



© 2017 Cybelec S.A.

All Rights Reserved

Copying, reproduction, modification, distribution, display or transmission of any of the contents of this manual for any purpose without the prior consent of Cybelec S.A. is strictly prohibited.

TABLE OF CONTENT

Safety	1
General Safety.....	1
Signs and Icons appearing in this Manual.....	2
<i>General warning</i>	2
<i>Information</i>	2
<i>Settings</i>	2
<i>Navigation</i>	2
Getting started with CybTouch 12	3
Screen Map	4
General navigation	6
<i>Menu Button</i>	6
<i>Status Pages Zone</i>	6
<i>Screen Cleaning</i>	6
Status page	7
User Preferences	7
<i>Language</i>	8
<i>Length Units</i>	8
<i>Force Units</i>	8
<i>Force/Length Units</i>	8
<i>Sigma Units</i>	8
<i>Show axes position values</i>	9
<i>Show Graphical part page</i>	9
<i>Show page L-alpha num</i>	9
<i>Show bent part</i>	9
<i>Bend numerical page</i>	10
<i>Show manual button</i>	10
<i>Materials</i>	11

<i>BDC correction</i>	12
<i>RFLink</i>	12
<i>Clear indexation</i>	12
<i>USB Explorer</i>	12
<i>Default retraction</i>	12
<i>Set Clock</i>	13
<i>Touchscreen Calibration</i>	13
<i>Offset Pinch Point function</i>	13
<i>Brightness xx% Eco xx%</i>	13
<i>Program counter mode</i>	14
<i>Show L first</i>	14
<i>Disable recalculation at product load</i>	14
Manual Axes Movement	15
<i>Desynchronized Beam</i>	15
Service Page	16
<i>Set Axis</i>	16
<i>Maintenance</i>	16
Defrag.....	17
Format.....	17
Internal backup	17
<i>Information</i>	17
<i>Configuration options</i>	18
Serial number	18
Computer ID.....	18
Option list	18
New option code.....	18
USB Transfer	19
<i>Precautions</i>	19
<i>Importing parts from other Cybelec NCs</i>	19

Basic Page Description	20
Bend Numerical Page	20
<i>Available functions on the Bend Num page.....</i>	<i>20</i>
Program number	20
Current step (sequence) number.....	20
Set-point beam (Y axis) / bend angle	21
Set-point back gauge (X axis) / flange length	21
<i>Extra Bending Parameters.....</i>	<i>22</i>
Set-point back gauge (R axis - optional)	22
Material	22
Material thickness.....	22
Material sigma.....	23
Back gauge retraction	23
Speed change threshold and bending speed	23
Bending length.....	23
Step bending.....	23
Slow speed return	24
Dwell time	24
Force.....	24
Opening (TDC).....	24
Crowning.....	25
Back gauge finger type	25
Number of parts	25
Back gauge manual control	25
Offset Pinch Point	26
Copy to All Function	26
Hemming.....	26
Tools Management	27
Punches.....	27
<i>How to create or modify a punch?.....</i>	<i>28</i>

Dies.....	29
<i>How to create or modify a die?</i>	30
Naming Tools.....	31
<i>Punches</i>	31
<i>Dies</i>	31
Back Gauges.....	32
Creating a Part Program.....	33
TouchProfile Mode	33
Bend Sequencing	34
Numerical Mode.....	36
L-Alpha Mode	37
Bending and Corrections.....	38
<i>Semi-Automatic mode</i>	38
<i>Angle Correction (Y axis)</i>	39
<i>Crowning</i>	39
<i>Back gauge (X & R axes) Correction</i>	40
Next-Part Function	40
<i>Cycles and quantities</i>	41
CybTouchConverter	42
<i>How to Convert a Part-Program?</i>	42
Managing Programs.....	44
Saving a Program.....	44
Creating a Program	44
Loading a Program.....	45
Deleting a Program	45
Browsing Programs.....	45
EasyBend Page	46
Making a bend on the EasyBend Page	46

Error and Warning Messages	47
Warning Messages	47
Error Messages.....	50
Beam Error Messages	56
Internal Reset Error Messages	57

SAFETY

GENERAL SAFETY



The users must have **READ** and **UNDERSTOOD**, but most of all must **RESPECT** the directives described in this manual.

All people coming into contact with the machine on which the numerical control is installed, whatever their function or whatever state the machine is in (assembly, disassembly, start-up, production, maintenance, repairs) must have read and understood the requirements concerning the security and the entirety of the directives of operation described in the manuals delivered with the machine.



The operator must be properly trained to work with the machine on which the numerical control is installed. Improper use of the numerical control can cause heavy damage on equipment and/or injuries to people.



Modification of machine parameters can cause important material damage or lead to irregular product quality.

Do not expose the numerical control to excessive humidity so as to avoid any risk of electrocution and any deterioration of the equipment.

Make sure the numerical control is disconnected from the mains power before carrying out any cleaning. Do not use liquids based on alcohol or ammoniac.

In case of malfunction of the numerical control, call a technician.

Do not expose the numerical control to direct sun rays or any other heat source.

Do not place the numerical control in the neighborhood of magnetic equipment such as transformers, motors or devices which generate interference (welding machines, etc.)

SIGNS AND ICONS APPEARING IN THIS MANUAL

While using this manual, you will come across the signs and icons represented here below: they are directly related to the safety and security of persons. Carefully follow this advice and inform others about it.

General warning



This warning sign appears in the manual whenever it is necessary to pay attention to rules, instructions or advice. The correct sequence of operations is to be followed in order to avoid damage to the machine.

Symbolizes a serious personnel danger

Information



This warning sign appears in this manual whenever an important information needs to be taken into consideration. Pay attention to this sign and follow the instructions given.

Settings



This sign appears in this manual whenever setting instructions are given. Pay attention to this sign and follow the sequence of instructions given.

Navigation



This icon appears in this manual to give navigation information, to give the path to the subject treated in the chapter.

GETTING STARTED WITH CybTOUCH 12

Depending on software evolutions and the press brake controlled by the CybTouch (configuration/capabilities), the present manual may not fully correspond to the CybTouch that you currently have. However, differences are only minor.



This manual describes all features of CybTouch 12 PS software version CybT12b_Press_PS_V5.4.2 and above.



Touchscreens are pressure sensitive.

Do not press down hard on the screen.

Pressing hard on the screen will damage the display. Such damage is not covered by manufacturer warranty!

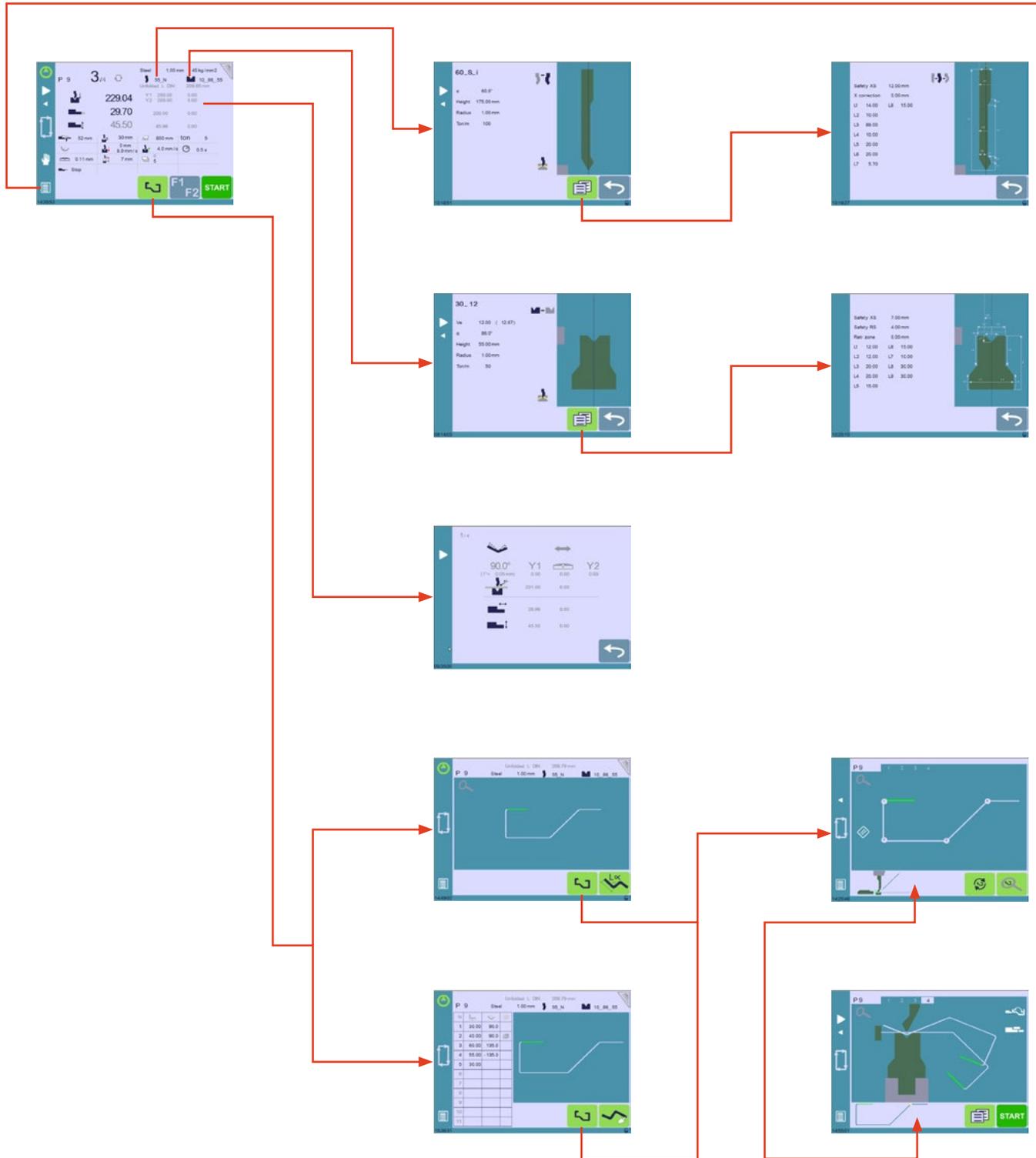


Do not use sharp and/or pointed objects (sheet metal, screwdriver, metal pen ball, etc.) to touch the screen; only use your fingers (with or without gloves on) or a plastic pen. Make sure that your gloves do not have metal particles encrusted in the finger tips as they may also damage the screen.

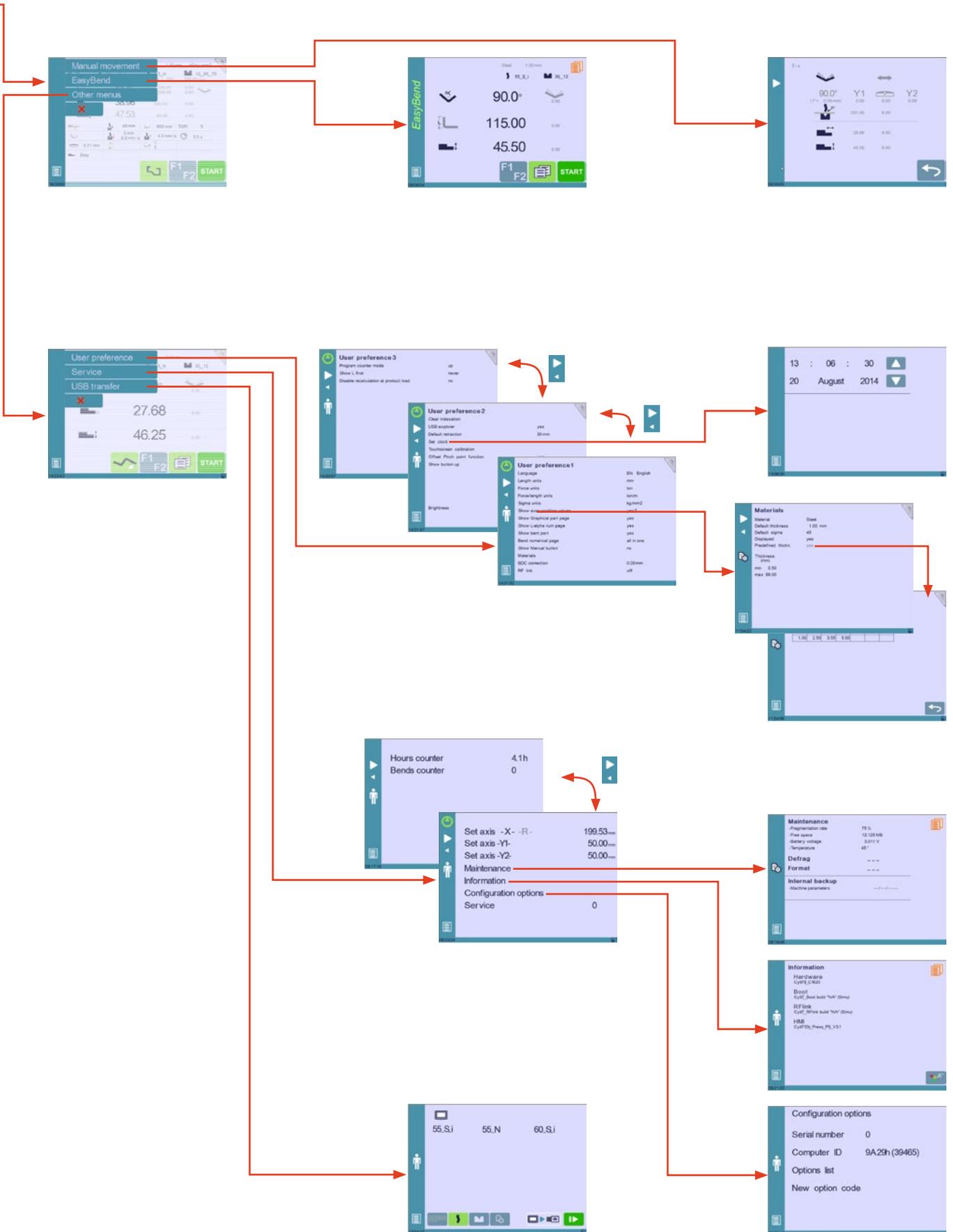
Take a few minutes to practice pressing gently on the screen, you will find that the screen is very reactive and it is pleasant to use.



SCREEN MAP



SCREEN MAP (CONTINUED)



GENERAL NAVIGATION



Menu Button

The Menu button  allows you to directly select (jump to) the desired screen. The content of the menu changes contextually.



Status Pages Zone

Briefly touching the Status pages zone gives access to the [Status page](#) (see page 7). This is really a zone that is active at any moment from any page.



Touching this zone for more than 2 seconds will activate the [Semi-Automatic mode](#) (see page 38).

Screen Cleaning

To clean the screen while the CybTouch is on, touch the  button. Use only a damp and smooth cloth with soap or a neutral detergent.



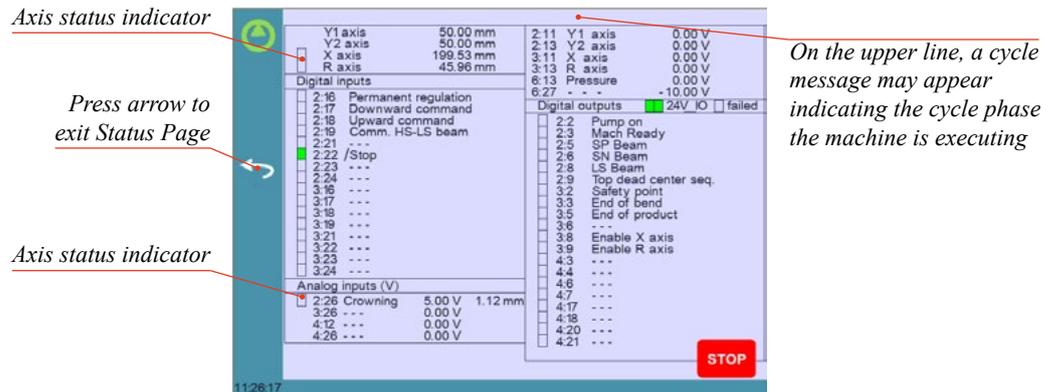
NEVER use solvent, petrol, benzene, alcohols, etc.

STATUS PAGE

The Status page shows the status of all inputs and outputs and axes positions of the NC. This feature is very useful during setup or during phone service with a machine installed in the field.

This page is accessed from anywhere by pressing the [Status Pages Zone](#) (see page 6).

To exit the Status page, press on the arrow on the left.



Axes have specific indicators. Their color indicates the position of the axis relative to the tolerance.

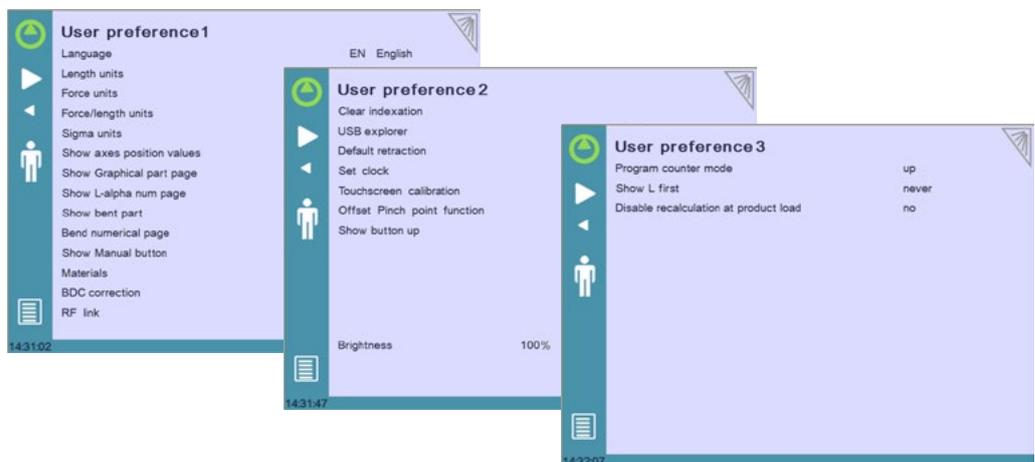
Possible states are :

- : Device not active and in the tolerance.
- : Device moving.
- : Device not active and outside of the tolerance.

USER PREFERENCES



(Menu Button) → Other menus → User preference



To exit the User Preference page, touch the button.

Language

To browse through the available languages, simply touch **Language** on the screen.

Available languages are:

- **EN** English.
- **BR** Brazil.
- **CN** 中文.
- **CZ** Český.
- **DE** Deutsch.
- **ES** Español.
- **FI** Suomi.
- **FR** Français.
- **IT** Italiano.
- **NL** Nederlands.
- **PL** Polski.
- **PT** Português.
- **RU** Русский.
- **TR** Türkçe.
- **TW** 台灣.



The list of available languages is subject to change and may increase over time.

Length Units

This parameter allows choosing between **mm**, **inch** and **none** for the length unit to be used in the CybTouch.



When **none** is selected, the units used are millimeters.

Force Units

This parameter allows choosing between **ton**, **kN** and **tons** for the force unit to be used in the CybTouch.

Force/Length Units

This parameter allows choosing between **ton/m**, **kN/m** and **tons/ft** for the force/length unit to be used in the CybTouch.

Sigma Units

This parameter allows choosing between **kg/mm²**, **N/mm²** and **psi(*1000)** for the sigma unit to be used in the CybTouch.

**Show axes
position values**

This function will display the axes positions on the [Bend Numerical Page](#) (see page 20).

- When set to **no**, the position of axes Y1, X and R is displayed during the beam's movements.
- When set to **yes1**, the positions of the axes (Y1, Y2, X and R) are displayed during their respective movements.
- When set to **yes2**, the positions of the axes (Y1, Y2, X and R) are constantly displayed under their respective set-point values.

**Show Graphical
part page**

When set to **yes**, this button  is displayed on the [Bend Numerical Page](#), and gives access to the [TouchProfile Mode](#) (see page 33).

**Show page
L-alpha num**

When set to **yes**, this parameter gives access to the [L-Alpha Mode](#) (see page 37). This button  is then displayed on the [TouchProfile Mode](#) page.

Show bent part

This parameter, when activated, lets the operator see the state of the part before and after the bend in the [Bend Sequencing](#) page.

Bend numerical page

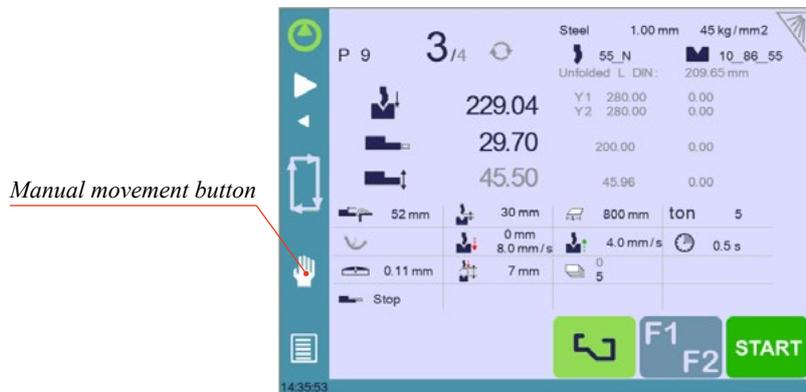
This parameter allows choosing the display on the [Bend Numerical Page](#) (see page 20). When set to **all in one**, all the parameters related to the part or the current bend (see [Available functions on the Bend Num page, page 20](#)) are displayed on the [Bend Numerical Page](#).

When set to **“More page”**, this button  appears on the [Bend Numerical Page](#), and gives access to the special page shown below.



Show manual button

When set to yes, this parameter displays a manual button on the left side of the screen, giving direct access to the [Manual Axes Movement](#) page.

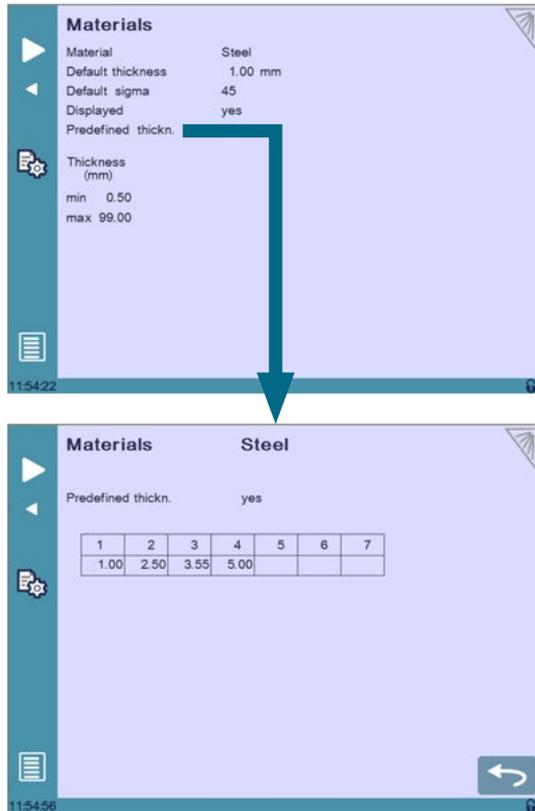


Materials

Touching **Materials** opens the Materials page, where the default characteristics for each material can be changed, or a new material can be configured.



This page may not be available, depending on the machine parameters' configuration. To be allowed to access the Materials page, a level 2 password is required.



The Materials page displays:

- **Material:** Selected material (here **Steel**).
- **Default thickness** for the material.
- **Default sigma:** Default sigma for the material (here **45**).
- **Displayed:** If the material will be available to be selected for use (here **yes**).
- **Predefined thickn.:** Allows defining up to 7 different predefined thicknesses for the selected material.
- **Thickness min/max:** Determines the maximum and minimum accepted thickness for the selected material.

Three predefined default materials are available (steel, stainless steel, aluminum), but others can be added.

To add a material:



1. Touch **Material** and select a non-configured material (Mater X) from the list.
2. Enter the new material's characteristics.
3. Touch the name (Mater X) to display the keyboard and enter the name of the new material.

BDC correction

This parameter allows the operator to apply a permanent correction to the Bottom Dead Center position.



Please note that this correction is always applied and is NOT shown in the correction page. If you experience big differences from what you expect to get, maybe check the value of this parameter.

RFLink

When activated, this function allows communication between the CybTouch and a laptop computer, onto which Cybelec's RFLink dongle is plugged in. This function's default status is **off**, and it is automatically reset to **off** every time the NC is turned on.

Clear indexation

When activated, this function clears the index and the machine will search for them, as it does when turning the power on, allowing the operator to re-index its machine without turning it off.

USB Explorer

When this parameter is set to **yes**, it is possible to browse the USB key from the USB transfer screen (see [USB Transfer](#), page 19).

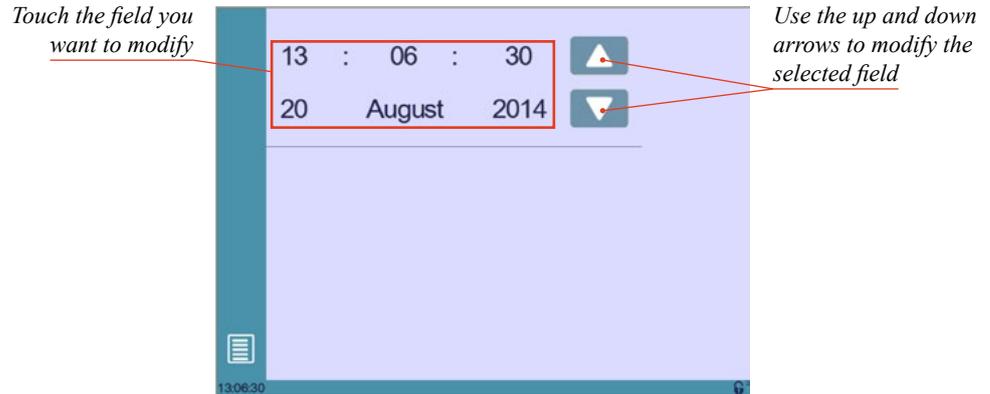
Default retraction

This parameter allows defining the default value displayed in the [Extra Bending Parameters](#) (see page 22) when activating the back gauge retraction function.



In [TouchProfile Mode](#) (see page 33), when a retraction is necessary, the numerical control will automatically calculate and activate the function. The calculated value can naturally be manually modified.

Set Clock Allows the user to set the time and date on the CybTouch.



Touchscreen Calibration

As a tall operator will tend to touch higher than a smaller one on the screen, this function allows the calibration of the touch screen, and also makes sure that it is operating correctly.



SETTING INSTRUCTIONS:

Simply follow the instructions on the page to calibrate the touchscreen.



Use your finger or the plastic tip of a pen to calibrate the Touchscreen. Never use sharp objects as this will damage the screen.

Offset Pinch Point function

When this parameter is set to **yes**, the [Offset Pinch Point](#) (see page 26) function is available in the [Extra Bending Parameters](#).

**Brightness xx%
Eco xx%**

Here the brightness of the screen for normal mode and Eco mode can be defined:

1. Touch the mode for which you want to modify the brightness.
2. Use the   buttons to set the brightness.

Program counter mode

This parameter allows defining the counting mode of the part counter. When set to **up**, the counter will count up to the desired number. When set to **down**, the counter will count from the desired number down to 0.



The counter stops the cycle when total is reached.

The value of the counter is automatically memorized when the numerical control is shut down.

Show L first

This function allows choosing the data entry mode for the back gauge on the [Bend Numerical Page](#) or [EasyBend Page](#). When set to **yes**, the length of the flange icon  is displayed by default. When set to **no**, the icon for the position in mm of the back gauge  is displayed.

When set to **never**, the position of the back gauge is no longer linked to the bending angle, i.e. the position is never recalculated when the bending angle is modified.



The default value for this preference is **never**. Note that completely formatting the machine will reset this preference to its default value.

The  mode is not available for graphical parts (see also [Set-point back gauge \(X axis\) / flange length, page 21](#)).

Disable recalculation at product load

This parameter controls the recalculation of the axes position values when a part is loaded. It is useful when there is a possibility that something has changed, like dimensions of tools or back gauge, since the last time the program was run.

The default value for this parameter is **no**.



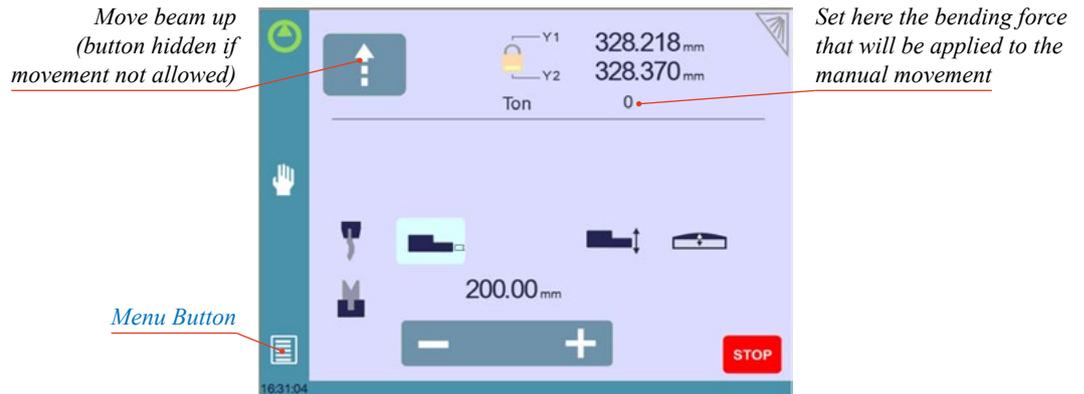
Corrections remain the same. We emphasize again the good practice of using corrections (see [Bending and Corrections, page 38](#)) to fine tune a program. In this case, even if modifications have happened, the part will be recalculated and properly bent.

MANUAL AXES MOVEMENT

In the course of setting up a machine, it is sometimes necessary to be able to move the axes manually, for example when changing the tooling. This can be done on this page.



(Menu Button) → Manual movement



SETTING INSTRUCTIONS:

1. Select the axis that you want to move:
 - for the back gauge X axis.
 - for the back gauge R axis.
 - for the crowning axis.
 - or for the tool clamping devices (if available).
2. Touch the buttons to move the selected axis.



The button needs to be kept pressed for two seconds to change the status of the clamping devices.

3. Use the foot switch (Low Speed Down movement) and this button (High Speed Up) to move the beam.

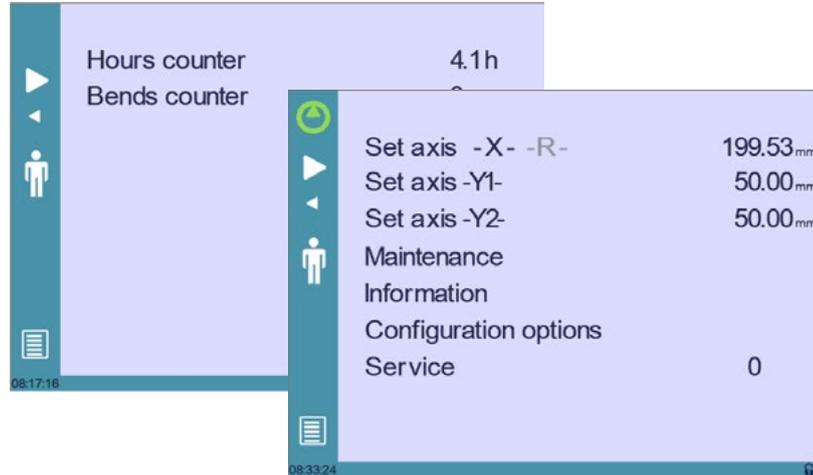
Desynchronized Beam

When the padlock is open , it is possible to select and move (Low Speed Down movement) Y1 or Y2 only. This is an easy way to return an unsynchronized beam back to parallel to the table.

SERVICE PAGE



(Menu Button) → Other menus → Service → Service



Set Axis

Allows the operator to manually adjust the position of the back gauge (axes X and R) and the beam (axes Y1 and Y2).



A level 2 password is required to use this function. It must be used with utter care and only by experienced personnel. Wrong settings may mechanically damage the machine.

Settings are lost after indexing the machine.

Maintenance

The Maintenance page displays the hardware status of the CybTouch and lets the operator perform different maintenance actions.



(Menu Button) → Other menus → Service → Service → Maintenance



All the following actions require codes and should only be performed by technicians or upon request of a technician.

Defrag

This function will rearrange the memory space of the CybTouch. Simply touch it and follow the instructions given in the yellow pop-up window.

Format

This function will erase all data in the CybTouch. Only use this with the help of a technician.



Before using this function, make sure that all your files have been transferred outside the CybTouch (see USB Transfer, page 19).

Internal backup

This function is specially designed for OEM and support.

Usually a machine parameters' backup is made by the machine manufacturer or the company who services the machine. This backup allows a maintenance technician to restore original working parameters if necessary.

Should there be a need to restore parameters, call on a maintenance technician and follow his instructions.

Do not try to use this function unless you are in dire need.

Information

The Information page displays the names and versions of the softwares installed on the CybTouch. Pressing the Advanced button shows more detailed information.



(Menu Button) → Other menu → Service → Service → Information



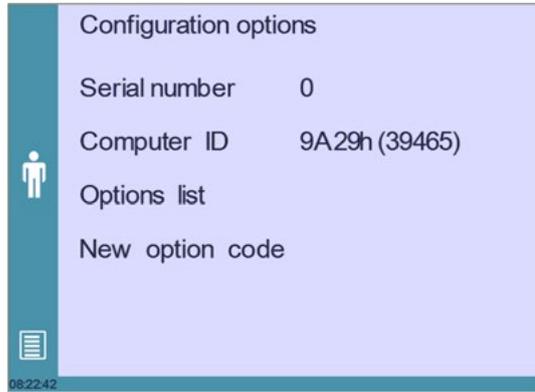
To Advanced information

Configuration options

Touching this menu opens the following page, where one can find the computer's identification and manage the machine's options.



(Menu Button) → Other menus → Service → Service → Configuration options



Serial number

This is the serial number of the CybTouch. It is entered at the factory at the end of the machine's initial setup and is related to the machine's option list.



Changing the serial number means that all the options installed on the machine can be lost.

Computer ID

This line displays an identification code that is unique to each CybTouch and guarantees, together with the serial number, a correct identification of the machine.

Option list

This function opens a yellow pop-up window where all the options installed on the CybTouch are displayed.

New option code

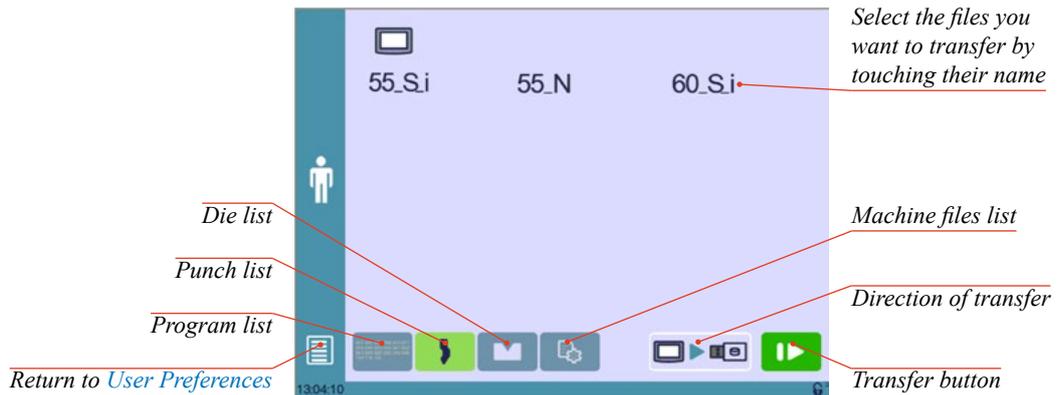
The function opens an alphanumerical pad where the code of the new option must be entered. The format of an option code is: ABC-DEF-GHI-JKLM

USB TRANSFER



(Menu Button) → Other menus → USB transfer

This function opens the USB transfer screen, which allows the operator to transfer data between the CybTouch and a USB key.



Copying parameters from USB to NC requests a level 3 password or, for end-users, the restore password.

Precautions

For a smooth transfer, please take the following precautions into consideration:

- The USB port of the CybTouch is meant to be used only with a storage media of the "Memory stick" type.
- The port is a standard USB 2.0 port, USB 1.1 compatible.
- USB 3.0 key that are USB 2.0 compatible, should normally work. There's however no guarantee, as it depends of the firmware of these new keys.
- CD and external HDD are not accepted.
- Although unlikely, it is possible to find a USB key that is not compatible with the CybTouch. If so, try with another one.



3 rules for a fast and easy transfer:

- ➔ Use a USB key dedicated to files transfer with the CybTouch.
- ➔ Less files means a faster transfer.
- ➔ The file's path on the key is limited to 255 characters.

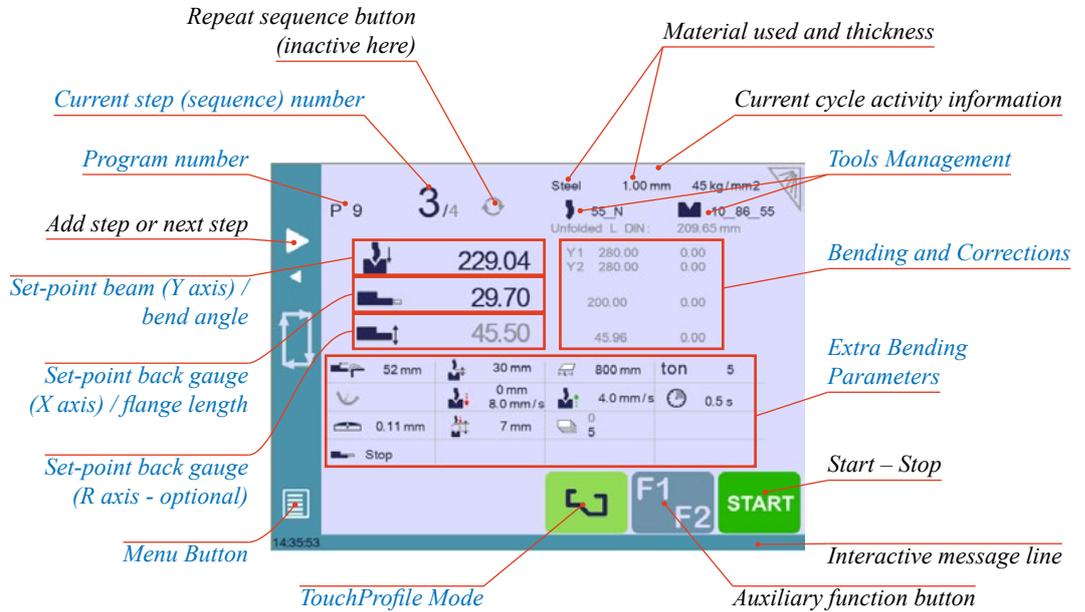
Importing parts from other Cybelec NCs

CybTouch 12 is delivered with Cybelec's off-line software PC-ModEva, which controls most of the other numerical controls produced by Cybelec.

From version T on, PC-ModEva includes [CybTouchConverter](#) (see page 42), which allows you to convert parts to the CybTouch format, and import them into your machine with the above mentioned function.

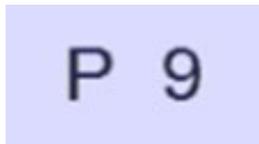
BASIC PAGE DESCRIPTION

BEND NUMERICAL PAGE



Available functions on the Bend Num page

The Bend Numerical Page is normally the working page, from which the bends are executed, and most of the navigation originates from and leads to.



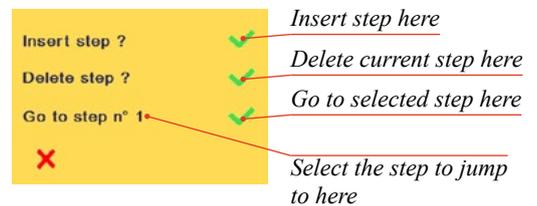
Program number

Touching the program number allows creating a new one, saving it and more. See [Managing Programs, page 44](#) for more information.



Current step (sequence) number

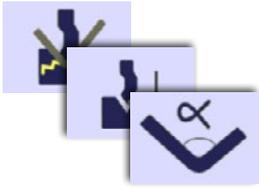
Touching the step number will open a yellow pop-up window as shown here, with 3 different actions to choose from:



- **Insert step**: this function will insert a step after the current one.
- **Delete step**: this function will erase the current step.
- **Go to step**: this function allows jumping directly to the desired step.



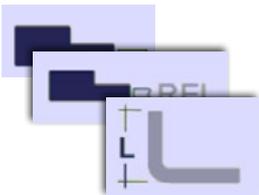
The **Insert step** and **Delete step** functions cannot be used on parts created with the **TouchProfile Mode** (see [page 33](#)).



Set-point beam (Y axis) / bend angle

The CybTouch has three different modes to manage the Y axis. Touching one of the icons on the left allows going from one mode to another. The three modes are:

- : in this mode, the operator simply enters the desired angle and the CybTouch will calculate the position at which the beam will stop (BDC).
- : in this mode, the operator can see the calculated target of the angle. It is possible to change the value.
- : this is the bottoming mode. When the beam stops, or when the target value is reached, the bending cycle continues (dwell time, return).
When activated, the calculated bottoming value is displayed in gray. It is however possible to modify this value, which will then be displayed in black. It can only be used if both the tools accept bottoming (see [Punches, page 27](#) and [Dies, page 29](#)).



Set-point back gauge (X axis) / flange length

The CybTouch has two different modes to manage the back gauge X axis. Touching one of the icons on the left allows going from one mode to another. The two modes are:

- : in this mode, the operator simply enters the desired length for the flange and the CybTouch will calculate the position for the back gauge X axis. This mode is only available when programming a part with the [Numerical Mode \(see page 36\)](#).
- : this mode shows the targeted position for the back gauge X axis.
- : this is the relative mode. The back gauge makes a relative movement of the value entered. When the value entered is negative, the back gauge moves in the operator's direction.



The relative mode is activated/deactivated by pressing the back gauge icon for 2 seconds.

Relative movement is not possible on the first sequence (neither on Easy Bend) or on graphical parts.



User Preference [Show L first \(see page 14\)](#) allows choosing which mode is displayed by default first.

Set-point back gauge (R axis - optional)

The value of the vertical position of the back gauge (R axis) is calculated by the CybTouch. It is displayed in gray. It can however be manually modified, but is then displayed in black.



It is possible to restore the calculated value simply by erasing the manually entered value in the numerical pad . The message "Default/Recalculate" appears at the top of the screen. Just press validate  on the numerical pad.

Extra Bending Parameters

The lower part of the [Bend Numerical Page](#) displays parameters related to the part, and depending on the CybTouch configuration and the type of action performed, it also displays various settings for the current bend.

Steel

Material

This is not a sequence parameter, but of course a part parameter. Each touch on the material's name selects the next available from the list of [Materials](#) (see page 11).

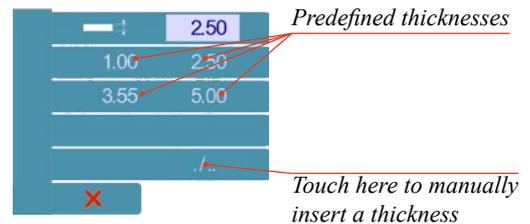
 1.00 mm

Material thickness

The default thickness, defined in [Materials](#) (see page 11), is automatically displayed when changing material. It is however possible to change it simply by touching this icon.

If on the other hand, the parameter **Predefined thicken.** (see page 11) is set to **yes**, a touch on this icon will open a numerical pad as show to the right, where the operator will be able to select directly one of the predefined thicknesses.

This is a part parameter.





Material sigma

The default sigma, defined in [Materials \(see page 11\)](#), is automatically displayed when changing material. It is however possible to change it simply by touching this icon. This is also of course a part parameter.



Back gauge retraction

The back gauge retraction can be activated/deactivated at its [Default retraction \(see page 12\)](#) value using this icon. It is possible to modify the value by touching it. This is a sequence parameter, meaning it can be modified with each step of the program.



Speed change threshold and bending speed

The distance parameter allows the operator to increase the height of the speed change point. The speed parameter allows decreasing the bending speed from its maximum value defined in the machine parameters.



Bending length

This parameter defines the width of the sheet metal part that will be pinched between the tools. It is used to calculate the bending force.

If this parameter is not activated (gray), the CybTouch will not calculate the bending [Force](#) and the [Crowning](#).



Step bending

When a large radius bend has been programmed (see [L-Alpha Mode, page 37](#)), its parameters are displayed here. It is possible to modify them directly here.

Programming 99 x will automatically calculate the maximum possible step bends. The resulting value may be reduced. However, if it is increased over the maximum calculated value, the resulting radius and angle will be drastically affected.

The large radius bending function is deactivated when this field is grayed.



Slow speed return

This parameter allows slowing down the speed of the beam after the bend, and is generally used when the part has a long flange and the operator tries not to let it “fall down too fast”.

The beam will return at low speed up while the operator holds the foot pedal. It switches to high speed up when either the pedal is released or Pinch Point is reached, whichever comes first.

When this field is deactivated (grayed), the beam moves back up directly at HSU (High Speed Up).



Dwell time

Allows defining the duration of the dwell time, meaning the time during which the punch remains at BDC before coming back up.

When the icon is crossed , it means that there is no automatic return of the beam after TDC. This is a sequence parameter, meaning it can be modified with each step of the program.

When this field is deactivated (grayed), the default Dwell Time value defined in the machine parameters is applied.



Force

The force is automatically calculated by the CybTouch, according to the [Material](#), the [Material thickness](#), the [Material sigma](#) and the [Bending length](#). The value can also be manually modified here.



Opening (TDC)

For parts created with a profile (see [TouchProfile Mode](#) or [L-Alpha Mode](#)), this value is automatically calculated to allow the operator enough room to extract its bent part from between the tools. It is however possible to change it manually.

For part created manually (see [Numerical Mode](#)), this parameter uses the default minimum value defined in the machine parameters. It can however be programmed here. Creating a “New part” reinitializes it to its default value.

When this field is deactivated (grayed), the beam moves back all the way up to its maximum limit switch.



Crowning

The crowning function is activated here. It is automatically calculated, according to the [Material](#), the [Material thickness](#), the [Material sigma](#) and the [Bending length](#). If the crowning needs correction, use the [Crowning \(see page 39\)](#) function in the correction page.

The value can be manually changed by operator. It will however be automatically recalculated if any of the values used for its calculation is changed.



When the crowning function is deactivated (gray icon), the crowning system physically remains to its last position and does not automatically return to 0.0 mm. Keep that in mind when using this function – or not – between one sequence and the following.



Back gauge finger type

The back gauge dimensions are defined in the machine parameters. This function allows browsing through the available support and stop positions of the back gauge. This function may be available, or not, according to the machine configuration.



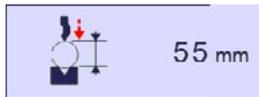
Number of parts

The operator can enter here the total amount of parts to be produced. Every time all the sequences of the program are executed, hence a part is completed, this counter is updated of one unit (increased or decreased, see [Program counter mode, page 14](#)). When the amount of parts is reached, a yellow pop-up window signals it to the operator.



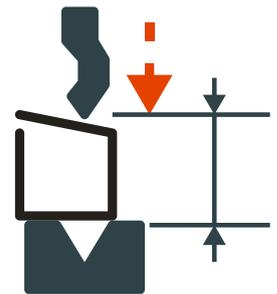
Back gauge manual control

Activating this parameter gives manual control over the back gauge movement. This means the operator must personally give the start to the back gauge movement, using for example the foot switch or the start button.



Offset Pinch Point

This function allows making a bending cycle further away from the matrix, for example to close the opened side of a box, as shown on the right.



Once activated, this function forces the CybTouch into bottoming mode (see [Set-point beam \(Y axis\) / bend angle, page 21](#)).

With this function, the Speed Change Point occurs higher, and the operator can inch the beam down with short impulses on the pedal, until the proper height is reached. The parameters [Speed change threshold and bending speed \(see page 23\)](#) can also be useful in this situation.

Proceeding this way is normally used for unit or small series. If however the number of parts is large, the operator can enter a target value in the bottoming field, which will then turn black. This way, the CybTouch will execute a normal bending cycle, stopping the beam at the programmed value, i.e. saving a lot of production time.



Hemming

Allows making conventional die hemming bends.

COPY TO ALL FUNCTION

This function allows copying a defined value to all the steps of the current program. It appears in the numerical pad of relevant fields, such as [Bending length](#), [Force](#), [Bending and Corrections](#), etc.



Copy to all button

It is also possible to use the Copy to All function with fields that do not require a numerical pad, such as [Back gauge manual control \(see page 25\)](#), by simply holding the button for 2 seconds. The Copy to All button is then displayed on top of the screen as shown here.



TOOLS MANAGEMENT

Tools management allows the creation and configuration on the CybTouch of the tools to be used on the machine. These tools are then taken into account in bend calculations.

Once a punch and die are created and selected, you can select the bend angle you require as well as the flange length (L). The CybTouch then calculates the positions for axes X and Y for the bend.

PUNCHES



Bend Numerical Page →

Punch mounting direction

Punch name → 60_S_i

Browsing through existing punches → [Left Arrow]

Basic punch data

α	60.0°
Height	175.00 mm
Radius	1.00 mm
Ton/m	100

Bottoming → [Bottoming Icon]

Return to Bend Numerical Page → [Back Arrow]

Types of punches → [Punch Type Icon]

Punch more specific data

Safety XS	12.00 mm		
X correction	0.00 mm		
L1	14.00	LB	15.00
L2	10.00		
L3	89.00		
L4	10.00		
L5	20.00		
L6	20.00		
L7	5.70		

Touch the punch to display its default shape, allowing an easier identification of the various dimensions → [Punch Shape Icon]



SETTING INSTRUCTIONS:

To select a punch, simply browse through the existing punches in your library using the arrows buttons, and then return to [Bend Numerical Page](#).

How to create or modify a punch?

If no punch is yet created, the punch will have no name (???) is displayed). If a punch already exists, then the last punch used will be selected, here **60_S_i** (modifications will not alter the existing punch as they will be saved under another name).



1. Touch the punch icon to access the punch details.
2. Enter the basic characteristics (α (punch angle), **Height**, **Radius** and **Ton/m**) for the new punch to be created.
3. Touch the button to invert the punch if necessary.
4. Select the or icon to define the tool as resistant for bottoming.
5. Touch the button to display the More page.
6. Select the punch type (straight, normal or gooseneck) with this icon .
7. Enter the more specific dimensions L1 to Lx by referring to the graphic representation on the right of the screen. Touching this image will display the default representation of the tool, making it easier to identify the various dimensions.



The dimensions L1 to Lx are the same as the ones used in PC 1200, DNC 880S, ModEva's. If you use the same tools, print them from PC1200 to easily program them in the CybTouch. Of course consider giving the same tools the same name.

8. Enter the following values:
 - **Safety XS:** Security distance between the tool and the back gauge for X axis.
 - **X correction:** If the punch is not perfectly aligned.
9. Return to previous page .



To be allowed to save a tool, a level 2 password is required.

10. Touch the punch name (here **60_S_i**).
11. Touch **Save punch** to overwrite the existing tool or **Save punch as** if you want to save your tool under another name.
12. Enter the name of the new punch using the alphanumerical keypad.



We recommend that you follow the naming conventions explained in **Naming Tools** (see page 31).

13. Touching the button brings you back to the program page, with the punch you just saved being selected and ready to be used.

DIES



Bend Numerical Page →

Die mounting direction

Die name → 30_12

Browsing through existing dies → [Left arrow icon]

Basic die data

Ve	12.00	(12.87)
α	86.0°	
Height	55.00 mm	
Radius	1.00 mm	
Ton/m	50	

Bottoming → [Bottoming icon]

Return to Bend Numerical Page → [Back arrow icon]

Die more specific data

Safety XS	7.00 mm		
Safety RS	4.00 mm		
Retr zone	0.00 mm		
L1	12.00	L6	15.00
L2	12.00	L7	10.00
L3	20.00	L8	30.00
L4	20.00	L9	30.00
L5	15.00		

Touch the die to display its default shape, allowing an easier identification of the various dimensions



SETTING INSTRUCTIONS:

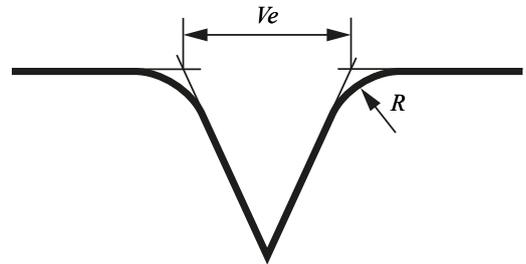
Selecting a die is the same as selecting a punch; simply browse through the existing dies in your library using the arrows buttons, and then return to [Bend Numerical Page](#).

How to create or modify a die?

If no die is yet created, the die will have no name (???) is displayed). If a die already exists, then the last one used will be selected, here 30_12 (modifications will not alter the existing die as they will be saved under another name).



1. Touch the die icon  to access the die details.
2. Enter the basic characteristics (V_e (die width), α , Height, Radius and Ton/m) for the new die to be created.
3. Touch the  button to invert the die if necessary.
4. Select the  or  icon to define the tool as resistant for bottoming.
5. Touch the  button to display the More page.
6. Enter the more specific dimensions L1 to Lx by referring to the graphic representation on the right of the screen. Touching this image will display the default representation of the tool, making it easier to identify the various dimensions.



The dimensions L1 to Lx are the same as the ones used in PC 1200, DNC 880S, ModEva's. If you use the same tools, print them from PC1200 to easily program them in the CybTouch. Of course consider giving the same tools the same name.

7. Enter the following values:
 - **Safety XS:** Security distance between the tool and the back gauge for X axis.
 - **Safety RS:** Security distance between the tool and the back gauge for R axis.
 - **Refr. zone:** Retraction zone.
8. Return to previous page .



To be allowed to save a tool, a level 2 password is required.

9. Touch the die name (here 30_12).
10. Touch **Save die** to overwrite the existing tool or **Save die as** if you want to save your tool under another name.
11. Enter the name of the new die using the alphanumerical keypad.



We recommend that you follow the naming conventions explained in [Naming Tools \(see page 31\)](#).

12. Touching the  button brings you back to the program page, with the die you just saved being selected and ready to be used.

NAMING TOOLS

It is recommended that you use naming conventions for your tools.

Below you will find a simple convention allowing you to precisely identify a punch or die through its name.

Of course, depending on your needs you may need to create more rules for punch and die naming.

Punches

The name of the punch should be built in the following manner: first its angle, followed by its type, and then whether it is inverted or not.

PUNCH ANGLE (°)		PUNCH TYPE		INVERTED OR NOT
30		N = Normal		i = If inverted
60	-	S = Straight	-	
90		G = Gooseneck		

Following these rules, here are some examples of punch names: **90_N_i**, **60_G**, **30_S**, and so on, and so forth.

Dies

The name of the die should be built in pretty much the same manner: first its width (Ve dimension), followed by its angle, and then whether it is inverted or not.

Ve (mm)		DIE ANGLE (°)		INVERTED OR NOT
12		30		i = If inverted
16	-	86	-	
20				

Following these rules, here are some examples of die names: **12_86_i**, **16_86**, **20_30**, and so on, and so forth.

BACK GAUGES

If the operator needs to intervene on the machine's back gauges, he can use the back gauge cycle function, which will bring them all the way forward. He will then be able to work on them without having to put his hands and arms between the tools.



This function may not be available, depending on the machine's configuration. The function's icon is only visible when the machine is indexed.



Back gauge cycle button



SETTING INSTRUCTIONS:

1. Make sure the machine is indexed. This button  is not available when it is not the case.
2. Press the back gauge cycle button  for more than 2 seconds to initiate the cycle.
3. The back gauges position themselves all the way forward, between the tools. If the beam is not at TDC, it will move first all the way up before the back gauges begin to move.
4. Once the intervention is done, touch the cycle button  again to deactivate the function, and press the  button.

CREATING A PART PROGRAM

There are three ways to create a program part: with the [TouchProfile Mode](#), with the [Numerical Mode](#) (see page 36), and with the [L-Alpha Mode](#) (see page 37).

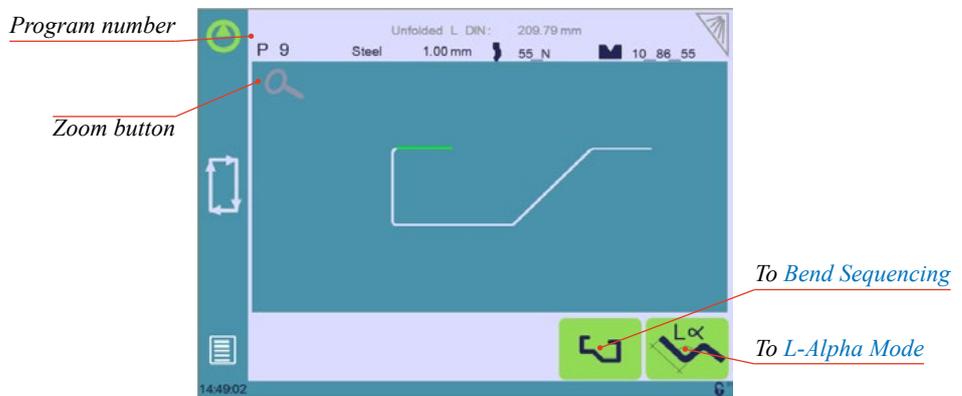


In this chapter the machine is considered operational: machine parameters, tools (see [Tools Management, page 27](#)), materials (see [Materials, page 11](#)), etc. are already configured and programmed.

TOUCHPROFILE MODE



Bend Numerical Page →

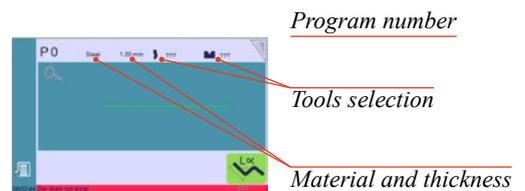


In this mode, the operator can very intuitively draw a profile directly on the screen.

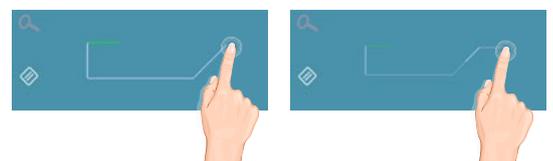


SETTING INSTRUCTIONS:

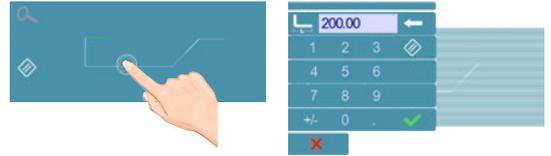
1. Touch the program number, select **New program** in the list, and touch **Graphical**.
2. First select the material, the [Material thickness](#) (see page 22) and the tools to be used for the part by touching their respective icons.
3. Draw the profile by touching the screen where you want to add a segment.



Touching this icon allows erasing the last segment added.



4. Touch on the middle of a segment to modify its value.



5. Touch on an angle to modify its value. Touching the  icon will open a yellow pop-up, in which it possible to:



- Activate (and deactivate) a special tool by touching its icon, and select it from a list by pressing **???**.
- Change the value of the radius (see also [Step bending, page 23](#)) and define how many steps the NC will make to execute it.



Special tools

Maximum distance to ideal curve

Number of steps to execute the step bending

Large radius value

6. Should it prove difficult to select a particular segment or angle, it is possible to zoom on an area by touching this button  and then on the desired zone.



Activate/deactivate zoom

Toggle zoom in/zoom out



One can also choose to enter the length and angle values in the **L-Alpha Mode** page by pressing this  button.

7. Once all the segments and angles are adjusted, press this button  to access **Bend Sequencing**.

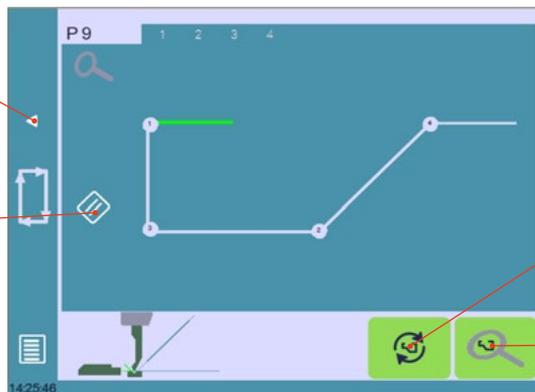
BEND SEQUENCING



Bend Numerical Page →  → 

Delete last bend

Delete all bends



Launch automatic bend sequence calculation (optional)

Automatic bend sequence calculation criteria (optional)

Once the shape of the part has been defined, the bending sequence for the part can be determined.



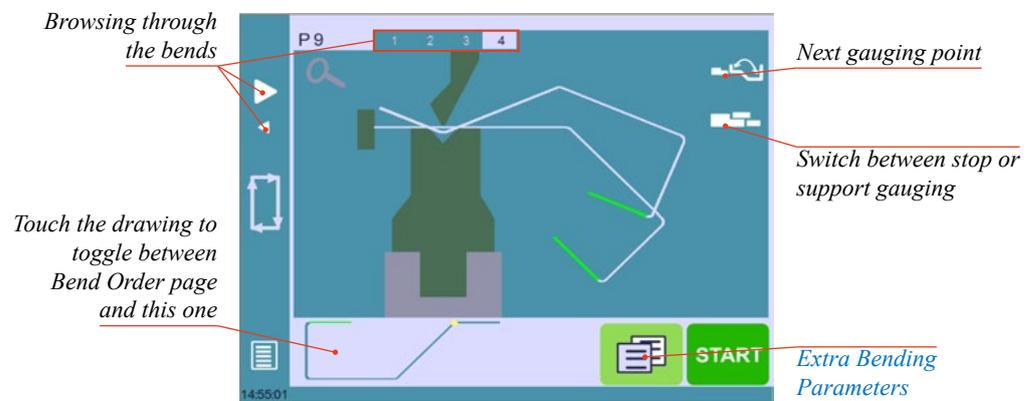
SETTING INSTRUCTIONS:

1. If available, touch the bend sequence calculation button . Once the sequence is calculated, the Bend 2D screen is displayed.
2. If the option is not installed, simply touch the bend in the order that you want them performed. A small number is displayed on the bend, indicating its position in the sequence.



Pressing the back button will erase the last bend in the sequence. If you wish to delete all bends, simply touch the button.

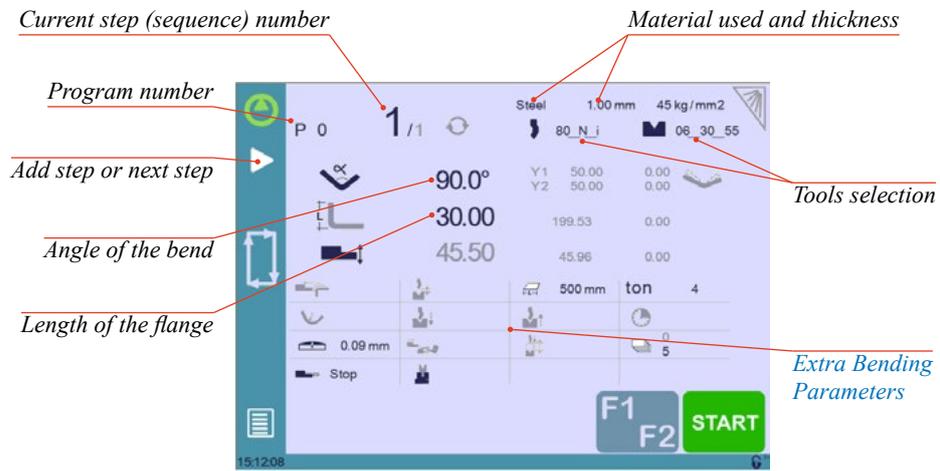
3. Immediately after determining the last bend to be selected, the Bend 2D screen here below is displayed.



4. In the Bend 2D page, the operator must determine the back gauge and part position for each bend, and if necessary [Extra Bending Parameters](#) (see page 22).
5. Press and select **Bend num page** to return to the [Bend Numerical Page](#).
6. Start the hydraulic pump motor (by pressing this button if available. It turns red when the motor is running).
7. Press the button to position the machine according to the data that were just entered.
8. When the machine is ready to bend, a button is displayed.
9. If you want to repeat the same step in order to apply all the necessary corrections to it, switch to the [Semi-Automatic mode](#) (see page 38).
10. Press the foot switch to execute the bend.

NUMERICAL MODE

It is also possible to simply create a part program directly on the [Bend Numerical Page](#).



SETTING INSTRUCTIONS:

1. Touch the program number, select **New program** in the list, and touch **Numerical**.
2. First select the tools to be used for the part by touching their respective icons (see [Tools Management, page 27](#)).
3. Touch the **Material's** name (here **Steel**) until the one used is displayed.
4. Touch the **Material thickness**, and select or enter the proper one.
5. In the **Extra Bending Parameters'** section, enter the **Bending length**, and other sequence parameters (**Opening (TDC)**, **Slow speed return**, etc.).
6. Touch the numerical value next to the angle icon , and enter the value for the first bend you wish to create (here **90.0°**).
7. Touch the numerical value next to the segment length icon , and enter the value for the first segment you wish to create (here **30.00 mm**).
8. Add the next bend to the program by touching .
9. Proceed in the same manner to create the other segments of the part.
10. Touch  to return to the first bend.
11. Start the hydraulic pump motor (by pressing this button  if available. It turns red when the motor is running).
12. Press the **START** button to position the machine according to the data that were just entered.
13. When the machine is ready to bend, a **OK** button is displayed.
14. If you want to repeat the same step in order to apply all the necessary corrections to it, switch to the [Semi-Automatic mode](#) (see [page 38](#)).
15. Press the foot switch to execute the bend.

L-ALPHA MODE



N	L	α
1	30.00	90.0
2	40.00	90.0
3	60.00	135.0
4	55.00	-135.0
5	30.00	
6		
7		
8		
9		
10		
11		



This page is only available if user preference **Show page L-alpha num** (see page 9) is set to **yes**.

In this mode, the operator can define each step (length or angle) of a profile on a table.



SETTING INSTRUCTIONS:

1. Touch the program number, select **New program** in the list, and touch **Graphical**.
2. Press this button to access the L-alpha mode.
3. First select the material, its thickness and the tools to be used for the part by touching their respective icons.
4. Touch the first value in the table (here **200.00**). The window to the right is displayed.
5. Enter the value for the first segment's length and touch the button to add another step.



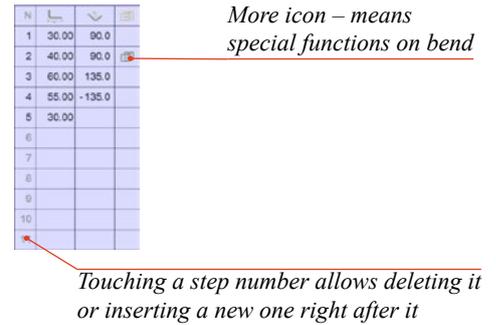
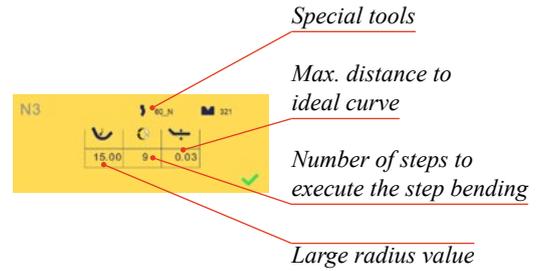
Click on OK when prompted to create new step.

6. Proceed in the same manner for all steps and angles of the profile.
7. If a bend needs special parameters (large radius, special tool), press in the more field of the corresponding sequence.



A special punch or /and die means there must be an additional working station.

8. A yellow pop-up is displayed, in which it possible to:
 - Activate (and deactivate) a special tool by touching its icon, and select it from a list by pressing ???.
 - Change the value of the radius (see also [Step bending, page 23](#)) and define how many steps the NC will make to execute it.
9. A small More icon shows that something special is programmed on the bend.
10. To remove the icon (and special parameters), return to the yellow pop-up, touch all the activated functions to make them turn gray, and select OK.
11. Once all the segments and angles are adjusted, press this button  to access [Bend Sequencing](#) (see [page 34](#)).



BENDING AND CORRECTIONS

All program corrections are made in the [Bend Numerical Page](#) (see [page 20](#)). According to his preferences, the operator can choose to execute all the steps of the program one after the other, making corrections along the way. Or he can choose to apply all the corrections necessary to the same step before moving to the next one using the [Semi-Automatic mode](#).

Corrections can be made to:

- The angle (Y1 + Y2, see [Angle Correction \(Y axis\), page 39](#) and/or crowning if available, see [Crowning, page 39](#)),
- The back gauge position (X and R, if available), see [Back gauge \(X & R axes\) Correction, page 40](#).

Semi-Automatic mode



The semi-automatic mode allows repeating the same sequence indefinitely. It is used when the operator wants to apply corrections to his part one bend after another. He can thus execute the same step until he gets the desired result, before moving to the next one by means of the  button.

The semi-automatic mode is activated (and deactivated) by touching for more than one second on the  button.

Angle Correction (Y axis)

After physically measuring the angle, if corrections are to be made, they must be done on this page, and not directly in the program step.



Bend Numerical Page →



The screenshot shows the Bend Numerical Page interface with the following labels and corresponding icons:

- Angle correction:** Points to the angle input field showing 90.0° (1" = 0.05 mm).
- Y1 axis correction:** Points to the Y1 input field showing 0.00.
- Y2 axis correction:** Points to the Y2 input field showing 0.00.
- Crowning correction:** Points to the crowning icon (a curved line).
- Pinch Point correction:** Points to the pinch point icon (a vertical line with a horizontal bar).
- X axis correction:** Points to the X input field showing 0.00.
- R axis correction:** Points to the R input field showing 0.00.



SETTING INSTRUCTIONS:

1. Touch the angle correction  icon, and enter the physically measured value of the angle. The numerical control will automatically calculate the Y1 and Y2 axes corrections.

The keypad shows the following elements:

- Input field: Enter the measured angle, 90.0
- Buttons: 1, 2, 3, 4, 5, 6, 7, 8, 9, +/-, 0, ., and a green checkmark.
- Reset corrections button: A red 'X' icon button.

Reset corrections button



Pressing this  button will reset all angle corrections.

2. Depending on the position of the sheet metal in the press, it may be necessary to fine-tune the beam's corrections. It is possible by simply touching this icon  or this  one, and entering the desired values.

Crowning

3. Corrections to the crowning value can be made by touching this icon .
4. The numerical control automatically calculates the Pinch Point height. If it needs correction, it can be done by touching this icon .



The grayed value next to the Pinch Point correction icon is the value calculated by the numerical control, and thus the value onto which the correction will be applied.

**Back gauge
(X & R axes)
Correction**

5. Touching this icon  allows applying a correction to the back gauge X axis position.
6. Touching this icon  allows applying a correction to the back gauge R axis position.



The grayed values next to the back gauge correction icons are the value calculated by the numerical control, and thus the values onto which the corrections will be applied.

The back gauge positions corrections can also directly be made on the [Bend Numerical Page](#) (see page 20).



NEXT-PART FUNCTION

This function allows the operator to run two, or several, part-programs one after another. This is very handy when one wants to make a three dimensional part, like a box for example, or make a final product composed of several parts.

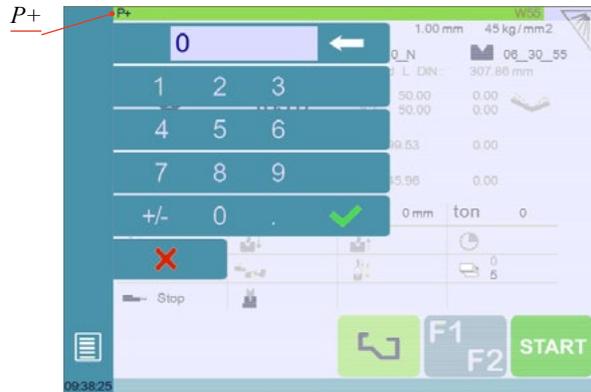
The CybTouch will execute the current program. At the end of the last sequence, instead of returning to the first sequence of the current program, the CybTouch switches to the program selected as **P+nn** (i.e. the next one). It goes on like this, as long as a part is programmed with a next one.





SETTING INSTRUCTIONS:

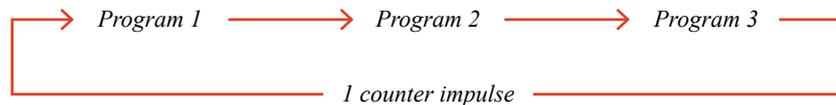
1. To activate the Next-Part function, touch the **Program number** (e.g. **P1**) and keep it pressed until the following numerical pad is displayed.



2. Enter the number of the program that must be executed at the end of the current one.
3. Save the program (see [Saving a Program, page 44](#)).

Cycles and quantities

It is naturally possible to cycle the programs, meaning that the program following the last one is the first one. There are however some specificities to take into consideration when one wants to produce certain amount of cycled programs.



In the diagram above, an assembly needs 3 programs to be completed: program 1 is followed by program 2, which in turn is followed by program 3. To complete the cycle, program 3 is programmed to be followed by program 1. In such a case, the value of the part counter will be updated (see [Number of parts, page 25](#)) of 1 unit when it goes from program 3 to program 1.

The CybTouch is designed like this: every time the program following the current one has a smaller number, the counter value is changed.



When a series of programs corresponds to one single part, make sure that their intrinsic numbers increase chronologically.

CYBTouCHCONVERTER

CybTouchConverter is a tool allowing the conversion of products/parts files from PC-ModEva (formerly PC1200) to CybTouch. Owners of Cybelec's ModEva type machines can thus produce the same parts on a ModEva or a CybTouch.

How to Convert a Part-Program?



1. Check that the version of PC-ModEva installed on your PC is recent enough to contain CybTouchConverter.

MENU → Welcome

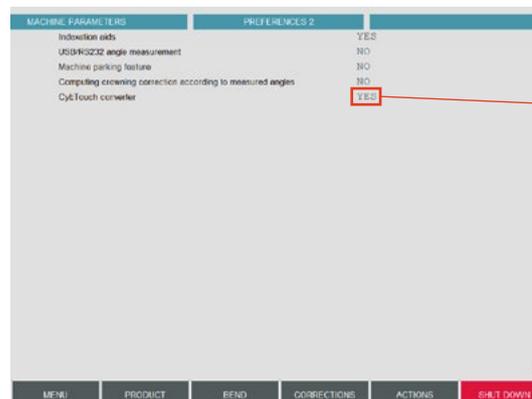
The software version must end with a 'T'



2. Activate the **CybTouch Converter** function in the machine parameter.



MENU → Machine parameters → MENU → Preferences 2



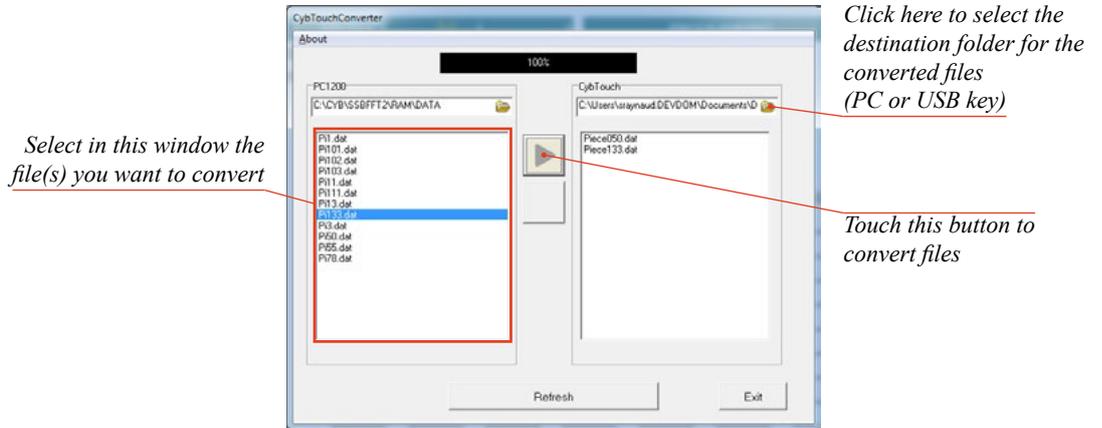
*Parameter **CybTouch Converter** must be set to YES*

3. In the List of Products' page, start the CybTouch Converter.



MENU → Cancel → List of products → ACTIONS → CybTouch Converter

- 4. Select the destination folder for the converted files.
- 5. Select the part(s) that you want to convert and touch this button.



- 6. The program-parts are now converted and located on your PC or USB key. They can be directly transferred on the CybTouch, using the [USB Transfer](#) (see page 19).



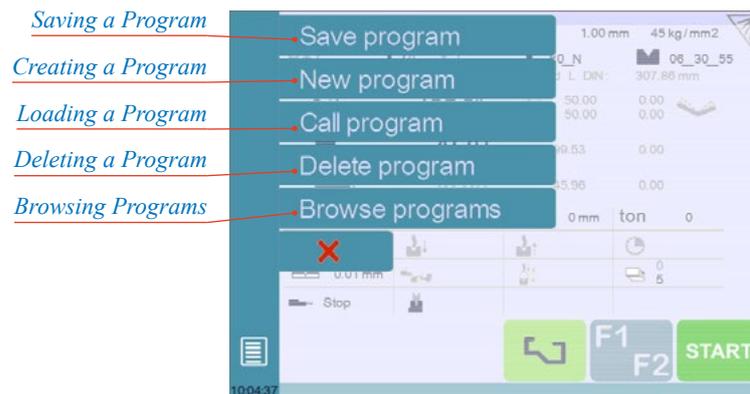
(Menu Button) → Other menus → USB transfer



MANAGING PROGRAMS

Because the production of parts follows cycles, the programs related to those parts need to be managed. They need to be stored, called into function, and so on.

On the CybTouch, the [Program number](#) (see page 20) is displayed on the top left corner of different pages. Touching it will open the following menu:



SAVING A PROGRAM

After creating a program, an operator can save the program in order to use it again:

1. Touch the [Program number](#) (e.g. **P0**).
2. Touch **Save program**.
3. Enter the number you wish to give to the program (e.g. **1** for **P1**), followed by .
4. The program is now called P1 and is saved in the CybTouch.

CREATING A PROGRAM

To create a new program:

1. Touch the [Program number](#) (e.g. **P1**).
2. Touch **New program**.
3. A yellow pop-up windows asks what kind of program:
 - » Selecting **Graphical** will open a new program in the [TouchProfile Mode](#) (see page 33).
 - » Selecting **Numerical** will open a new program in the [Numerical Mode](#) (see page 36).



LOADING A PROGRAM

To call (load) a program:

1. Touch the **Program number** (e.g. **P1**).
2. Touch **Call program**.
3. Select the program to be loaded from the list (e.g. **002** for **P2**).
4. The selected program (**P2**) is then loaded into the work memory and is ready to be used.

DELETING A PROGRAM

To delete a program:

1. Touch the **Program number** (e.g. **P1**).
2. Touch **Delete program**.
3. Select from the list the program to be deleted.
4. Touch  to confirm.

BROWSING PROGRAMS

This function allows browsing visually through the programs, and is particularly useful for graphic parts.

To browse programs:

1. Touch the **Program number** (e.g. **P21**).
2. Touch **Browse programs**.
3. Browse through programs using the  and .
4. Validate your choice by touching the browsing icon .



The browsing function remains active as long as the browsing icon  is displayed. Touching anything else but the  and  arrows will close the browsing mode.

EASYBEND PAGE



(Menu Button) → EasyBend



The EasyBend page is used for individual bends, for example when an external worker needs to interrupt production just to make a single bend (usually with the same tools).



The program currently being used for production is only temporarily interrupted (no need to save it) when switching to the EasyBend page, and then resumed again when returning to the program page (Menu Button → Current program).

MAKING A BEND ON THE EASYBEND PAGE



See [Basic Page Description, page 20](#) for more information on the different controls on the EasyBend page.



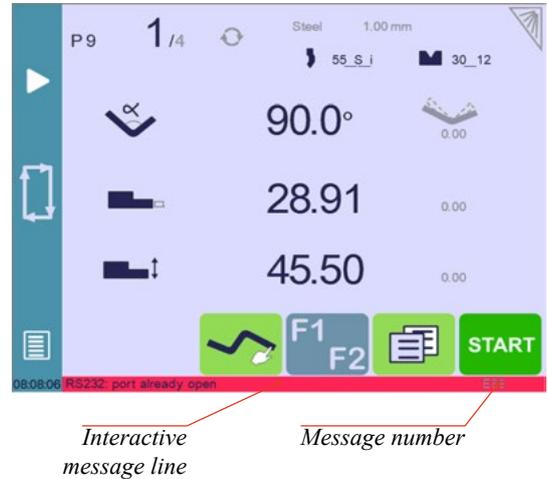
SETTING INSTRUCTIONS:

1. Touch the material's name (here **Steel**). The [Extra Bending Parameters \(see page 22\)](#) are displayed. Select the [Material \(see page 22\)](#) used, enter the [Material thickness](#) and the [Bending length](#).
2. Touch one of the tool's icon ( or ) to select a punch or die. To learn how to configure tools, please refer to section [Tools Management, page 27](#).
3. Enter the angle  for the bend you wish to create (here **90°**).
4. Enter the flange's length  (here **115.00 mm**).
5. Enter the back gauge height value  (here **45.50 mm**)
6. Press this button  to go to the [Extra Bending Parameters \(see page 22\)](#) and enter the eventually required extra data for the bend.
7. Start the hydraulic pump motor (by pressing this button  if available. It turns red when the motor is running).
8. Press the **START** button to position the machine according to the data that were just entered.
9. When the machine is ready to bend, a **OK** button is displayed.
10. Press the foot switch to execute the bend.

ERROR AND WARNING MESSAGES

Following is a list of warning and error messages which may be displayed on the interactive message line of the CybTouch. There are two types of messages:

- **Warning Messages**, which are displayed on a green background. They are information or instructions that will disappear automatically.
- **Error Messages** (machine or NC errors), which are displayed on a red background. They inform the user of an error occurring on the machine or NC, and sometimes require intervention by the end user or a technician.



When reporting error messages, please **ALWAYS** indicate the complete message, including eventual numbers between brackets [], and of course the error number at the end of the line. This number also refers to the first column in the section below.

WARNING MESSAGES

MSG NR.	MESSAGE	DESCRIPTION
W01	Not implemented	This message is displayed when a function is in development but not yet fully implemented.
W02	Ignore	This message appears when the desired action makes no sense, like for example clearing the indexation when the indexation was not made.
W03	Code accepted	This message is displayed when the correct password has been entered.
W04	Please press 2 seconds	This message reminds the operator to keep the start pump button  pressed for 2 seconds.
W05	The pump is on	This message appears after the pump starting cycle has been correctly executed.
W06	The pump is off	This message indicates that the pump has been stopped.
W07	Select at least one bend	This message is displayed when the operator tries to go to the Bend 2D screen from the Bend Order screen before defining the first bend of the sequence (see Bend Sequencing, page 34).
W08	Touch OK to continue	Indicates that a validation is required to continue.
W09	Not possible with graphic part	This message appears when trying to insert or delete a step in a graphic part.
W10	Cycle in progress	While this message is displayed, the machine cycle is in progress and the screen is locked, except for the Stop button.
W11	Machine is indexed	Indicates the indexation cycle was successful.
W12	Identification OK	In the Axis Wizard, indicates the identification cycle was successful.
W13	Touch the zoom center	When the zoom function  is activated, indicates that the HMI will zoom around the next touch on the screen.

MSG NR.	MESSAGE	DESCRIPTION
W14	Touch to create a new segment	When creating a new part in the TouchProfile Mode (see page 33) , this message reminds the operator to touch the screen to create a part's segment.
W15	Touch one length or angle to start the profile	When creating a new part in the L-Alpha Mode (see page 37) , reminds the operator to touch the table to begin creating a part.
W16	Eco mode	This message appears when the Eco mode starts, after the timer defined in the machine parameters is over.
W17	Enter the measured angle	This message is displayed when the operator must enter the physically measured angle value in the Bending and Corrections page.
W18	Empty field	Operator did not enter a value.
W19	24V I/O power on	24V to the inputs/outputs is now available.
W20	Please select a field	This message is displayed when trying to set the time (see Set Clock, page 13) and no field (minute, seconds, etc.) has been selected.
W21	Set seconds	In the Set Clock (see page 13) page, when the corresponding field is selected, indicates that it can be set using the up and down arrows.
W22	Set minutes	
W23	Set hour	
W24	Set day	
W25	Set month	
W26	Set year	
W27	Touch the small graphic to toggle screen	This message appears once at the beginning of the bend manual sequencing, telling the operator that he can toggle from the Bend Order page to the Bend 2D page simply by touching the graphic at the bottom of the screen (see Bend Sequencing, page 34).
W28	Diary empty	This message is displayed when the Fiessler security module's diary is empty. Something is not programmed correctly in the machine parameters. Ask a technician.
W29	End of list	This message is displayed when reaching the end of the list in one of the different menus  .
W30	Serial number from 100'000 thru 231'071	This message appears only when entering the serial number. It indicates the range of the number to be entered. Attention, this operation is normally done at the factory, with a serial number is related to the options installed on the machine. Do not change it!
W31	New option code	When installing a new option in the Service Page (see page 16) .
W32	Data entry in progress	Operation impossible: data entry in progress. Finish entering data and try again.
W33	Indexation in progress	Operation impossible: indexation in progress. Wait until the indexation is finished, or stop it, and try again.
W34	RFLink disconnected	When the RFLink connection to a laptop has been shut down from the latter.
W35	Access not allowed	Operator needs another level password.
W36	Machine not indexed	Beam is indexed, but not the rest of the machine.
W37	Moving direction has been inverted	Wizard message: Rotary direction of the motor has been changed.
W38	Counting direction has been inverted	Wizard message: Counting direction of the axis has been changed.
W39	Moving and counting directions have inverted	Wizard message: Both the rotary direction of the motor and the counting have been changed.

MSG NR.	MESSAGE	DESCRIPTION
W40	OK	Indicates a cycle or operation has properly ended.
W41	No movement executed	Axis Wizard message: Operator pressed  but no movement was made.
W42	Enter next measured angle	When using double or triple angle measurement, tells the operator to enter the next measured value.
W43	OK, correction done	Appears when an angle correction has been entered and applied.
W45	Enter unlock interface password	This message is displayed when parameter P02.04 Level 0 Lock HMI is set to yes and the screen is touched.
W46	Enter password level 1 or greater	This message is displayed when a password of level 1 or higher is needed to execute a specific operation.
W47	Enter password level 3	This message is displayed when a password of level 3 or higher is needed to execute a specific operation.
W48	Enter new password	These messages are displayed when changing passwords.
W49	Confirm new password	
W50	Enter password for backup	This message is displayed when trying to create a backup.
W51	Enter password for restore	This message is displayed when trying to restore a backup.
W52	Enter password for init	This message appears on the page displayed after the system crashed (soft or hardware problem), when the operator tries to format the machine.
W53	Enter password for delete all backups	This message is displayed when trying to delete all backups.
W54	Graphical part	This message is displayed when trying to display the flange's length on the Bend Numerical Page when the part was created with the TouchProfile Mode . With such parts, it is only possible to display the position value (X) of the back gauge.
W55	P+	Message displayed when using the Next-Part Function (see page 40).
W56	Limiting number of step bending	The total number of steps for the Step bending (see page 23) function is limited to 176.
W57	Filter Y1 to check	
W58	Filter Y2 to check	
W59	Enter password level 3 or restore	Message displayed when trying to transfer machine parameters from a USB key onto the CybTouch (see USB Transfer, page 19).
W60	Press next or previous button to browse the programs	Message displayed when Browsing Programs (see page 45).
W61	Default/Recalculate	Message displayed when touching  on a calculated field (see Set-point back gauge (R axis - optional), page 22).
W62	Press + or - to move	Message displayed when touching a specific axis on the manual movement page (see Manual Axes Movement, page 15).
W63	Only one bend	Message displayed when pressing the  arrow in the Bend 2D page (see Bend Sequencing, page 34), and the graphic part has only one bend.

ERROR MESSAGES

MSG NR.	MESSAGE	DESCRIPTION
E02	Pump motor off	The pump motor needs to be on for the sequence to start.
E03	Buffer Full	The part-program memory is full, you cannot add another sequence.
E04	Code refused	The level code to access the selected page is not correct. Try again or ask for it if you do not have it.
E05	File not compatible	The loaded part-program is incompatible with the NC. This part should be deleted.
E06	Machine parameter file problem	This file is corrupt and cannot be saved. Try to restart the NC. If the problem persists, format the memory.
E07	Machine parameters not compatible, please format data	This message appears when a software update has been made over a much older version and the parameters are no longer compatible. It can also appear if the uploaded parameters (with Rflink) are much older or newer than the current software version and they are not be compatible. A new start up of the machine must be made. Contact your dealer.
E08	Lismisc File not compatible	Information message, which will disappear when restarting the NC.
E09	Save program problem	This file is corrupt and cannot be saved. Try to restart the NC. If the problem persists, format the memory.
E10	File not found []	A file is missing and the code indicates which one. Call Cybelec with this code to know which file is missing.
E11	Write to file problem	This file is corrupt and cannot be saved. Try to restart the NC. If the problem persists, format the memory.
E12	X under minimum limit	Operator entered a value under the limit, or a memorized value in the program is under the limit. The wrong value flashes and must be corrected.
E13	X over maximum limit	Operator entered a value over the limit, or a memorized value in the program is over the limit. The wrong value flashes and must be corrected.
E14	Fw SetVar Error []	May occur when a feature is configured, but the dedicated input/output is not configured. Usually this is solved by loading the default input/output configuration (see the machine parameters).
E15	Bottoming not allowed	This message is displayed when a bottoming sequence is programmed and bottoming is not allowed with either one of the tools.
E16	Fw Axes Error [] ...	Axis manager error. The number gives more information. Most common errors are described in messages E55 to E68 . If other error numbers are listed, please send conditions of problem, traces and parameters to the Cybelec Technical support for assistance.
E17	Programming error	Machine parameters incorrectly configured, the error page is displayed.
E18	No Backgauge defined	All the back gauges programmed in Machine Parameters have none selected for mode. Select at least one mode other than none.
E19	Quantity = 0	When pressing start, the programmed amount of parts to be made is '0'. See Number of parts, page 25 for more information.
E20	Cycle repeat = 0	Cannot start cycle because repeat cycle function is set to "0".

MSG Nr.	MESSAGE	DESCRIPTION
E21	No material defined (define one or more in MP)	No material programmed in the Materials (see page 11) page. A material must be selected to perform calculations.
E22	"Safety speed" input activated	The beam stopped because this input was activated while the beam was still in HS down. The beam must first switch to LS down before this input can be activated.
E24	Identification Error []	During the Axis Wizard, there was an error identifying one of the axes. The error number (typically E55 , E56 or E57) gives more information. See also message E16 .
E25	No FAST task running []	Switch OFF the machine for 1 min and restart it again.
E26	NULL pointer to axis struct.	This message indicates a software bug. Write it all down and contact Cybelec.
E27	MUTEX Error []	This message indicates a software bug. Write it all down and contact Cybelec.
E28	I/O no 24V or overload (output in safety off)	The 24V power supply for the inputs/outputs is no longer present or an output is overloaded. Reset any safety device on the machine, check protection grids and rear guards are closed, etc. If the problem persists, switch machine OFF for 3 min and restart it again. If the problem still persists, check the machine manual and/or ask a service technician to check your machine.
E29	Radio link error, code []	The RLink chip has detected an error. Check the environment for disturbances (cell phone, wi-fi) and that the material works properly. If the problem persists, write the error number down and send it Cybelec.
E30	Touchscreen error, code []	Please contact your machine dealer with this specific code and details.
E31	"Crowning" analog input not configured	While configuring (setting up) the NC, the Crowning dedicated input was not configured but is requested to run properly.
E32	"Crowning" analog output not configured	While configuring (setting up) the NC, the Crowning dedicated outputs were not configured but are requested to run properly.
E33	Syntax error in XML file	This file is corrupt and cannot be used. Try to restart the NC. If the file is a part-program, try to delete it.
E34	Memory allocation problem (xml)	There was a problem while trying to read a file in the memory. The file is probably corrupted. The number gives more information, write it down.
E35	Endless loop on process task	Process error. Please restart the NC and inform your dealer.
E37	WARNING: Overloop intern	This error should normally never happen on the machine. It means there are too many elements in a coded list.
E38	Unknown key	There is a list of known screen zones, and the pressed zone is not in it. This error can normally not happen in the field.
E39	"Start" input refused	The start command is not accepted in this page/situation.
E40	R under minimum limit	The programmed position value for axis R is below the value of the minimum limit switch position.
E41	Thickness smaller than min	The thickness entered for the material is below the minimum value defined in the User Preferences (see Materials, page 11).
E42	Thickness greater than max	The thickness entered for the material is above the maximum value defined in the User Preferences (see Materials, page 11).
E43	Configuration error	Input/output incorrectly configured in machine parameters, the faulty input/output page is displayed. Check for unauthorized doubled outputs or inputs. This message can also be displayed if the chosen configuration requires more icons on the first page than there room available.

MSG NR.	MESSAGE	DESCRIPTION
E44	"External stop" input is active	External stop may be caused by safety devices, emergency buttons, rear protection guards, etc. See machine instructions.
E45	Error: backgauge "rest" mode and retract	It is not allowed to program a Back gauge retraction (see page 23) with a support type of back gauge finger.
E46	"Pressure" analog output not configured	While configuring (setting up) the NC, dedicated input or output are not configured but are requested to run properly.
E47	R over maximum limit	The programmed position value for axis R is above the value of the maximum limit switch position.
E48	Stop doesn't exist	This error means that the back gauge in the program no longer exists when the operator tries to make a bend. Select a valid back gauge.
E49	Beam locked	The back gauge cycle (see Back Gauges, page 32) cannot start because the beam is locked.
E50	Value out of limit	This message is displayed when the value the operator is trying to be program is bigger than the maximum authorized value.
E51	Error[][][]...	Internal management error. Write the error's codes down and the software's number (see Information, page 17) and call Cybelec.
E52	Punch does not exist	This message is displayed when trying to create a new part before having selected a punch in the list (see Punches, page 27).
E53	Die does not exist	This message is displayed when trying to create a new part before having selected a die in the list (see Dies, page 29).
E54	Beam too low for crowning movement	The beam is still in its secure zone, which is defined in the machine parameters. If this happens during a cycle, it means that the current step TDC is not high enough and needs to be manually programmed in the Extra Bending Parameters (see page 22).
E55	Identification Error 1 (No motion detected)	No motion detected. Should not happen if you started the Wizard from the beginning. If the error remains, check limit switches, drive, wiring, etc.
E56	Identification Error 2 (Not enough oscillations)	Not enough oscillations Increase the identification time. See machine parameters manual.
E57	Identification Error 3 (Amplitude of the oscillation)	Oscillation amplitude. Increase the identification voltage. See machine parameters manual.
E58	Fw Axes Error 32 [Trajectory tracking error]	This is a regulator error. The axis could not follow its trajectory. This may be due to high friction, resistance or an obstacle on the axis movement. This may also be a drive problem. Call a technician.
E59	Fw Axes Error 311 [MaxSpeed too high !]	Max speed or encoder resolution too high.
E60	Fw Axes Error 312 [MaxSpeed too small !]	Max speed or encoder resolution too low.
E61	Fw Axes Error 313 [Acceleration too small or MaxSpeed too high !]	Acceleration too low (mm/s ²) or max speed too high. This needs to be corrected. Please note that acceleration is not a ramp distance.
E62	Fw Axes Error 314 [Acceleration too high or MaxSpeed too small !]	Acceleration too high (mm/s ²) or max speed too low. This needs to be corrected.
E63	Punch pressure	The pressure calculated for the current bend (in Ton/m) is higher than the maximum limit allowed for this tool.
E64	Die pressure	

MSG Nr.	MESSAGE	DESCRIPTION
E65	The beam is not indexed	This message appears when trying to move the crowning manually and the beam is not indexed.
E66	Fw Axes Error 33 [Maximum voltage time exceeded (10V)]	This is a regulator error. The axis could not follow its trajectory. This may be due to higher friction, resistance or an obstacle on the axis movement. It may also be a drive problem. Call a technician.
E67	Fw Axes Error 39 [Speed tracking error]	This is a regulator error. The axis could not follow its trajectory. It may be due to higher friction, resistance or an obstacle on the axis movement. It may also be a drive problem. Call a technician.
E68	Fw Axes Error 316 [MinPosition or MaxPosition outside limit !]	Axis position counter is out of max or min limit. Verify physical axis position and set the axis counter accordingly.
E69	The machine is not indexed ! No limit	Before the machine is indexed, the NC doesn't know where the axes are. In manual page, movements are authorized but the electronic stroke limits are not activated. Operator is responsible for stopping axis movement before mechanical limit is reached.
E70	Progr. angle is smaller than the die angle	The programmed angle is smaller than the die angle. Change die.
E71	Progr. angle is smaller than the punch angle	The programmed angle is smaller than the punch angle. Change punch.
E72	Depth safety	The calculated angle causes the punch, material, and die to collide and approach coining mode. Operator must confirm to proceed.
E73	Collision back gauge-tool	This message appears to warn that, with the parameters cannot be entered as such, for they would cause a collision between the back gauge and the tools. Check your parameters' values.
E74	Beam error No [][]	This message indicates an error of the beam. The first number refers to the message number (see Beam Error Messages, page 56).
E75	Beam: static gains not found	
E76	Error in the parameter beam (you must correct it before continuing)	
E77	Error in the sequence parameter (you must correct it before continuing)	
E78	RS232: parity error	
E79	RS232: overrun	
E80	RS232: framing	
E81	RS232: noise	
E82	Security module event No xx	
E83	Security module error time out	
E84	Security module error checksum	
E85	Security module error buffer rec to small	
E86	File access error	There was an error when trying to access a file while programming an option. Make sure that the code was entered properly. If it is not working try and restart the NC. If the problem persists, contact Cybelec.

MSG Nr.	MESSAGE	DESCRIPTION
E87	Unknown error	An unknown error occurred while trying to program an option. Contact Cybelec.
E88	RS232: port already open	
E89	Solution not found	This message indicates that the bending sequence automatic search did not find any solution. Try to create the bending sequence manually. If not possible, it means that this part is not feasible on this machine.
E90	Too many buttons on bend num page	Too many buttons are configured for the available space on the Bend Numerical Page (see page 20) . The last one, and all the following will be refused. A technician must check the configuration.
E91	Next program error	This message appears when there has been a problem with the Next-Part Function (see page 40) . The problem appeared when going to the next part: it doesn't exist, or could not be read.
E92	Step bending-too many steps	The programmed step-bend cannot have so many steps. Try reprogramming it, and eventually the whole part.
E93	Beam drive error	There is an error on one of the drives controlling the beam. Check them and resolve the problem. They must be ready to run.
E94	F_CycStopAxe failure, please make a "Tracer"	Should not happen. If it does, call a technician to make a trace and have him send it to support@cybelec.ch .
E95	Combined machine IO configuration	(Option K1 - combined machine) – configuration of the outputs/inputs is not correct when option K1 is activated.
E97	Negative face length	Flange length is too short for the radius programmed.
E98	Incorrect face	This message means that the face number programmed in the bend sequence is outside the limits. This message can normally not happen on a part programmed on the CybTouch, but could on an imported part. Reprogram the part entirely.
E99	Illegal stop	This message means that there is a problem between the stop number and the face number programmed in the bend sequence. One or the other is outside the limits. This message can normally not happen on a part programmed on the CybTouch, but could on an imported part. Reprogram the part entirely.
E100	Stop does not suit	No gauging solution was found for the current bend. This bend cannot be made with the actual machine configuration.
E101	Station error	(Option K1 - combined machine) – this message is displayed when trying to go in auto mode, and no station is defined. Or if one of the switches defining the current station changes state while working.
E102	Cycle station Time out	(Option K1 - combined machine) – the duration of the station change cycle exceeded the maximum allowed. Check if the station movement is not impeded in any way.
E103	Clamping opened	The machine cannot start if the tools are not properly clamped. Check the status of the tools on page Manual Axes Movement (see page 15) .
E104	Station locked	(Option K1 - combined machine) – it is not possible to work on the station, because the tooling is still locked (input Unlocked not active). Check the locking device of the station.
E105	Cylinder not in position	(Option K1 - combined machine) – the station cannot change position, because one of the cylinder's status is unknown, i.e. it is neither closed nor opened. Correct the situation from the page Manual Axes Movement (see page 15) .

MSG NR.	MESSAGE	DESCRIPTION
E106	Machine Err code D2. Contact your dealer	Contact your dealer.
E107	X relative impossible	This message is displayed when trying to select the relative mode for the positioning of the back gauge on a graphic part, or on the first step of a program (see Set-point back gauge (X axis) / flange length, page 21).
E108	Beam under minimum limit	When pressing Start, the CybTouch checks if the targeted position for the beam is within the limits. Check your program.
E109	Beam over maximum limit	

BEAM ERROR MESSAGES

MSG NR.	DESCRIPTION	COMMENT
13	Command refused because pinch point is below Y1 and Y2 bottom dead center	Check the value of the PP correction, and the value of the BDC correction in the User Preference page.
14	Command refused because the speed commutation point is below the pinch point	
15	Command refused because low speed up speed out of limits	
16	Command refused because speed out of limits	
17	Command refused because Bottom Dead Center Y1 and Y2 are too different	Difference between target values of Y1 and Y2 bigger than the limit value defined in the machine parameter. Call you dealer.
18	Command refused because pinch point is outside stroke limits	
19	Command refused because beam is moving	
20	Command refused because Bottom Dead Center is outside the beam limits	
26	Command refused because beam is in emergency	For example, beam out synchronism tolerance.
27	Command refused because Synchronism controller parameters not in range	Problems with limit parameters (max and min) of the beam. Contact your dealer.
28	Command refused because stop is active	
37	Command refused because zero has not been found yet	Beam is not indexed.
40	Command refused because bend data have never been loaded	
44	Command refused because beam is in emergency. Only a manual down of the upper axis is allowed	Only the axis with the highest position is allowed to be moved manually down.
49	Command refused because beam is not at high end of stroke position	
52	Command refused because the beam is requested to go to TDC	
53	Command refused because a beam cycle is not running	
248	Synchronism limit reached, Y1: {0} Y2: {1}	Height difference between Y1 and Y2 bigger than the limit value defined in the machine parameter. Call you dealer.
265	Error with Oil Leakage Control. Canceling operation	
267	Beam not indexed, no move	
606	High speed down stopped by safety speed switch	

MSG Nr.	DESCRIPTION	COMMENT
607	Command refused because other command still active	Order of priority is as follows : 1. Stop Command 2. Go to TDC MAX 3. Go to next TDC 4. Up command 5. Down command



This table does not contain an exhaustive list of all the errors related to beam. If the message number is not explained here, please write it down and contact your dealer.

INTERNAL RESET ERROR MESSAGES

This kind of messages appears if a serious error occurred. The display automatically switches to a page as shown below.



If such an error occurs,

- Write down the error number and press the Restart button.
- If the error remains, turn the machine off and let it cool down before restarting it.
- If the error still remains, write down the error number and contact your dealer.

