Live out your dreams

DONATONI
HIGH INNOVATION STONE MACHINES

SECTION B

Software manual

Rev.Doc. 0.0 – Vers.SW 4.9.2

TRANSLATION OF ORIGINAL INSTRUCTIONS

Donatoni Macchine Srl
Via Napoleone 14 - 37015 Domegliara
S. Ambrogio di Valpolicella - (VR) Italy
Tel. +39 045 686 25 48

Fax. +39 045 688 43 47
info@donatonimacchine.eu
donatonimacchine@pec.it
www.donatonimacchine.eu

Cod. fisc./P.IVA VR 03767560232
Capitale sociale: 1.000.000€ i.v.
Numero R.E.A.: 363818

SOFTWARE MANUAL

A publication by Donatoni Macchine S.r.l.

Document: MN SW_sw4.9.2_rd0.0_it.docx

Date: 11/04/2022

Great care was taken to give accurate and comprehensive information in this manual. Any errors, omissions and/or improvements can be communicated to the following address:

DONATONI MACCHINE S.r.l
Via Napoleone, 14
37015 PONTON – DOMEGLIARA
VERONA (ITALY)

Sales Office	*Engineering Office*	*Assistance*
Tel. +39 045 68 62 548 *Fax +39 045 68 84 347*	*Tel. +39 045 68 88 604*	*Tel. +39 045 68 62 899*

Email address: info@donatonimacchine.eu

List of updates

Rev. Doc.	Vers. SW	Description	Approved by	Date	Page
--	4.9.2	First issue		--	All

Code Issuing body: Engineering Department	**Drawn up by**	**Verified by**	**Pages**
File: MN SW_SW4.9.2_RD0.0_IT.DOCX	P. Novello	N. Chesini	

TABLE OF CONTENTS

1 INTRODUCTION

1.1 GENERAL INFORMATION

The information in this document is only applicable to the following software versions shown on the cover page.
It is possible that not all the functions the product can carry out are described in this documentation; Where the above is the case, Donatoni Macchine shall not necessarily guarantee these functions or keep them in future versions.

This document is supplied as annex "Section B", software part, of the machine use and maintenance manual.

1.2 PURPOSE OF THE MANUAL

This manual contains the instructions for use required to become suitably familiar with the Parametrix software and useful information to identify solutions to malfunctions.
The smooth operation, durability and economy of operation of the machine depend on compliance with the instructions described in this manual.
The machine use and maintenance manual and this software manual must always be available in the place where the system is installed, in a location that is safe and easy to access. If a copy is lost, contact the Prevention and Protection Service Manager.
Failure to comply with the instructions in this manual, negligence, incorrect or inappropriate use of the machine, can be grounds for cancellation by the Manufacturer of the warranty accompanying the machine. In addition, the benefits for which this machine was built will not be fully exploited.
DONATONI MACCHINE SRL is at your disposal to ensure prompt and precise technical support and everything that can be useful to maximise operation and to obtain the highest performance from the machine.
The Manufacturer reserves the right to modify, at its discretion, this publication and the information contained therein if it deems necessary for the technical or commercial improvement of the product.

1.3 RETENTION OF THE MANUAL

This Software Manual must be kept for the entire operating life of the machine, even in the event of sale to a third party.
In order to ensure correct retention of the Software Manual, it is recommended to:

- employ the Manual in such a way as to avoid damaging in whole or in part, the contents; in particular, it is advisable not to lose the manual during use and to ensure it is returned to the assigned place immediately after its consultation;
- do not remove, tear or rewrite parts of the Manual for any reason whatsoever;
- keep the Manual in an area protected from moisture, heat, and other environmental agents that could impair its integrity or durability;
- in the event of loss or damage to all or part of the contents of this Manual, request a copy of this documentation from the manufacturer.

1.4 RESPONSIBILITY

Responsibility for this document is entrusted to Donatoni Macchine S.r.l.

1.5 GRAPHIC SAFETY SYMBOLS

The texts and descriptions of particular importance for the safety of personnel and the proper use of the product, including the inappropriate behaviour to be avoided and the related obligations and prohibitions are highlighted in **BOLD CAPITAL LETTERS**.
The following symbols are used in the instruction manual:

	**DANGER** This indicates a situation that may result in injury, death, or serious damage to the health of persons.
	**CAUTION** This indicates a situation that may result, directly or indirectly, in damage to persons, property and to the environment with consequences also of financial nature.
	**WARNING** This indicates a situation that may result, directly or indirectly, in damage to persons, property and to the environment or to the machine as a result of incorrectly performed operations.

1.6 GLOSSARY

RTCP	Rotation Tool Centre Point
FRC	Flange Rotation Centre
BKSP	Back Space
CSV	Comma Separated Values
DXF	Drawing Exchange Format

1.7 USERS OF THE MANUAL

This document contains information aimed at:

- machine operators,
- Engineers authorized to provide support

1.8 STRUCTURE OF THE MANUAL

The document is subdivided into chapters that describe:

- Knowledge of the software structure
- Functions of the machine
- Interlocks and rules for correct usage of the device

2 INFORMATION ON TECHNICAL ASSISTANCE

In case of necessity, or when clarifications are required, customers can contact the technical/commercial support service of the local agents or importers, who are always in direct contact with the manufacturer.
In most cases, many of the technical problems can be resolved with minor interventions. We therefore advise you to read this manual carefully before contacting the Assistance service.

DONATONI MACCHINE S.r.l.
Via Napoleone, 14
37015 – DOMEGLIARA
VERONA (ITALY)

Sales Office	*Engineering Office*	*Assistance*
Tel. +39 045 68 62 548 *Fax +39 045 68 84 347*	*Tel. +39 045 68 88 604*	*Tel. 045 68 62 899*

E-mail addresses: info@donatonimacchine.eu

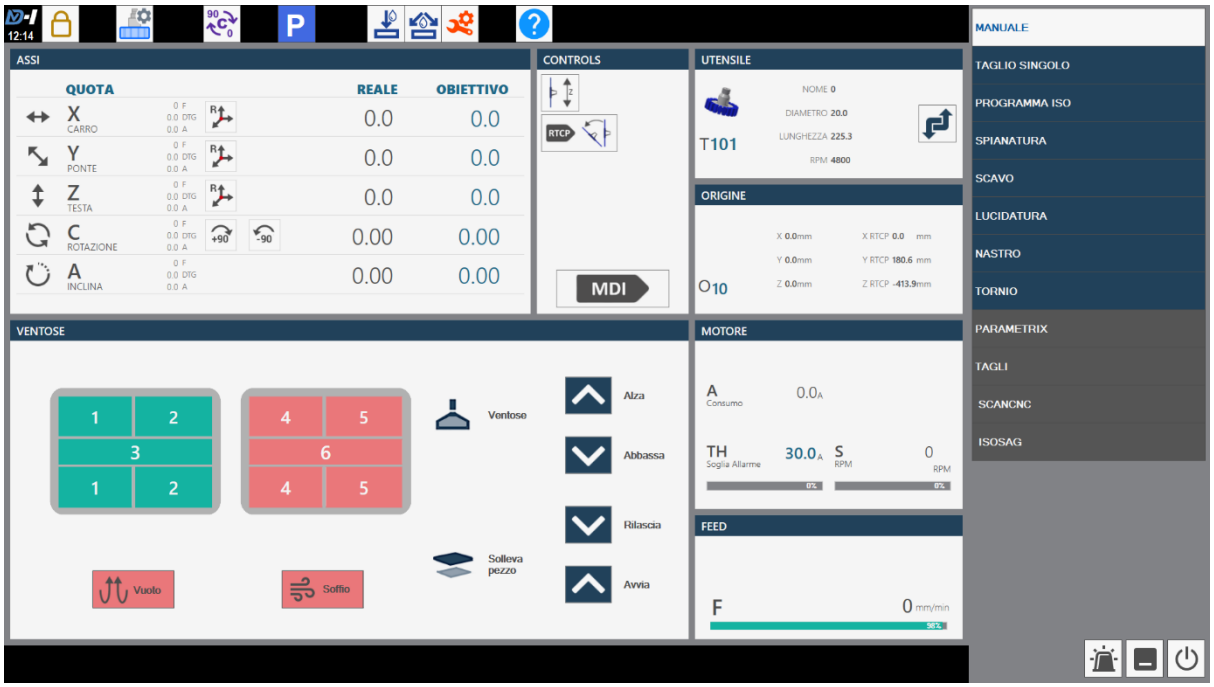
In the case of faults or malfunctions that cannot be resolved, the user can contact the manufacturer directly.

3 ON BOARD THE MACHINE

The system permits interaction with the machine by pressing buttons on the monitor and by operating the commands on the console.
In order to help the user, the pages are first of all illustrated with an overall view and then the individual buttons are explained. An attempt has been made to associate an image to these in order to render their function intuitive and they are used on different pages but always actuate the same command.
If there is no explanation of the meaning following the overall view of the page, try leafing back through the manual from the current page.

3.1 STARTING THE MACHINE

To start the machine move in front of the control panel:
1. Turn the main switch into the "ON" position.
2. Move the 0-1 switch to "1". When the PC and CN start cycle is completed a light LED comes on inside the switch
3. Once the LED of the 0-1 switch has come on, press the "Enable controls" button which in turn will become green.



With ignition of the machine, the control system is started automatically and displays the page shown in the figure to allow the operator to start the work phases.

WARNING: *Before moving the axes of the machine, reset them to zero as described in paragraph "2.1 Manual"*

3.2 SWITCHING THE MACHINE OFF

After pressing the "Off" button, the following message appears: “Confirm switching off of the machine?”.

- With the button "*OK*" the machine starts the switch off cycle.
- With the "*Cancel*" button the program returns to the start page.

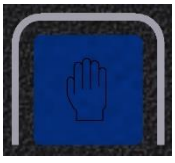
Once shutdown is confirmed wait until the monitor is completely black and move the switch on the electrical panel to 0.

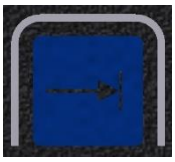
WARNING: *Before moving the switch to 0 wait until the monitor is completely off (black) to prevent damaging the management software or breaking the PC of the machine.*

3.3 USING THE BUTTON PANEL

The manipulators, potentiometers, functions buttons and the emergency mushroom button for immediate control of the machine are located on the button panel. There are other buttons on the top part, between the keyboard and the monitor, with various functions including starting and stopping the cycle.

Manual movement is possible using the manipulators on the button panel, while the speed can be adjusted using the potentiometers alongside them. The speed of axes X and Y is adjusted using independent potentiometers; the potentiometer marked as “INTERP. AXES” is used to control the speed of axes Z, A and C.

“CYCLE STOP” button: Pressing this button blocks any movements and puts the machine into standby mode. When the start button is pressed movement resumes from where it was interrupted. NOTE When the button is pressed for more than 5 seconds, the alarms are reset and the movement command is cancelled completely.	
"CYCLE START" button: the automatic programs are launched or restarted after a cycle stop when this button is pressed (see above).	
"RESET" button: this button cancels all movements and any programs that are running. It is also used to reset any active alarm. NOTE This button must not be pressed simultaneously with a manual movement of the axes to prevent sudden and harmful blocking.	
"LOW SPEED" button: pressing the button activates movement of the axes at low speed (3 m/min). Activation of this mode is indicated by the continuing orange light on the column above the monitor and the message "Slow JOG active" on the monitor. WARNING: With slow JOG active, the speed of all the axes is controlled by the interpolated potentiometer, including the X and Y axes.	

"Laser" button: pressing this button switches the “Cut marker” laser on/off.	
“Cross-shaped laser” button: pressing this button switches the cross-shaped laser on/off.	
“Cutting length acquisition” button: this button has 2 functions: If the button is pressed when performing a cut using the "Single cut" function a new cutting length is acquired. See the Single Cut chapter (in the Basic Programs on Board the Machine section).	

3.3.1 STARTING, STOPPING AND RESUMING AN AUTOMATIC WORK CYCLE

Starting a work cycle
Starting an automatic work cycle is subordinate to the selection by an operator of one of the automatic enabling buttons. The next stage envisages turning on the water (not obligatory) and the blade or tool (obligatory) and enabling the automatic work cycle by means of the CYCLE START button.
In the event a work cycle to be started without water, which will then be activated at a later stage, the system allows axis movement to be temporarily stopped. This is possible by pressing the CYCLE STOP button for no more than two seconds; to restart after a temporary stop, the CYCLE START button must be pressed again.

Stopping a work cycle definitively
A work cycle can be stopped definitively at any time by pressing the associated button on the control panel for more than five seconds or the RESET button.

Resuming the automatic work cycle
It is possible to suspend the automatic work cycle temporarily at any time by pressing the CYCLE STOP button for no more than two seconds.
The machine automatically stops in the temporary STOP position and waits for the restart command, given by the CYCLE START button that must be pressed for two seconds.

3.4 AXES ZERO-SETTING

Whenever the machine is switched off, it is then necessary to zero-set the axes to restore the correct reference points with the following procedure:

press the "Zero" button	
Wait for the zero indicators to turn green before moving the machine axes, as shown in the images.	

3.5 HARDWARE AND SOFTWARE SAFETY DEVICES

The machine is fitted with safety mechanical systems (described in the dedicated manual) and software systems which trip automatically if there are any program faults or incorrect operating procedures, stopping any operation.

3.5.1 AMPERE THRESHOLD

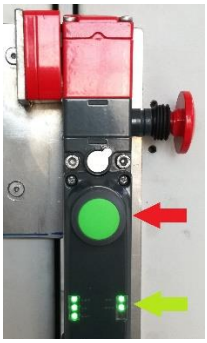
The Ampere Threshold parameter is worth a mention among the options available. This parameter is present in several interface pages and is to be used to protect the machine and the operator against faults in the tools or if wrong parameters have been set. If the spindle consumption exceeds the set threshold, automatic programs are paused. We recommend setting the value based on the installed tool and the ongoing machining process.

CAUTION: *When the machine is left unsupervised by the operator, it is advisable to set a medium-low value*

When the safety device is active, the "spindle extra current" message is displayed; once the causes have been verified and solved, it is possible to restart the operation by pressing the "start" button several times.

3.5.2 ACTIVATING THE SAFETY MODE USING THE FRONT PROTECTION BARRIERS

With the "open doors" status which is defined by the electric condition of the electric lock (yellow arrow), a limited operating mode is available to comply with the safety regulations in force.



The following operations are disabled:

- Switching the spindle on
- Switching the coolant on
- Starting any program on the machine
- Starting axes zero setting
- Jogging for inclined cuts using the associated mechanical manipulator
- the RTCP function
- The automatic parking function
- The manual tool change function

The following functions are enabled:

- Tilting bench ascent and descent
- Movement of the X-Y-Z-C-A axes with speed reduced to 30% of the nominal speed of each axis
- Programming the work parameters

Whenever the safety mode is activated with open barriers, any programs in progress can be automatically stopped by pressing the button indicated by the red arrow and put in "WAITING" mode until the safety condition is deactivated. Then, the "Cycle Start" button must be pressed.

3.5.3 INTERRUPTION DUE TO WATER FAILURE

The machine is fitted with a pressure switch which measures the pressure of the water entering; this must be set on the basis of the water required for the machining operations you intend to carry out.
During the automatic cycles the pressure switch is activated to determine if there is sufficient water pressure. Then, if the water supply fails the current cycle will be stopped automatically and the error message activated. The error must be reset by the operator when they find that there is sufficient pressure to continue with the current machining operation. The cycle in progress can be restarted by pressing the Cycle start button.

3.5.4 COLLISION VALUE REQUEST

The machine is fitted with a software control system checking for the destination value set for the various axes and notifying the operator with any error for possible collision.
To reset the error, the “Cycle stop” button needs to be pressed for more than 5 consecutive seconds, or any axis needs to be moved to update the position values.

3.5.5 EMERGENCY BUTTON

In the case of serious and imminent danger for persons or property, press the emergency button on the operator panel (Figure below).
Pressing the button:

1. immediately stops the axes
2. stops the spindle
3. disables several optional commands such as: coolant water.

A few fractions of a second after the button has been pressed, all the electrical devices in the machine are disconnected from the power supply.
From this point on, the brake is activated in the braking axes (Z, C and A), while the other axes (X and Y) are free to move (unless mechanical impediments or motion transmission limitations exist).

When the emergency situation is over, work can be resumed. To do so, release the emergency button by gently turning it anticlockwise and reset the control buttons by pressing the “Enable controls” button featured on the door of the electric panel.

3.6 MACHINE PLC ALARMS

PLC alarms normally trip for causes in connection with incorrect operation of the hardware components in the machine, for instance motor drives and the pressure switch, or also in connection with missing water or a power failure to the machine.

Alarm / warning message description	**Causes**
Direct Jog On	It indicates that the axes are still to be zero-set and that no checks are performed on the axis strokes. Unless there are special cases, avoid any movement and proceed with the zero-set procedure. The same message is displayed when the machine is the warehouse area.
Slow Jog On	Slow JOG mode is enabled when the relevant button is pressed and it is highlighted by an orange light turning on steadily. The mode sets 2m/min as the maximum speed for all the axes.
Cycle start request	When the Start Cycle button is pressed in the various work cycle setting pages, the PLC checks that the spindle has started and sends a signal to the operator if the result is positive so that he can then definitively start the cycle safely by pressing the Cycle Start button on the operator control panel once again.
Operator hold	“STANDBY” mode It is enabled whenever an ongoing automatic cycle is stopped after pressing the Stop cycle button. If the button is pressed for a while, the message appears. When the Start cycle button is pressed, the work program is restarted from the point where it had stopped; if the Stop cycle button is held down for over 5 seconds constantly, the cycle is stopped completely.
Standby for spindle starting	this message is displayed while waiting for the spindle to reach the revolutions set in the parameters.
Blade not started and Water not started	It may be necessary to reactivate it manually in case of water interruption.
Override zero	This is displayed when one of the potentiometers for setting the speed is at zero.
General emergency	The General Emergency alarm message is triggered when the red emergency button on the operator control panel has been pressed. To reset the error, it is necessary to release the emergency button and re-enable the controls by means of the button on the door of the power board.
Controls not enabled	This is displayed when the machine is turned on or after pressing the emergency button. It reports that it is necessary to press the Controls enabling button on the power board door.

Alarm / warning message description	**Causes**
Spindle extra current	It indicates that the threshold set in the parameters has been exceeded. Verify the cause and press start or blade start to delete the alarm and press again to restart.
Pump thermal relay	The pump thermal relay alarm is triggered by incorrect operation of the oil pump during the ascent or descent of the tilting bench (optional). Switch the machine off and de-energise the entire machine. Wait for 5 minutes and open the electrical panel to check the condition of the protection system. Contact specialised personnel.
Blade thermal relay tripped	It indicates an alarm on the inverter that controls the spindle rotation. Press the emergency button, wait about one minute and resume operation. If the problem persists, contact the technical assistance.
Inverter alarm	It may be due to multiple checks performed by the software. Switch the machine off and de-energise the entire machine. Wait for 5 minutes and open the electrical panel to check the condition of the protection system. Contact specialised personnel.
Collision limit switch on	When this message is displayed, it means that the Z axis has lowered and the anticollision system has consequently triggered. Check the machine condition and cautiously move it upwards.
Power board overheating	A thermostat in the power board signals that the set threshold has been exceeded. If an alarm is triggered, check the effective temperature and the operation of the air-conditioning system the power board is fitted with.
Open door	It indicates the "not closed" status of the electric lock. Close the door and press the dedicated button to lock it.
Incorrect "A" axis position	The system generates the "Incorrect A axis position" message whenever the "Parking" or "Manual tool change" buttons are pressed and the "A" axis is not in the correct position (zero degrees).
Emergency from remote control unit	The "Emergency from remote control unit" is triggered by the system when the mushroom button on the remote unit (optional) for positioning the axes is pressed.
Bench zero acquired	The "Bench Zero Acquired" alarm message is displayed when the bench limit acquisition button is pressed; the software memorizes the position beyond which when the bench block is set, the Z axis will not pass the set limit.

Alarm / warning message description	**Causes**
Error of the drives	This message indicates a malfunction or a power error affecting the drives. Other specific messages may appear. NOTE No operation must be performed with the machine, including resetting of the axes and full machine restarting; wait a few minutes before switching the machine back on again.
Tilting bench not in position (bench optional)	In order to carry out any machine movement it is necessary for the bench to be completely down so that the limit switch under the bench enables the use of the machine and allows the movement to be made.
Lubrication system failure	The lubrication system failure alarm appears when the grease control unit notifies any fault. Make sure that the minimum quantity of grease is present inside the tank and then focus on the distribution path. To reset the system after this error activate the pump using the dedicated button in the "statistics" page and press the reset button on the pump.  
Spindle fan thermal relay tripped	The alarm “thermal relay active in spindle electric fan” is triggered when there is an electrical fault in the panel. Solution Check that the remote switch in the panel is active. If it continues to trip, it must be checked.
Pressurization failure	There are various alarms associated with the air pressure depending on the devices to be controlled. In general the alarm is triggered by a machine stoppage caused by hazards that may occur. If the alarm appears, check the pressure available in the system and that there are no leaks along the air path.
Active vacuum pump thermal switch	It indicates that the vacuum pump motor protection has tripped via the circuit breaker upstream of the motor. Switch off the machine, open the electrical panel to verify the operation of the device and contact specialist personnel to check the motor.
Insufficient air	This message indicates an excessively low input pressure. Check the pressure gauge and if necessary contact the support service.
Vacuum not generated	When prompted for a command to check if the vacuum is active, if within the timeout set by the manufacturer the vacuum signal is not received, the alarm is generated. Check that all the suction cups selected are in contact with the material, check that there are no leaks and if necessary contact the support service. To release the situation, press the reset button and switch off the pump using the appropriate button.

Alarm / warning message description	**Causes**
Check the "..." Axis motor notch	It is displayed just after zero-setting. There is a specific message for each axis and notifies that there may be a difference with respect to previous zero-setting operations. Try zero-setting again, check the position of the rotary axes and if necessary contact the technical assistance. It is not a prohibitive error and it is still possible to operate the machine when it is present.
A axis calibration error	During zero-setting the procedure is stopped if the reference sensor is not detected within a certain time. Check the zero-setting sensor.
TOOL+ not in parking position	If the optional device is present on the machine, a sensor detects the position of the additional spindle outside the work area (in parking position) Check the actual position of the additional spindle and check the sensor, if necessary.
TOOL+ pressure failure	The pressure switch for checking the air on the additional spindle does not detect sufficient pressure. Check that there is air in the system and that air comes out of the front of the tool+.
TWIN - Lifting bloc lifted	With the twin optional a mechanical lock is present to secure the position of the tables. Some sensors detect the position of the pins, which must be in the low position during the movement. Make sure that they are in the low position and check the sensors, if necessary.
No tool mounted	(Optional) It indicates that the motor clamp is closed and no tool is inserted in the motor. Rotation is disabled.
Tool not defined	(Optional) It indicates that a tool is present on the motor but its data have not been set. Set the number of the tool mounted.
Tool present in the selected workstation	During the tool unloading phase, a check is carried out to ensure that the position to be used is free. Cancel the cycle and check the positions in the tool holders.

4 BASIC PROGRAMS ON THE MACHINE

This section presents the basic programs that are present on each machine, i.e:

1. Control pages
2. Manual Page
3. Single Cut
4. Iso program
5. Levelling
6. Emptying (with Cutter)
7. Basic optionals
8. Polishing

For further information or if you have questions about certain parts of subsequent explanations please contact Donatoni support.

4.1 CONTROL PAGES

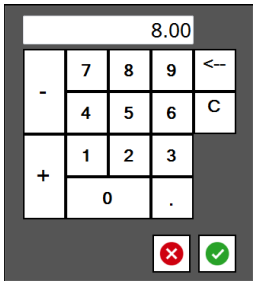
Some of the interface pages for general, support or internal software control purposes are described below.
They can be brought up from various points of the interface to make it possible to change the parameters of the installed tools or, for example, check for the presence of alarms or faults.

4.1.1 SEMI-AUTOMATIC OPERATION

The machine envisages movements in semi-automatic mode, making positioning and displacements when the desired values are entered. This displacement system is useful for making precise movements.

WARNING: *The displacement of the axis does not include the thickness/diameter of the blade. The operator must therefore take account of any increases or decreases in the values entered.*

WARNING: *The speed of the X and Y axes is controlled by their potentiometers, while the speed of the Z, C and A axes is controlled by the "interpolated" potentiometer.*

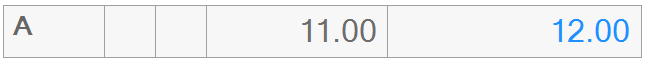
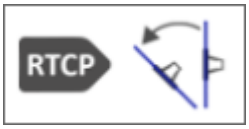
<table>
<tr><th colspan="2">Operating procedure</th></tr>
<tr><td>The operator, to set the displacement of one or more axes, must enter the value, with respect to the active origin, to be achieved in the box dedicated to each axis. To enter the values, keep the "target level" item pressed for the axis you wish to displace. A numerical keypad appears on which you can enter the new value.
 

Important! The minus sign before the number is pressed to subtract a value from the setpoint.
Example: if you are in position 10 and you enter -50 you will obtain the value
10-50 = - 40

The image to the side shows the "X" axis target level to be changed.</td><td> 
</td></tr>
<tr><td>"MDI" button: when you press this button, the machine will move to the target positions entered.</td><td> 
</td></tr>
</table>

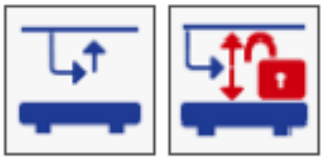
4.1.1.1 RTCP – ROTATION TOOL CENTRE POINT

The RTCP function allows the operator to make a positioning for an inclined cut while keeping the tool height unchanged even though the inclination is varied during positioning. The inside edge of the blade will remain in the same position in which it is found at the start of the operation.

Operating procedure	
Set the target level in the area dedicated to the "A" axis (inclination) or "C"	
When you press the RTCP button, the inclination or rotation function gets started while the lowest point of the tool remains fixed. **NOTE** Machine operation by the RTCP button is permitted exclusively when the coded key is fitted in the right workstation, which is displayed on the operator panel.	
Wait until the machine gets placed in the set position.	

4.1.1.2 TOOL/BENCH SAFETY LOCK/RELEASE

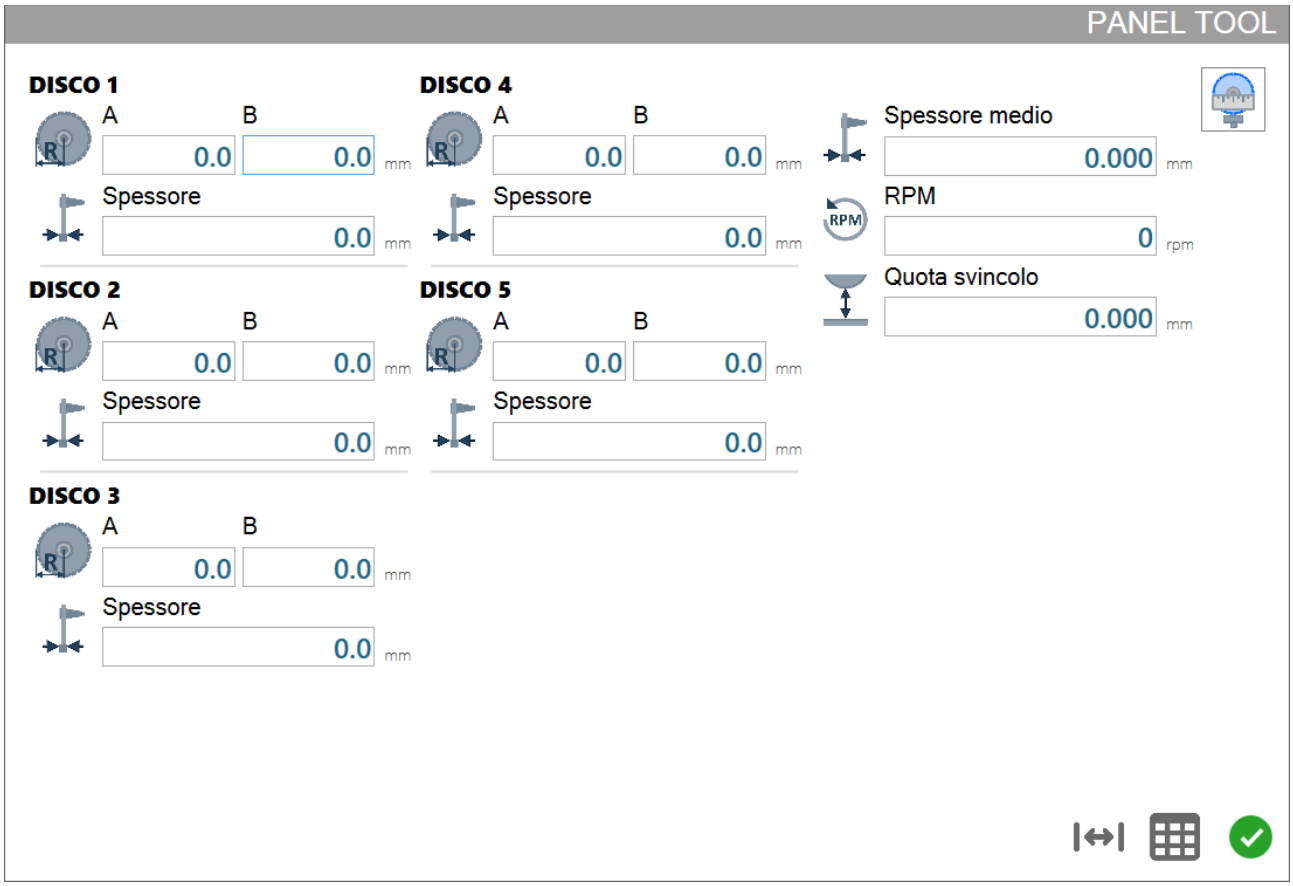
The machine envisages the tool/bench software limit switch safety function; it is necessary to set the minimum level of tool approach to the bench beyond which the machine envisages complete stopping of the axes involved in the movement.

Button Description	
Select the button  located in the upper left of the program	
The "Bench safety position Lock/Release" button (figure on the right): this activates the "Z" axis software limit switch; the lock acts indifferently to the origin for manual or semi-automatic movements only. When the button is activated, the collision control is carried out for any type of movement; both for interpolated axis movements and for single movements of the "Z" axis". The figure on the right indicates that the lock is not on: therefore the movement of the "Z" axis takes place freely without any constraint by the limit-switch; the figure to the left on the other hand indicates that the software limit switch will be respected.	

The "Set bench zero" button: This is the button for acquiring the software limit switch position. When this button is pressed, the maximum depth achievable during "Z" axis descent with the "bench lock/release" button on is set". **CAUTION:** *The position found when this button is pressed is critical during automatic cuts.*	

4.1.2 TOOL CONFIGURATION

Selecting the button  on the monitor it is possible to access the TOOL CONFIGURATION page

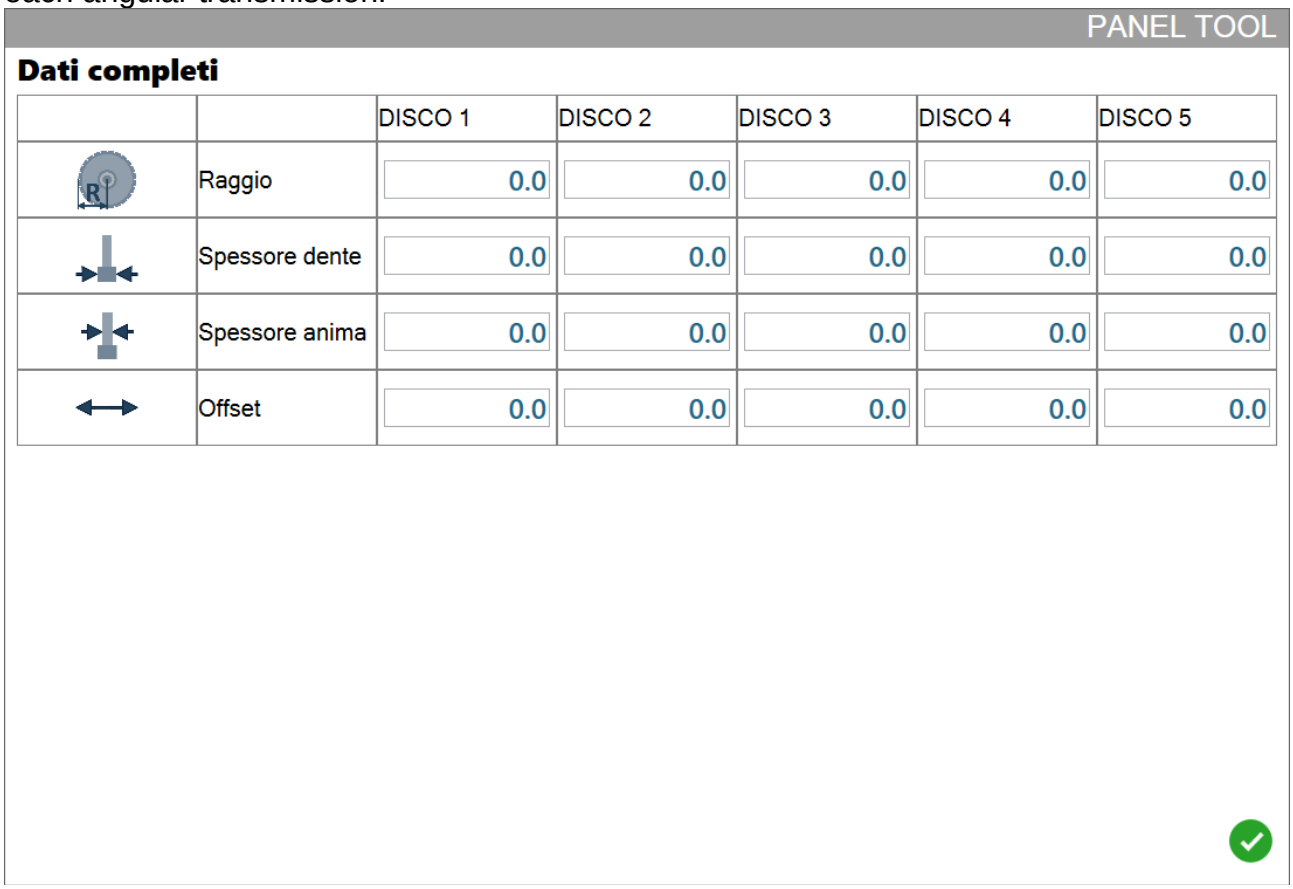


that allows the operator to set the data relating to the tools installed on the machine. These measurements are fundamental to calculate the geometries, to operate the machine in either RTCP or interpolated mode, and for the software applications installed on board the machine.

Parameter description	
BLADE THICKNESS: indicates the size of the blade thickness	
TOOL RADIUS (applies to all tools): indicates the size of the tool radius.	

4.1.2.1 COMPLETE TOOL DATA

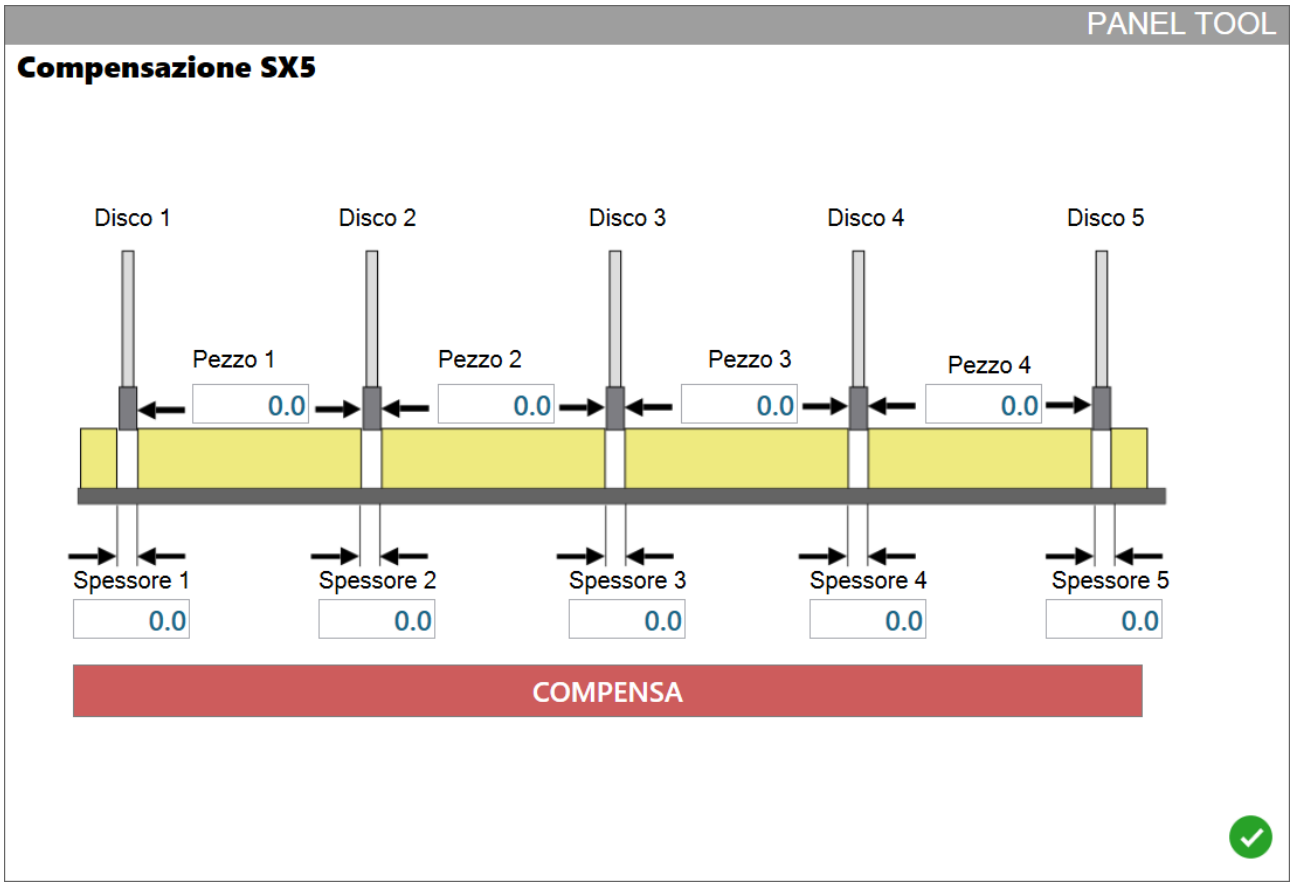
A click on the button  gives access to the page showing all the tool parameters for each angular transmission.



Description of the parameters	
TOOL RADIUS (applies to all tools): indicates the size of the tool radius.	
BLADE THICKNESS: indicates the size of the blade thickness	
CORE THICKNESS: indicates the thickness of the blade core.	
OFFSET: indicates the correction value achieved through offsetting.	

4.1.2.2 TOOL OFFSETTING

When on the TOOL CONFIGURATION page, a click on the button  gives access to the page where the angular transmission distance can be offset. This page is used to offset the distances between the various blades with a view to achieving the correct size of the final workpiece.



The user has to cut 4 workpieces (2 if it is SX3), each 300 mm large. Then the user has to measure the cut workpieces thoroughly and write the measured value in the relevant box. A click on the button “Offset” causes the software to calculate the offset value to be applied to achieve the desired size.

4.1.2.3 TOOL SYNCHRONISATION

This feature allows the synchronisation of tools, reading them from another machine connected to the network. It is particularly useful in the Office-Machine configuration.

To configure the connection, go to user parameters and press button below. The following window opens:

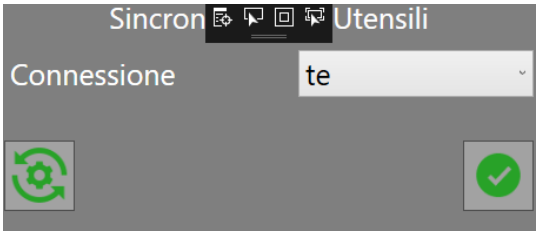
The three buttons allow:

- A connection to be added, inserting the necessary data in the relative fields
- A selected connection to be removed
- A selected connection to be modified

Fields:

- Connection: allows a saved connection to be selected
- Name: Name of connection
- IP: IP address of the machine on which Parametrix is running, and from which the tool data is to be copied. It is also possible to use the name of the computer instead of the IP address
- Port: Port number on which the Parametrix service is running (default 51333)

It is now possible to go to the tool management page and press the button  The following window opens:



In the “Connect” field, select the computer on board machine from which to copy the tools.

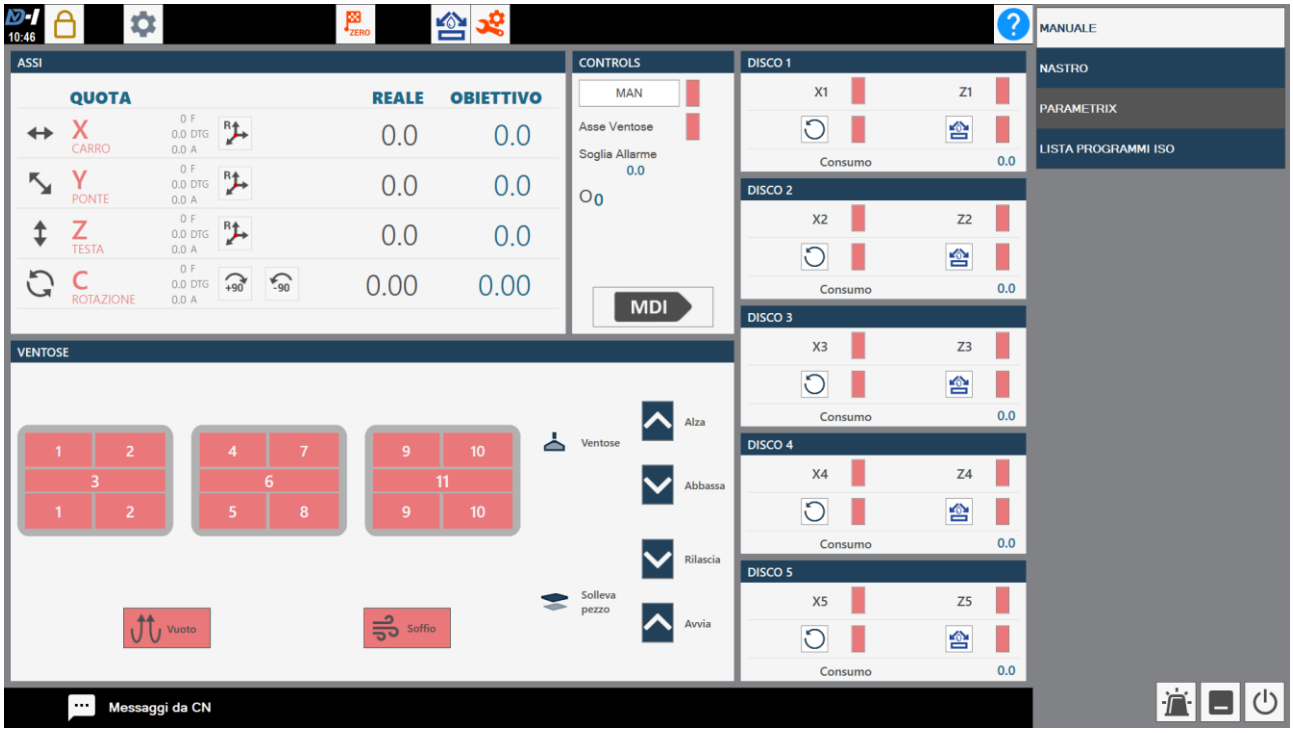
Then press the button  to perform the synchronisation cycle.

The synchronisation function works differently depending on the type of machine:

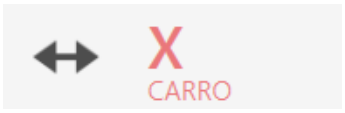
- Without Tool Change: Only the data of the three tools in use by Parametrix will be copied
- With Tool Change: All tools with a valid corrector will be copied

Caution: the procedure may overwrite one or more tools.

4.2 MANUAL PAGE

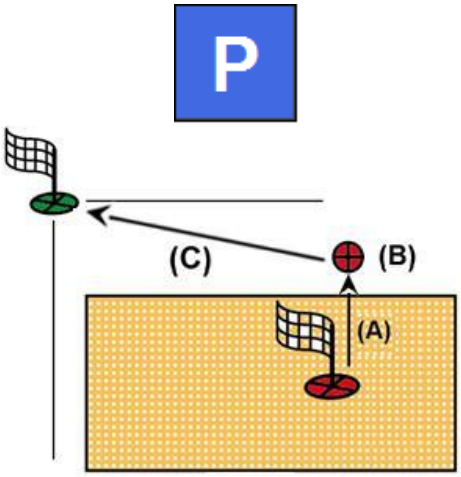


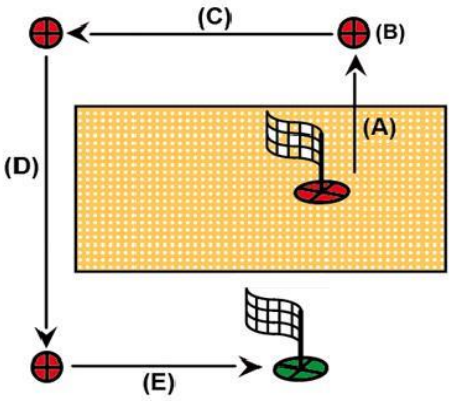
CAUTION: *Before the axis zeroing function it is essential to pay utmost attention when moving the machine. Otherwise mechanical collisions could occur*

Zero" button": when this button is pressed, the axes are reset (this operation is necessary when the machine is started or when the rectangular indicators are coloured red). When the zero set cycle is complete, the indicators will be coloured grey. The zero setting stage envisages sequential movement of the axes: 1. Z-axis 2. A axis 3. C axis 4. The X and Y Axes simultaneously 5. B axis (Optional: Controlled lathe) N.B. Zero setting is only permitted when the coded key is in the appropriate station, identifiable on the operator console.	  Axis status indicator. If red the axis is not zeroed

"AUTOMATIC PARKING" button: when this is pressed the machine automatically displaces the "X", "Y", "Z" and "C" axes to the zero position.
The displacements are made by the machine in the following order:
(A) - "Z" axis to the absolute zero position;
(B) - "C" axis to the absolute zero position;
(C) – The "X" Axis and "Y" Axis simultaneously to the absolute zero position;

WARNING: *Several safety conditions must be respected in order to carry out the automatic parking:*
- The position of the "A" Axis must be at 0°.
- Check that the movement area involved in positioning (see the figure to the side) is free.



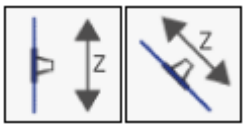
"MANUAL TOOL CHANGE" button: when this is pressed the machine completes a displacement in automatic mode in the relevant manual tool change area. **WARNING:** *Several safety conditions must be respected in order to carry out the manual tool change:* *- The position of the "A" Axis must be at 0°.* *- Check that the movement area affected by the positioning (see figure to the side) is free.*	 
"Water On/off" button": pressing this button activates the main water pipe. The indication above the button changes colour: Green = On; Red = Off.	
The "0/90 degree automatic rotation" button: the "C" axis is automatically moved to 0 or 90 degrees when the button is pressed.	
The "Tool Parameters" button: pressing this button offers access to the tool parameters page. Refer to the chapter "Tool Configuration" for a detailed description of the parameters page.	
Actual level: this indicates the position of the axis in relation to the active origin. *Target level*: this indicates the level to be set in order to move the axis by pressing either the MDI or RTCP button.	

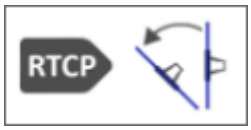
<table>
<tr><td>“Tool active” button: a click on this button causes the tool currently installed on the machine to be selected. The button has 2 states:
1. Blade
2. Cutter or hollow bit</td><td>  </td></tr>
<tr><td>The "Origin" field allows selection of the active origin to work with (in the example to the side the origin number 10).</td><td> 
</td></tr>
<tr><td>The "Axis Reset" button: pressing this button sets the zero of the active origin at the point in which the relative axis is found (X, Y or Z).
The axis reset operation can only be carried out on origins other than 0 (zero).

In order to use the tool coordinate (zero) point (TCP), the correct measurements of the installed tool must have been set before pressing the Axis Reset button.</td><td> 
</td></tr>
<tr><td>“Interpolation mode” button: when this button is enabled (figure on right-hand side), the axes Z, C and A are moved manually considering the rotation and inclination values.

Movement of interpolated Z axis: by actioning the JOG of the Z axis the machine will move aligned to the tool fitted.
N.B. the type of tool selected has an effect on the direction of movement.
Movement of interpolated C axis: when you activate the C axis command, the X and Y axis will also move as well as the C axis itself, in such a way that the inside edge of the blade (or the tip of the tool) remains stationary.
Movement of interpolated A axis: when you activate the A axis command, the X, Y and Z axis will also be moved as well as the A axis itself, in such a way that the inside edge of the blade (or the tip of the tool) remains stationary in the position it finds itself in.

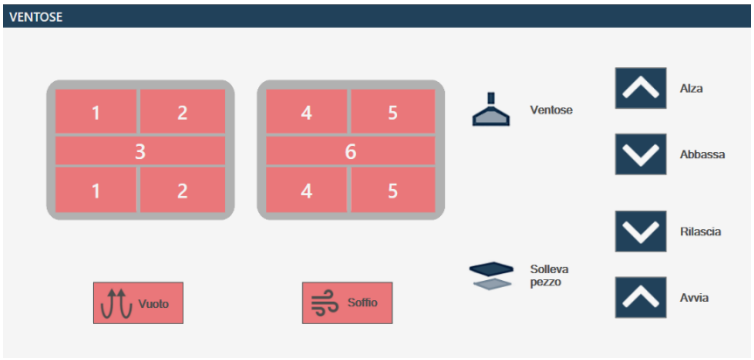
Simultaneous movements of the Z, C and A axes are not permitted with the interpolated mode active.

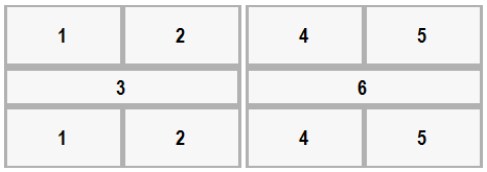
WARNING: Pay particular attention to the state of the button as the machine movements may be different from expected.</td><td> 
</td></tr>
<tr><td>“Automatic +90 degree rotation”: the position of axis “C” is increased by 90 degrees clockwise.
Example: the machine is at 12 degrees, pressing the "Automatic +90° automatic rotation" button the machine rotates to 102 degrees: 12 + 90</td><td> 
</td></tr>
</table>

“Automatic -90 degree rotation”: the position of axis “C” is decreased by 90 degrees anticlockwise. Example: the machine is at 102 degrees, pressing the "Automatic -90° automatic rotation" button the machine rotates to 12 degrees: 102 - 90	
RTCP interpolation button. This carries out a positioning to the A or C level set in the "Target " while maintaining the inside edge of the blade (or the tool tip) stationary. For further specifications see paragraph: “*RTCP semi-automatic functionality Rotation Tool Centre Point*”	
"MDI" button: when pressed, the machine performs movement until reaching the levels set in the target levels (see above). The speed of the X and Y axes is controlled by their potentiometers, while the speed of the other axes is controlled by the interpolated potentiometer.	

4.2.1 MANUAL CONTROL OF SUCTION CUPS (OPTIONAL)

It is possible to use the suction cups in manual mode to move the workpieces from one work area to another.



Button Description	
Enabling/disabling of suction cups. The screen shows the suction cup group seen from above in the various separate areas. Pressing on the relevant area changes the status. If green this means that the area is prepared for vacuum mode. If red this means that the area is prepared for blowing.	
Park suction cups. When the button is pressed, the suction cups are brought in to take them into the rest area/parking area.	 Up
Lower suction cups. When the button is pressed down the suction cups descend from the parking area as far as the limit switch. It is important that the suction cups have the space necessary to completely exit.	 Down
Workpiece unloading. Pressing the button lowers the Z axis until contact of the material with the work surface; the vacuum pump is turned off and the blower from all the suction cups is turned on to detach the workpiece. After a timeout wait, the axis is lifted to a safety height.	 Release

Button Description	
Workpiece lifting. With this command the suction cups are brought into contact with the material, the vacuum pump is turned on in the areas marked in green and the blower in the areas marked in red, and when the vacuum switch indicates that the vacuum was generated, the axis is raised to the position set for handling. **WARNING:** *Pay attention to which suction cups are active in order to use the largest possible area to lift the workpiece.*	 Start
Blower control. The button turns the blower from the suction cups that are not prepared for vacuum on or off. The indicator shows if the blower is on (green) or off (red).	 Blow
Vacuum pump control. This button switches the vacuum pump on and off. The indicator shows if the pump is on (green) or off (red).	 Vacuum

Safety Conditions

Pay particular attention to the active suction cups during lifting, ensuring that the areas used are sufficient to lift the workpiece being moved. Try to always use the greatest vacuum area available for gripping of the workpiece and make sure there are no defects or breakages on the material which could create hazards or problems in operation of the machine.

Having turned on the vacuum pump, its switching off is only possible through the automatic functions provided for this purpose. Pressing the reset button or the emergency button has no effect on the vacuum pump which will continue to operate. Switching off the pump can be forced by using the relevant button in the "Additional commands" page.

With the automatic procedures, before lifting the workpiece, the status of vacuum is checked through a dedicated sensor: if a sufficient vacuum value is not detected the "no vacuum" alarm appears. The pump will continue to remain active until the vacuum switch detects a correct value, or if will be disconnected from the "Additional commands" page.

CAUTION: *any energy failure to the vacuum pump will cause a rapid loss of vacuum and, as a result, falling of the material lifted by the suction cups.*

4.2.2 ALARMS

Pressing the alarm button  it is possible to open the page to manage and view the alarms.

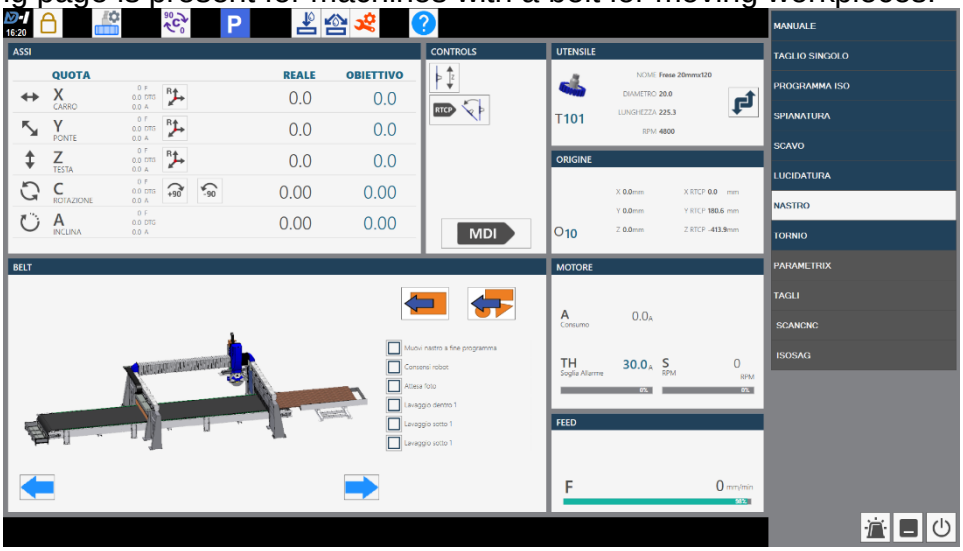
The alarms page indicates the Alarms/Messages that are active in the system.
If no alarms are active in the machine, the page will appear empty otherwise there will be a line for each error present. This line consists of: error number, a brief description, the date and time of the event.

Function	
The "Refresh" button is for updating the machine alarm status.	
"Delete" is for eliminating the list of historical alarms present.	
This shows a historical report of the alarms signalled by the machine. It is useful for the engineers from the Donatoni Engineering Workshop to trace a timeline of the machine's operation.	
The "Save" button saves the alarms on disk in the situation in which it is pressed.	
It allows choosing and displaying the alarms or messages.	

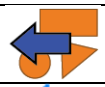
To exit the page, select an item from the list displayed on the right side of this page.

4.3 CONVEYOR BELT

The following page is present for machines with a belt for moving workpieces:



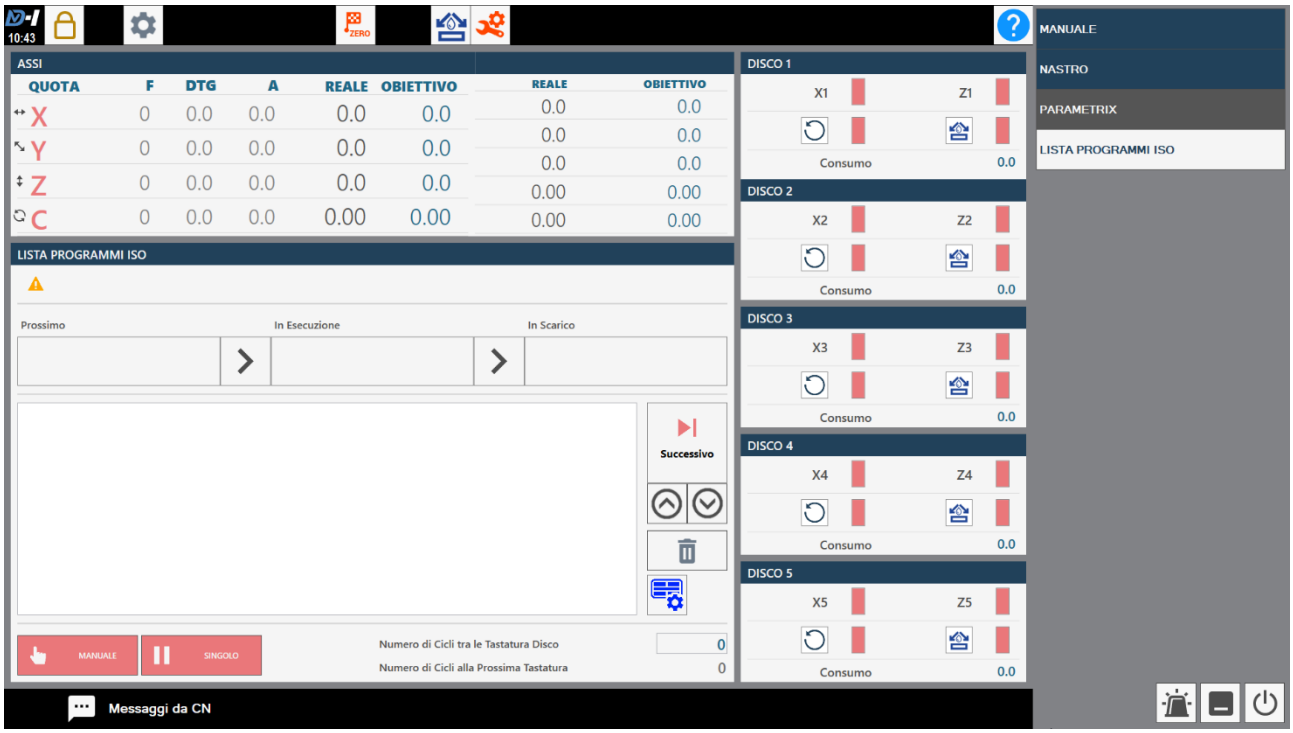
The control buttons contained in the page are listed below:

	Button for loading slabs
	Button for unloading pieces
	Manual movement of the belt to the left
	Manual movement of the belt to the right

The page also contains flags that can be ticked to select machine characteristic options.

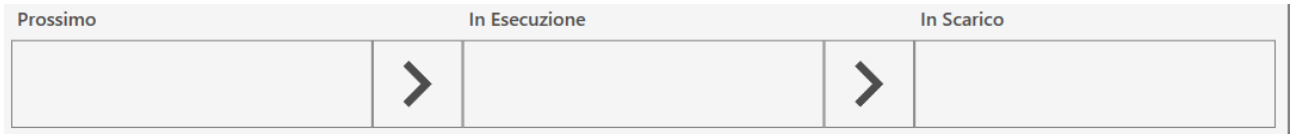
4.4 LIST ISO PROGRAMS

The page below is provided for program execution:



The programs the user has created are displayed in an execution list.

The user can see the program currently executed (the program the machine is running at present), the previous program, and the future program to be executed.



Description of buttons	
Next program This button is pressed to drop the ongoing program and to upload the next program.	
Up/Down These buttons are pressed to sort the programs in the execution list.	

Delete program This button is pressed to delete a program from the execution list.	
Suction cups This button is pressed to open the screen where the suction cup parameters are managed.	
Manual/Automatic When the machine switches to automatic operation, this button turns green, thus indicating that the selected program can be executed.	
Single program/Next in automatic If the button is red, a single program is executed. If the button is green, the next program is automatically executed at the end of the ongoing program.	
Automatic blade gauging It is used to set the number of programs that need to be executed before a gauging cycle, which is required to measure the blade diameter.	

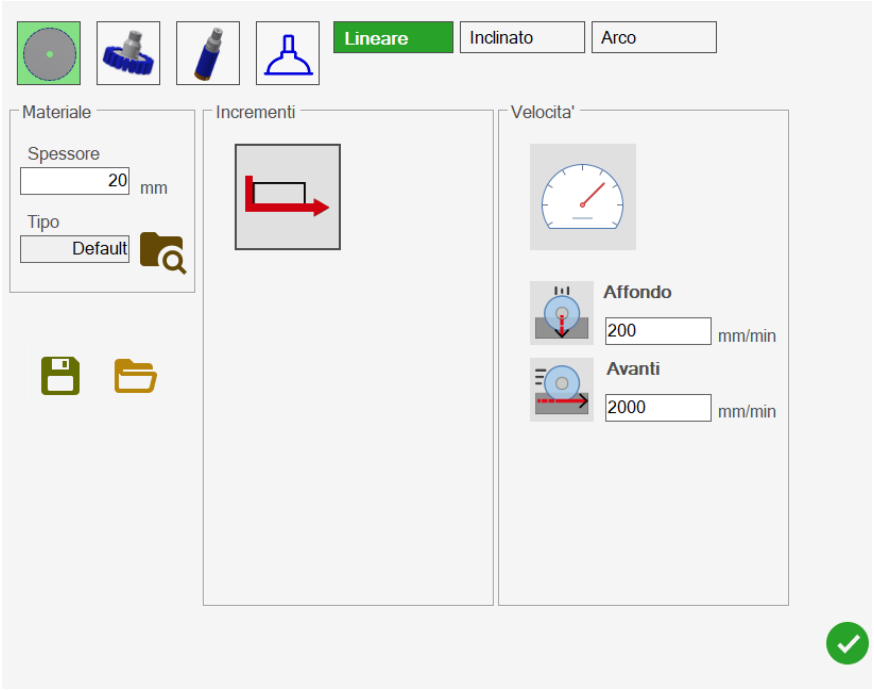
5 PARAMETRIX

This system was designed to assist the operator in programming automatic cuts. The software helps:

- determine the position of the slab in the working area of the machine and create the material machining perimeter;
- Create workpieces through parametric figures
- Import figures made with other CAD systems
- Generate a list of workpieces
- Identify the best position of the pieces on the slab
- Set special cutting functions
- Program the cutting order and create the machine movement
- Monitor on the screen the execution of cuts

5.1.1 MACHINING PARAMETERS

The machining parameters screen offers the operator management of the data relating to the machining and it is possible to view it by selecting the button  .



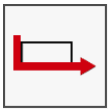
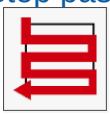
The machining data can be set and modified on the parameter panel.
The editable parameters refer to the blade, the milling cutter and the core bit, if the user is enabled to use them. As for the suction cup parameters, only speed can be edited.

5.1.1.1 PARAMETER DESCRIPTION

Except for the hollow bit in relation to which only the penetration speed can be set, the blade and the cutter have different machining and speed modes. In fact, it is possible to decide whether to perform *full pass*or *Step machining.*

To choose which type of working mode to select, it is necessary to select the image visible under the increments item, i.e.: 

The image varies and each one has its meaning:

Blade	Full Pass 	Step pass 

5.1.1.2 PASSES

Full Pass
The Full Pass includes all the machining in a single pass. The tool therefore cuts through the whole material from the very beginning of the cycle and performs the machining process.

Step pass
With this type of passes the tool does not cut through the material immediately, but gradually.
This involves the performance of manufacturing multiple times at different heights up to the bench height (end of material) or to the decided height.

5.1.1.3 INCREMENT PARAMETERS

The increments section is only used by *Step* or *Spiral*machining and indicates the various penetration to be maintained during execution of the work.

Description of the parameters	
Indicates the depth of the cut on the material when the machine cuts in the direction of blade rotation	FORWARD
Indicates the depth of the cut on the material during the cutting stage in the opposite direction to that of blade rotation	BACK
This indicates the cut depth of the last tool pass through the material. This cut is always in the direction opposite to the blade movement.	LAST

5.1.1.4 SPEED PARAMETERS

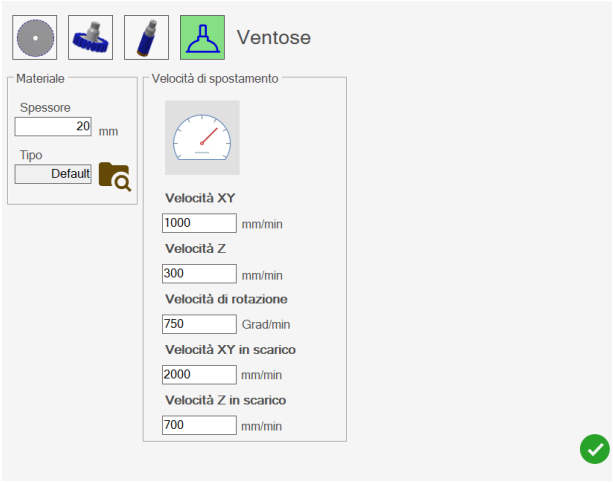
The speed section is used by all types of pass excluding some in case of full pass.

Description of the parameters	
This determines the speed during penetration of the tool into the material. This value is expressed in millimetres/minute. It is used by all types of passes	BREAK
This indicates the desired speed for the forward cutting stage. This value is expressed in millimetres/minute. It is used by all types of passes	FORWARD
This indicates the desired speed for the return cutting stage. This value is expressed in millimetres/minute. Only used only by Step or Spiral passes.	BACK
This indicates the speed that the machine must have for the last pass cut. This value is expressed in millimetres/minute. Only used only by Step or Spiral passes.	LAST

Note	Regarding the increments and speeds the "Back" and "Last" parameters may be present or not or only one of the two. Normally these choices are made at the time of installation of the program and are decided according to the customer's machining. It is possible to change these options. For other work modes, contact Donatoni Support.

5.1.1.5 SUCTION CUP SPEED PARAMETERS

The suction cup speed section sets the speed of workpiece movement with the suction cups.

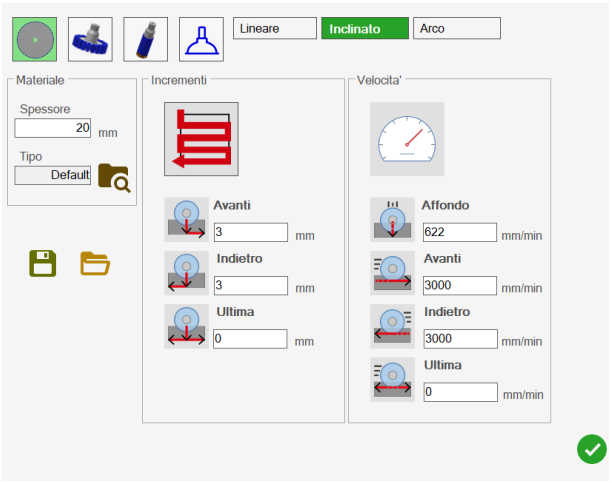
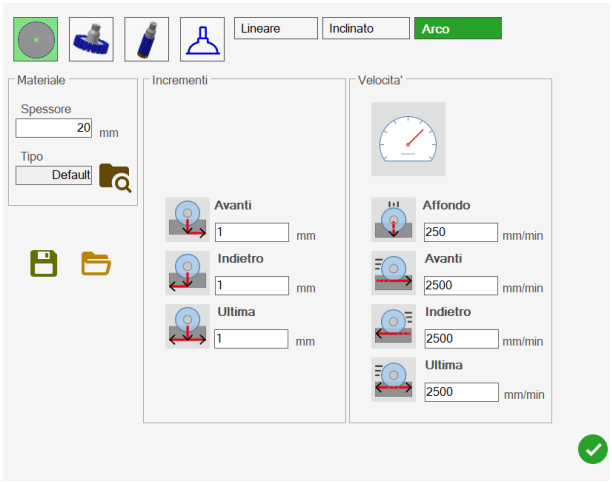


Description of the parameters	
Determines the speed of workpiece movement with the suction cups. This value is expressed in millimetres/minute.	XY Speed
Determines the speed of workpiece movement with the suction cups in Z. This value is expressed in millimetres/minute.	Z Speed
Determines the speed of workpiece rotation with the suction cups. This value is expressed in degrees/minute.	Rotation speed
Determines the speed of workpiece movement with the suction cups during the Piece Offload or Cut Below stage. This value is expressed in millimetres/minute.	XY Speed while unloading
Determines the speed of workpiece movement with the suction cups in Z during the Piece Offload or Cut Below stage. This value is expressed in millimetres/minute.	Z Speed while unloading

Note	With regard to the “XY Speed while unloading” and “Z Speed while unloading”, these may or may not be present. Normally these choices are made at the time of installation of the program and are decided according to the customer's machining. It is possible to change these options. For other work modes, contact Donatoni Support.

5.1.1.6 BLADE OPTIONS

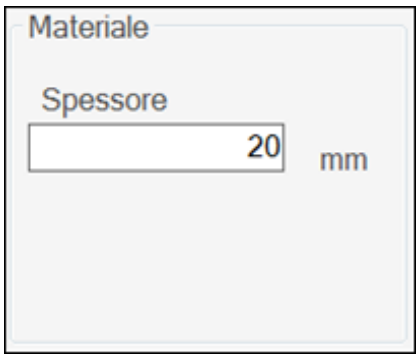
For the blade it is possible to also set the machining properties for certain cut types. Depending on the type of machining being performed, it will use the parameters present in the respective type. If for example the blade must make an inclined cut, it will use the **increment** and **speed** parameters present in the “inclined” section.

Inclined Mode (Axis A > 0)	Arc Mode
	

The Arc mode is always performed in pass mode.
As can be seen from the images above the various types may have different increments and speeds.

5.1.1.7 MATERIAL

Within this screen it is also possible to change the thickness of the material

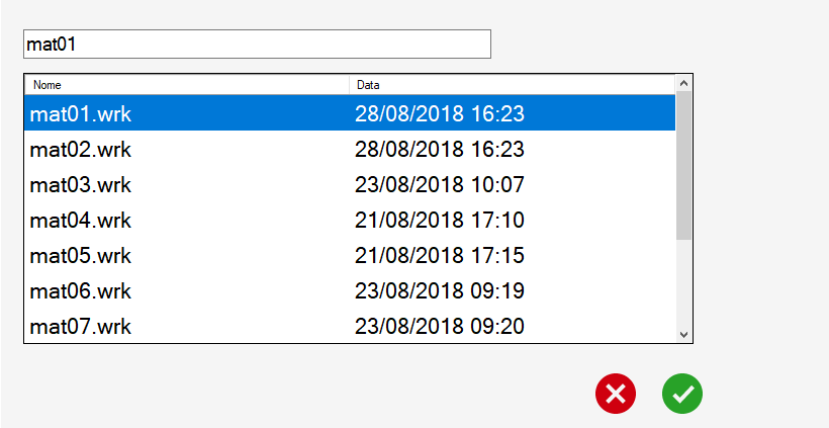


The value entered in this section is essential for all the calculations performed by the program.

5.1.1.8 OPEN AND CLOSE

It is possible to save the machining parameters entered to then use them again during subsequent machining.
In fact, it is probable that these parameters may be modified constantly, changing the material to be processed.
Using the save button allows saving of this configuration and, with the Open button to use it at another time

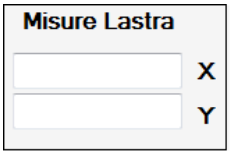
When the button  the following panel will appear:



Insert the file save name at the top, or select an existing file to overwrite it.

5.1.2 SLAB ACQUISITION

To introduce the workpieces onto the slab, it is necessary to know the location of the material on the workbench. Acquisition of the slab can be performed with 4 systems:

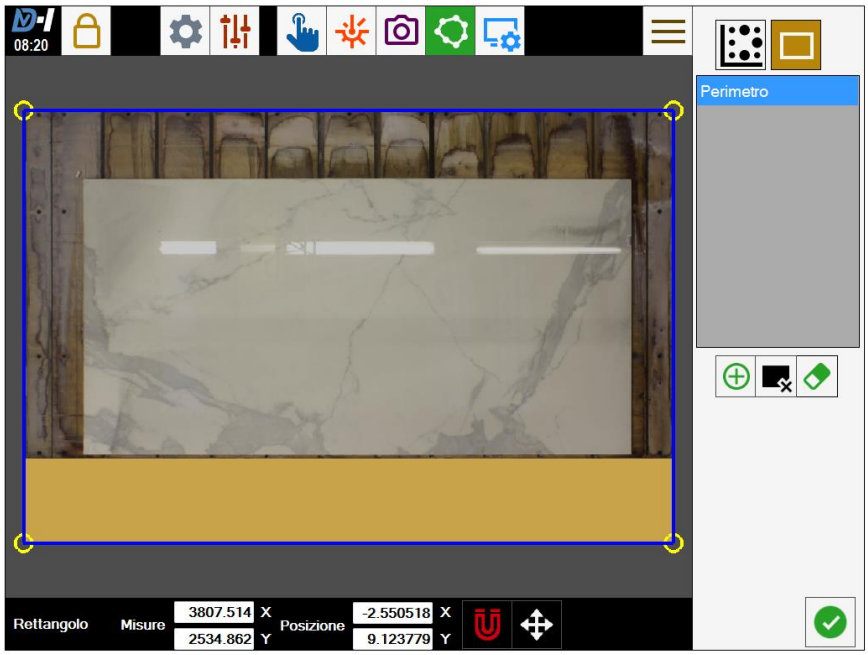
1. Photography (Optional) 
2. Cutting laser 
3. Cross laser 
4. Slab width and height measurements 

The photo shows the user the material with defects, if any. The laser system, on the other hand, does not make this check possible. The slab acquisition systems are analysed below and a description of their operation is provided.

By measurements it is meant creating the perimeter material by inserting the measurement of the Width and Height of the slab. To set the sizes, press the material perimeter button  and in the data entry panel it is possible to enter the X and Y values.

5.1.2.1 PHOTOGRAPHY (OPTIONAL)

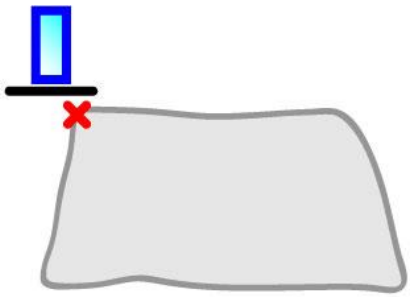
To upload a photo within the program, press the button  . If the software is connected to a photographic device, a photo is taken of the slab otherwise there will be a prompt to select a photograph from the PC.
The photograph is uploaded into the work area and there will be a prompt for the perimeter. When the image is ready, it is possible to position the workpieces inside the perimeter.



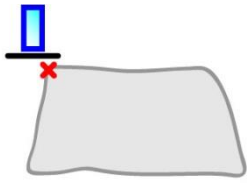
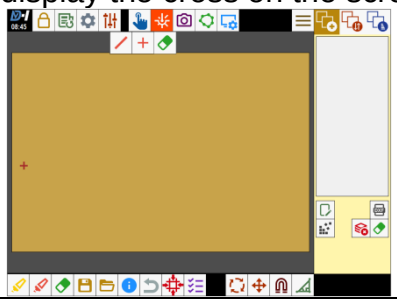
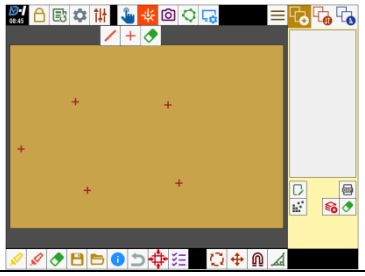
It is possible to assign an **area to be avoided** (causing cracks on the material, unsightly veins or other issues) in which the program prevents insertion of the pieces. To create it, it is necessary to select the button  under the table.
This area can be inserted in two ways: **by points** or **using "Rectangle" mode,** as is the case when the perimeter is created.
Similarly to perimeter creation, elements can be moved, vertices added at a later stage, etc., as explained in the section titled *"Material perimeter".*

5.1.2.2 CROSS-SHAPED LASER POINTER

To detect the position of the slab with the cross laser (see illustrative photo) move the laser indicator on the perimeter of the slab and digitise the position thereby obtaining the coordinate of the laser on the monitor.



A tap on  causes the software to detect the position of the laser cross and to reproduce it on the work area. The position of the slab is acknowledged using multiple points. The procedure is illustrated below.

With the button  on the home page of the program enable the slab acquisition menu.	 Initial page buttons  Slab acquisition menu
Move the laser pointer onto the vertex top of the slab. 	Press the laser position acquisition button  to display the cross on the screen. 
Example of slab perimeter acquisition with the laser. 	***NOTE:*** *There are no point limits to be used for acquisition; the more laser positions acquired, the more accurate the perimeter will be.*

5.1.3 MATERIAL PERIMETER

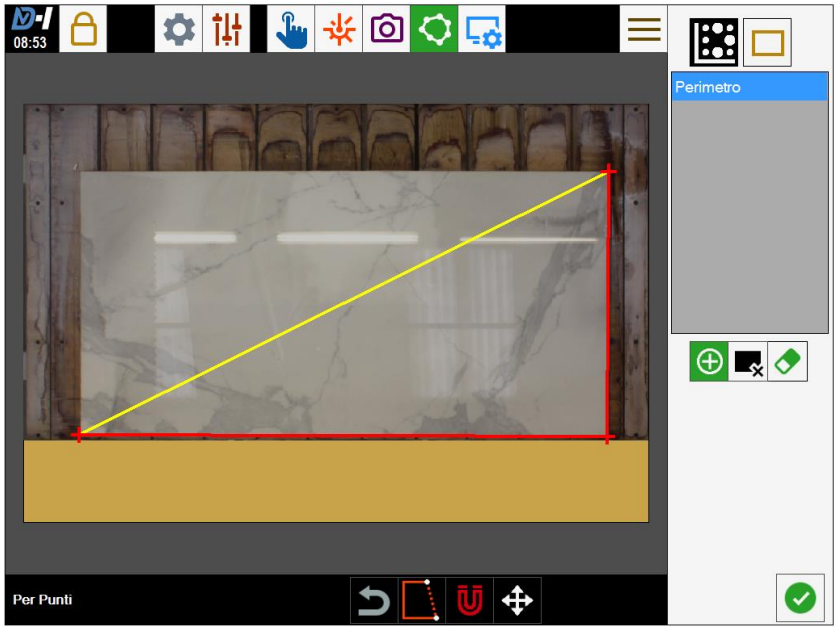
The program must recognise the edge of the slab. Selecting  it is possible to choose how to create the perimeter:

1. **For points**: It allows drawing of the points that form the perimeter in the desired locations
2. **For rectangle**: A rectangular shape appears on the screen and it will be necessary to move the vertices into the desired position.

For points

Having selected the mode for points  and pressed the button  simply select the points required on the work area to begin creating the perimeter (Figure below).

Having taken all the necessary points, pressing  the perimeter will be closed.
If this needs to be modified, or points need to be added or deleted, simply press the button  , to open the selected perimeter and to change the shape of the latter. Continuing to press it deletes in reverse the previously created points.



Both during creation and in the presence of a closed perimeter, it will always be possible to move the vertices to improve the precision of the perimeter if necessary. Simply select the vertex and drag it to the desired position.
If in the presence of "*Laser Crosses*" on the work area to give our perimeter the vertices exactly above these crosses it will be necessary to press the button  . When this button is enabled and in the process of being created, by drawing the vertex of the perimeter near the cross, the latter will automatically be placed on top of it. If the perimeter is closed and this mode is enabled, all that is required is to drag the vertex, highlighted with a yellow circle, near the cross and the program will automatically place the vertex in the desired position.

Using Rectangle mode

When the button is pressed,  "Rectangle" mode  will create a rectangular area having the size and position specified in the bar below.

Also for this mode, it will be possible to move the vertices to the position of the cross lasers by selecting the button  .

Special features common to the two modes:

1. It is possible to add as many perimeters as required by selecting the button 
2. It is possible to move the entire figure to a new position by pressing the button 
3. Switching from the mode **for points** with a perimeter selected for **rectangle** mode there will be a prompt to transform the perimeter into a rectangle. Either proceed or remain in the section for points.
4. The perimeter/area to be avoided presents blue outlines and yellow vertices; those instead that are not selected are green (perimeter) or orange (area to be avoided).
5. To eliminate a perimeter/area to be avoided, press the button  below the table and select the figure to be deleted on the work area or directly from the table
6. To select the perimeter, simply select it on the work area or in the table
7. It is not possible to have two perimeters on top of each other

5.1.3.1 MAPASCAN INTEGRATION

The Mapascan software runs on a machine called scanner that scans the slab and produces a file.
Parametrix is able to read this file and import the photo of the slab, its perimeter and thickness.

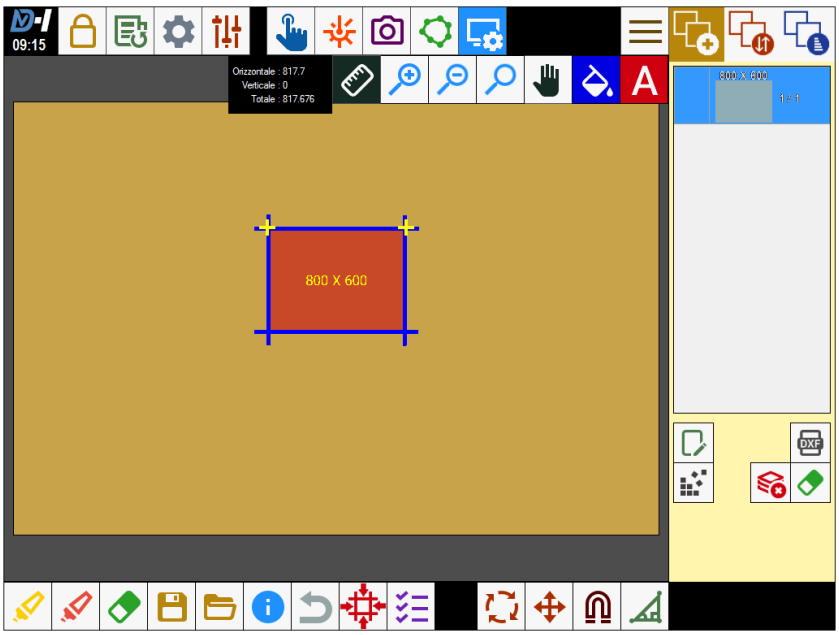
5.1.4 PROGRAM UTILITIES

At the top of Parametrix it is possible to note the following button  . Pressing it, the following panel appears:

The buttons are used to show some information or utilities while the software is being used. The panel stays visible until the button is pressed again.  .
As a basic option or if all functions are disabled, the Zoom function is considered active.

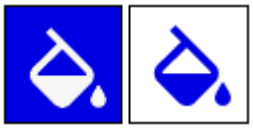
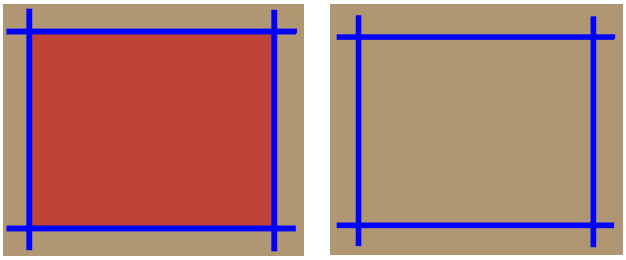
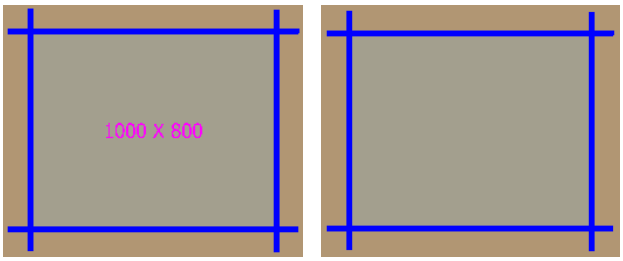
5.1.4.1 METRO

The metro function is used to measure approximately the distance between two points on the work area  .
To use it, it is necessary to press the button, specify the first point (a yellow cross will be inserted) and the second point (also indicated with a yellow cross).



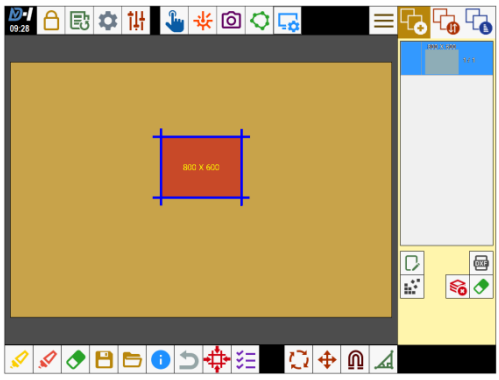
Under the "Metro" button appears a small panel that shows the distance between the two points

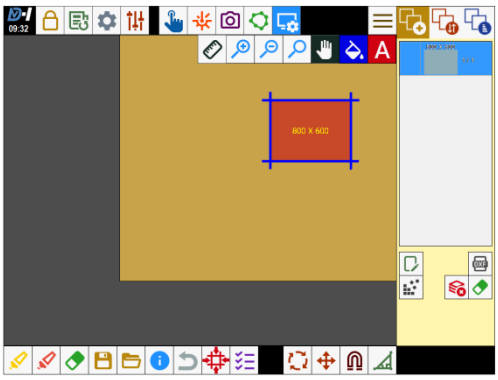
5.1.4.2 LENS AND WORKPIECE VIEW OPTIONS

	These three buttons are respectively used to: expand, reduce the work area, or resize it to its initial size.
“Background” enabled / disabled button 	
"Name" button enabled / disabled 	

5.1.4.3 MOVEMENT

Selecting the button  will make it possible to move the work area with all the workpieces on it.





This function is very helpful when combined with the Zoom function illustrated in the previous section as, when the size is remarkably expanded (instance), some workpieces in the work area may not be visible. This function helps move the area and bring the workpiece back into it to view it.

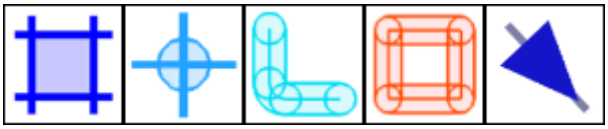
5.1.4.4 MANUAL BUTTONS IN PARAMETRIX

At the top of Parametrix it is possible to note the following button  . Pressing it, the following panel appears:

These buttons allow some movements of the machine directly from Parametrix. The features that can currently be used, visible in the image above, are: "Rotation of the head by 90°", "to bring the machine into Parking", "Exchange of Benches". The last button is only visible if the machine has been provided with several benches.

5.1.4.5 BUTTONS TO HIDE ENTITIES

In the top part of Parametrix, pressing of the button  evokes the following panel which is used to enable and disable viewing of some entities in the work area. This notably includes cuts, holes, milled and lowered parts in the first tab plus arrows in the third tab only.



The buttons indicate respectively: hide cuts, hide holes, hide milling, hide lowering, hide arrows. The "hidden" modes cannot be selected, thus making it easier to manage the selection in the case of overlapping entities.

5.1.5 WORKPIECE CREATION

Parametrix is designed to machine flat and closed geometrical figures. There are 6 possible ways to create a workpiece.

1 - Using an external Cad software and importing a DXF file	
2 - Insertion of Rectangles	
3 - Parametric Figures	
4 - Excel	

5 - History	
6 - Favourites	

When a workpiece is either created or imported into the program, it is included in a list of workpieces. An explanation of how the list works will be given later on in this manual.

5.1.5.1 DXF IMPORTING

Note	• *To understand the chapter it is necessary to have a basic knowledge of technical drawing.*
	• *The drawing software requirements include:*
	  ○ *export of DXF file;*
	  ○ *polygons exported in polylines: they must not be exploded in the export file.*

The figures the machine will be required to cut can be created with a technical drawing software. The drawing must conform to the following characteristics:

- The workpieces must be closed polygons.
- The design must be free from scroll or similar as these could interfere with the importation and scanning of the workpieces by Parametrix.
- Where labels need to be added to the workpiece, the lettering must be within the perimeter of the workpiece.

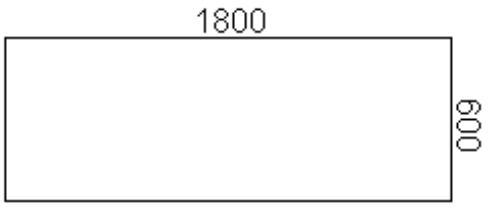
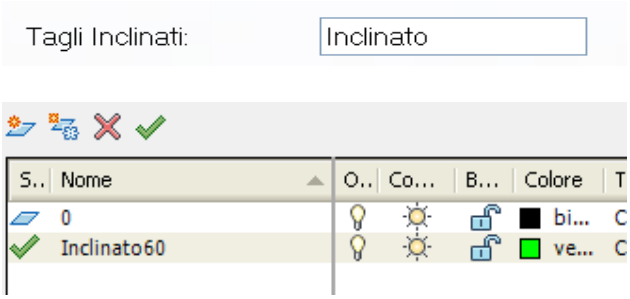
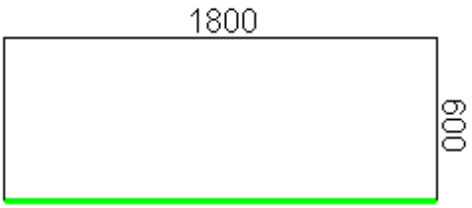
Furthermore, at the time of design it is possible to indicate certain types of special machining that will be recognised automatically by the software. The list of machining options are:

- inclined cuts;
- Raiser
- Backsplash
- Offset
- Drip
- Milling
- Lowering

Parametrix uses the name of the layer for special functions. The name of the layer associated to the machining type can be edited on the parameters page of the software. On the drawing, special machining processes are established by creating a layer with the same name as the parameter, for example:

Parametrix	Technical drawing
Name: backsplash machining: *Rive_par*	Creation of a new layer with name: *Rive_par*
Result: Backsplash machining will be assigned when the Rive-par. layer is used.	

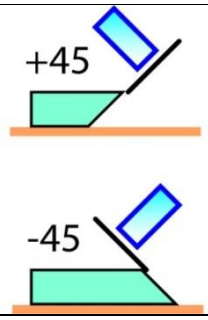
To apply the machining to a piece, firstly draw a line over the cut to be modified. The applied line must be part of the adapted layer. Here is an example of programming of an inclined cut on a rectangle.

1. Creating a polyline with the sizes and shape of the piece to be worked.	
2. In Parametrix the Layer name for inclined cuts is set to *Inclined* On the drawing create a layer and call it *Inclined60* in order to cut with the axis A = 60°	
3. Activate the layer *Inclined60* and draw a line over the side to be 60° **NOTE:** The colour of the layer does not affect the machining, Parametrix only takes into account the name	
4. Save the drawing in DXF format and the amount within the program. Upon reading the drawing, the program automatically recognises the machining on the cut with a 60° angle	

Below are listed the special types of machining with the necessary characteristics:

- *Inclined cuts:* to cut one side with the A axis of the machine other than 0. The inclination is expressed in degrees and must be entered in the layer name: **nameLayer+Degrees** (see example).

Inclination direction:	**Example: obtain the machining with axis A = 30°** On Parametrix – Inclined Cuts: Incline

	On the program create a new layer: Incline30

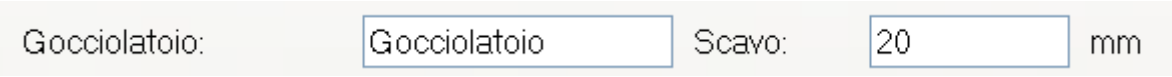
- *Raiser*: to add a rectangular workpiece, the length of which is equal to the cut. The width of the rectangle must be written in the name of the layer: **nameLayer+mmWidth**.

Incline the A axis on which the function is inserted and the relevant side of the rectangle created. The degrees of the axis A should be set in the parameters, under the item Degrees *(*see figure below).

- *Backsplash*: as for "Raiser" add a rectangular workpiece the length of which is equal to the actual cut. The width of the rectangle must be written in the name of the layer: **nameLayer+mmWidth** the difference lies in the common cut which in this case is not inclined.

- *Offset:* to create a shift of the cut and to increase the size of the workpiece by the number of millimetres shown on the layer name. The name must be: **nameLayer+mmOffset**.

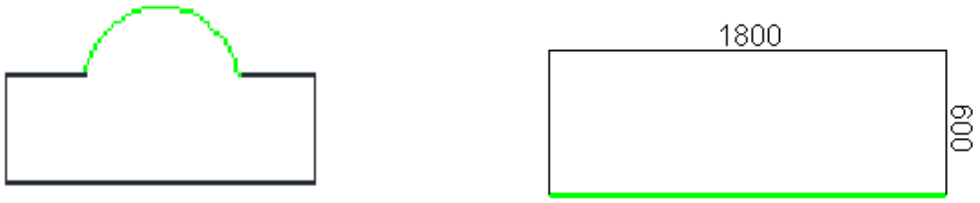
- *Drainer:* the Drainer function is designed to enter a cut with a greater width than the thickness of the blade and a smaller depth than the thickness of the material. The depth of the cut is set in the parameters under the item 'Emptying' (*see figure below*), while the width is indicated inside the layer name **nameLayer+mmWidth.**



- *Milling*: Milling is a particular feature in that it can completely replace a blade cut or can be added to finish certain processes performed with it.

To insert a cut with the cutter, firstly create a Layer with the same name present in Parametrix (under the milling item) and move the corresponding entity (line with straight side, arc with curve, ...) over the side to be milled. If the figure added completely covers the side, Parametrix will only use the cutter to cut this part of the workpiece, otherwise on the same side there will be both cut with blade and cuts with Cutter.
E.g. DXF where the milling (selected in green) fully covers the side of the workpiece. When they are imported into Parametrix, only the cut with the cutter will be applied on those two sides.

1. 2.



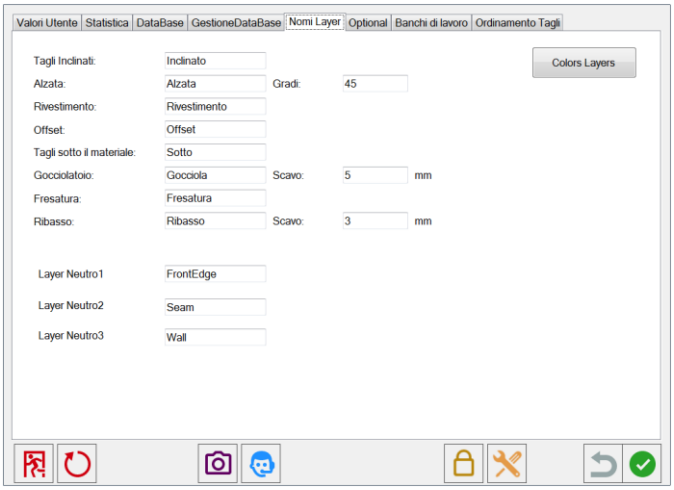
E.g. DXF with milling that does not replace the cut with the blade but it finishes the machining.

Note	For cuts with the cutter, which do not replace cutting with the blade but assist it, the length with which they are drawn in the DXF file remain for the entire machining process, meaning they will not receive lengthening or shortening following the Paramterix operations. The only thing that changes is the thickness of the cut as it depends on the installed tool.

- ***Lowering:*** the Lowering function is mainly used to lower the holes in kitchen tops by a preset amount in order to make room for sink installation. To create the lowering in the DXF file, the name of the layer must be as follows: **nameLayer+mmWidth**, where the **nameLayer** is equal to the name in Parametrix (in the Lowering item). With reference to "**mmWidth**", value 0 is used by default if it is not included in the name. Value 0 means that the milling cutter centre is right above the side, in other words half of the cutter will cut through the material.

For a Lowering cut to be correct, all the sides of a polygon must be above the Layers of the Lowering, as the lowering is only applicable to the perimeter of a polygon.
The depth of the cut is set in the parameters page in Parametrix:

To change the "nameLayer" used to assign the machining processes, press the button  and select the "Layer Names" window where the machining processes are listed and the corresponding names assigned.



"Colour layers" help view certain coloured sides of the workpiece in a different way: Layer colours do not modify the machining processes of the side, but identify the side on screen. To set the name of the layer and to link it to the drawing fill in the **Layer change colour table**

Note	*The identifying name of the layer colour should not have numbers or points but only alphabetic characters*

The "neutral layers" allow the entering of three layer names that will be considered neutral. That is, everything that is put on neutral layers Parametrix ignores it.

Core bit

Another particular function is cutting with a core bit. This function is normally used to finish certain machining operations (in particular points of the workpiece), for instance at corners the blade cannot reach as it would damage the workpiece (*blade avoidance area*), or to create a hole in the material.
This function works differently from the others as it does not need a particular layer, but it can be created directly on the main layer.
To draw a hole that finishes the corners of the workpiece (where the blade cannot cut) it is necessary to draw inside the DXF file a circle at this angle, making sure that the circle intersects the angle in question.
On the other hand, if you wish to use a core bit to cut a circle inside the material, simply draw this element inside the workpiece and check that it does not intersect with any side.
In both cases, the software recognises that the cut must be performed with the core bit tool, rather than with the blade tool, if the value in parameter "Core bit maximum diameter" in the section is not exceeded.  .
Generally, this value is set when the machine is installed and it does not require any further editing.

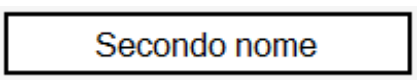
Importing the DXF file

Importing of the dxf file takes place on the workpiece creation page (see figure below). To access this page press  ; and, after it has opened, press the button  .
The list of files included in the work folder of the PC appears. The following options exist to change the folder or to delete files:

-  All files in the USB memory fitted in the PC
-  All the files present in the work folder of the PC
-  It is possible to explore all the folders of the PC
-  Permanent erasure of a file

Press on the file name to select it and enter the workpieces in the drawing with  .

It is possible to change the number of identical workpieces to be made under the heading "Number of Pieces", next to the button  (basic is 1): the program will repeat the pieces in the drawing for the number of times specified in '"Number of Pieces'.

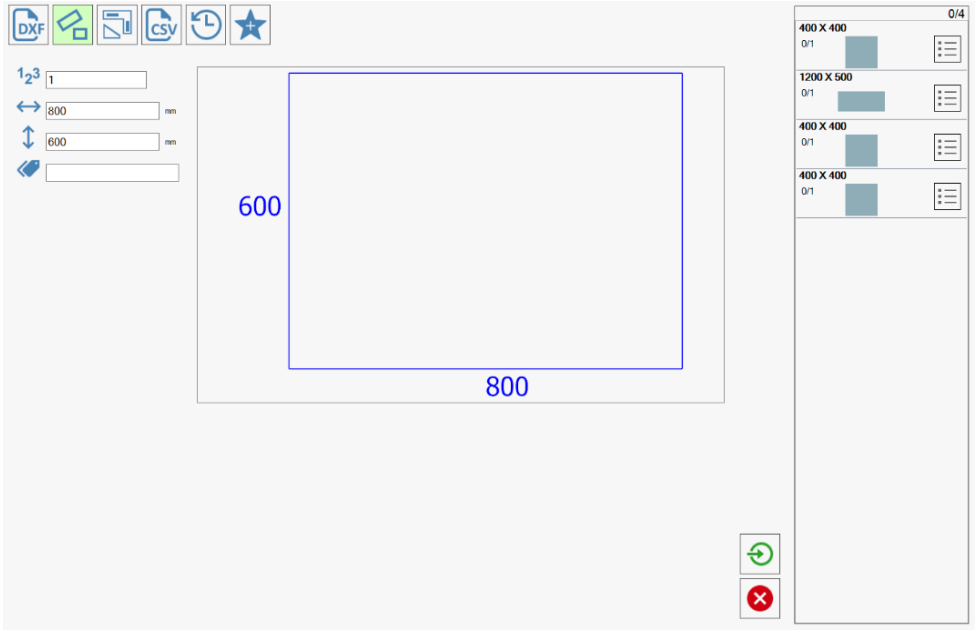
Added Buttons	
Ordering the files present in the basic list by date. By default they are sorted from the most recent to the oldest and the button appears like this:  . Pressing the button again the files are sorted from the oldest to the most recent and will appear like this:  . It is possible to repeat the operation several times, passing from the most recent to the oldest and vice versa	
The files in the basic list are sorted by file name. By default, the files  are sorted by name when the button is pressed.	
It associates a second name to the file, if the function is enabled (green). A keyboard will appear that allows insertion of the name when pressed  .	

5.1.5.2 CREATE RECTANGLES

With Parametrix it is possible to create rectangular pieces entering the measurements of length and width and to place them on the list of pieces to be worked. The program also allows writing of the brand (or label) of the piece and the number of repetition of equal-sized rectangles.

To access the rectangle creation page press on  , on the piece creation page activate the button 

which will display the following screen:



For insertion of the rectangles the following information is required:

- *Number of Pieces:* number of rectangles with the same measurement.
- *Width:* Width of the finished part
- *Height:* Height of the finished part
- *Brand:* Lettering to be inserted on the piece. This is an optional parameter as it is not essential for creation of the piece

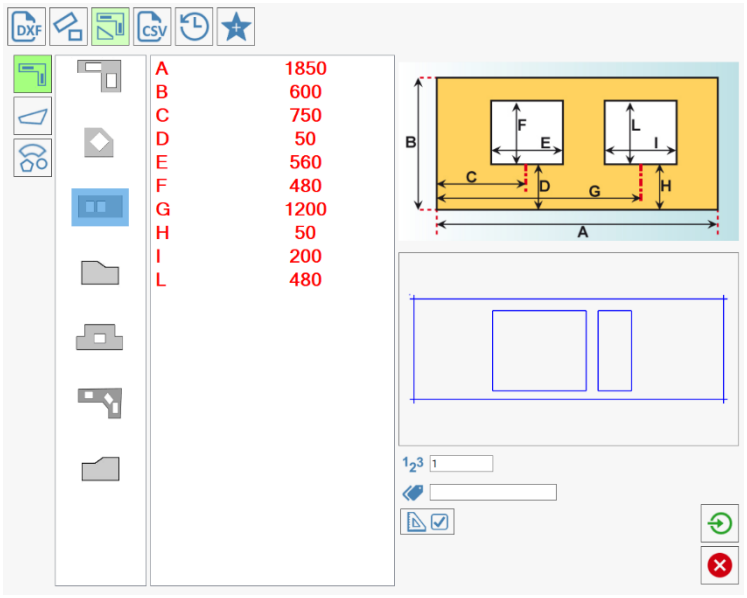
When all the values have been entered, press the button  to insert the rectangle in the list of pieces to be cut.

Note	*It is possible to modify certain information of already created pieces, such as: "Name", "Pieces Inserted", "Total Pieces" directly from this page with the button*  *, visible in the lower right corner of the image above*

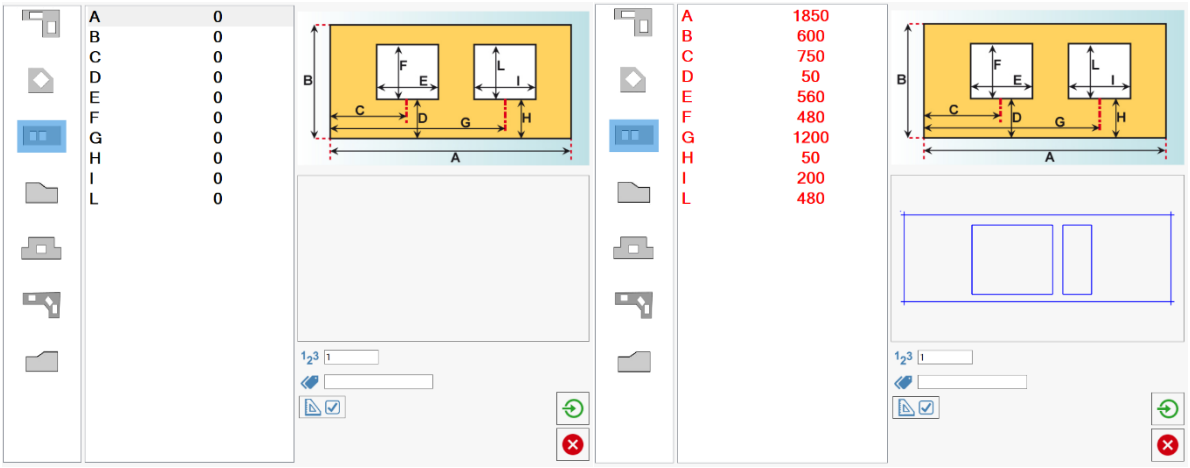
5.1.5.3 CREATE PIECES WITH PARAMETRIC FIGURES

It is possible to create the pieces by entering only the dimensions of the sides without having to draw them. To access the parametric figures library of the program, press on the button 

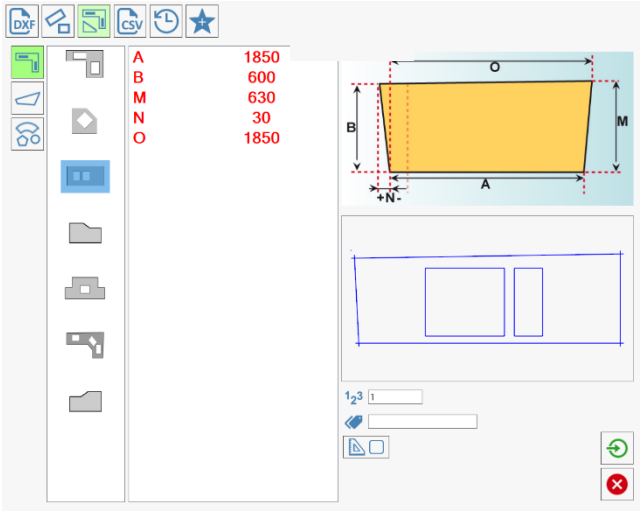
on the piece creation page, activate “FIGURES”, which will display the following screen:



To load the workpieces, select the type of work “KITCHEN”, “STAIRCASE” or “FIGURES”: the relevant figures will be displayed accordingly. Select the desired shape by clicking the list of workpieces: the image will be uploaded with the dimensions necessary for its creation. Then fill in the value boxes with the actual measurements of the workpiece. To do so, select the letter for which the measurement is to be entered. A numeric keypad will appear that allows insertion of the values.



For some pieces it is possible to enter non-square corners (other than 90 degrees). Simply press “NON-SQUARE” and additional values will be requested to allow the program to calculate the corners of the piece. Example of a non-square piece:



Once the values have been entered it is possible to enter the brand (or label) that appears on the piece when it is inserted into the work area.

Press the button  creates the piece in the list of pieces inserted.

5.1.5.4 CREATION OF CORRECT EXCEL FILES FOR IMPORTING OF PIECES

To successfully create an Excel file to be imported into Parametrix, the file must be structured in a certain way:

1. The file saving format must be "csv"
2. The ORDER of the columns must not be changed following the settings made by the installer.

Numero	Larghezza	Altezza	Nome	Spessore
2	200	500	a	
5	300	800	b	
0	300	800	c	
5	0	800	d	
5	300	0	e	
0	0	0	f	
			Pezzo 3	
5	300	800	h	

This image shows an example of a csv file.

Explanation of the columns
Number: It indicates the number of pieces to be created with those sizes.
Width: It indicates the width that the piece should have.
Height: It indicates the height that the piece should have.
Name: Name to be associated with the piece.
Thickness: thickness to be associated with the workpiece. If this parameter is not filled, the thickness of the current material will be taken.

Note	*The name of the columns may change, e.g.* Number *can become* numPieces; *what is important is that the meaning of the column does not change, i.e. the first column identifies the number of workpieces to be created with those sizes and with that name.*

The example above shows some lines that are not correct, which is why only some of them will be imported into Parametrix, namely lines 2, 3, and 9.

Errors in the example which prevent workpiece import

Line 4: the number of the workpieces is zero
Line 5: the workpiece width is 0
Line 6: the workpiece height is 0
Line 7: all the fields are 0 (except for name)
Line 8: all the fields are 0 (except for name) If both Height and Width are blank, the table is deemed to have been completed.

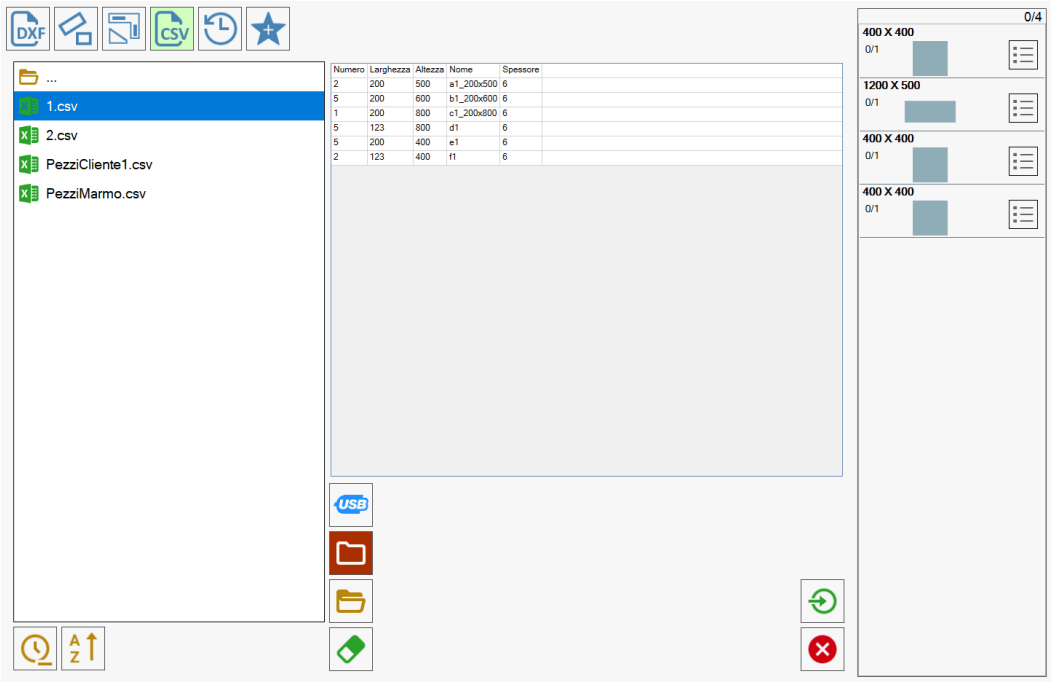
Note	If the name field of a single line is empty the piece is still created and the name is associated with "WidthxHeight"

5.1.5.4.1 IMPORTING OF EXCEL FILE (.CSV)

The pieces can be imported from Excel by selecting the button  on the first screen of the program and the button  .

The Excel files which are present inside the folder set during installation of the program will be shown within the table.

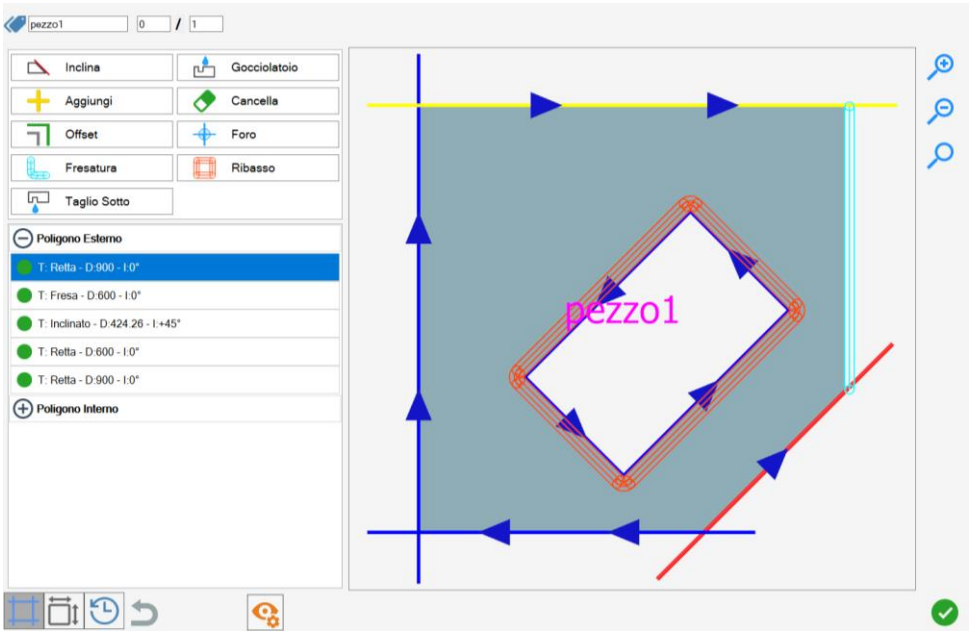
The buttons  ,  ,… behave in the same way as the buttons described in *DXF Import section*



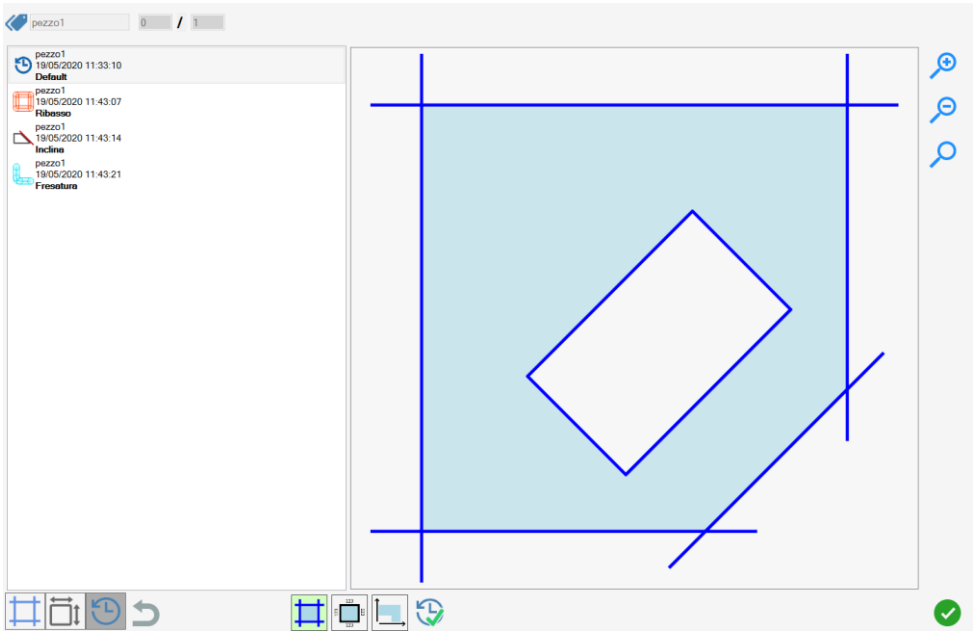
Press the button  creates the piece in the list of pieces inserted.

5.1.5.5 HISTORY

It is used to view all the changes made to the current workpiece and, where required, it helps take the workpiece back to its previous state.



The following screen will be displayed after pressing the button  .



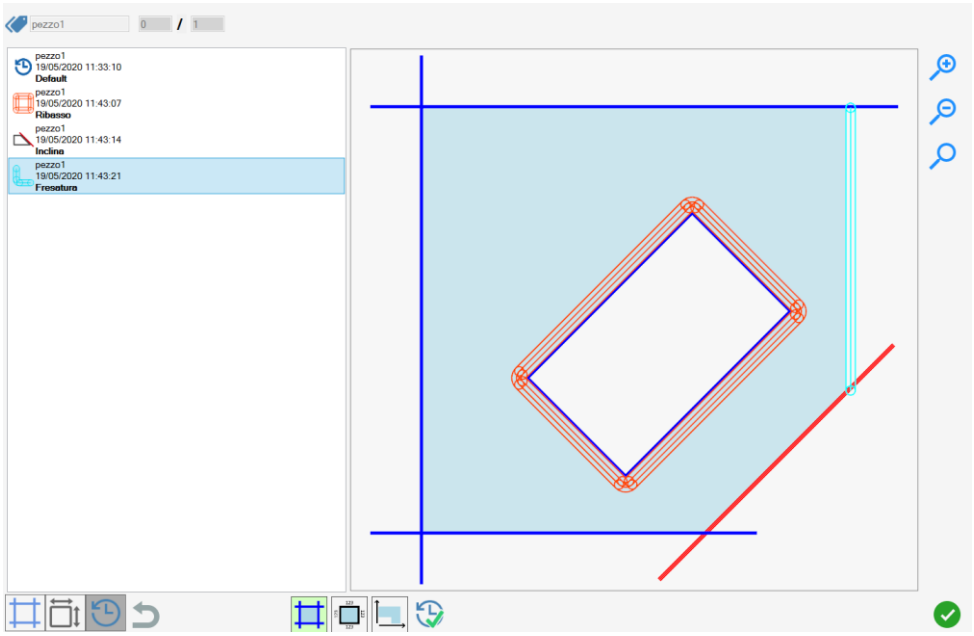
The state of the current workpiece without changes is called “Default”. The bottom part of the panel contains these four buttons. Three of them are display utilities:

	The piece is displayed showing the cuts
	The piece is displayed showing the values
	The workpiece is displayed together with its overall dimensions.

The button  restores the current workpiece to the state selected in the list on the left. When a previous state is applied, any changes made after the selected change are removed, except for the default state which is never removed. Each change shows:

- the name of the current piece
- the change time
- the name of the applied change
- the icon of the specific change

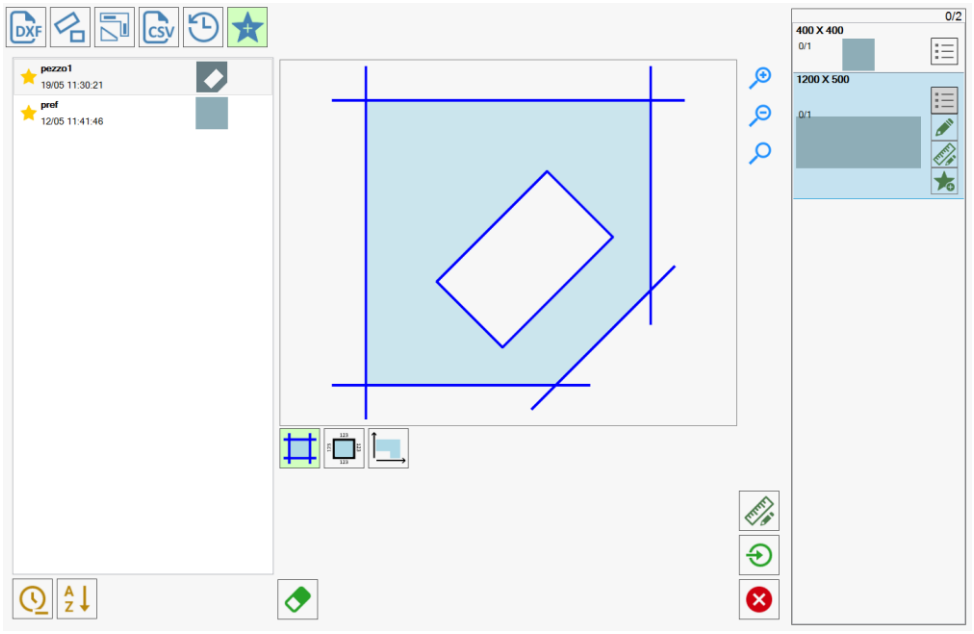
If changes have been applied to the current piece, they will be displayed in the list on the left



5.1.5.6 FAVOURITES

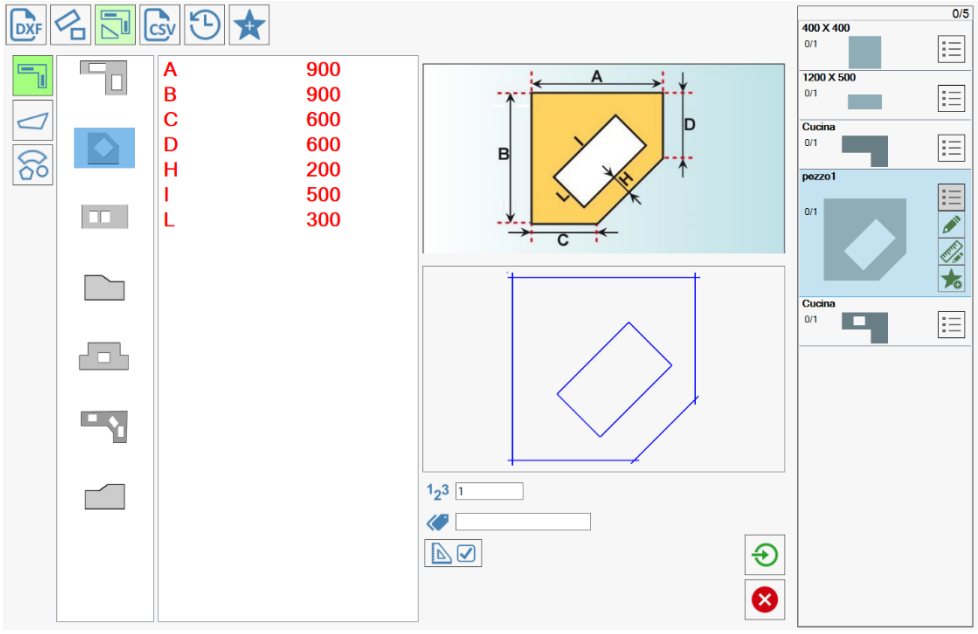
You can add a piece to the list of favourites by clicking the button  next to the piece and then the button 

A piece in the favourites list can be reused by pressing the button  and select it in the list on the left.



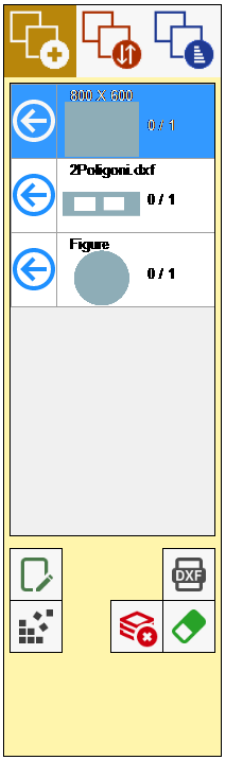
5.1.5.7 CHANGE SHAPE

It is possible to create a piece starting from an existing piece by pressing the button next to the piece and then the button



5.1.6 LIST OF ADDED WORKPIECES

When a workpiece is created (see Chap. 4 “Workpiece creation”), the software adds it to the list of workpieces. The workpieces can be placed from the list onto the work area or they can be subjected to some changes; additionally, the number of workpieces missing for the machining process can be checked.

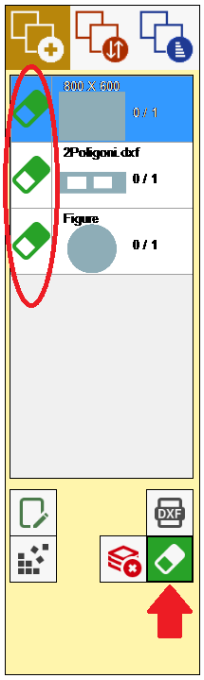


Each line of the workpiece list contains 3 pieces of information on workpiece production.

Example of line: 	
• Shape preview	
• Brand or label. **Note:** if there is no brand, the software writes the name of the file (dxf) or the measurements of the rectangle.	

• Number of workpieces added/to be added	
• Using the arrow insert the piece within the work area. Having reached the maximum number of pieces, the arrow disappears as no further cutting of pieces is required with the characteristics shown on the line.	

To delete a line in the pieces list press  . The program will prompt selection of the row to be deleted, drawing the rubber instead of the arrow (see figure below)



When the line is selected Parametrix deletes the entire contents and all the pieces.

CAUTION: *With this function*  *the program also cancels the pieces connected to the deleted line present in the work area.*

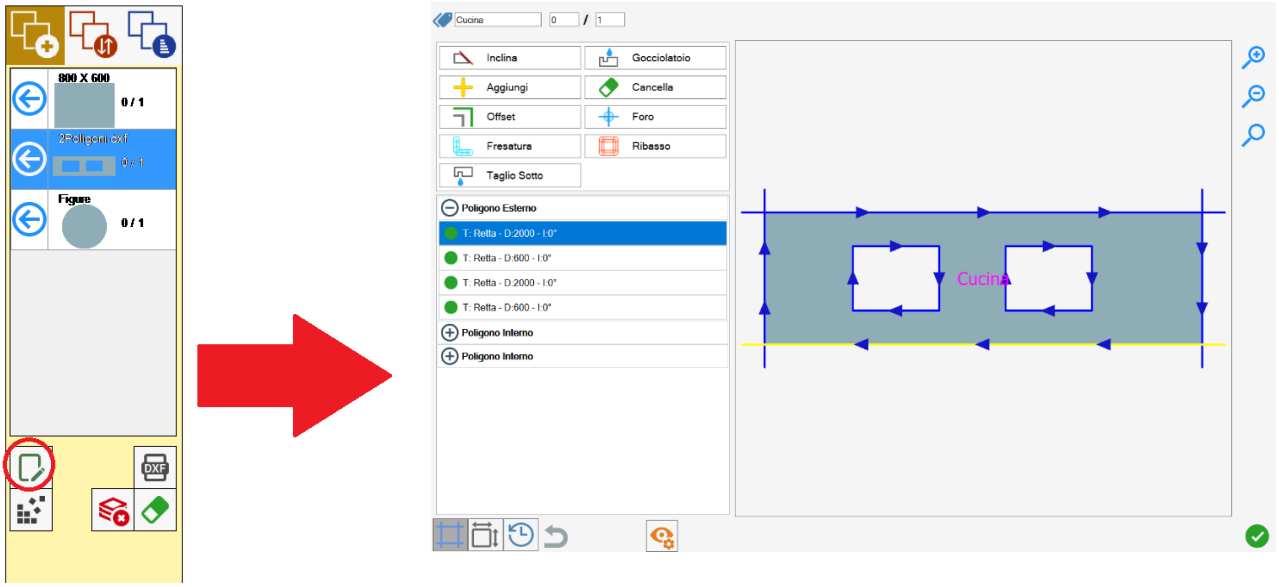
To cancel all the table contents, press the button  . Confirmation is requested for this operation.

5.1.7 MODIFY PIECES

Within the program it is possible to modify or add certain machining modes to the piece:

- Inclined cuts
- Drip cut
- Offset cut
- Added cut
- Holes added with the hollow bit
- Added cutter cuts
- Adding the Lowering with the cutter (for the recessing of sinks)
- Delete cut

To access the edit page, select the line from the parts list and press the button  .

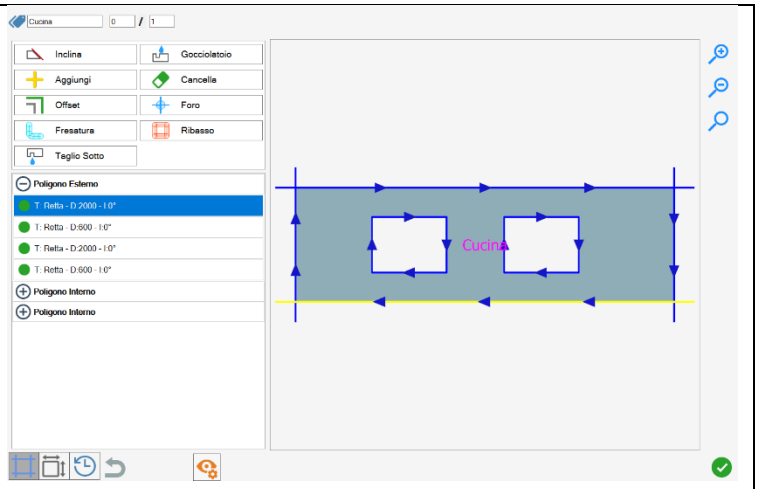
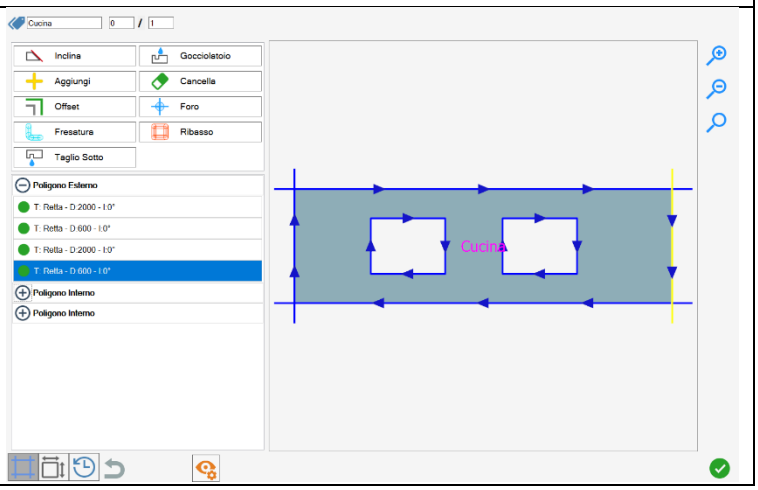


Certain functions are also present on the main page of Parametrix, though by changing cutting through the "Modify Piece" screen all the pieces of that line will have the desired machining whereas if the machining is set on the work area only the selected piece will be changed.

Example: we create a line with 10 measurement rectangles 1000 X 500

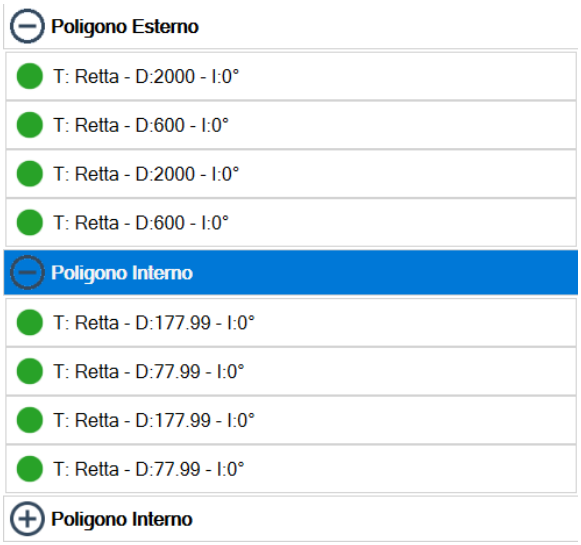
"Modify Piece" page	**Work Area**
Modifying the angle of one side, *all* the rectangles will have the angle changed at the time of insertion into the work area.	If we modify the inclination on the work area the individual piece will be changed, not all the pieces of the line.

To modify a machining mode, it is necessary to select the cut on which to apply it. There are Systems for selection of the cut:

1. clicking on the drawing of the piece will select the closest cut. *This selection system does not include confirmation for insertion of the function.*	
2. Selecting a line on the list of cuts will result in the relative cut being selected.	

5.1.7.1 LIST OF CUTS

The list of cuts shows which are the cuts of the piece in question also indicating if they are part of the external or internal perimeter.
This however is only the case if the piece is composed of multiple polygons. If the piece is composed of only one polygon, only its cuts will be shown.



Pressing on the "Polygon" lettering hides/shows the cuts of that particular section of the piece. This makes it easier to view the cuts of the piece when it possesses many of them and offers the possibility of finding a particular cut quickly, already knowing to what portion of the piece is belongs.
The dot visible within the list can be used to disable the cut where it doesn't need to be performed. Making this change means making a change to all the pieces of that particular line, as already explained above, when the changes on the "Work area" and "Modify" were evident.

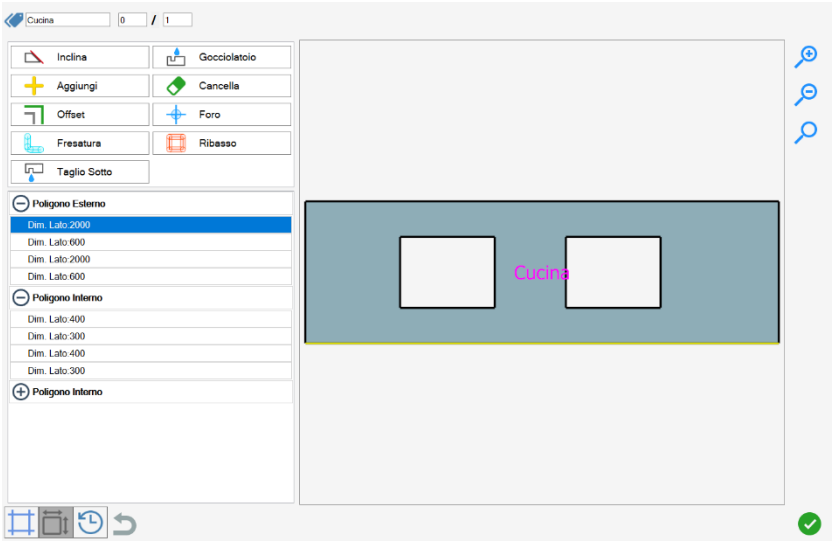
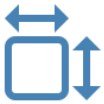
Information on lines

When the “Sides of Piece” button is deselected (not green), the list is shown as per the images above and provides various information regarding the cut. In fact for every line it is possible to find the following information (if the particular cut has):

1. **T**: This indicates the type of the cut (Straight, Curved, Circle, Milling, ...)
2. **D**: Size of the cut
3. **I**: inclination of the cut

If on one part of the piece there is no cut, this is indicated by "None".

5.1.7.2 OPTIONAL

Button Description	
Zoom Buttons: these are used to increase, reduce or restore the initial conditions and sizes of the image of the Piece.	
It enables cancellation of the changes made to the Piece. If the Cancel button is held down, the user can go back to the initial conditions of the workpiece. **Note:** the button works for as long as the “Modify workpiece” panel remains open on the workpiece in question. If the panel is closed, the opportunity to cancel the changes to the workpiece in question will be lost, even if the panel is opened again.	
It allows confirming of all the changes made to the Piece.	
This button allows displaying of the length of the sides of the Piece within the list rather than the length of the cut. Selection directly on the image or on the list will result in the side of the piece selected turning yellow.  In this mode it is not possible to use the functions be applied to the piece. To exit this mode select the button 	

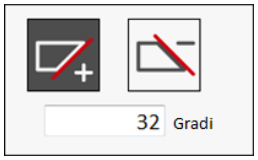
5.1.7.3 FUNCTIONS

On the pieces it is possible to apply particular functions to perform certain types of machining, such as milling cuts, adding holes to remove material that the blade was unable to remove, ...

These functions are not available for all the workpieces; where this is the case, a message will be shown on screen.

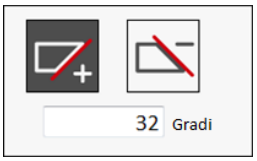
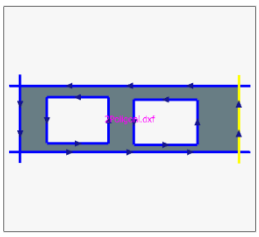
5.1.7.3.1 INCLINATION (OPTIONAL)

The “Inclination” function defines a machining process where the A axis is different from 0. A click on the button causes the following window to appear on the bottom right-hand side.



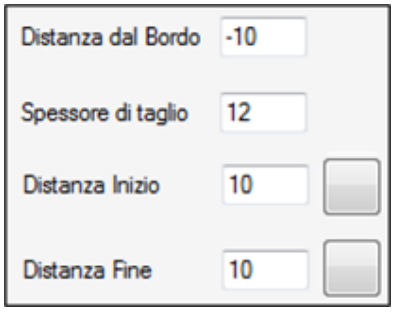
The buttons are pressed  to decide the direction of the set square to the workpiece, while the text below indicates the machining value in degrees. After entering the parameters, select the cut to be modified. The inclined cut appears in a different colour.

Example of cut set at 32 degrees

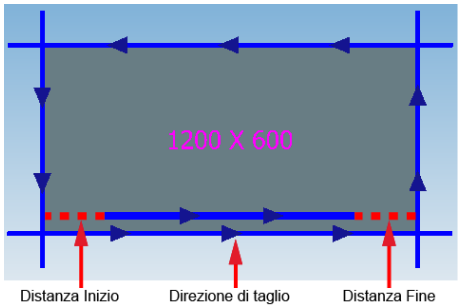
On the "Modify Piece" page, press the "Inclination" button.	
1. Select the square and enter the desired value.	
2. Select the cut on which to set the inclined cut, directly on the piece or on the list as explained before.	

5.1.7.3.2 DRIP

The 'Drip' function allows adding of a cut with characteristics different from the normal cut. It is possible to set the cut width (it must be equal to or greater than the thickness of the blade) and the penetration depth into the material.
The Drip is inserted parallel to a cut of the piece. It is necessary to select the desired cut and to set the distance (actual distance of material) on which to insert the drip.
When calling up the Drip function in the bottom right the following data request appears that is discussed below:

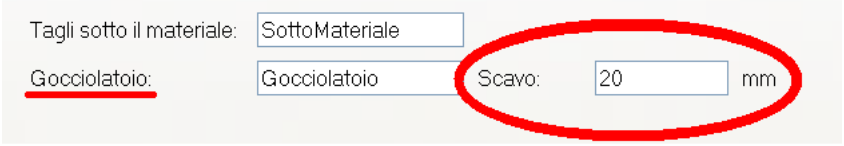


Distance from the edge is the distance from the selected cut to the drip machining.
Cut thickness is the measurement of the machining width. As already stated it is possible to perform a Drip of width greater than the thickness of the blade. The program will make parallel cuts until reaching the thickness selected.
Start Distance is the internal distance with which the machining takes place with respect to the start of the cut. See figure below for greater clarity.
End distance, similar to the start distance only that it refers to the end of the creation cut.



Start Distance and *End Distance* can be activated or deactivated with the buttons to the right of the value. Where they are grey (off) Parametrix does not consider them whereas when they are green (lit - as shown in the figure) the program calculates the displacement.

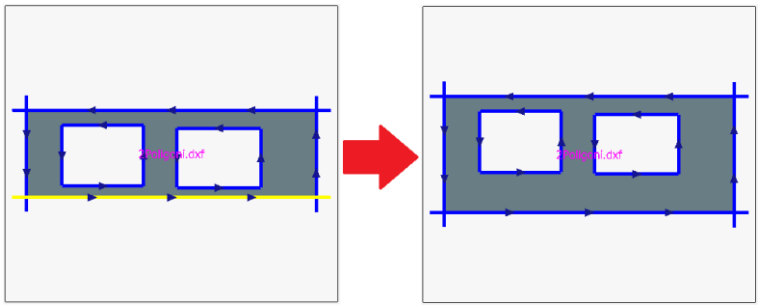
Penetration into the material is set within the parameters of the machine. To modify the data, follow the procedure shown in the table:

Press the button  located in the upper left of the program	
"***Layer Name***" and change the **emptying** value found after the Drip layer name.	

5.1.7.3.3 OFFSET

Note	*Where an offset is entered all the changes previously inserted on the piece will be lost!*

The 'offset' function expands the piece moving parallel to the cut selected. The displacement measurement is entered as a parameter. In the figure below is an example of the offset of the side below (highlighted yellow)

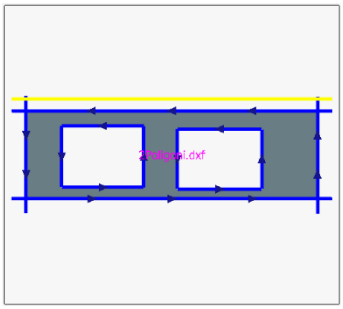


To activate the function press the “Offset” button and enter the displacement value in the box to the right.

Having inserted the measurement select the cut on which the offset is to be made.

5.1.7.3.4 ADD CUT

The "Add Cut" function is used to add a cut parallel to the cut selected. The distance between cuts is defined by the parameter. Where the value is greater than 0 the cut is calculated out of the piece, while if the value entered is less than 0 then the cut will be made inside the piece.

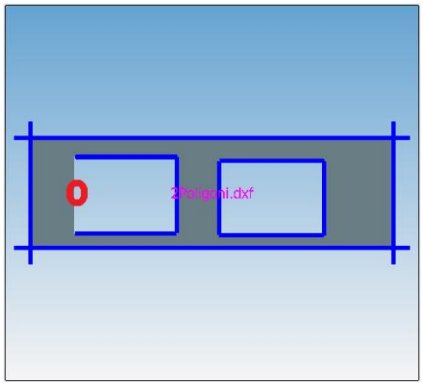


To add a cut, press the “Add cut” button, set the distance between two consecutive cuts (figure below), and select the cut with respect to which the new cut must be created.

5.1.7.3.5 CANCEL CUT

The "Cancel Cut" function allows the user to cancel a cut of the piece.

To use this function, press the “CANCEL CUT” button and select the cut that needs to be cancelled.



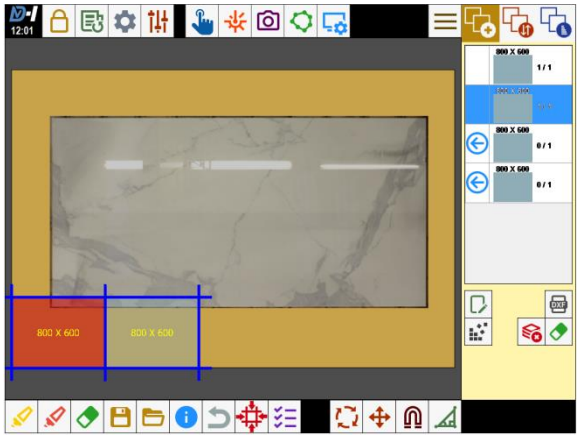
5.1.8 WORK AREA: POSITIONING OF PIECES AND OPTIONALS

Parametrix is designed to position the workpieces inside the work area of the machine to create the cut file. After the slab is acquired, the perimeter and the workpieces have been created, the workpieces must be imported into the slab. The button  , which is featured on the workpieces table, is pressed to place the workpiece within the work area. The machining cycle can be run with 1 or multiple workpieces at the same time.

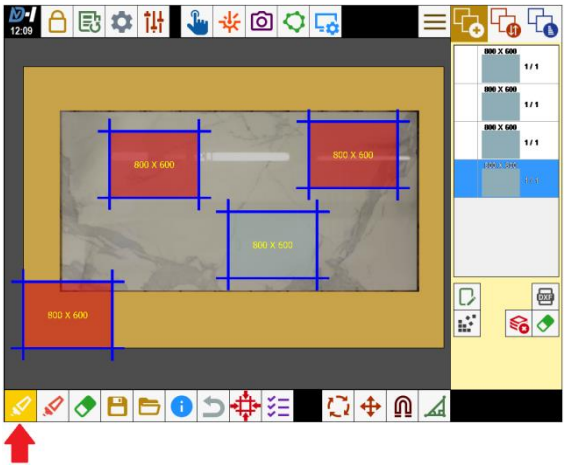
If the pieces are not perfectly inside the slab, a caution/warning signal  will appear in the top left.
The appearance of this signal indicates that problems may arise during the cutting stage, however none of the program functions will be blocked.

5.1.8.1 WORKPIECE SELECTION

When the workpiece is placed in the work area, its colour is light grey. The workpiece is selected by a click on it: the selected workpiece turns red.



It is possible to select 1 workpiece at a time. For this reason when pressing on a piece the active one is automatically deselected.
To enable selection of multiple pieces press the button  . With this system active, multiple pieces will be selected at the same time. In this case to deselect 1 piece press inside a perimeter that has already been selected. The figure to the side shows an example of 3 pieces selected at the same time. To select all the pieces present in the area press the button  .

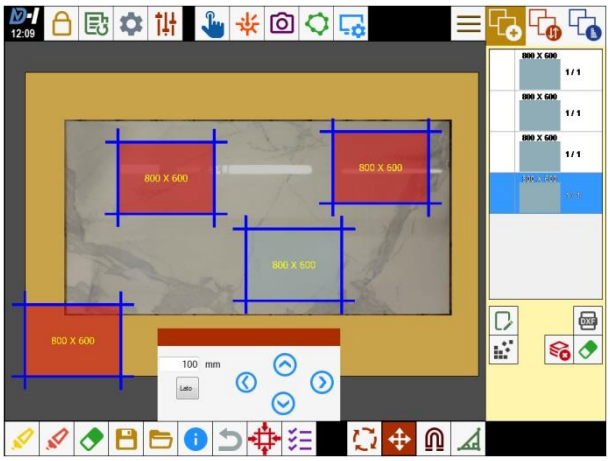


5.1.8.2 DISPLACEMENT OF PIECES

There are two ways to move the piece within the work area:

1. ***Movement*** of the piece takes place by clicking **inside** the perimeter of the piece and by moving the finger in the desired direction.

2. Pressing  , a panel with 4 arrows appears to move the selected pieces. The distance of movement for each click is given by the value under the arrows.



The “SIDE” button in the Panel allows the execution of parallel or orthogonal movements with respect to the side of the piece selected. If multiple pieces are selected at the same time all of them will move consistently with respect to the side selected.

5.1.8.3 ROTATION OF PIECES

There are two ways to rotate the piece within the work area:

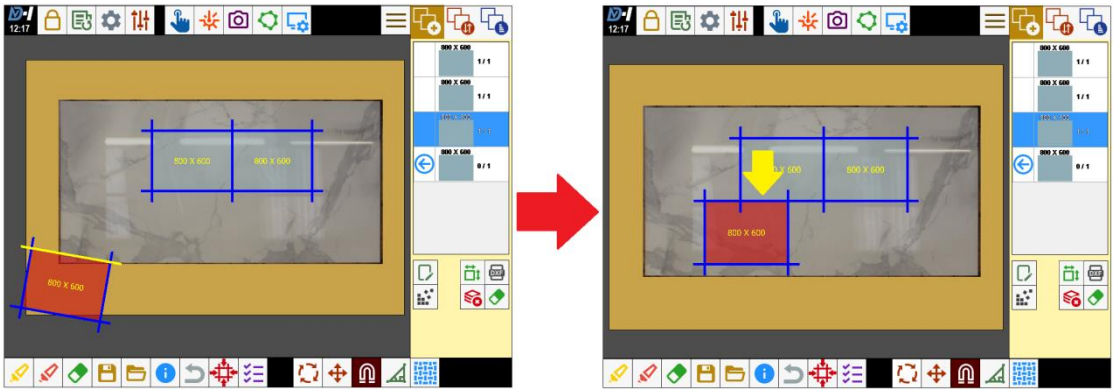
1. **Rotation** of the piece takes place by clicking **outside** the perimeter of the piece and by moving the finger in the desired direction.

2. Pressing  , a panel with 2 arrows appears to rotate the selected pieces. Rotation in this case is shown by the value under the arrows.

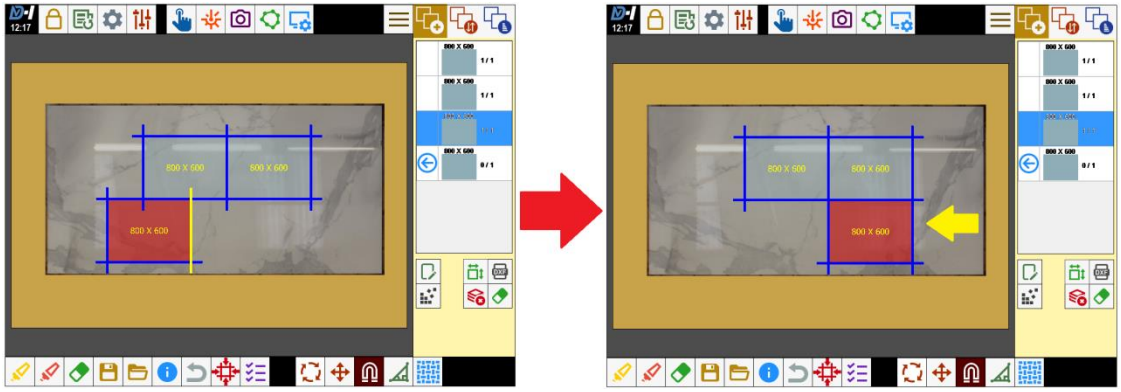
Note: *the Move **and** Rotate functions are applied to all the workpieces selected. Hence, it is possible to move multiple workpieces simultaneously.*

5.1.8.4 MAGNET FOR CUTS

A magnet system for the cuts makes it possible to join 2 cuts together. Where the cuts are not parallel, the workpiece is rotated parallel to the cut. To activate the function, press the button  , select the cut to be moved and the cut on which to magnetise the workpiece: the **FIRST** cut selected moves towards the **SECOND** cut selected (see example below).



Cuts on adjacent workpieces can also be magnetised, as shown below.



The above option is not possible if the cuts are made by milling cutter or by blade and milling cutter.

5.1.8.5 INCLINED CUT (OPTIONAL)

The software offers the possibility to set cuts made by an inclined blade. For greater understanding refer to *Paragraph 7.6.3.1 Inclination*.

5.1.8.6 DELETE PIECE

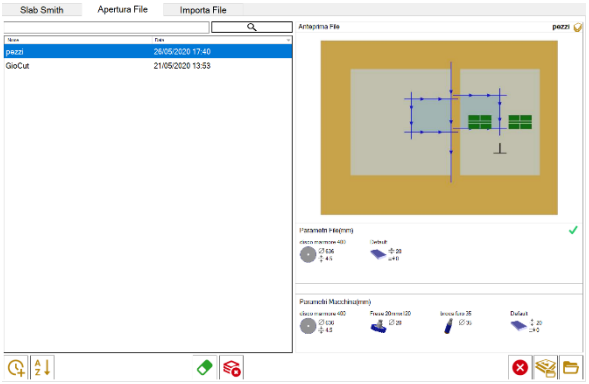
By pressing the button  , the selected pieces are removed from the work area and are inserted in the list of pieces to make them available later.

5.1.8.7 REMOVE PIECES

Press the button  in the top left of the page of the program it is possible to permanently remove the pieces present in the work area and the slabs. It should therefore be used when all the pieces are considered cut or no longer needed.

5.1.8.8 SAVE AND OPEN

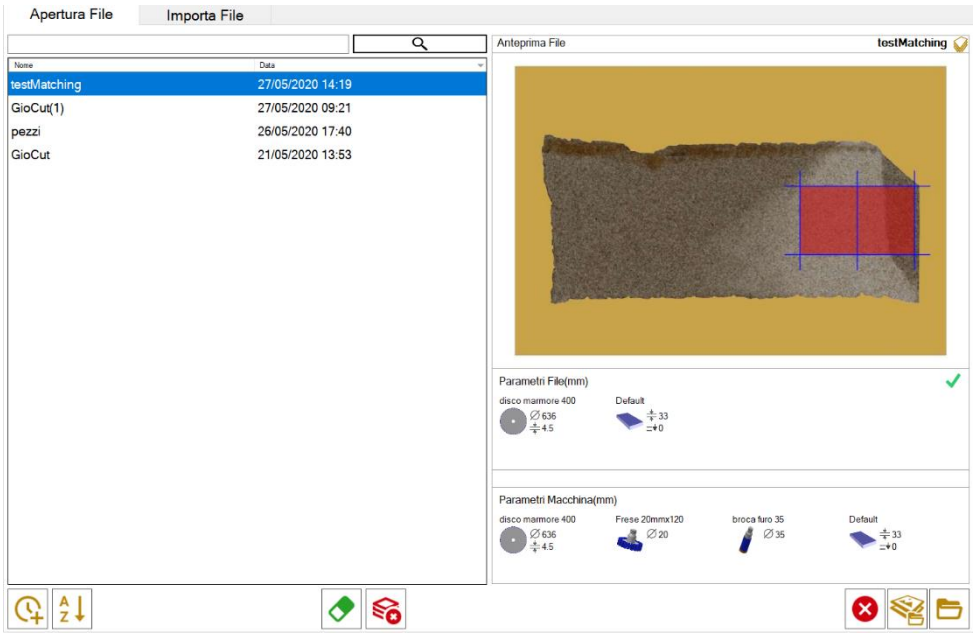
It is possible to save the ongoing machining process: workpieces, plates, modified cuts, ... with the button  . Button  is pressed to choose the configuration to be retrieved, which the software will open as it was when it was saved. The following window opens:



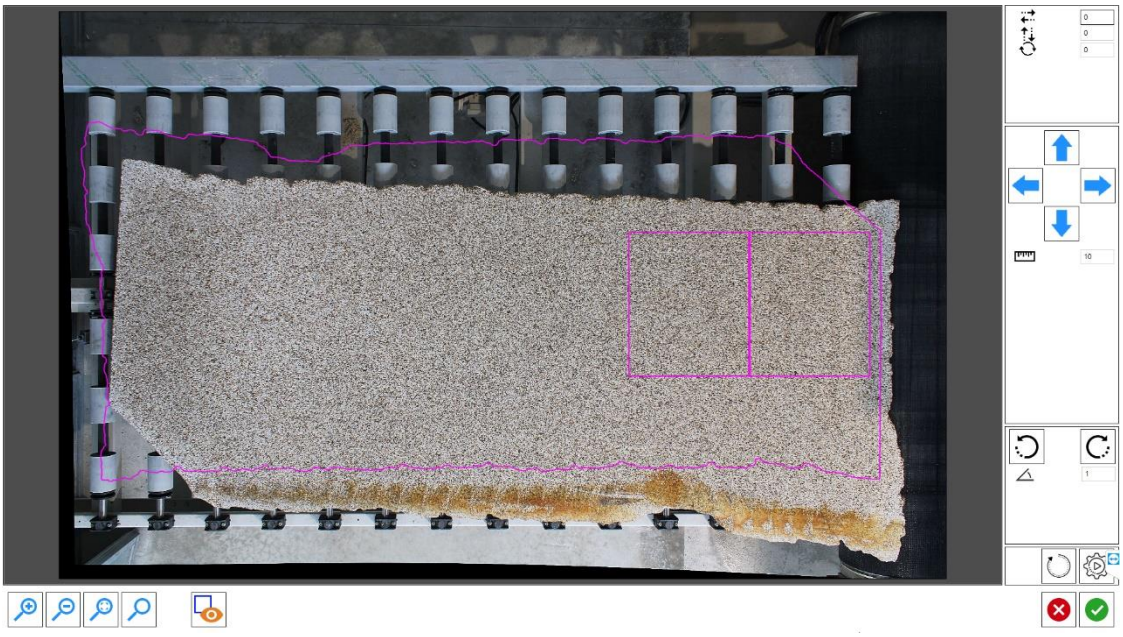
it is possible to view a preview of the saved process, the tool materials and material with which machining was carried out
To fully open a machining configuration, the file parameters must be identical to the machine parameters. If they are identical, a green tick will appear on the right, otherwise a red cross will appear. The file parameters that are different from the machine parameters are also indicated in red.
If the parameters are not the same, the machining configuration will only partially open - workpiece arrangement only. A message warns the user that modifications to cuts cannot be made.

5.1.8.8.1 SLAB MATCHING

This function allows users to position pieces in the Parametrix office installation using an image of the slab, then realign the perimeter and pieces with the actual position of the slab itself on the machine's workbench. To open a saved job using the slab-matching function, the user can press the relevant button in the Open / Import File dialogue box.



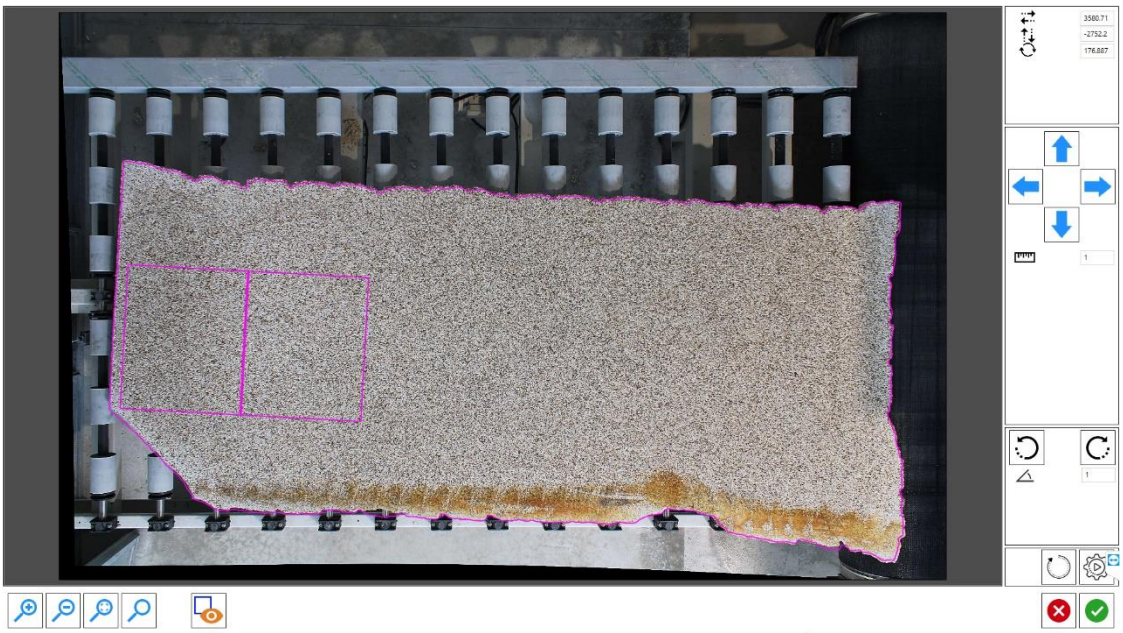
The following slab-matching window is opened when button is pressed.



Red line in the figure	Perimeter of the slab imported from the office
	X Offset to be applied to the perimeter
	Y Offset to be applied to the perimeter
	Rotation to be applied to the perimeter
	Buttons to move the perimeter
	Step of the displacement applied with the movement

	Buttons to rotate the perimeter clockwise or anticlockwise
	Step in degrees applied using the rotation buttons
	Reset button to reset the perimeter position on the one imported from the office
	Automatic alignment of the perimeter
	Zoom Buttons: inside, outside, window, fit
	It hides the pieces present in the work area
	Interrupt and confirm the buttons

Automatic alignment only works if the photo imported from the office and the photo taken on the machine are images of the same slab. The function cannot work in any other case, even if the slab appears similar.
The perimeter can be aligned manually: it can be moved horizontally and vertically by clicking inside the perimeter and dragging it in the desired direction. It can be rotated by clicking outside the perimeter and dragging it.
It can also be moved using the appropriate buttons on the right.
When the perimeter is aligned (see figure below) the user can confirm



The photo taken on the machine will be cut out using the perimeter imported from the office and the pieces will be placed on the slab accordingly.

5.1.8.8.2 SLAB SMITH

It is a software for the management of kitchen tops processing in general; Parametrix is able to import the cutting layout created with the software.
The software produces a dxf, the perimeter of the slab is put on a particular layer, all the pieces are put on different layers. Parametrix imports the dxf and performs the processing respecting the arrangement of the pieces on the slab. To import the dxf file press the button  and press the "Slab Smith" tab on the window that opens up.

5.1.8.9 IMPORTING AND EXPORTING

If a machining configuration needs to be transferred from Parametrix in one PC to Parametrix in another PC, it can be exported and then imported. To export, press the button  and open the “Export File” tab. When the Confirm button is pressed, a .prx file will be generated. Transfer the file to the second PC and import it by pressing  and opening the “Import File” tab.

5.1.8.10 CANCEL

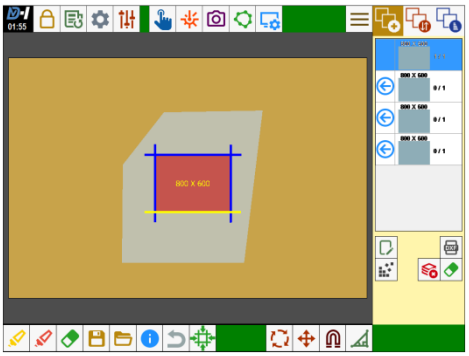
This button is pressed to cancel the last operation performed. 

5.1.8.11 INFORMATION PANEL

If visible, by selecting the button  , it will be possible to obtain certain information on the tools currently used, the name of the current processing and some statistics on the Slab.

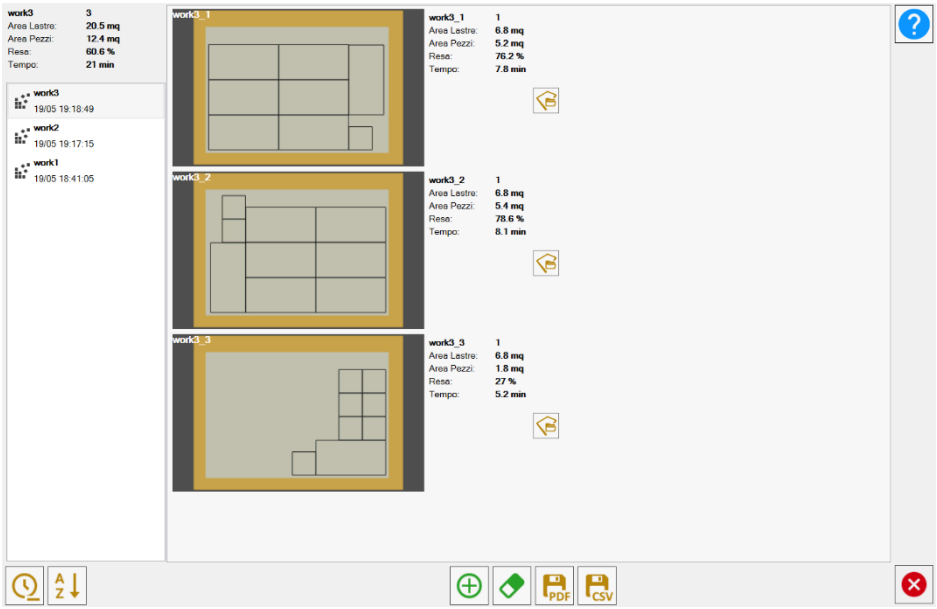
5.1.8.12 MACHINE ON CUT

This function is intended to control the position of the workpiece in the work area to its position in the machine. When “Machine on cut” is enabled, a cut is selected on screen and the machine moves to the corresponding position.
This operation is useful to check if a cut is included in the material.



5.1.8.13 COMPLETE NESTING

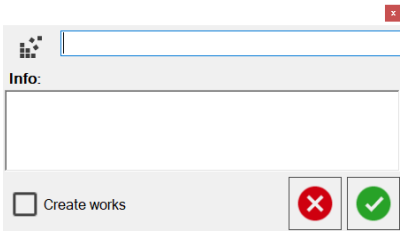
This function is designed for the nesting of the workpieces in the list on multiple slabs, until there are no more workpieces. The following window is opened when button is pressed.



It shows the number of slabs used for the machining process, selected in the list on the left. Information are present for each slab and for the entire job, such as:

- Slab area
- Piece area
- Performance
- Time

The following window is opened when button is pressed.



A new evaluation can be started by entering the name, optional information, and a flag to create the corresponding machining processes.

A click on button deletes some machining processes from the list on the left.

Use buttons and to create reports of the processing phase in a pdf or csv format.

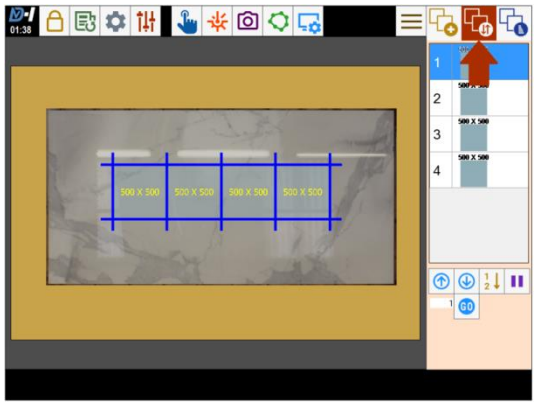
Use button near the slabs to take the processing directly to the work area.

5.1.9 PIECE CUTTING ORDER (OPTIONAL)

Having positioned the workpieces in the work area, press  (visible in the image) to access the system that sorts the workpiece cutting sequence.
The arrangement of the workpieces in the list on the right indicates the order in which they are cut. The order can be changed in 3 ways:

1. Arrows
2. Positioning
3. Selection by click

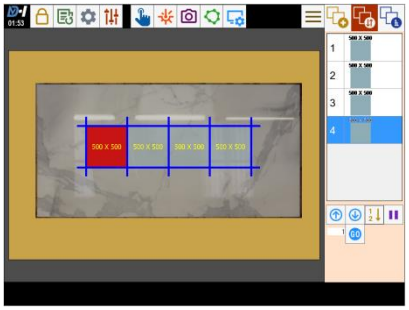
The following sections explain the various sorting systems.

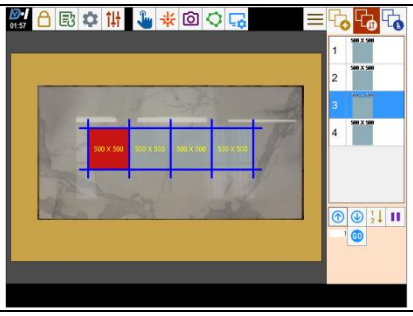


While sorting the workpiece cutting sequence, a pause can be set up between two consecutive workpieces. The software will perform all the cuts on the workpiece set to be made before the pause and then it will stop the machine and wait for the Start button to be pressed.

5.1.9.1 ARROWS FOR WORKPIECE POSITIONING

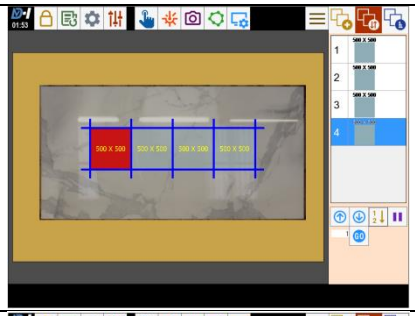
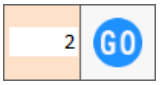
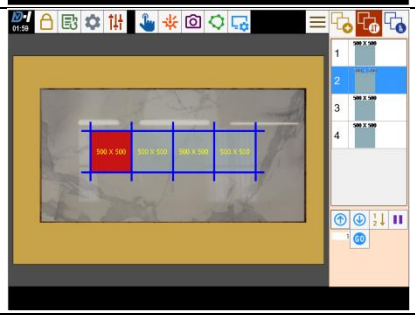
With the arrows below the list  it is possible to move the piece to another position.

1. Select from the list the piece to be moved. The selected piece in the image is 4.	

2. Press on the arrow  . The selected piece will be moved to position 3 of the list.	

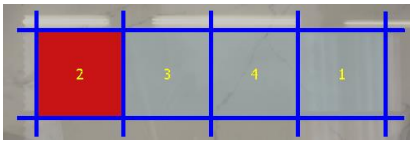
5.1.9.2 POSITIONING

The button  moves the workpiece currently selected in the list to the desired position.

1. Select the workpiece to be moved. In this case the workpiece is no. 4, as is visible from the image.	
2. Specify the position that the workpiece should fill in the text box. 	

5.1.9.3 SELECTION BY CLICK

It is possible to choose the order of the pieces directly from the work area using the button  . When pressed, the program writes the order of cutting on the pieces:

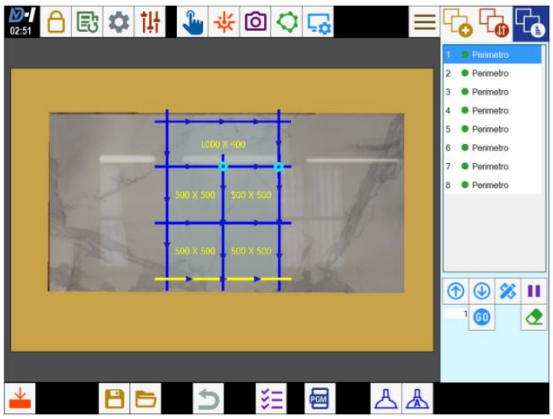


press on the pieces in the order in which they are to be cut. The currently applied order indicates that the first piece cut is the rightmost one, the second the leftmost one, etc.; to change the order, click **inside** the piece to be first, second,...

Once finished, press the button  to validate the choice made.

5.1.10 MANAGEMENT OF CUTS AND DISPLACEMENT

Press the third button on the top right-hand side  to access the cuts manager page.
The cuts are loaded into the list on the right in order of execution; furthermore, the directional arrows of the cut are drawn on the screen.



The chronological order of the cuts can be changed in 2 ways:

- Arrows 
- Positioning 

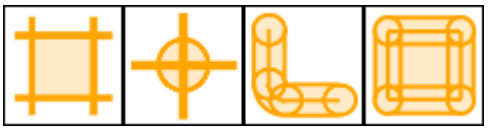
The operation to modify the priority is equal to the number of pieces; for this reason, refer to the explanation in paragraph '7.8.1Piece positioning *arrows'* and '7.8.1 *Positioning'.*

There are two ways to select the cut (coloured yellow):

1. On the "Cuts list" press the line for the same cut
2. Clicks on the work area (when the work area is pressed the cut closest to the click is enabled).

5.1.10.1 MULTIPLE CUT SELECTION

The following panel opens up when pressing the  button featured in the third tab:



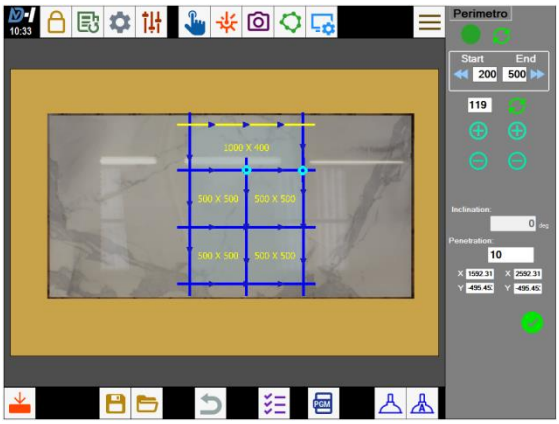
These buttons are pressed for multiple cut selection, respectively:

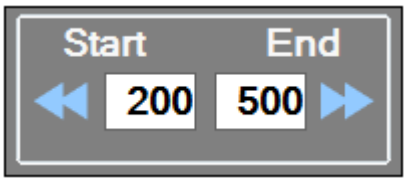
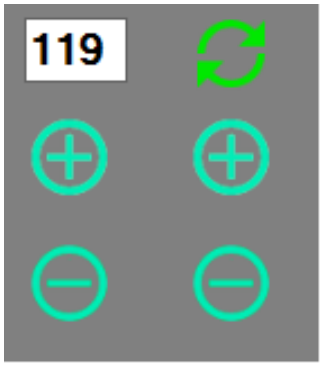
- Selection of cuts
- Selection of holes
- Selection of milling
- Selection of lowering

Once you have selected all the cuts of the chosen type, you can deselect some of them by clicking either on the viewer or in the cut list.
This feature is useful for editing multiple cuts at once. The procedure for modifications is described in the following paragraph.

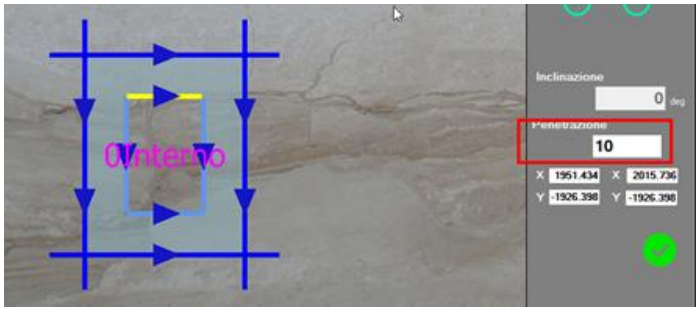
5.1.10.2 MODIFY CUT

Select the cut to be modified and press the button  to access the cut edit screen (figure to the right).
The arrows define the direction and indicate the start and end of the cut. The table below explains the cut edit panel buttons.



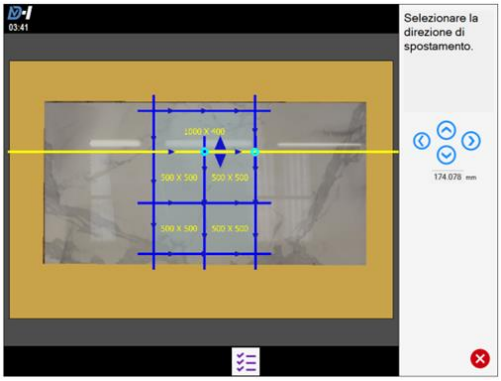
	The button  reverses the start of the cut with the end and vice versa.	
	The arrows are used to extend either the start or the end of the cut, as far as the edge of the slab. The values in the boxes are used to extend the cut outside the slab for a predefined measure.	
	Buttons  and  are used to lengthen or shorten the cut. The left buttons are used to shorten/lengthen the start part of the cut while those on the right change the end part. The shortening/lengthening distance is inserted by the user within the data panel  . By pressing the button  the program calculates the cut length value (based on the blade diameter and material thickness) and enters the value on the data panel.	

After changing the cut, the Confirm button must be pressed.  .
The installer parameters include the “Internal Cuts Penetration” parameter, using which it is possible to set a fixed penetration for all internal cuts. An internal cut with different penetration will be highlighted in a different colour.



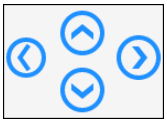
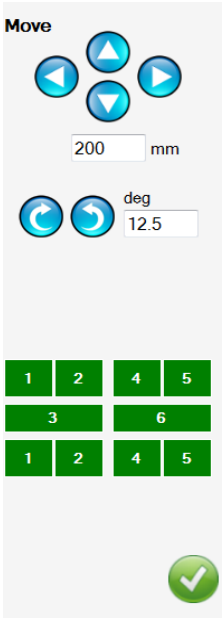
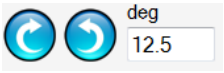
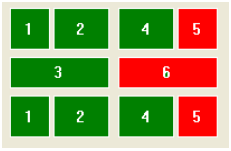
5.1.10.3 MATERIAL TRANSFER WITH SUCTION CUPS

Press the button  the selected cut is used to divide the slab and move the material with the suction cups.



To move the slab it is possible to use the 4 Arrows visible on the right, or by selecting one of the two arrows on the work area. The amount of movement of the slab in both types is written in the text box below the arrows.

To change the position of the suction cups or areas of vacuum use  .

	With the directional arrows it is possible to change the position of the suction cups when the movement is made.	
	Rotation of the suction cup group can be changed with the arrows on the left. The value indicates by how much the suction cup group rotates with every click on the arrows	
	The arrangement of the suction cup group allows the operator to select empty areas to be enabled or disabled during displacement. In the event that an area is green the machine will activate the vacuum, if it is red it will not. To the left an example with area 5 and 6 off.	

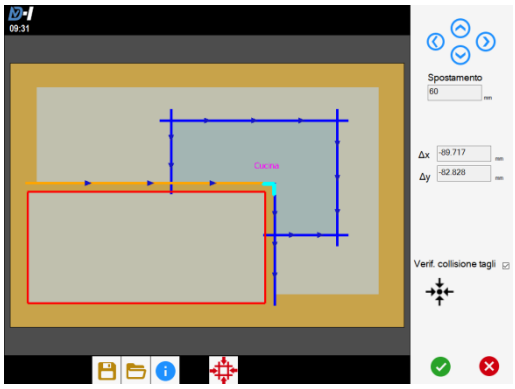
It is possible to use **automatic movement**  .
This means that the program automatically calculates which cuts are to be used for the required movement(s). This function does not always solve all the issues but it is possible to act in manual mode to improve the situation.
The program will attempt to solve the problem of pieces damaged by cuts with movements of the suction cups.

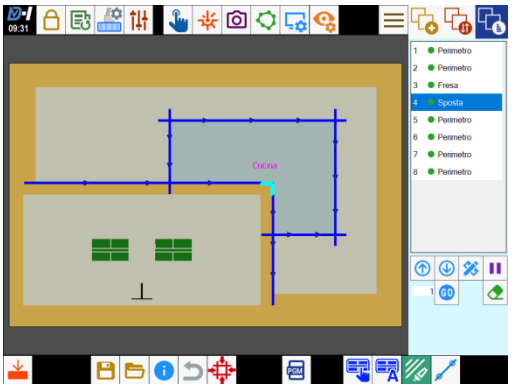
5.1.10.4 N-CUTS MODE

In N-Cuts mode it is possible to do the same thing described in the previous paragraph, but selecting more than one cut.

	**CAUTION:** *N-Cuts mode allows the lifting of pieces with a generic shape and therefore also L-pieces. The operator is responsible for checking the feasibility of the movement.*

To activate this mode, press the button  , select the cuts, then press the button 
Cut selection rules: cuts can be made by either blade (straight lines only) or milling cutter. The first and the last cuts must be blade cuts.
The program extends cuts with blade until it finds a solution allowing the slab to be split. Milling cuts on the other hand are never modified.





In the example in the figure, milling is required on the internal corner in order for the piece to detach.

Graphic interface

After having applied the N-Cuts function, the program shows a preview of the division, showing the displaced slab within a red perimeter.
On the right we find an interface allowing the displacement to be modified.
The slab can be moved using the arrows and by setting the value of the desired displacement.
The possibility is given to directly enter the absolute values of X and Y with respect to the initial position, or to drag the selected slab directly onto the screen.
After pressing the green button to confirm, the software runs a check to make sure the slabs and cuts are not overlapping. If “Check for cut collision” is deselected, the software only checks for slab overlapping.

The button  brings the displaced slab back to the initial position.
When the operator has identified the desired displacement, the selection must be confirmed using the green button below, or cancelled using the red button.

Operation Not Possible message

If the message “It is not possible to divide the slab with the selected cut” appears, the causes may be the following:

1. The selected cuts are not sufficient to create a path that divides the slab in two.
2. The selected cuts intersect instead of being contiguous.
3. The selected cuts cannot detach the piece due to the blade avoidance area.
4. The length of the selected cuts will ruin other pieces.

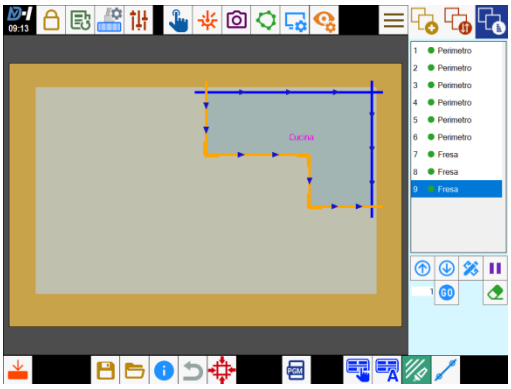
Check for non-overlapping slabs and cuts

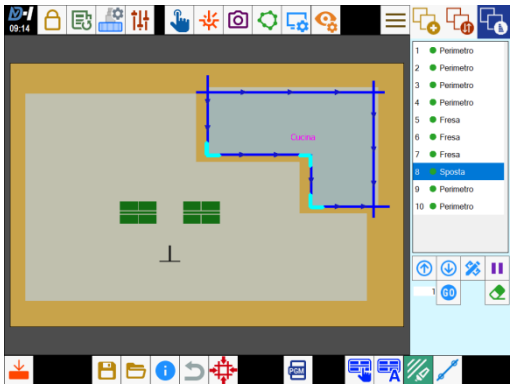
If the program finds that the displaced slab will overlap with another slab, or the cuts to be made will overlap with other slabs, it automatically adds an additional displacement in order to prevent such overlapping.

	**CAUTION:** *the operator is responsible for checking that the movement proposed by the software is feasible.*

The user parameters include the “N-Cuts Suction Cup Mode” option. If “manual” is selected, the machine does not automatically modify cuts made with the blade.

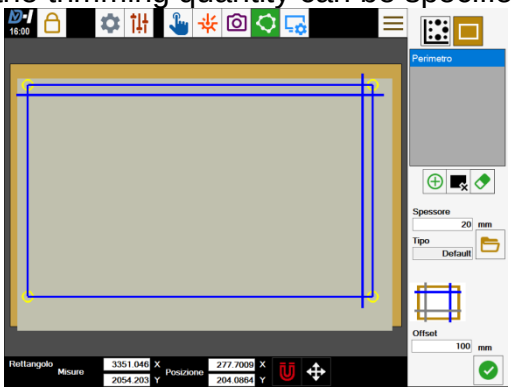
To avert blade avoidance areas in the leftover material, milling can also be performed on the external corners and cuts can be shortened using the “Modify Cut” function.



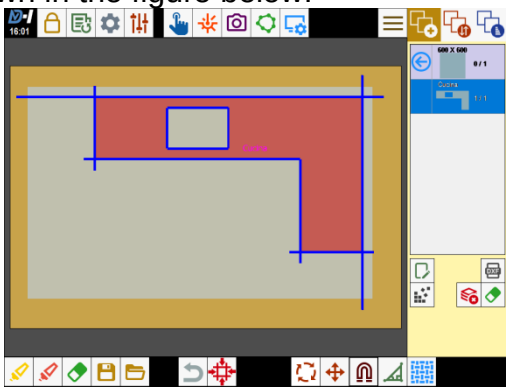


5.1.10.5 SLAB TRIMMING

Slab trimming is a feature that forces trimming cuts into the machining process. The purpose of these cuts is to trim the slab along its edges.
The feature can be activated in the perimeter selection panel. In this screen we find the button  which shows the four cuts that can be selected, one for each side of the slab. After selecting the cut, the side will be highlighted on the button. Under the button we find the Offset field, where the trimming quantity can be specified.

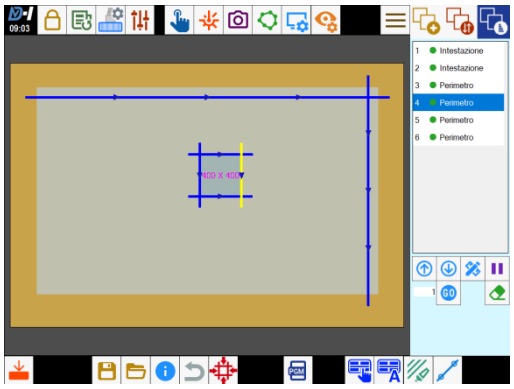


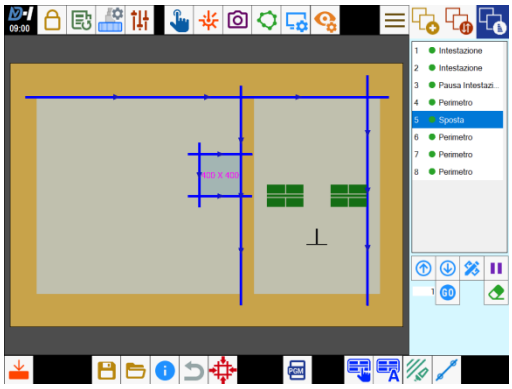
Upon confirmation, the slab is trimmed. It is possible to insert pieces and magnetise them to the trimming cuts as shown in the figure below.



Multiple slabs can be trimmed in different ways. The slab can only be rectangular, trimming does not work on perimeters formed by points.

In the list of cuts, trimming cuts are always at the start. If a displacement is inserted, a trimming pause is automatically inserted. During the trimming pause, trimming cuts can be removed in order that all displacements are carried out without being obstructed by the trimming cuts.

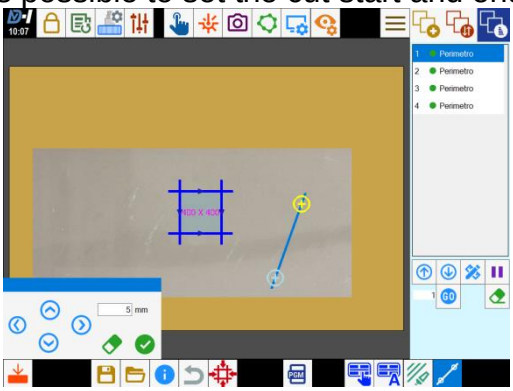




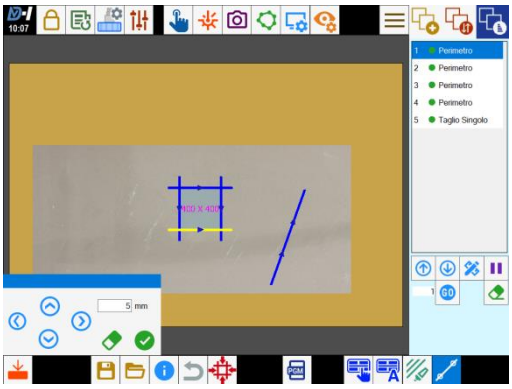
5.1.10.6 SINGLE CUT FROM PARAMETRIX

The single cut is a function allowing cuts to be inserted in the list of cuts window.

The feature can be activated in the third tab using the button  , found in the bottom bar. When the button is pressed, a functions panel appears. From that moment on, when the work area is clicked, it is possible to set the cut start and end points.



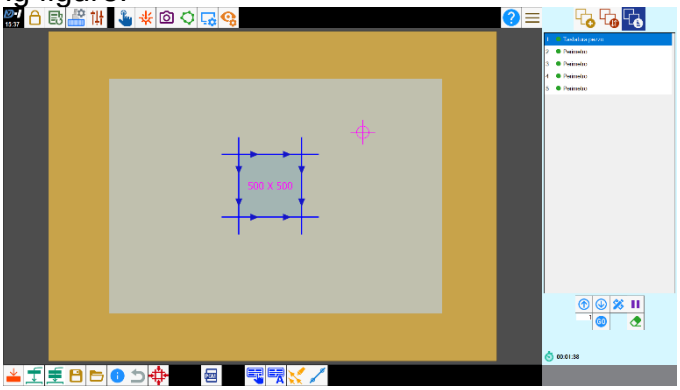
Single points can be selected and moved using either the drag function or the arrows in the relative panel. The displacement value using the arrow is set in this field.
When positioning is complete, the preview can be confirmed using the green button or rejected using the red button. If the preview is confirmed, it becomes a cut for all intents and purposes. Once the preview has either been confirmed or removed, a new cut can be added.



To complete the additions, press the button 

5.1.10.7 THICKNESS GAUGE MANAGEMENT (OPTIONAL).

The slab gauging process can be entered on the Cuts page. To enter the operation press the button  then press a point on the slab. A fuchsia symbol will appear in the pressed point as in the following figure:



That will be the point where probing will be performed. The operation in inserted at the beginning of the cuts list.

5.1.10.8 CUT SPEED UP/SLOW DOWN

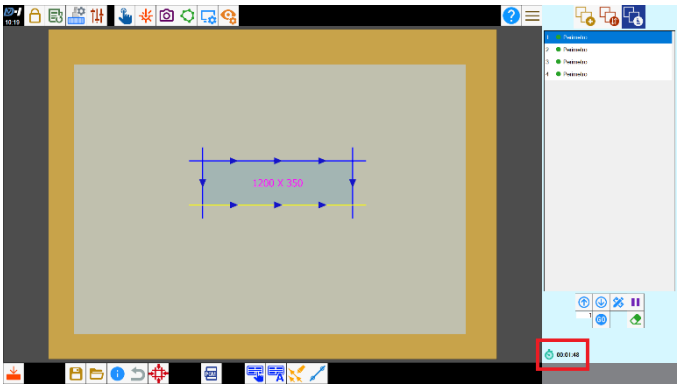
In user parameters there are two parameters for the management of accelerated cuts:

- Speed up cuts
- Slow down cuts

The acceleration is linear from a minimum speed; the user can manage the distance required for the tool to run at full speed. If Speed up cuts and Slow down cuts are at 0, the cuts are made normally.

5.1.10.9 WORK TIME SIMULATION

On the Cuts page, the estimated cutting time is shown on the bottom end of the page.

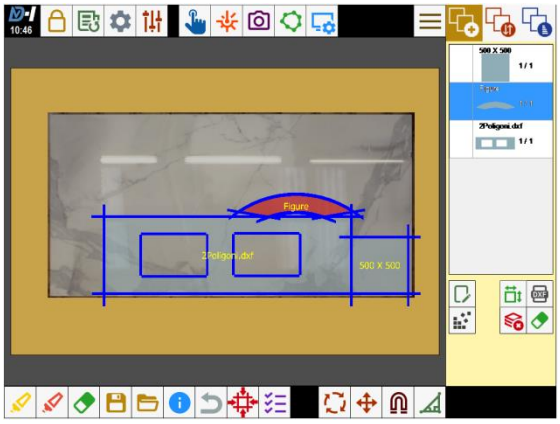


5.1.11 OPTIONALS AND OTHER FUNCTIONS

Some optionals and functions are described in this chapter

5.1.11.1 NESTING FUNCTION

The Nesting function is used to automatically arrange the pieces within the perimeter of the slab. To use this function it is necessary to create a list of pieces and to delimit the perimeter on which to insert them. Then press the button  . The program will arrange the pieces inside the perimeter.



The Nesting function arranges the pieces following various insertion logics. For this reason, each time the  button is pressed, the program changes the arrangement of the pieces, up to a maximum of 4 solutions (upon the fifth attempt, the first solution is reproposed). The software arranges the workpieces in 4 different ways in order for the user to compare and decide the best arrangement.

5.1.11.2 OPEN MARK

Through the Open Mark function it is possible to see, in real time, the background that will take the pieces to the original position in which they were designed, moving them on the workbench over a slab.

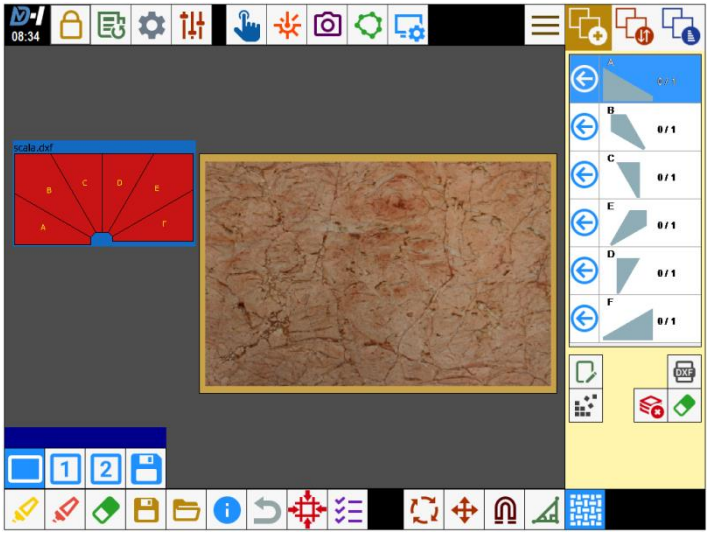
Proceed as follows:

- Import the DXF file to be used
- Select a piece from the piece list visible on the right of the program

The program will display a preview of the open mark relative to the selected piece and to all the pieces present in the same DXF file

Note: The Open Mark function only works for workpieces imported from a DXF file.

The screen below appears after pressing  to enable the *Open mark* function.



A preview of the open mark is given on the left of the workbench. In this case, all the workpieces coming from the “scala.dxf” file are displayed in the exact position in which they were designed. The size of the preview is in a 1:1 scale to the size of the workpieces and the workpieces appear on a red background to indicate that they have not yet been added to the workbench or that they have not yet been machined.

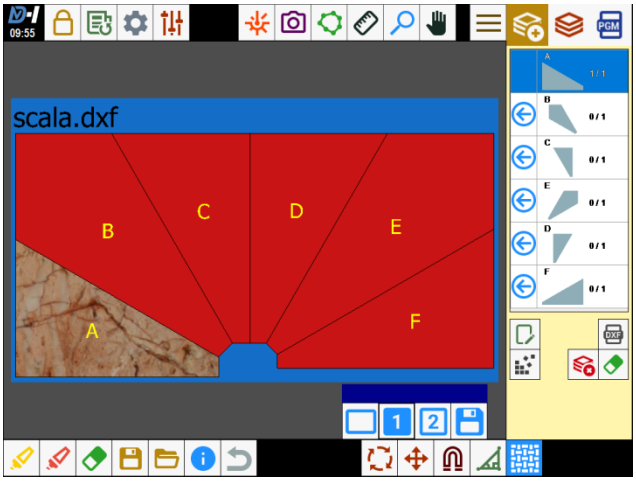
From the Preview it is possible:

- Add/Select a piece on the work area by clicking inside the piece to be modified
- Move a piece directly on the workbench
- Delete a piece from the preview by selecting the rubber present under the list of pieces and then selecting the piece on the preview to be deleted (**Note**: The piece will also be eliminated from the bench and from the parts list).
- Change the background colour of the preview (in this case blue)
- Remove the perimeter display

The Open Mark function also brings up the following panel:

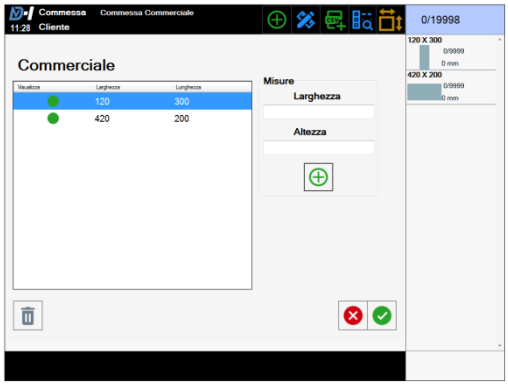
Side view: Default view as soon as the "Open Mark" function is activated. It provides an immediate view of the background that the workpiece will have by moving it onto the workbench.	
Preview Display: view focused on the preview. In this way, the workpiece will be positioned in the desired position with greater precision.	
Bench View: view of the workbench. The workpieces are moved on the workbench using the whole screen. The preview will be updated in real time.	
Save: to save the **Preview** in a folder already set up. Saving is possible at any time.	

The figure below shows an example of the **Preview Display**



5.1.11.3 COMMERCIAL JOB ORDER

Job order in which there are pieces that can be used to fill the empty areas of the slab, not used by the main pieces; in the upper bar the button  will be visible and pressing it the following screen will be visible:



Explanation of the buttons

1.  It allows Enabling/Disabling of the use of a particular piece
2.  and  simply allow closing of this screen
3.  It allows cancelling of a piece

 It allows adding of a piece with the values expressed in the fields

5.1.11.3.1 "ENABLE/DISABLE JOB ORDER" IN CERTAIN MACHINING PROCESSES.

During certain machining processes the workpieces of the "Job Order" may not need to be considered, while in others they may. To avoid this problem, the button  is featured on the home page. When the button is green, the job order can be used during the machining processes; if it is red (), commercial workpieces cannot be used until it is enabled again.

5.1.11.3.2 MODIFICATION OF WORKPIECE SIZE FROM WORK AREA

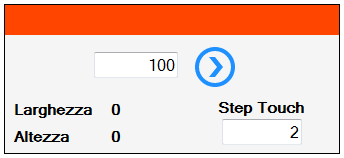
It can only be used if the button  is present, in the lower bar of the first Parametrix screen. This feature allows changing of the size of the ***RECTANGLES*** created as "commercial pieces" from the work area (Paragraph "Sales Order").
There are two ways to increase/decrease the size, but both comply with these characteristics:

1. **Selection of the side** of the piece to be modified, directly from the work area
2. As soon as workpiece modification is completed, a new workpiece will appear in the list on the right-hand side, including the dimensions of the workpiece that has just been modified.
3. If a new machining process is carried out  , these workpieces are deleted.
4. A function is featured that controls potential collision with the other workpieces.
5. It is possible to modify only one piece at a time

6. The **height** and **width** values indicate the dimensions of the currently selected piece

Modification mode:

1. From panel



a. In the box, where “100” currently appears, enter the value by which the piece must be enlarged/decreased with respect to that side

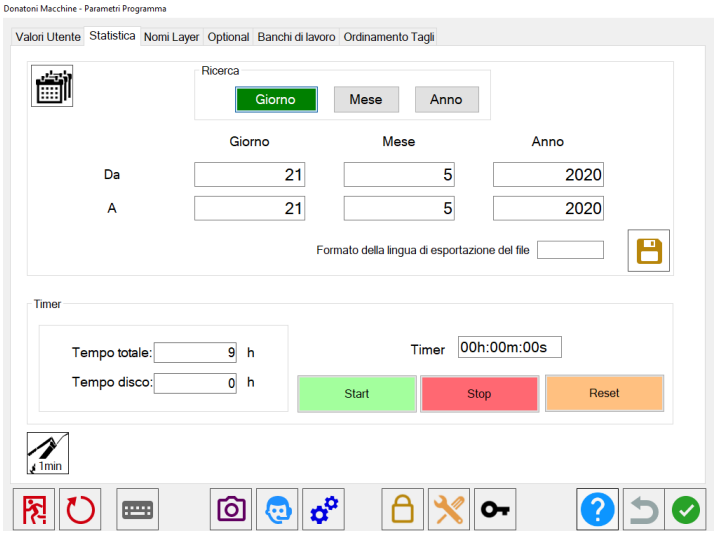
b. Press the button to apply the piece change

2. From touch

a. Select the side to be changed and drag it to the desired direction. Each movement has the same size as the "Step Touch" value visible in the panel. To cover a triple distance (e.g.) with respect to the one reported by the "Step Touch", simply continue dragging the side until reaching the desired size (visible under the heading **width** and **height** of the panel).

5.1.12 STATISTICS

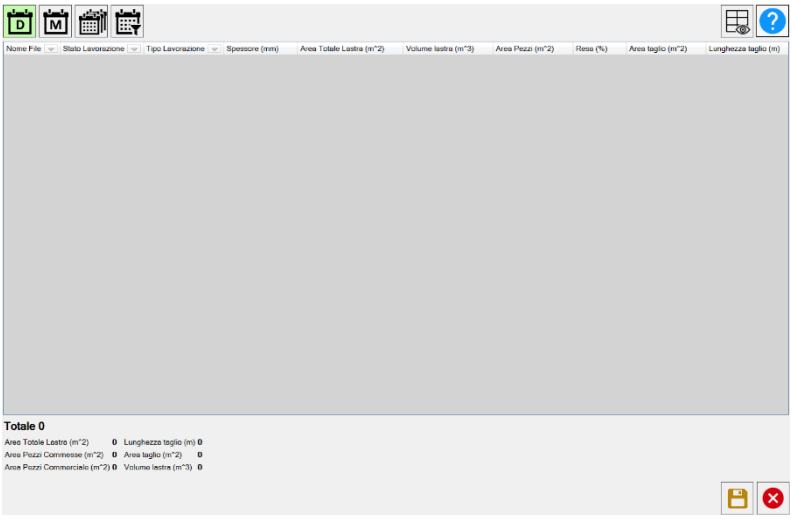
Regarding statistics, there is the possibility of having a temporal statistic rather than for the individual job order. To obtain then, press from the initial page of Parametrix and go to the "Statistics" tab, shown below



There is the possibility of entering the start and end date to define the time range relating to the statistics. Then pressing the "save" button will generate a csv file containing the statistical information. The name of the file with which these statistics will be saved will consist of: Today's date + hour, minutes, seconds when the file was created.

For daily statistics, press  from the initial Parametrix page and open the “Optionals” tab, enable the “View statistics” button. This will allow the automatic generation of daily statistics.
Statistics also include Single Cut, Wheel milling, Levelling, ISO processes.

To view statistics on screen, press the  button and the following window will open up.



The window shows statistics in table form. Below is the meaning of the button:

	It shows daily statistics
	It shows monthly statistics
	It shows all statistics
	It shows statistics in a time interval that can be set
	It allows the selection of the columns to be displayed
	It saves statistics on a file

The bottom left of the screen displays summation data for selected statistics such as:

- Number of the statistics displayed
- Slab total area
- Job order pieces area
- Commercial pieces area
- Cut length
- Cut area
- Slab volume

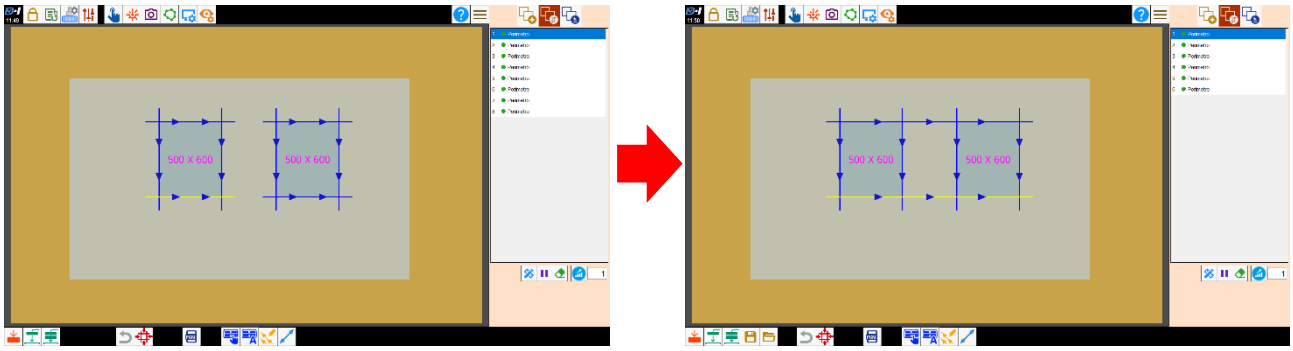
5.1.13 SX CONTROL AND OPTIMISATION

Below is a description of the different ways to optimize the cuts.
This function is useful to put the cuts together and make them with a lower pitch using the SX3/5 machine.

To set the desired mode, press the button  after entering the mode number in the field next to the button.

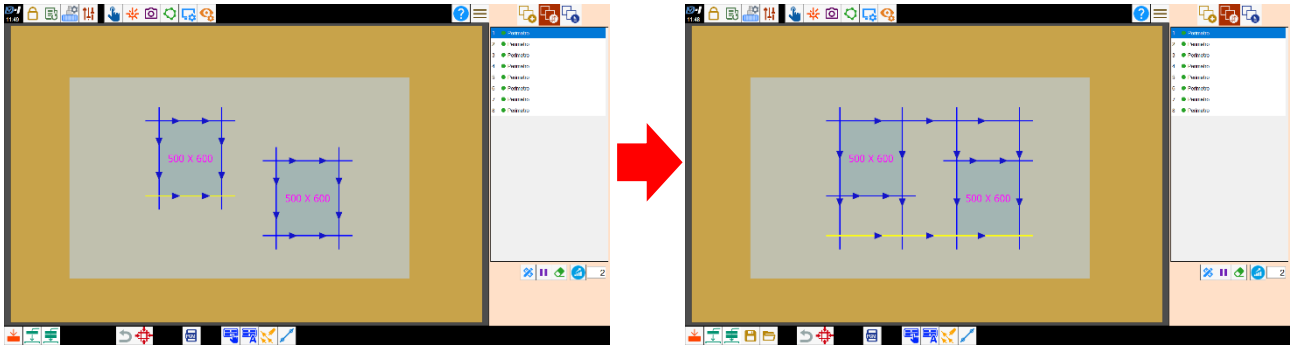
5.1.13.1 MODE 1

Two different cuts on the same straight line become one cut



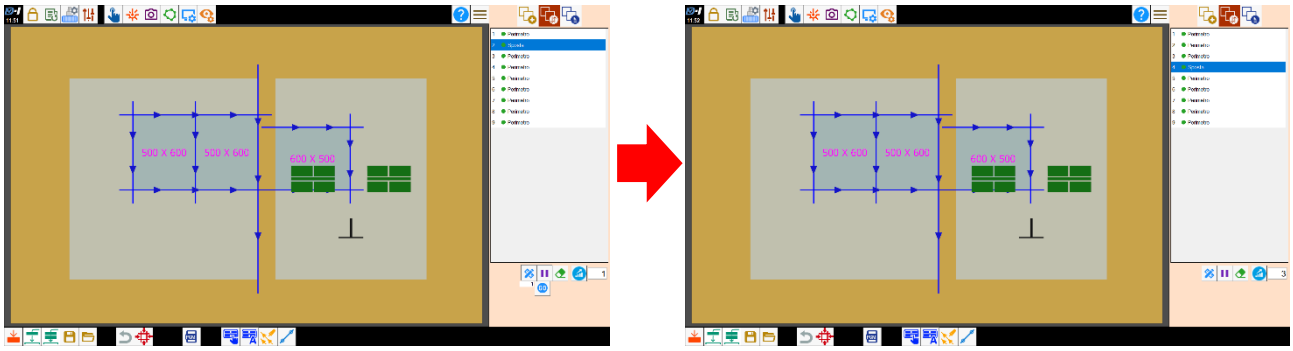
5.1.13.2 MODE 2

Try to make the whole cut the same length (maximum).



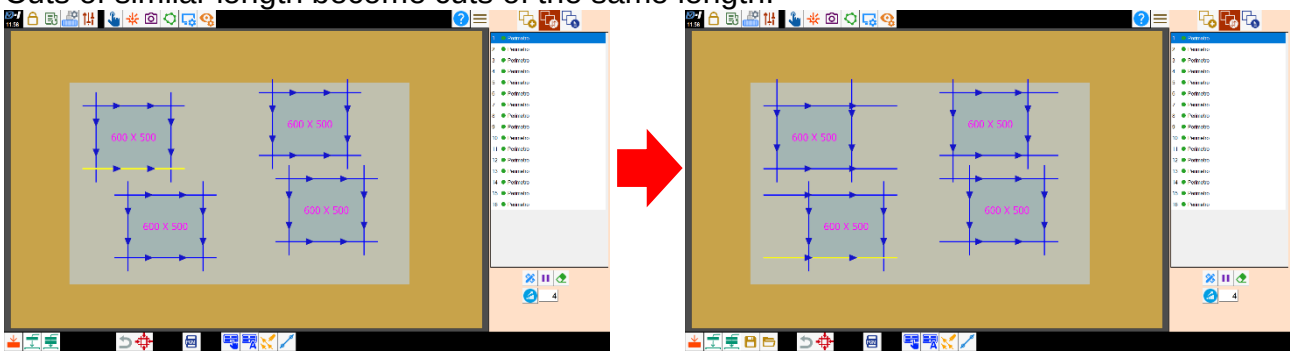
5.1.13.3 MODE 3

With mode 3 the software performs all the cuts it can before moving.



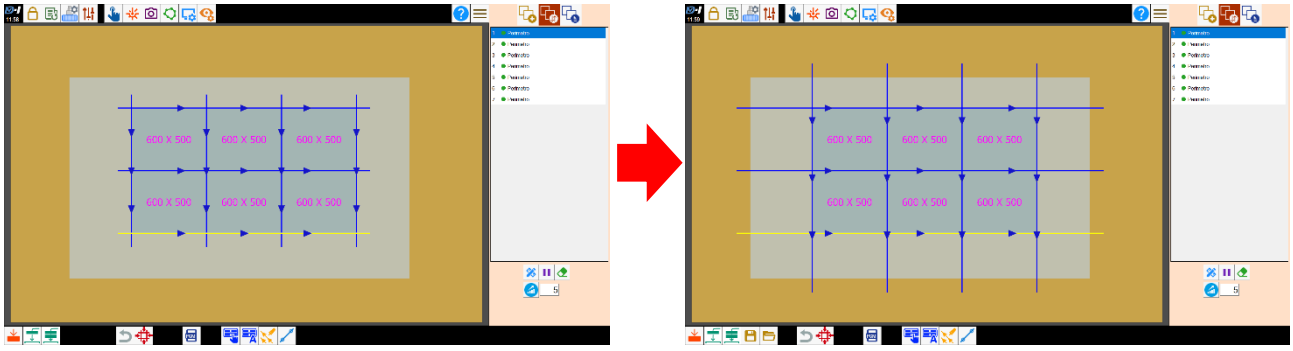
5.1.13.4 MODE 4

Cuts of similar length become cuts of the same length.



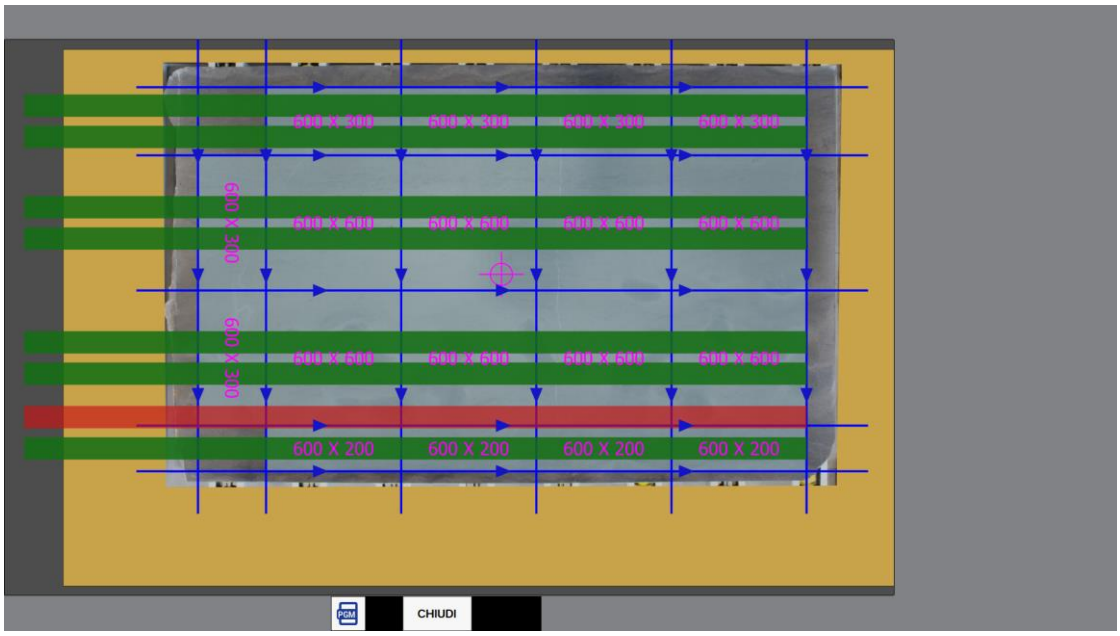
5.1.13.5 MODE 5

It extends all cuts beyond the edge of the slab.



5.1.14 UNLOADING SYSTEM (OPTIONAL)

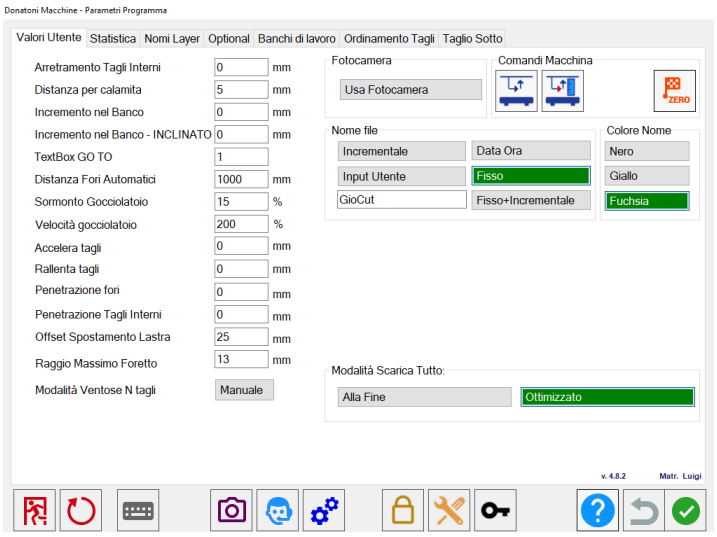
The strip unloading system is controlled by the software which creates an unloading program to position the rows of workpieces on the relevant belt conveyors.



The software checks that the slab is divided into horizontal rows (with a tolerance of a few degrees on the direction of the cuts), that the dimensions of the pieces are within the limits and sets the order of movements and the row of suction cups to be used.
Movements with suction cups to the right or left, intersecting parts and other similar conditions cancel the creation of the program which will not be unloaded automatically.
The parameters are set by the technician during machine start-up and remain fixed.
You can check the gripping positions and the areas used by pressing the "Show unloading" button. The gripping areas are displayed in green and the non-gripping areas are displayed in red.

5.1.15 USER VALUES

The user values page is accessed by pressing the button .



Below is a description of the individual fields.

Internal cut withdrawal	Minimum approach distance between the blade and the workpiece for internal cuts
Distance for magnet	Distance between two pieces before the magnet gets activated
Increase in the bench	Tool breakthrough depth in the bench
Increment in the bench - Inclined	For isoparametric mode only.
TextBox GO TO	Default value for GOTO to sort lists
Distance between automatic holes	It indicates the distance between one hole and another when inserting multiple holes.
Drip overlap	Percentage of drip overlap, 99% to 0%
Drip speed	Percentage of drip speed compared to the speed of normal cuts
Speed up cuts	Acceleration distance from start
Slow down cuts	Deceleration distance from the end
Hole penetration	It sets the penetration of all holes, set to 0 for through-hole
Penetration of internal cuts	It sets penetration of all internal cuts, set to 0 for through-cut
Slab displacement offset	Additional value for the move function
Hollow bit maximum radius	For importing the dxf, maximum radius for a circle to be imported as a hole
Suction cup N cuts mode	Manual mode ON: the user must manually lengthen the cuts on the slab. Manual mode OFF: it automatically lengthens the cuts
Tool probing at the beginning of the program	Flag ON: it performs tool probing at the beginning of the program
Use camera	Image of the slab from camera or file
	It unlocks bench height acquisition and deactivates the safety control

	It sets the current z value of the machine as the bench value
	The machine performs the axis zero cycle
Incremental	It increases the iso filename as follows: 100, 101, 102...
Date Time	Name of the iso file in date-time format, e.g. 20191206_080124
User input	The user can enter the file name
Fixed	Fixed iso file name; enter the desired name in the adjacent field.
Fixed + incremental	Name of the iso file in incremental-fixed format, e.g. File_101
Black	Black piece text
Yellow	Yellow piece text
Fuchsia	Fuchsia piece text
At the end	"Unload everything" mode with the piece being unloaded at the end of the machining process
Optimized	"Unload everything" mode with the piece being unloaded when it is completely cut
	It closes the Parametrix software
	It restarts the Parametrix software
	It opens the Windows keyboard
	It opens the management software of the camera
	It opens the connection for remote assistance
	It opens a software for viewing cnc/plc registers

	It opens the page of technical parameters
	Tools for tests on the machine
	Entering the password
	Activation of the Help mode. A question mark appears on each button; the relative information is displayed when pressing it.
	You leave the page without saving changes
	Changes are applied and the page is exited.

6 REMOTE SUPPORT

Donatoni macchine S.r.l. makes a remote support service available in order to carry out maintenance at software level by means of an internet connection with the machine.
In order to make the connection it is necessary to call the Donatoni Macchine support service and follow the indicated procedure:

1. Call the Donatoni Macchine S.r.l. support service.
2. Open the Reserved Area page in the software on board the machine
3. Check that there is an internet connection to the machine (USB pen drives in the end user's possession), network connection by router (optional)
4. Press the Tele-servicing button and wait until the Donatoni Macchine S.r.l. support program starts.

5. Provide the User ID and password to the connected engineer and wait for instructions

From this moment until the conclusion of the tele-servicing session, the Donatoni engineer is connected to the machine and takes remote control of it.