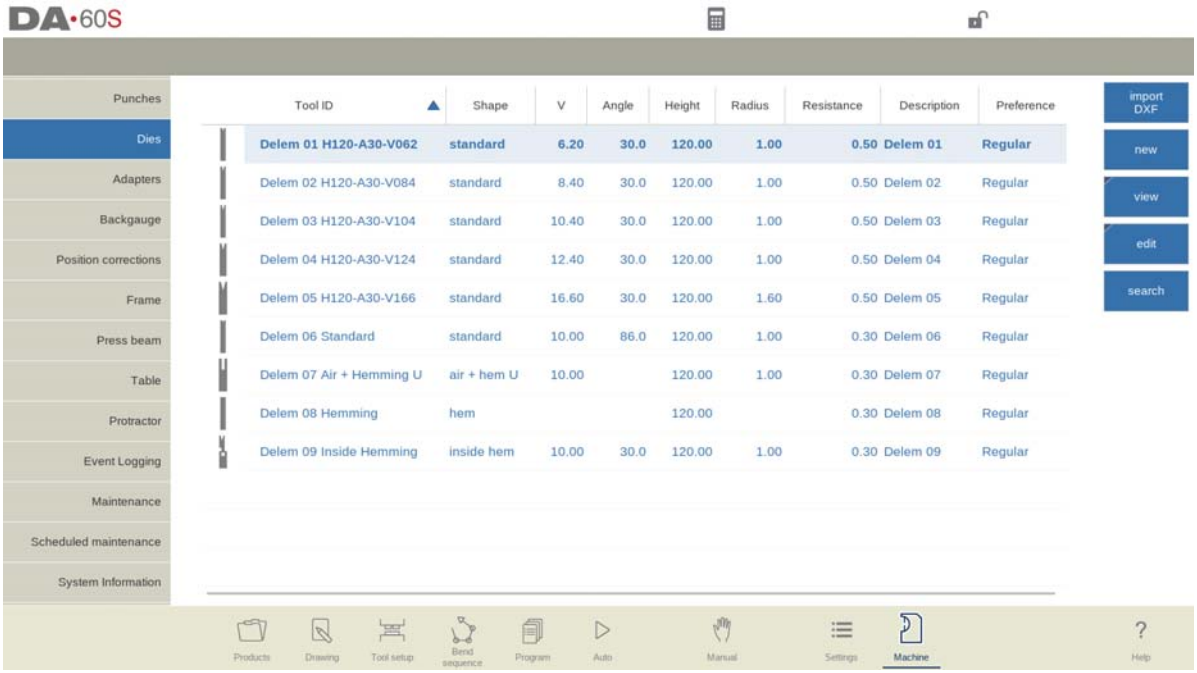
Top width
The width of the tool on the top side of the punch.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.


DA·60S
Tool: example
Shape: big radius
5.00
100.00
12.32
12.32
30.00
20.00
Angle
-90.0 °
Radius
0.00 mm
delete
auto finish
reset drawing
properties
tool properties
end
Products
Drawing
Tool setup
Bend sequence
Program
Auto
Manual
Settings
Machine
Help

10.3. Programming of Dies

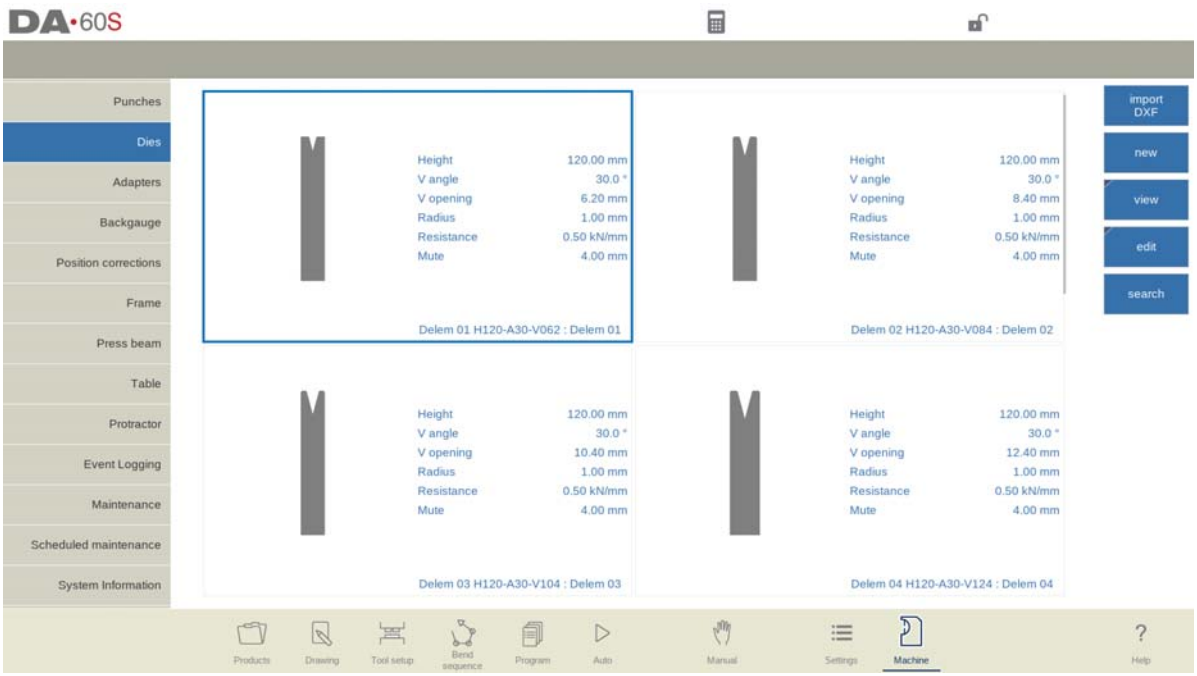
In this tab, the dies used in the machine can be programmed. New dies can be added; existing dies can be edited, copied, renamed and deleted.



10.3.1. View

In the main page, a list of available dies is shown. By using the View function, similar to Products mode, different views can be selected. Next to the default view Normal also Graphical is available.

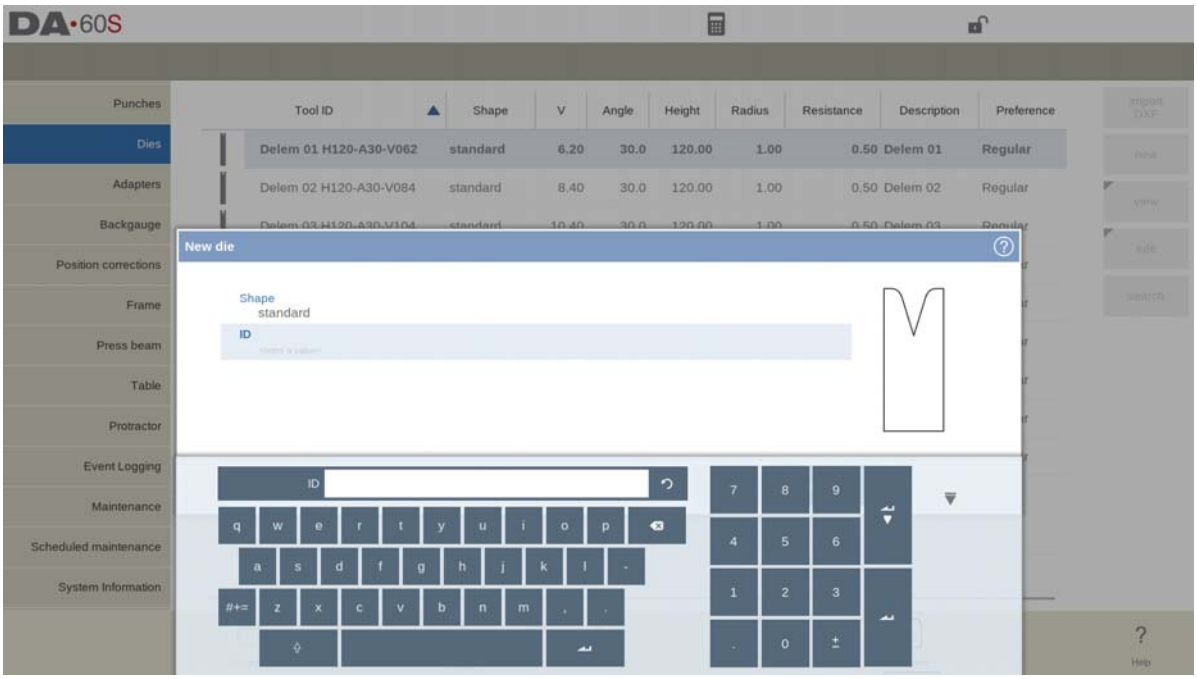
Graphical directory



In Graphical the geometry of the tools is shown as well as the main properties.

10.3.2. 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 and drawing 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.
- Multi-V Die for dies with multiple V and or U openings.
- Vario-V Die for dies with variable V or U opening (only available if a Vario-V system is present).
- Multi-V Inside Hem Bend Die for dies with multiple V or U openings including an integrated hemming system.
- Doorframe Hem Bend Die for dies specifically for doorframe production, V-die combined with a separate inside hemming feature.
- Wingbend Die for special dies with rotating radius segments.

ID

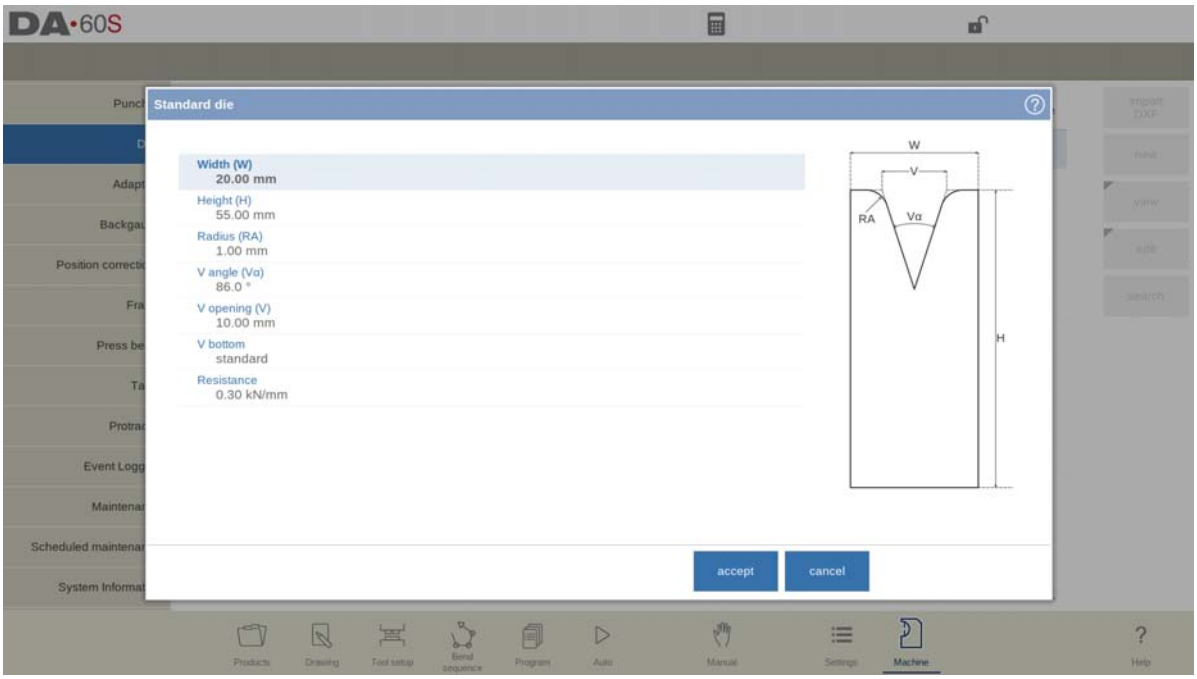
A unique name or number to identify the tool. The maximum length is 25 characters.

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 starting with the initial 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.

DXF import (only available when DXF option has been installed)
The shape of a die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

10.3.3. 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

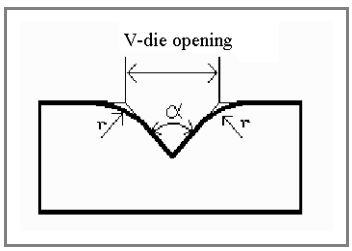
The radius of the edges of the V-opening.

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

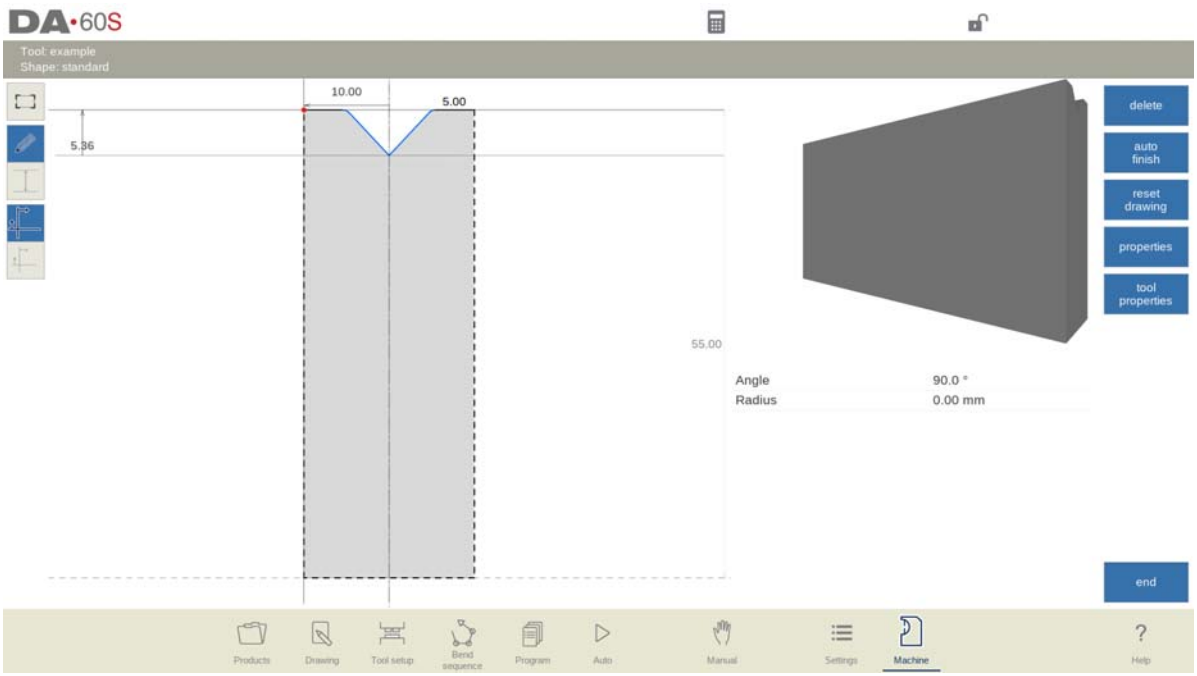
Maximum allowable force on the tool.

Orientation of the die on the screen

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

Drawing

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.



Mount point (optionally programmable)

Within the drawing one also finds the mount point of the die. This point, indicated with the triangle arrow, shows at what point the die will be connected and placed on the table or in an adapter. For both normal and turned a separate mount point is available. Mount points can also be found in adapters and in the table. When this function is not enabled, the indicators are not shown.

Following functions are available while drawing

Delete Line

To delete a line segment.

Change Height

To change the height dimension of the tool.

Auto Finish
Finishing the tool outline to the top of the tool automatically.

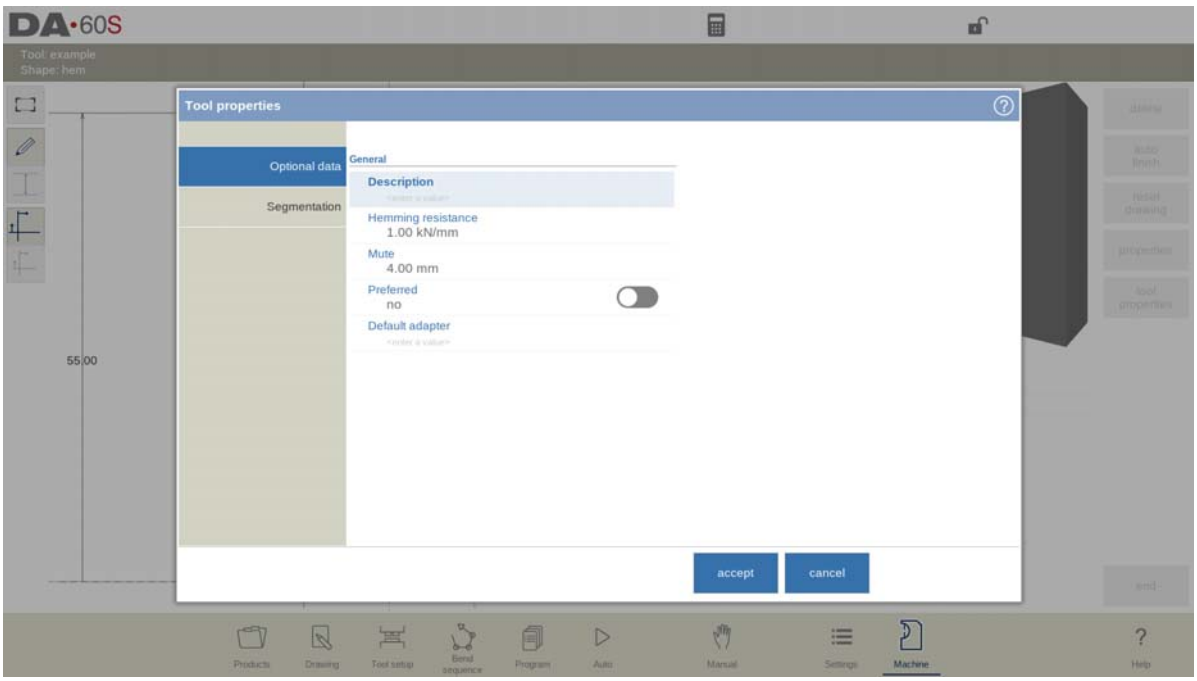
Reset Drawing
To reset the programmed drawing of the tool till the basic, initial shape, when creating a new die.

Reload Drawing
To reload the programmed drawing of the tool till the basic, initial shape, when changing an existing die.

Properties
To change the specific properties of the line or angle, add or remove a radius, change the length, etc. It is e.g. possible to add a radius in the outline of the tool.

Hemming can be part of the property of lines. Within the shape of a tool a surface can be appointed as a hemming surface. This will enable the tool for hemming operation.

Tool Properties
To change the generic tool data and description.

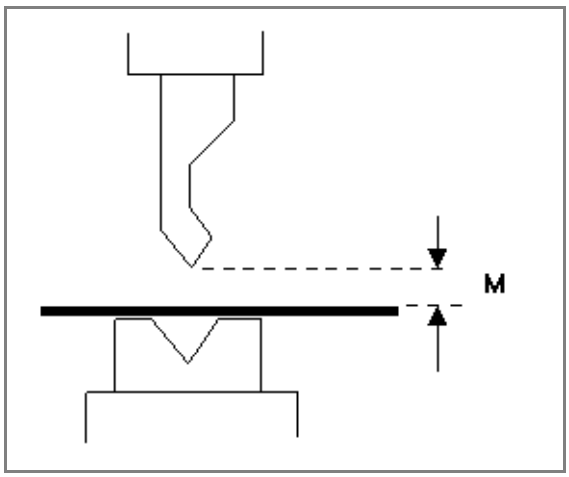


Description
A name or description of this tool. The maximum length is 25 characters. This description has already been entered in the beginning defining the tool, but can be edited in this field. The description is listed in the tool overview of the library.

Resistance
Maximum allowable force on the tool.

Mute

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



Preferred

For automatic tool selection, based on equivalent fitting of a tool for a specific bend, this parameter can sign a tool to be preferred above others. The automatic tool selection will prevail the tools being signed as preferred.

Default adapter

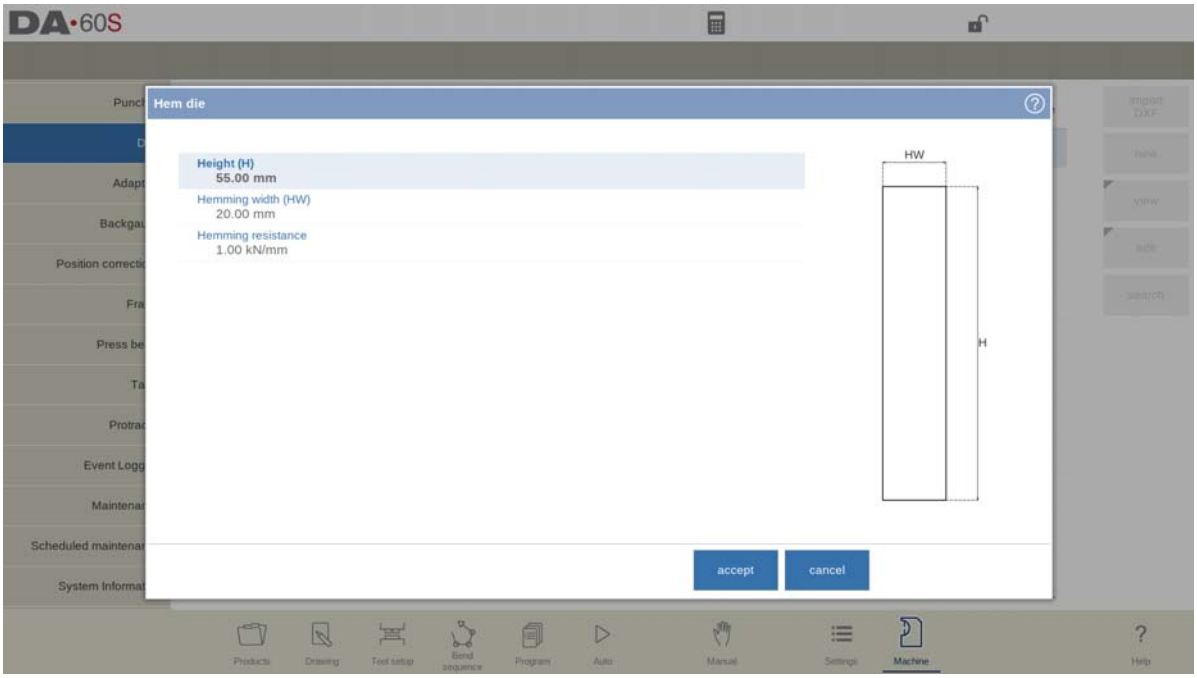
When a tool is used with a specific adapter and this combination is used commonly, the adapter can be set as default in the properties of the tool. Every time the tool is selected, automatically also the adapter will be loaded in the tool setup.

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 drawing facilities.

A round upper side can be created by dragging the end of the surface, connected to the radius, downwards.

10.3.4. Hem bend die



Height

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

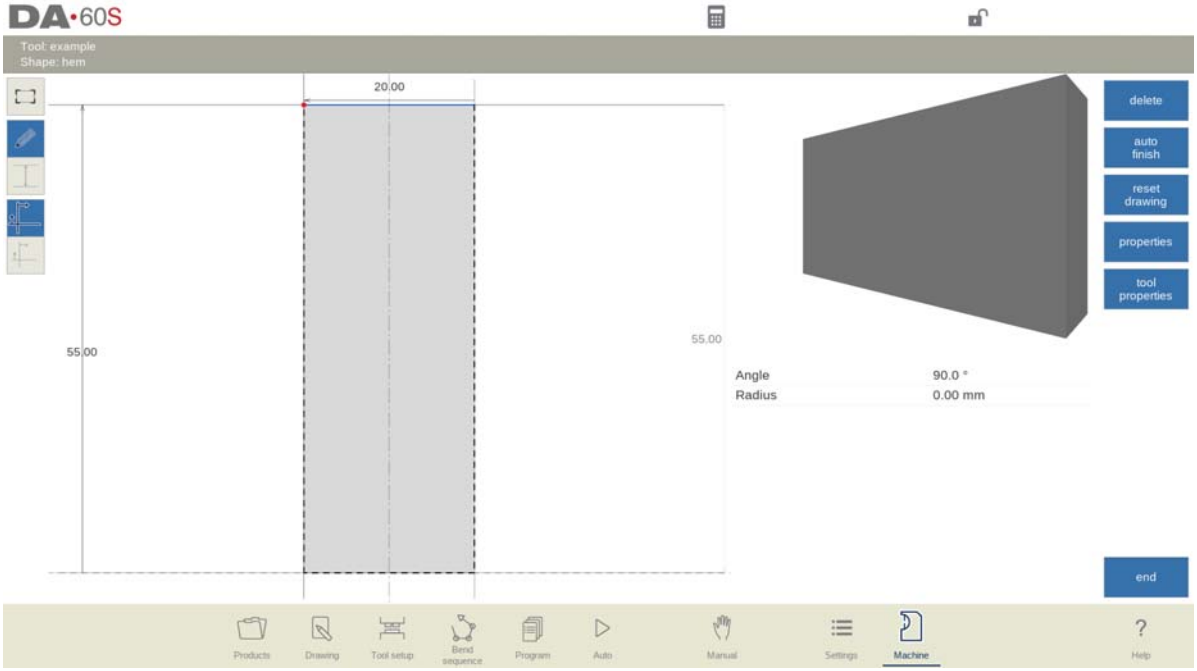
Hemming width

The width of the tool to program.

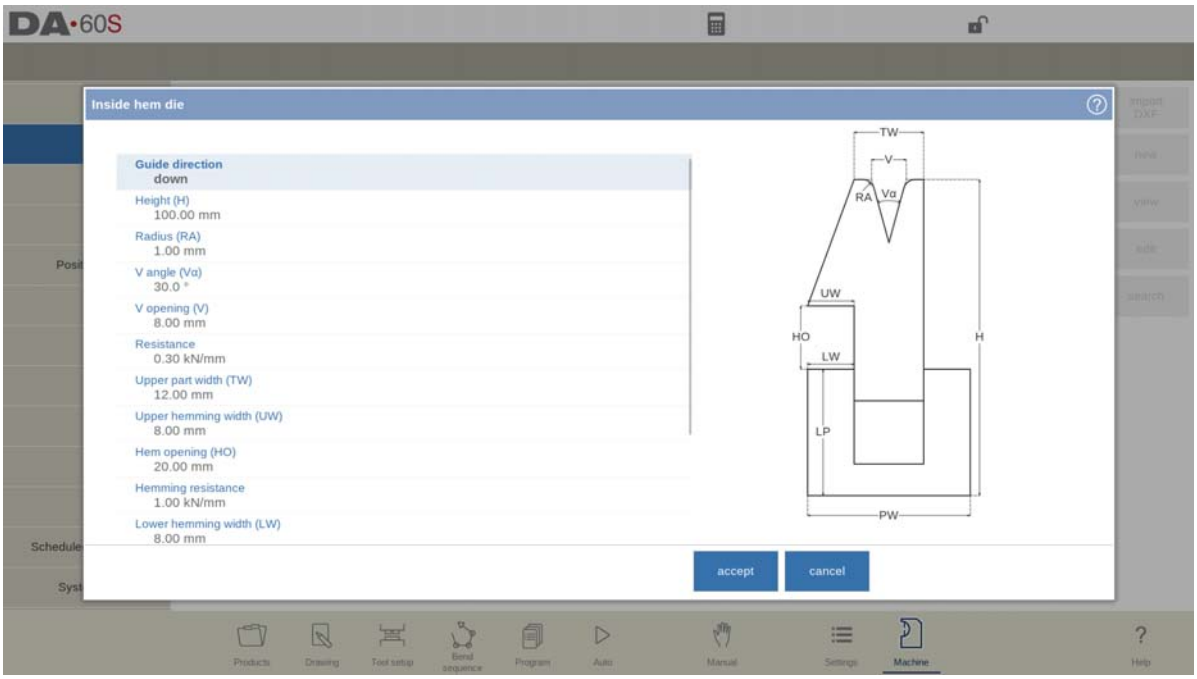
Hemming resistance

Maximum allowable force on the tool during hemming.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.


DA·60S
Tool: example
Shape: hem
20.00
55.00
55.00
delete
auto finish
reset drawing
properties
tool properties
Angle
90.0 °
Radius
0.00 mm
end
Products
Drawing
Tool setup
Bend sequence
Program
Auto
Manual
Settings
Machine
Help

10.3.5. Inside hem bend die



Guide direction

Defines the mechanical construction of the inside hem 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.

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.

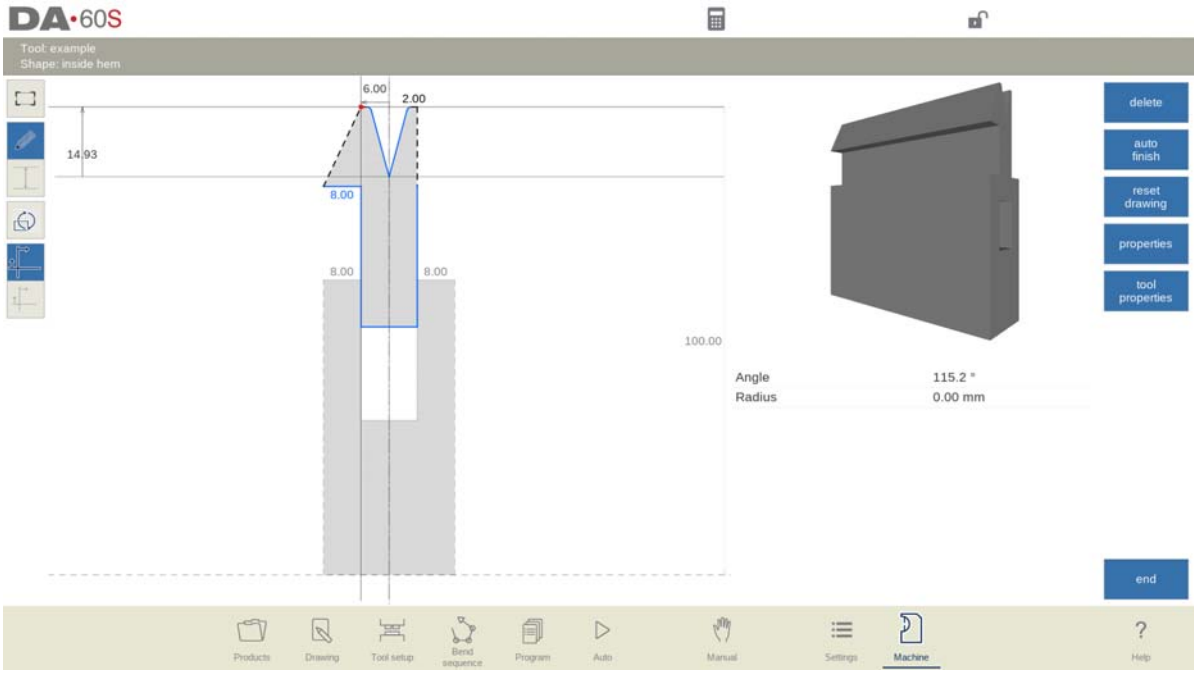
Adapt decompression

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).

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

Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.

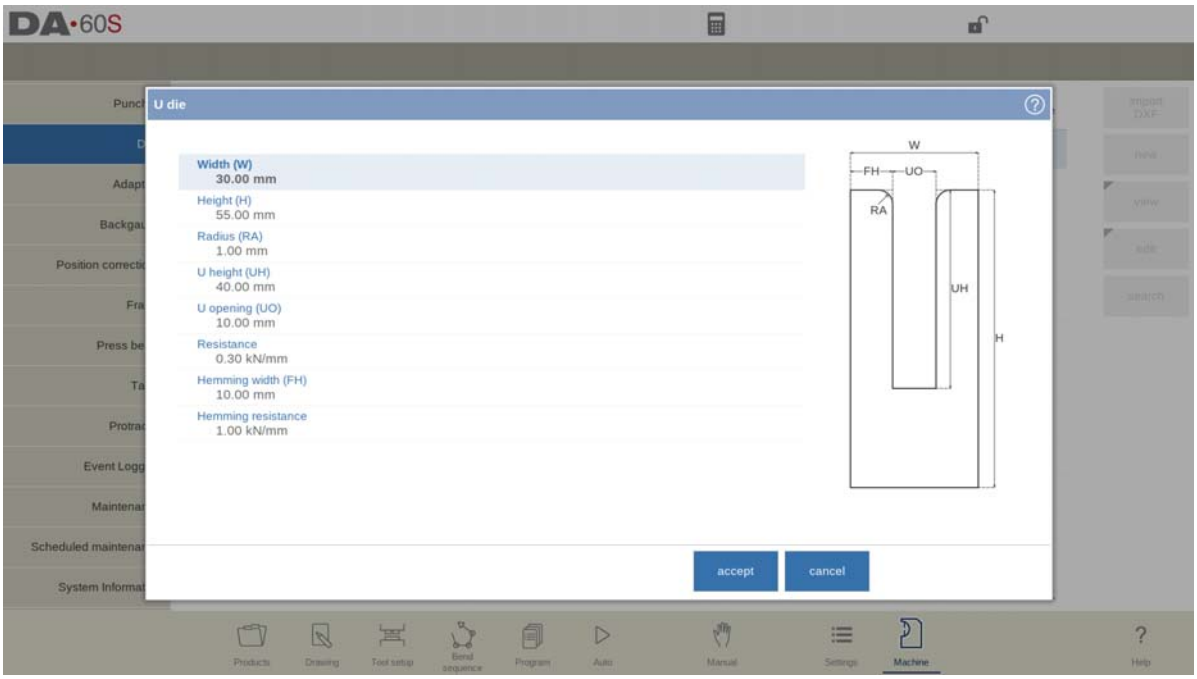


DXF import (only available when DXF option has been installed)

The shape of an inside hem die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Important for the import of an inside hem die is that the body (lower part) of the die must be in layer 0 within the DXF drawing and the moving part (upper part) should be in a different layer. If this layer is called ‘inside hem’ the importer can automatically select the correct tool type; otherwise it has to selected manually. Both parts in the DXF need to be closed contours.

10.3.6. 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.

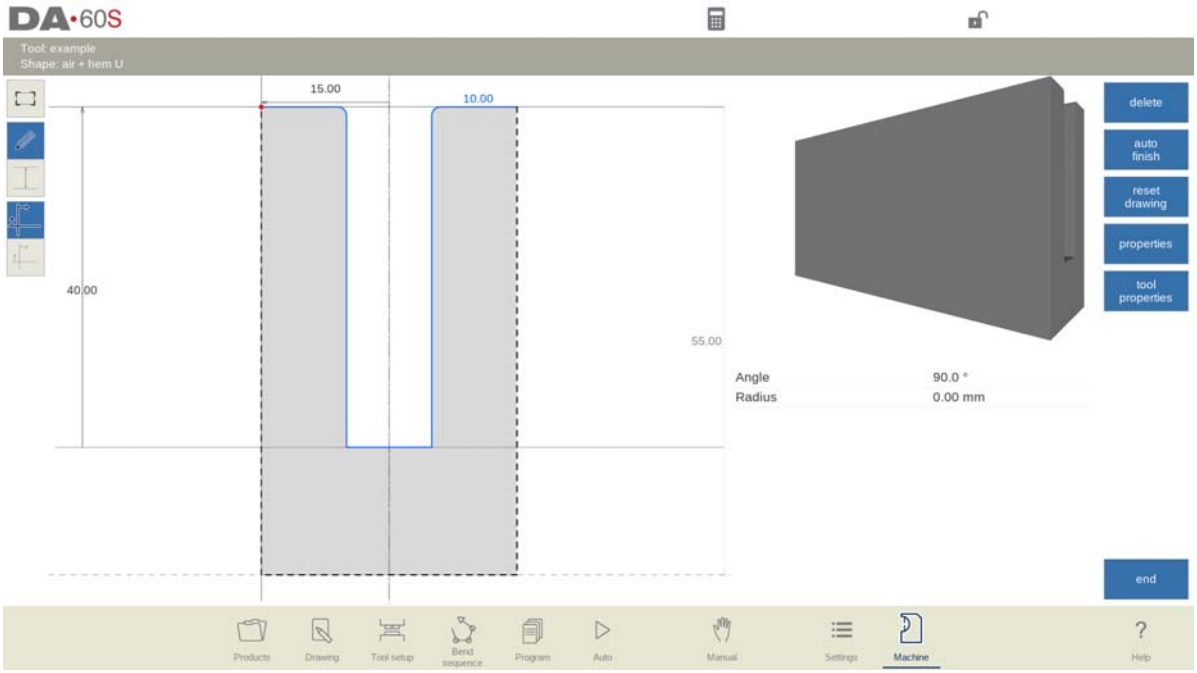
U opening
The width of the U-opening of the die.

Resistance
Maximum allowable force on the tool.

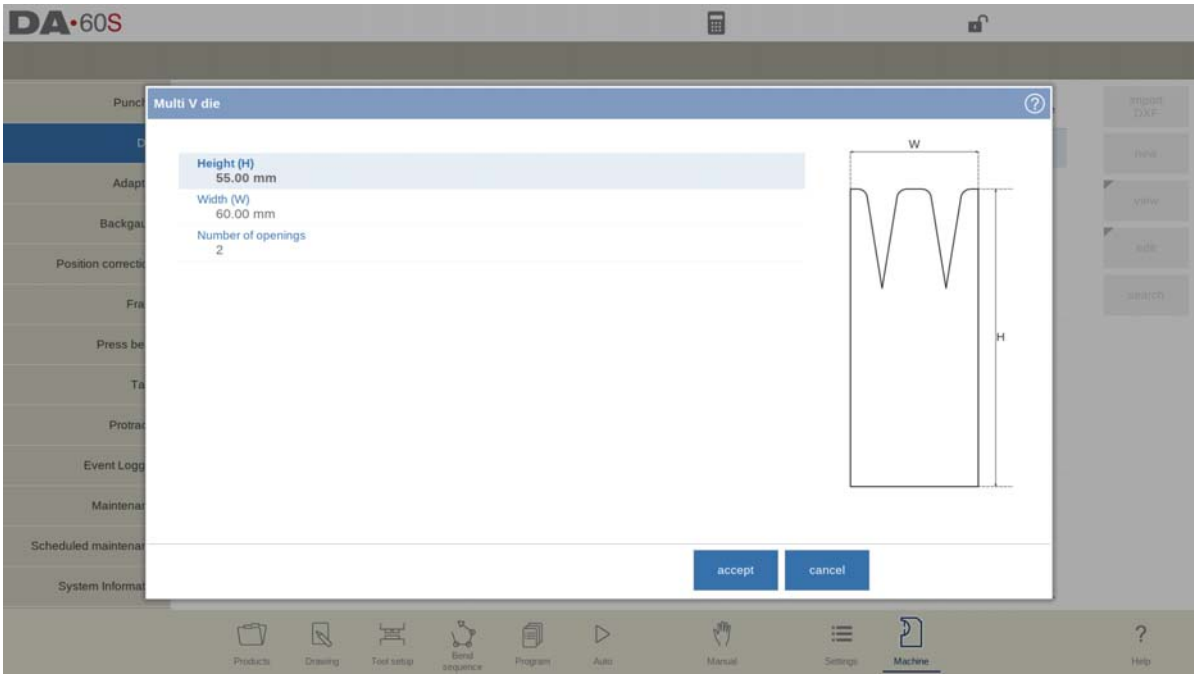
Hemming width
The front width of the die, meant as support for the hem bend.

Hemming resistance
Maximum allowable force on the tool during hemming.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.


DA•60S
Tool: example
Shape: air + hem U
15.00
10.00
40.00
55.00
delete
auto finish
reset drawing
properties
tool properties
Angle
90.0 °
Radius
0.00 mm
end
Products
Drawing
Tool setup
Bend sequence
Program
Auto
Manual
Settings
Machine
Help

10.3.7. Multi V die



Height

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

Width

The width of the tool to program.

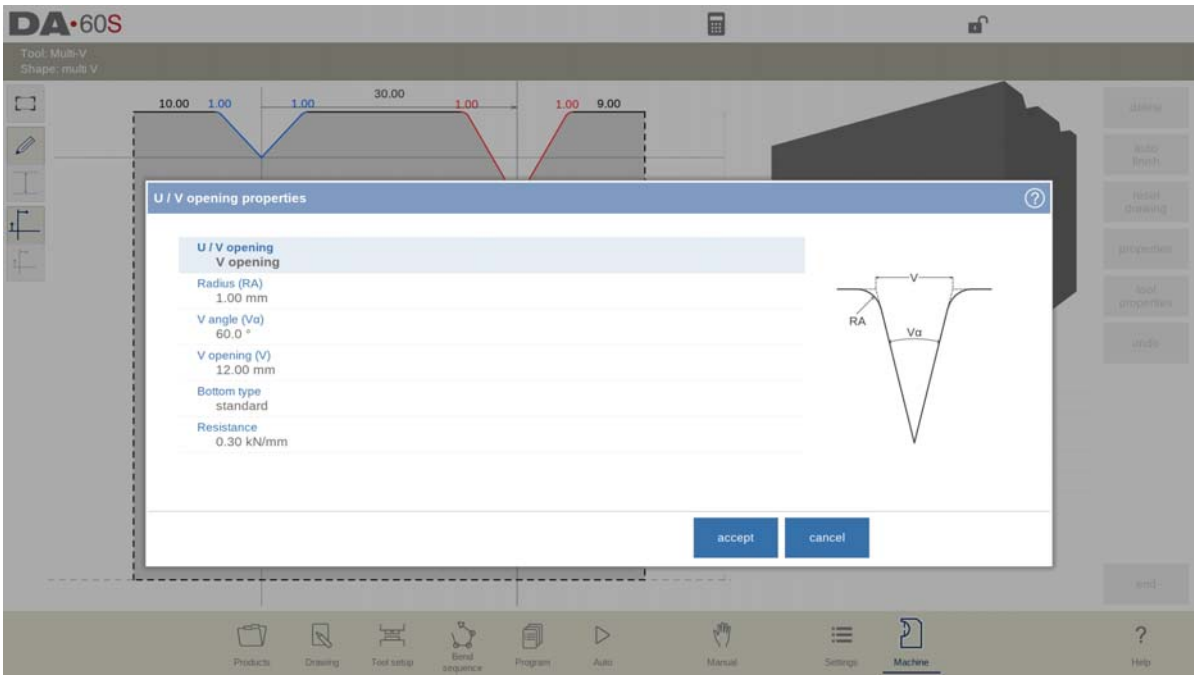
Number of openings

Number of V or U openings.

U / V opening properties

After specifying the number of tool V or U openings, the specific tool features must be programmed under Properties when the V or U opening is selected.

Each U or V opening's properties can be programmed. Tool resistance can be overall or specifically programmed.



U / V opening

Type of V or U opening.

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

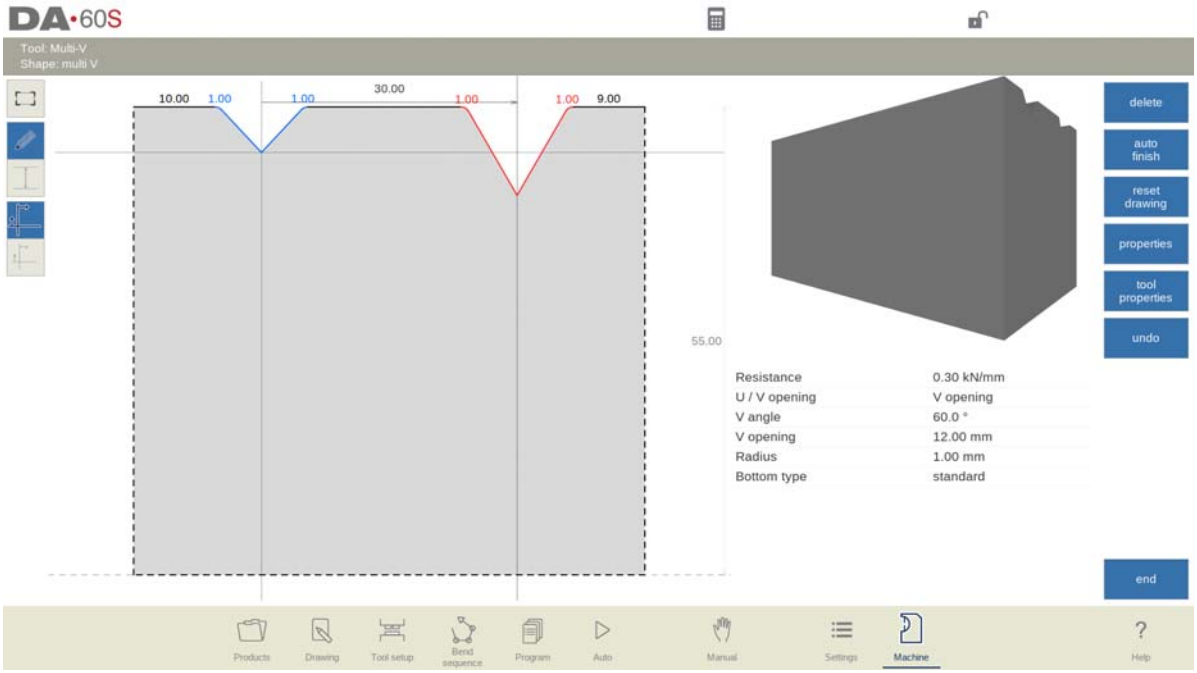
Herewith the different possible bottoms inside the V opening can be defined:

- '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'.

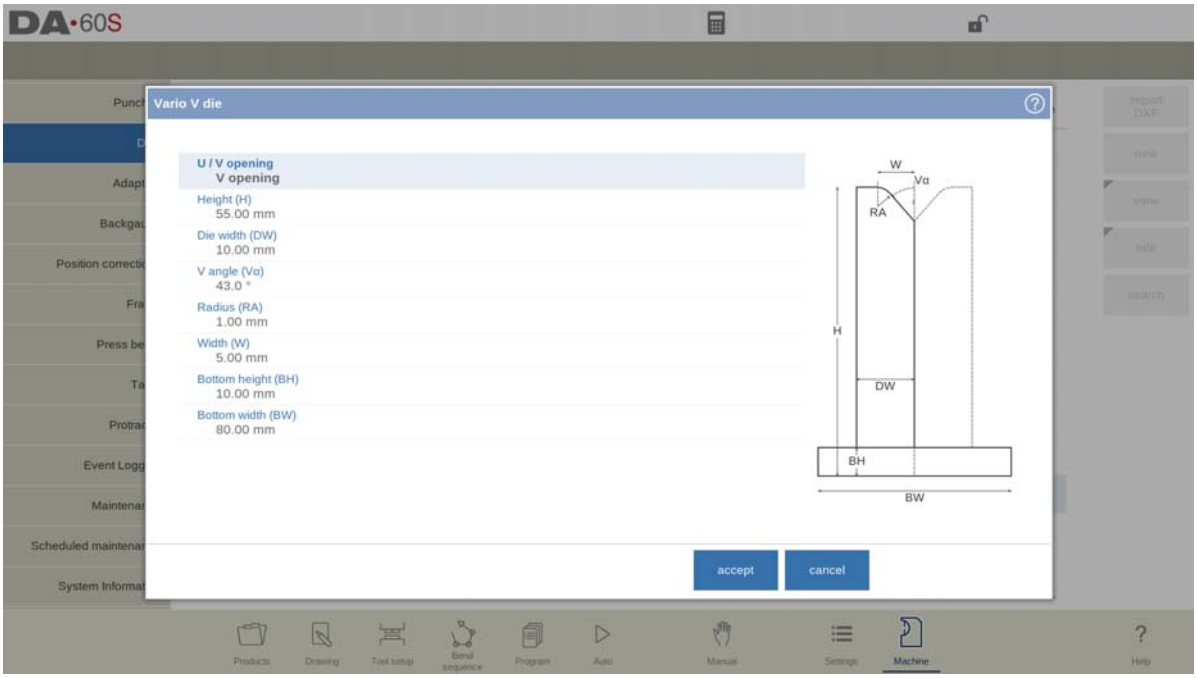
Resistance

Maximum allowable force on the tool, specifically for this V or U opening. In case this is equal for all, no resistance should be filled out. The programmed value under Tool Properties will do.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.


DA·60S
Tool: Multi-V
Shape: multi V
10.00
1.00
1.00
30.00
1.00
1.00
9.00
55.00
delete
auto finish
reset drawing
properties
tool properties
undo
Resistance 0.30 kN/mm
U / V opening V opening
V angle 60.0 °
V opening 12.00 mm
Radius 1.00 mm
Bottom type standard
end
Products
Drawing
Tool setup
Bend sequence
Program
Auto
Manual
Settings
Machine
Help

10.3.8. Vario V die (only available if a Vario-V system is present)



U / V opening
Type of V or U opening.

Height
The total height of the tool. Important: this height value will be used in the bend depth calculation. In the Vario V die this includes the bottom part of the system.

Die width
The width of a single Vario V side (i.e. half of the width of the die).

V angle
Half of the V angle of the die.

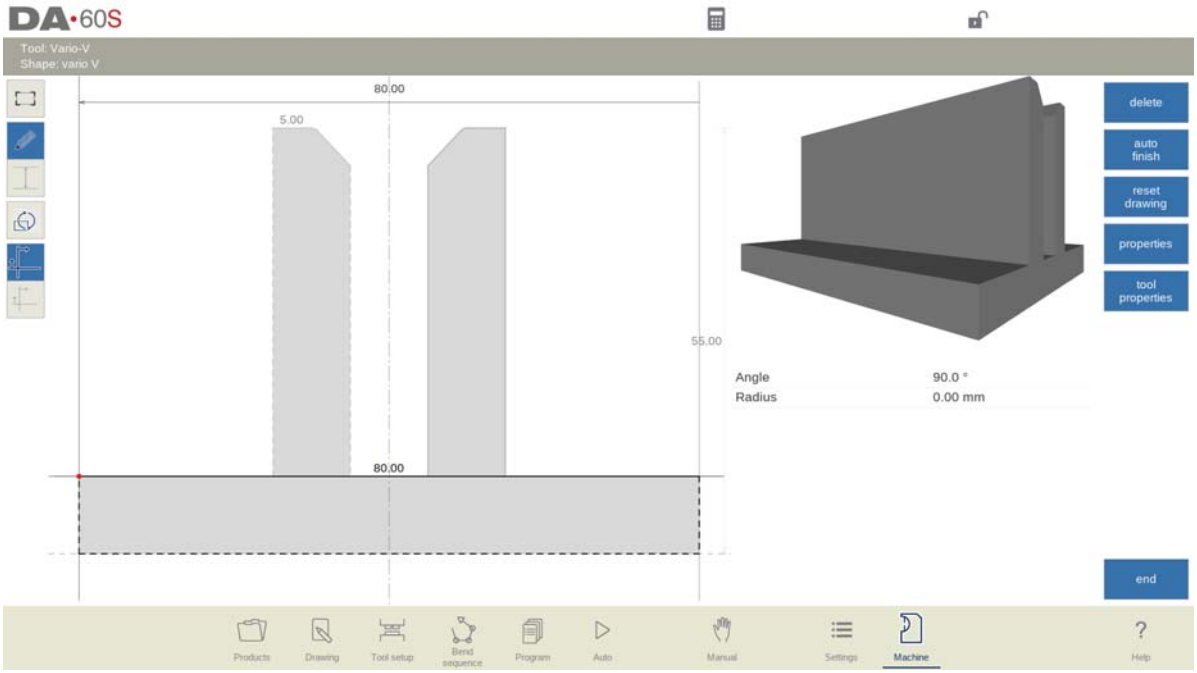
Radius
The radius of the edges of the U or V opening.

Width
The width of the V angle of a single Vario V side (i.e. half of the minimum V-opening).

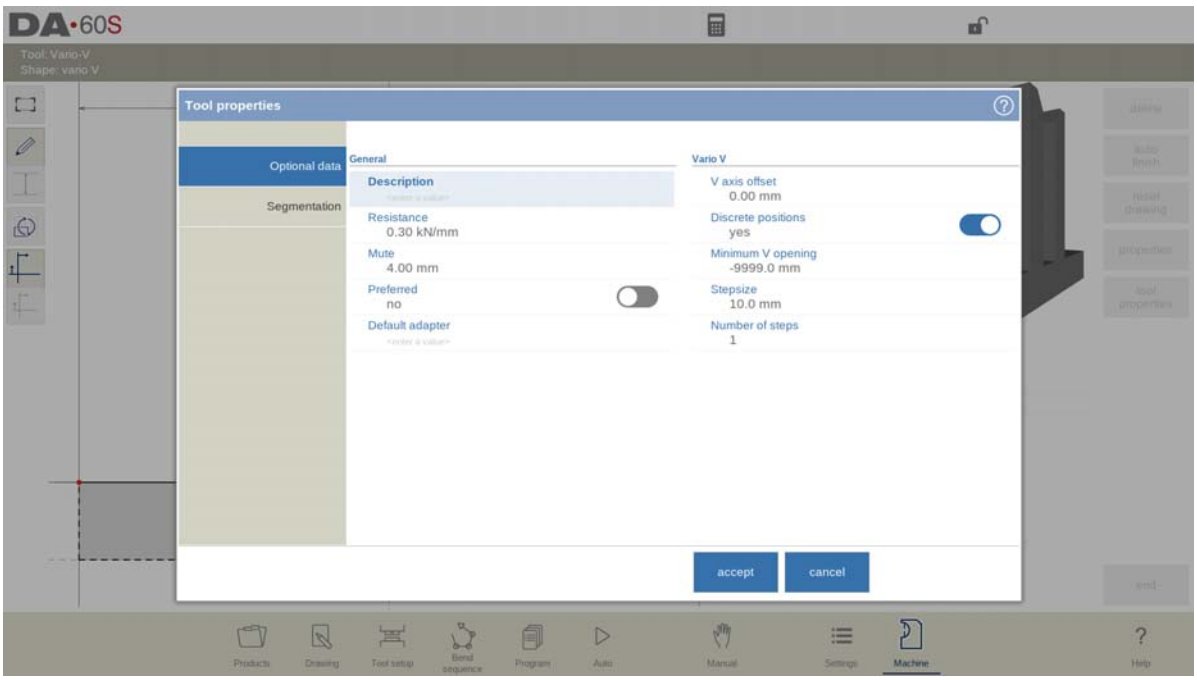
Bottom height
The height of the bottom part of the die system.

Bottom width
The width of the bottom part of the die system.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.



Tool properties



Description

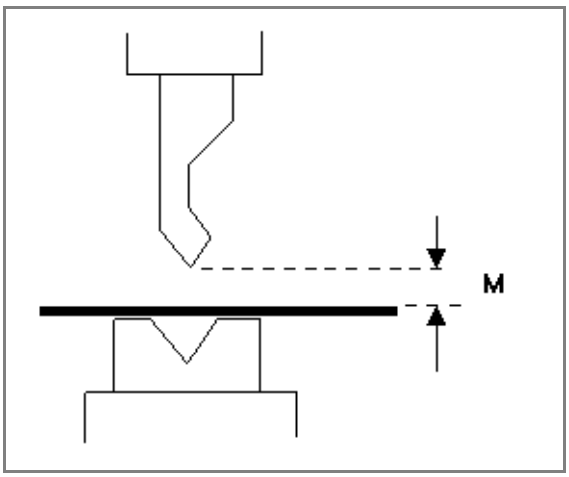
A name or description of this tool. The maximum length is 25 characters. This description has already been entered in the beginning defining the tool, but can be edited in this field. The description is listed in the tool overview of the library.

Resistance

Maximum allowable force on the tool.

Mute

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



V axis offset

The width difference, in case of identical V-axis positions, between the V-opening of the current Vario V system and the Vario V system with which the machine was commissioned and calibrated.

Discrete positions

Indicates if the V-opening can have any value (no) or only a value from a fixed interval (yes). Only available in case of an analog Vario-V system.

Minimum V opening

The minimum V opening in case of the smallest discrete position.

Stepsize

The interval size between the possible V opening values in case of discrete positions.

Number of steps

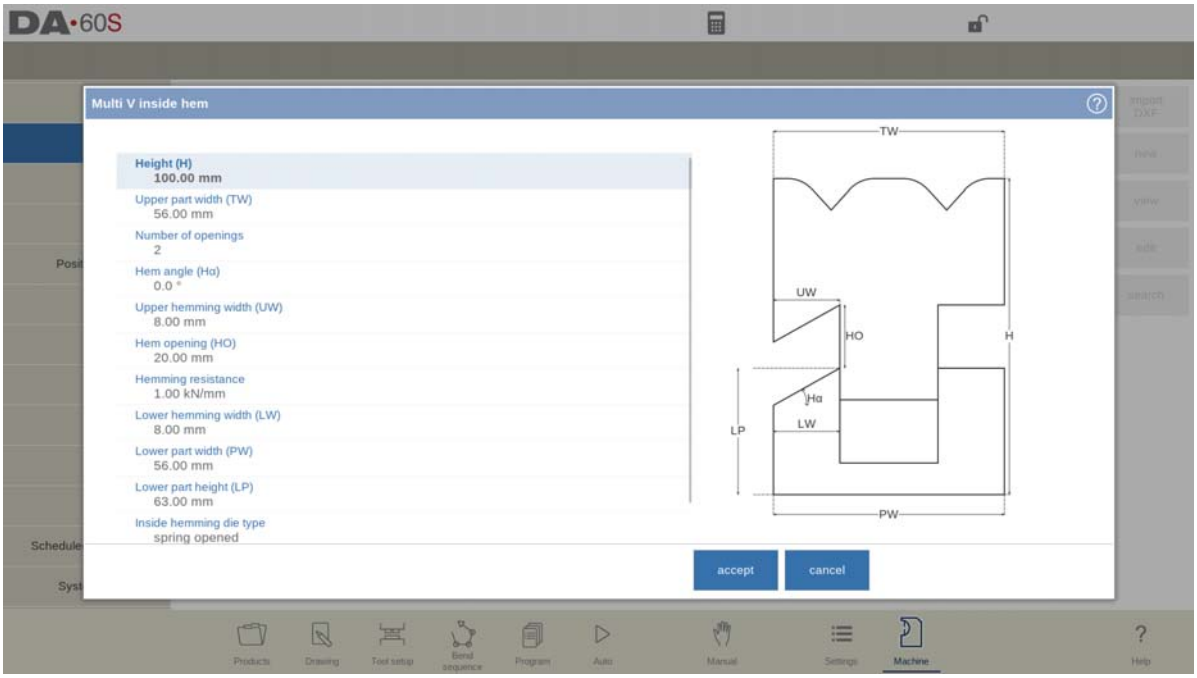
The number of possible V opening values in case of discrete positions.

DXF import (only available when DXF option has been installed)

The shape of a Vario V die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Important for the import of a Vario V die is that the body (lower part) of the die must be in layer 0 within the DXF drawing and the moving parts (upper parts) should be in a different layer. If this layer is called ‘Vario V’ the importer can automatically select the correct tool type; otherwise it has to selected manually. Both parts in the DXF need to be closed contours.

10.3.9. Multi-V inside hem die



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

Upper part width (TW)
The width of the upper part of the die.

Number of openings
Number of V or U openings.

Hem angle (Ha)
Angle of the hemming unit.

Upper hemming width (UW)
The width of the segment in the upper part of the die used for the hemming action.

Hem opening (HO)
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.

Lower hemming width (LW)
The width of the segment in the lower part of the die used for the hemming action.

Lower part width (PW)
The width of the lower part of the die.

Lower part height (LP)

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).

U / V opening properties

After specifying the number of tool V or U openings, the specific tool features must be programmed under Properties when the V or U opening is selected.

Each U or V opening's properties can be programmed. Tool resistance can be overall or specifically programmed.

U / V opening

Type of V or U opening.

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

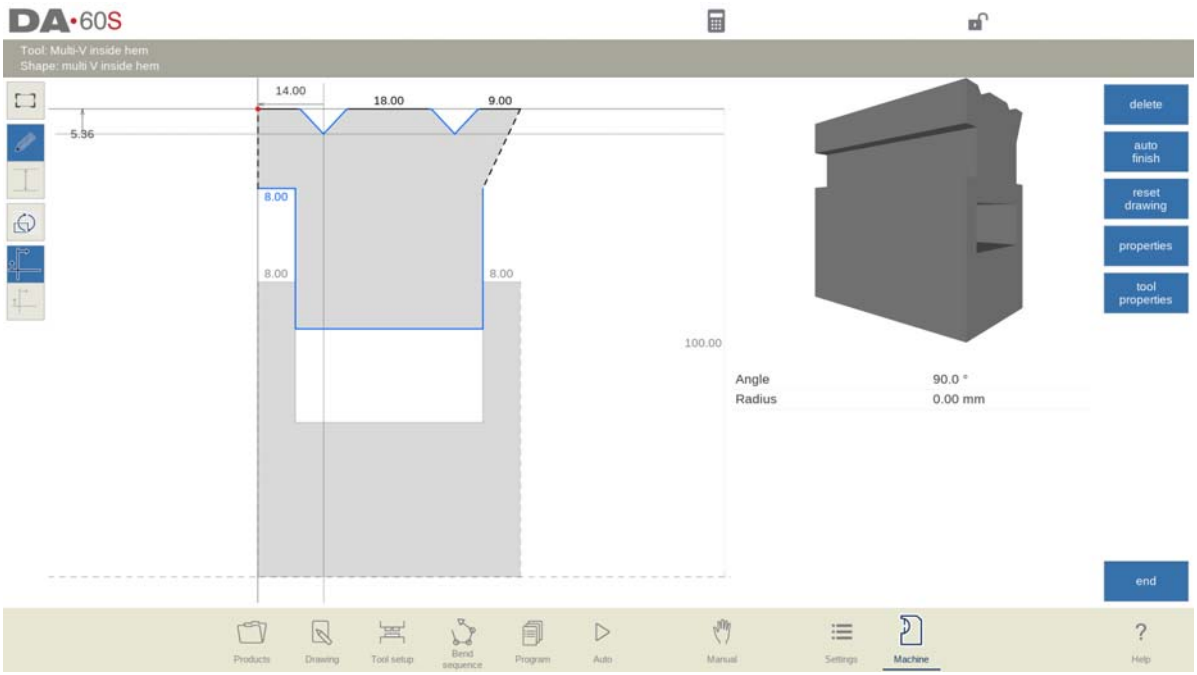
Herewith the different possible bottoms inside the V opening can be defined:

- '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, specifically for this V or U opening. In case this is equal for all, no resistance should be filled out. The programmed value under Tool Properties will do.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.

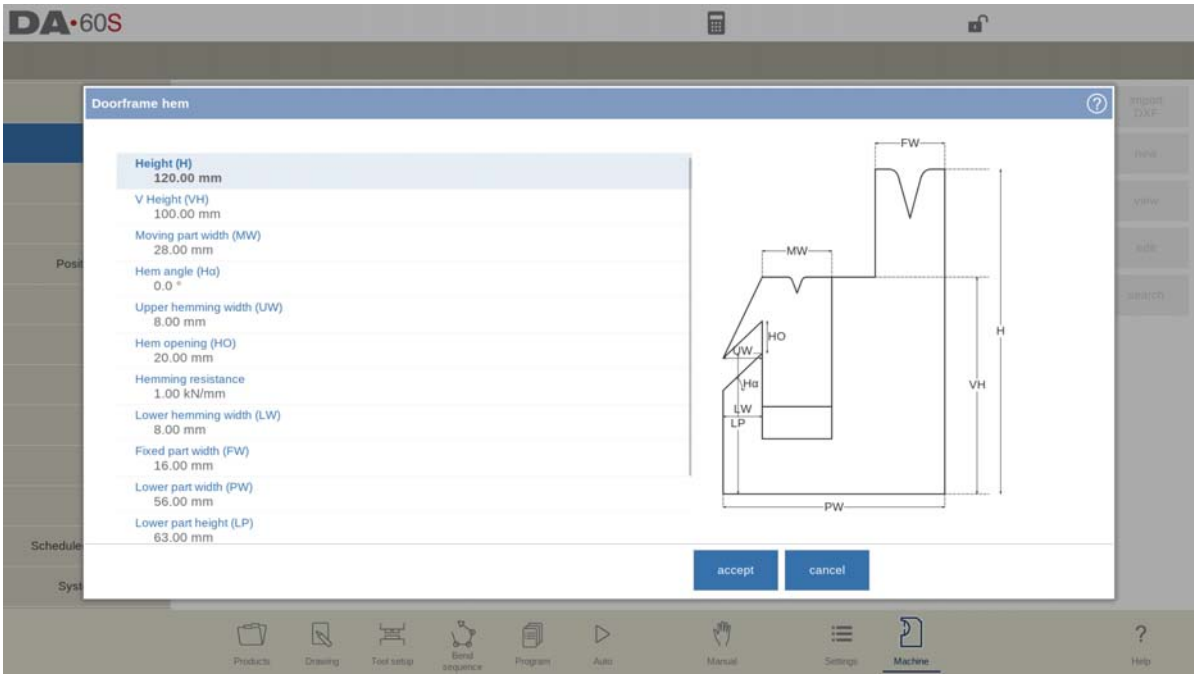


DXF import (only available when DXF option has been installed)

The shape of a multi V inside hem die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Important for the import of a multi V inside hem die is that the body (lower part) of the die must be in layer 0 within the DXF drawing and the moving part (upper part) should be in a different layer. If this layer is called ‘inside hem’ the importer can automatically select the correct tool type; otherwise it has to selected manually. Both parts in the DXF need to be closed contours.

10.3.10.Doorframe hem die



Height (H)

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

V Height (VH)

The height of the hemming unit. Important: this height value will be used for the hemming operation.

Moving part width (MW)

The width of the hemming unit/moving part of the die.

Hem angle (Ha)

Angle of the hemming unit.

Upper hemming width (UW)

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

Hem opening (HO)

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.

Lower hemming width (LW)

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

Fixed part width (FW)

The width of the fixed part of the die.

Upper part width
The width of the upper part of the die.

Lower part width (PW)
The width of the lower part of the die.

Lower part height (LP)
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).

V opening properties
Within the V-opening properties details can be programmed

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

V angle
The angle of the die.

V opening

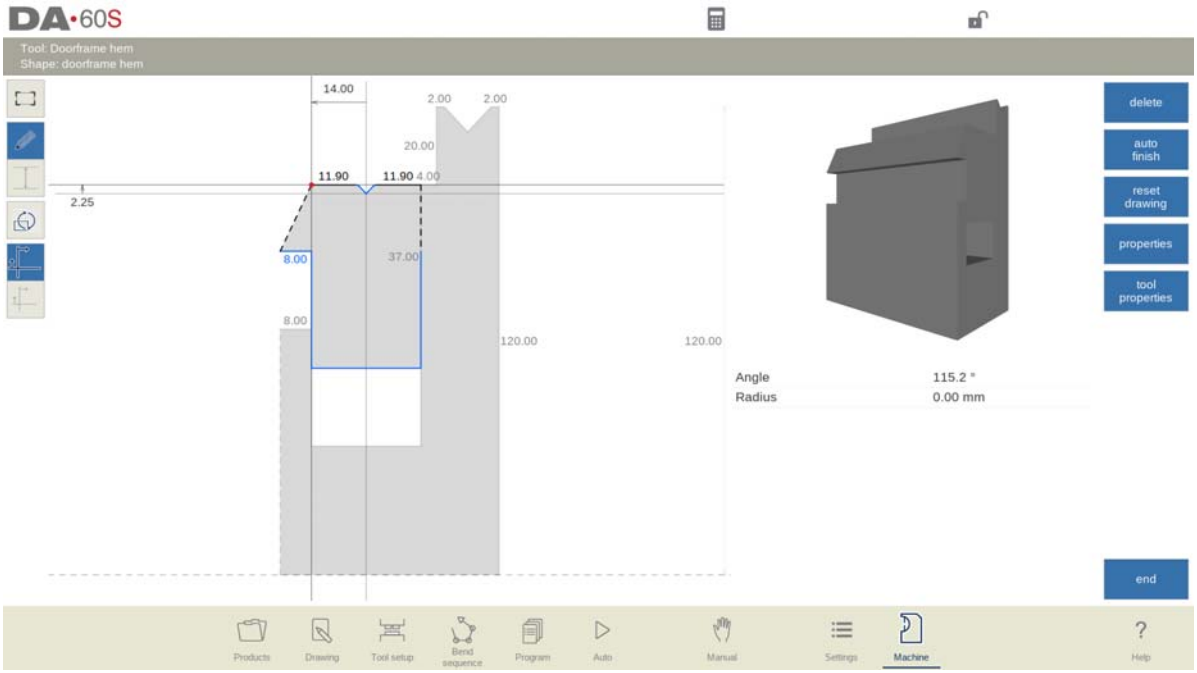
The V-opening of the die.

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'.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.

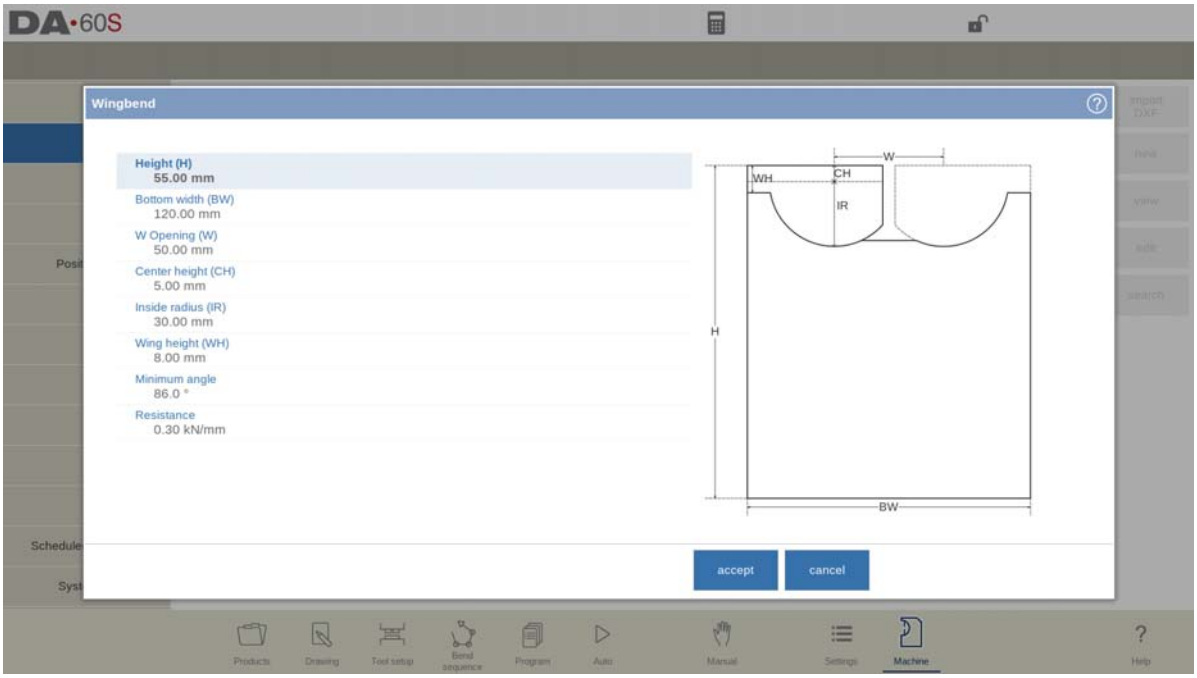


DXF import (only available when DXF option has been installed)

The shape of a doorframe hem die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Important for the import of a doorframe hem die is that the body (lower part) of the die must be in layer 0 within the DXF drawing and the moving part (upper part) should be in a different layer. If this layer is called ‘inside hem’ the importer can automatically select the correct tool type; otherwise it has to selected manually. Both parts in the DXF need to be closed contours.

10.3.11.Wingbend die



Wingbend dies, dies with moving elements in the V-opening, can be programmed using the wingbend die preset. Specific parameters have to be filled out to establish the correct functioning of the system. As for all other templates the basic shape of the die can be freely edited upon choice.

Most important parameters for this type of die are the distance from the rotation point to the top of the die and the radius information of the two wings. Others as the Wing width are important for the retract calculations. The features of the die as well as the moving elements can be edited as part of their specific features.

Height (H)
The total height of the tool including the possibly extending wings. Important: this height value will be used in the bend depth calculation.

Bottom width (BW)
The width of the tool to program.

W Opening (W)
Distance between the rotation points.

Center height (CH)
Height of the wing above the inner rotation point.

Inside radius (IR)
Radius of the wing from its inner rotation point.

Wing height (WH)
The height of the wing within the tool height definition (part of the tool height moving).

Minimum angle

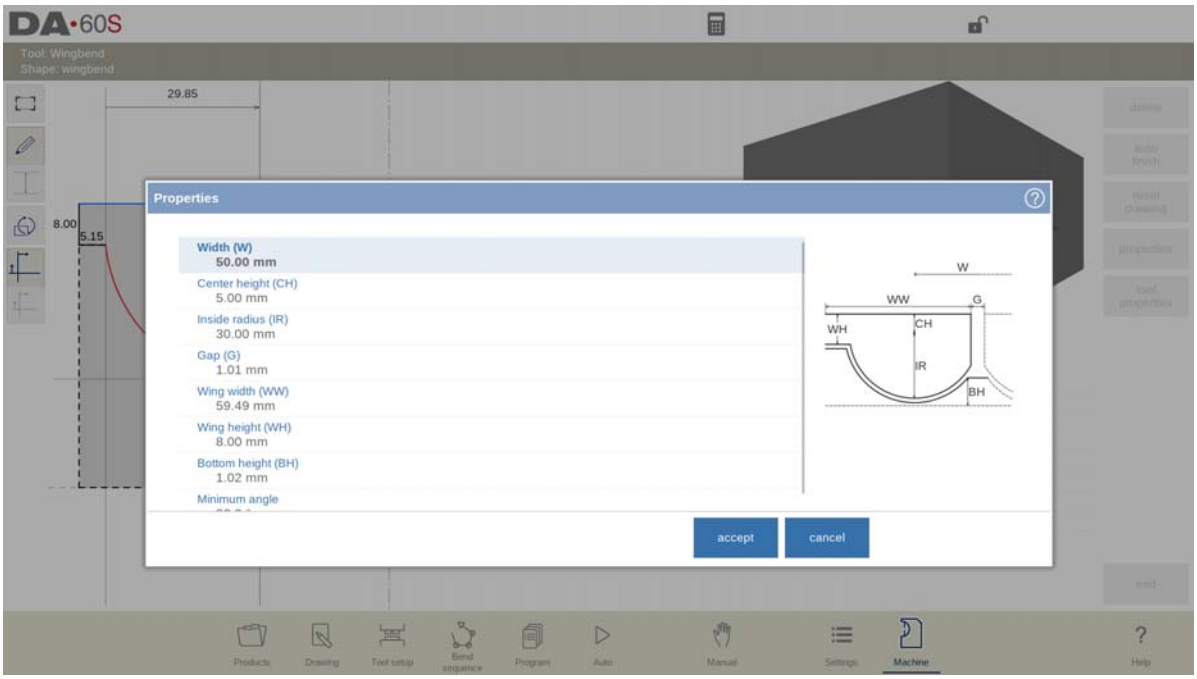
Smallest angle which can be bent with this wingbend tool.

Resistance

Maximum allowable force on the tool.

Wingbend properties

Within the wingbend tool definition the tool properties can be editted. When selecting the wing feature one can use Properties to view or modify.The Contour select button toggles over the multiple contours in the wingbend shape.



Width (W)

Distance between the rotation points.

Center height (CH)

Height of the wing above the inner rotation point.

Inside radius (IR)

Radius of the wing from its inner rotation point.

Gap (G)

Distance between both wings.

Wing width (WW)

Width of the wingtop.

Wing height (WH)

The height of the wing within the tool height definition (part of the tool height moving).

Bottom heigth (BH)

Height of the middle part representing the depth of the V-opening.

Minimum angle

Smallest angle which can be bent with this wingbend tool.

After entering these typical values you can create the tool drawing with the drawing facilities. Drawing a tool profile is done by entering angle values and line length values. Also the Touch drawing tools are available as with the product drawing method.

DXF import (only available when DXF option has been installed)

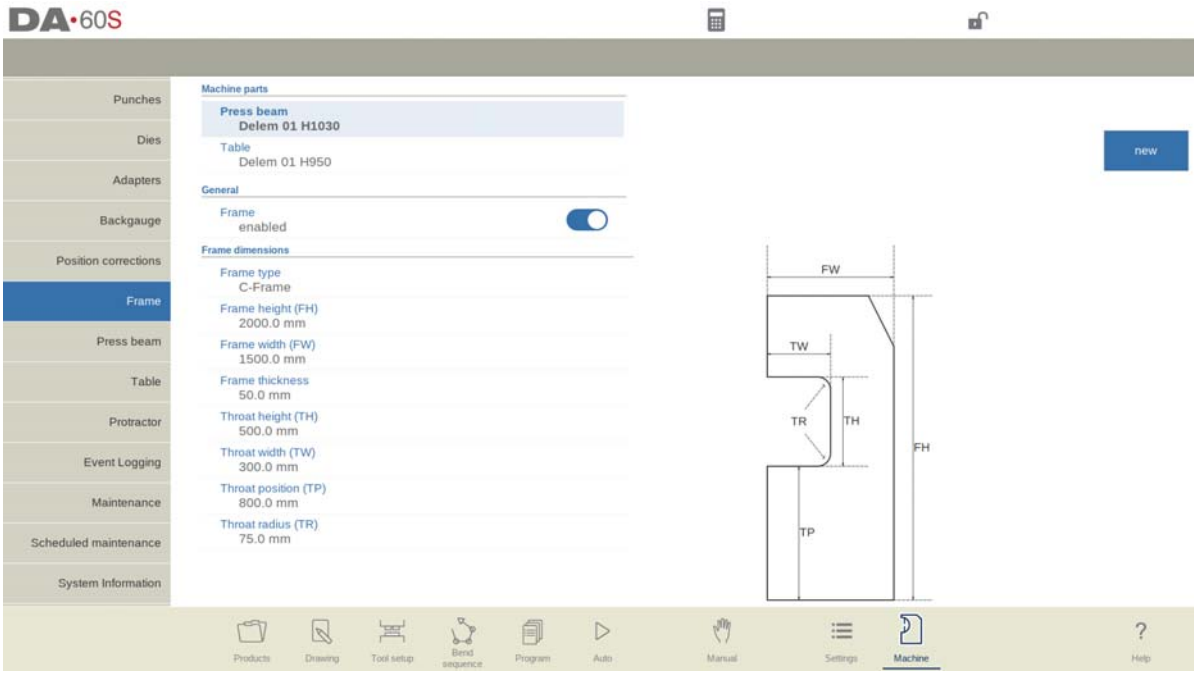
The shape of a wingbend die can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Important for the import of a wingbend die is that the body (lower part) of the die must be in layer 0 within the DXF drawing and the rotating part (upper part), the wing, should be in a different layer. If this layer is called ‘wing’ the importer can automatically select the correct tool type; otherwise it has to selected manually. Both parts in the DXF need to be closed contours.

10.4. Frame

On this tab the active machine geometries, the press beam and table and the side frames, can be selected and set. Also the machine identification can be programmed here.

Next to the Press beam and Table, which are chosen from those available, the Frame dimensions can be programmed in this page.
The machine shape is shown in the simulation screen during graphical programming and used for the collision detection for work piece against machine.



At the top of the screen the possible machine parts can be selected. The selected machine parts are used when a new tool configuration is programmed.

Press beam

Select the relevant machine press beam.

Table

Select the relevant machine table.

Frame

The side frames of the machine are part of the total machine design and important for the bend sequence of a product in relation to possible width. Some machines have side frames outside the bending zone and are not of influence for the collision detection. In that case the frames do not need to be shown in the bend simulation process. With this kind of machines the frame design can be disabled. By default this parameter is set to Enabled.

Frame type

The default side frame type is a C-frame. An alternative O-frame can be chosen if mounted. The O-frame is symmetrical and positioned at each side of the table.

Frame height
Total height of machine frame.

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

Frame thickness
The thickness of the side frame.

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

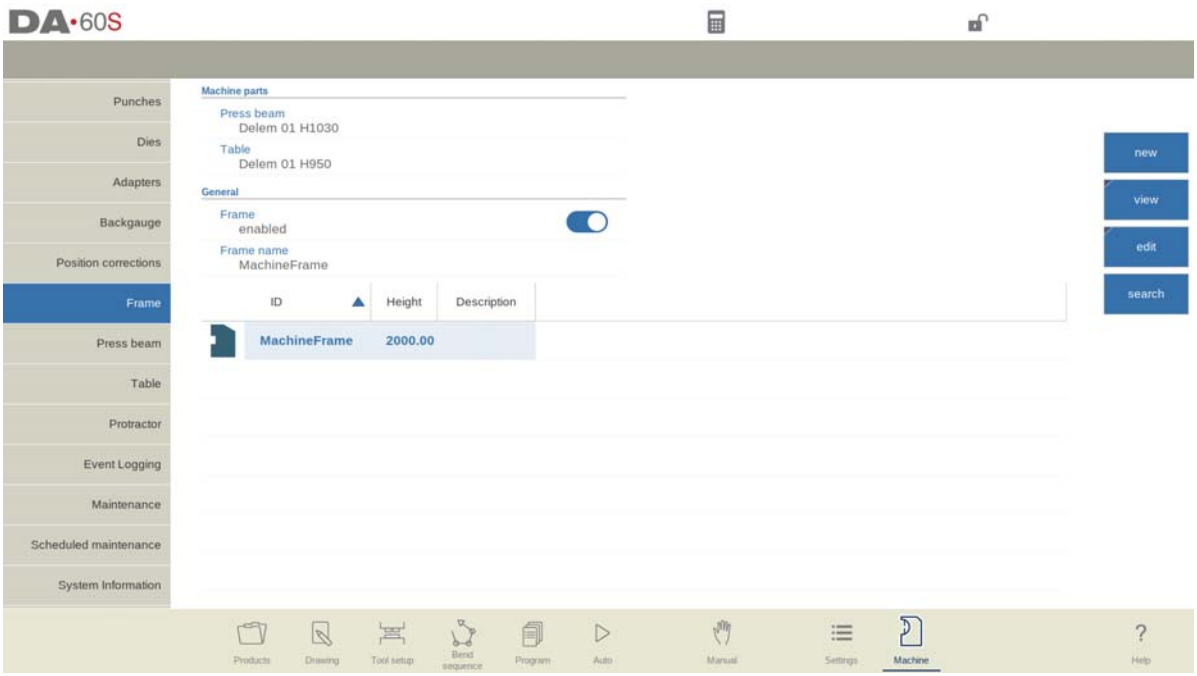
Throat width
The width (or depth) of the workspace.

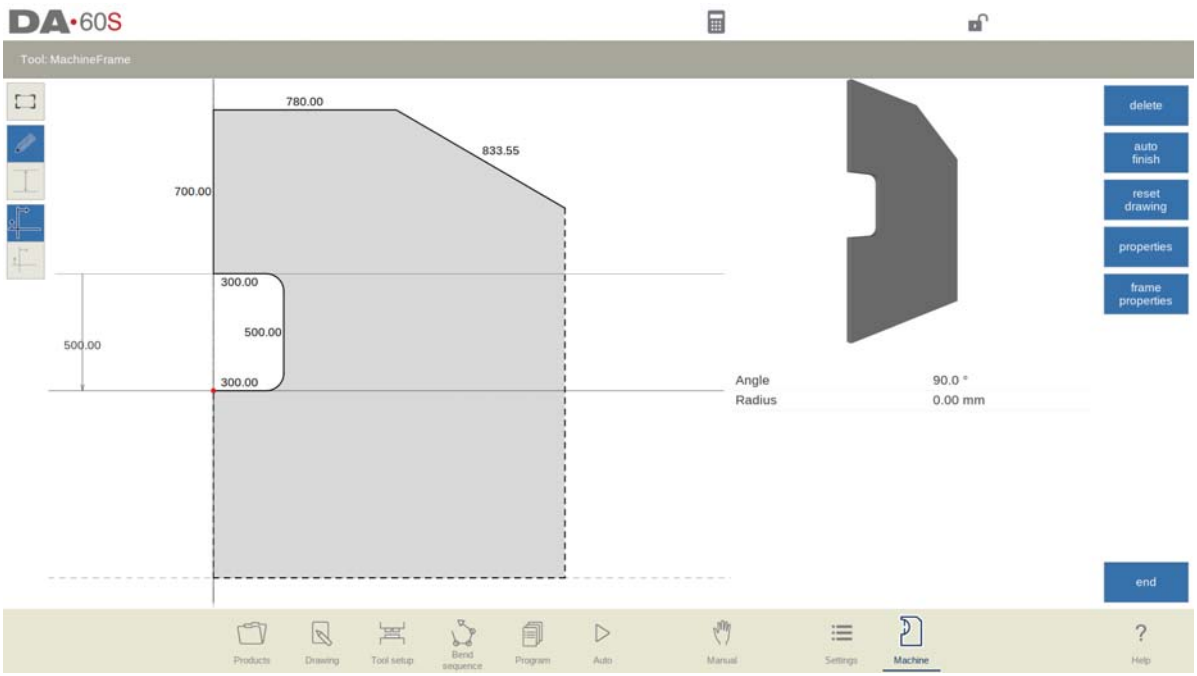
Throat position
The position of the workspace in the machine.

Throat radius
The radius of the workspace corner.

10.4.1. C-frame contour editor

Alternatively the C-frame shape can also be drawn as a contour. Within the contour editor the C-frame contour details can be drawn in detail. Starting from the basic shape, one can draw the frame as is; e.g. radii can be added in between lines as well as into lines themselves.

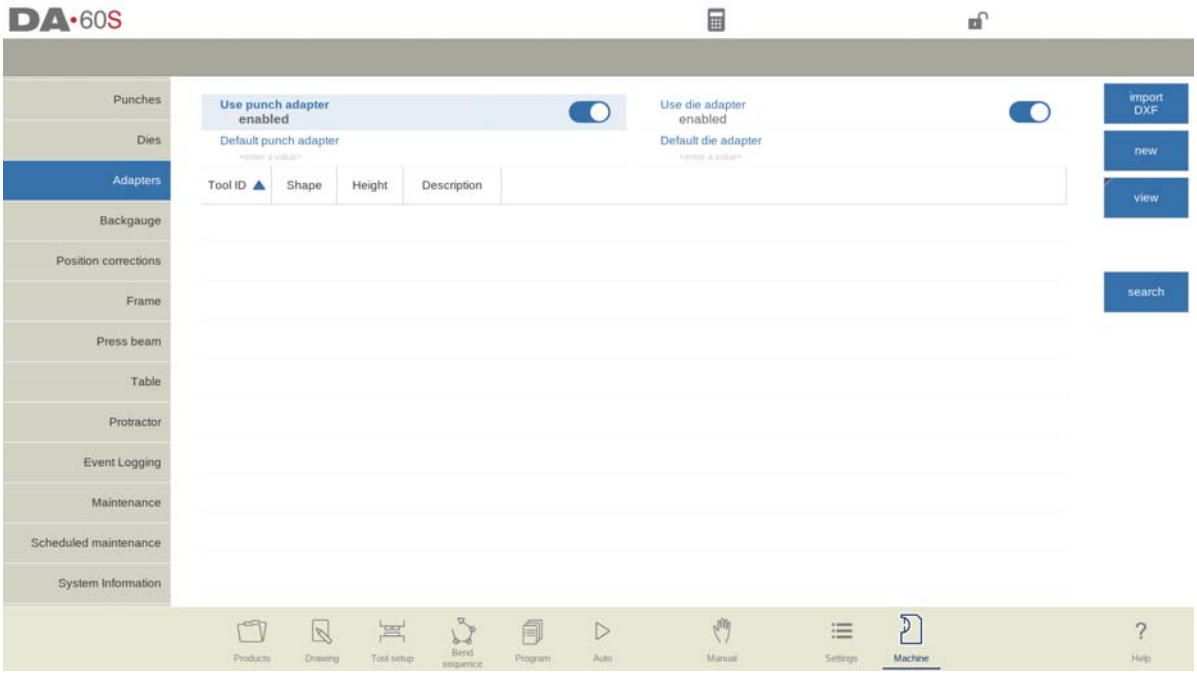




When using this graphical programming function it replaces the parameterized frame programming.

10.5. Adapters

On this page the tool adapters can be enabled and programmed.
Upon choice upper adapters as well as lower adapters can be enabled. The default adapter, which will be chosen when an adapter is added to the tool setup, can also be set.



When adding an adapter, first basic parameters need to be given based on a template. In the second stage the adapter details can be drawn like any other punch or die.

Shape

A selection must be done from the different available basic adapter shapes. Options are:

- Punch adapter, to be mounted to the press beam.
- Die adapter, to be mounted to the table.

ID

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

Description

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

Support type

Switch parameter, to account for differently mounted adapters. Such adapters 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', similarly to punches.

Resistance

Maximum allowable force on the adapter. A value of 0 means no check on the resistance of the adapter.

DXF import (only available when DXF option has been installed)

The shape of an adapter can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Mount point (optionally programmable)

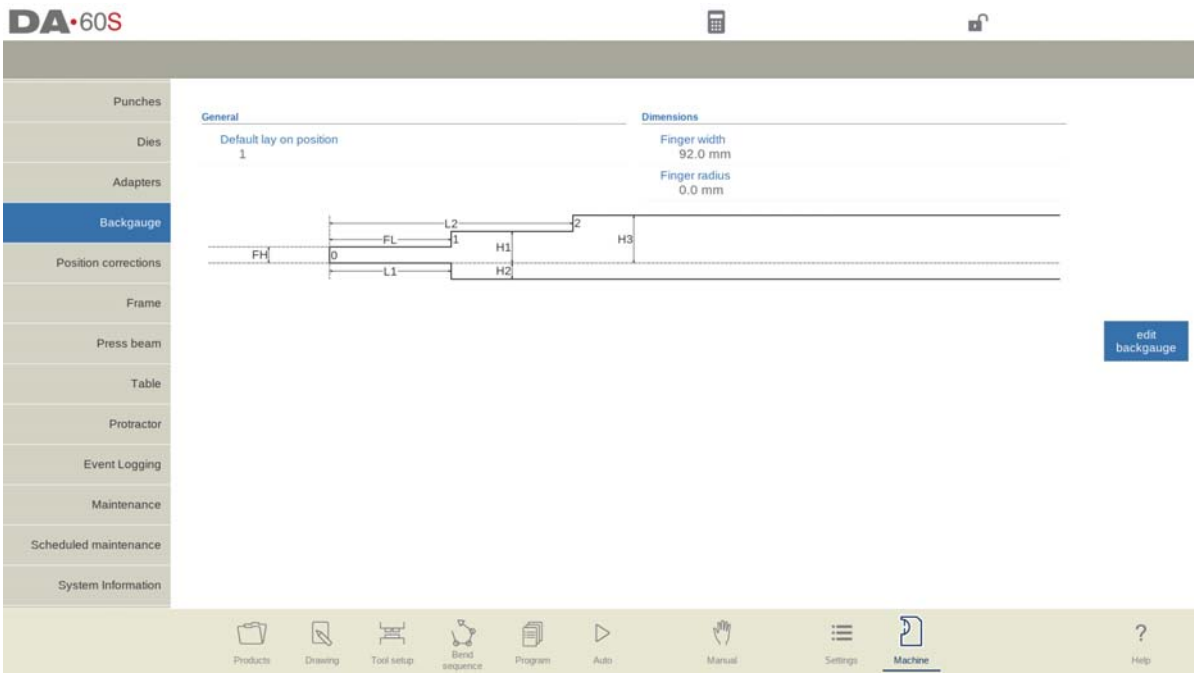
Within the drawing one also finds the mount point of the adapter. This point, indicated with the triangle arrow, shows at what point the adapter is connected to the press beam or table. The second arrow indicates where the punch or die will be placed in this adapter. Mount points can also be found in punches/dies and in the press beam/table. When this function is not enabled, the indicators are not shown.

Default adapters for tools

When a tool is used with a specific adapter and this combination is used commonly, the adapter can be set as default in the properties of the tool. Every time the tool is selected, automatically also the adapter will be loaded in the tool setup.

10.6. Backgauge

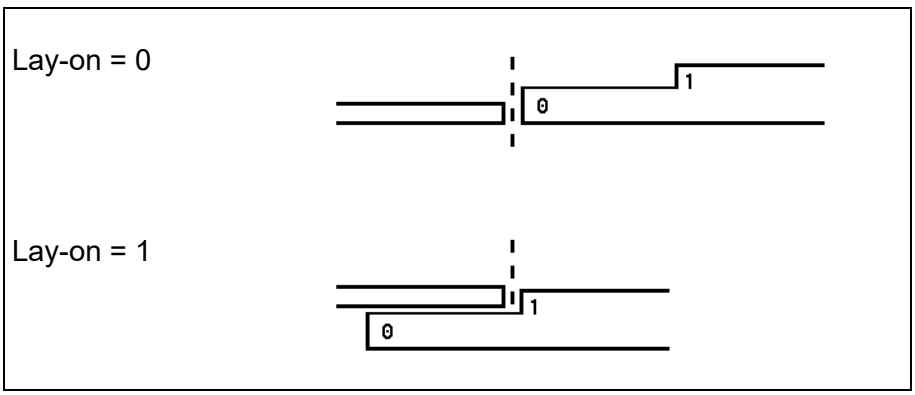
With the backgauge finger dimensions the R-axis movement and related X-axes movement is taken into account. Also the workpiece / backgauge collision is computed using the dimensions.



Default lay on position

This is the default lay-on position in case a lay-on position must be used during automatic bend sequence computation, e.g. in case the X-axis position is outside its allowed range or larger than the 'Lay-on backstop limit'; it is not used when selecting a lay-on level manually.

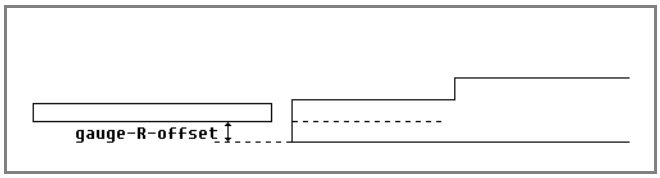
Meaning of lay-on numbers:



Gauge R offset

An offset value for the R-axis can be set if the backgauge is positioned against the sheet

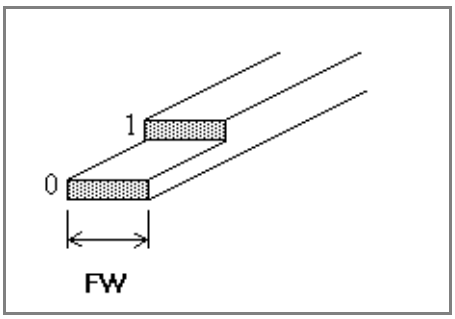
edge and the X-axis position is outside the die safety zone.



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

Finger width

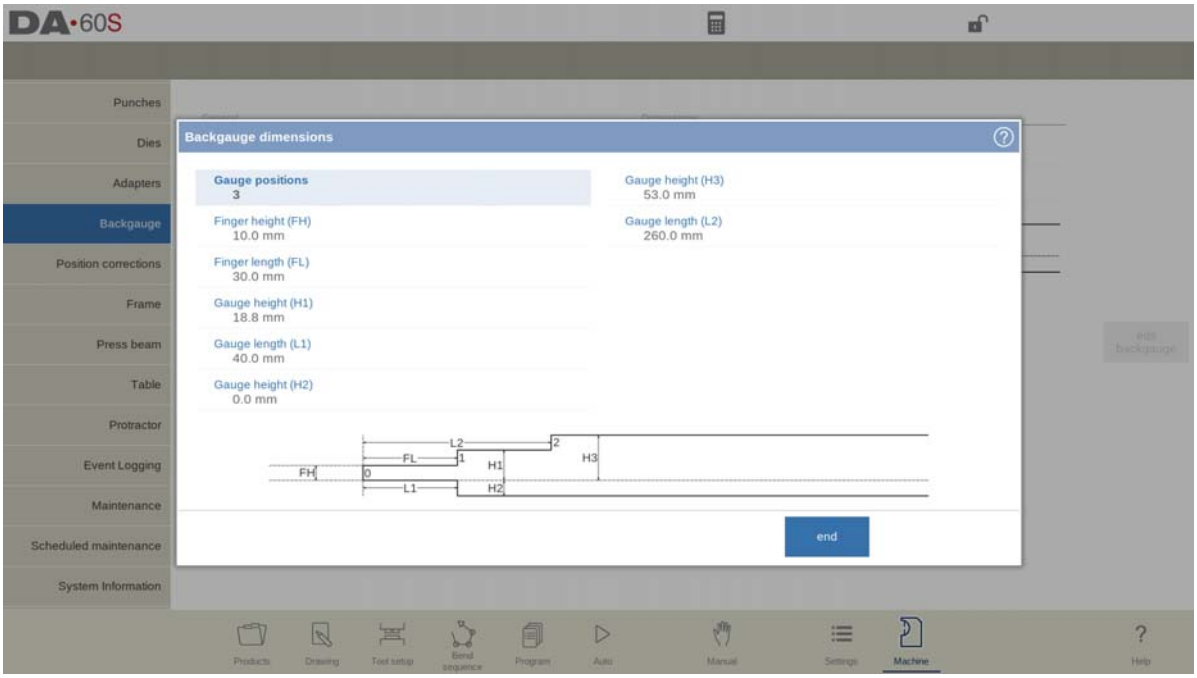
The width of the backgauge finger. Only available when automatic Z-axes have been installed.



Finger radius

The radius of the tip of the backgauge finger. A value of 0 means no radius. Only available when automatic Z-axes have been installed.

Tap Edit backgauge 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/H3/H4)

The height of the different lay-on levels.

Gauge length (L2/L3)

The length of additional lay-on levels.

Gauge height (H2)

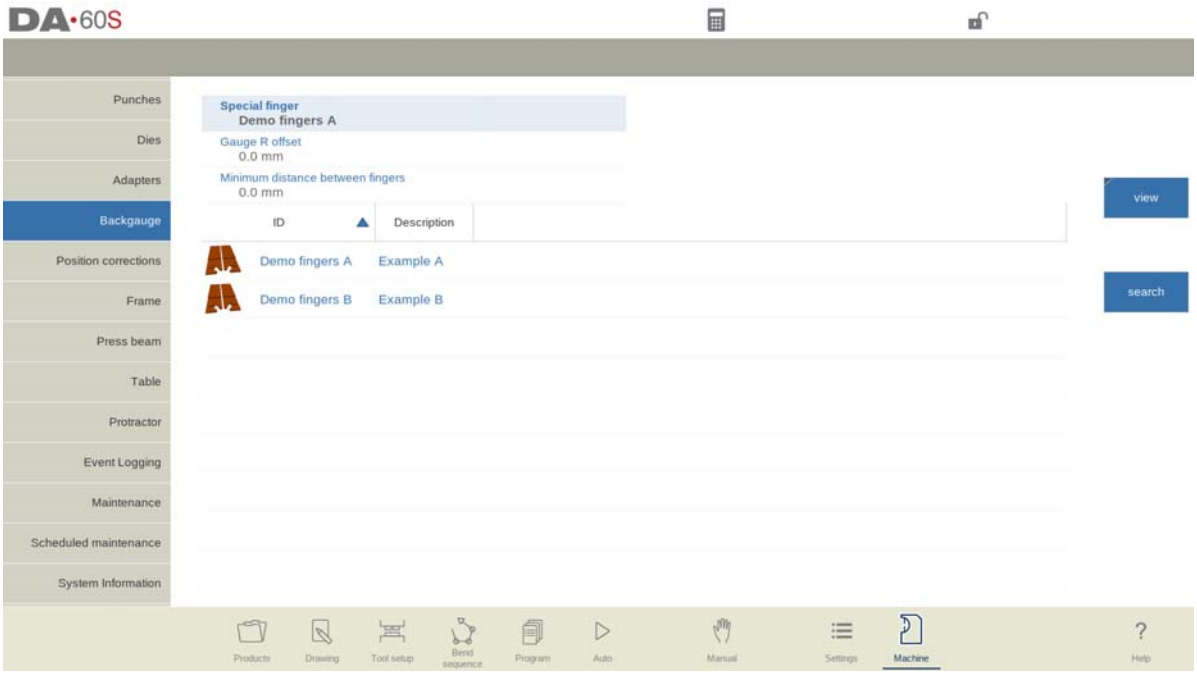
The height of the bottom of the gauge.

Gauge length (L1)

The length of the finger level at the bottom of the gauge.

10.6.1. Backgauge finger library

When more than one set of backgauge fingers are programmed, they are shown in a library of fingers. When special fingers are installed, the database is activated enabling more finger sets.
Upon choice the user can select the backgauge fingers in the machine. The specific finger geometries are taken into account for the calculation of finger positions.
When selecting a product one must make sure the appropriate fingers with which the CNC program has been generated have been selected in the machine.



Special finger

The name (ID) of the selected backgauge fingers set. Tap to modify or select from the backgauge finger library.

Minimum distance between fingers

An absolute distance that must be maintained between fingers. A value of 0 means this parameter is not used.

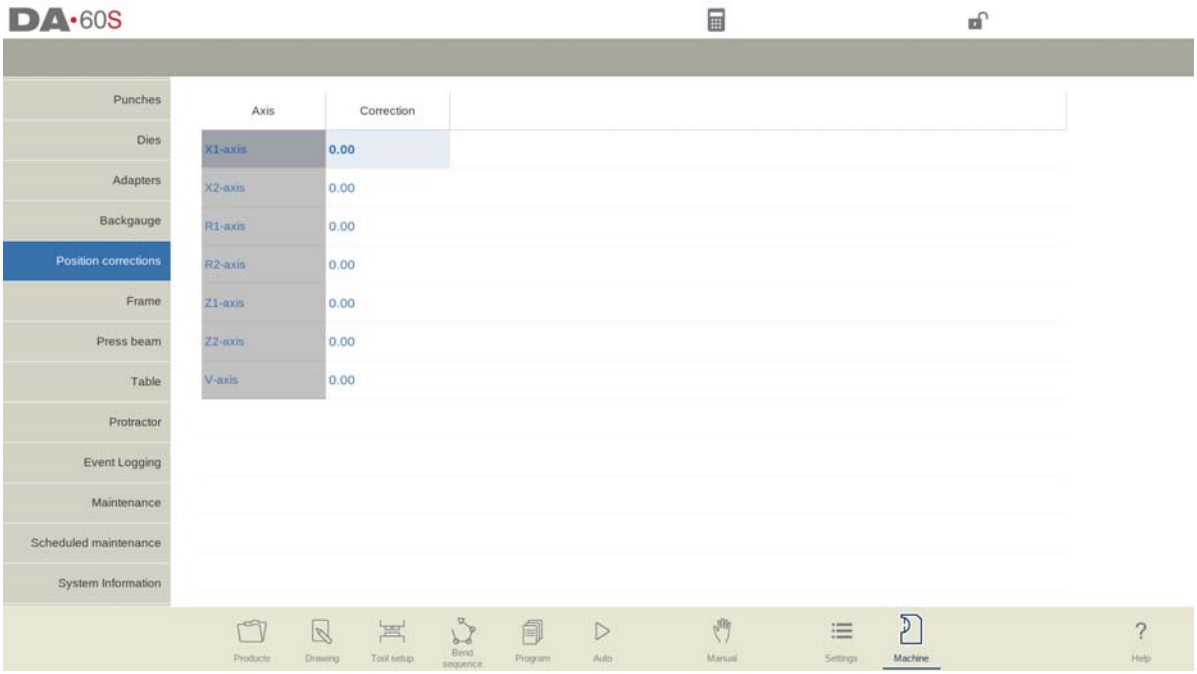
When all finger sets are deleted, the parameterized finger definition is re-activated.

The following additional parameter is available when editing a special finger from the backgauge finger library:

X offset

The length difference between the front of the selected finger and the front of the finger with which the machine was commissioned and calibrated.

10.7. Position corrections



Position correction

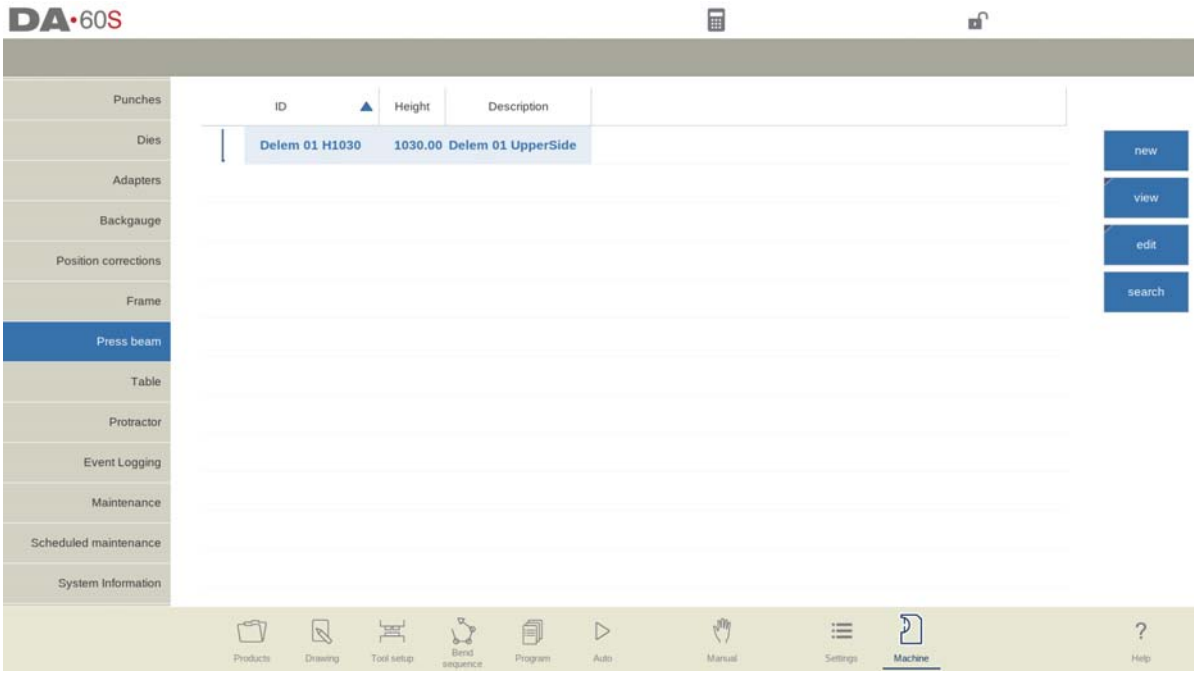
When the actual, mechanical axis position is not corresponding with the displayed value it is possible to correct the position with this parameter. Program the calculated difference.
Example:
- When the programmed and displayed value = 250 and the actual, mechanical position value = 252 the correction 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 auxliliary axes.

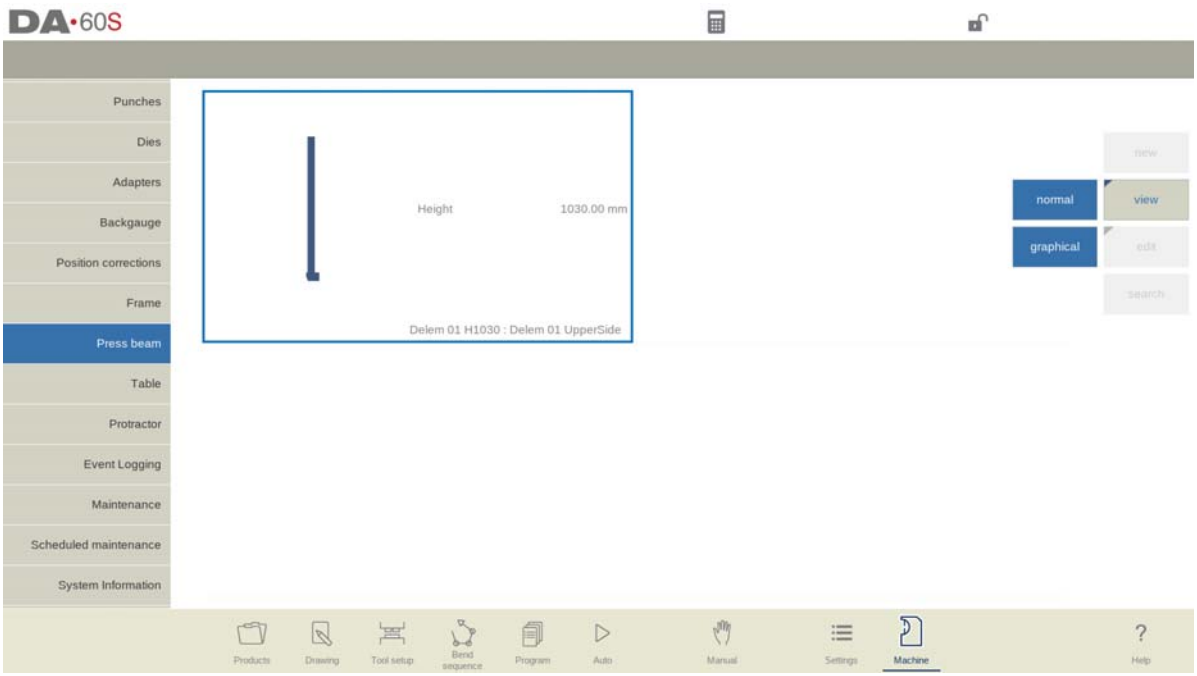
Position corrections should only be used temporarily. In case machine positions have changed after commissioning or service, positions of axes can be corrected. In normal situations these corrections should be 0.

10.8. Press beam

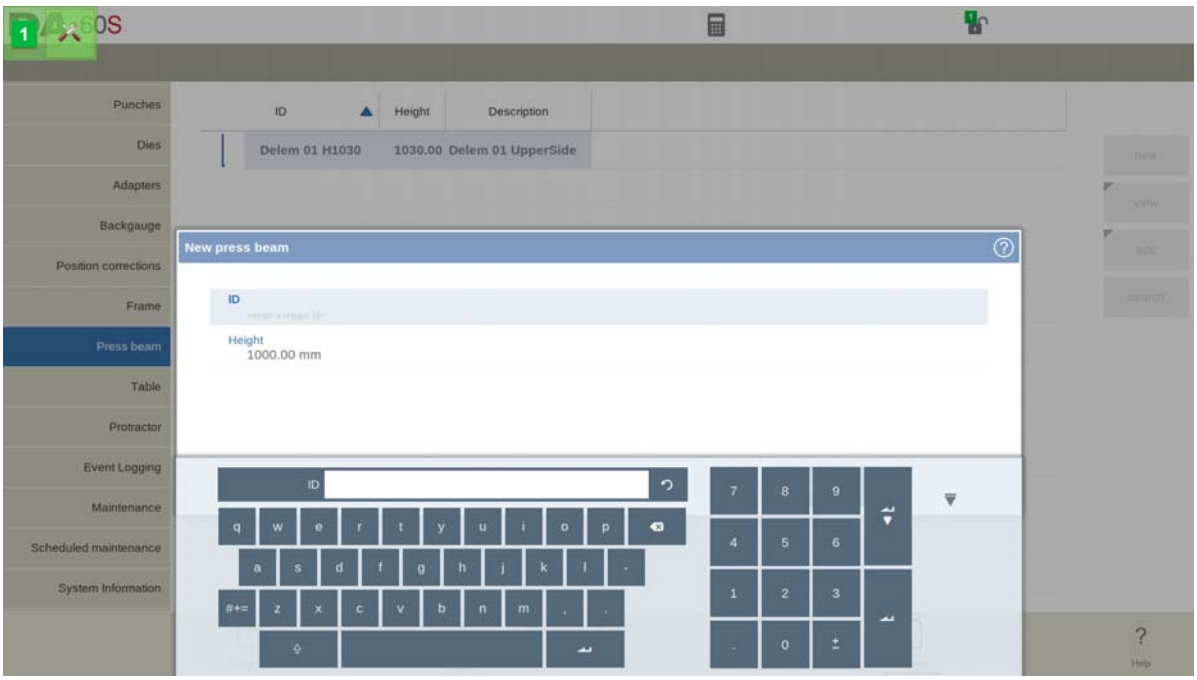
In this tab the machine geometry for the press beam, as a profile, can be programmed. This information is used in the collision detection of collisions with product and machine.
When e.g. utilities are added to the machine in special cases, these can be programmed as a special machine shape to enable the collision calculations to take this into account.
In most instances there is only one shape programmed.



The shapes of your machine are drawn in the same way as for the punches and dies. Like with the tools the right hand side of the drawing is the backgauge position of the machine.
To have a more detailed view in the library tab, tap View and use Graphical.



To create a new machine part, tap Edit in the library and subsequently use Add.
The control will start by asking for an ID, a description and a height.



ID

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

Description

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

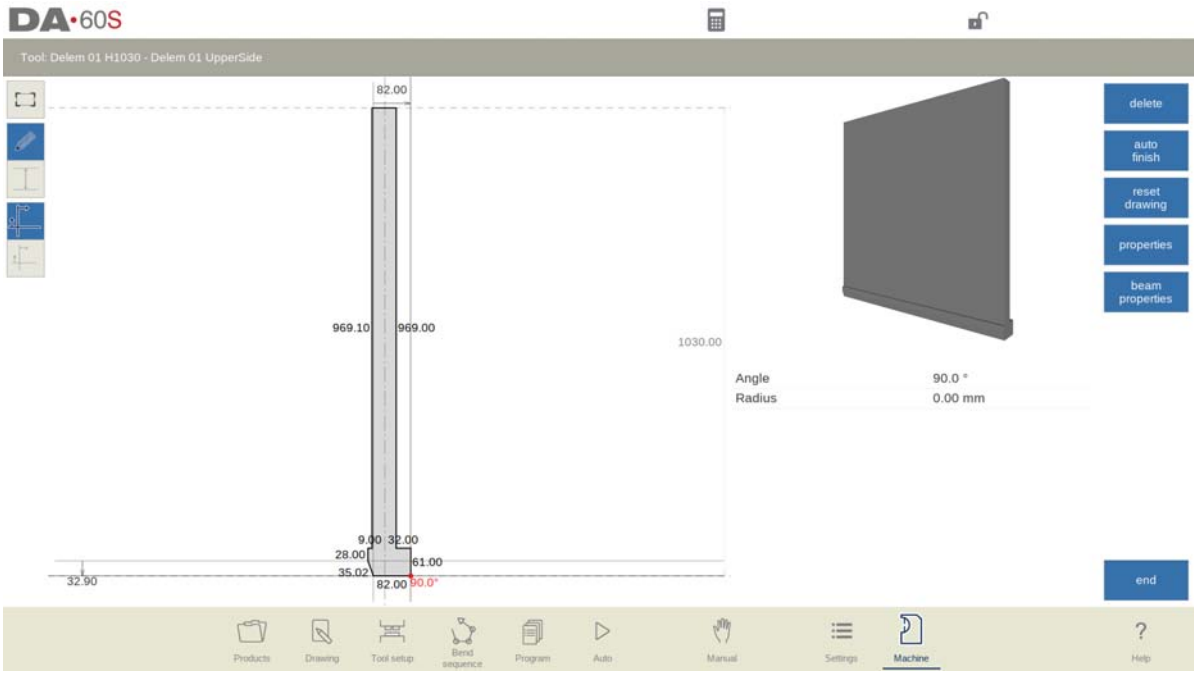
Height

The total height of the machine part.

Tool holder resistance

Maximum allowable force on the tool holder. A value of 0 means no check on the resistance of the tool holder.

For the press beam the height is defined as the height of the part when the moving part of the upper side is in its mechanical top dead center.



After giving in the base parameters for the specific machine part, the drawing editor appears. Similar to drawing of tools, also the details of the machine parts can be drawn. Either by tapping and sketching or by giving in the length of sides and pointing the direction of the next side.

DXF import (only available when DXF option has been installed)

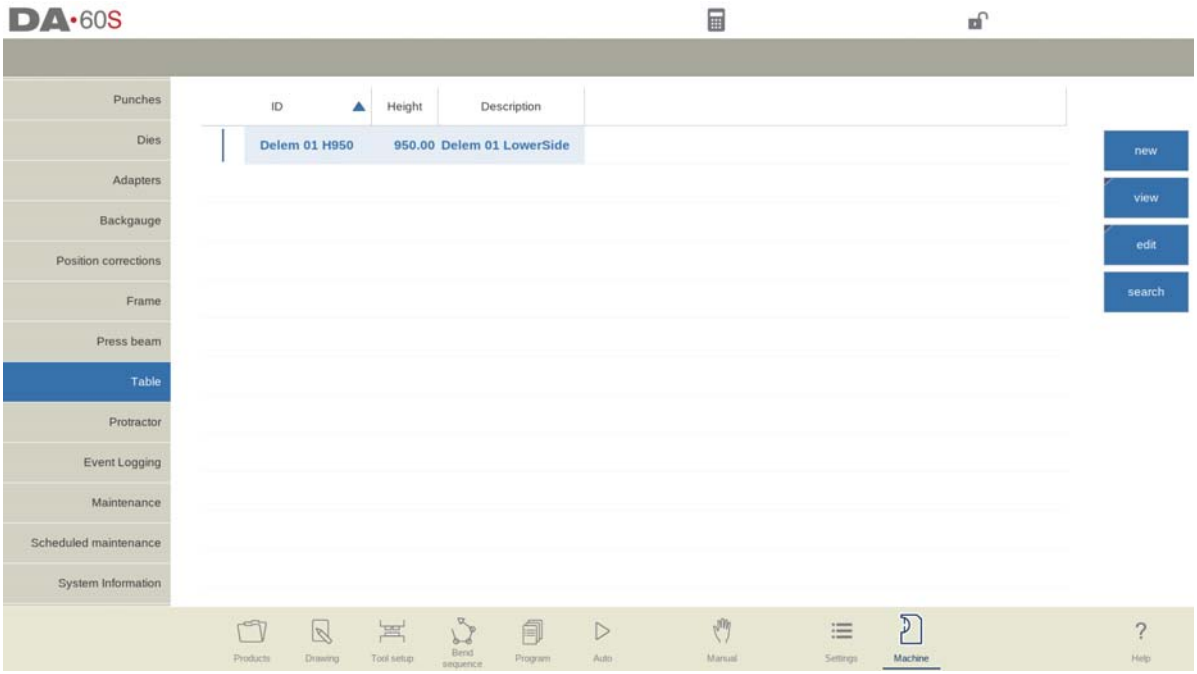
The shape of a press beam can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Mount point (optionally programmable)

Within the drawing one also finds the mount point of the press beam. This point, indicated with the triangle arrow, shows at what point the punch or adapter will be connected and placed on the press beam. Mount points can also be found in punches and adapters. When this function is not enabled, the indicator is not shown.

10.9. Table

In this tab the machine geometry for the table, as a profile, can be programmed. This information is used in the collision detection of collisions with product and machine.
When e.g. utilities are added to the machine in special cases, these can be programmed as a special machine shape to enable the collision calculations to take this into account.
In most instances there is only one shape programmed.



To create a new machine part, tap Edit in the library and subsequently use Add.
The control will start by asking for an ID, a description and a height.

ID

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

Description

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

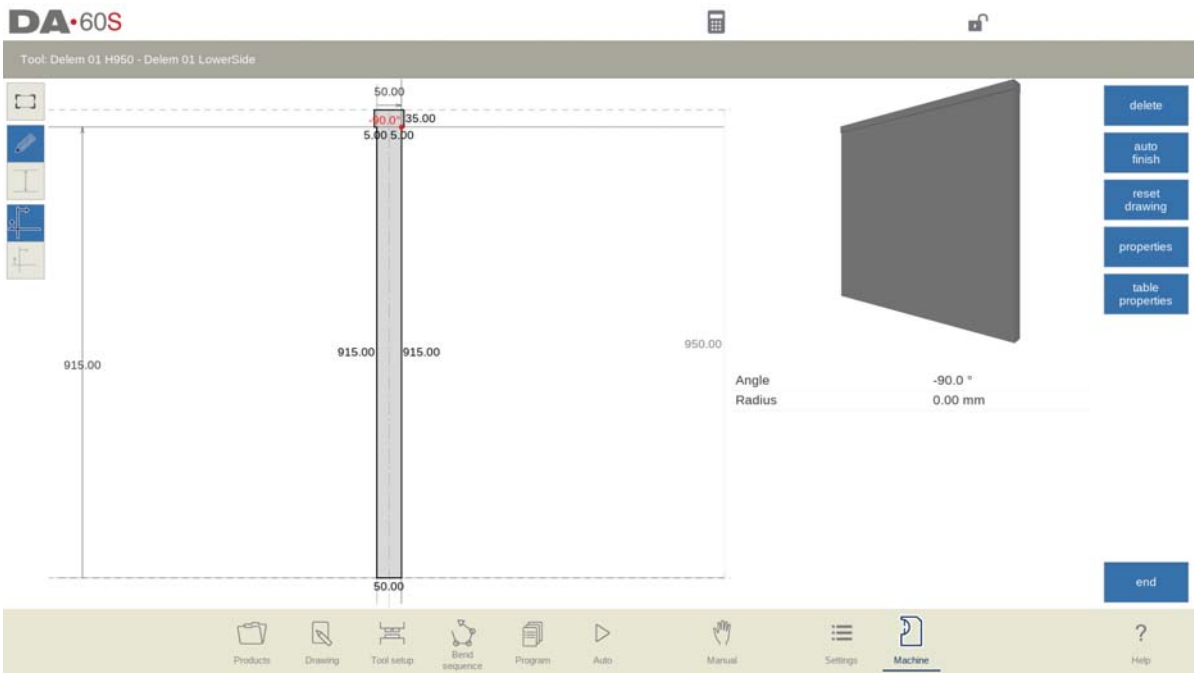
Height

The total height of the machine part.

Tool holder resistance

Maximum allowable force on the tool holder. A value of 0 means no check on the resistance of the tool holder.

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



After giving in the base parameters for the specific machine part, the drawing editor appears. Similar to drawing of tools, also the details of the machine parts can be drawn. Either by tapping and sketching or by giving in the length of sides and pointing the direction of the next side.

DXF import (only available when DXF option has been installed)

The shape of a table can also be loaded via the optional DXF import function. For details see paragraph 10.11.

Mount point (optionally programmable)

Within the drawing one also finds the mount point of the table. This point, indicated with the triangle arrow, shows at what point the die or adapter will be connected and placed on the table.
When an I-axis is present in the system also the I-axis division line will be shown. This line, by default positioned at the top level of the table, can be pulled down on levels in the drawing. This level can be the division between the tool holder and the actual table. The division line defines the stationary and dynamic part of the table. All above the dashed line, e.g. the tool holder, will move with the I-axis.

Specifically with the mount point of the table also the I-axis position is defined. By default this I-axis value is 0, meaning when the I-axis position is set to 0 the mount point will be positioned at the center of the machine (X=0). Upon choice one can change this value giving an offset to the I-axis range.
Mount points can also be found in dies and adapters. When this function is not enabled, the indicator is not shown.

10.10. Drawing functionality for tools, adapters and machine shapes

In programming punches, dies, adapters and also machine shapes, after the main data the control provides the functionality to freely draw the desired shape in the object. This functionality makes the objects appear more realistic but most of all enables the control to do an accurate collision prevention.

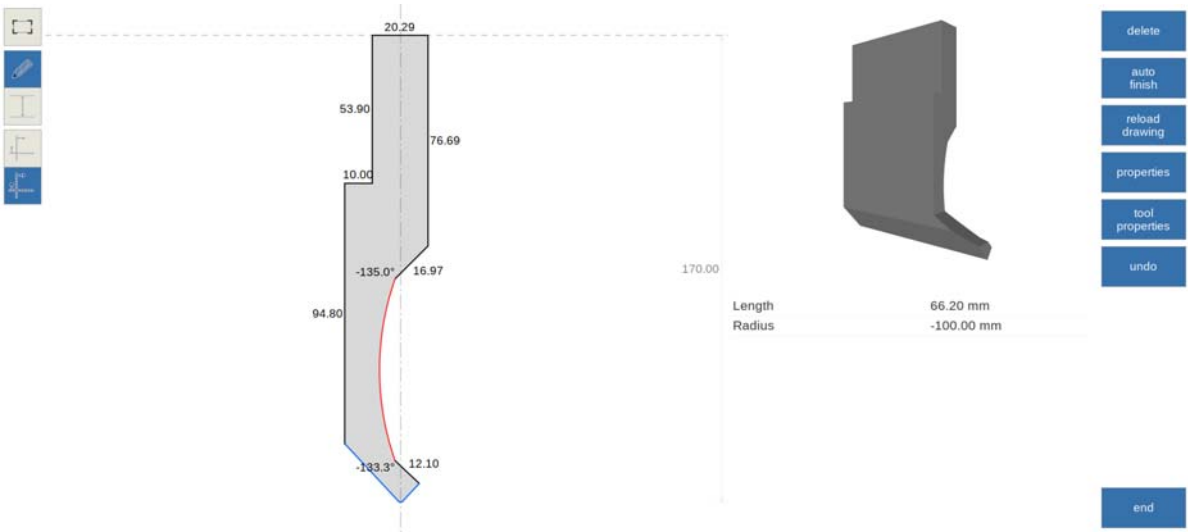
In this drawing functionality multiple methods can be used to have the desired shape.
One can sketch the desired shape and after that change segments to program their accurate value. One can also start doing this from the first drawn segment, step by step.

Important to know is the following:

- In the end these shapes should be closed. The auto-finish function can help with that.
- The height of the programmed object is used in the bending calculations. Keep in mind that this is very important to get to the desired results.

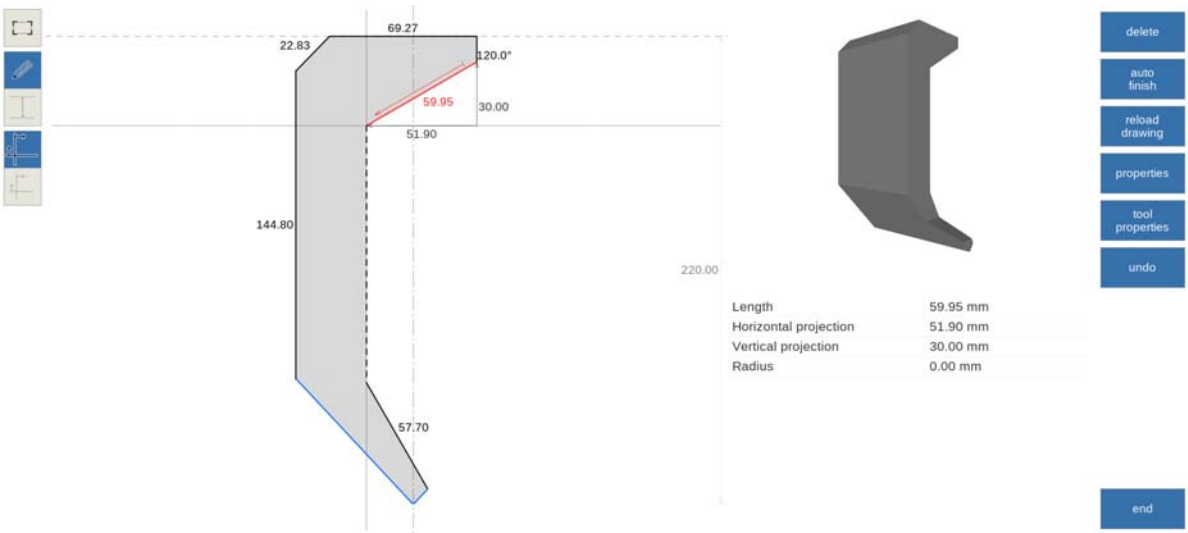
Helpful to know can be the following:

- A line as well as an angle can be given a radius.

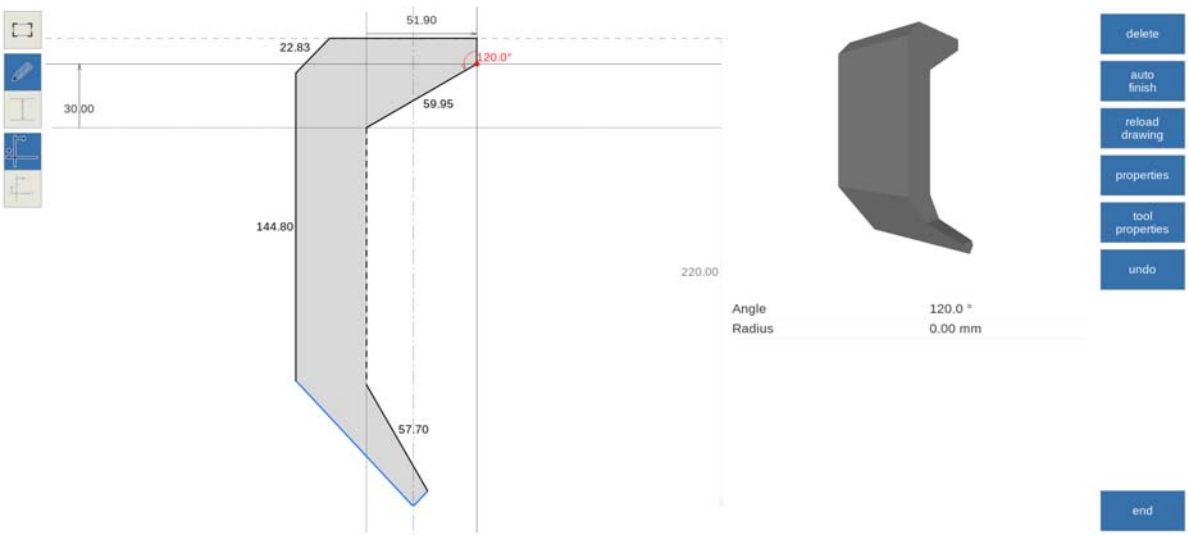


- Snapping will help to align lines and angles to its surroundings.

- A line has length as well as projection dimensions. These, along with its angle, can be used programming the desired line. The control will help in this by trying to accept your last input and adjusting the necessary values around. This is very helpful since drawings might alter in what dimension information is given at each shape detail.



- Help lines are provided to measure inter point distances and also enable to change the point based on the desired distance. When selecting a point, help lines appear. When you move help lines to reference your desired value, just change the dimension and the point will move where you need it.



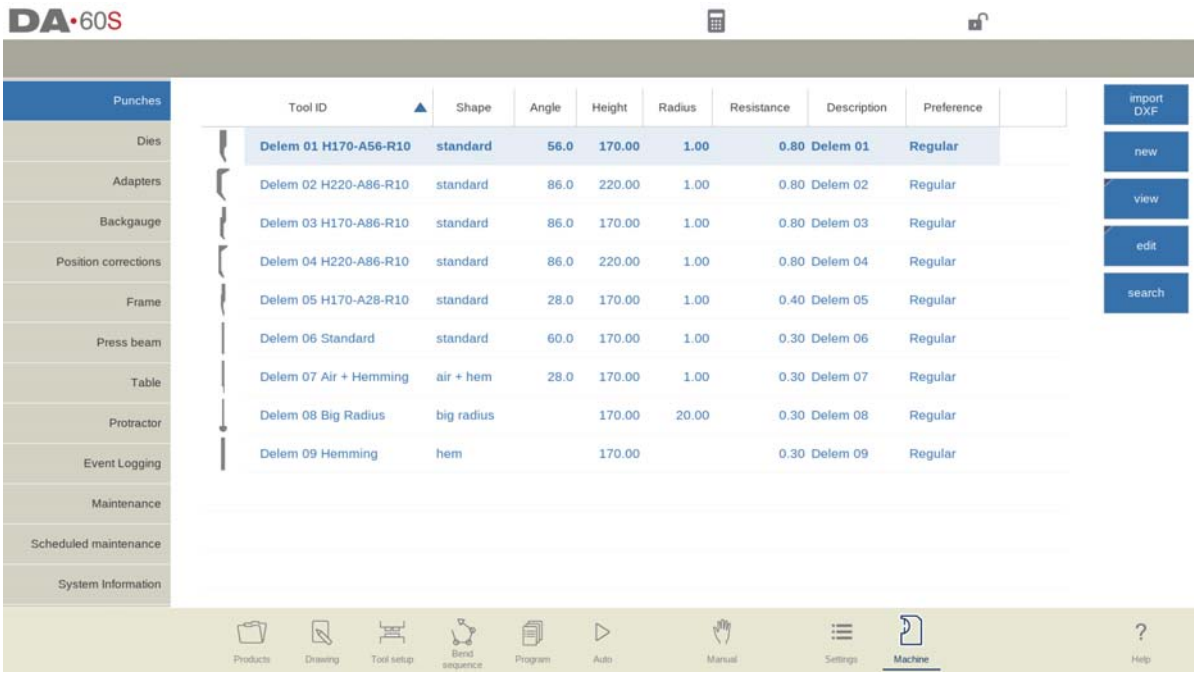
- For your convenience the help lines can be switched off, giving a non disturbed view to what you have drawn.

- In case the control is using mount points in the tool details, the mount points can be toggled with the drawing. This will prevent from accidental non desired changes to either mount point position or shape details.

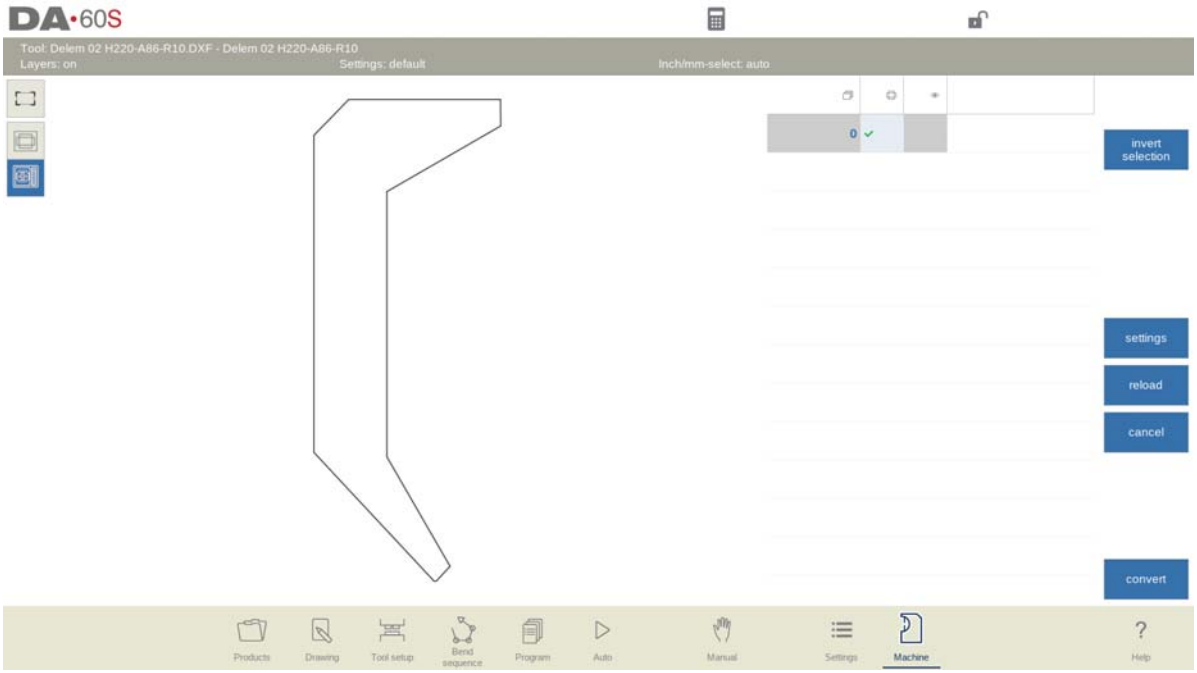
10.11. DXF import for tools, adapters and machine parts (only available when DXF option has been installed)

Within the tool library, Punches, Dies, Adapters as well as the machine shapes, Press beam, Table and Frame, one can find the optional DXF import function. This function enables the import of the contour from a DXF file.

Import DXF opens the file browser enabling the selection of the DXF file of the desired shape.



When selecting the DXF file, depending on the specific screen this is chosen from, the system will show the DXF file. The conversion settings can be changed upon desire. E.g. if layer selection is required or if not if only contour information is available. If layer selection is on, one has to designate layers to the desired content.



The DXF tool import functionality is similar to the DXF product import, explained in paragraph 2.3.

When Convert is chosen, automatic shape detection or specific shape selection can be selected as well as the name ID of the new tool can be given.

From this point, the conversion will be done and the new tool will be shown in edit mode within the tool editor. Specific tool functions as Tool height, Mount points (if applicable) and tool properties can be set.

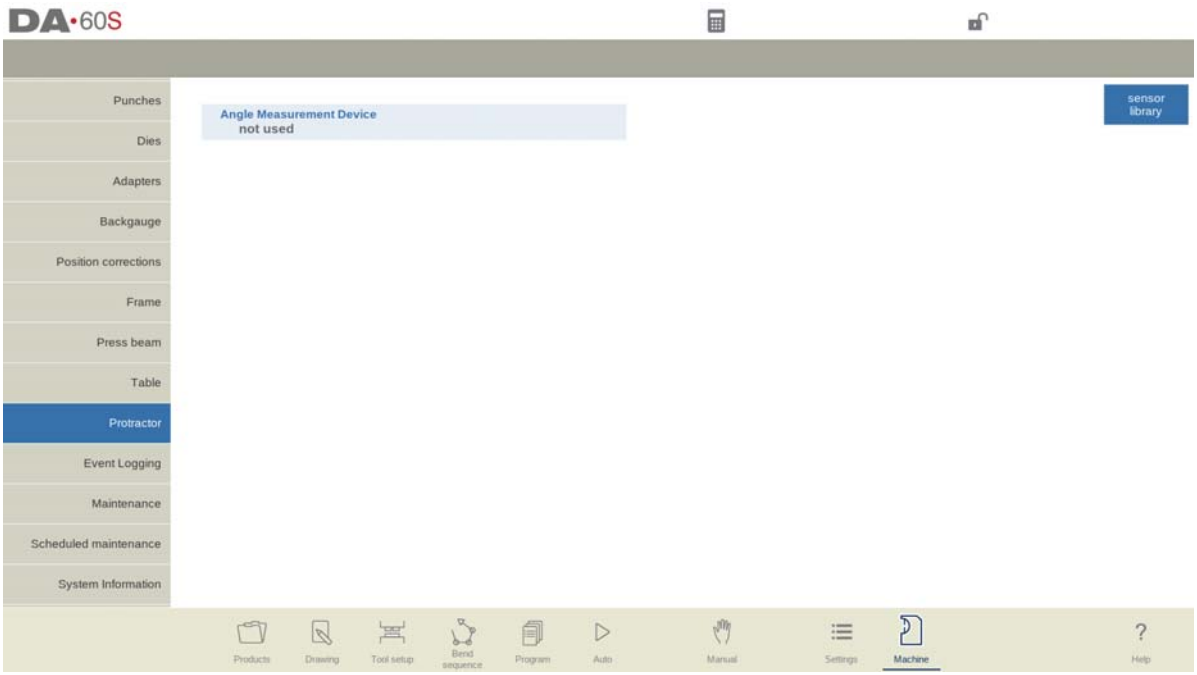
After closing the new tool has been added to the library and can be used and edited as any other tool.

Machine DXF import functions
This same functionality, only differing based on the desired import, can be found in following screens:

- Punches
- Dies
- Adapters
- Frame
- Press beam
- Table

Conversion of special tools
Important for multi contour shape importing is that the body of the tool must be in layer 0 within the DXF drawing. The top part of a inside hem die, the wing of a wingbend or the movable part of the vario v, should be in a different layer. If this layer is called resp. ‘Inside hem’, ‘Wing’ or ‘Vario V’ the importer can automatically select the correct tool type, which otherwise can also be selected manually. Both parts in the DXF need to be closed contours.

10.12. Protractor



With this parameter you can select a digital angle measuring device when the Protractor option has been installed.

Angle Measurement Device

- Not Used
- Mitutoyo 187-50x
- Mit.187-50x U-WAVE (this option will only be displayed if the wireless receiver has been detected during start-up)
- Mitutoyo 187-50x IBR
- KEBA KeMes A205 (this option will only be displayed if the wireless receiver has been detected during start-up)

This device can be used in the production modes. Place the cursor on either the alpha correction field or in the pop-up window alpha corrections on the measured angle field and press the transmit button on the angle measuring device. The control will display the received angle on the screen. By selecting the enter key this value is accepted and the control will calculate the required correction based on this received angle value.

Auto enter corr.α

off => After a measurement, the angle correction will be calculated and displayed on the screen. The machine operator has to select the angle correction field and press the enter button to copy the correction.

on => The angle correction field will be selected automatically. From the measured angle, the correction is calculated and copied into the correction parameter automatically without selecting any button on the DA-control.

Angle tolerance

Maximum allowed deviation between the programmed angle and the measured angle. When the measured angle differs too much from the programmed angle an error

message will appear on the screen.
In case the ‘Auto enter corr.α’ parameter is switched on, the correction will not be copied automatically when the deviation is too large.

Angle calculation

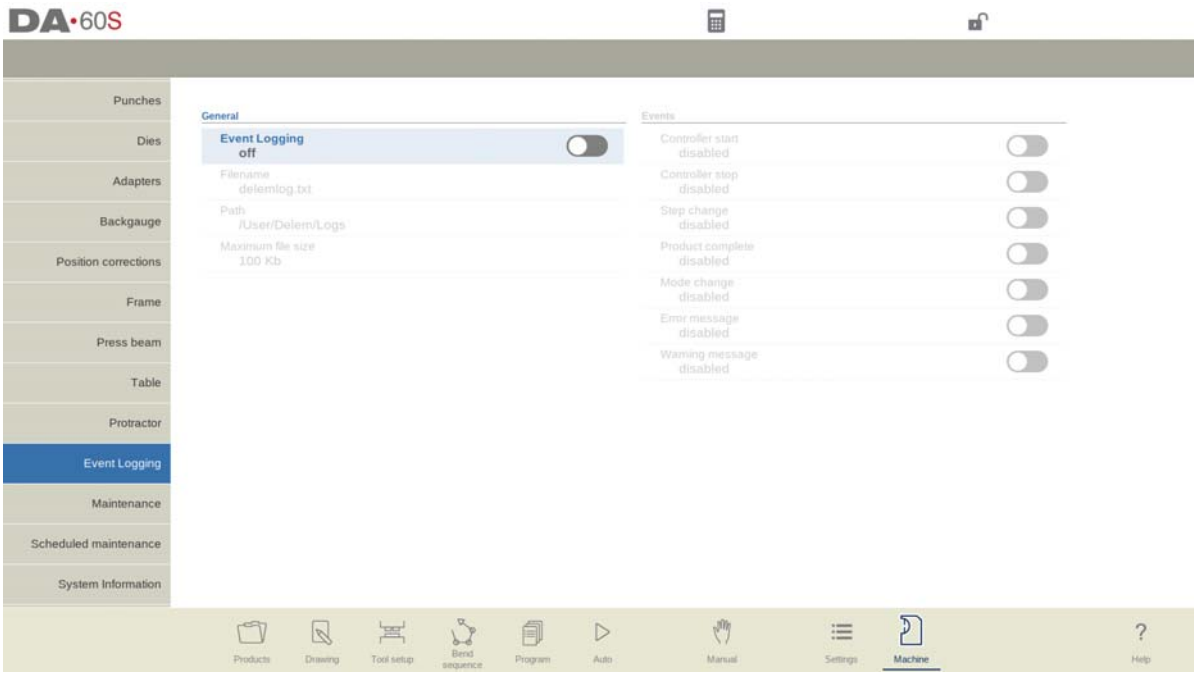
180-α => The measured angle is the supplement angle of the actual product angle.

α => The measured angle is the actual product angle.

10.13. Event logging

10.13.1.Parameters

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



Event logging

To switch the event logging function on or off.

Filename

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

Path

The folder (directory) where the log file will be stored. This path can be on the internal memory of the DA-control, a USB stick or even a network drive, if connected. When the cursor is on this parameter, use the function Select Path to select the desired location for the log file.

Maximum file size

The maximum size of the log file in kilobytes. If the current log file reaches this size it will be closed and renamed automatically. Immediately a new file with the programmed name is created and opened to continue logging.

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

should be switched on for logging.

Controller Start
Log event: the control has been started.

Controller Stop
Log event: the control has been stopped.

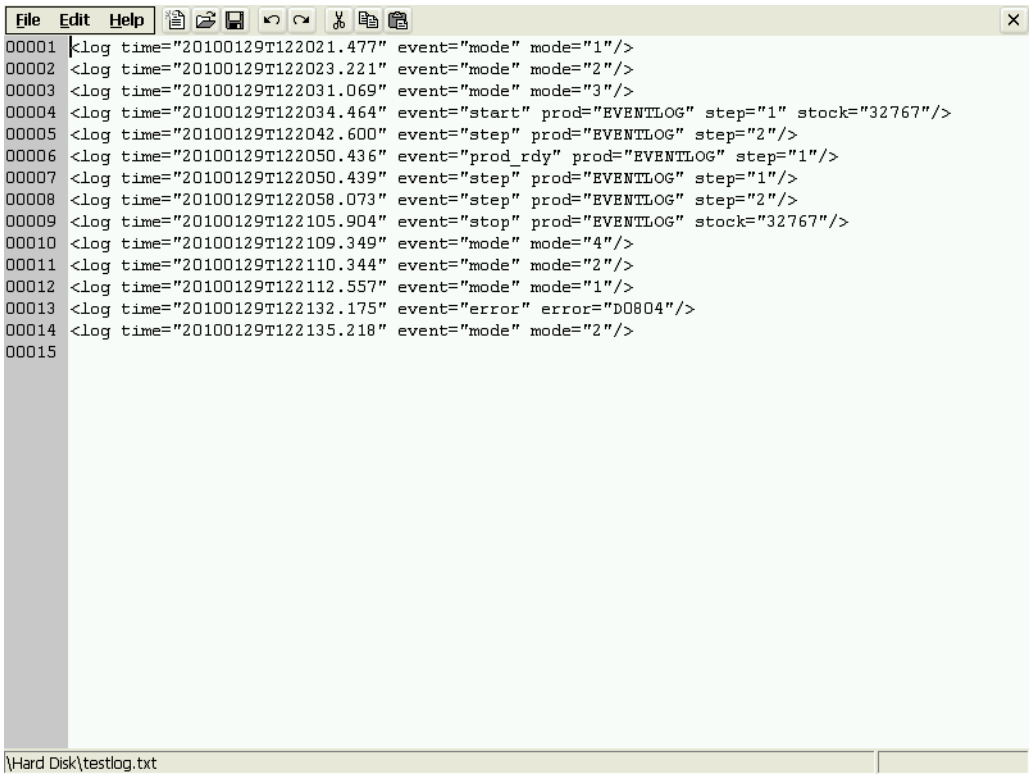
Step Change
Log event: the control has switched to the next bend step.

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

Mode Change
Log event: a different mode has been selected.

Error Message
Log event: an error message has occurred.

10.13.2.Explanation



```
00001 <log time="20100129T122021.477" event="mode" mode="1"/>
00002 <log time="20100129T122023.221" event="mode" mode="2"/>
00003 <log time="20100129T122031.069" event="mode" mode="3"/>
00004 <log time="20100129T122034.464" event="start" prod="EVENTLOG" step="1" stock="32767"/>
00005 <log time="20100129T122042.600" event="step" prod="EVENTLOG" step="2"/>
00006 <log time="20100129T122050.436" event="prod_rdy" prod="EVENTLOG" step="1"/>
00007 <log time="20100129T122050.439" event="step" prod="EVENTLOG" step="1"/>
00008 <log time="20100129T122058.073" event="step" prod="EVENTLOG" step="2"/>
00009 <log time="20100129T122105.904" event="stop" prod="EVENTLOG" stock="32767"/>
00010 <log time="20100129T122109.349" event="mode" mode="4"/>
00011 <log time="20100129T122110.344" event="mode" mode="2"/>
00012 <log time="20100129T122112.557" event="mode" mode="1"/>
00013 <log time="20100129T122132.175" event="error" error="D0804"/>
00014 <log time="20100129T122135.218" event="mode" mode="2"/>
00015
```

One line in the log file can look as follows:

<log time="20100129T122021.477" event="mode" mode="1"/>

or:

<log time="20100129T122034.464" event="start" prod="EVENTLOG" step="1" stock="32767"/>

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

log time=<date>T<time>

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

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

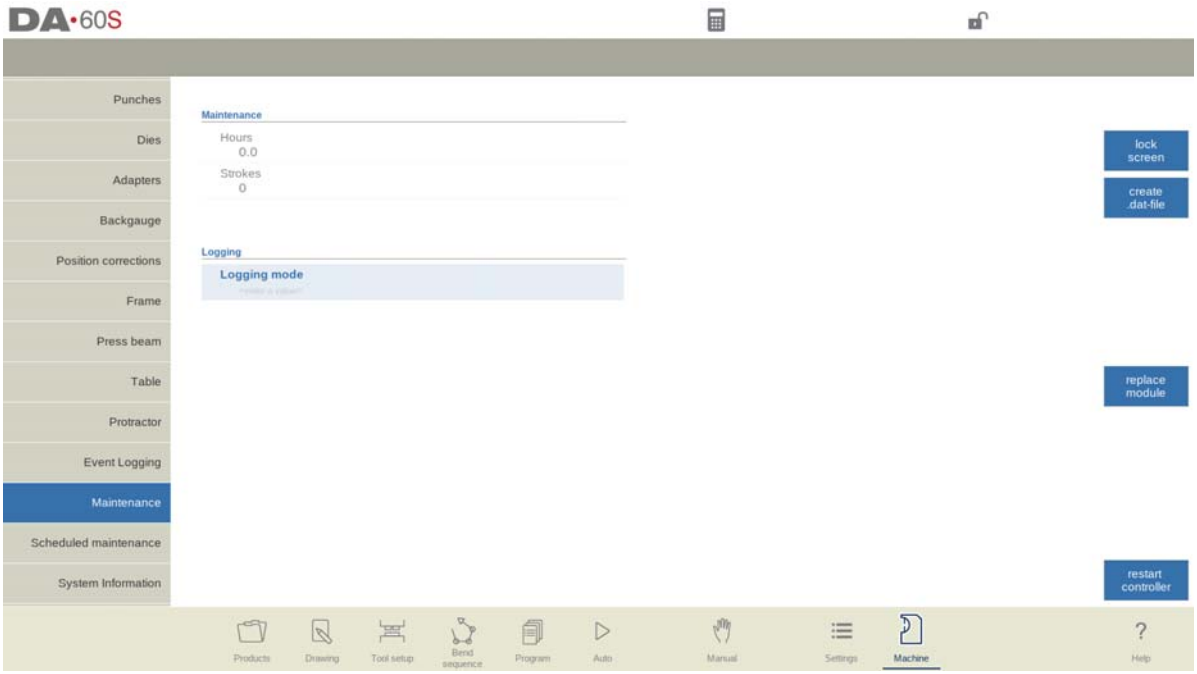
Each attribute is indicated by a keyword. The following keywords are possible:

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

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

10.14. 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 press beam has executed.

Logging mode

Activate or deactivate a logging mode for service purposes. The logging mode is activated by programming a special code here. The logging mode is de-activated by programming 0.
To activate the logging 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. To unlock the screen, press the STOP button.

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.

Replace module

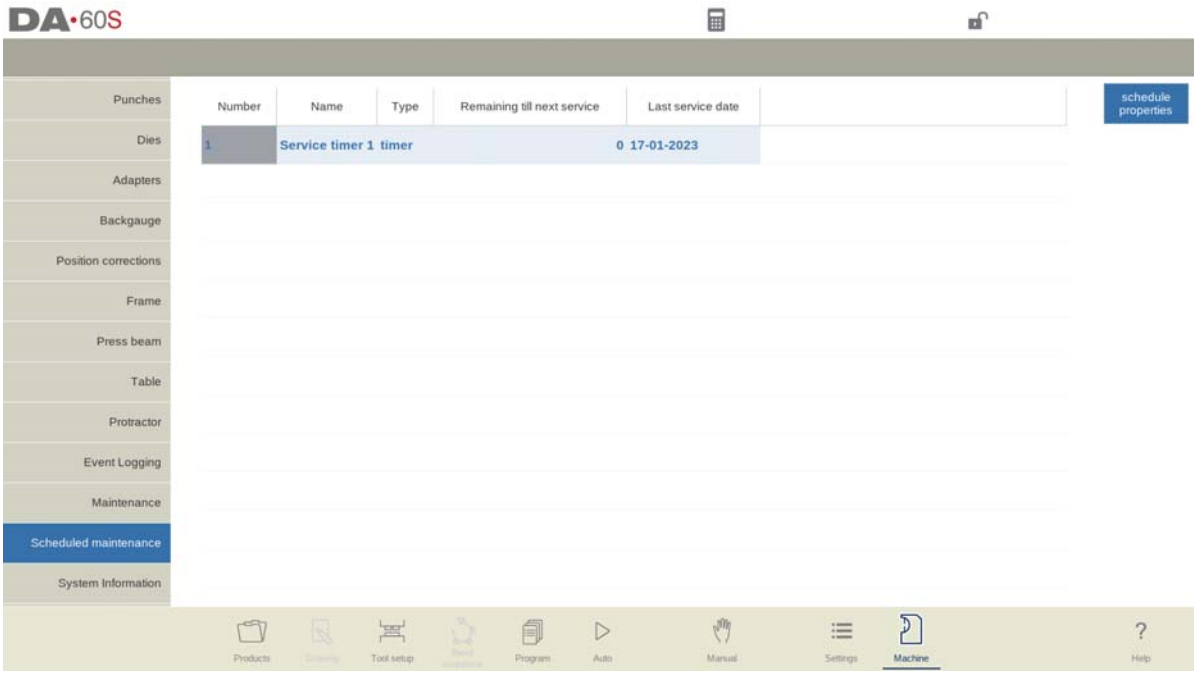
An axis module can easily be exchanged without special access following the exchange procedure. Only when applicable, this function will be available.
The module exchange procedure is as follows:

1. Switch off the DA-control and disconnect the DM-module that needs to be replaced.
2. Connect the new DM-module of the same type and switch on the machine.
3. An error message with the ID of the original module will appear on the screen.
4. Select the Machine mode and go to the Maintenance tab.
5. Tap Install Module.
6. A Window appears asking to replace the module with the new one. Tap Yes to accept.
7. The control will reboot to initialize the new module. If necessary the software of the new module will be updated automatically.

Restart controller
Restart the application.

10.15. Scheduled maintenance (optionally available)

On this tab scheduled tasks for maintenance are shown, when enabled by the machine manufacturer. Upon the set-up of the machine, maintenance related tasks are listed. These tasks are shown to the user based on a timer or machine usage (strokes). The user can, depending on the task, either do the maintenance himself and check the task done, or call a service technician to do the required maintenance. Upon the set-up, the task can be checked done with or without a specific code. In the list one can see the tasks, when they are planned to be done and when the last service has been given.



Schedule properties
The user can view the status of each task, and can also mark the task as done.

10.16. 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

KO-table

The KO-table number and description

Disk space

The available user disk space and total user disk space

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.

11. Profile-60S

11.1. Introduction

11.1.1. General

The Profile-60S offline program offers a similar user interface as a DA-60S line Delem control. The previous chapters, describing the operation and use of the DA-60S control should be used as a reference for using this offline software. This chapter will focus on a few special functions that are only available in the Profile-60S software.

11.1.2. System requirements

To run Profile-60S on a computer it should at least be equipped with the following features:

- IBM-compatible PC;
- Minimum Screen resolution of 1920x1080;
- Windows 10;
- video drivers supporting OpenGL V3.3 or higher;
- Free USB port.

11.1.3. Profile-60S distribution & manuals

Depending on the purchase package the Profile-60S software is distributed on a USB memory stick or can be generated at the control.

The complete Profile-60S package contains the following items:

- Profile-60S installation stick;
- USB license dongle;
- Content info and installation steps printed inside the Profile-60S box.

Profile-60S software distributed together with the control consists of:

- USB stick for installation and back-up purposes;
- USB license dongle;
- Installation steps printed inside the Profile-60S box.

11.2. Profile-60S installation

Before you can work with the Profile-60S software you must install it on the hard disk of your computer. The Profile-60S software contains a set-up program which facilitates the installation of the package on your hard disk. It is not required to have the USB license dongle connected to your PC to complete the installation procedure.

To install the Profile-60S software perform the following steps:

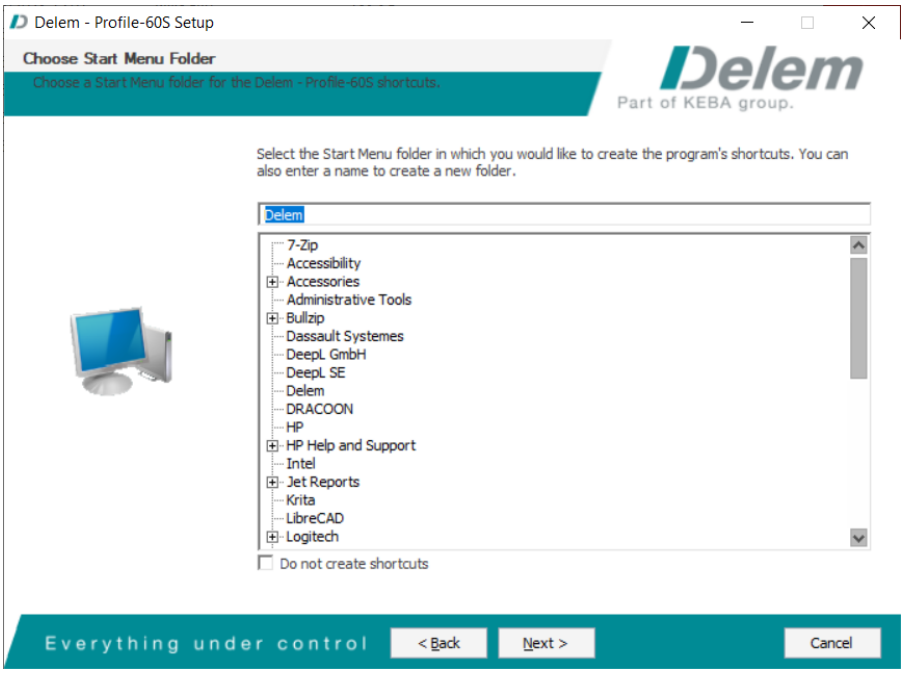
1 Insert the Profile-60S installation stick in the USB-port. Follow the instructions. This will automatically download the applicable Profile-60S installation files to your computer. After you can follow the proceeding steps as described below.

2 A ‘welcome’ window appears, offering to install the Profile-60S software. You can proceed by clicking the Next button. If you want to abort the set-up, you can click the Cancel button. If the set-up program is not automatically started, you can do so yourself by browsing to the Profile-60S installation stick in Windows Explorer and double-click the program ‘Delem.Profile-60S.setup.Vx.y.z.exe’ (where x.y.z is the version number) in the root directory of the Profile-60S installation stick.

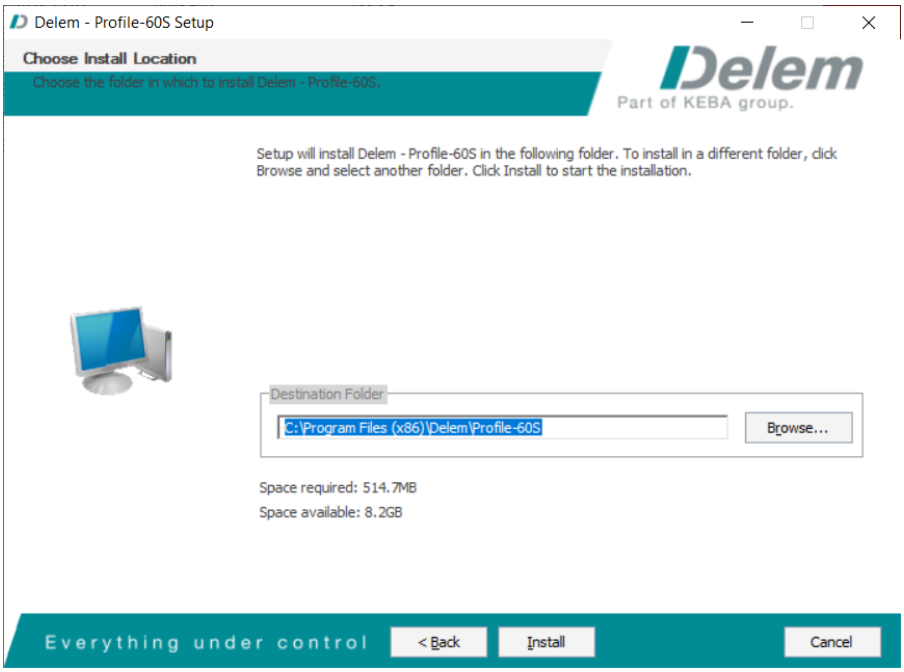
3 On the next window, the license for using Profile-60S is displayed. Read this license carefully. Select ‘I accept...’ to accept the terms of this license or leave it at ‘I do not accept...’. to abort the installation. If you have accepted the license, click ‘Next’ to continue the installation.

.

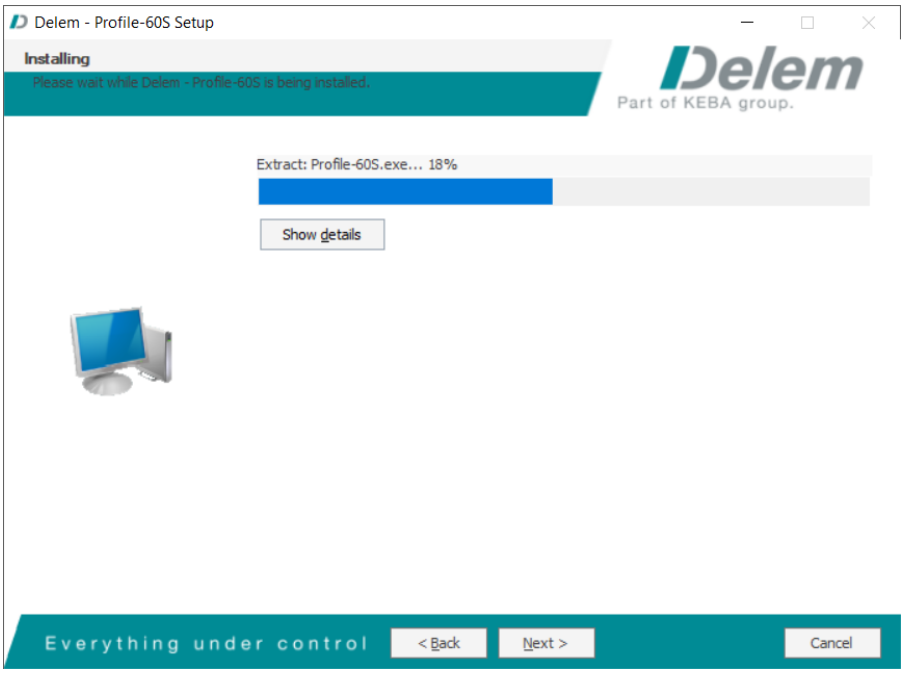
4 The set-up program will create a folder in the Windows start menu. In this page you can change the folder name and if required include it in an existing folder. Click ‘Next’ to continue.



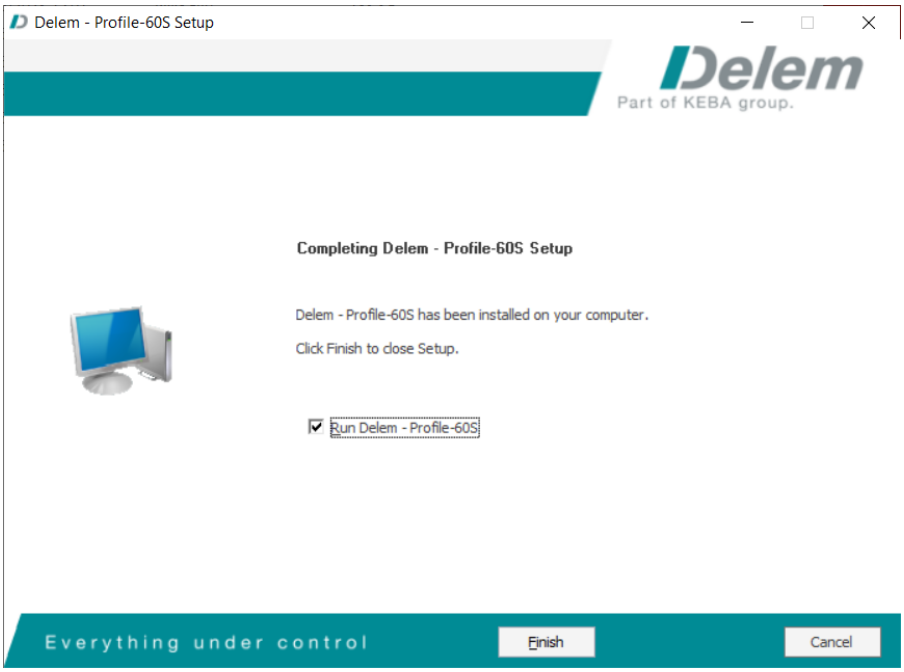
5 The next step lets you choose the folder where Profile-60S will be installed. The default folder is ‘C:\Program Files (x86)\Delem\Profile-60S’. You can change this easily by clicking the ‘Browse’ button and changing the path. Click “Install’ to start the actual installation of the software.



6 The set-up program will now start copying the necessary files to your hard disk. When finished the set-up program reports that the installation has been completed.



7 Press the ‘Finish” button to complete the installation. A reboot of your PC is not necessary.



Now you are ready to use Profile-60S. If the USB license dongle is connected to the PC Profile-60S will be launched. If the USB license dongle is missing an error message will be displayed.

After a default installation the following icons are present on your Windows Desktop:

Profile-60S software

Operation manuals of the DA-60S control

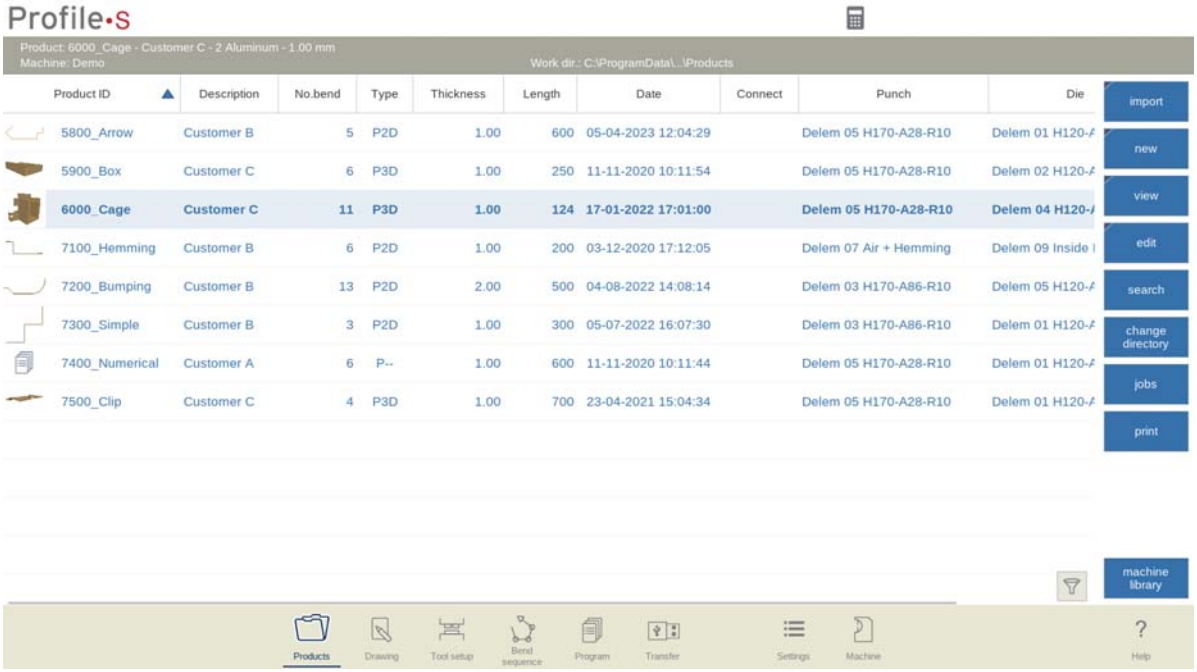
The installation program has also placed a new item named Delem, if no changes were made to the installation, in the Windows Start menu. The following items are shown in this menu:

Profile-60S	Profile-60S software
Profile-60S manuals	Operation manuals of the DA-60S control

11.3. Profile-60S operation

11.3.1. Profile-60S principle

The next figure shows the start page of Profile-60S:



The buttons on the bottom part of the screen are the mode selection keys. They are organized such that they indicate the work flow. On the left there is the products mode, here you can select an existing product or generate a new product. From here you work your way to the right.

For programming of products, programs and tools we refer to the built-in Help (? key) and the operation manual of the control. Operation manuals are included with the standard installation of Profile-60S.

Unlike a DA-60S control, Profile-60S has the following extra functions:

- With button 'machine library' (‘install machine’ in Profile-SL) you change the machine data. This data should match the settings and values as programmed in the control that is on the machine.
- When a product is selected in 'Products' mode they can be printed by pressing the button 'print'.
- In 'Transfer' mode the generated product including tooling can be transferred to a hard disk location, a USB memory device or a network location.

11.3.2. Machine library

It is important that the parameters in Profile-60S are the same as in the control on the machine. In this way the offline generated programs are 100% compatible with the control system. In Profile-60S you install several machines in order to use 1 offline programming station for the complete machine range on the production floor.

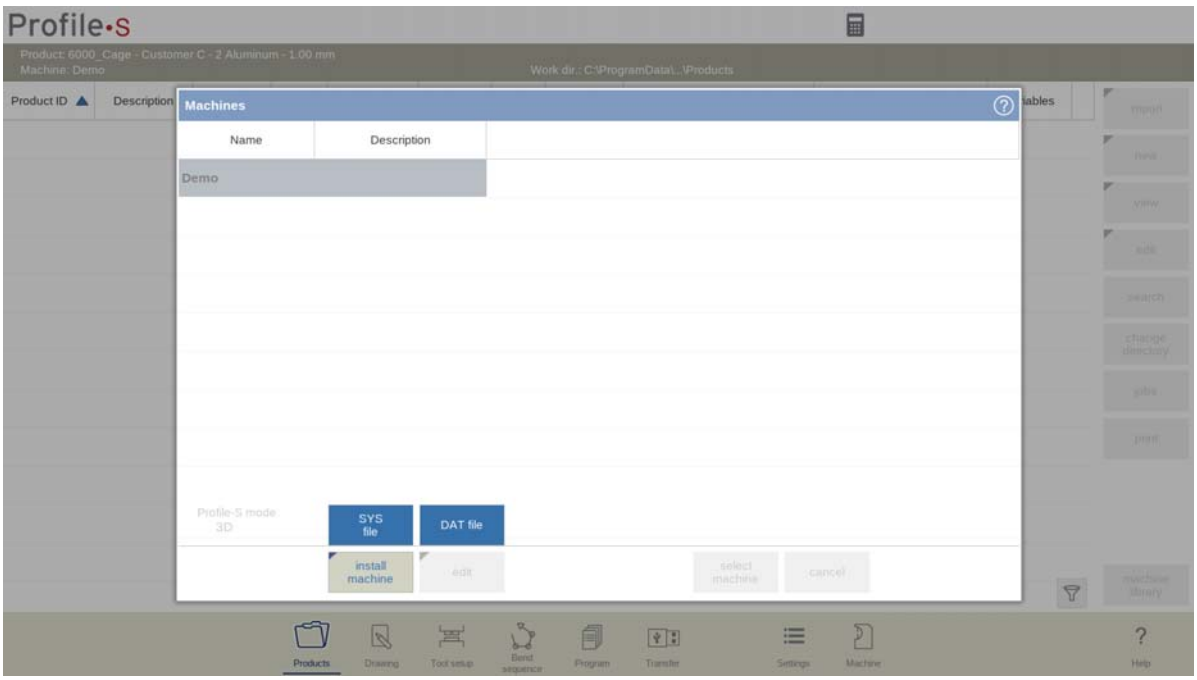
To add a machine in Profile-60S (replace the machine in Profile-SL, which supports only one machine) press the button 'machine library' (‘install machine’ in Profile-SL) in Products mode. A new window will pop-up and here press the button 'install machine':

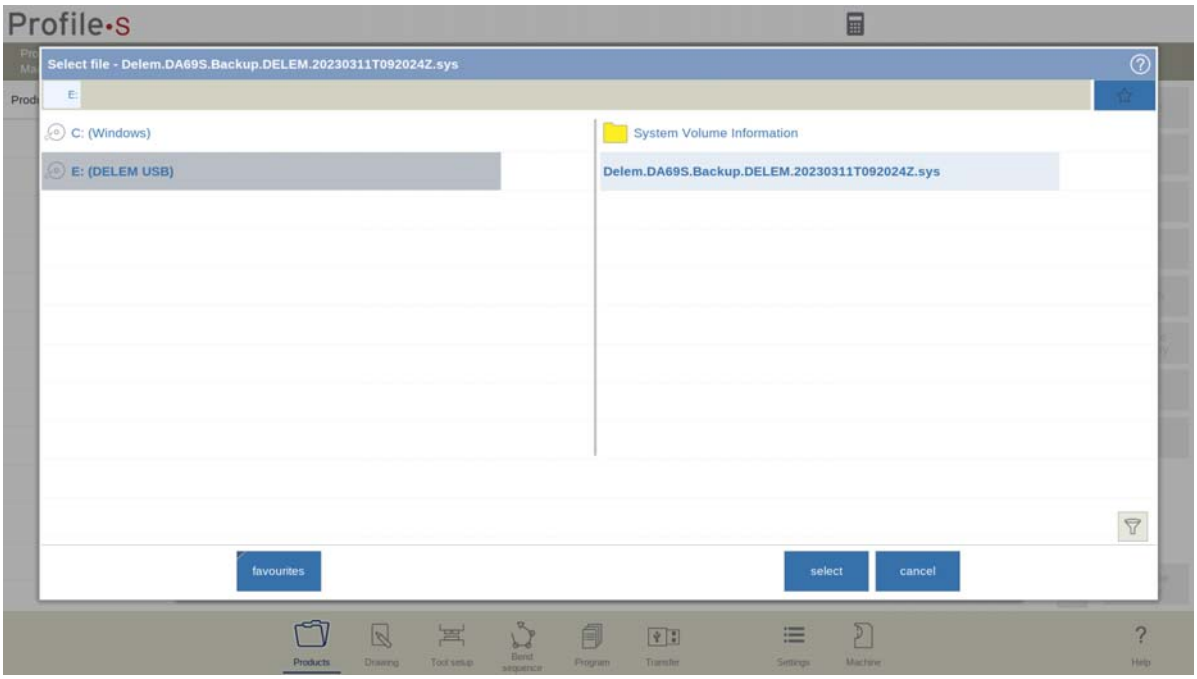
SYS file	A back-up file from a DA-60S control.
DAT file	A .dat file from a DA-60S control.

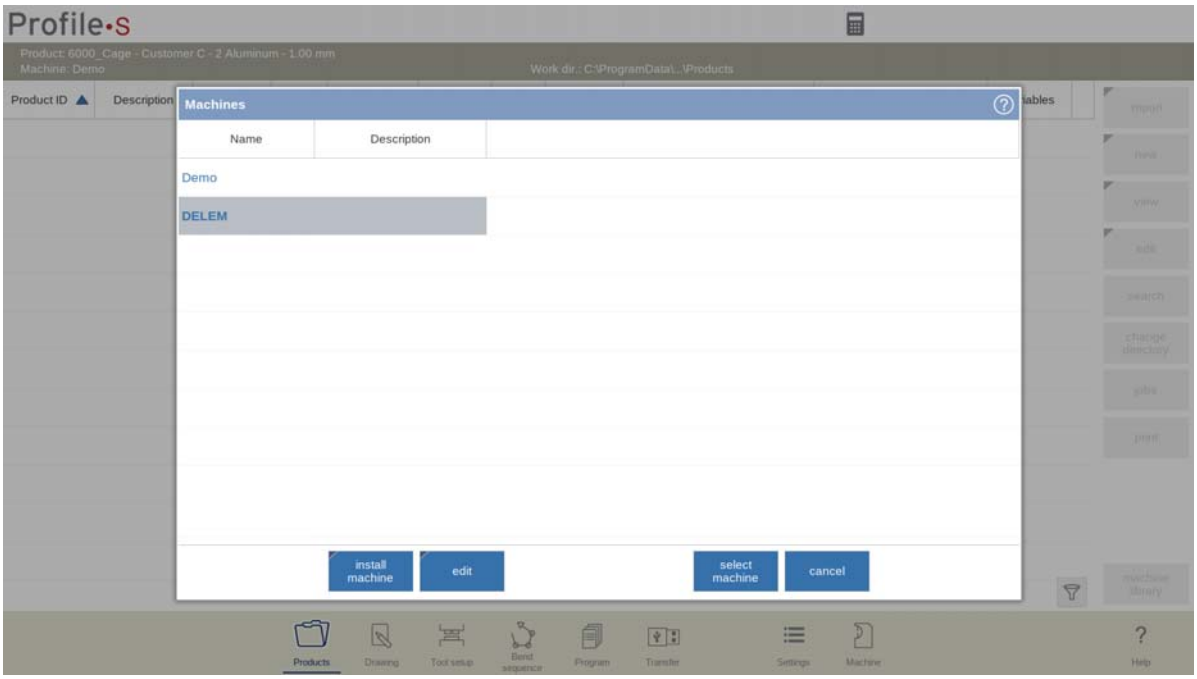
To create the SYS file on a DA-60S control select ‘backup system’ on the system information page in the Machine mode.

When Profile-60S is generated from the control a machine data file (i.e. a .dat file) is generated automatically and copied to the USB installation stick, and is loaded automatically into the Profile-60S software during installation.

In the example below a SYS file from a DA-60S control is imported. The name of the machine in Profile-60S is the same as the name of the control on the machine. After installation of the machine in Profile-60S the machine name can be changed (edit -> rename machine) if necessary.





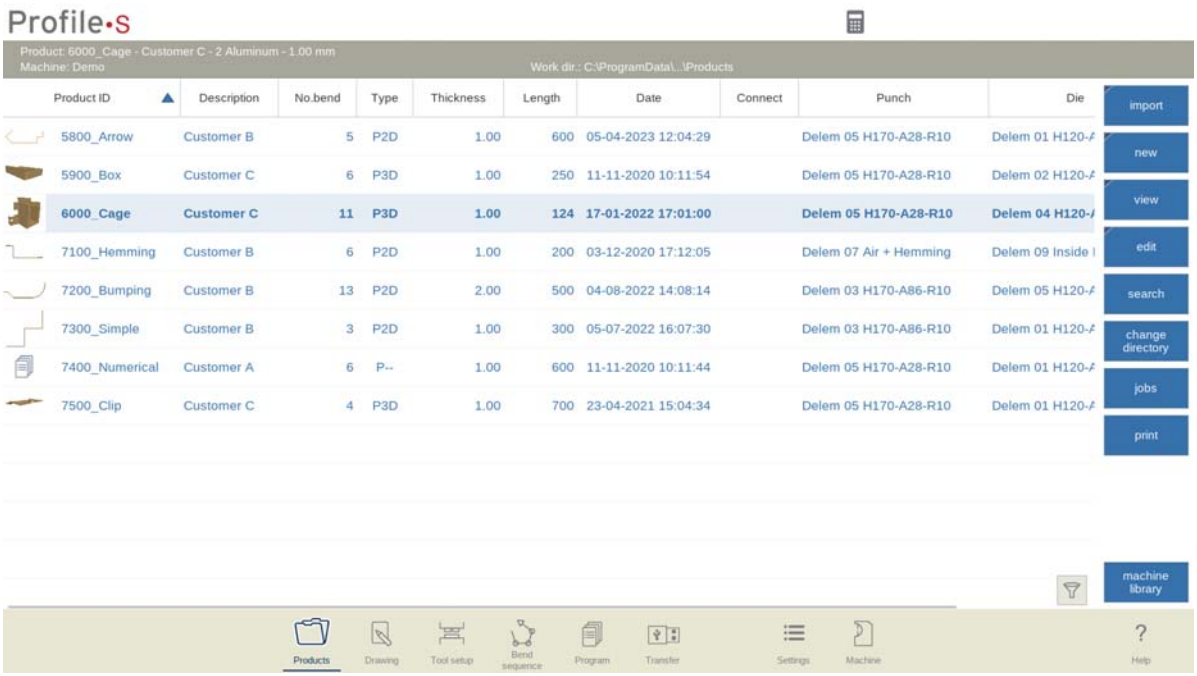


You can switch between the installed machines by selecting the required machine name and then press the button 'select machine'.

With the button ‘edit’ the selected machine can be renamed, removed or copied.

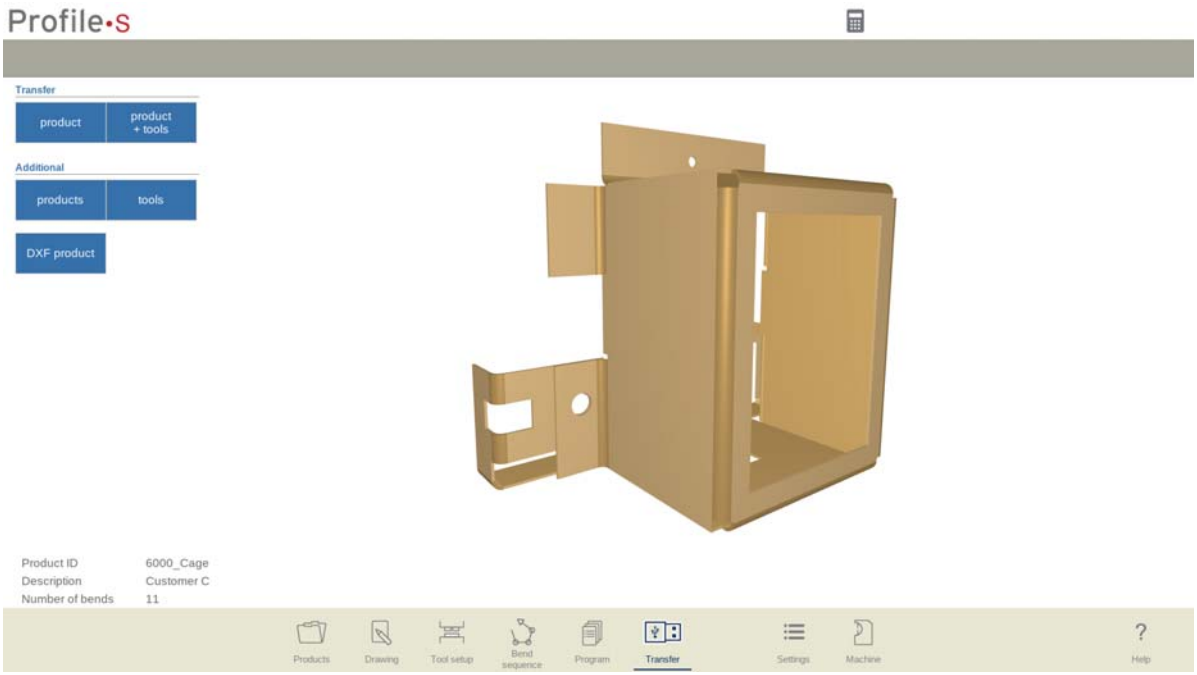
11.3.3. Print functionality (Profile-S2D and Profile-S3D only)

In Products mode it is possible to generate a print out of the selected product (Profile-S2D and Profile-S3D only). When pressing the button 'print' the standard Windows printer menu is opened to select the desired printing device and printer settings.



11.3.4. Transfer mode

When the product design and CNC program generation are finished, this data can be transferred in Transfer mode to e.g. a USB memory device.



In this mode there are 3 options:

product	Copy the program data to a specified directory.

product + tools	Copy the program data and used tooling data to a specified directory.
DXF product	Export the product including the bend deductions as a contour in a DXF file and copy it to a specified directory (Profile-S2D and Profile-S3D only).

Additionally other products besides the selected product, and other tools can be transferred as well.

11.3.5. Key combinations

The following key combinations are possible:

<alt> + <enter>
To enable/disable full screen mode.

<F11>
To disable/enable the Windows title bar and Windows frame.

A. Parameter index

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

90 Degree tolerance 5.17
Active bend allowance table 9.19
Adapt decompression 10.26
Adapt decompression 10.37
Adapt decompression 10.40
Angle 10.11
Angle 10.6
Angle 3.11
Angle 6.24
Angle 6.5
Angle 6.9
Angle 8.2
Angle 8.7
Angle / Angle 2 . . . . . .7.3
Angle 1 / Angle 2 . . . . . .7.2
Angle calculation 10.66
Angle correction database 9.14
Angle correction programming 9.15
Angle definition 2.29
Angle Measurement Device 10.65
Angle sel. 6.18
Angle tolerance 10.65
Application 10.74
Auto bend change mode step 9.22
Auto enter corr.α 10.65
Auxiliary axis 6.8
Auxiliary axis 7.3
Auxiliary axis 8.6
Backstop against sharp angle allowed . .5.16
Backstop-die, intermediate bend 5.16
Bend allowance 3.3
Bend type 3.11
Bending length 3.2
Bending length 6.5
Bending length 8.2
Blank length 6.19
Bottom height 10.33
Bottom heigth (BH) 10.44
Bottom width 10.33
Bottom width (BW) 10.42
Bottoming force factor 9.20
Boundary 1/2/3 . . . . . .9.25
Bumping 6.24
Center height (CH) 10.42
Center height (CH) 10.43
Central angle 3.13
Clamping correction 9.23
Code 6.7
Command panel side 9.3
Computed radius 3.11
Controller Start 10.68
Controller Stop 10.68
Conversion of special tools 10.64
Corr.α 1, Corr.α 2 . . . . . 8.2
Corr.Y 8.3
Correction 8.7
Data preparation bend allowance 9.20
Date format 9.26
Decomp speed 6.7
Decomp speed 8.4
Decompression 6.6
Decompression 8.4
Default adapter 10.22
Default adapter 10.9
Default angle 2.29
Default bend allowance 9.18
Default decompression speed 9.17
Default delay time 9.17
Default dimensions 9.18
Default dwell time 9.17
Default hem opening 9.18
Default lay on position 10.50
Default material 9.18
Default part support return speed 9.23
Default prebend angle 9.18
Default pressing speed 9.17
Default radius 9.18
Default step change code 9.17
Default wait for retract 9.17
Default X-axis speed 9.17
deflect 7.3
deflect 7.3
Deflect method 6.5
Deflect method 8.2
Delay time 6.7
Depth 7.3
Description 10.21
Description 10.34
Description 10.48
Description 10.56
Description 10.58
Description 10.8
Die 6.8
Die 8.5
Die adapter 6.8
Die adapter 8.5
Die width 10.33
Dimension type 2.29
Dimensions 3.2
Discrete positions 10.35
Disk space 10.74
Display time 9.26
Dwell time 6.6
Dwell time 8.4
E module 9.4
Edge tolerance 5.17
Effective bend length 6.5
Equal bumping-segments 3.13
Equal bumping-segments 6.24

Error Message 10.68
Event logging 10.67
Fast hemming 6.5
Fast hemming 8.3
Filename 10.67
Finger (1/2/3/4) 6.8
Finger (1/2/3/4) 8.6
Finger height (FH) 10.52
Finger length (FL) 10.52
Finger radius 10.51
Finger width 10.51
Fixed part width (FW) 10.39
Force 6.6
Force 8.4
Frame 10.45
Frame height 10.46
Frame thickness 10.46
Frame type 10.45
Frame width 10.46
Front extend ratio 5.13
Front extend ratio accept 5.13
Gap (G) 10.43
Gauge height (H1/H3/H4) 10.52
Gauge height (H2) 10.52
Gauge length (L1) 10.52
Gauge length (L2/L3) 10.52
Gauge positions 10.52
Gauge R offset 10.50
General angle correction programming 9.15
Guide direction 10.25
Height 10.10
Height 10.11
Height 10.13
Height 10.19
Height 10.23
Height 10.25
Height 10.28
Height 10.30
Height 10.33
Height 10.57
Height 10.58
Height 10.6
Height (H) 10.36
Height (H) 10.39
Height (H) 10.42
Height 1 . . . . . 10.9
Height 2 . . . . . 10.9
Help language 9.2
Hem angle (Ha) 10.36
Hem angle (Ha) 10.39
Hem opening 10.26
Hem opening 3.14
Hem opening 6.5
Hem opening 8.3
Hem opening (HO) 10.36
Hem opening (HO) 10.39
Hemming height 10.11
Hemming load opening 10.10
Hemming load opening 10.11
Hemming resistance 10.10
Hemming resistance 10.12
Hemming resistance 10.23
Hemming resistance 10.26
Hemming resistance 10.28
Hemming resistance 10.36
Hemming resistance 10.39
Hemming width 10.10
Hemming width 10.11
Hemming width 10.23
Hemming width 10.28
Horizontal projection 3.8
Hours 10.71
ID 10.17
ID 10.48
ID 10.5
ID 10.56
ID 10.58
Inch/mm-select 2.29
Inch/mm-select 9.2
Initial angle 6.10
Initial angle 8.7
Initial angle correction 9.15
Initial angle speed 6.10
Initial angle speed 8.7
Inside hemming die type 10.26
Inside hemming die type 10.37
Inside hemming die type 10.40
Inside radius (IR) 10.42
Inside radius (IR) 10.43
Intermediate R for X-movement 9.24
Intermediate R for Z-movement 9.24
Intermediate X for Z-movement 9.23
Key sound 9.3
Keyboard layout 9.3
KO-table 10.74
Language 9.2
Layer selection 2.30
Lay-on backstop limit 5.17
Length 3.23
Lock touch screen when started 9.22
Logging mode 10.71
Lower hemming width 10.26
Lower hemming width (LW) 10.36
Lower hemming width (LW) 10.39
Lower part height 10.26
Lower part height (LP) 10.37
Lower part height (LP) 10.40
Lower part width 10.26
Lower part width (PW) 10.36
Lower part width (PW) 10.40
Machine description 9.16
Machine DXF import functions 10.64
Machine ID 9.16
Machine ID check 9.16
Manual mode store angle corrections . . 9.15
Material 3.2

Material 6.18
Material 8.5
Material name 9.4
Max inter segment distance 4.9
Maximum file size 10.67
Maximum length 3.23
Message sound 9.3
Method 6.4
Method 6.9
Method 8.2
Method 8.6
Minimal circle diameter 2.29
Minimum angle 10.43
Minimum angle 10.44
Minimum distance between fingers . . .10.53
Minimum finger-product overlap 5.17
Minimum length 3.23
Minimum V opening 10.35
Minimum Y opening 5.14
Mode Change 10.68
Moving part width (MW) 10.39
Mute 10.22
Mute 10.35
Mute 6.6
Mute 8.3
Name 3.23
Number of openings 10.30
Number of openings 10.36
Number of segments 3.13
Number of segments 6.24
Number of steps 10.35
Opening 6.6
Opening 8.4
Optimization degree 5.13
Option ID 10.74
Parallel 6.6
Parallel 8.3
Parallelism offset 9.22
Path 10.67
Position correction 10.54
Prebend angle 3.14
Precision 3.9
Preferred 10.22
Preferred 10.9
Preferred Radius 3.11
Preferred radius 6.5
Press beam 10.45
Pressure correction 9.23
Product Complete 10.68
Product description 3.2
Product description 6.18
Product ID 3.2
Product ID 6.18
Product position 6.5
Product position 8.2
Product-product collision 5.14
PST 7.3
PST 7.3
PST-axis 6.8
PST-axis 8.6
Punch 6.8
Punch 8.5
Punch adapter 6.8
Punch adapter 8.5
Radius 10.11
Radius 10.13
Radius 10.19
Radius 10.25
Radius 10.28
Radius 10.31
Radius 10.33
Radius 10.37
Radius 10.40
Radius 10.6
Radius 10.9
Radius 3.13
Radius 3.3
Radius 6.24
Radius factor 5.13
Radius height 10.13
Recomputation of numerical values . . . 9.15
Repetition 6.7
Repetition 7.4
Resistance 10.11
Resistance 10.13
Resistance 10.20
Resistance 10.21
Resistance 10.25
Resistance 10.28
Resistance 10.31
Resistance 10.34
Resistance 10.38
Resistance 10.43
Resistance 10.48
Resistance 10.6
Resistance 10.8
Resistance per m/mm selection 9.2
Retract 6.8
Retract 8.6
Retract optimization 5.18
Return speed 6.9
Return speed 8.7
R-position 6.9
R-position 8.6
Safety stop angle 6.9
Safety stop angle 8.7
Scaling factor 2.29
Sequencer 10.74
Shape 10.17
Shape 10.48
Shape 10.5
Sheet thickness limit 5.15
Show notes automatically 9.24
Side length 3.15
Special finger 10.53
Speed 6.7

Speed 6.8
Speed 6.9
Speed 8.4
Speed 8.6
Speed 8.7
Step Change 10.68
Step mode 7.4
Stepsize 10.35
Stock 7.4
Stock count mode 9.22
Strain hardening exponent 9.4
Strokes 10.71
Support type 10.48
Support type 10.8
Table 10.45
Tensile strength 9.4
Thickness 3.2
Thickness 6.18
Thickness 7.3
Thickness 8.5
Throat height 10.46
Throat position 10.46
Throat radius 10.46
Throat width 10.46
Tilt angle 6.9
Tilt angle 8.6
Tilt clearance 6.9
Tilt clearance 8.7
Tilt speed 6.9
Tilt speed 8.7
Time format 9.26
Tolerance 2.29
Ton/kN select 9.2
Tool holder resistance 10.57
Tool holder resistance 10.58
Tool length tolerance 5.14
Top width 10.13
U / V opening 10.31
U / V opening 10.33
U / V opening 10.37
U height 10.28
U opening 10.28
Unfolded dimensions limit 5.14
Upper hemming width 10.26
Upper hemming width (UW) 10.36
Upper hemming width (UW) 10.39
Upper part width 10.25
Upper part width 10.40
Upper part width (TW) 10.36
Use in auto mode 3.23
Use tool length tolerance 4.9
V angle 10.19
V angle 10.25
V angle 10.31
V angle 10.33
V angle 10.37
V angle 10.40
V axis offset 10.35
V bottom 10.19
V bottom 10.25
V bottom 10.31
V bottom 10.38
V bottom 10.41
V Height (VH) 10.39
V opening 10.19
V opening 10.25
V opening 10.31
V opening 10.38
V opening 10.41
Vertical projection 3.8
W Opening (W) 10.42
Wait for retract 6.7
Wait for retract 8.5
Width 10.11
Width 10.19
Width 10.28
Width 10.30
Width 10.33
Width 10.6
Width 10.9
Width (W) 10.43
Wing height (WH) 10.42
Wing height (WH) 10.43
Wing width (WW) 10.43
X correction programming 9.15
X offset 10.53
X-axes 7.3
X-safety offset 9.23
Y opening default 9.17
Y1 / Y2 . . . . . 7.2
Y1/Y2 independent 9.15
Y-axis 8.3
Y-axis (Bend position) 6.6
Z-distance 9.20