

DA-41

The compact press brake control

Operation Manual V1, GB

Table of contents

1. Introduction1

2. Hardware.....3

2.1 Front panel.....3

2.2 Display4

3. Operating modes5

3.1 Introduction5

3.2 Basic navigation.....5

3.3 Product programming6

3.4 Tool programming.....7

3.5 Program constants.....8

3.6 Manual movement9

3.7 Product selection10

3.8 Key lock13

4. Programming products and tools15

4.1 Introduction15

4.2 Tools programming15

4.3 Product programming18

4.3.1 Program screen18

4.3.2 Angle programming20

4.3.3 Program edit21

4.3.4 Direct axis programming.....21

5. Production mode22

1. Introduction

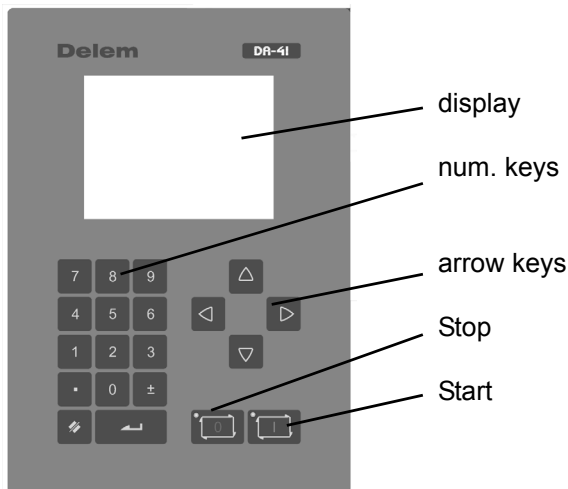


The DA-41 is a programmable control for conventional press brake machines.

2. Hardware

2.1 Front panel

The following picture shows a representation of the front panel. The panel contains one display and several keys for programming and axis control. The display will be explained in the next section.

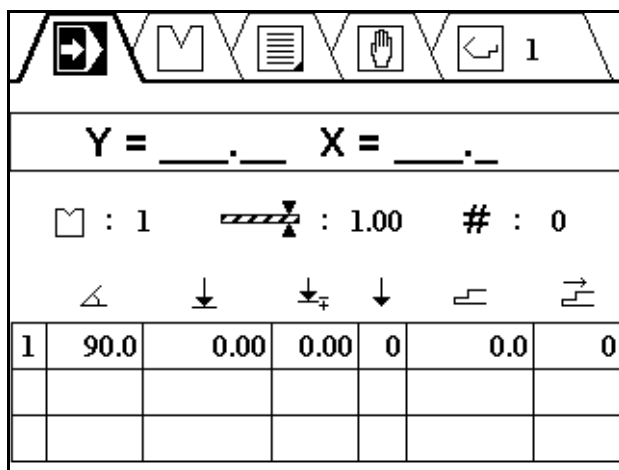


The keyboard contains the following keys:
<https://www.machinemfg.com/>

- 10 numerical keys (0 - 9)
- decimal point
- plus/minus toggle
- clear key, to clear a parameter value
- enter key, to enter a typed value
- arrow keys, to select various parameters
- stop key (0) (incl. status LED)
- start key (1) (incl. status LED)

2.2 Display

The DA-41 has a monochrome LCD display, 320 x 240.



At the top of the screen, the different operation modes are represented by symbols.

3. Operating modes

3.1 Introduction

The control has several modes for programming and operation:



product programming and execution



tool programming



program constants



manual movement



bend program selection

It is indicated in the top of the screen which mode is active.

3.2 Basic navigation

- To switch between modes, press the <arrow up> key until the mode symbol is highlighted. Then use the <arrow left> and <arrow right> key to move to the required mode.
- Within a mode, use the arrow keys to move around between parameters and fields.
- To program a parameter, move the cursor to the appropriate symbol and type the required value. Press ENTER to confirm the value. A programmed value is also confirmed when














an arrow key is pressed. In some cases, the <arrow left/right> keys can also be used to change values.

- Steps and their parameters can only be programmed and viewed when the control is stopped.
- To start execution of a program, press the 'Start' key.

3.3 Product programming



Select this symbol for product programming and execution

     1						
Y = ____ X = ____						
 : 1  : 1.00 # : 0						
     						
1	90.0	0.00	0.00	0	0.0	0

This programming screen is divided into three areas:

- current Y and X position
- general product properties
- table with bend steps, each step has several parameters

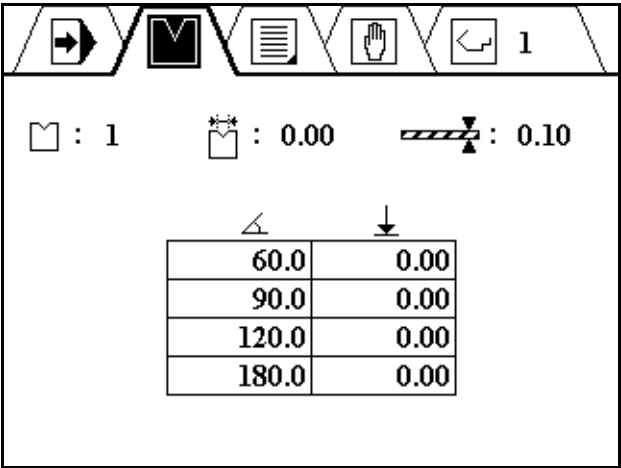
In the upper right corner of the screen, the number of the currently active program is shown.

In this screen, a product program can be edited and executed. Programming of products will be explained in chapter 4.

3.4 Tool programming



Select this symbol for tool programming.

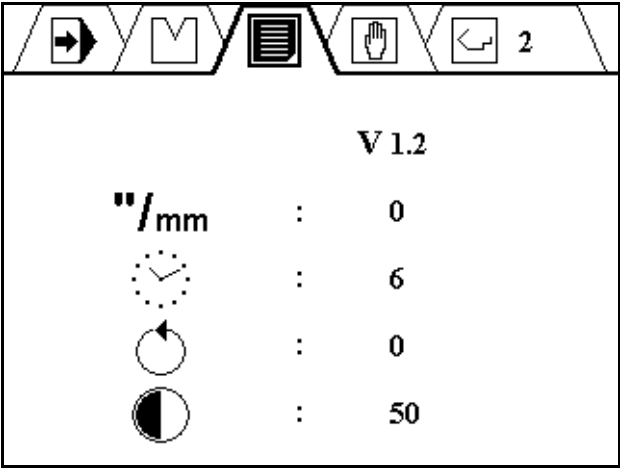


In this window, a set of tool properties can be programmed. During product programming, these tool properties are used for the calculation of Y-axis values.
Programming of tools is explained in chapter 4.


3.5 Program constants








Select this symbol for the program constants




In this screen, several control settings can be viewed and changed.

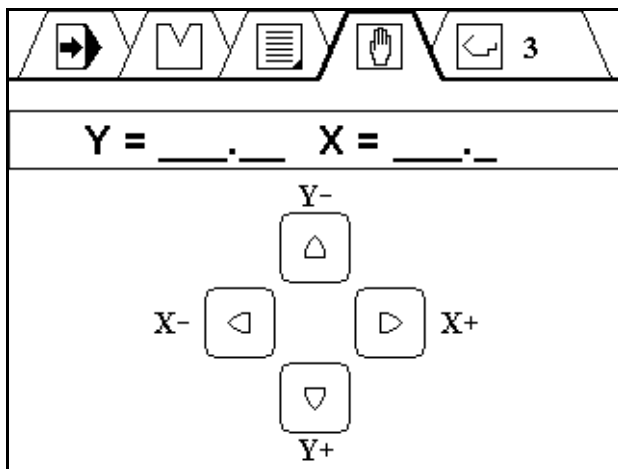
V x.x		Software version of this control
"/mm	:	Switch to select units: 0 = millimeters 1 = inches
	:	hours: the number of hours the machine has been running. This parameter is read-only.

	:	strokes: the number of bend strokes the machine has executed. This parameter is read-only.
	:	contrast (%)
	:	brightness (%)
	:	Current X-axis position. Program the current position of the X-axis. The control will retain this position value after power-down and automatically assume the same position value after power-on. After power-on, verify the axis position and correct this parameter when necessary. This parameter is only available in case there is no reference system for this axis installed on the machine. https://www.machinemfg.com/
	:	Current Y-axis position. This parameter works the same as for the X-axis.

Parameters (except the read-only ones) can be edited using the arrow keys (left and right) or they can be programmed numerically.

3.6 Manual movement

	Select this symbol for the manual movement screen
---	---











In this screen the X- and Y-axis can be moved manually with the arrow keys. This can only be done when the control is not started. On the screen is indicated how an axis can be moved with a particular arrow key.

To move an axis gradually, press the arrow key once. To move an axis at high speed, press the arrow key continuously. Press the 'stop' key to leave this mode and return to the product programming screen.

3.7 Product selection



Select this symbol for the product selection screen

				 1
 N 				
1	1	1.00	1	
2	0	1.00	1	
3	0	1.00	1	
4	0	1.00	1	

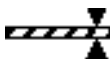
The symbols above each column have the following meaning:



Product number.

N

The number of bends in this product.



Sheet thickness of this product.



Used tool in this product.

There are 100 product programs available in the control. To edit a program, it must be selected first. By default, all programs consist of 0 steps. As soon as a program has been selected, one step is added to the program.

To select a program, move the cursor to the required program number and press ENTER to select it. This program number is also displayed in the upper right corner of the screen, beside the mode symbol.

To select a program with a high number (e.g. 74), press the first digit of this number (e.g. 7). The cursor will automatically jump to the group of numbers that begin with this digit. Then use the arrow keys to get to the correct program number.

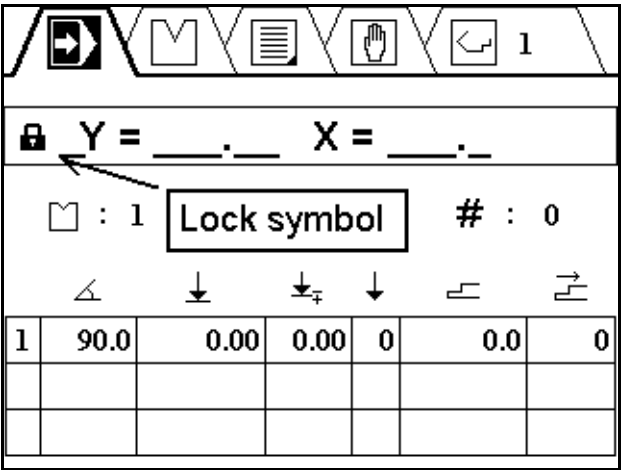
When a program has been selected, the control automatically switches to the programming mode.

A program remains active until another program is selected or until it is deleted.

To delete a program, move the cursor to the program number and press the clear key. The program number remains, but the number of steps is reset to 0.

3.8 Key lock

The control is equipped with a key lock function, to prevent unauthorised programming.



It depends on machine settings whether the key lock is implemented.

If the control is unlocked, it can be programmed as described in this manual.

If the control is locked, it is indicated by a lock symbol in the upper left corner of the programming window. When locked, the following restrictions apply:

- programs cannot be created or edited
- programs cannot be deleted
- tools cannot be created or edited

The following actions are still possible when the control is locked:

- programs can be selected (if they consist of one or more steps)
- programs can be executed

-
- Y-axis corrections within programs can be changed
 - program constants can be changed
 - axes can be moved through the manual movement screen

4. Programming products and tools

4.1 Introduction

Programming of products and tools can be done very straightforward.

Normally, bend programs can only be programmed when tools are available, so tools must be programmed first.

4.2 Tools programming

: 1

: 0.00

: 0.10

60.0	0.00
90.0	0.00
120.0	0.00
180.0	0.00

A tool is programmed on the control by entering a few essential tool values. These values will be used for automatic bend depth calculation during bend programming.

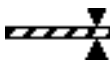
At the top of the tools window, three general tool properties are shown:



Tool number. Range: 1 - 15.



V-opening of the die.



Intended product thickness for this tool.

The thickness programmed here will be important during product programming. The V-opening value serves for reference only and will not be used for calculations.

At the bottom of the tool window, a table is shown with angle values and Y-axis values.



Angle set point

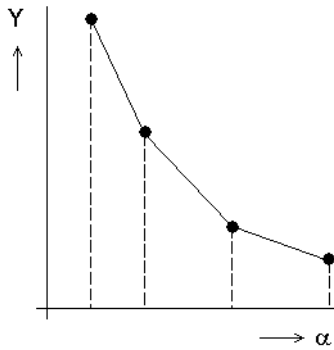


Y-axis value (bend depth) at this programmed angle value.

This table will be used to calculate the necessary bend depth to obtain a certain angle.

<https://www.machinemfg.com/>

You must program four set points in the table. For angle values which are not programmed in this table, the control will use interpolation between two adjacent set points to calculate a corresponding Y-axis value.



In case there are less than four set points, program the required set points and leave the Y-value at zero for the invalid set points. The angle values in the first column must be programmed in increasing order.

4.3 Product programming

4.3.1 Program screen

Y = ____ X = ____

: 1

: 1.00

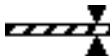
: 0

1	90.0	0.00	0.00	0	0.0	0

As already explained in section 3.3, this programming screen shows the following information: current axes positions, general product properties and a table with bend steps. In middle of the screen, the general product properties are displayed:



Number of the tool which is used for this product.



The thickness of this product.



Stock counter: count the number of products.
If programmed to 0, the counter will increase after each finished product. If programmed higher than zero, the counter will count down. When it has reached 0, the control will stop and the stock counter will be reset to the programmed value.

Below this line, a table is shown with the bend steps of this product. Each line represents one bend step. The first column contains the bend number. For each bend step, the following properties can be programmed:



Angle of this bend.
Program here the required angle value.
This parameter is available when a tool is programmed.



Y-axis bend depth.
When an angle is programmed, this value is calculated from the tool properties. A higher value means a lower beam position.



Correction on the Y-axis bend depth.
When angle programming is used, this correction is used to correct faulty Y-axis values. A positive correction means a lower beam position.



Y-axis opening time.
The required opening time after a bend, programmed as a percentage of the maximum opening time. This parameter depends on machine settings.



Beam high speed yes/no (1/0).
Select high travel speed for the beam during this bend, yes or no. This parameter depends on machine settings.



Back gauge position.
The required back gauge position for this bend.



Retract distance of the back gauge.
This parameter depends on machine settings.

A maximum of 25 bends can be programmed.

For quick programming:

- put the cursor on the angle or Y-axis parameter
- type the required value and press ENTER: the cursor automatically jumps to the backgauge parameter
- type the required value and press ENTER
- repeat these steps for other bend steps

Use the arrow keys to move to a specific parameter.

4.3.2 Angle programming

Angle programming is possible when a tool is programmed. When an angle value is entered, the control automatically searches the tool data for a matching angle value to obtain the corresponding Y-depth from. If no matching value is found, the control will calculate a Y-depth value from the existing tool data. To program a different Y-depth for a bend, simply change the bend depth value and press ENTER. Notice that the Y-depth value will not change, but the correction of the Y-depth will change to reflect recent modifications. Of course, the same can be achieved by programming the correction directly. If the product thickness in the bend program is different from the thickness programmed in the tool then this difference will also be added to the resulting Y-axis value. So a larger thickness value

in the product program will result in a lower Y-axis value (the beam will travel less deep) and vice versa.

4.3.3 Program edit

To add a bend to the program:

Go to the last bend of the current program, move the cursor to the first column (bend number) and press ENTER. A new bend is added, which has the same parameter values as the previous bend.

<https://www.machinemfg.com/>

To delete one or more bends:

Go to the last bend you wish to keep, move to the first column (bend number) and press the 'clear' button. All bends after the current bend are deleted.

The first bend cannot be deleted in the programming screen. To delete the program, go to the program selection screen and delete the program with the clear key.

4.3.4 Direct axis programming

To create a bend program without using the programmed tools, program 0 for the tool number. The angle parameter will no longer be available. Now it is possible to enter plain axis values.

5. Production mode

Program execution is carried out from the programming screen. From here, it is possible to execute one or more bend steps.

Y = ____ X = ____						
□ : 1 : 1.00 # : 0						
↓ ↓ ₊ ↓						
1	90.0	117.00	0.00	0	0.0	0



To start a program, press 'Start'. The control will begin execution of the step on which the cursor is placed.



To stop execution, press 'Stop'.

A program is repeated until 'Stop' is pressed or until the stock counter (#) has reached 0 after downcounting. Programs can only be executed on the machine when the pump for the hydraulic system is running.



This symbol indicates the pump is not started.

Delem

The DA-41 is a compact unit for control of conventional press brakes.