Live out your dreams

DONATONI
Stone Tech Creators

SECTION B

Software manual

Doc.Rev. 3.2 – SW Vers. 4.8.2

TRANSLATION OF THE 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_sw4.8.2_rd3.2_EN

Date: 10/12/2024

Care has been taken to ensure that the information contained in this manual is accurate and complete. 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	*Technical Office*	*Assistance*
Tel. 045 68 62 548 *Fax 045 68 84 347*	*Tel. 045 68 88 604*	*Tel. 045 68 62 899*

Email address: info@donatonimacchine.eu

List of updates

Doc. Rev.	SW Vers.	Description	Approved by	Date	Page
--	1.015	First issue	N. Chesini	--	All
--	2.01	--	N. Chesini	--	--
--	2.0	--	N. Chesini	--	--
--	3.0	--	N. Chesini	--	--
--	4.0	--	N. Chesini	30/01/2018	--
1.0	4.0	-Reference to "Original Instructions" inserted on the first page -Email hypertext links inserted -Company address inserted for communication of any changes to be made to the manual -Reference to the manual's reproduction rights inserted -List of updates inserted and document drafting table inserted in footer -Headers and footers modified -Insertion of chapter 1 - INTRODUCTION -Modification to chapter 2 - INFORMATION ON TECHNICAL ASSISTANCE -Document layout updated and optimised -Reordering of chapters/paragraphs -Format of chapters and paragraphs aligned with standards of other manuals -Removal of paragraph “Piece machining” in chapter 5.3 - SCANCNC	N. Chesini	16/03/2018	All

Doc. Rev.	SW Vers.	Description	Approved by	Date	Page
2.0	4.6	-New version of Parametrix 4.6.0.0 -Addition of Parametrix management with automatic tool change. Section 4.7.2 -Addition of paragraph 4.1.10.4 (N-Cuts mode) -Addition of paragraph 5.1.1.5 (Suction cup speed parameters) -Modification to section 4.1.2 (Tool Configuration) -Modification to section 5.1.8 (Work area: piece positioning and optionals) -Modification to section 5.1.1.8 -Modification to table in paragraph 5.1.1.3 -Addition of paragraph 4.7.3 (Tool synchronisation) -Modification to paragraph 5.1.1 (Working parameters) -Modification to paragraph 5.1.7.3.5 (Hole) -Modification to section 5.1.8.9 (Save and open) -Addition of paragraph 5.1.8.10 (Importing and Exporting) -Modification to section 5.1.10.1 (Modify cut) -Addition of paragraph 5.1.10.5 (Slab trimming) -Addition of paragraph 5.1.10.6 (Single cut from Parametrix) -Addition of paragraph 5.1.14 (Cut below) -Addition of paragraph 5.1.15 (Hole below) -Modification to section 5.1.19 (Statistics)	N. Chesini	10/05/2019	All

Doc. Rev.	SW Vers.	Description	Approved by	Date	Page
3.0	4.8.2	-New version of Parametrix 4.8.2.0 -Modification to paragraph 3.5 (Hardware and software safety devices) except 3.5.6 (Mushroom-head emergency push-button) -Modification to paragraphs 3.7 and 3.8 -Paragraph 4.7.3 (Tool synchronisation) moved to 4.1.2.2 (Tool synchronisation) -Modification to paragraph 4.2 (Manual mode page) -Modification to paragraph 4.2.1 (Manual control of suction cups (Optional)) -Modification to paragraph 4.4.3 (Graphics) -Modification to paragraph 4.7 (Basic optionals) -Addition of paragraph 4.7.3 (Belt) -Addition of paragraph 4.7.4 (Lathe) -Addition of paragraph 5.1.3.1 (Mapascan integration) -Addition of paragraph 5.1.4.5 (Buttons to hide entities) -Modification to paragraph 5.1.5 (Piece creation) -Addition of paragraph 5.1.5.5 (History) -Addition of paragraph 5.1.5.6 (Favourites) -Addition of paragraph 5.1.5.7 (Modify shape) -Modification to paragraph 5.1.7 (Modify pieces) -Modification to paragraph 5.1.7.2 (Optionals) -Addition of paragraph 5.1.7.3.9 (Cut below) -Modification to paragraph 5.1.8 (Work area: piece positioning and optionals) -Modification to paragraph 5.1.8.8 (Delete pieces) -Addition of paragraph 5.1.8.9.1 (Slab matching) -Addition of paragraph 5.1.8.9.2 (Slab smith) -Modification to paragraph 5.1.8.13 (Machine on cut) -Addition of paragraph 5.1.8.14 (Complete nesting) -Addition of paragraph 5.1.10.1 (Multiple selection of cuts) -Addition of paragraph 5.1.10.8 (Touch probe control)	N. Chesini	10/06/2020	All

Doc. Rev.	SW Vers.	Description	Approved by	Date	Page
3.0	4.8.2	-Addition of paragraph 5.1.10.9 (Cut speed up/slow down) -Addition of paragraph 5.1.10.10 (Work times simulation) (Management of job orders) -Addition of paragraph 5.1.12.2.1 ("Unload everything" mode) -Modification to paragraph 5.1.13 (Statistics) -Addition of paragraph 5.1.12 (Optionals and other functions) -Paragraphs 5.1.12 (Nesting function)-5.1.16 (Open book) and 5.1.18 (Commercial job order) moved to 5.1.12 (Optionals and other functions) -Deletion of paragraph 5.1.17 -Addition of paragraph 5.1.14 (Left side control and optimisation) -Addition of paragraph 5.1.14.1 (Mode 1) -Addition of paragraph 5.1.14.2 (Mode 2) -Addition of paragraph 5.1.14.3 (Mode 3) -Addition of paragraph 5.1.14.4 (Mode 4) -Addition of paragraph 5.1.14.5 (Mode 5) -Addition of paragraph 5.1.14.6 (Unloading device) -Addition of paragraph 5.1.16 (User values) -Modification to paragraph 5.2 (Multiple cuts) -Modification to paragraph 5.2.2 (Side bar) -Modification to paragraph 5.2.4 (Head rotation confirmation) -Modification to paragraph 5.2.5 (Deletion of cuts) -Modification to paragraph 5.2.6 (Update cuts) -Modification to paragraph 5.2.7 (+/-90) -Modification to paragraph 5.2.8 (Execution of cuts)	N. Chesini	10/06/2020	All
3.1	4.8.2	- Removed paragraph 3.8 “Remote unit for manual positioning” in chapter 3 – On board the machine	N. Chesini	01/12/2023	-
3.2	4.8.2	- Chapter "ISOSAG" removed	N. Chesini	10/12/2024	-

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; in these cases, however Donatoni Macchine is not obliged to guarantee these functions or to 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 necessary for proper knowledge of the Parametrix programme and useful information in order 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 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 leave the manual during use and to ensure it is returned to the assigned place immediately following its consultation;
- do not remove, tear or rewrite for any reason parts of the Manual;
- 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:

- The machine operator
- 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	*Technical Office*	*Assistance*
Tel. 045 68 62 548 *Fax 045 68 84 347*	*Tel. 045 68 88 604*	*Tel. 045 68 62 899*

E-mail addresses: info@donatonimacchine.eu

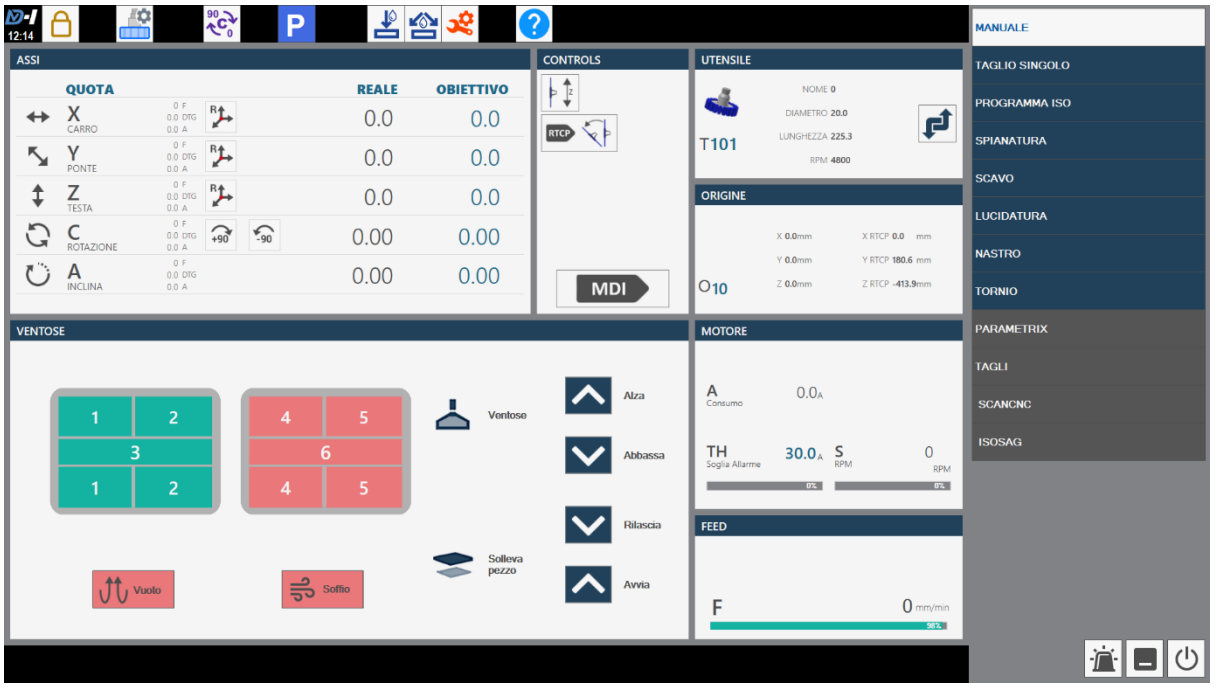
In the case of faults or malfunctions that cannot be resolved, the user can contact the manufacturer directly.

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 and:
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.

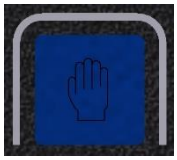
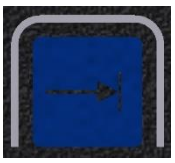
Once shutdown is confirmed wait until the monitor is completely black and move the switch on the electrical panel to 0.

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 the X and Y axes is adjusted by independent potentiometers while the “INTERP. AXES” potentiometer is used to control the speeds of the Z, A and C axes.
The lathe (OPTIONAL) can be controlled both using selector switch and by activating rotation using the button. The speed is controlled by the "LATHE" potentiometer.

“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. N.B. 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 also resets the active alarms. N.B. 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 switches the "Cut marker" laser on/off.	
"Cross laser" button: pressing this switches the cross 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

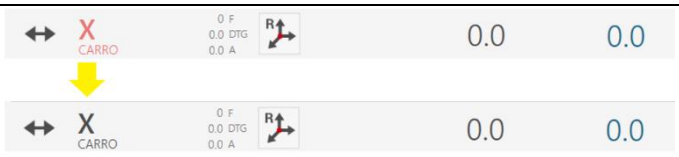
Starting a work cycle
Starting an automatic work cycle is subordinate to the selection by an operator of one of the automatic enabling buttons. The next stage envisages turning on the water (not obligatory) and the disc or tool (obligatory) and enabling the automatic work cycle by means of the CYCLE 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

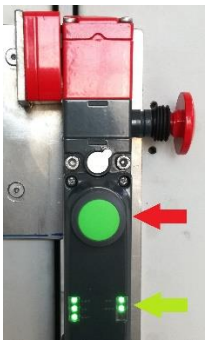
The Ampere Threshold parameter is worth a mention among the options available. This parameter is present in several interface pages and is to be used to protect the machine and the operator against faults in the tools or if wrong parameters have been set. If the spindle consumption exceeds the set threshold,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

ATTENTION: *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.
In order to reset the error, it is necessary to press the "Cycle stop" button for more than 5 consecutive seconds or carry out a movement with any axis to update the position values.

3.5.5 COLLISION LIMIT SWITCH

A dedicated sensor monitors the Z axis movement and notifies any collision by blocking all the axis movements, so as to prevent dangerous situations and possible damage.
After checking for the cause, to remove the error it is necessary to manually move upwards the Z axis which is the only movement enabled.

3.5.6 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 in the braking axes (Z, C and A) the brake is activated, while the others (X and Y) are free to move (unless in the case of mechanical impediments or motion transmission limitations).

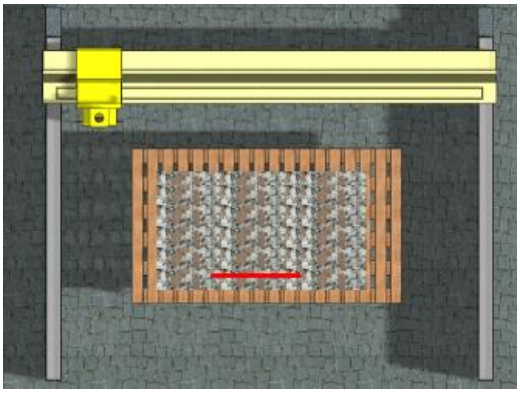
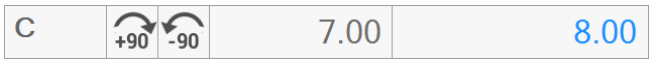
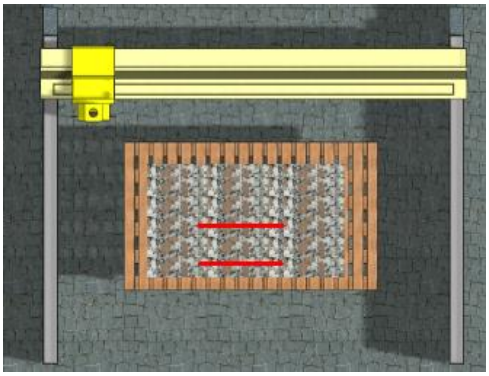
It is possible to resume work when the emergency has ceased. To do so, it is necessary to release the emergency button with a slight anti-clockwise rotation of the same and to re-enable the controls by means of the relevant "Enable controls" button on the door of the electrical panel.

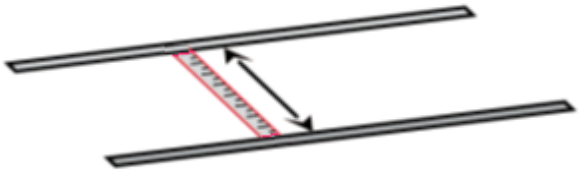
3.6 MACHINE GEOMETRIES

This chapter describes some of the measurement techniques for the tools, whether they are cutters or discs.
It should be remembered that the correct measurement of the tools (radius, length and thickness) is a fundamental operation for achieving correct machine behaviour and obtain pieces of the desired dimensions.

3.6.1 FRC (FLANGE ROTATION CENTRE) CALCULATION

The term FRC is intended as the distance from the inside edge of the disc tooth to the rotation axis C.

Procedure for measuring the FRC of the disc	
Make a longitudinal cut with respect to the X axis, while maintaining the rotation axis (C) and the inclination axis (A) at 0. C axis equals 0° A axis equals 0° The cut must be made as shown in the figure.	
At the final position of the cut just made, raise the "Z" axis in safety conditions and rotate the "C" axis 180 degrees. To do this, in the "Target level" field of the "C" axis set the required value, in this case 180. Then press the button MDI. It is fundamental for the other machine axes not to move; otherwise the measured FRC distance will not be correct or ONLY modify the field of the C axis.	
With the "C" axis at 180° make a new longitudinal cut compared with the "X" axis; this cut must be parallel to the first cut made (see the figure).	

Procedure for measuring the FRC of the disc	
Measure the distance between the two marks previously made, using a measuring instrument (ruler or gauge) (see figure). After measuring the length divide it by two. The result of the division is the value to set as the Flange Rotation Centre.	

3.6.2 TOOL LENGTH MEASUREMENT (OPTIONAL)

The tool length must be expressed in relation to axis of rotation C.
The length becomes essential for 5-axis machining operations or when using several tools with the same origin.
In order to facilitate the operator with the measurement operations, a measurement basis in which to insert the tool and on which to carry out the measurements is provided to those who have 5-axis CAD-CAM software available and an ISO coupling motor.

Some of the machine specifications are calibrated in line with the measurement basis provided. The length must be measured between the flat base of the cone-holder and tool tip as shown in the figure below. It is advisable to use a very precise tool and to ensure that the measuring rod is parallel to the tool.

3.7 MACHINE PLC ALARMS

"PLC" alarms are alarms that occur for reasons caused by the malfunction of the machine's hardware components. For example, activation of the motors, the pressure switch or even as a result of insufficient water or power failure to the machine.

Description of the alarm/message	**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	The Slow Jog mode can be activated by pressing the dedicated button and is also signalled by the steady orange light which remains on while this mode is active. 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 gets activated when any running automatic cycle is stopped by pressing the Stop cycle button. The message appears if the button is pressed for an instant; the program resumes execution from the point at which it stopped; otherwise if the Cycle Stop button is pressed constantly for more than 5 seconds the cycle is stopped completely.
Spindle start standby	this message is displayed while waiting for the spindle to reach the revolutions set in the parameters.
Disc 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.

Description of the alarm/message	**Causes**
Spindle extra current	It indicates that the threshold set in the parameters has been exceeded. Verify the cause and press start or disc 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, disconnect it from the power supply, wait 5 minutes and open the electric panel to check the condition of the safety device. Contact specialised personnel.
Disc 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, disconnect it from the power supply, wait 5 minutes and open the electric panel to check the condition of the safety device. 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.
Error of the drives	It indicates a power supply failure to the drives. Other specific messages may be displayed. N.B. Do not carry out any kind of operation with the machine, including resetting the axes. Restart the machine completely and wait a few minutes before turning it on again.

Description of the alarm/message	**Causes**
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 "spindle fan thermal relay alarm" is triggered when there is an electrical fault in the board. Solution: check that the remote switch in the board is on. If it continues to trip it must be checked.
Pressure 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	It indicates insufficient pressure at the machine inlet. 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.
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.

Description of the alarm/message	**Causes**
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 verify the positions in the warehouses.

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

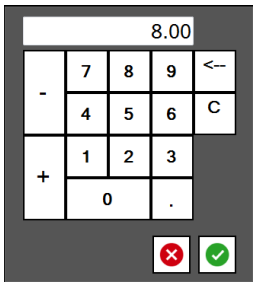
For further information or if you have questions about certain parts of subsequent explanations please contact Donatoni support.

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.

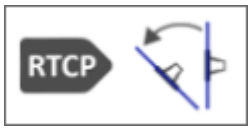
Operating procedure	
The operator, to set the displacement of one or more axes, must enter the value, with respect to the active origin, to be achieved in the box dedicated to each axis. To enter the values, keep the "target level" item pressed for the axis you wish to displace. A numerical keypad on which you can enter the new value will appear.  **Important:** A minus sign before the numerical data sets the subtraction from the value entered. 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.	
"MDI" button: when you press this button the machine will move to the target levels entered.	

WARNING: *The displacement of the axis does not include the thickness/diameter of the disc. 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 the respective potentiometers while the speed of the Z, C and A axes are controlled by the "interpolated" potentiometer.*

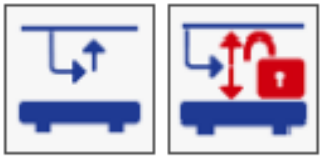
4.1.1.1 RTCP – ROTATION TOOL CENTRE POINT

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 disc 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"	A 11.00 12.00
When you press the RTCP button, the inclination or rotation function begins while the lowest point of the tool remains fixed. **N.B.:** Movement by means of the RTCP command is only permitted when the coded key is inserted in the appropriate station, identifiable on the operator console.	
Wait until the machine positions itself at the desired level.	

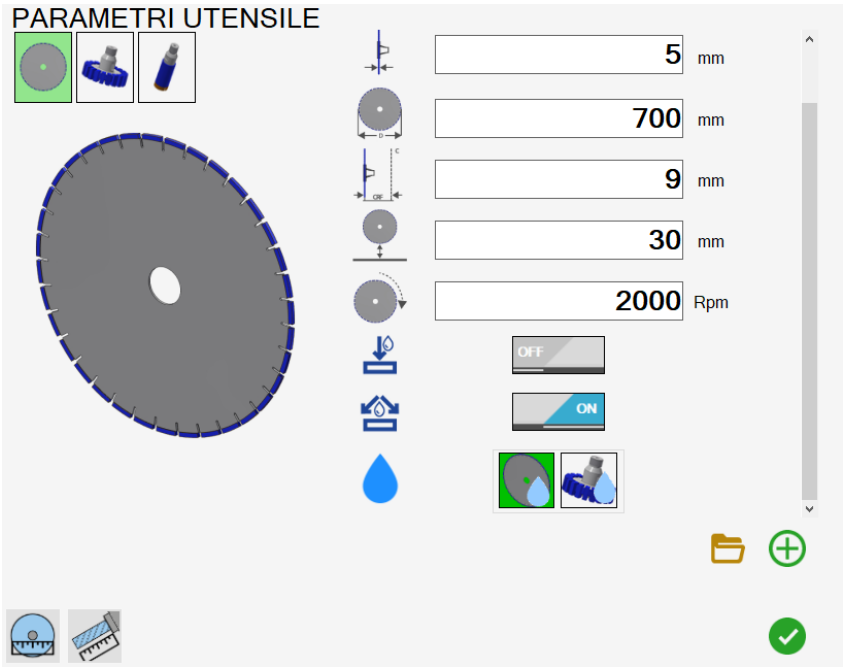
4.1.1.2 TOOL/BENCH SAFETY LOCK/RELEASE

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.

<table>
<tr><th colspan="2">Button Description</th></tr>
<tr><td colspan="2">Select the button  located in the upper left of the program</td></tr>
<tr><td>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.</td><td>  </td></tr>
<tr><td>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”.
  ATTENTION: The position found when this button is pressed is critical during automatic cuts.</td><td>  </td></tr>
</table>

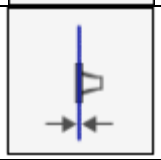
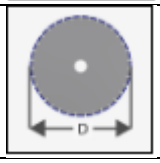
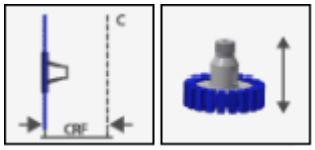
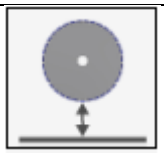
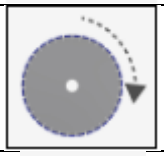
4.1.2 TOOL CONFIGURATION

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 for calculating the geometries, movement in RTCP or in interpolated mode and for the program on board the machine.

The button with the green cross allows the creation of a new tool, opening a creation window in which all data relative to the new tool can be entered. The newly created tool will not automatically replace the present tool, rather the open button must be pressed, and the new tool must be selected.

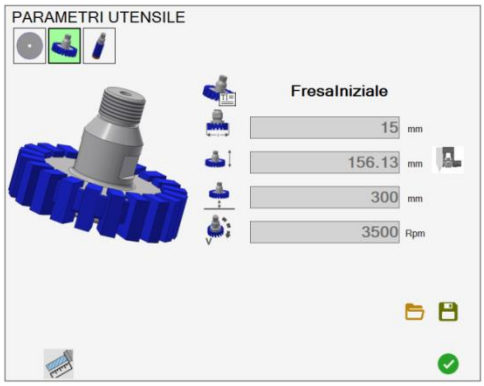
Description of the parameters	
<u>*DISC NAME (valid for all tools)*</u>: Name of the tool with which any parameters that are currently visible are associated.	
<u>*DISC THICKNESS (only valid with the disc)*</u>: It indicates the disc thickness measurement	
<u>*TOOL DIAMETER (valid for all tools)*</u>: It indicates the maximum tool diameter.	
<u>*CRF (only valid with the disc)*</u>: for calculation of the CRF, see the chapter "CRF Calculation" (*Chapter: On Board the Machine paragraph CRF Calculation*) <u>*TOOL LENGTH (only valid for tools)*</u>: it indicates the distance of the tool tip to a point taken as a reference. As this depends on the machine model, please contact Donatoni support for more information on this matter. For those who have the "*tool measuring kit*" option simply take the measurement by inserting the cone into the kit. In this case refer to the chapter TOOL LENGTH MEASUREMENT (OPTIONAL)	
<u>*MATERIAL RELEASE LEVEL*</u>: This is the safety height the machine takes into consideration when carrying out automatic programs.	
<u>*ROTATION SPEED*</u>: this indicates the maximum rotation speed of the spindle. This value is linked 100% to the disc control potentiometer.	
<u>*INTERNAL WATER*</u>: allows the enabling or disabling of internal water use during machining. For disc type tools, this will always be disabled.	
<u>*EXTERNAL WATER*</u>: allows the enabling or disabling of external water use during machining. For disc type tools, this will always be enabled.	
<u>*EXTERNAL WATER TYPE*</u>: used to select the type of external water to be used during machining: • <u>*Disc type*</u> • <u>*Nozzle type*</u>	

TOOL TYPE: it is possible to select the "disc type" or the "cutter type". The type of tool selected affects the spindle tool acceleration and the interpolation mode. The "interpolated Z" movement is made by following the blade direction in the case of "disc type" while it is made along the spindle axis direction in the "cutter type" case.

4.1.2.1 CHOICE OF SECOND TOOL (OPTIONAL)

The machine may be able to use two tools simultaneously. It therefore becomes necessary to be able to decide which will be the second tool used in addition to the disc, therefore either the cutter or the hollow bit.
To choose which tool to use it is necessary to enable a button that is only found in the screens that relate to the cutter and the hollow bit, that is  .

Hollow Bit page



Cutter page

As shown in the images, the second tool mounted on the machine is considered as the hollow bit and not the cutter. In fact, when a tool is not considered to be mounted, the screen is shown with all the parameters greyed out.

To change this setting, simply select the button  on the cutter page resulting in the reverse situation, i.e. the machine will consider the cutter tool fitted and the hollow bit will have all the parameters greyed out.

4.1.2.2 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:

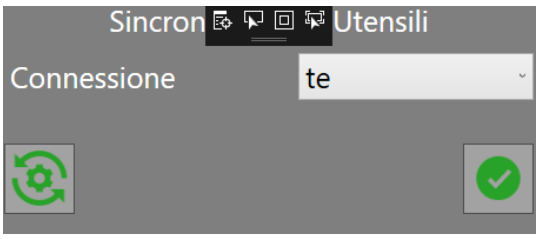
The three buttons allow:

- 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 connection field, select the on board machine computer from which to copy the tools. Then press button to synchronise.

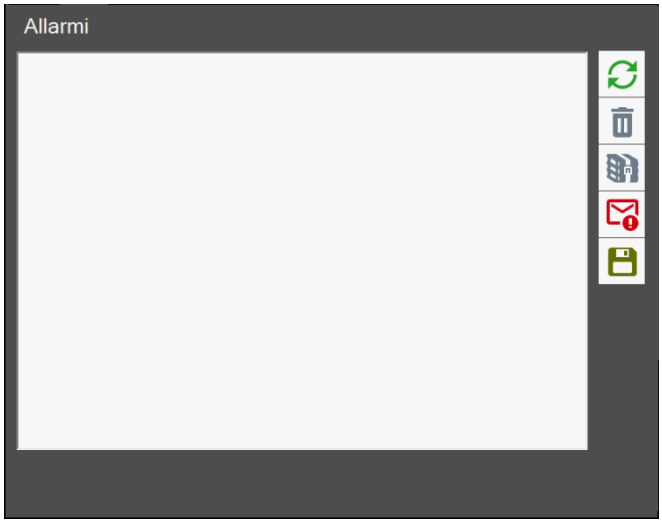
The synchronisation function works differently depending on the type of machine:

- 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

Attention: the procedure may overwrite one or more tools

ALARMS

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

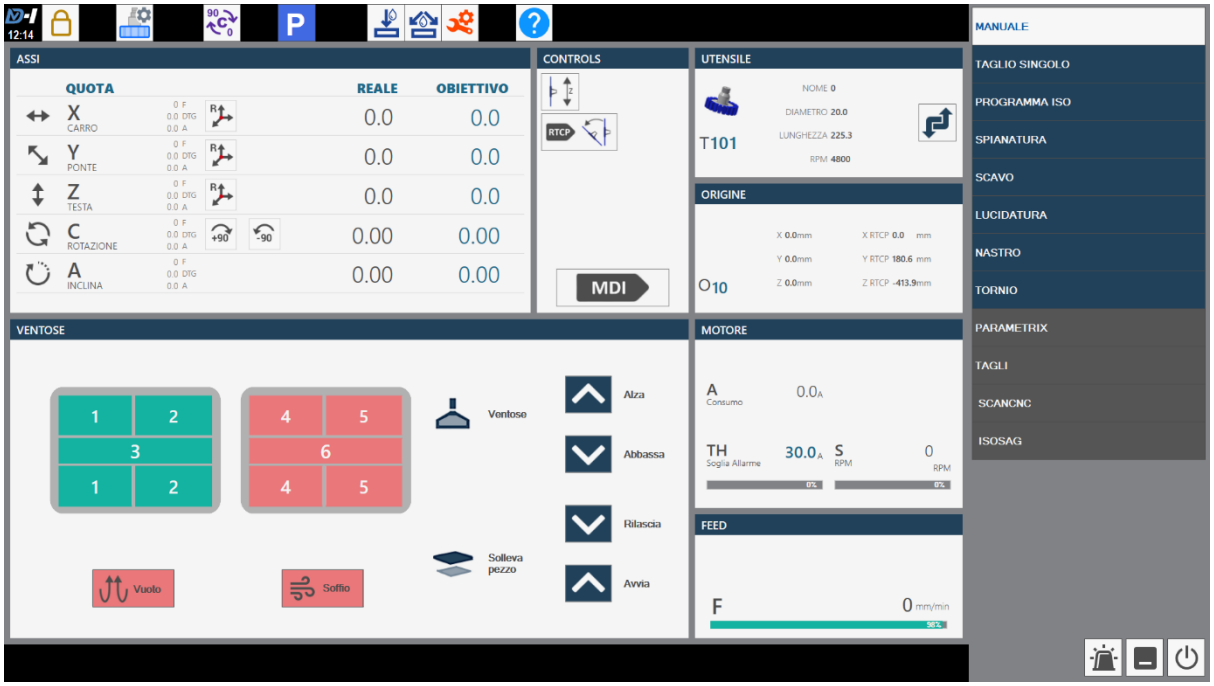


The alarms page indicates the Alarms/Messages that are active in the system.
If no alarms are active in the machine, the page will appear empty otherwise there will be a line for each error present. This line consists of: error number, a brief description, the date and time of the event.

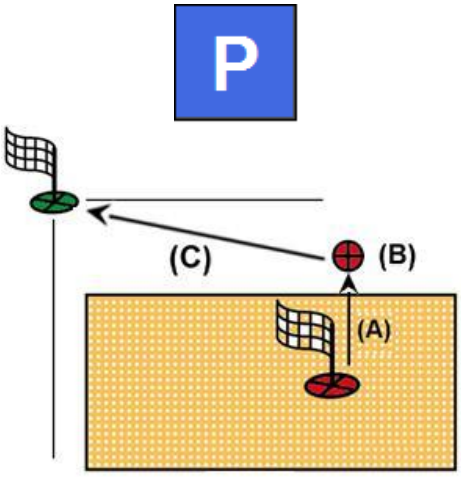
Function	
The "Refresh" button is for updating the machine alarm status.	
"Delete" is for eliminating the list of historical alarms present.	
This shows a historical report of the alarms signalled by the machine. It is useful for the engineers from the Donatoni Engineering Workshop to trace a timeline 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.	

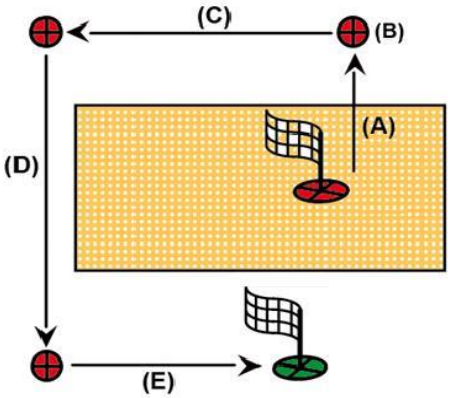
To exit the page select a list item visible on the right side of this page.

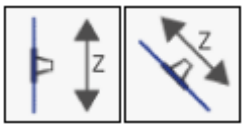
4.2 MANUAL MODE PAGE

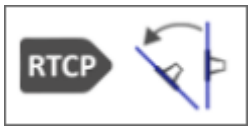
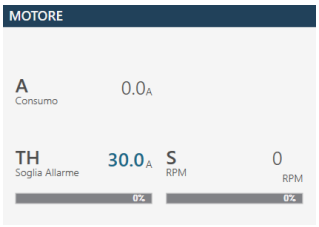


ATTENTION: *Before the axis zeroing function it is essential to pay utmost attention when moving the machine. Otherwise mechanical collisions could occur*

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.*	

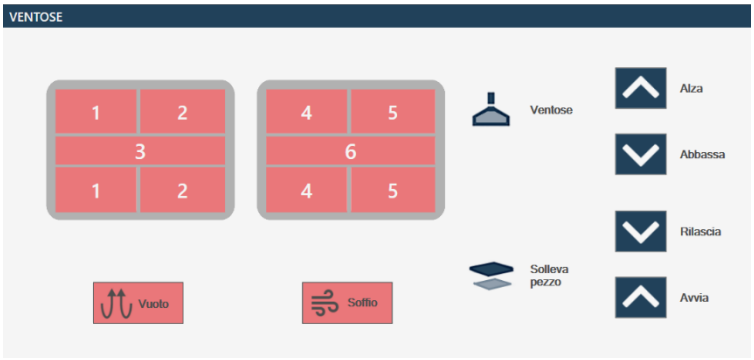
"MANUAL TOOL CHANGE" button: when this is pressed the machine completes a displacement in automatic mode in the relevant manual tool change area. **WARNING:** *Several safety conditions must be respected in order to carry out the manual tool change:* *- The position of the "A" Axis must be at 0°.* *- 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.	
Internal "Water On/off" button": pressing this button activates the pipe from inside the tool. 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 to move the axis by pressing the MDI or RTCP button.	

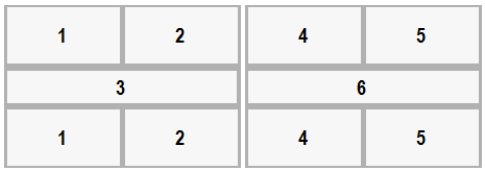
The "Active tool" button: clicking on the button selects the tool currently mounted in the machine. The button has 2 states: 1. Disc 2. Cutter or hollow bit	
The "Origin" field allows selection of the active origin to work with (in the example to the side the origin number 10).	
The "Axis Reset" button: 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 origin in tool coordinates (TCP) the correct tool measurements must have been set before pressing the Axis Reset button.	
"Interpolation Mode" Button: When the button is active (figure to the right) the manual movements Z, C and A are executed considering the rotation and inclination. *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 could be different from those you expect.*	
"Automatic +90° rotation" button: the position of the "C" axis is increased by 90° clockwise. Example: the machine is at 12 degrees, pressing the "Automatic +90° automatic rotation" button the machine rotates to 102 degrees: 12 + 90	

"Automatic -90° rotation" button: the position of the "C" axis is decreased by 90° anti-clockwise. 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 the respective potentiometers while the other axes are controlled by the interpolated potentiometer.	
RPM: this indicates the spindle motor speed in RPM (revolutions per minute). If the rotation direction is clockwise the value will be positive and if it is anticlockwise the sign will be negative. *Consumption (A):* this indicates the current intake of the spindle motor; *Alarm threshold (A)*: this indicates the threshold beyond which the alarm is triggered and spindle rotation is blocked. The ideal Ampere Threshold values for correct spindle motor operation are described in the use and maintenance manual "Annex A".	

4.2.1 MANUAL CONTROL OF SUCTION CUPS (OPTIONAL)

It is possible to use the suction cups in manual mode to move the pieces 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
Piece 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 piece. After a timeout wait, the axis is lifted to a safety height.	 Release

Button Description	
Piece 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 piece.*	 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 piece being moved. Try to always use the greatest vacuum area available for gripping of the piece 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 piece, 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.

ATTENTION: *a lack of power to the vacuum pump will cause a rapid loss of suction and the subsequent falling of the material lifted with the suction cups.*

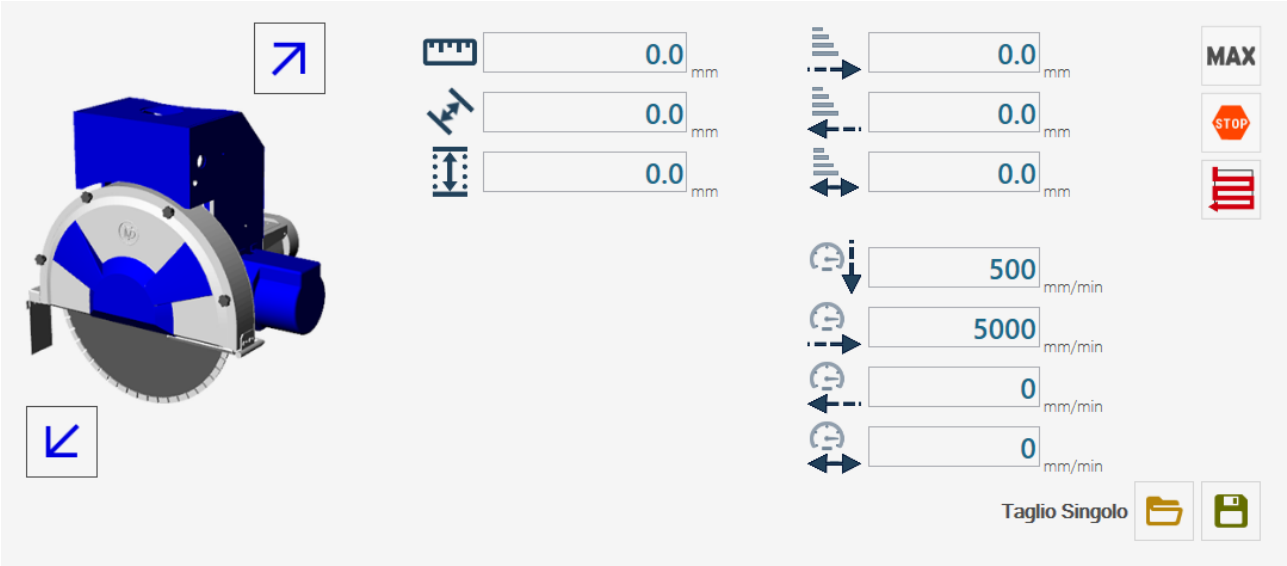
4.3 SINGLE CUT

In the newer versions of Parametrix it is possible to use the "Single Cut" function.
Selecting from the list the item "Single Cut" produces a page similar to this.
With the Single Cut option, it is possible to perform a straight (or even inclined) cut on the slab starting from the position at which the machine is and ending at the length requested by the user. With the cut it is NOT possible to cut arcs or circles.

Starting Cutting

1. Put the machine into the starting position.
2. Indicate the length in the "Length" field visible in the image below
3. Turn on the disc from the monitor pushbutton panel
4. Press  to start the machining.

The machining will be performed in the direction in which the machine is located at the start of the cut.

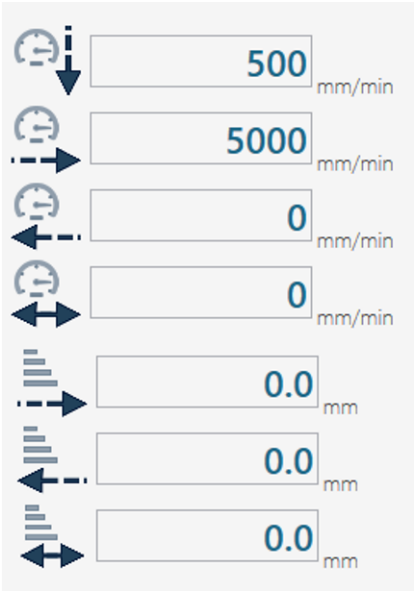


This page allows modifying of all the working parameters of the cutting, to indicate how long the cut must be, whether to repeat the same cut at a certain distance from it and whether to save or retrieve information about a cut.

4.3.1 PARAMETERS

Depending on the type of material to be worked it is possible to set the parameters differently and to save them in case of subsequent machining operations.

If the work mode chosen is multiple pass  the parameters to be set are as follows:



Material input speed

Working speed moving forwards

Working speed moving backwards

Working speed of the last pass

Penetration into the material moving forwards
Penetration into the material moving backwards

Penetration into the material for the last pass

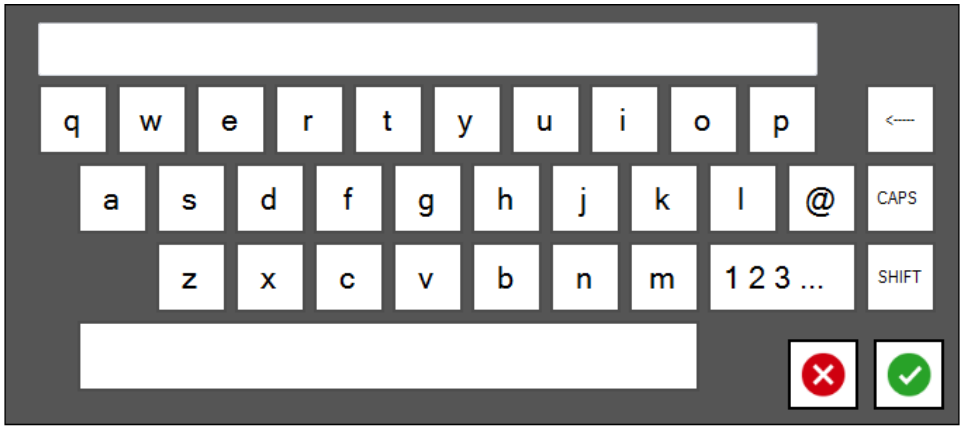
If instead full pass working mode  is chosen, the only parameters that will need to be set will be:

Material input speed

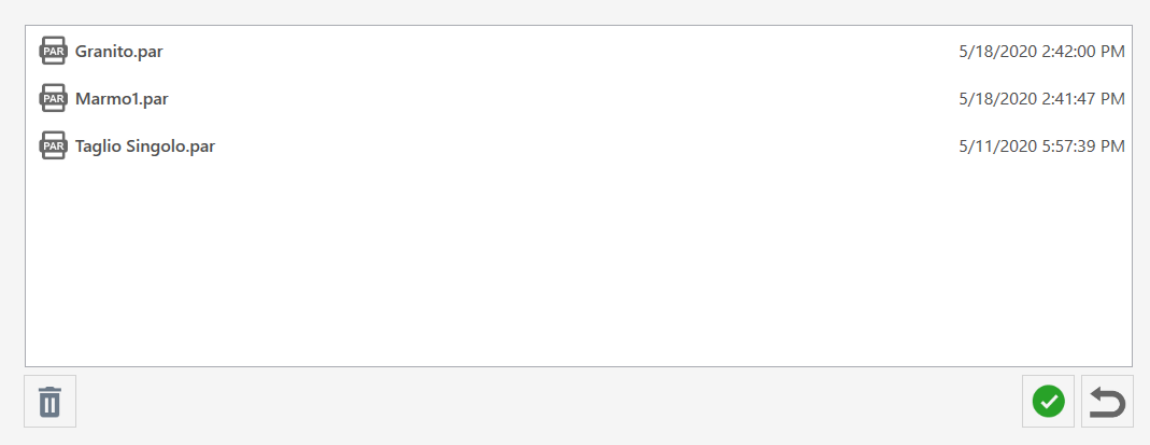
Working speed moving forwards

4.3.2 OPEN AND SAVE

All these parameters can be saved in a file. To retrieve them at a later date, press the button .
Pressing the button will display a keyboard for typing in of the File name:



To recover the files it is instead necessary to press the button .
This will display all the files saved during previous machining and that the user has decided to keep.



Having selected the desired file it is possible to delete it or to load the values saved by pressing .
If instead the button to open the file is pressed by accident it is possible to go back by pressing the button .

4.3.3 LENGTH, MAX AND STOP

It is necessary to enter the length of the cut to be made in order to perform the machining. If of course the value exceeds what can actually be cut, the machine stops before.

MAX

Pressing of the button 

by the user, means that the actual cutting length that can be cut will be given from the current position of the machine and the end of the slab (the end of the slab is relative to the direction that the machine has at that moment).
This value may differ from that entered by the user in the text box.

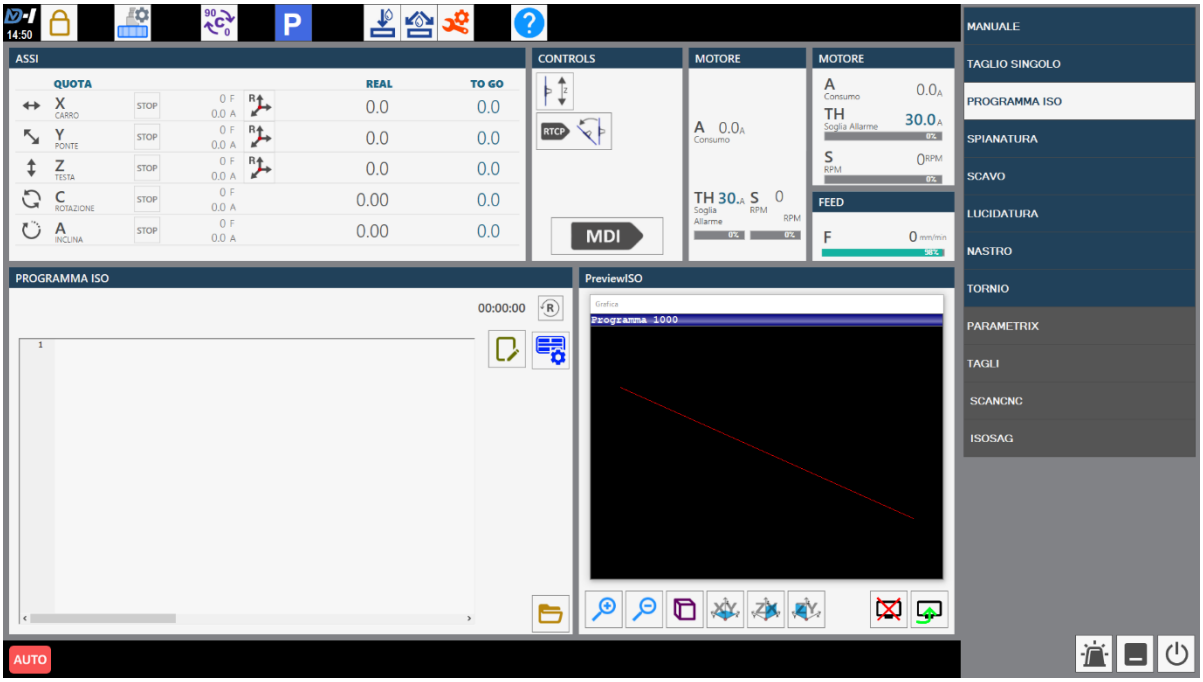
STOP

Instead the 

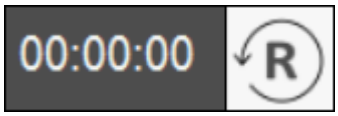
button is used once the machine has started to cut and when it has been decided to shorten the length of the cut. By pressing this button, the machine stops and in the "Length" text box the size of the cut shape made to now is shown.

Alternatively, it is possible to press the  button on the console to obtain the same behaviour of the button 

4.4 ISO PROGRAM

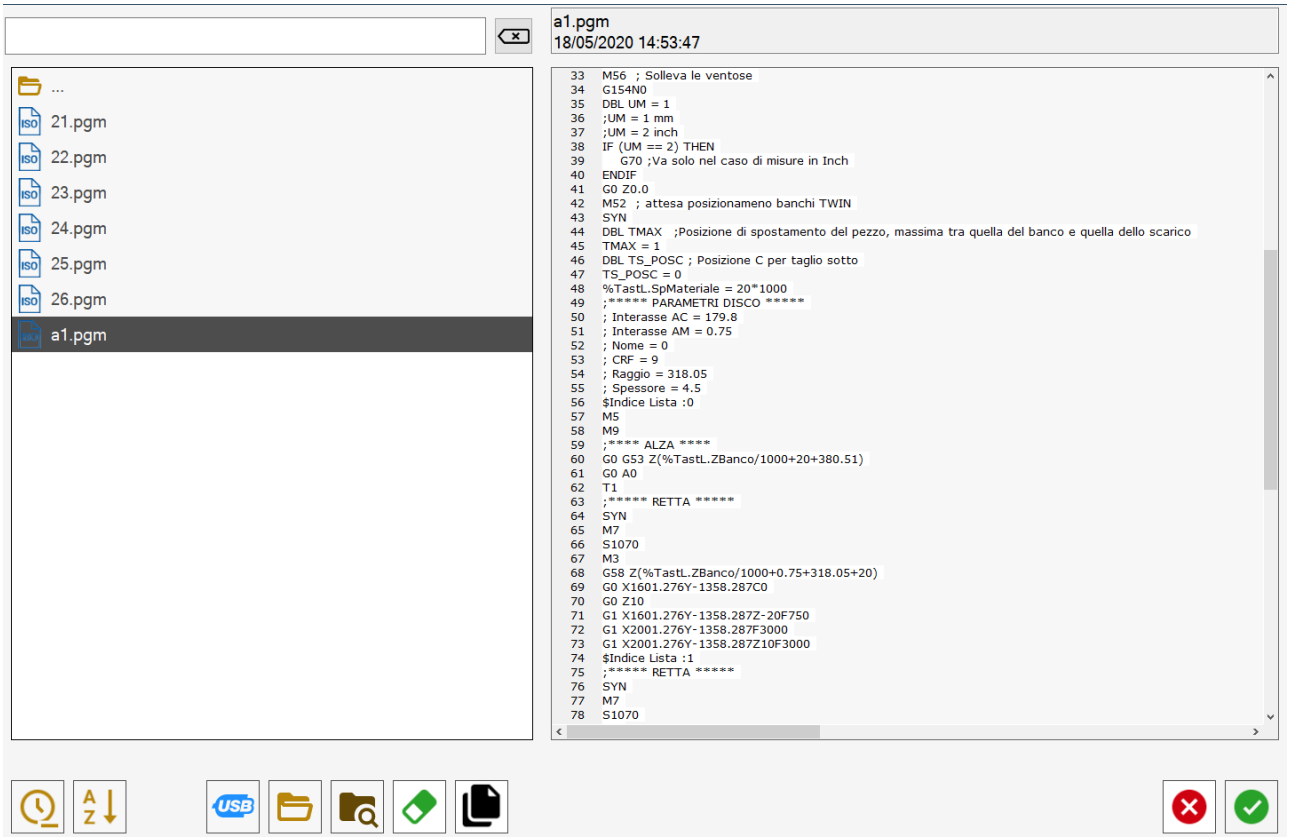


The ISO file programming page allows you to manage and start machining operations by means of .pgm files created beforehand through CAD-CAM programs.

Page Description	
It allows opening of saved. pgm files	
Pressing the button displays a keyboard that allows editing of the loaded .pgm File, before sending it into operation.	
It gives the execution time of the entire .pgm file. Pressing the button  resets the Timer	
It indicates the speed at which machining is performed. It cannot be modified.	
It opens the zoom handling and views icons.	

4.4.1 FILE OPENING

By pressing the button  , the screen in front of us allows us to choose the file to be opened:

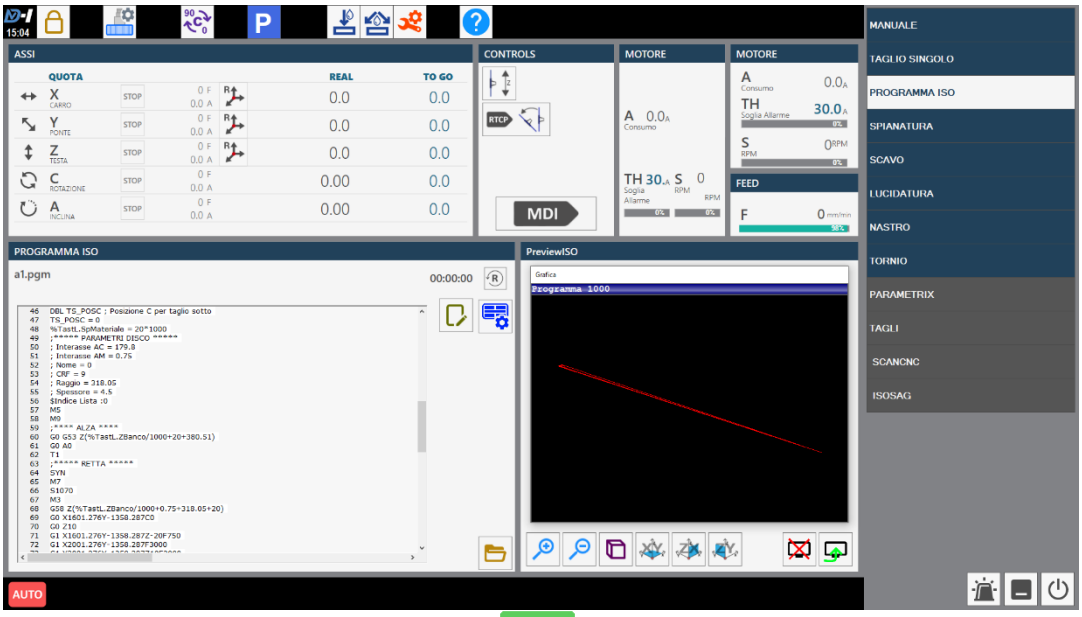


On this page we can search the file system to select the file to be opened. When a file is selected, various data is shown on the right-hand side of the screen:

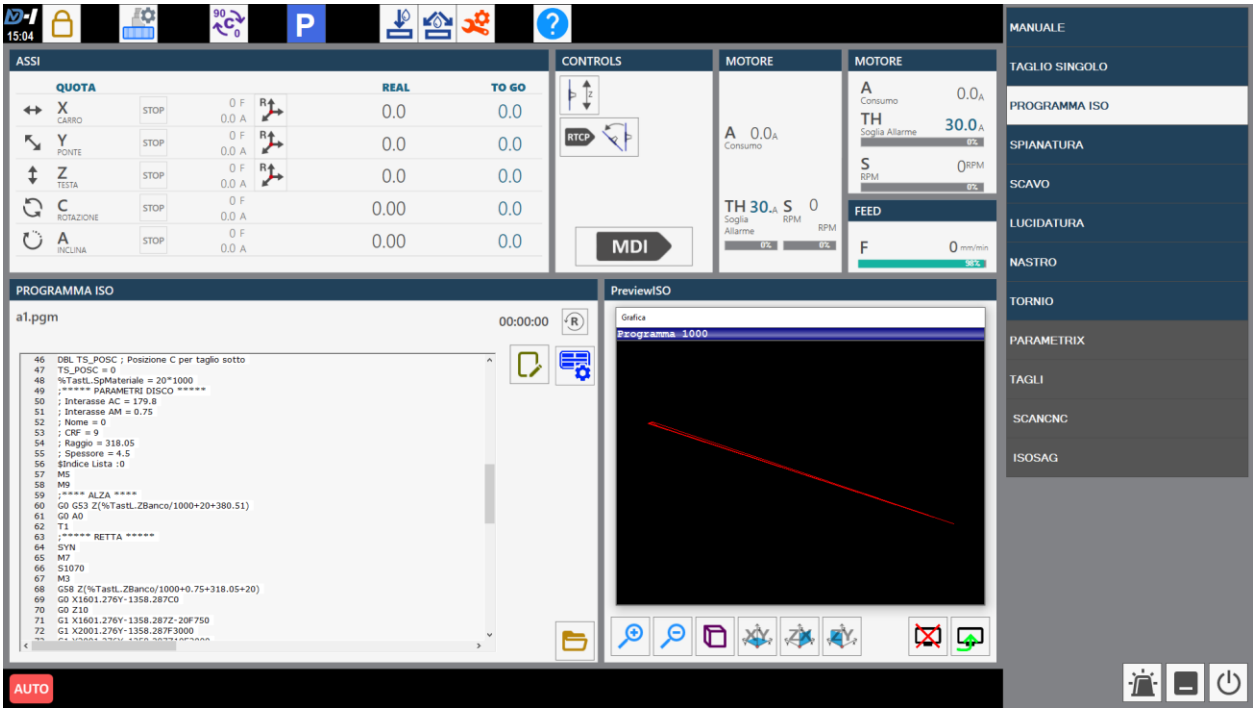
- Name of selected file
- Date last modified
- File content (reduced in case of particularly large files)

In the top left, it is possible to filter the file names.
The lower buttons, starting from the left:

- Sort in date order (increasing or decreasing)
- Sort in alphabetical order (increasing or decreasing)
- View all files with .pgm extension in memory stick
- View contents of ISO folder
- View discs present on PC and Desktop folder
- Enables/disables the delete function
- Enables/disables the copy function (to copy the selected file in the ISO folder)
- Closes the page without opening any files
- Closes the page and opens the selected file



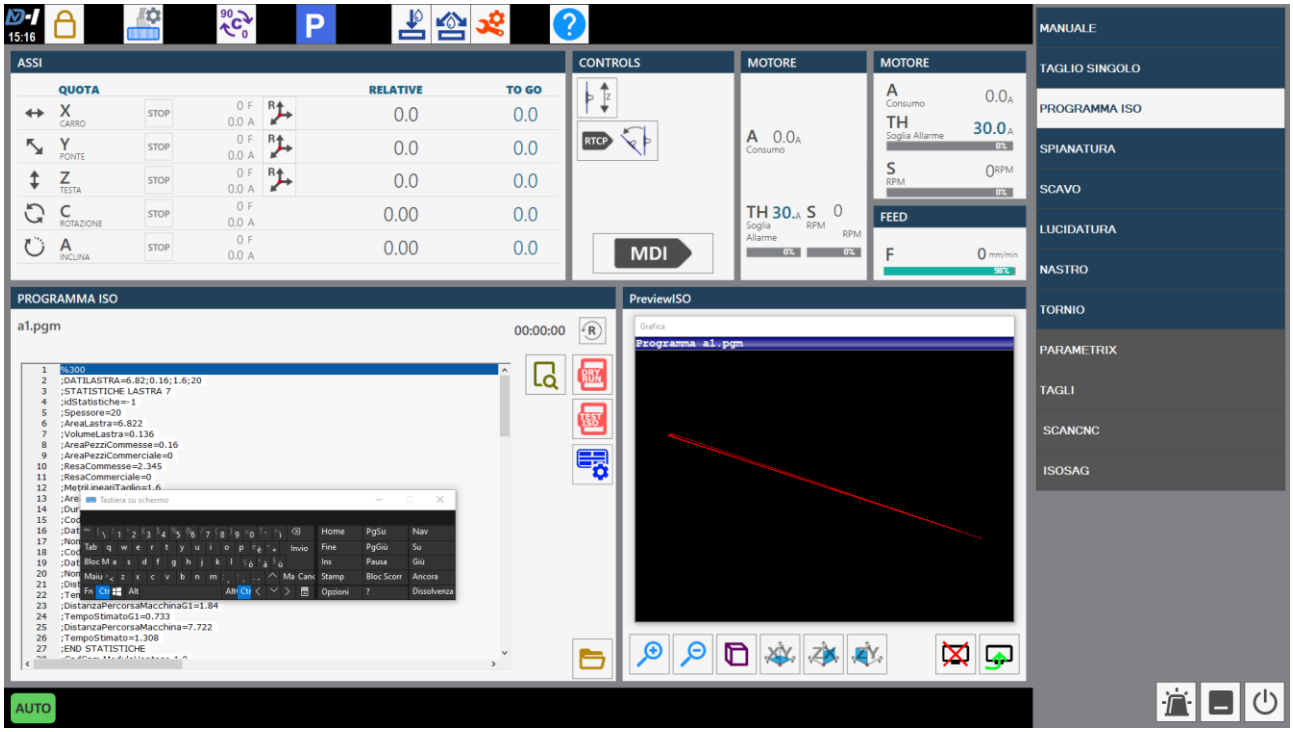
Note below the appearance of a button: AUTO, this indicates that the file is ready to be executed. By pressing the button, it will turn red, preventing execution AUTO and modifying the page as follows:



Note on this page that it is possible to open another file, edit or preview it with the graphics button.

4.4.2 MODIFY

By selecting Modify, it is possible to change the code before launching it into execution adding or removing instructions from the file.



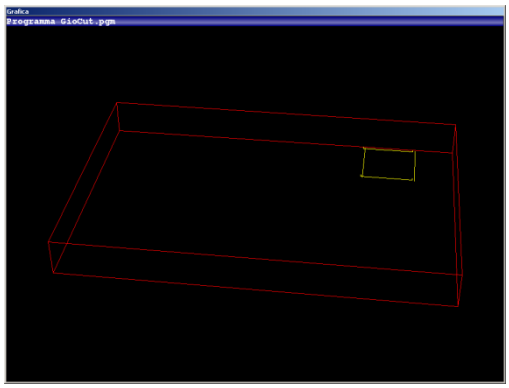
As is deducible from the image it is possible to find the point at which to edit the file by using the side scroll bar.
Having found the desired point, click on the area where the lines are and use the keyboard visible in the image to write or delete the line.

With the button it is possible to save the changes (the original file is also changed).

With the button it is possible to return to the previous page without making any changes or to delete them where any have been made.

4.4.3 GRAPHICS

The interface displays on the screen a graphic representation of the file being executed.



The following buttons are related to the image displayed above:

Page Description	
It enlarges the image.	
It reduces the size of the image.	
It provides a 3D representation of the image	
It provides a representation of the piece, with respect to the XY plane	
It provides a representation of the piece, with respect to the ZX plane	
It provides a representation of the piece, with respect to the ZY plane	
It hides the graphic representation	
It shows the graphic representation	

4.4.4 ISO AND DRY RUN TEST

Button Description	
When you select this option, it is possible to carry out the test of the file loaded to check that there are no syntax errors. This function is useful when it is necessary to carry out a machining test before starting to work on the material definitively. Press the "CYCLE START" button and the machine will execute virtually all the code lines of codes without moving the axes.	
When you select this work mode, it is possible to carry out a machining operation at the maximum machine speed. **DANGER: Use of this function means that NO material must be used during machining, as the speeds in the ISO program are not respected during execution; the numerical control will execute the program at the maximum nominal speed achievable with interpolated axes.** This function is useful when it is necessary to carry out a machining test without material in the work area. Procedure for using the function: press the "DRY RUN" button (the button will be coloured green), make the selection and load the program. Press the "CYCLE START" button;	
“FIND BLOCKAGE”: see paragraph below.	

4.4.5 FIND BLOCKAGE

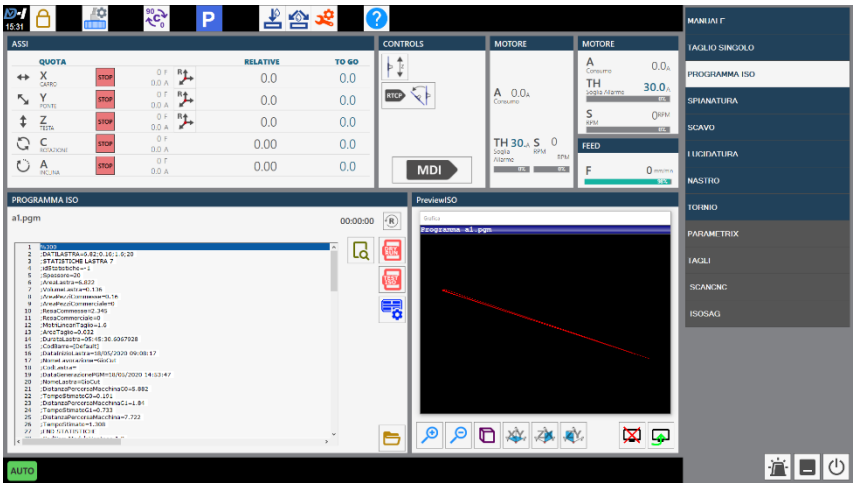
The find blockage function could be useful during the execution of files (.Pgm) to reposition the tool at the point where the machine interrupted the execution of the program if the tool gets blocked inside the material or if other types of blockage occur.

The find blockage function can be activated via  .
Having pressed the button it is possible to search for the line from which to continue/start machining via the line number by pressing the  button or by scrolling the file via the scroll bar located next to the file and selecting the desired line.

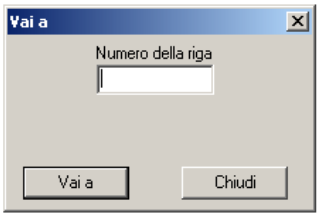
After selecting a point in the program from which the machine will continue the interrupted operation, press the "Cycle start" button; the machine will recalculate all the lines of code in the .Pgm file without making any movement. The speed of this operation depends on the number of lines of code that must be generated without making any movement.
When the recalculation is complete, the machine activates an automatic pause and asks the operator to press the "Cycle start" button again to start the machining.

It is advisable before starting the cycle to consider the initial positioning of the machine that involves all the axes simultaneously.

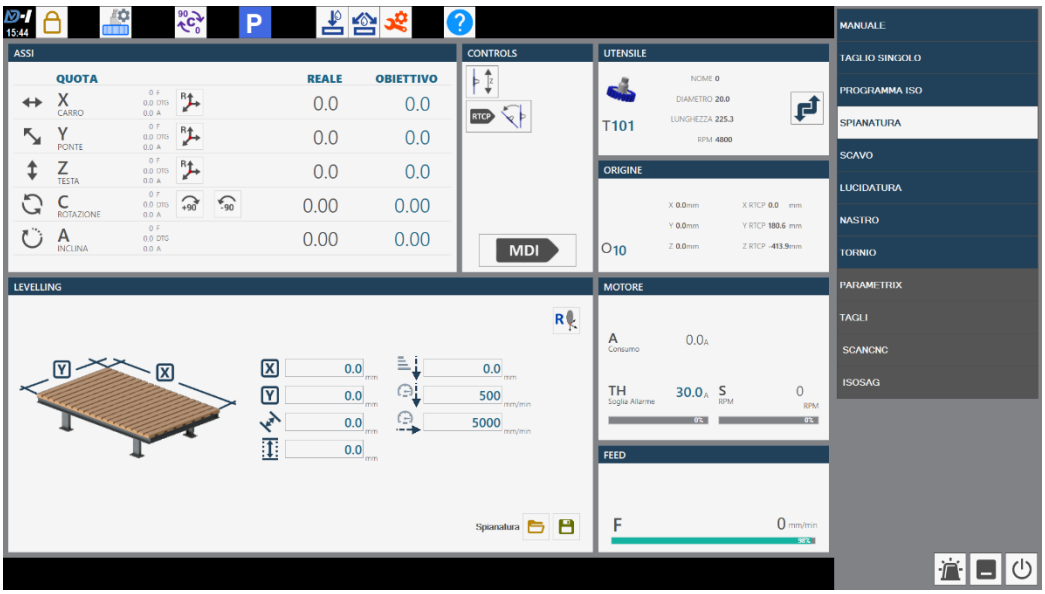
This is necessary as the machine moves from the current position to the point where it begins to work looking for the shortest route and, to do this, all the axes move to arrive exactly at that point.
This means that e.g. the Z axis does not lower having arrived at the starting point but is gradually lowered during the path. This could damage the machine or the material if the latter were to collide on material in the central of the route.
It is therefore possible, depending on the situation, to deselect the movement of one or several axes to avoid causing damage.
To do so, simply select the buttons above each axis that change from green to red.



The buttons will turn green automatically once "Cycle Start" is pressed.

Description of the Buttons	
Selecting this button will show on the screen a panel on which to type the line to go to  	

4.5 LEVELLING



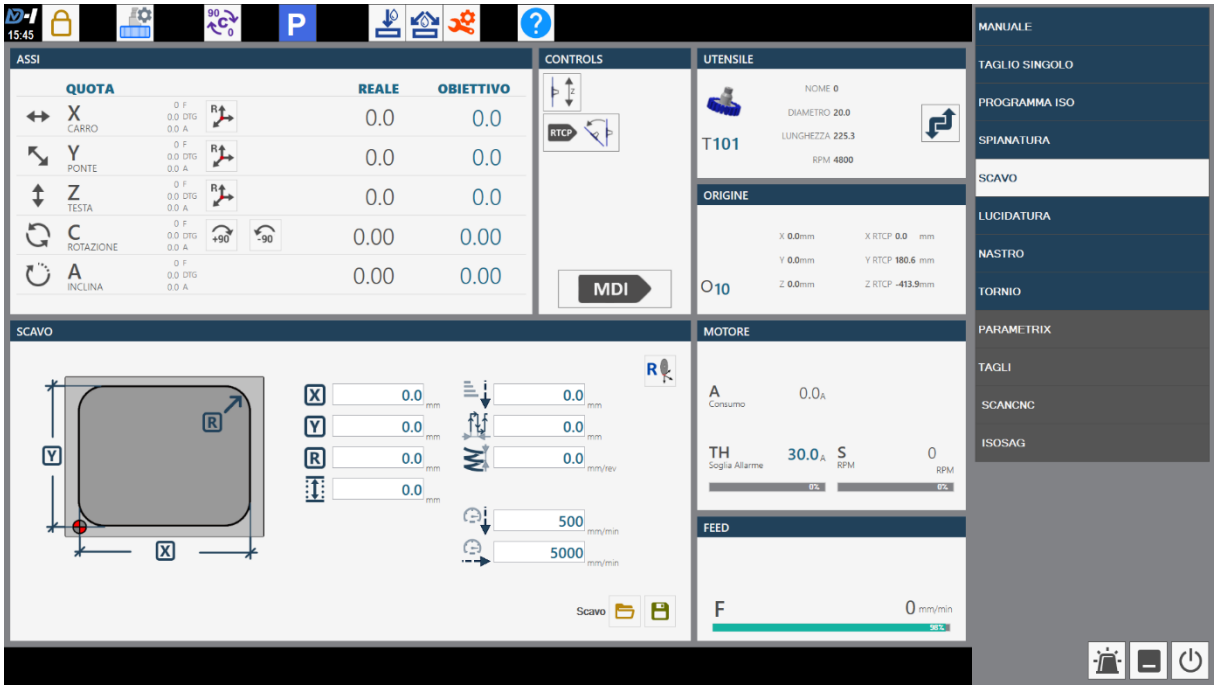
The levelling cycle makes it possible to level the bench or material. This program is useful when the bench is rendered uneven by the cuts or if you want to even out the thickness of the material.

Program operating procedure:

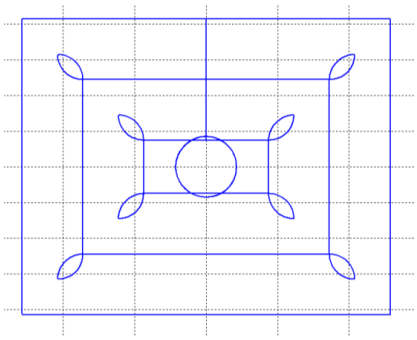
It is not necessary to set the X, Y and Z origin to zero for the "LEVELLING" program.
The position of the tool at the moment in which the cycle is started is considered in order to determine the starting position.
After setting the various parameters, move with the disc onto the maximum desired depth of the bench or material and start the cycle by pressing the CYCLE START button.
The program can be used with the A axis inclined at ZERO degrees or to NINETY degrees in order to be able to use tools such as discs or cutters.

Description of the parameters	
It indicates the displacement distance with respect to the Y axis. It is possible to set positive and negative values.	**Y**
This indicates the displacement distance compared with the X axis. It is possible to set positive and negative values.	**X**
“Step increase" indicates the distance between one cut and the next, in this field it is only possible to set positive values. To carry out the levelling operation it is advisable to set a "step increase" of about 80% of the tool thickness.	**STEP**
The "Cutting speed" (mm/min) indicates the feed speed during the automatic cycle.	**SPEED**

4.6 EMPTYING



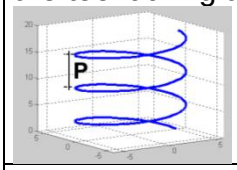
The emptying page makes it possible to carry out an emptying operation with rectangular shape with fillets and flat bottom. The "bench level" is considered the bottom level.
The cycle envisages the multiple pass mode with helical penetration at the centre and pass steps settable by parameters. The tool path is shown below.
The parameters are in any case described below.



WARNING: *In order to make the cut, the axis inclination must be A=90°*

Utilisation procedure	
Set the characteristic dimensions of the emptying form you wish to obtain. Then check the correctness of the tool data, particularly the diameter.	
Acquire the "bench level" at the emptying bottom level. In order to carry out an emptying operation at the desired depth, move to the top edge of the material, set the Z position to zero and make a displacement downwards (outside the material) equivalent to the depth of the cut and then acquire the bench level by means of the associated button. In order to use tools of the cutter type, measure the bench height with the axes A=90° in order to prevent errors in the measurement of the tool length. It is possible to find the button shown to the side by pressing  in the top left.	
Reset the X and Y axes to zero by selecting the corresponding reset button and positioning the cutter centre at the point indicated in the figure by the symbol. 	
In the case of machines with "laser pointers" it is possible to obtain the zero point in X and Y, by referring to the position pointed by the laser itself.	
Position the tool at the top level of the material, turn on the spindle and the internal and external water by means of the associated buttons, and press the "Cycle start" button.	
The emptying depth depends on the "bench height" level and on the position of the Z axis at the moment in which the "Cycle start" button is pressed.	

The emptying operation is carried out by helical penetration at the centre of the depth set in the "Z step" and then empty everything by means of a concentric operation at the distance set in the parameters.
The displacements for initial positioning are carried out at the "Release level" which can be set in the tool parameters.

Description of the parameters	
Length of emptying to be performed	**X**
Width of emptying to be performed	**Y**
Size of the connection to be obtained on the vertices of the rectangle created	**R Connection**
This indicates the maximum step of the penetration into the material. Depending on the emptying depth, the Z step can drop in order to carry out all the passes at the same depth.	**Z Step**
This indicates the lateral displacement between one pass and the parallel. A pass distance equivalent to 60% of the cutter diameter should be entered in order to prevent the displacements on the edges (that can be seen in the image above).	**Pass Distance**
This indicates the depth increase made with one revolution of the tool during the material penetration procedure. 	**Inlet**
This indicates the linear speed programmed for the machining. It is advisable to set a value that is appropriate for the material, for the type of tool and for the thickness you want to cut.	**Operating speed**
This speed is used during the penetration procedure.	**Descent Speed**

4.7 BASIC OPTIONALS

Below are a few basic optionals that can be supplied with the machine.

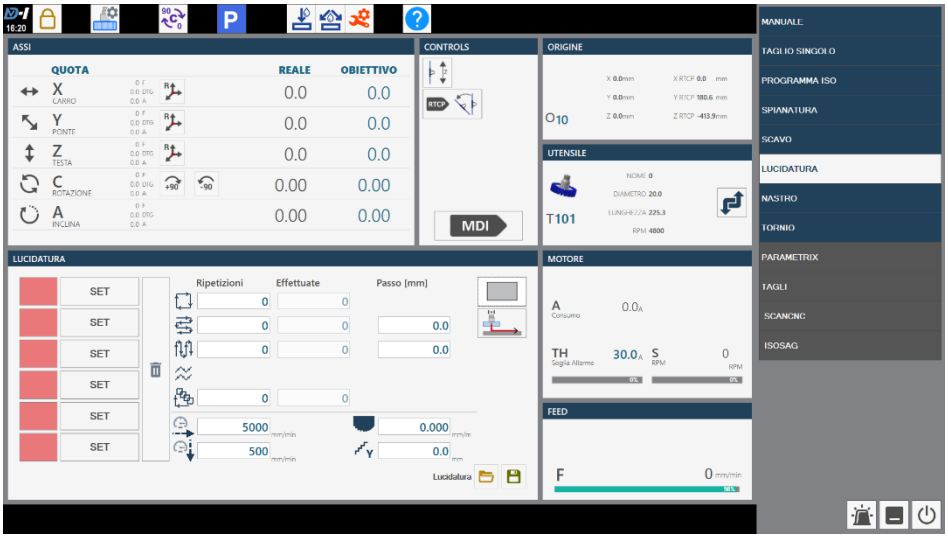
1. Polishing
2. Warehouse
3. Belt
4. Lathe

4.7.1 POLISHING

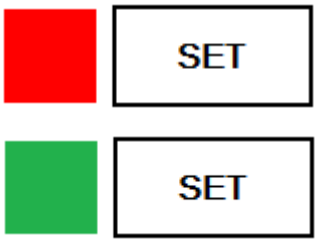
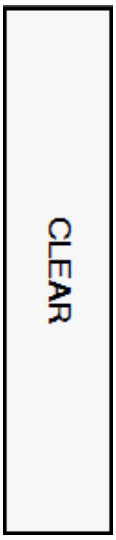
The Polishing program carries out polishing by means of different types of machining. It is also possible to choose the slab geometry acquisition mode: by detecting the points that determine the vertices of the surface (from three up to a maximum of six) or by using a regular surface (rectangle) inserting the value of zero and the value of the two sides.

By pressing the button shown in the figure it is possible to switch from the work mode in which acquisition of the surface takes place through the choice of points to one that uses the rectangle as the default figure	
By pressing the button in the figure it is possible to switch from the work mode that uses the rectangle as the default figure to that in which acquisition of the surface takes place through the choice of points.	

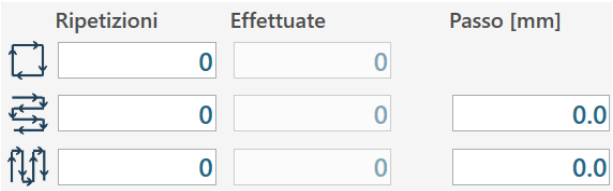
4.7.1.1 POINT ACQUISITION MACHINING



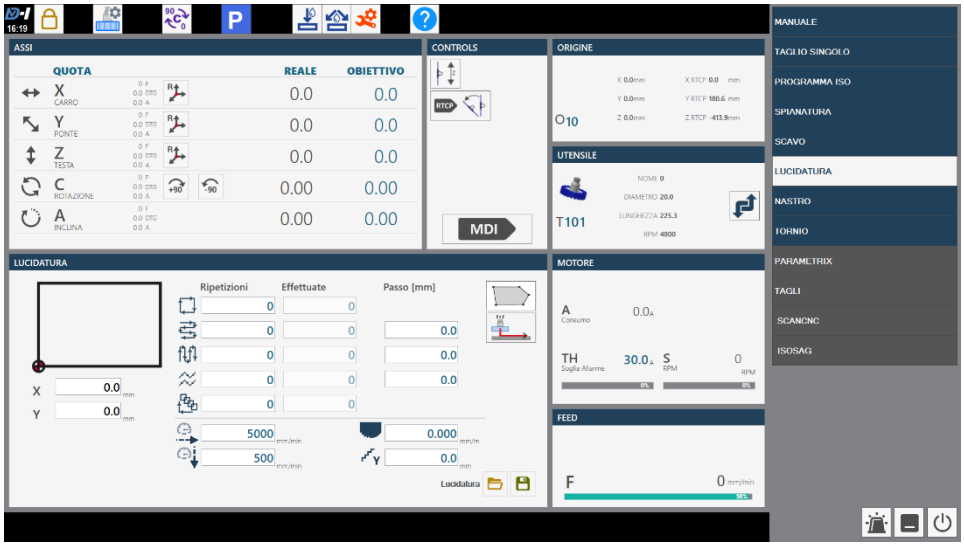
The left side of the screen is reserved for the management of points that determine the figure on which to work.

The insertion of coordinates of each point is permitted via the "SET" button. When the position has been acquired, the status of the adjacent figure changes from red to green. The insertion of points ranges from a minimum of three to a maximum of six.	
If the setting is incorrect and therefore to acquire the points again, the "CLEAR" button cancels all the values saved.	

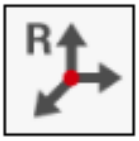
The right side of the screen is dedicated to management of the machining methods.

There are three possible types of machining (contour, horizontal passes, vertical passes), repeated for the number of times selected in the relevant box. If the value is set to "0", the machining in question is not performed. The "Performed" column displays the number of repetitions performed. The “Step” value indicates the distance between one pass and the next, selectable for each type of machining process. **WARNING:** *Between one machining process and the next the machine performs displacements without being raised to the safety level.*	

4.7.1.2 MACHINING ON RECTANGLE



The left side of the screen is dedicated to acquisition of the lengths of the rectangle.

The figure on which machining will be performed is determined by the values attributed to **Y** and **X** that correspond to the sides of the rectangle. The lower left vertex coordinates of the rectangle are acquired through the respective reset buttons, located under each axis. Both the values must be reset.	

The right side of the screen is dedicated to management of the machining methods.

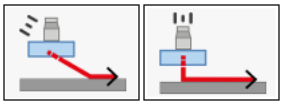
There are four types of machining (contour, horizontal passes, vertical passes and zigzag), repeated for the number of times selected in the relevant box. If the value is set to "0", the machining in question is not performed. The "Performed" column displays the number of repetitions performed. The “Step” value indicates the distance between one pass and the next, selectable for each type of machining process. **WARNING:** *Between one machining process and the next the machine performs displacements without being raised to the safety level.*	

4.7.1.3 REPETITION OF ENTIRE POLISHING PROGRAM

It is possible to start the machining cycles again by entering the number of repetitions in the relevant text box (see figure). The box for the column "Performed" shows the number of repetitions performed for complete machining.	

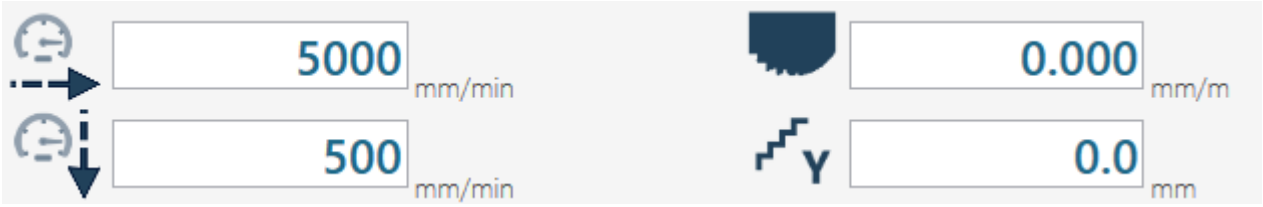
4.7.1.4 INPUT/OUTPUT MODE

To change the input mode, select the button 

The button in the figure enables (green) or disables (red) the progressive input and output of the tool on the piece. Enabling this function when the program starts, the machine will move from the Z coordinate to the Z safety coordinate then moving to the first point of the surface to be polished (for the rectangle it is the one in the top left). Then it will move along the perimeter of the surface, gradually lowering the Z coordinate so that at the end of the path, the tool is at work level. After this phase, the machine will perform polishing on the perimeter after which the processes selected will be performed in sequence. Upon completion of the work sequences, the machine will carry out a pass on the perimeter of the surface to be polished, then taking the same path rising progressively in the Z axis until reaching (at the end of the path) the safety level. Disabling this function when the program starts, the machine will move from the Z work coordinate to the Z safety coordinate then moving to the first point of the surface to be polished (for the rectangle it is the one in the top left). Thereafter, maintaining the X and Y axes still, the Z axis will lower as far as the work level after which the polishing tasks selected will start. Upon completion of the works sequences set, the machine will move to the first point of the surface to be polished then climbing in the Z axis to the safety level.	

4.7.1.5 MACHINING PARAMETERS

To change the Machining parameters, select the button  .
This will enable viewing and editing of the machining parameters



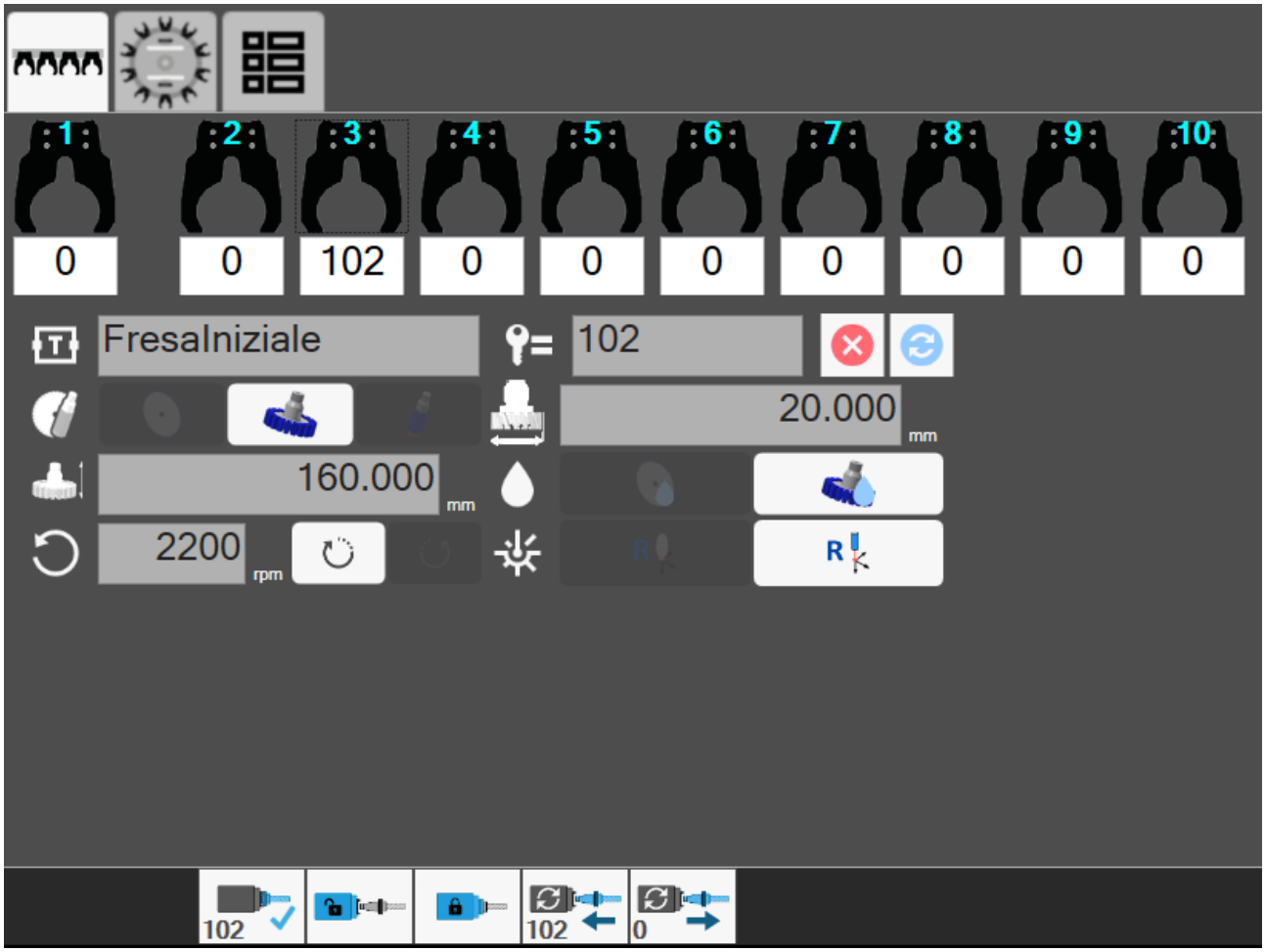
Description of the parameters	
It indicates the speed at which machining is performed.	**Working speed**
It indicates the desired speed in positioning along the Z axis at the coordinate selected.	**Descent speed**
It allows taking into account of the tool wear, during manufacturing with entering of the compensation value.	**Tool wear**
For zigzag polishing mode, it is possible to modify the machining step in Y, enabling splitting of the work trajectory into several passes. Keeping the value to zero or by setting a value greater than the one attributed to the B parameter, the polishing step along the Y axis will coincide with the value of B.	**Y axis step for ZigZag**

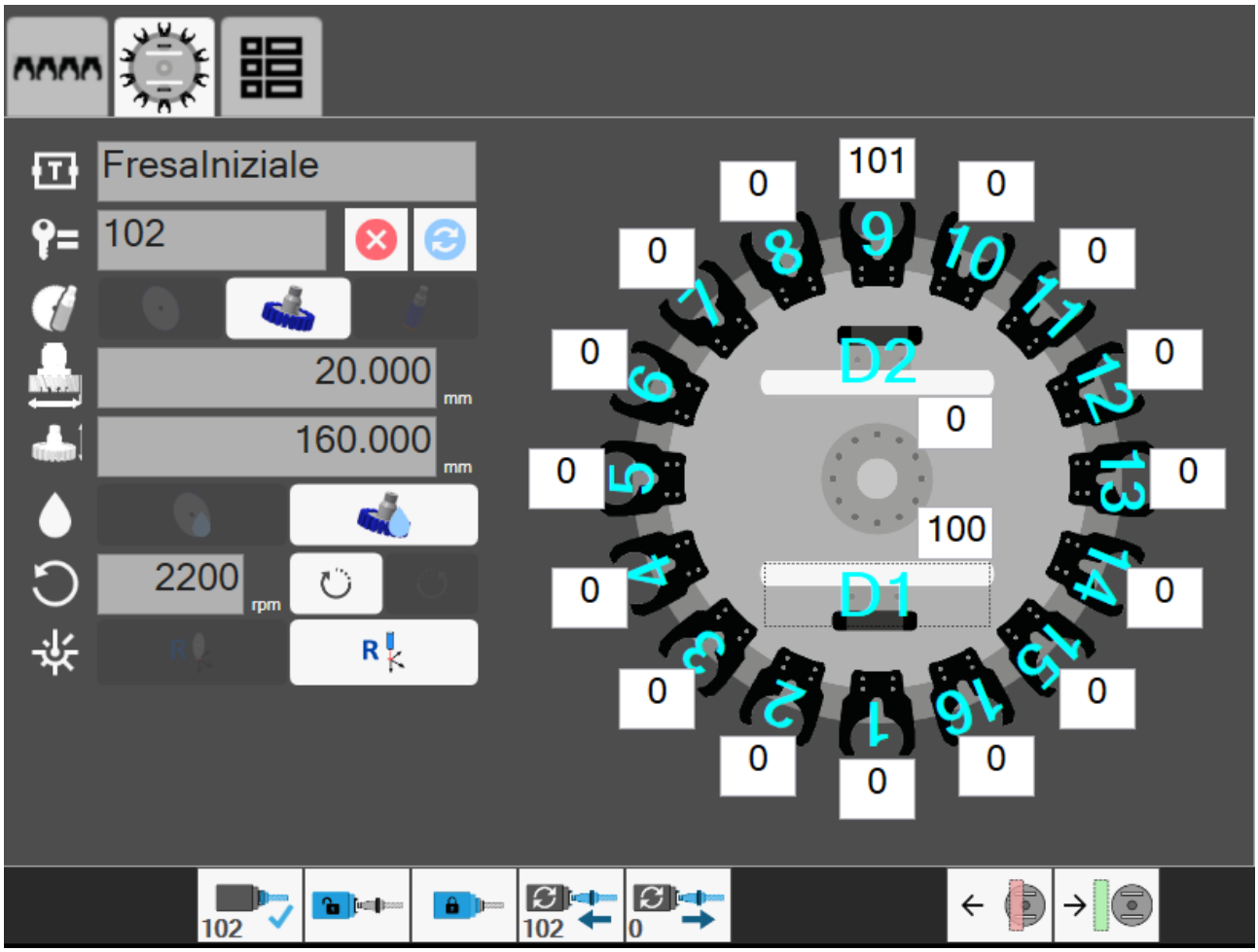
4.7.2 WAREHOUSE

The tool warehouse allows the management of tools and warehouses defined in the Parametrix program, enabling several buttons for tool change, probing and warehouse open/close commands. This new feature is only visible on machines with tool change; once the Parametrix has been properly configured, the “WAREHOUSE” option will appear in the list of programs.

In the WAREHOUSE page, a number of tabs will be generated equal to the number of warehouses present, plus an additional tool management tab. For example, in the figures below, three tabs have been generated insofar as two warehouses have been defined:

- One rack type warehouse
- One carousel type warehouse





- By clicking on a warehouse tab, the warehouse will be shown either as a carousel or rack.
- By clicking on a position, it is possible to view data regarding the tool in that specific position.
- By clicking on the text associated with a position, it is possible to assign a tool corrector to the selected position. If the corrector number keyed in is not linked to any tool, or if the tool doesn't have the required characteristics (e.g. Diameter > 400), the adjusted position is reset.

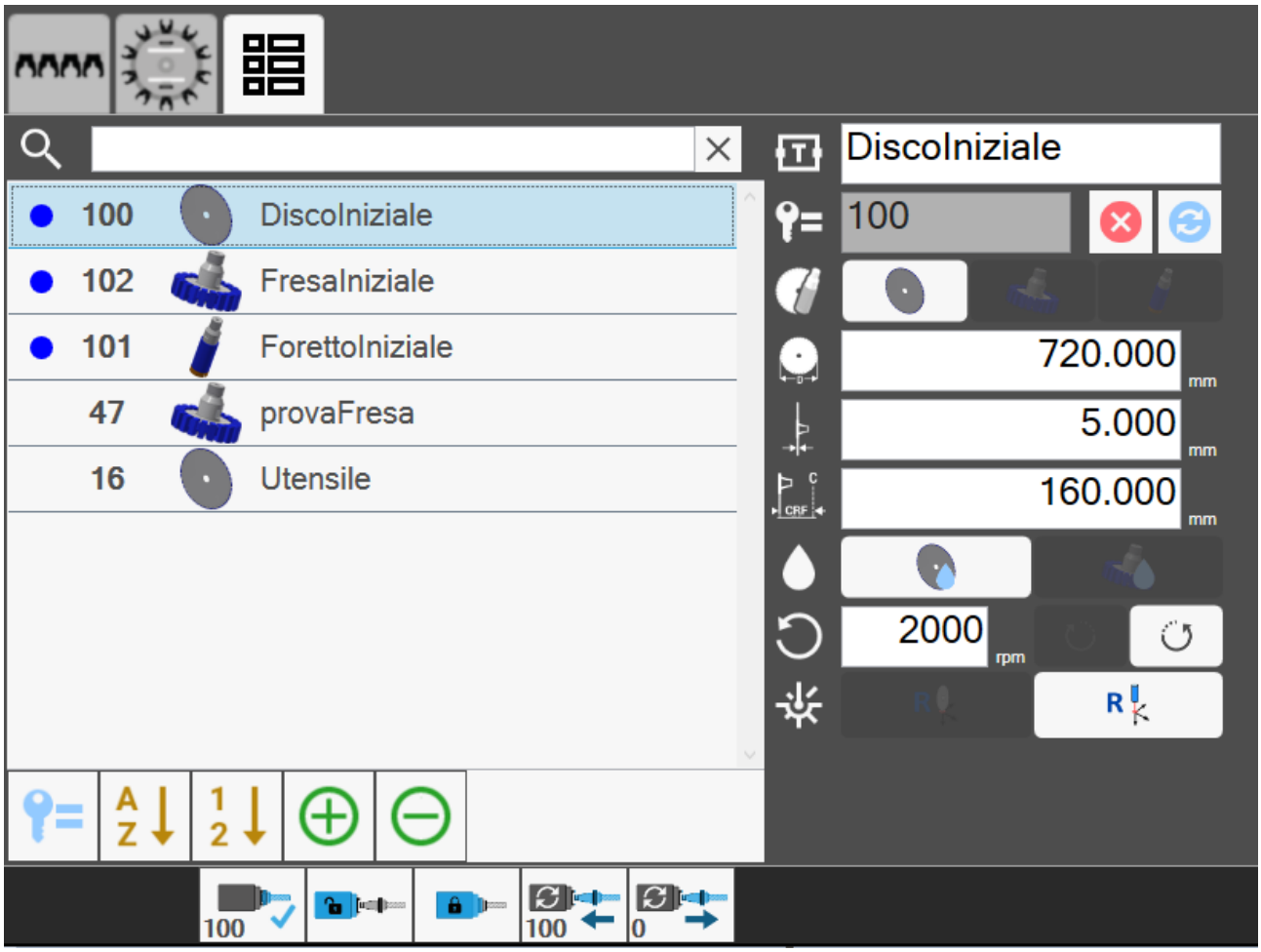
ATTENTION: The fact that a tool can be inserted in a warehouse position (software side) does not mean the tool can be physically placed in the set position. Decisions must therefore be made with care.

N.B. The tool data shown in the warehouse pages is not editable.

The last tab shows the list of tools. By selecting an element from the list, it is possible to view the tool data and eventually modify its characteristics.

N.B. it is not possible to change the type of tools in use from Parametrix.

There are **sort functions** (to sort the list in increasing alphabetical or number order) and **list filter** functions based either on a particular alphanumeric sequence, or whether or not to include tools with a zero corrector.
Other tool list management functions include: delete and create tool.



Delete tool this deletes the selected element, but cannot be performed on tools in use by the Parametrix programme or on tools not currently mounted on the machine.

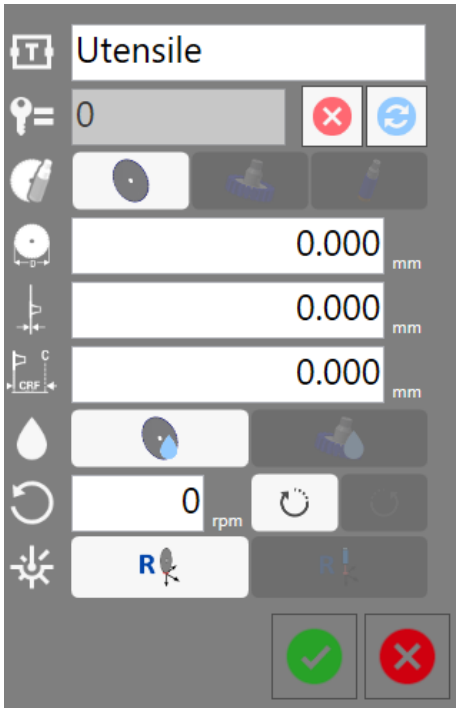
Create tool allows a new tool to be created and the definition of all its defining characteristics. Once the confirm button is pressed, the tool will be created **if and only if** the set values respect any limits defined for the tools, for example, the thickness of a disc type tool must always be greater than zero.

In the figure showing the list of tools, we can see blue dots alongside certain tools.

The blue dots identify special tools:

- Tools in use by the Parametrix program
- Tools not currently mounted on the machine

Tool creation window:



The only parameter that cannot be set during tool creation is the **release level**; this parameter will be automatically set depending on the release value of the tool currently mounted, or alternatively of the tool in use by Parametrix. The release level can be modified from within Parametrix.

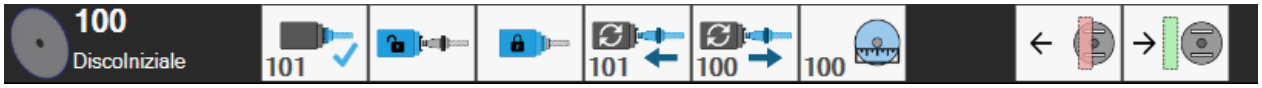
The corrector is assigned to a tool through a dedicated window, in which it is possible to select the corrector number to be assigned to the selected tool.

- Numbers with a green background indicate a free corrector
- Numbers with a red background indicate a corrector that has already been assigned to a tool

It is possible to select a corrector already in use by other tools, in this case the tool with the selected corrector will be assigned a zero corrector (a warning message will appear).

N.B. Tools cannot be assigned a corrector already assigned to tools in use by Parametrix (a warning message will appear)

Below are the buttons used to manage tool changes



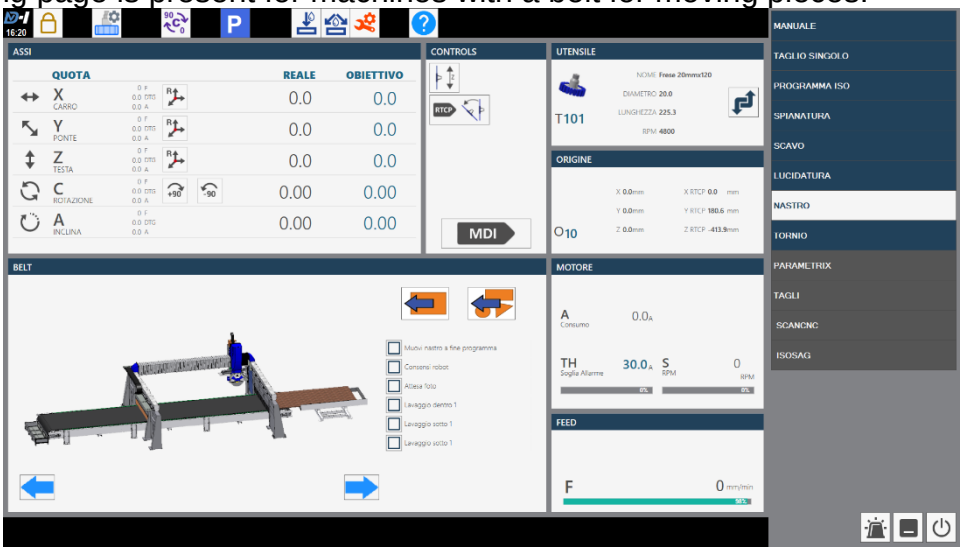
From left to right:

- **Currently mounted tool viewer:** when clicked, it is possible to view data relative to the mounted tool; if the space is empty, this means no tools are mounted.
- **Force tool**: Allows the selected tool to be forced as mounted; this is necessary in case of manual tool changes. The bottom number (e.g. 101) indicates the tool that will be set as mounted.
- **Open clamp**: Opens the clamp that blocks the tool. Requires operator confirmation.
- **Close clamp**: Closes the clamp that blocks the tool
- **Mount tool**: Starts the automatic tool change command; deposits the currently mounted tool and retrieves the selected tool (e.g. tool 101).
- **Remove tool**: Deposits the currently mounted tool (e.g. 100) in the assigned position
- **Probing**: Probes the diameter or length (the button to measure length is not visible insofar as not enabled) of the currently mounted tool (e.g. tool 100 diameter probing).
- **Open carousel**: Allows the carousel to enter the work area
- **Close carousel**: Moves the carousel out of the work area

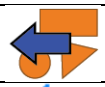
When the machine is running a cycle, all buttons will be disabled and all tools used by Parametrix will appear as non-modifiable; it will still be possible to change the positions of the warehouses.

4.7.3 BELT

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



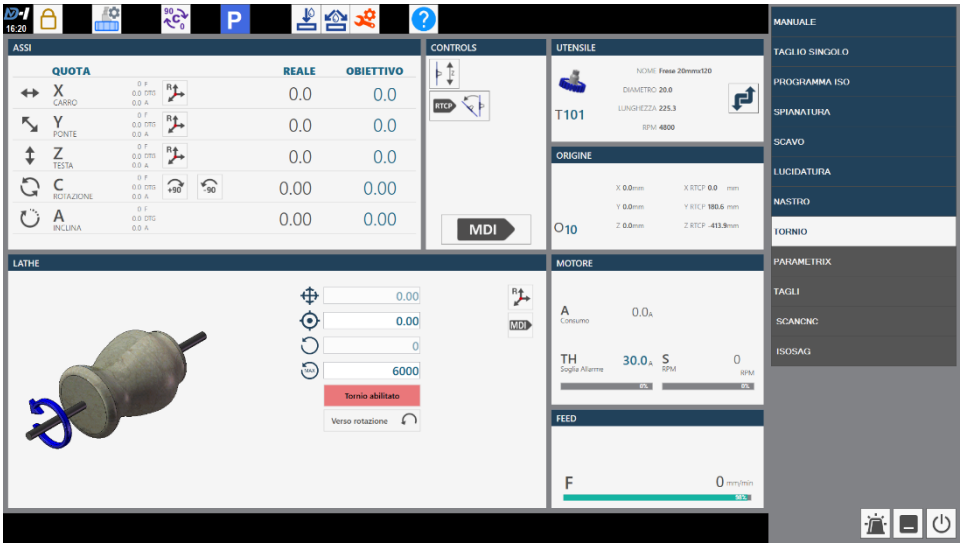
The control buttons contained in the page are listed below:

	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 the machine characteristics.

4.7.4 LATHE

The following page is present for machines able to perform machining operations with the lathe:



The control buttons and numeric fields contained in the page are explained below:

	This field shows the actual lathe value
	The lathe value can be set in this field
	This field shows the actual number of revolutions per minute
MAX	The maximum number of revolutions per minute can be set in this field
	Button to enable the lathe
	Button to set the rotation direction
R	Button to reset the axes
MDI	Button for the MDI movement

5 OPTIONAL SOFTWARE

The optional software that can be present in the program are:

1. Parametrix
2. Cutting
3. IsoSag
4. ScanCNC

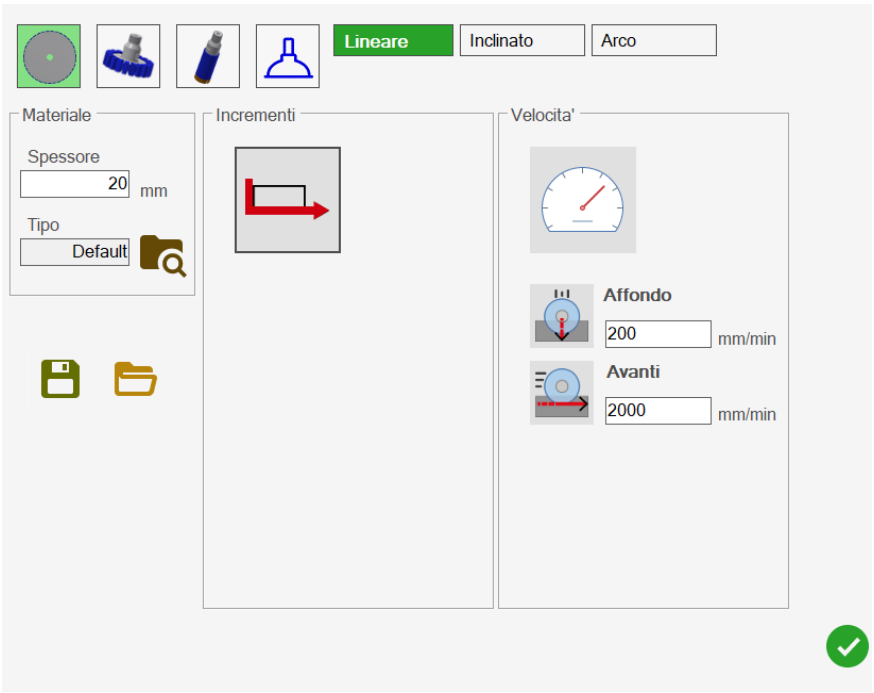
5.1 PARAMETRIX

The system is designed to facilitate the operator in programming of the automatic cuts. With the software it is possible to:

- Determine the position of the slab in the working area of the machine and create the material for machining
- Create pieces through parametric figures
- Import figures made with other CAD systems
- Generate a list of work pieces
- 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

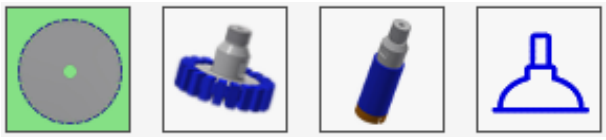
5.1.1 MACHINING PARAMETERS

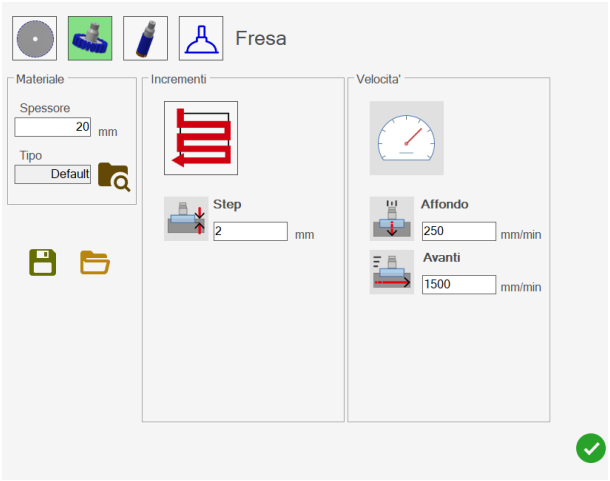
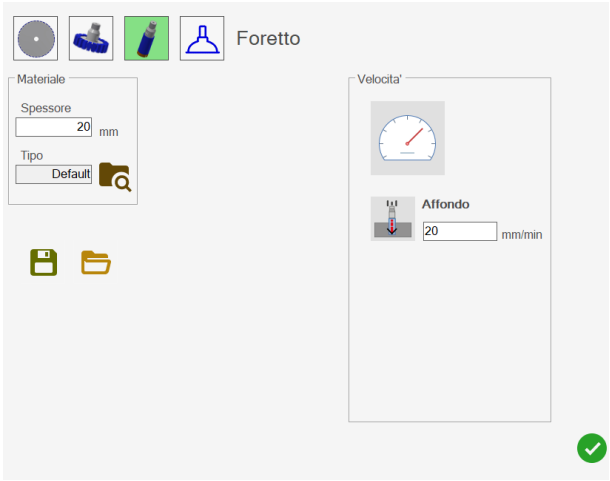
The machining parameters screen offers the operator management of the data relating to the machining and it is possible to view it by selecting the button  .



The machining data can be set and modified with the parameter panel.

It is possible to change all this data for the disc, the cutter and the hollow bit if the user is authorised to use them. For the suction cups, it is possible to modify the speed.
To decide on which tool to change the parameters it is necessary to select one of the following buttons visible in the upper left part of the panel.
Currently the disc is selected.

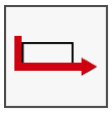
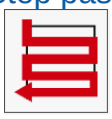
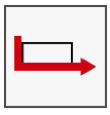
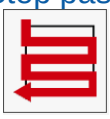


Modify the Cutter parameters	Modify the Hollow Bit parameters
	

5.1.1.1 EXPLANATION OF PARAMETERS

Except for the hollow bit in relation to which only the penetration speed can be set, the disc 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:

Disc	Full Pass 		Step pass 
Cutter	Full Pass 	Step pass 	Spiral pass 

5.1.1.2 PASSES

Full Pass
The Full Pass includes all the machining in a single pass. Therefore, the tool immediately enters all the materials and performs the machining.

Step Pass/Spiral Pass
These passes involve not penetrating to the bottom of 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 disc rotation	FORWARD
Indicates the depth of the cut on the material during the cutting stage in the opposite direction to that of disc 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 disc movement.	LAST

5.1.1.4 SPEED PARAMETERS

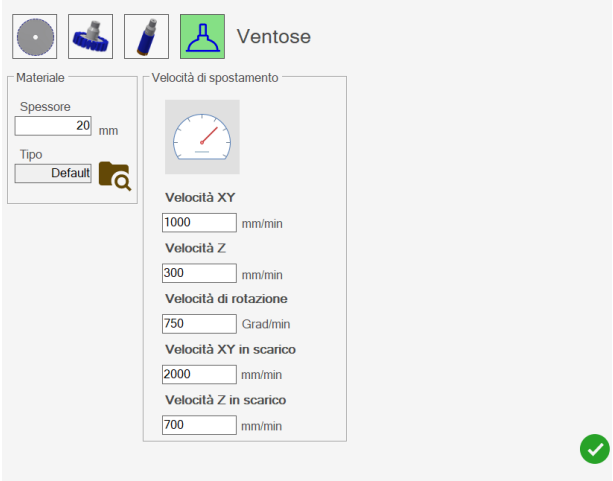
The speed section is used by all types of pass excluding some in case of full pass.

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 piece movement with the suction cups.

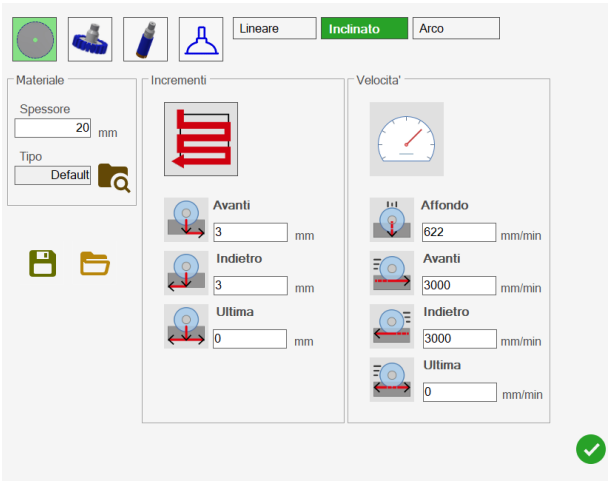
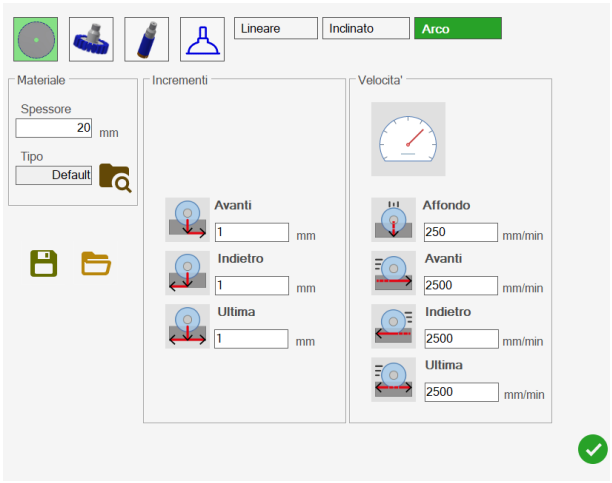


Description of the parameters	
Determines the speed of piece movement with the suction cups. This value is expressed in millimetres/minute.	XY Speed
Determines the speed of piece movement with the suction cups in Z. This value is expressed in millimetres/minute.	Z Speed
Determines the speed of piece rotation with the suction cups. This value is expressed in degrees/minute.	Rotation speed
Determines the speed of piece 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 piece 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 DISC OPTIONS

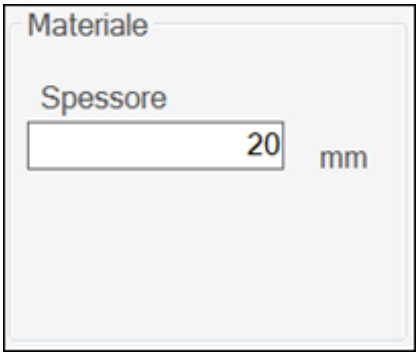
For the disc it is possible to also set the machining properties for certain cut types. Depending on the type of machining being performed, it will use the parameters present in the respective type. If for example the disc must make an inclined cut, it will use the **increment** and **speed** parameters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

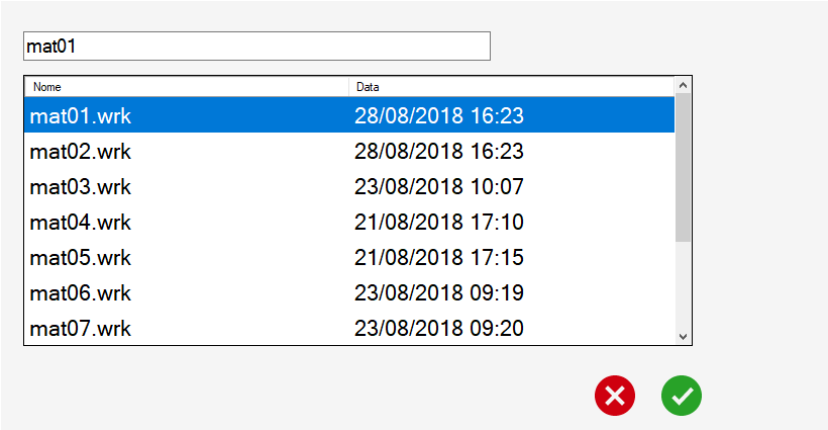


The value entered in this section is essential for all the calculations performed by the program.

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

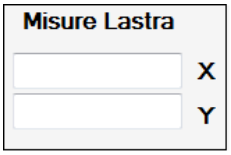
When the button  is pressed, the following panel will appear:



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

5.1.2 SLAB ACQUISITION

To introduce the pieces 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:

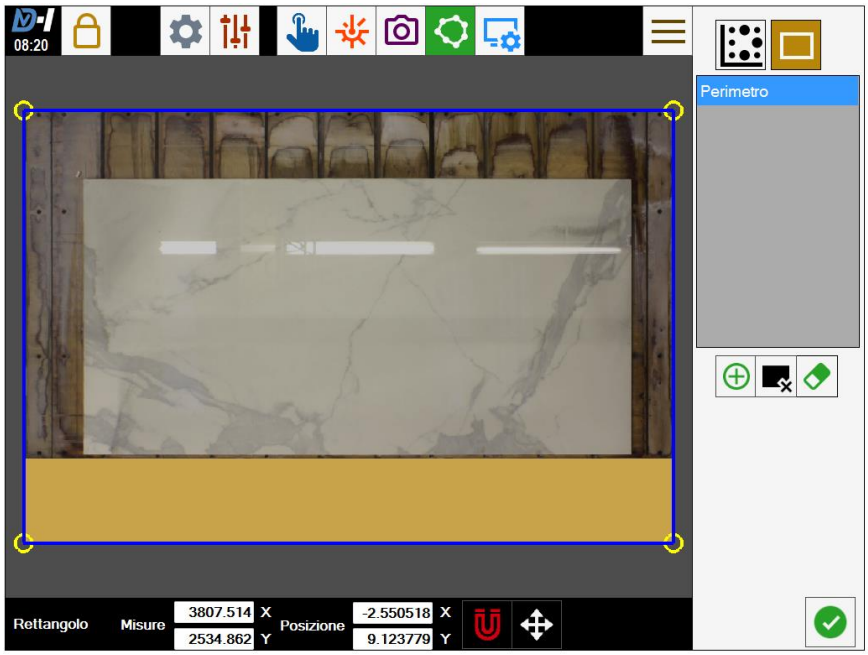
1. Photography (Optional) 
2. Cutting laser 
3. Cross laser 
4. Slab width and height measurements 

Through photography the user can see the material with any defects while with the laser system this inspection is not 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)

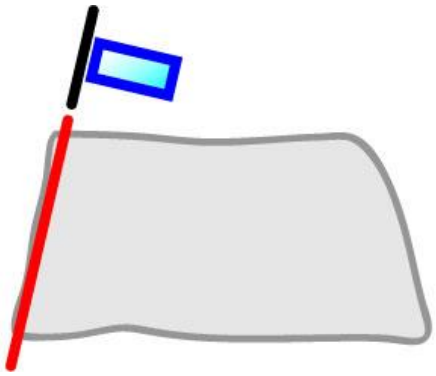
To upload a photo within the program, press the button . If the software is connected to a photographic device, a photo is taken of the slab otherwise there will be a prompt to select a photograph from the PC.
The photograph is uploaded into the work area and there will be a prompt for the perimeter. When the image is ready, it is possible to position the pieces 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: **for points** or **for Rectangle** similarly to the creation of the perimeter.
As the latter may be moved, vertices can be added at a later time, etc., as explained in section *Material Perimeter.*

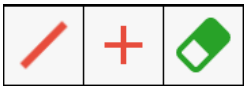
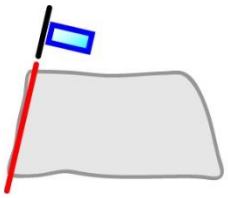
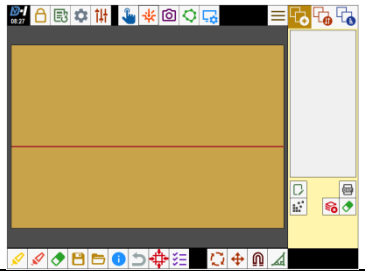
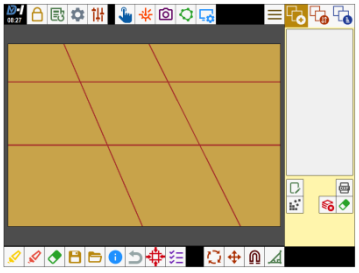
5.1.2.2 CUTTING LASER

To detect the position of the slab with the cutting laser it is necessary to move the machine on the edge of the material and to use the laser as a reference on the perimeter (*see figure*).



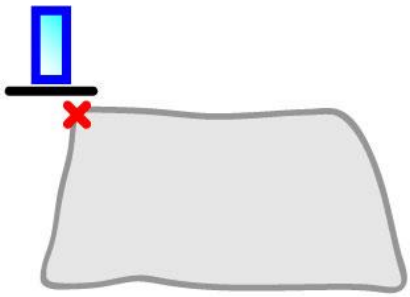
Pressing the button  the program detects the position of the line and shows on the screen, through the use of multiple lines how it is possible to recognise the perimeter and the position of the slab.

The procedure in points is as follows:

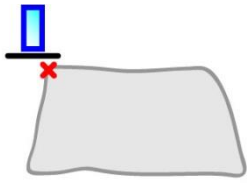
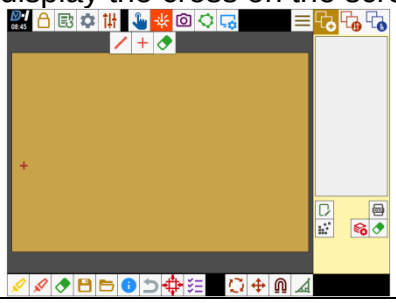
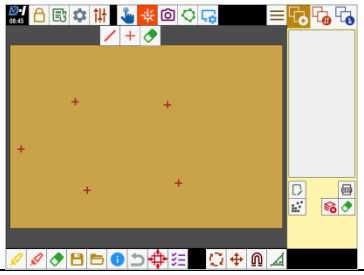
<table>
<tr>
<td>Press  on the home to activate the slab acquisition menu.</td>
<td> 
Initial page buttons
 
Slab acquisition menu</td>
</tr>
<tr>
<td>Move the machine onto the slab.
  </td>
<td>Press  and the screen shows the relevant line.
  </td>
</tr>
<tr>
<td>Repeat the operation for each side of the slab.
  </td>
<td>NOTE:
There are no line limits to be used for acquisition, the more laser positions acquired, the more accurate the perimeter will be.</td>
</tr>
</table>

5.1.2.3 CROSS 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.



Pressing  the program detects the position of the cross laser and reproduces it on the work area. With the use of multiple points it is possible to recognise the position of the slab. Follow this procedure.

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

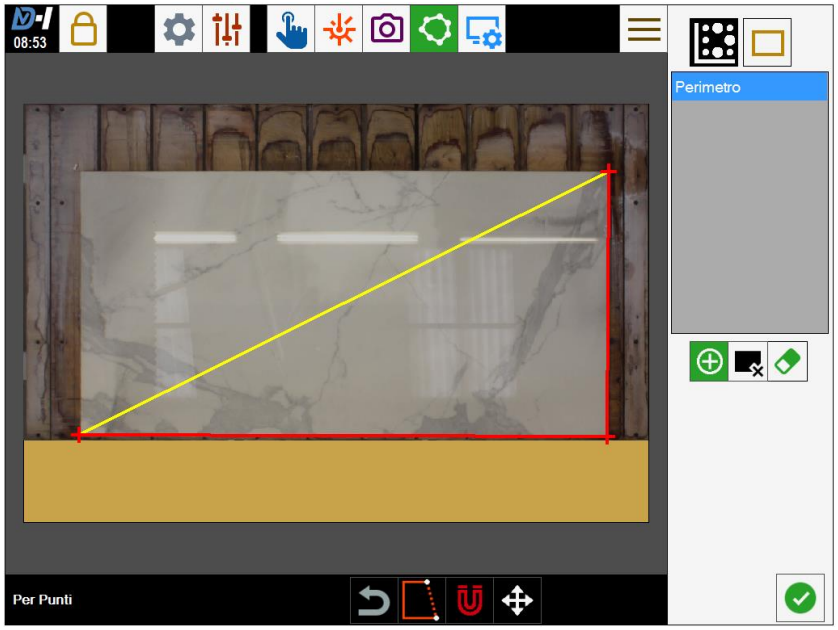
The program must recognise the edge of the slab. Selecting  it is possible to choose how to create the perimeter:

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.

For Rectangle

The "Rectangle" mode , when the button is pressed, will create a rectangle of the size and position indicated in the bar below

Also for this mode it will be possible to move the vertices to the position of the cross laser 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

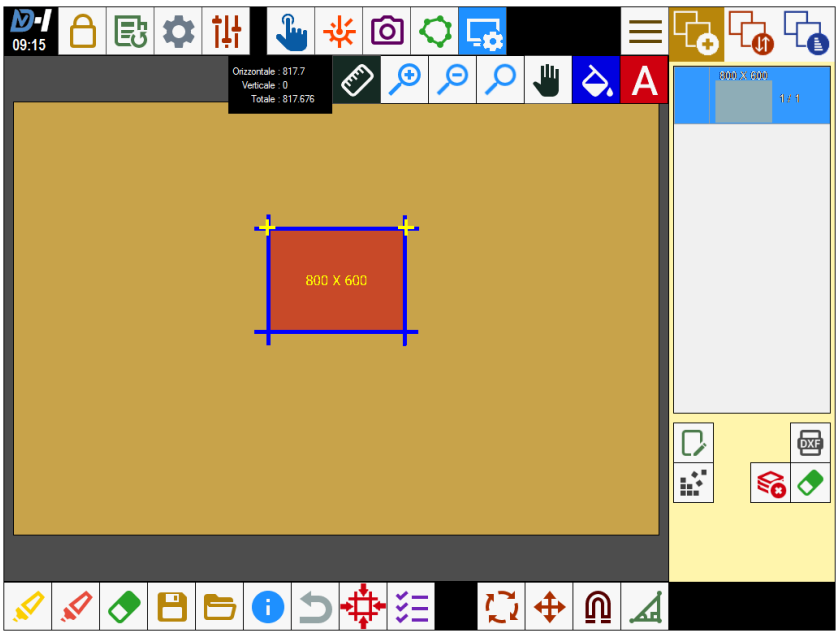
At the top of Parametrix it is possible to note the following button  . Pressing it, the following panel appears:

These buttons are intended to offer some information or utilities during use of the Program.

This panel remains visible until the button is pressed again  .
As a basic option or if all functions are disabled, the Zoom function is considered active.

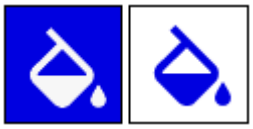
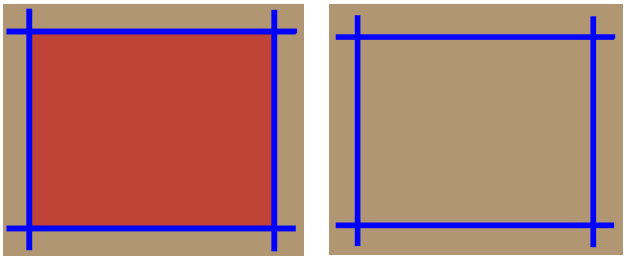
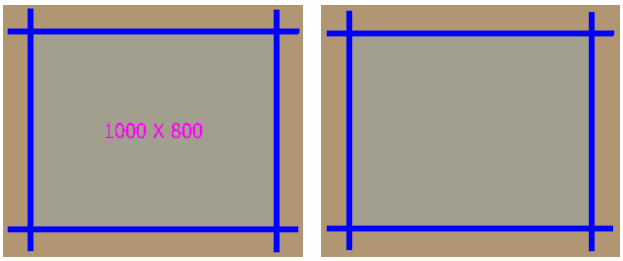
5.1.4.1 METRO

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).



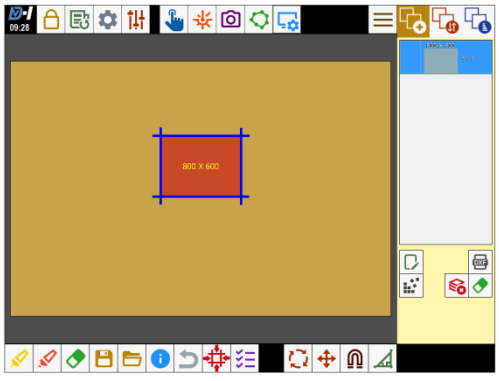
Under the "Metro" button a small panel appears, that shows the distance between the two points

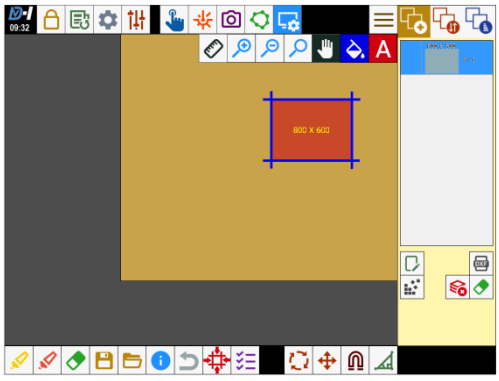
5.1.4.2 LENS AND VIEW OPTIONS ON THE PIECE

	These three buttons are used respectively to increase, reduce, or restore to the initial sizes the work area.
"Background" button enabled / disabled 	
"Name" button enabled / disabled 	

5.1.4.3 MOVEMENT

Selecting the button  it will be possible to move the work area with all the pieces above





This function becomes very useful associated with the Zoom of the preceding paragraph, as increasing the dimensions greatly, for example, it is possible that a few pieces inside the work area will no longer be visible. Using this function, it is possible to move to the area and relocate that piece.

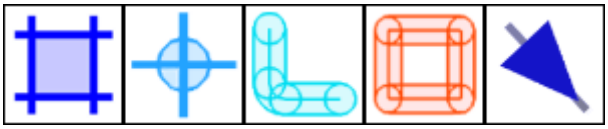
5.1.4.4 MANUAL BUTTONS IN PARAMETRIX

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

At the top of Parametrix the button  opens the following panel that allows you to enable and disable the display of certain entities in the workspace. Cuts, holes, milling and lowering are displayed in the first tab, and arrows are shown only in the third tab.



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 PIECE CREATION

Parametrix was designed for the processing of flat and closed geometric shapes. The creation of the pieces can be performed in 6 ways:

1 - Through an external Cad program and import of the DXF file	
2 - Insertion of Rectangles	
3 - Parametric Figures	
4 - Excel	

5 - History	
6 - Favourites	

When a piece is created or imported into the program, it is inserted in a piece list. Later in this manual the functioning of the list will be explained.

5.1.5.1 DXF IMPORTING

Note	• *To understand the chapter it is necessary to have a basic knowledge of technical drawing.*
	• *The drawing program requirements must be:*
	  ○ *DXF file export*
	  ○ *Polygons exported into polylines: there should not be explosions on the export file*

It is possible to create the figures to be cut in the machine with a technical drawing program. The drawing must conform to the following characteristics:

- The pieces must be closed polygons.
- The design must be free from scroll or similar as these could interfere with the importation and scanning of the pieces by Parametrix.
- Where there are labels to be inserted in the piece the lettering must be within the perimeter of the piece itself.

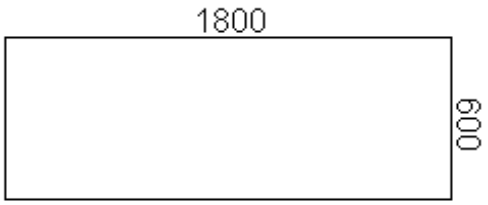
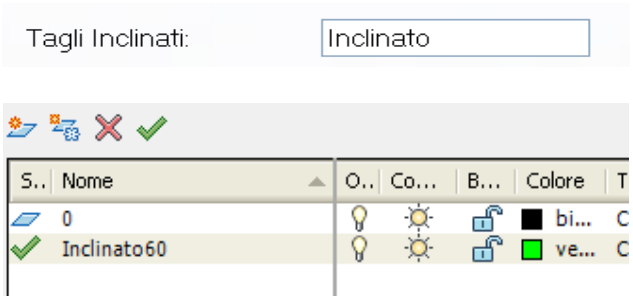
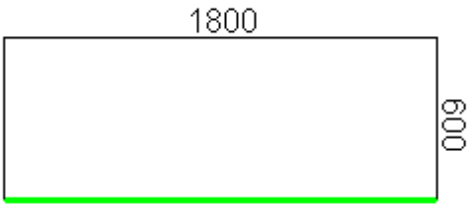
Furthermore, during the design phase it is possible to indicate certain types of special machining that will be recognised automatically by the program. The list of machining options are:

- Inclined cuts
- Raiser
- Backsplash
- Offset
- Drip
- Milling
- Lowering

For the special functions Parametrix uses the name of the layer. Therefore, on the program parameters page, it is possible to change the name of the layer relating to the machining. On the drawing, the special types of machining 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.	

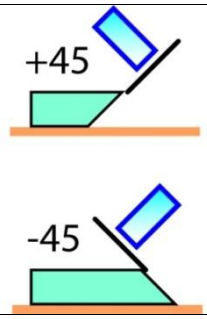
To apply the machining to a piece, firstly draw a line over the cut to be modified. The applied line must be part of the adapted layer. Here is an example of programming of an inclined cut on a rectangle.

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:* cut on one side with the A axis of the machine other than 0. The inclination in degrees is inserted into the name of the layer: **nameLayer+Degrees** (see example).

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

	On the program create a new layer: Incline30

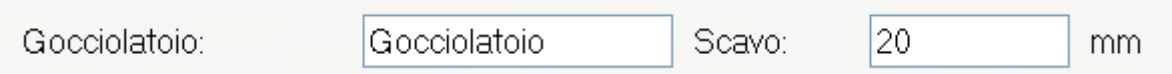
- *Raiser*: add a rectangular piece the length of which is equal to the actual cut. The width of the rectangle must be written in the name of the layer: **nameLayer+mmWidth**.

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 piece 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:* create a slit of cut and increase the size of the piece equal to the number of millimetres shown on the layer name. The name must be: **nameLayer+mmOffset**.

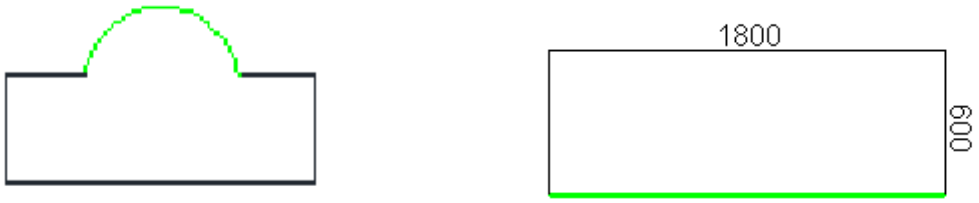
- *Drip:* the Drip function allows entering of a cut whose width is greater than the thickness of the disc and whose depth is less than the thickness of the material. The depth of the cut is set in the parameters under the item 'Emptying' (*see figure below*), while the width is indicated inside the layer name **nameLayer+mmWidth.**



- *Milling*: Milling is a particular feature in that it can completely replace a disc 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 piece, otherwise on the same side there will be both cut with disc and cuts with Cutter.
E.g. DXF where the milling (selected in green) fully covers the side of the piece. When they are imported into Parametrix, only the cut with the cutter will be applied on those two sides.

1. 2.



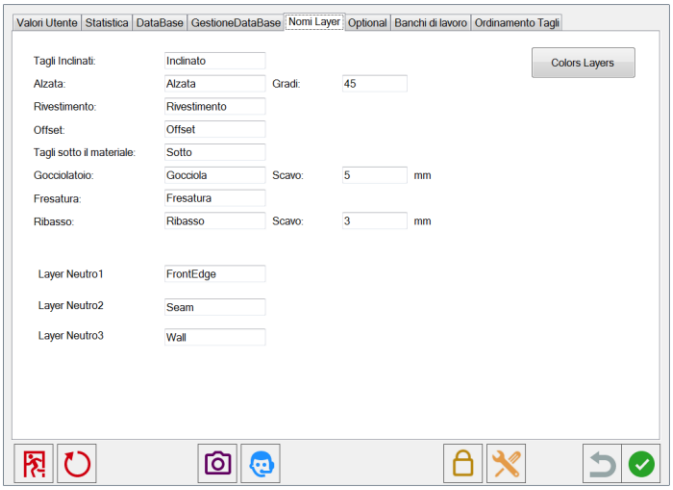
E.g. DXF with milling that does not replace the cut with the disc but it finishes the machining.

Note	For cuts with the cutter, which do not replace cutting with the disc 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 tool assembled.

- ***Lowering:*** The Lowering function is primarily used to lower by a certain amount the holes of the kitchen tops in order to able to insert the sinks. To create lowering within the DXF, the layer name must take the following format: **nameLayer+mm width**, where the **nameLayer** is the same as that present in Parametrix (under the Lowering item). With regard to the part "**mmWidth**" if it is not present in the name, by default the value 0 is taken. With a value of 0 this means that the cutter centre is right above the side, therefore half of the cutter will enter 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 within Parametrix:

To change the 'nameLayer' with which to assign the machinings, press the button  and select the "Layer Names" window within which are listed the machinings and the corresponding names assigned.



Through the "colour layer" it is possible to view certain sides of the coloured piece differently, therefore the Layer colours do not modify the machinings of the side but identify them on the 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.

Hollow Bit

Another particular function is the cut with the Hollow Bit; it is normally used to finish certain machining operations (in particular points of the piece), such as, in the corners where the disc cannot reach, as it would damage the piece (*Disc avoidance area*) or to create a hole inside the material.
This function does not behave like the others, that is, 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 piece (where the Disc 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.
Instead to use the core to cut a circle inside the material, simply draw this element inside the piece, making sure that it does not intersect with any side.
In both cases, the program recognises that the cut must be performed with the Hollow Bit tool, rather than with the Disc tool, if this value is not exceeded: "Hollow Bit Maximum Diameter", present in the section  .
Generally, this value is set when the machine is installed, and no further changes are required.

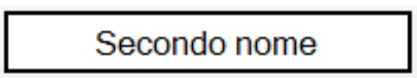
Importing the DXF file

Importing of the dxf file takes place on the piece creation page (see figure below). To access this page press  ; once opened press the button  .
A list of files present in the work folder of the PC appears. The following options exist to change the folder or to delete files:

-  All the files in the USB memory inserted into 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 pieces in the drawing with  .

It is possible to change the number of identical pieces 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'.

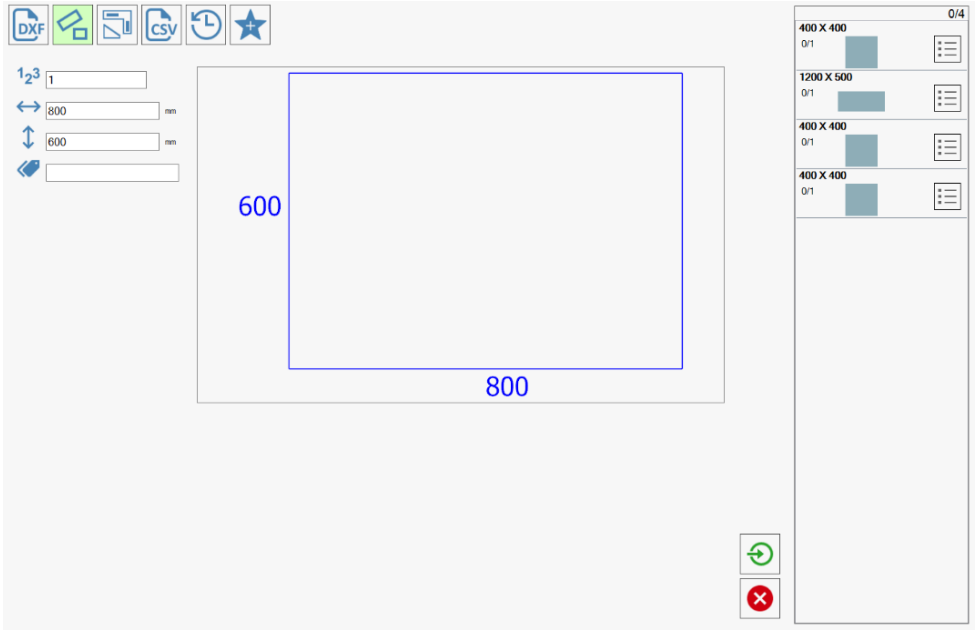
Added Buttons	
Ordering the files present in the basic list by date. By default they are sorted from the most recent to the oldest and the button appears like this:  . Pressing the button again the files are sorted from the oldest to the most recent and will appear like this:  . It is possible to repeat the operation several times, passing from the most recent to the oldest and vice versa	
Ordering he files present in the basic list by file name. By default the files upon pressing the button  are ordered by name	
This allows associating a second name to the file if 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 repetitions 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

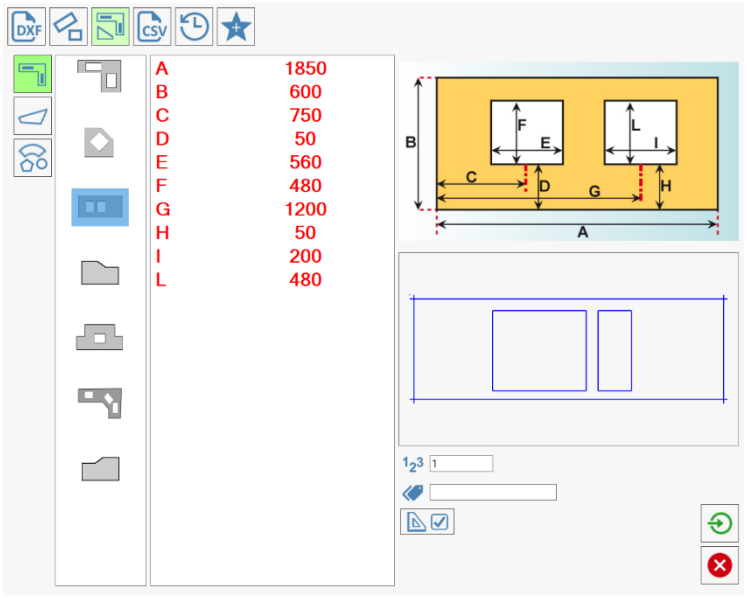
When all the values have been entered, press the button  to insert the rectangle in the list of pieces to be cut.

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*

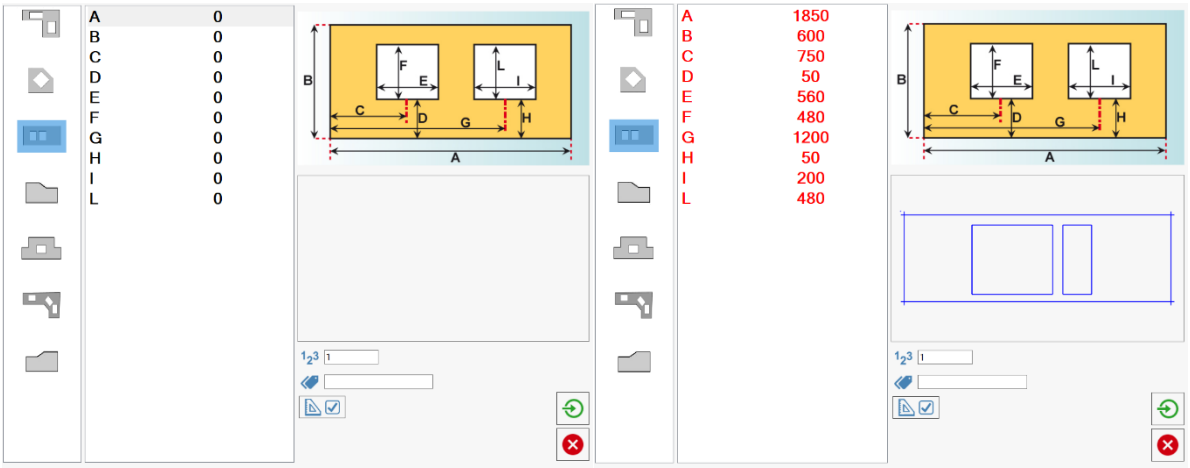
5.1.5.3 CREATE PIECES WITH PARAMETRIC FIGURES

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 

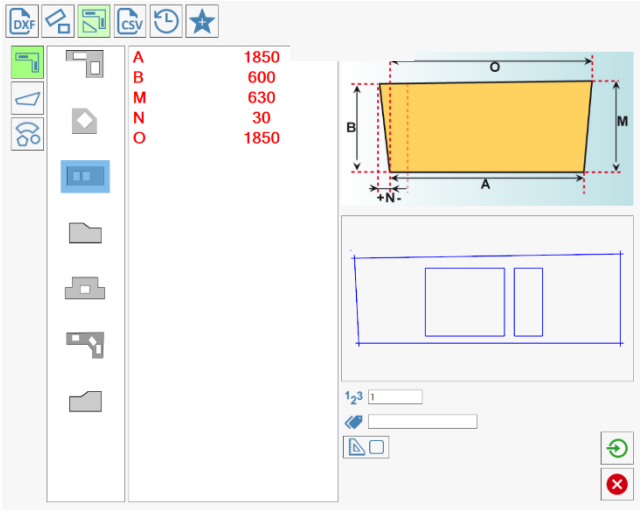
on the piece creation page, activate “FIGURES”, which will display the following screen:



To load the pieces, select the type of work “KITCHEN”, “STAIRCASE” or “FIGURES” and the relative figures will be displayed accordingly. Select the desired shape by pressing on the list of pieces and the image will be uploaded with the dimensions necessary for its creation: fill in the blanks of values with the actual measurements of the piece. To do this 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 associate with the piece; if left empty, the thickness of the current material will be taken.

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

In the example above, it can be seen that not all the lines are correct, and therefore only some will be imported into Parametrix, namely lines 2,3,9.

Errors in the example, which do not allow importing of the piece:

Line 4: The number of pieces is at zero
Line 5: The width of the piece is at 0
Line 6: The height of the piece is at 0
Line 7: All the fields are at 0 (except for the name)
Line 8: All the fields are empty (except for the name) If Height and Width are both empty, the table is considered terminated.

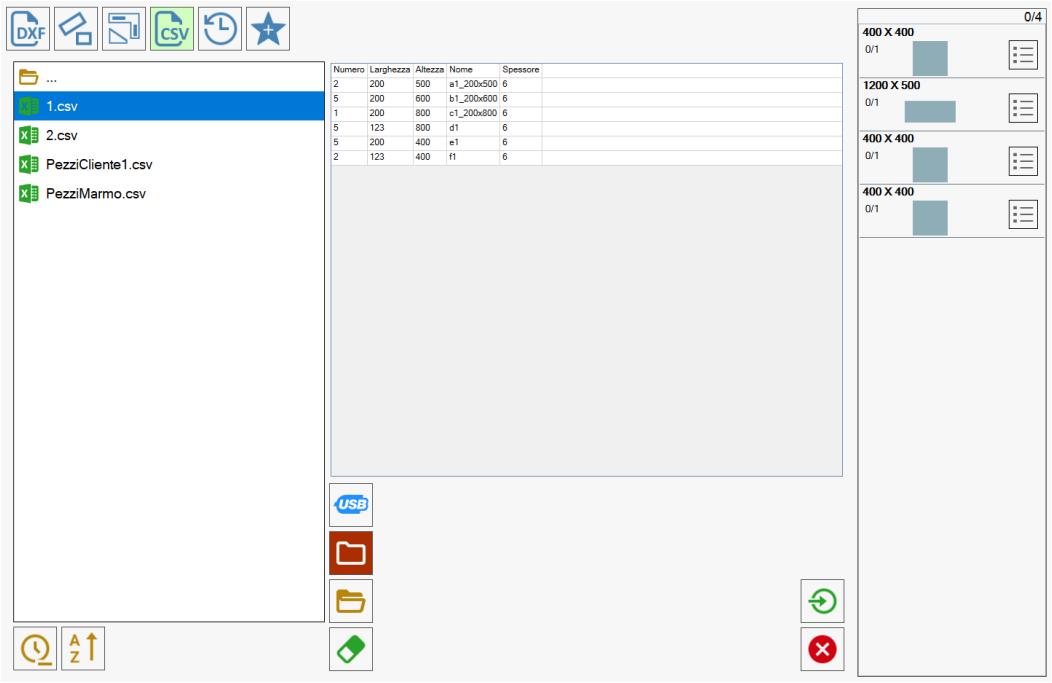
Note	If the name field of a single line is empty the piece is still created and the name is associated with "WidthxHeight"

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.

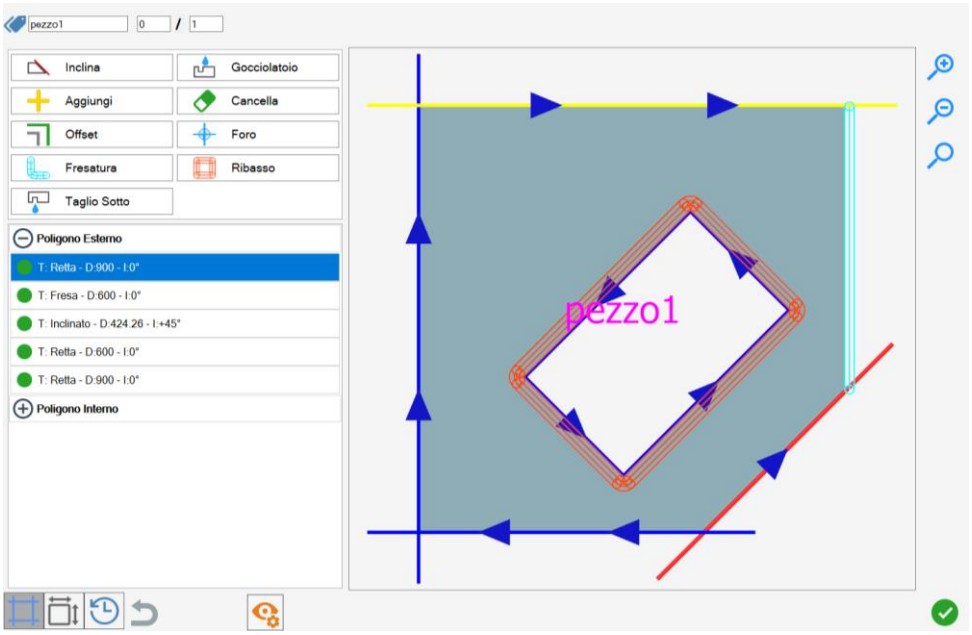
The buttons  ,  ,… behave in the same way as the buttons described in *DXF Import section*



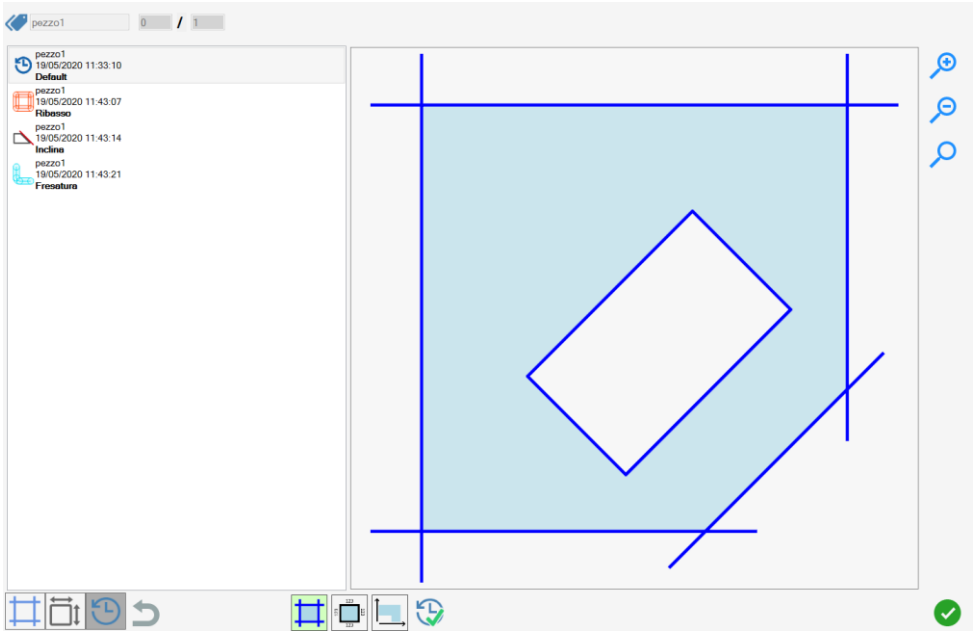
Press the button  creates the piece in the list of pieces inserted.

5.1.5.5 HISTORY

It allows displaying all the changes made to the current piece and to take it back to its previous state.



The following screen will be displayed after pressing the button 



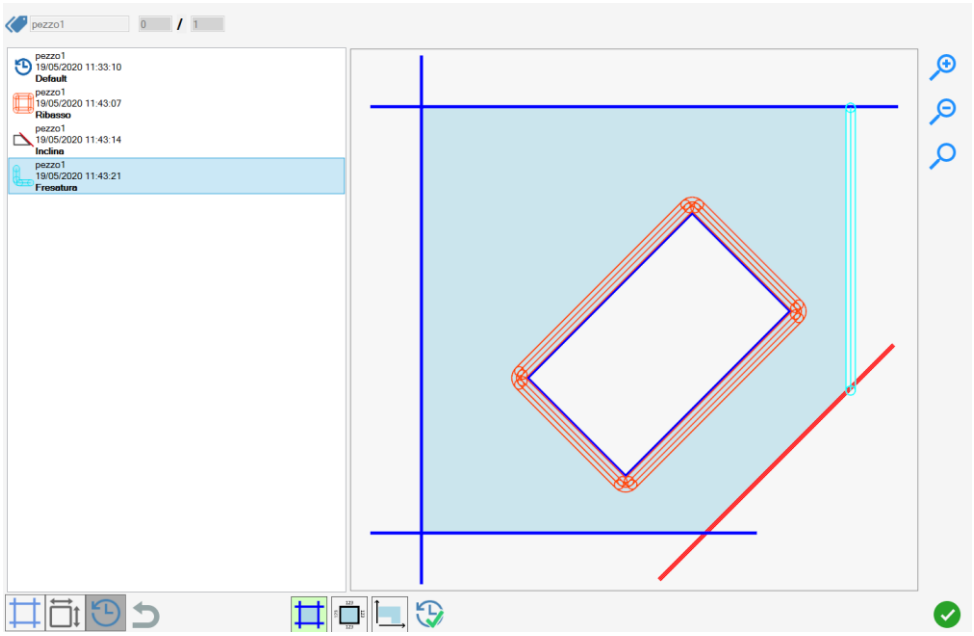
The status of the current piece without changes is called "Default". The bottom part of the panel contains these four buttons. Three of them are display utilities:

	The piece is displayed showing the cuts
	The piece is displayed showing the values
	The piece is displayed showing the overall dimensions

The button  restores the current piece to the state selected in the list on the left. When a previous state is applied, changes after the selected state are removed, except for the default state: this 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

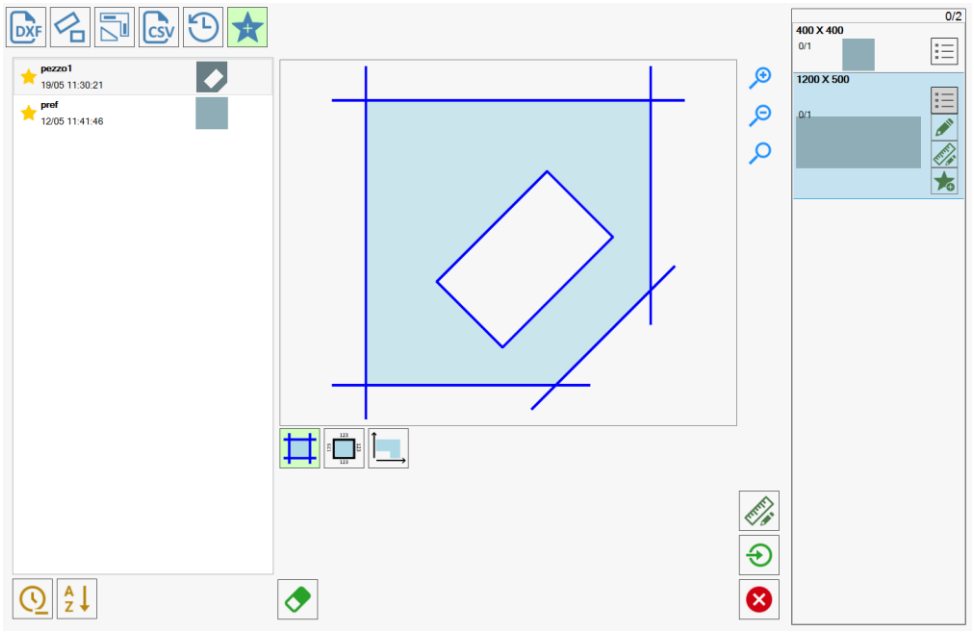
If changes have been applied to the current piece, they will be displayed in the list on the left



5.1.5.6 FAVOURITES

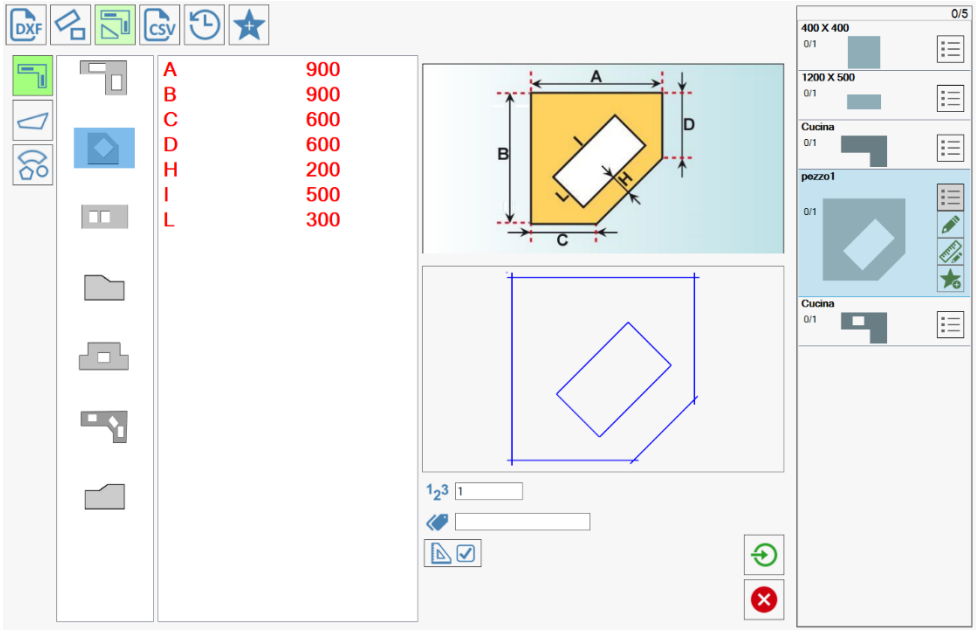
You can add a piece to the list of favourites by clicking the button  next to the piece and then the button 

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



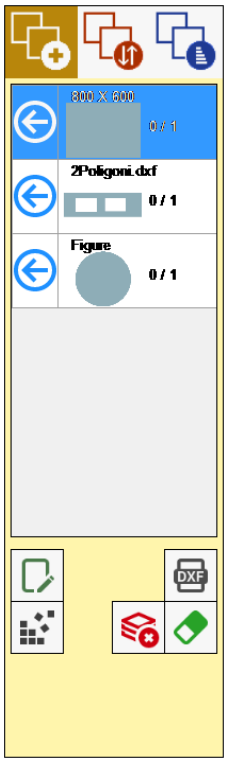
5.1.5.7 CHANGE SHAPE

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 INSERTED PIECES

When a piece is created (see chap. 4 Piece Creation) the program adds it to the list of pieces; from the list it is possible to insert them into the work area or make some changes to the piece and check the number of remaining requiring machining.

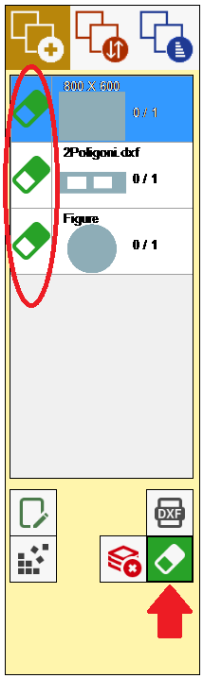


Each line of the list contains 3 pieces of information on the production of the piece:

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

• Number of pieces inserted/to be inserted	
• 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.

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

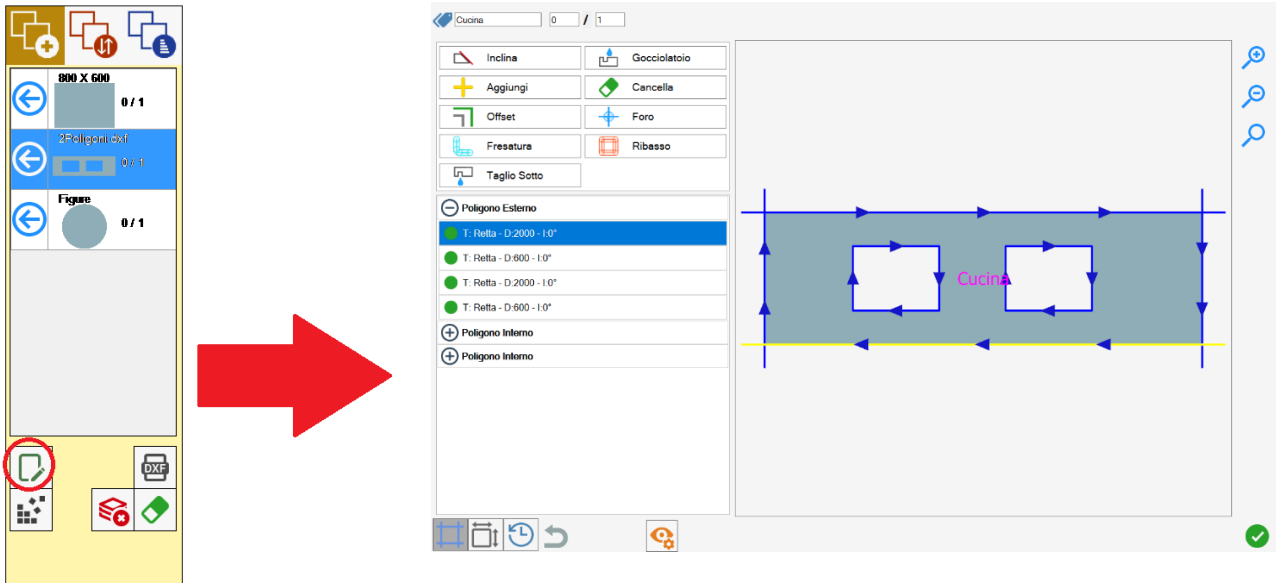
To cancel all the table contents, press the button  . Confirmation is requested for this operation.

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  .

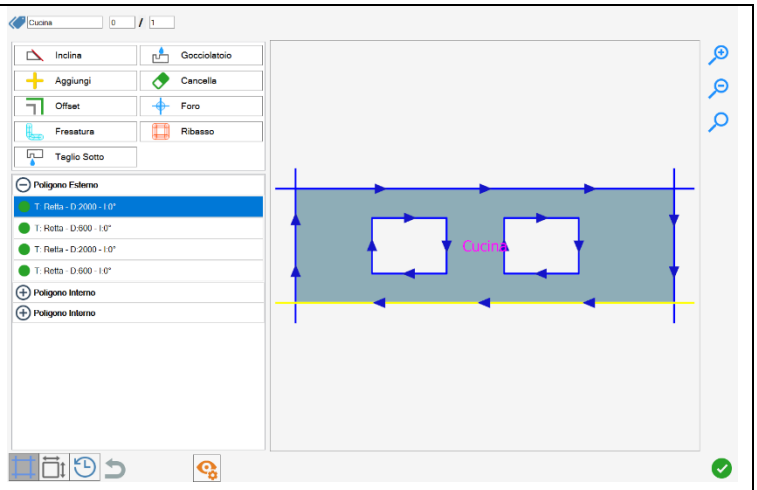
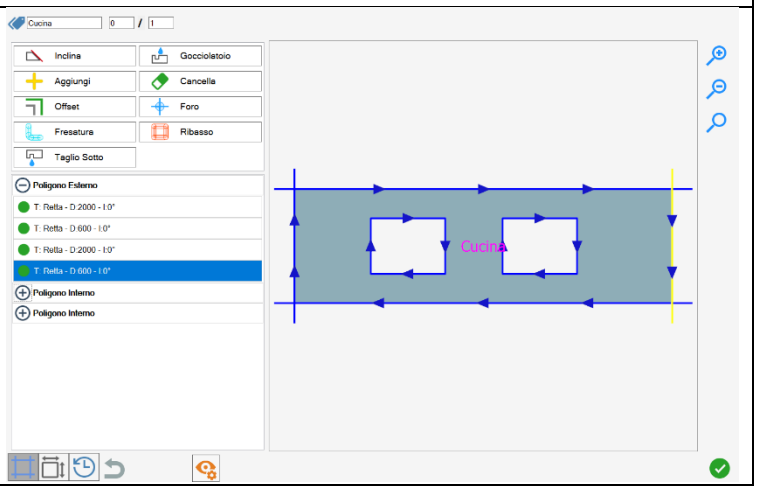


Certain functions are also present on the main page of Parametrix, though by changing cutting through the "Modify Piece" screen all the pieces of that line will have the desired machining whereas if the machining is set on the work area only the selected piece will be changed.

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

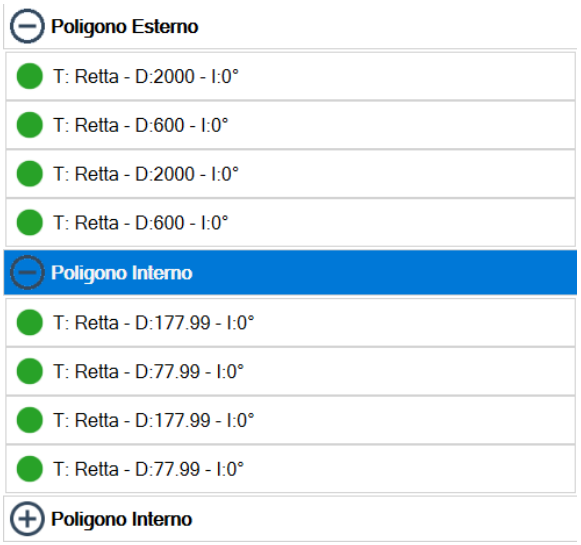
"Modify Piece" page	**Work Area**
Modifying the angle of one side, ***all*** the rectangles will have the angle changed at the time of insertion into the work area.	If we modify the inclination on the work area the individual piece will be changed, not all the pieces of the line.

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.

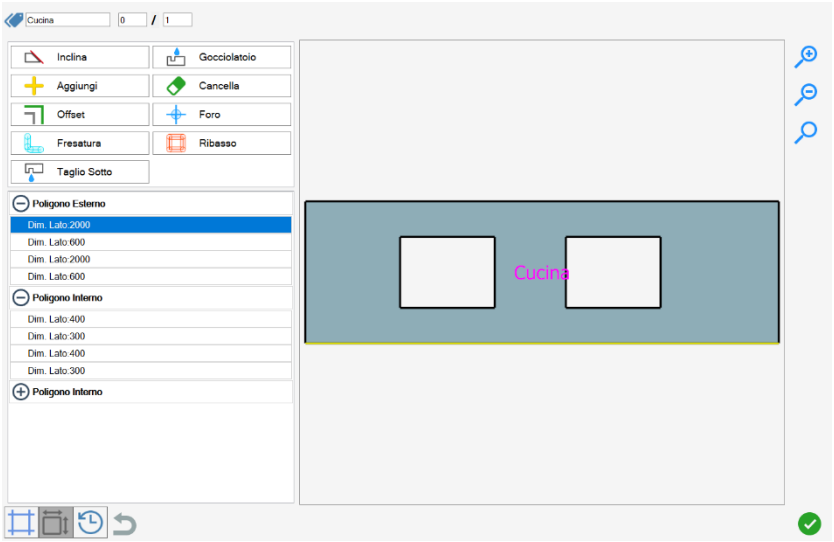
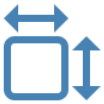
Information on lines

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 OPTIONALS

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. Continuing to press cancel it is possible to return to the initial conditions of the piece. **N.B.:** the button works for as long as the "Modify Piece" panel remains open on the piece in question. If the panel is closed, it will not be possible to cancel the changes to the piece in question, 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 	

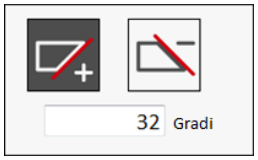
5.1.7.3 FUNCTIONS

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 disc was unable to remove, ...

Not all the pieces can use these functions but where this is possible a screen message will be shown.

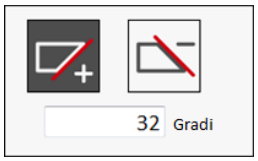
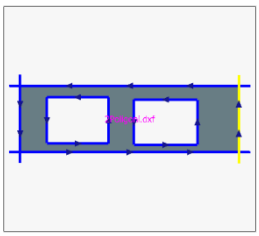
5.1.7.3.1 INCLINATION

The 'Inclination' function defines a machining mode with the A axis different from 0. Pressing the button displays in the bottom right the following window.



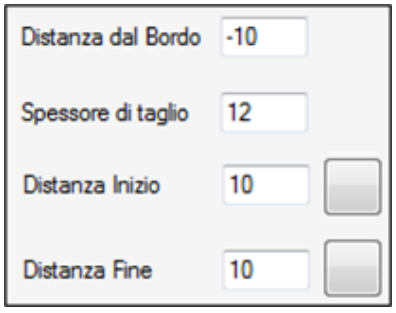
Via the buttons  decide the direction of the square with respect to the piece, while the text below indicates the value measured in degrees of the machining. After entering the parameters select the cut to be modified. The inclined cut appears in a different colour.

Example of 32 degree cut setting

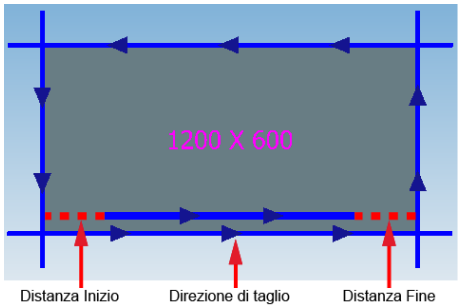
On the "Modify Piece" page, press the "Inclination" button.	
1. Select the square and enter the desired value.	
2. Select the cut on which to set the inclined cut, directly on the piece or on the list as explained before.	

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 disc) 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:

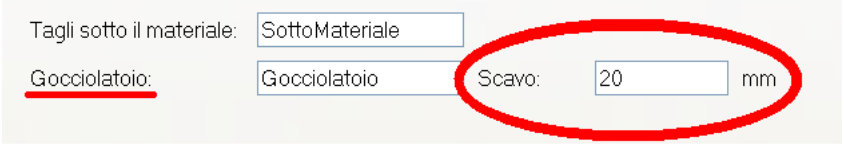


Distance from the edge is the distance from the selected cut to the drip machining.
Cut thickness is the measurement of the machining width. As already stated, it is possible to perform a Drip of width greater than the thickness of the disc. The program will make parallel cuts until reaching the thickness 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.

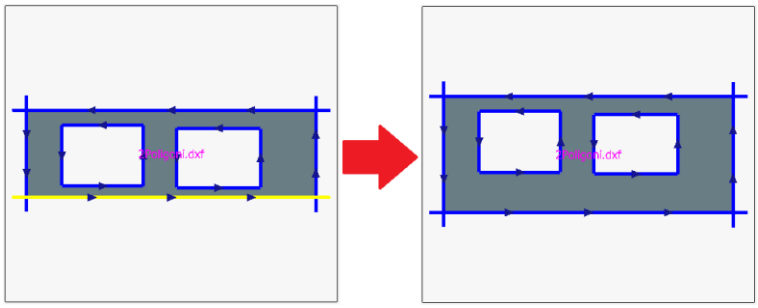
Penetration into the material is set within the parameters of the machine. To modify the data, follow the procedure shown in the table:

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)

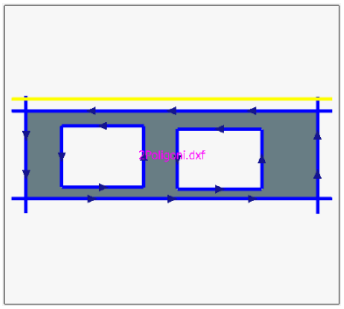


To activate the function, press the “Offset” button and enter the displacement value in the box to the right.

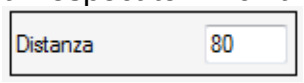
Having inserted the measurement select the cut on which the offset is to be made.

5.1.7.3.4 ADD CUT

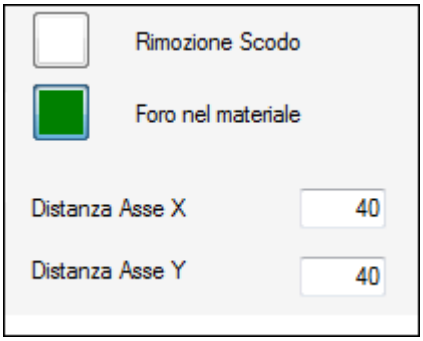
The "Add Cut" function is used to add a cut parallel to the cut selected. The distance between cuts is defined by the parameter. Where the value is greater than 0 the cut is calculated out of the piece, while if the value entered is less than 0 then the cut will be made inside the piece.



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

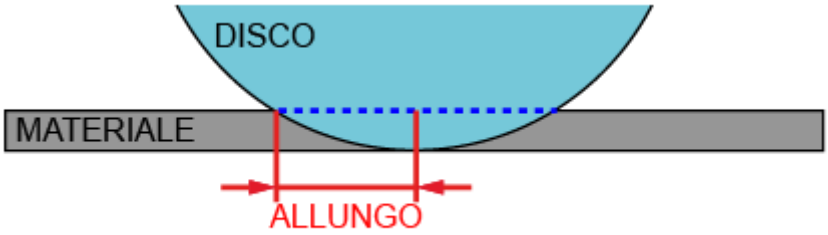


5.1.7.3.5 HOLE

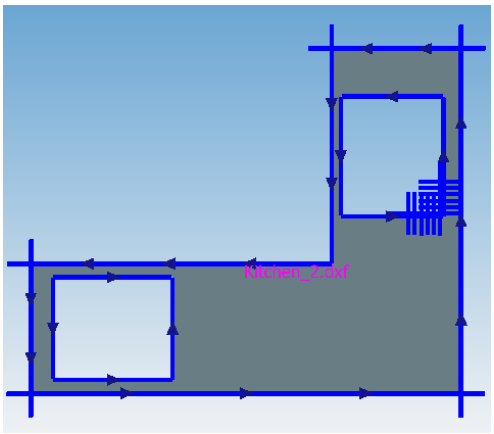


With the 'Hole' function it is possible to insert a series of holes to remove the material that the disc was not able to cut (as it would have damaged the piece), selecting "**Avoidance Area Removal"** or create a Hole in the material based on coordinates with "**Hole In The Material**".

Removal of material left by the Disc (Avoidance Area Removal)
This processing is only enabled on cuts Start-Edge and End-Edge and automatically calculates the number of holes according to the distance of the disc length (*see figure below*).

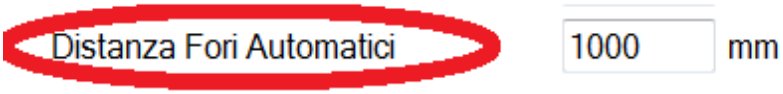


To insert a series of cuts and to complete the cut with disc, the function must be activated using the “Hole” button and selecting the cut to be finalised. Parametrix inserts a series of holes along the cut, calculating the length, as in the image.



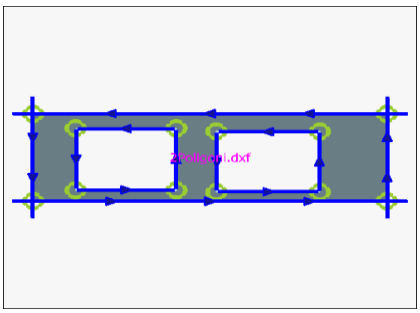
NOTE: *In the case of Start-Edge and End-Edge the holes will be inserted based on the selection position. Selecting the cut near the end will insert the holes at the end while otherwise they will be calculated at the start.*

The distance between one hole and another is set within the parameters of the program; to change the value, the following points are required:

Press the button  located in the upper left of the program	
After pressing the button, on the first page that appears, look for the line "*Automatic Holes Distance*".	

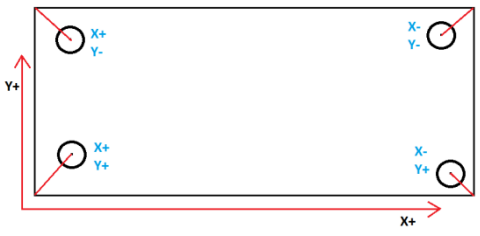
Again on the first parameters page, we find the “Hole Penetration” parameter, which allows the forced penetration of all holes.

Hole in the material
By enabling the "Hole in the Material" button it will be possible to add a boring operation, referring to the specific points highlighted in the figure (see small circles in the figure below).



By selecting one of the highlighted circles, a Hole will be created in the coordinates indicated in "X Axis Distance" and "Y Axis Distance" (visible in the first image of this paragraph).
The distance will always be in relation to the point being examined to create/modify the hole and the centre of the latter.
In the case of figures presenting arcs, their central points will also be highlighted.
In the case of circles, only the centre of the Piece will be displayed
If the hole can be built, as the inserted coordinates do not take it completely out of the piece, the cut will be highlighted in yellow and inserted in the list of cuts of the piece shown to the side.

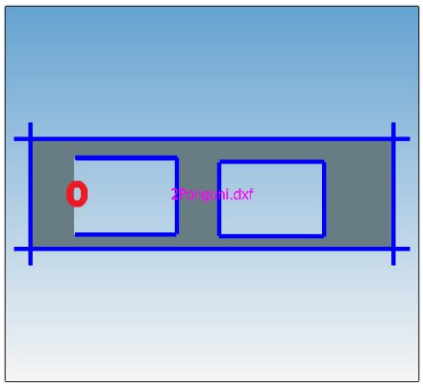
Note: With respect to each point, the part on the right of the circle (for the X) and the part above the circle (for the Y) is considered positive. In summary:



Modify Hole
Once the cut is created, as long as it remains selected, i.e. without moving to another cut or creating another "Hole in the material", it will be possible to change its position. As a starting point reference is always made to the original starting point that had the cut, i.e. the "circle" of the figure with which it was created.

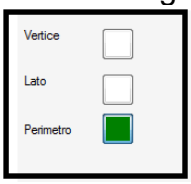
5.1.7.3.6 CANCEL CUT

The "Cancel Cut" function allows the user to cancel a cut of the piece.
To use this feature, press the “CANCEL CUT” button and select the cut that needs to be cancelled.

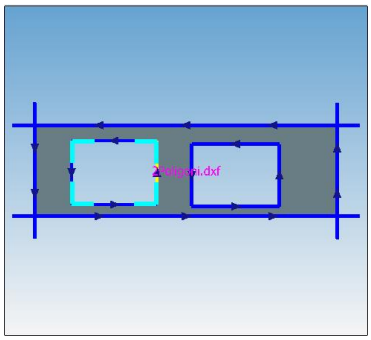
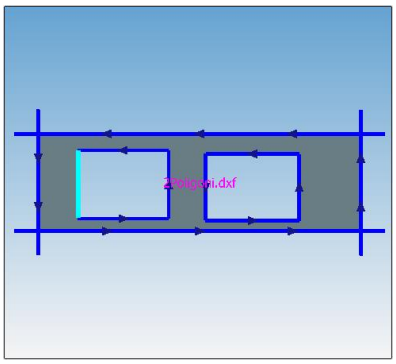
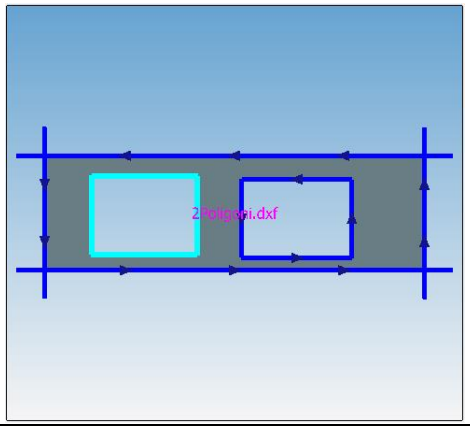


5.1.7.3.7 MILLING

With the milling option it is possible to use the cutting tool to perform the machining of some or all parts of the piece rather than using the disc.
After pressing the “MILLING” button, the machining mode must then be selected.

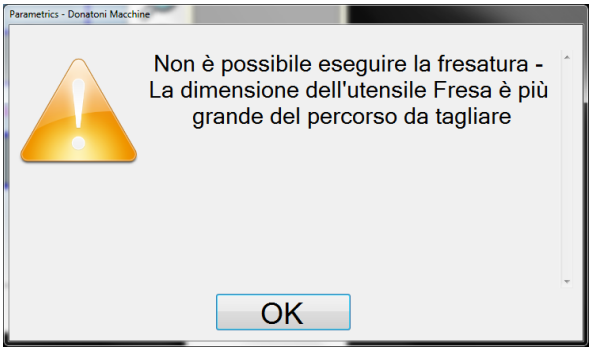


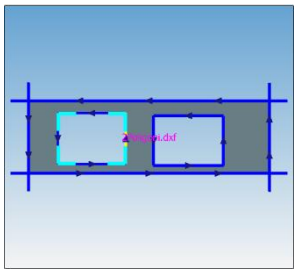
Having made the choice with which to perform the cut, select on the piece the side on which to apply it.

MILLING METHOD	
Vertex Mode 	This mode has the same purpose as the hollow bit, i.e. to remove the material that the disc was unable to without damaging the piece. This particular use of the cutter does not replace the cut with the disc but is only used to refine certain points of the piece. The size of the cut is taken as the distance between the end of the cut with the disc and the vertex of the piece.
Side Mode 	With this mode it is possible to use the cutter tool to cut one side of the piece. Side mode, compared to that of the vertex, does not accompany the cut with the disc (as can be seen from the image above), but replaces it completely.
Perimeter Mode 	Perimeter mode uses the Cutter tool rather than the disc tool to perform the cut over the entire polygon of the piece. Perimeter mode, compared to that of the vertex, does not accompany the cut with the disc (as can be seen from the image of Vertex mode), but replaces it completely.

Note:

1. For all the modes displayed, the cutting thickness is cutter is the tool diameter; varying this element automatically changes the cuts, updating them with the new measurement;
2. For Vertex mode, a variation in the thickness of the material or in the diameter of the disc involves the lengthening or shortening of the cut, thus adapting it to the new measurements. This, however, is only the case if the cut in vertex mode was created with Parametrix and was not imported from DXF. In that case the measurement does not change.
3. If the Cutter is too big to cut the side selected an error message will be displayed on screen:





5.1.7.3.8 LOWERING

The lowering function is primarily used to lower by a certain amount the holes of the kitchen tops in order to able to insert a sink. The tool used to perform this processing is the Cutter.
When the “LOWERING” button is pressed, the necessary distance from the SIDE of the piece must be selected, as well as the depth at which to make the Cut.

If the distance is positive the cut will move within the piece, while if it is negative, the cut will move outside the piece. If this distance exceeds the size of the diameter of the tool being used, the cut will be performed in pass mode until the required path is covered.
With regard to penetration the largest value that can be received is equal to the thickness of the material.

5.1.7.3.9 CUT BELOW

This type of cut is possible if the relevant mechanical optional is present.
Parametrix checks the overall dimensions of the piece. If the overall dimensions allow it, the machine takes the workpiece to the optional table where it performs the cut.
The parameters to be entered are the same as for the drip except for the thickness of the cut, which is not present.

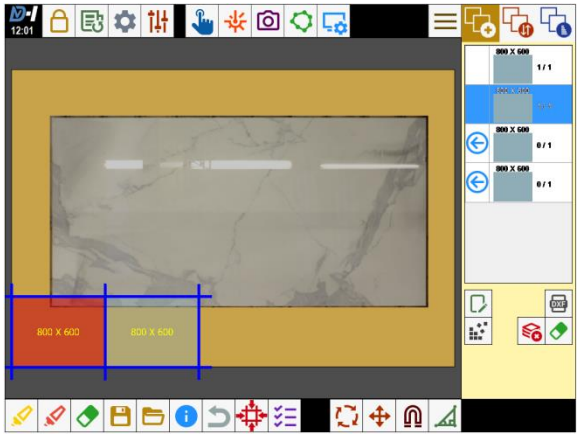
5.1.8 WORK AREA: POSITIONING OF PIECES AND OPTIONALS

Parametrix positions pieces inside the work area of the machine to create the cut file. Having acquired the slab, having created the perimeter and having created the pieces it is necessary to import the pieces inside the slab. The button  on the pieces table is used to place the piece within the work area. It is possible to work with 1 or more pieces at the same time.

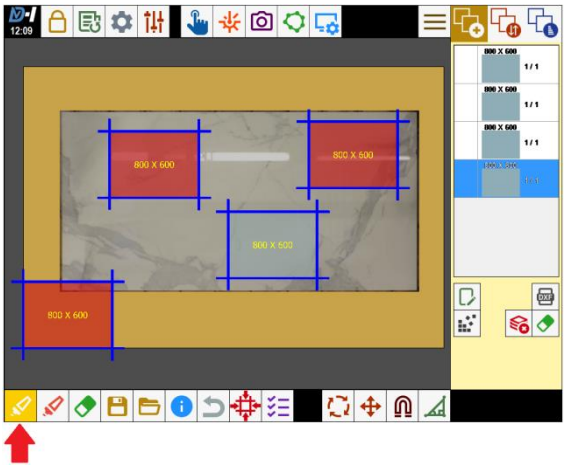
If the pieces are not perfectly inside the slab, a caution/warning signal  will appear in the top left.
The appearance of this signal indicates that problems may arise during the cutting stage, however none of the program functions will be blocked.

5.1.8.1 SELECTION OF PIECES

When the piece is placed in the work area, it is light grey. Selection of the pieces takes place by clicking on the piece itself: the selected piece turns red.



It is possible to select 1 piece 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  .

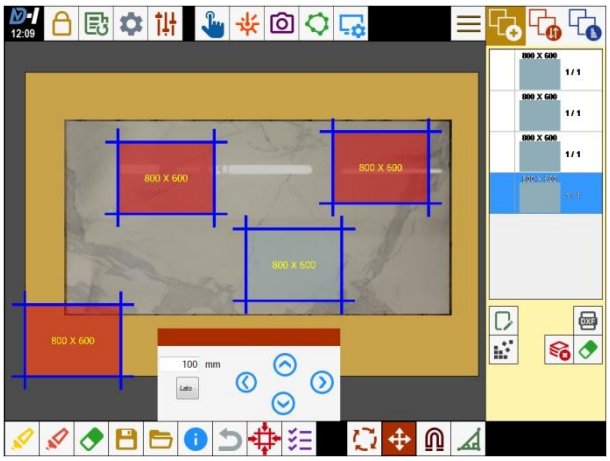


5.1.8.2 DISPLACEMENT OF PIECES

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:

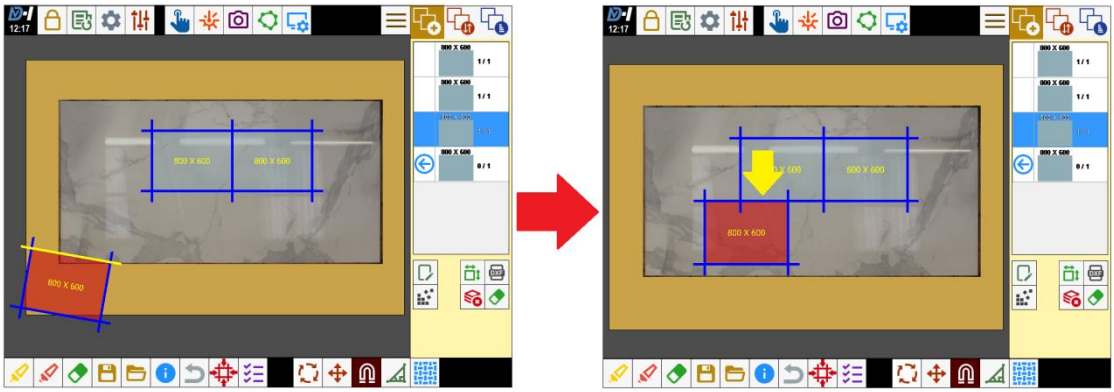
1. **Rotation** of the piece takes place by clicking **outside** the perimeter of the piece and by moving the finger in the desired direction.

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

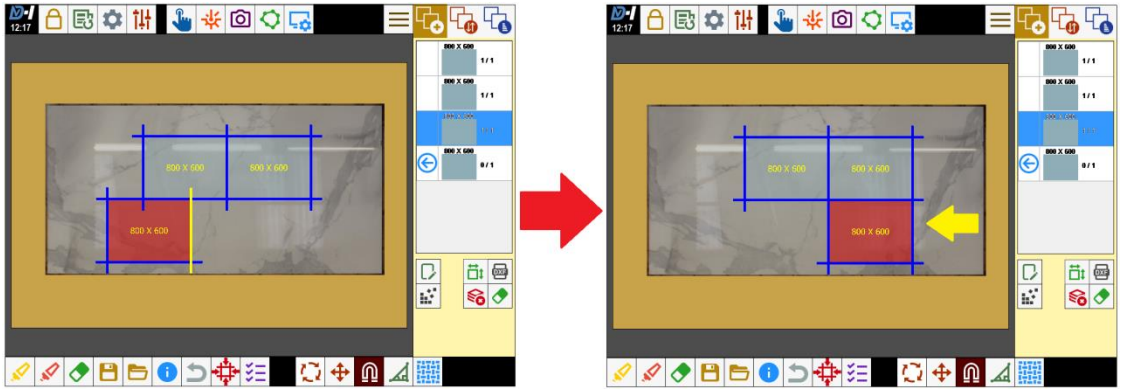
Note: *The **Move** and **Rotate** functions are applied to all the pieces selected. Therefore, it is possible to move multiple pieces simultaneously*

5.1.8.4 MAGNET FOR CUTS

With the cuts magnetic system it is possible to attach 2 cuts together. Where the cuts are not parallel, the piece 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 piece: the **FIRST** cut selected moves towards the **SECOND** cut selected (see example below).



It is also possible to magnetise cuts with adjacent pieces as shown below:



It is not possible to magnetise cuts together with the cutter or cuts with disc and cuts with cutter.

5.1.8.5 MAGNET ON LINE LASER

You can draw one side of a piece on line laser.
After activating the line laser, insert the piece in the work area. Activate the magnet function, click on the magnet cut and on the laser line of interest. The cut will be magnetized on the chosen laser line. The laser will coincide with the disc centre.

5.1.8.6 INCLINED CUT

In the program, it is possible to set the cuts with inclined disc. For greater understanding refer to *Paragraph 7.6.3.1 Inclination*.

5.1.8.7 DELETE PIECE

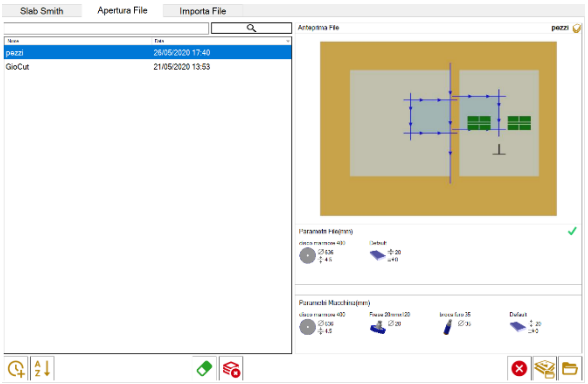
By pressing the button  , the selected pieces are removed from the work area and are inserted in the list of pieces to make them available later.

5.1.8.8 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.

5.1.8.9 SAVE AND OPEN

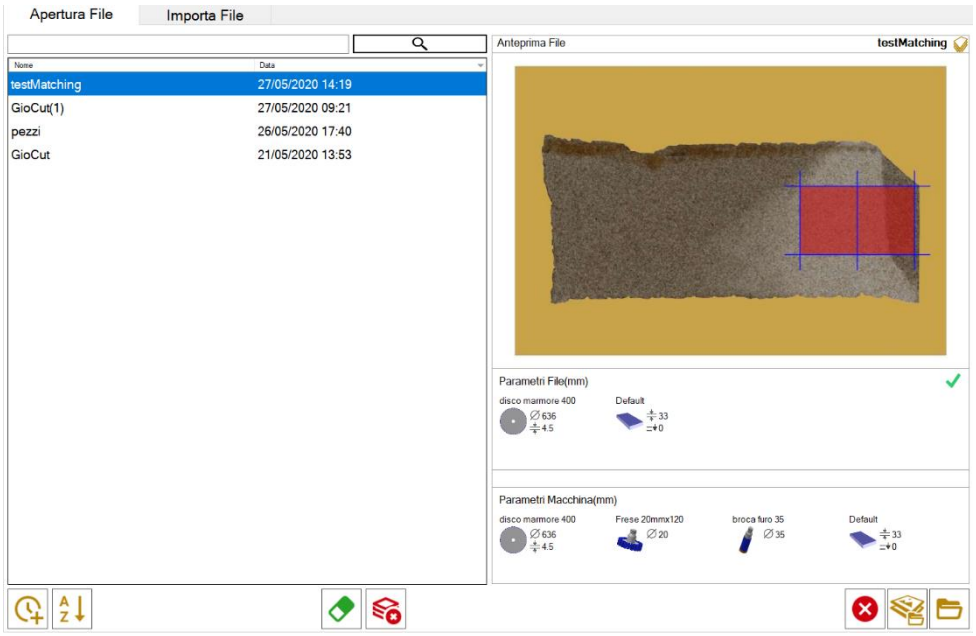
It is possible to save the current work in progress: pieces, plates, modified cuts, ... with the button  . With button  it is possible to choose which configuration to recover and the program will open it as it was when saved. The following window opens:



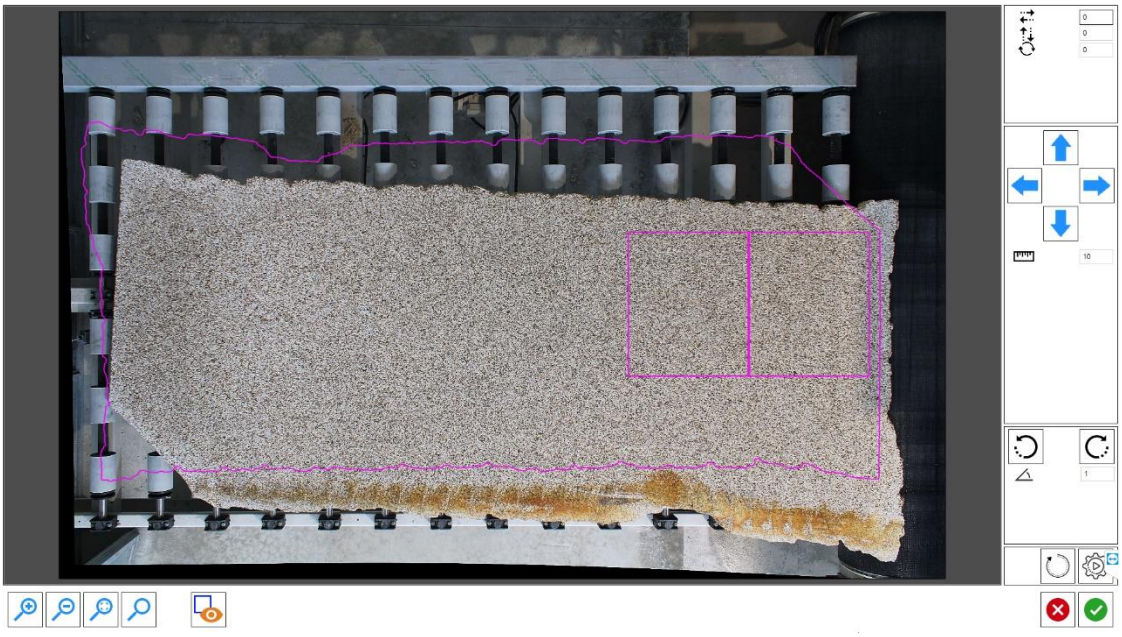
it is possible to view a preview of the saved process, the tool materials and material with which machining was carried out
To fully open a machining configuration, the file parameters must be identical to the machine parameters. If they are identical, a green tick will appear on the right, otherwise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

5.1.8.9.1 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 specific button in the Open / Import File dialogue box:



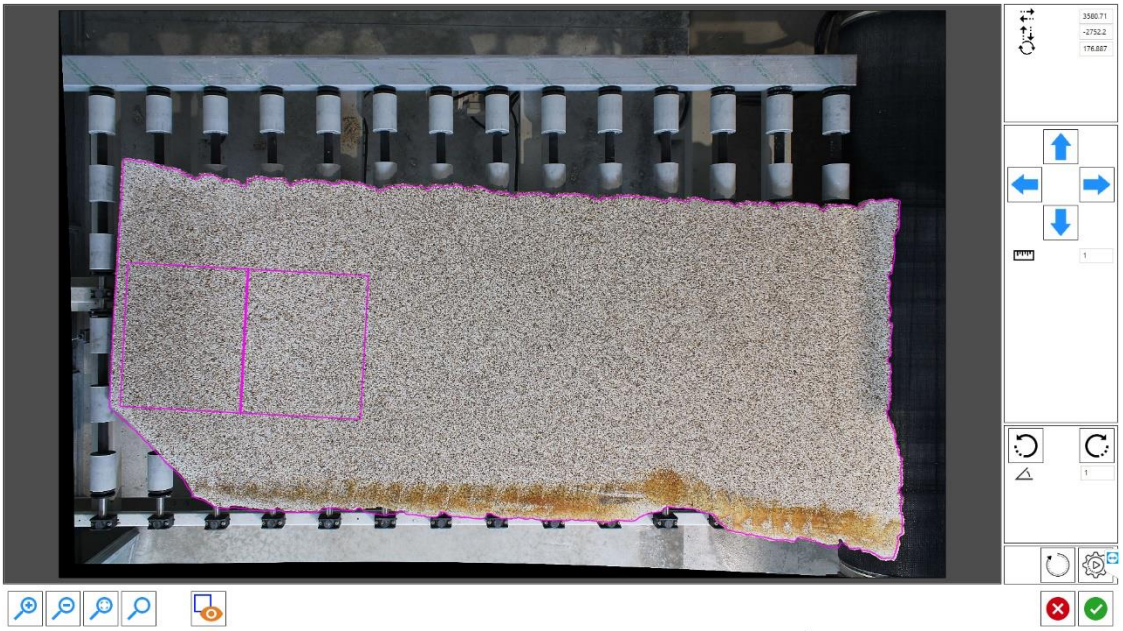
The following slab-matching window is opened when pressing the button  :



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.

5.1.8.9.2 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.

5.1.8.10 IMPORTING AND EXPORTING

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

5.1.8.11 DELETE

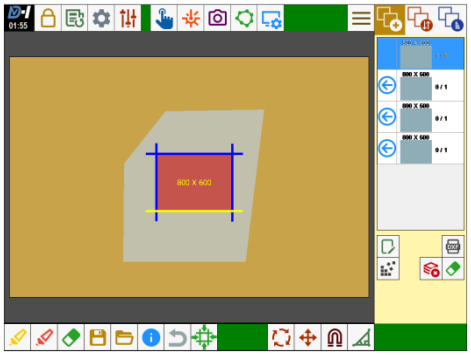
Allows deletion of the operation performed by pressing the button 

5.1.8.12 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.

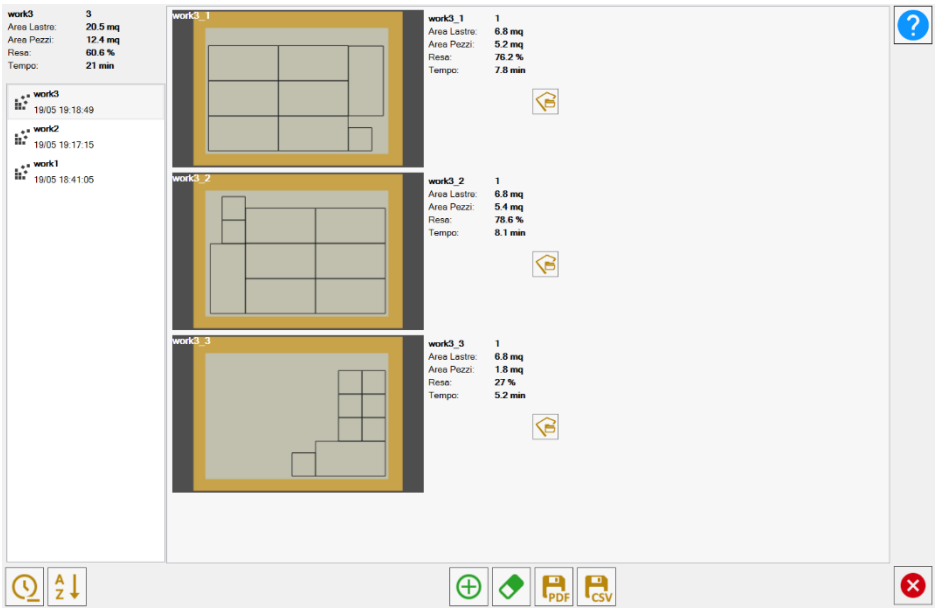
5.1.8.13 MACHINE ON CUT

The objective of the function is to control the position of the piece in the work area relative to the position in the machine. When "Machine on cut" is enabled, a cut is selected on the screen and the machine moves into the relative position.
This operation is useful in order to check if a cut is in the material.



5.1.8.14 COMPLETE NESTING

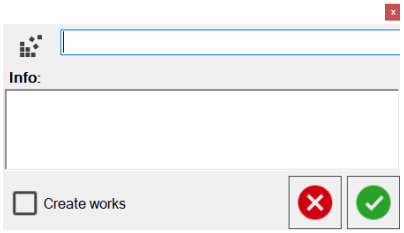
This function allows nesting to be performed on the pieces of the list on more slabs till pieces are exhausted. The following window is opened when button is pressed:



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

- Slab area
- Piece area
- Yield
- Time

The following window is opened when button is pressed:



This allows a new evaluation to be started, by entering name, any additional information, and a flag to create the relative processing phases.

Processing phases can be deleted from the list on the left by pressing the button .

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

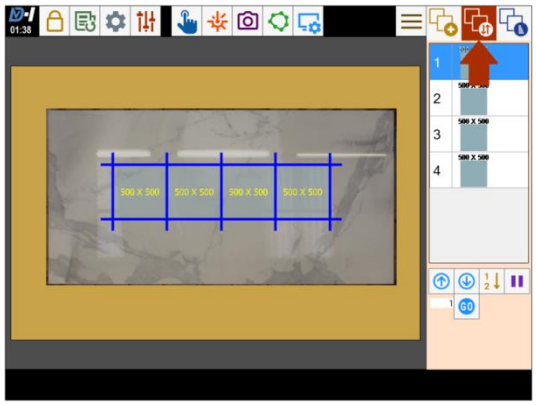
Use button near the slabs to take the processing directly to the work area.

5.1.9 PIECE CUTTING ORDER (OPTIONAL)

Having positioned the pieces in the work press  (visible in the image) to access the piece cutting order system.
The arrangement of the pieces in the right-hand list indicates the order in which they are cut. It is possible to change the order in 3 ways:

1. Arrows
2. Positioning
3. Selection by click

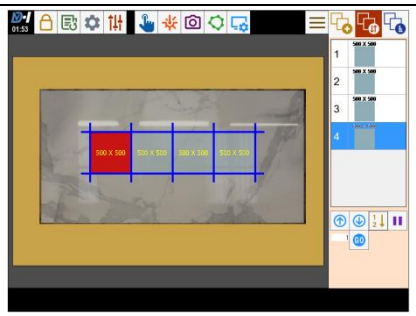
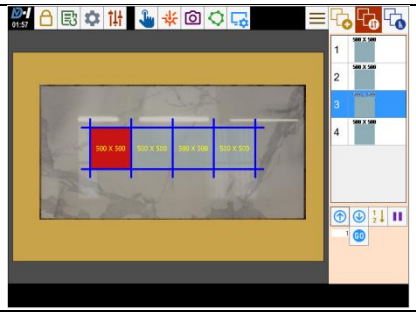
The following paragraphs explain the various sorting systems.



During the piece cutting ordering, it is possible to set a pause between one piece and another. The program will make all cuts of the piece before the pause and will then stop the machine and wait for a start.

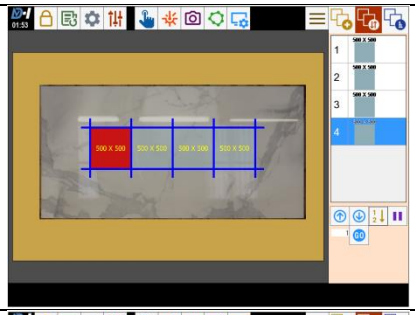
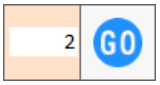
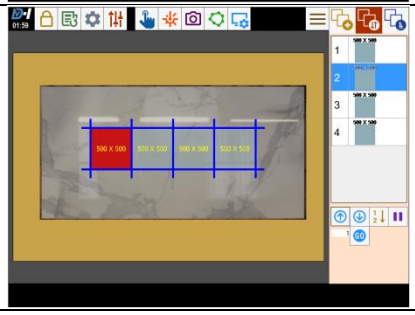
5.1.9.1 PIECE POSITIONING ARROWS

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.	

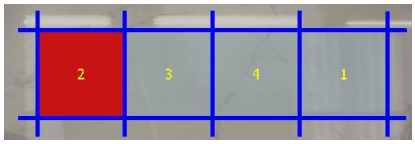
5.1.9.2 POSITIONING

The button  moves the piece that is currently selected in the list to the desired location.

1. Select the piece to be moved. In this case as visible from the image is 4.	
2. Indicate in the text box the position that the piece should fill 	

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:

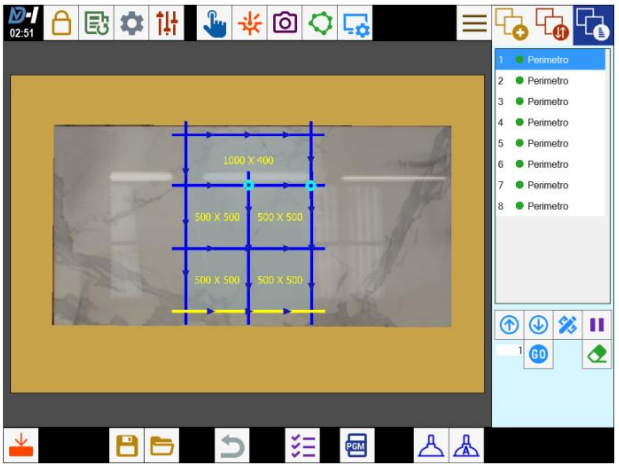


press on the pieces in the order in which they are to be cut. The currently applied order indicates that the first piece cut is the rightmost one, the second the leftmost one, etc.; to change the order, click **inside** the piece to be first, second,...

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

5.1.10 MANAGEMENT OF CUTS AND DISPLACEMENT

Use the third button in the top right  , to access cuts administration:
The cuts in order of execution are loaded into the list on the right; furthermore, the directional arrows of the cut are drawn on the screen.



It is possible to move the chronological order of the cuts 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 '7.8.1 *Piece 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 SELECTION OF CUTS

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

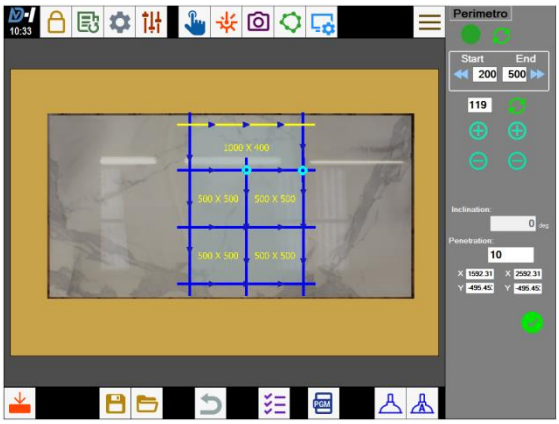
These buttons allow multiple selection of cuts, respectively:

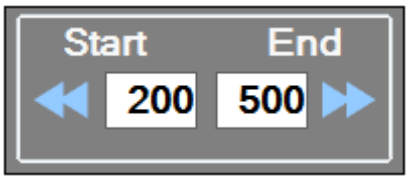
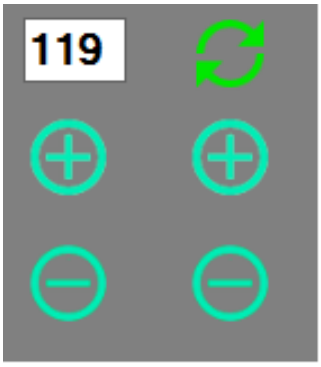
- Selection of cuts
- Selection of holes
- Selection of milling
- Selection of lowering

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

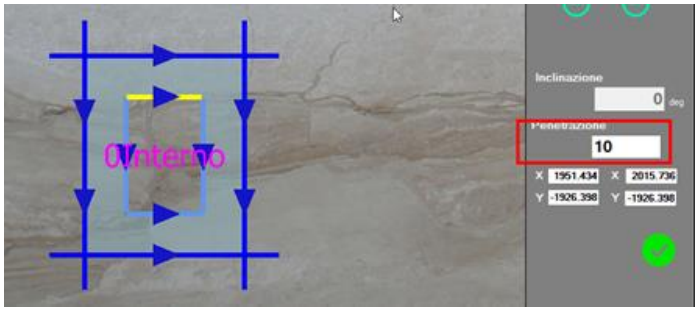
Select the cut to be modified and press the button  to access the cut edit screen (figure to the right).
The arrows define the direction and indicate the start and end of the cut. The table below explains the cut edit panel buttons.



	The button  reverses the start of the cut with the end and vice versa	
	The arrows allow lengthening of the start or the end of the cut, up to the edge of the slab. The values in the boxes are used to lengthen the cut of a certain quantity outside the slab	
	 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 disc diameter and material thickness) and enters the value on the data panel.	

Once the cut has been changed it is necessary to press the confirm button  .

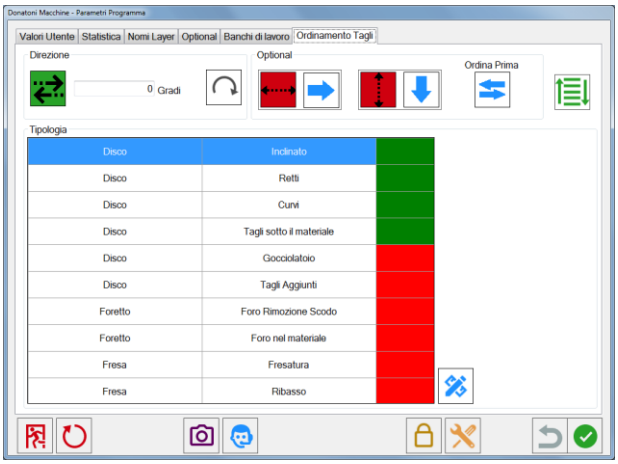
The installer parameters include the “Internal Cuts Penetration” parameter, using which it is possible to set a fixed penetration for all internal cuts. An internal cut with different penetration will be highlighted in a different colour.



5.1.10.3 ORDERING CUTS

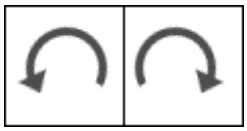
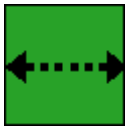
It is possible to order the cuts in such a way that the machine performs the fewest possible displacements or that a certain type of cut works first with respect to another. These sortings are only applicable to non-SX3/SX5 machines and must be in a situation where the cutting list has not yet been created, i.e., in the section for inserting the pieces  , or in the part for ordering of the pieces  .

From one of the two previous sections select  and move on the page: "Ordering Cuts”.



The buttons with RED background do not perform their function

	**Enabling Ordering use** It allows using of orderings on cuts.
	**Ordering by Direction** If enabled, according to the grades passed (used as a starting point) and to the orientation to be followed, all the cuts are

	ordered.
	It allows insertion of the degrees from which to start sorting by direction.
	According to the icon visible, the cuts are recovered in a clockwise or anti-clockwise direction with respect to the chosen angle.
	**Ord. Right\Left– Left\Right** All the cuts are sorted in the same direction. Depending on the arrow direction of the button to the side, the first order is from RHS to LHS  or from LHS to RHS 
	**Ord. Up\Down– Down\Up** As above. Cuts from above down  or from down upwards  .
	It allows indicating whether to execute sorting of the cuts first from right to left (or vice versa) or from up to down (or vice versa)

Sort by Type of Cut

Another sort of ordering available is *"Sort by Type"*. This ordering allows a certain type of cut to be executed before another; e.g.: perform *"Inclined Cuts"* first and then cuts with *"*Milling".
As can be seen from the initial image of this paragraph, in the table there are a number of green elements and in a certain position.
These two pieces of information determine:

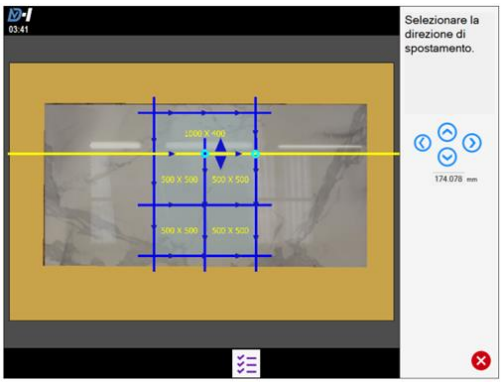
1. How the cuts are ordered (which is executed before the others)
2. Which is ordered (<u>only</u> those in green)

To modify this list it is necessary to select the button  . This will bring up other elements that will allow modification of the list.

	They allow movement of a type of cut above/below with respect to another
	It moves all the cuts highlighted in green at the beginning of the list
	It moves all the elements highlighted in green at the bottom of the list, also reversing the order of the elements inside it. That is, the first becomes the last, the last becomes the first,...

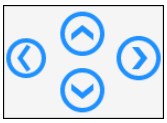
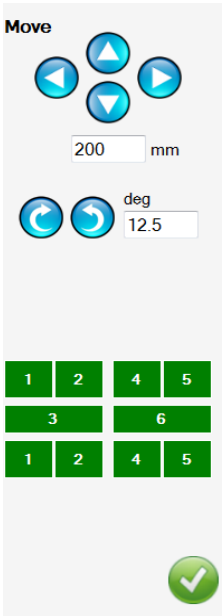
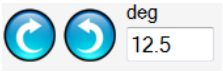
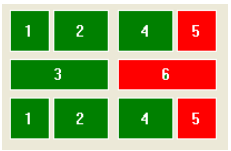
5.1.10.4 MATERIAL MOVEMENT WITH SUCTION CUPS

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.	

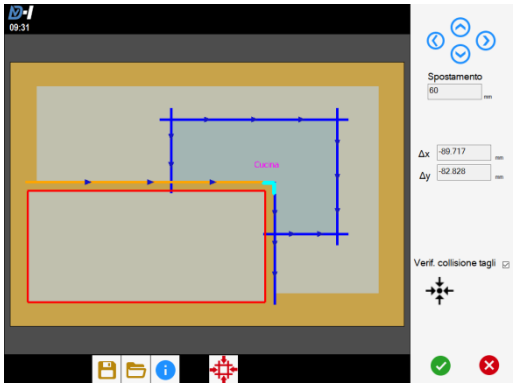
It is possible to use **automatic movement**  .
This means that the program automatically calculates which cuts are to be used for the required movement(s). This function does not always solve all the issues but it is possible to act in manual mode to improve the situation.
The program will attempt to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

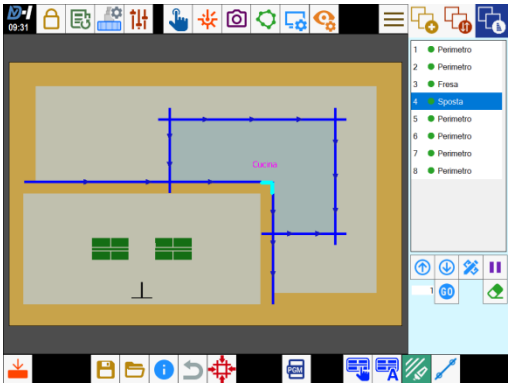
5.1.10.5 N-CUTS MODE

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

⚠	**ATTENTION:** *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: the cuts can be either cuts with disc (straight only) or with milling. The first and last cut must be cuts with disc.
The program extends cuts with disc 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.
It is possible to directly insert the absolute values of X and Y with respect to the initial position, or drag the selected slab directly onto the screen.
After confirming with the green button, the program runs a check to make sure the slabs and cuts are not overlapping. If “Check for cut collision” is deselected, the program 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.

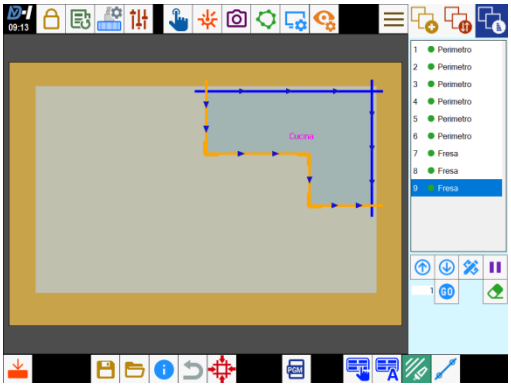
3. The selected cuts cannot detach the piece due to the disc avoidance area.
4. The length of the selected cuts will ruin other pieces.

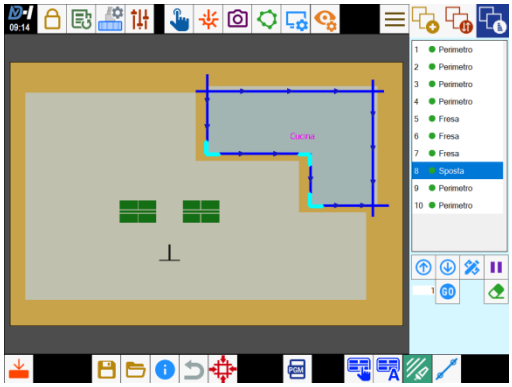
Check for non-overlapping slabs and cuts
If the program finds that the displaced slab will overlap with another slab, or the cuts to be made will overlap with other slabs, it automatically adds an additional displacement in order to prevent such overlapping.

	**ATTENTION:** *The operator is responsible for checking the feasibility of the movement proposed by the program.*

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

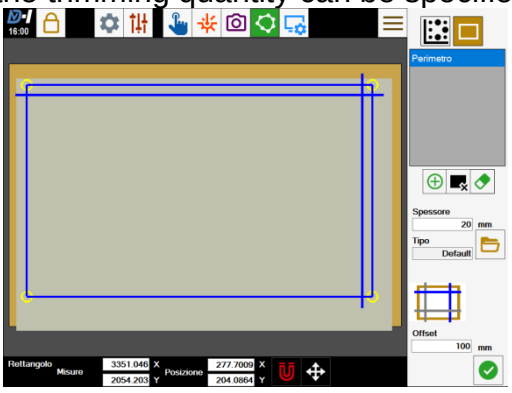
To avoid the disc avoidance areas in the leftover material, milling can also be performed on the external corners and cuts can be shortened using the “Modify Cut” function.



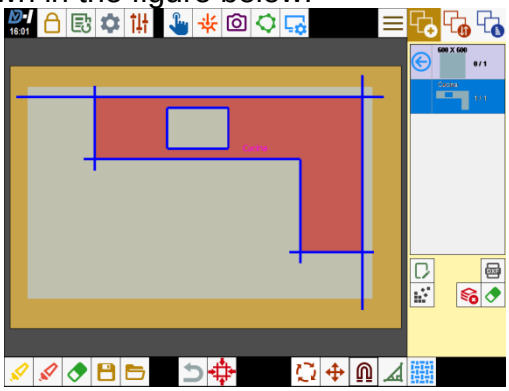


5.1.10.6 SLAB TRIMMING

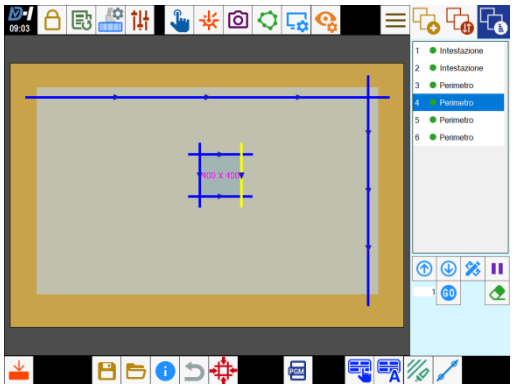
Slab trimming is a feature that forces trimming cuts into the machining process. The purpose of these cuts is to trim the slab along its edges.
The feature can be activated in the perimeter selection panel. In this screen we find the button  which shows the four cuts that can be selected, one for each 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.

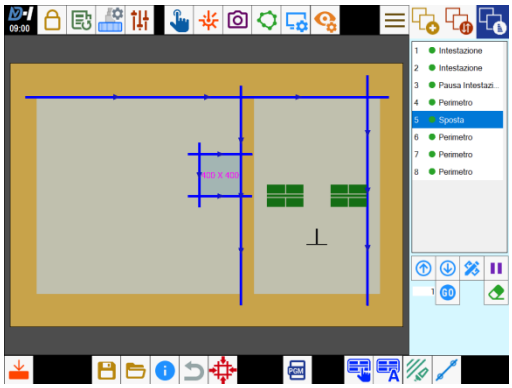


Upon confirmation, the slab is trimmed. It is possible to insert pieces and magnetise them to the trimming cuts as shown in the figure below.



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.

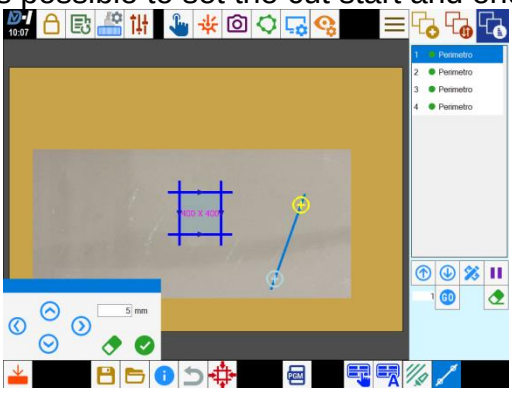




5.1.10.7 SINGLE CUT FROM PARAMETRIX

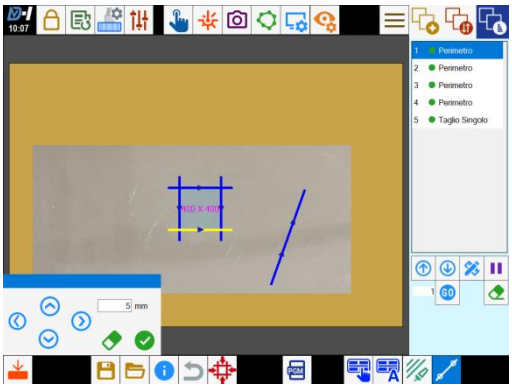
The single cut is a function allowing cuts to be inserted in the list of cuts window.

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.

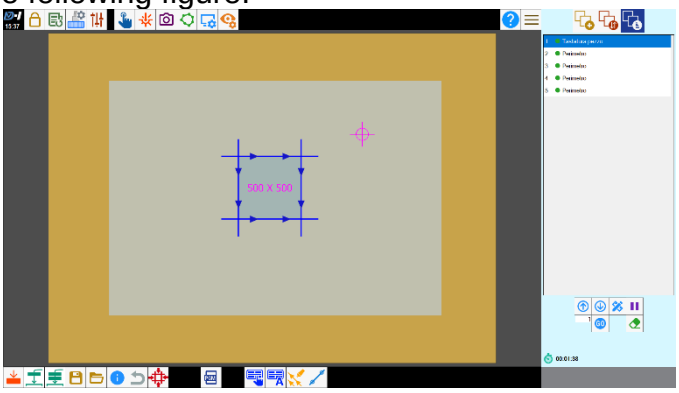
When positioning is complete, the preview can be confirmed using the green button or rejected using the red button. If the preview is confirmed, it becomes a cut for all intents and purposes. Once the preview has either been confirmed or rejected, a new cut can be inserted.



To complete the insertions, press the button 

5.1.10.8 CONTROL OF TOUCH PROBE

On the cuts page you can enter the operation of probing the slab. 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.

5.1.10.9 CUT SPEED UP/SLOW DOWN

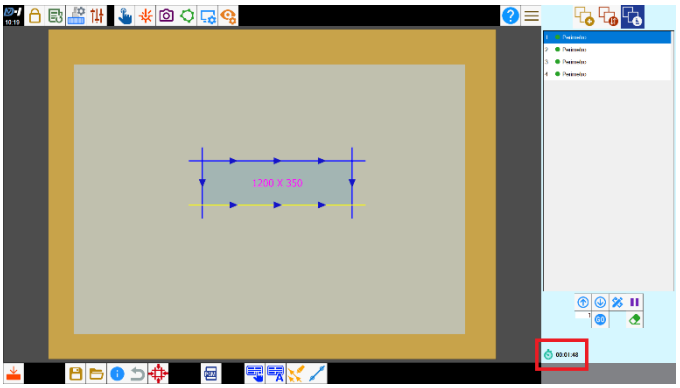
In user parameters there are two parameters for the management of accelerated cuts:

- 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.

5.1.10.10 WORK TIME SIMULATION

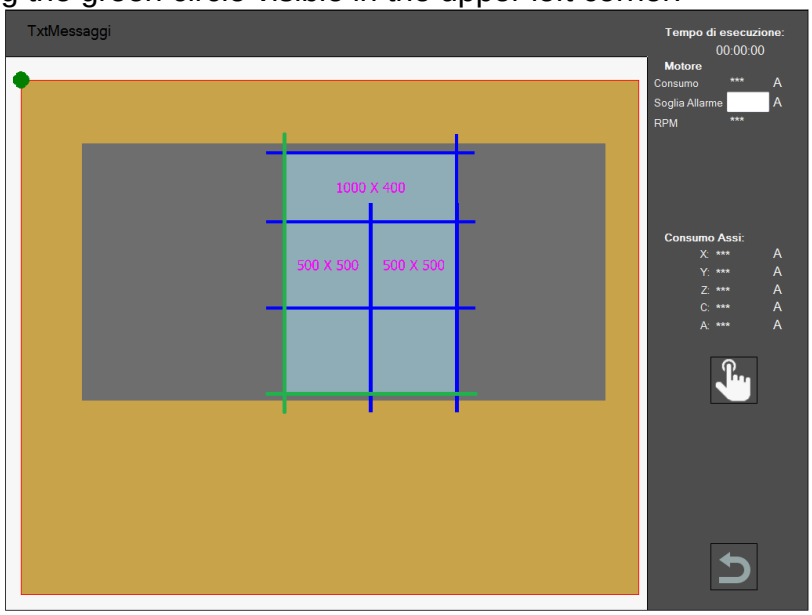
On the cuts page, the estimated cutting time is shown below:



5.1.11 ISO PAGE

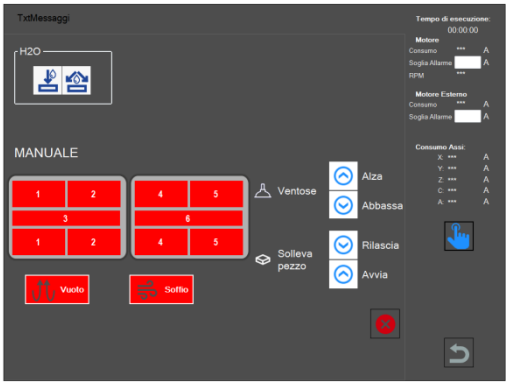
After finishing programming of the machining cuts, press the button 

which will transform the drawing and its characteristics into machine language. The program loads a screen to monitor the condition of the cuts. It is possible to see the movement of the machine following the green circle visible in the upper left corner.



During the processing it is possible to hide the page with the button  , to allow the operator to start programming the next processing.

Selecting  the program moves to the following page:



Here it is possible to turn the waters on or off (the internal and external one) or to perform operations on the suction cups (where they are present).

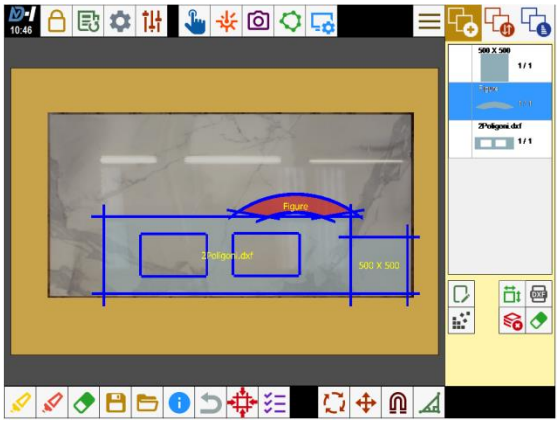
Selecting the  or  button again returns to the previous screen.

5.1.12 OPTIONALS AND OTHER FUNCTIONS

Some optionals and functions are described in this chapter

5.1.12.1 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.

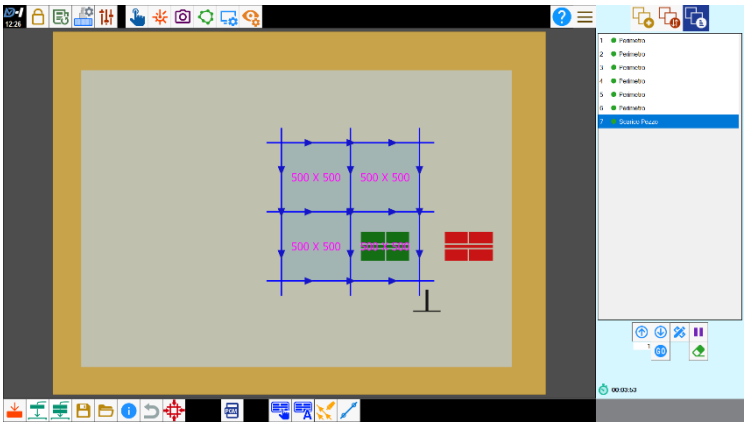


The Nesting function arranges the pieces following various insertion logics. For this reason, each time the  button is pressed, the program changes the arrangement of the pieces, up to a maximum of 4 solutions (upon the fifth attempt, the first solution is reproposed). The program arranges the pieces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

5.1.12.2 PIECE UNLOAD FUNCTION

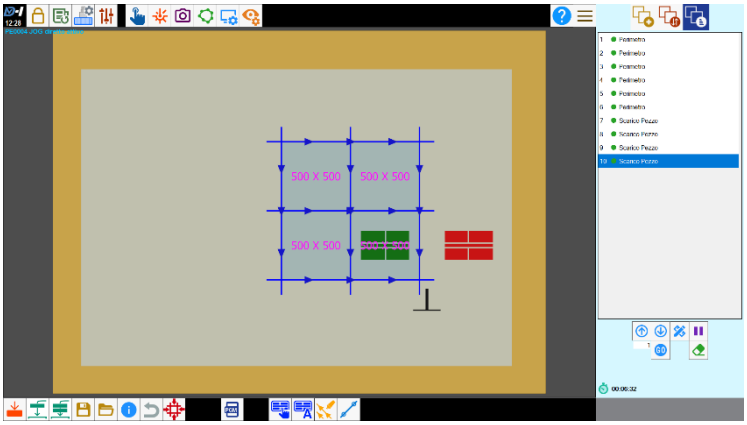
With the *Piece Unload* function it is possible to program the movement of a piece into a predefined position of the machine.
The program calculates the shift of the centre of the piece to a machine coordinate X, Y, and Z defined in the parameters.
To set the unloading of the pieces it is necessary to be inside the cutting management and to follow this procedure:

Activate the *Piece Unload* function using the button  and click on the piece to be discharged. In the image below it is possible to see the application of the functionality to the piece in the bottom right with the relative addition of a cut in the list on the right.



5.1.12.2.1 "UNLOAD EVERYTHING" MODE

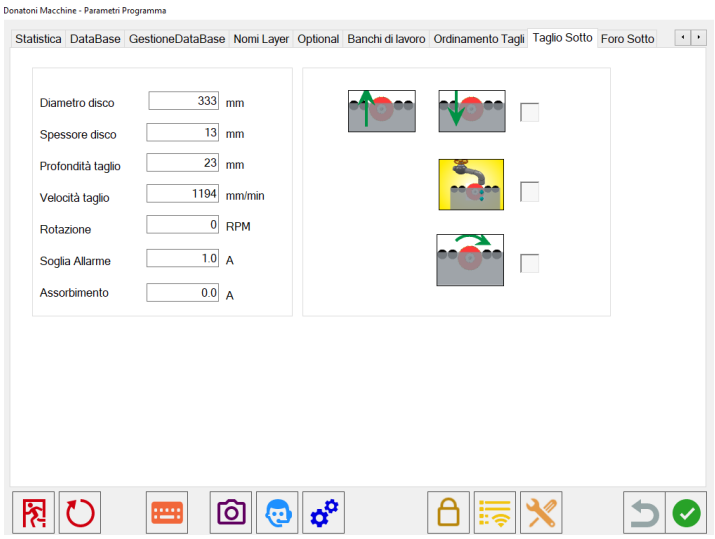
The "Unload everything" mode can be activated pressing the  button. In this case an Unload Piece cut will be added for each piece in the work area.



If you want to unload the piece as soon as possible, i.e. when all its cuts have been completed, you have to go to user parameters under "Unload everything mode" and select the "Optimized" flag

5.1.12.3 CUT BELOW

If the optional function *Cut Below* is present, press  on the initial Parametrix page and go to the “Cut Below” tab, which is the following:



Disc Up/Disc Down

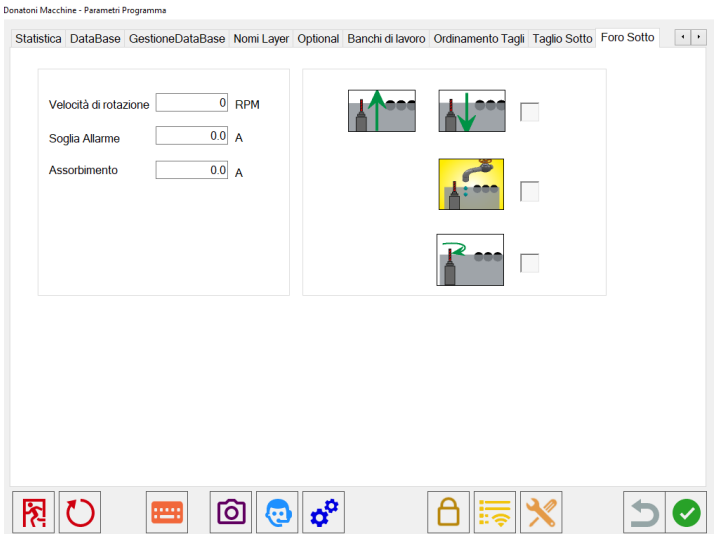
Water On/Off

Disc On/Off

There are certain parameters and buttons to control the outputs; various LEDs are linked to the buttons, indicating the status of the output.

5.1.12.4 HOLE BELOW

If the optional function *Hole Below* is present, press  on the initial Parametrix page and go to the “Hole Below” tab, which is the following:



Spindle Up/Spindle Down

Water On/Off

Spindle On/Off

There are certain parameters and buttons to control the outputs; various LEDs are linked to the buttons, indicating the status of the output.

5.1.12.5 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.

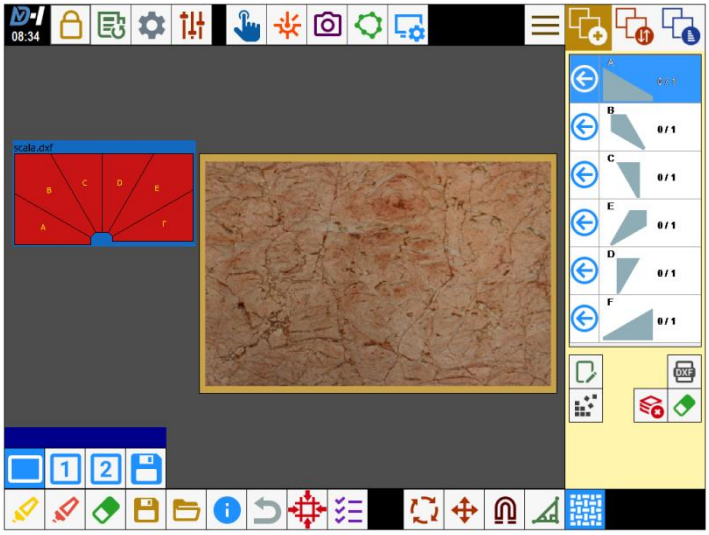
Proceed as follows:

- 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 functionality only works for parts imported from DXF files.

Press  to activate the *Open Book* function the following screen will appear:



To the left of the workbench there is a preview of the open mark. In this case, all the pieces 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 with the size of the pieces and appear with a red background to indicate that they have not yet been added to the workbench or that they have not yet been processed.

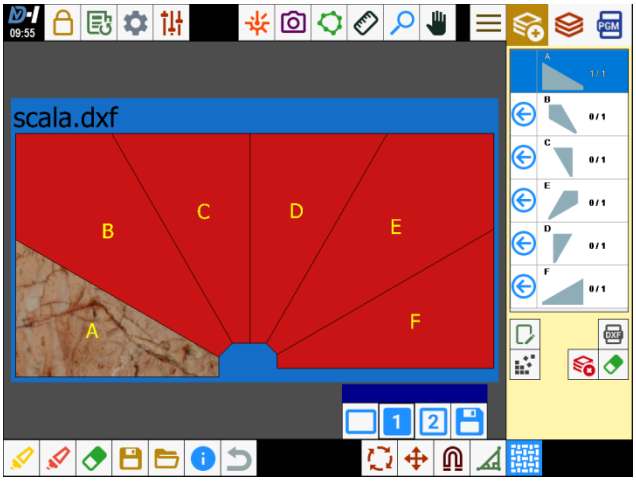
From the Preview it is possible:

- 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:

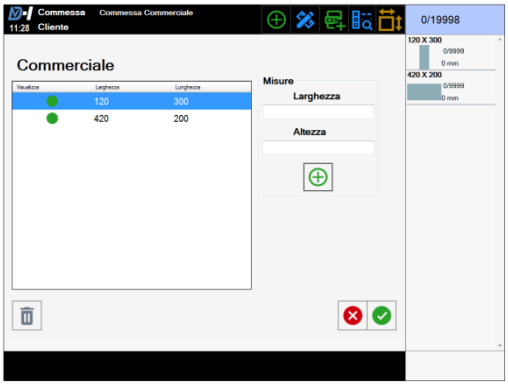
Side view: Default view as soon as the "Open Mark" function is activated. It provides an immediate view of the background that the piece will have by moving it onto the workbench.	
Preview Display: View focused on the preview. In this way it will be possible to position the piece in the desired position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

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



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

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

5.1.12.6.1 ENABLING/DISABLING A COMMERCIAL JOB ORDER IN CERTAIN PROCESSES

It is possible that during certain tasks the pieces of the "Job Order" do not need to be considered while in others they do; to avoid this problem, the button  is present on the main page. When the button is green it means that the job order can be used during processing; if it is red (), the commercial pieces cannot be used until it is re-enabled.

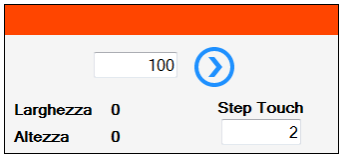
5.1.12.6.2 MODIFY SIZE OF PIECES FROM WORK AREA

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. At the end of modification of the piece, a new piece with the dimensions of the piece just modified will appear in the right hand list
3. If a new process is carried out  , these pieces are deleted
4. Collision checking with other pieces is present
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

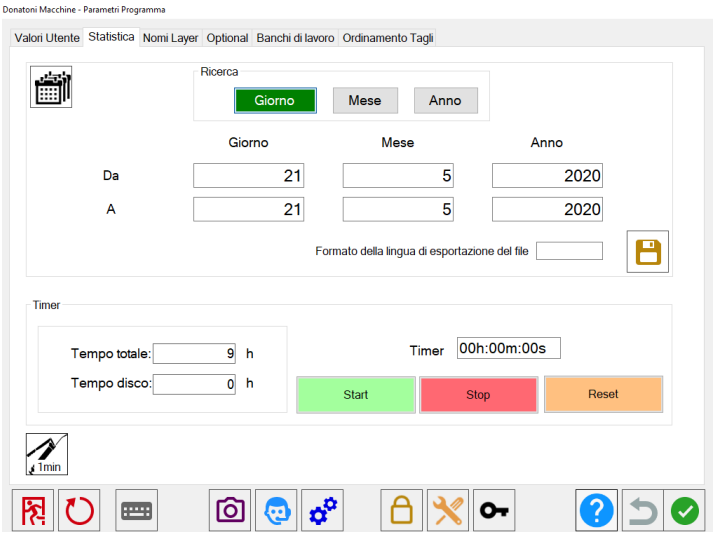
   b. Press the button  to apply the piece change

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).

5.1.13 STATISTICS

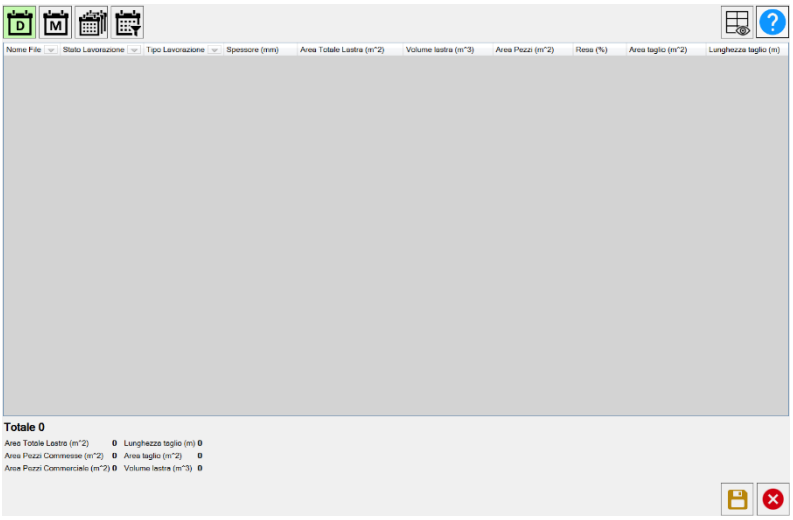
Regarding statistics, there is the possibility of having a temporal statistic rather than for the individual job order. To obtain then, press  from the initial page of Parametrix and go to the "Statistics" tab, shown below



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.
The statistics also include Single Cut, Emptying, Levelling, ISO processes.

If you want to view the statistics on screen, press the  button and the following window will open up:



The window shows the statistics table. 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

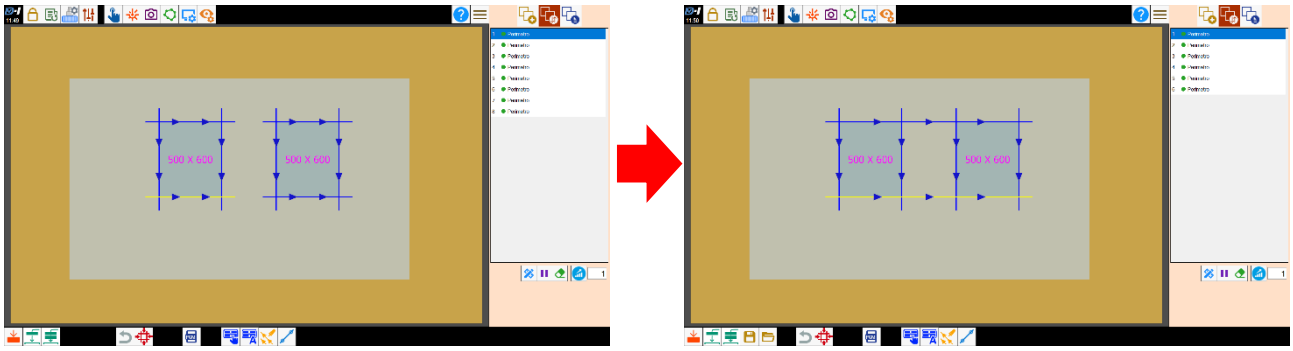
5.1.14 SX CONTROL AND OPTIMISATION

Below is a description of the different ways of optimizing cuts.
This function is useful to put the cuts together and make them with a smaller step with the SX3 machine.

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

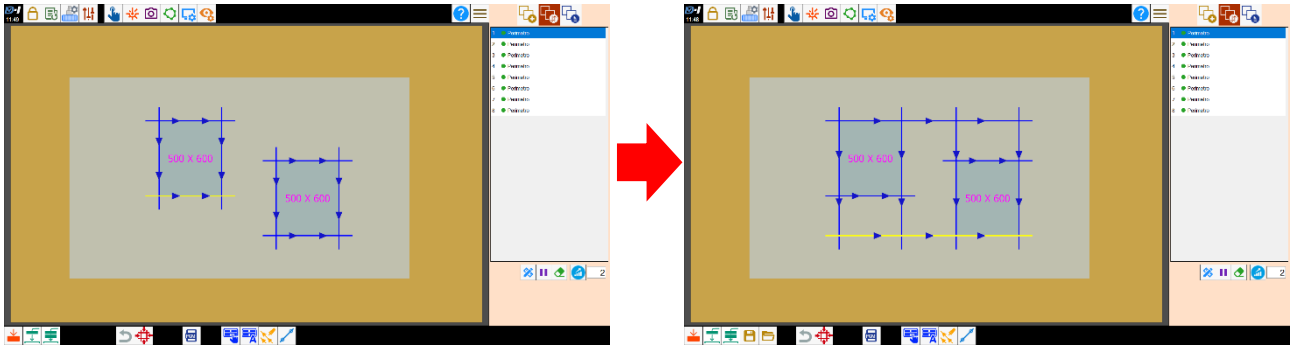
5.1.14.1 MODE 1

Two different cuts on the same straight line become one cut



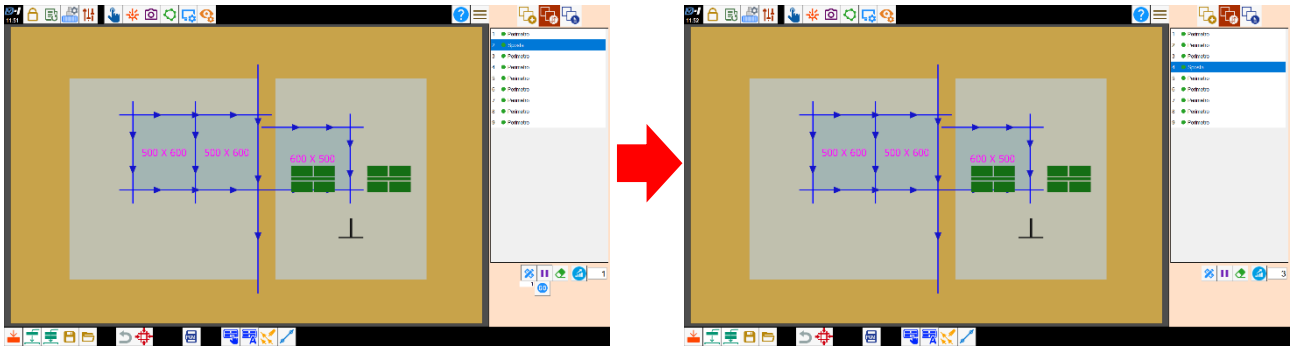
5.1.14.2 MODE 2

It tries to make the whole cut the same length (maximum).



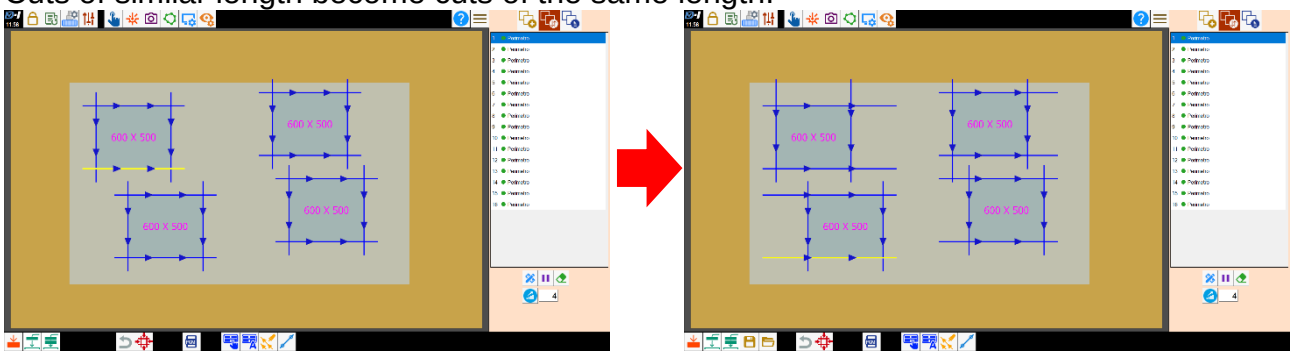
5.1.14.3 MODE 3

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



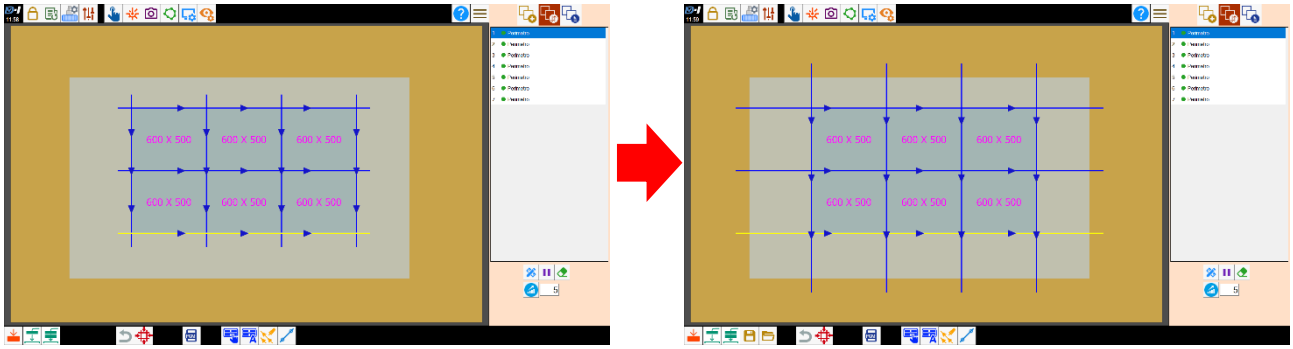
5.1.14.4 MODE 4

Cuts of similar length become cuts of the same length.



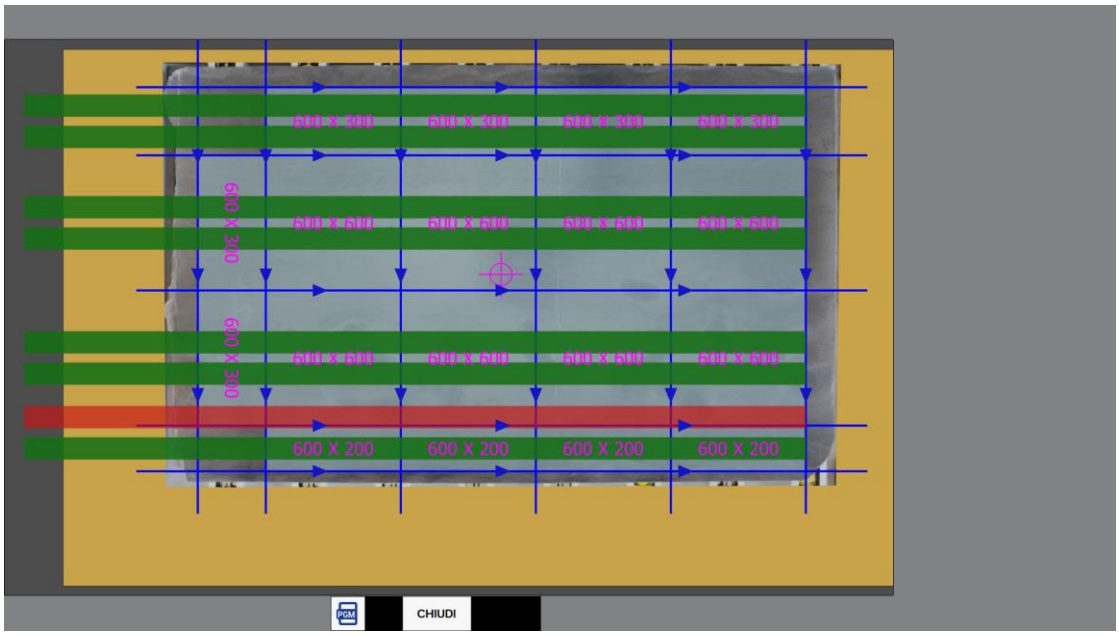
5.1.14.5 MODE 5

It extends all cuts beyond the edge of the slab



5.1.15 UNLOADING DEVICE

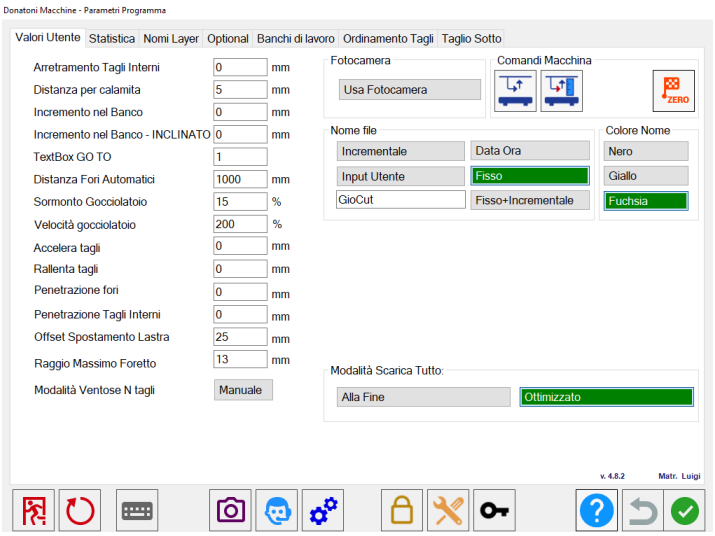
The unloading device is managed by the software by creating an unloading program to position the rows of pieces on the appropriate belts.



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.

5.1.16 USER VALUES

You can access the user values page by pressing the button .



Below is a description of the individual fields:

Internal cut withdrawal	Minimum approach distance between the disc 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 you leave the page

5.2 MULTIPLE CUTS

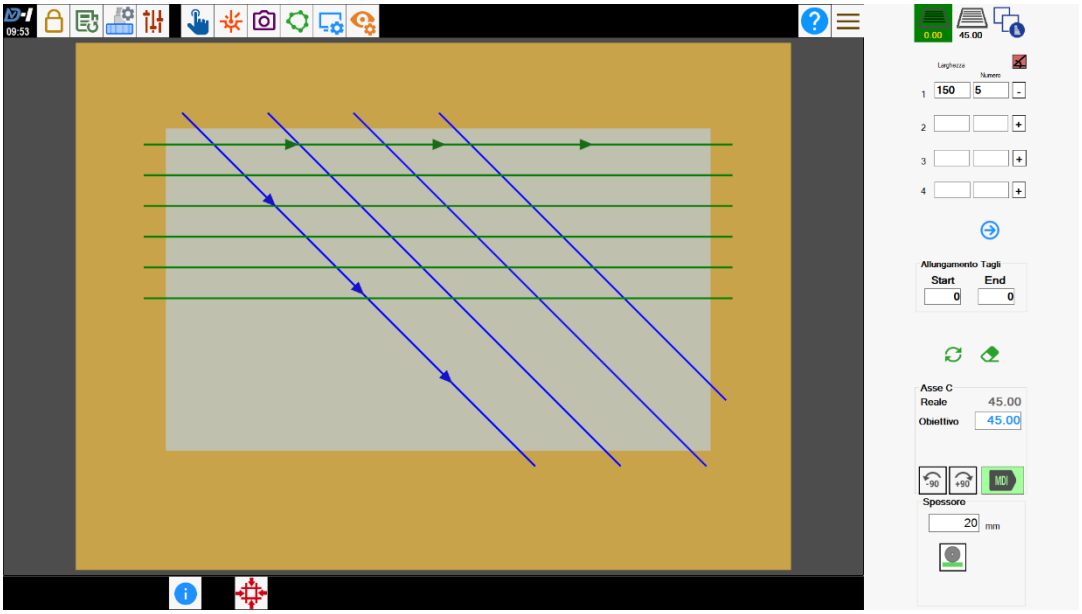
In Parametrix it is possible to add multiple cuts from the "Cuts" section in the main list. This particular type of cuts allows you to create a series of cuts parallel to each other in the same group, with the possibility to have two groups each with its settable direction.

5.2.1 GENERAL EXPLANATIONS

Once "Cuts" has been selected from the general list, if the perimeter of the slab has not yet been acquired, the program will bring the user directly into the section for its creation. Refer to section: "*Slab Acquisition*", present in this manual.

Note: *The creation of several perimeters in the case of multiple cut always involves use of the first to perform cuts while the others are ignored.*

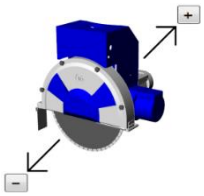
Once in the presence of the perimeter of the slab, it will be possible to proceed to the cuts creation section where a situation similar to the image below will be presented.



5.2.2 SIDE BAR

As can be seen from the image, there are three buttons at the top. These three buttons indicate:

1.  Creation of the first group of cuts: Parallel cuts are created with respect to the current position of the machine. The distance between one cut and another is taken from the "Width" text box and the quantity of parallel cuts to be created will be taken from the value found in the "Number" box (See image above). If to the side of these two values there is the button  then the parallel cuts will be created from the current position of the disc towards the "motor of the machine". If instead the button has this form  the cuts will always start from the current position of the disc but will be created in front of the disc.



Summary view of the +/- buttons

2.  Creation of second group of cuts: the second group of cuts is created in a similar way to the first, with the difference that the direction may be different. To decide the direction, the C axis must be positioned in the desired direction; the cuts created will have the set direction. The direction in degrees for the specific group of cuts is displayed below the button:

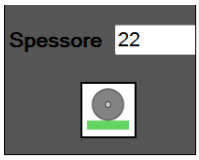
If the two groups of cuts have orthogonal directions to each other, the icon  otherwise this icon will be displayed 

3.  Management of Cuts: In this mode it is possible to make certain changes to the cuts before they are made. It is possible to: modify, disable, move, delete and add pauses before making the cut. Refer to section "*Parametrix –Management of Cuts and Displacements"*.

5.2.3 MATERIAL ACQUISITION

To calculate the thickness of the material two modes can be used:
1. Classic mode, that is using a meter or other measuring devices
2. Bringing the machine to the desired height for cutting

The following panel is used to set this information:



To modify the material thickness manually (first method), press on the text box and type in the new material thickness using the keyboard. To instead acquire the thickness using the machine, bring the height of the machine to the desired value and select the button . The distance between the bench and the disc will be considered as material thickness.

5.2.4 HEAD ROTATION CONFIRMATION

During the creation of Multiple cuts, the head is turned each time the selection between cuts of the first group and those of the second group is changed and each time the C-axis is changed from to one of the two groups, the C axis does not coincide with the rotation that the machine must have for the execution of the cuts.

5.2.5 DELETION OF CUTS

You can delete the cuts created to restart with a new layout by pressing the button . Pressing this button not only cancels cuts, but also aligns the direction with the current C axis direction.

5.2.6 UPDATE OF CUTS

Multiple Cuts are created based on the current position of the machine. It may be necessary for various reasons to move the machine to a different location on the bench.
In such a situation the cuts and the machine are no longer attuned to each other. It is therefore necessary to press the button which moves the cuts into the correct position with respect to the machine.
If the update button is not pressed, the machine performs the cuts related to the old position of the machine.

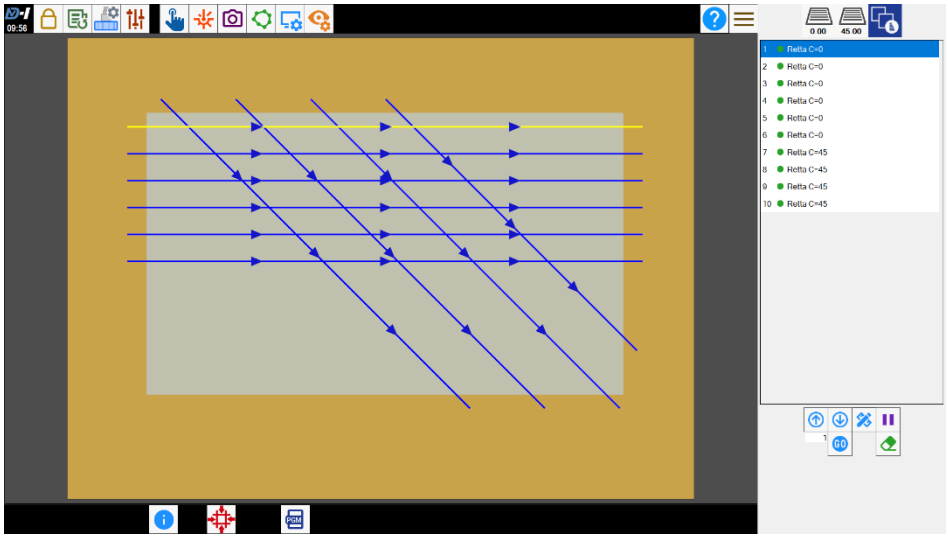
5.2.7 +/- 90

The buttons and are used to rotate the head by +/- 90 degrees with respect to the position they are in. These buttons do not show on the screen the message prompting confirmation of rotation, as is the case for the buttons of the first group and the second group.

5.2.8 EXECUTION OF CUTS

When pressing button, the file that will be sent to the machine is created and, if the procedure is successful, a message will be displayed, otherwise the machine cannot

perform the processing and shows a screen message indicating which cut created problems and pointing the user to the cut management page, where it is possible to resolve the problem that has occurred.  .



If the machine goes out of its stroke to make a cut, the cut cannot be made.
Faced with this problem it is possible to disable/remove the cut or select another perimeter (returning to one of the other two buttons:  of the first group and the second group).

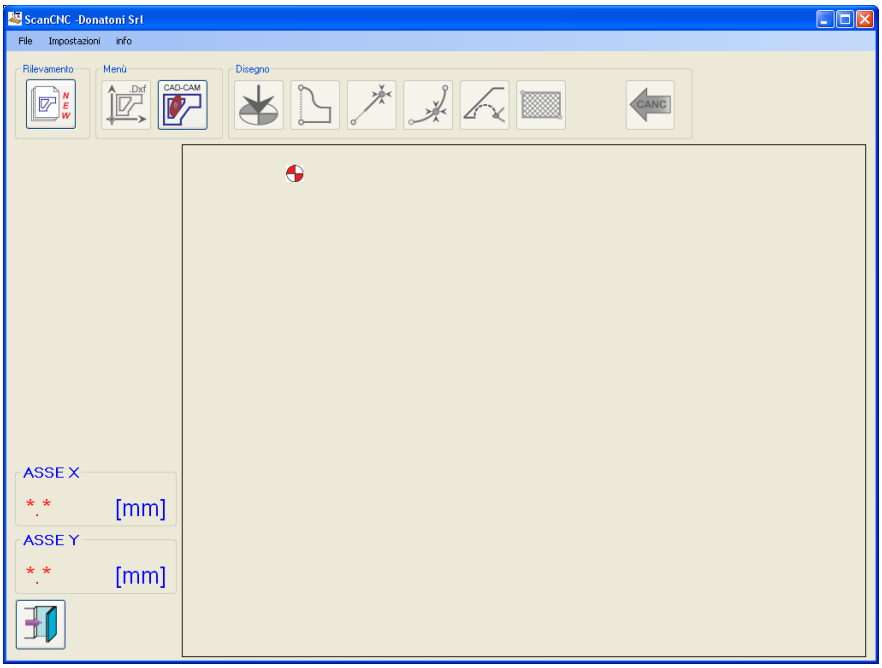
5.3 SCANCNC

ScanCNC is a software that allows manual shape perimeter detection. A classic example of using software detection functions might be the angular steps of a staircase obtained from a cardboard template. It is important to highlight that the program is connected to the numerical control of the machine that allows the creation of detections with pinpoint accuracy.

The logical operation of the program is as follows:

- Identification of the reference points along the perimeter of the template
- Template detection
- Creation of the DXF
- Use of the DXF created to associate the work tool
- Cutting of the template

During detection it is possible to see the result on the screen. This allows visual inspection of the operations performed. In addition, the software is accompanied by a screen to ensure ease of use.



5.3.1 DRAWING

When you press the *start scan* button the program begins to communicate with the machine's numerical control, thereby making it possible to scan the outline.

Detection takes place via movement of the machine over several points of the template. It is in fact necessary to insert into the program the data of the figure segments. This operation enables precision detection of the templates.

There are several commands that allow the insertion of predestined points on the perimeter of the figure:

Source: The design asks for an origin from which to start the scan. This will be defined as the origin of the piece until it is closed.
In order to set it, move the machine so that the laser pointer provided for ScanCNC is at the starting point of the outline then press the source button shown in the Figure.

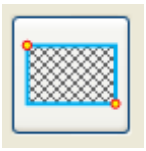
Line: To scan a line, the laser pointer must be moved to its final point and the line command must be given. The initial point will always be the last point inserted in the scan, which may be the origin or another scanned element (line or arc)

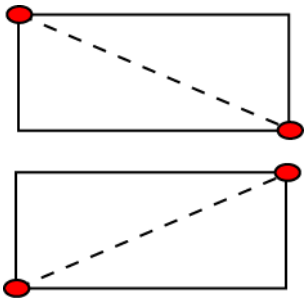
Arc: The arc command requires 2 acquisition points. Therefore, move the ScanCNC laser pointer to a central point of the arc to be scanned and press on the "Arc" button (first figure below). The software buttons change enabling insertion of the last point of the arc or cancelling of the operation. To determine the last point, move the pointer to the end point of the arc and give the command "Last arc point". The latter button is divided into 2 commands: clockwise arc or anti-clockwise arc. User choice must be based on the direction of the arc that is being scanned. Often many templates have no definite radii. It is therefore advisable to split the curve into several smaller arches. Doing so allows maximum precision on the curve.

Jump to the next point: This button makes it possible to generate a machining skip from one point in the scan to another. It may be useful when detecting several figures in the

same project. This command requires the source point of the next figure. Therefore, after giving the command, perform the source operations for the next piece.

Rectangle: This button makes it possible to create a rectangle by simply importing the position of the 2 diagonally opposite corners and is very useful in acquiring slabs or regular figures.



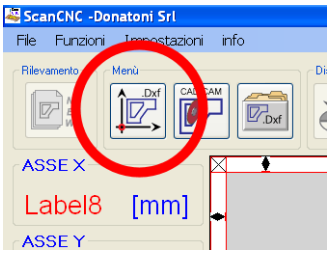


To use this function, press the "Rectangle" button after which move with machine to the first point in a corner of the rectangle and press the button "Acquire first point". Move the machine to the corner that is diagonal to the corner already acquired and press the command "Acquire last point". The program automatically calculates the length and position of all sides of the rectangle.

Close figure: this button makes it possible to close the figure being scanned with precision. It traces a line that starts from the last position entered in the program and arrives at the exact origin point of the piece. It is not necessary to take the machine to the origin point as this function does not take the position of the machine in to consideration.

5.3.2 DXF FILE CREATION

The ScanCnc program produces a file with a DXF extension, thus making it compatible with many CAD. To create the file, having acquired the desired figures, press the button "Generate DXF"



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.