IsoNs – Next Step
Use Of Interface

www.promax.it

The contained information in this handbook are only informative and they can being change without warning and they must not being understandings with some engagement from Promax srl. Promax srl does not assume responsibility or obligates for errors or inaccuracies that can be found in this handbook. Except how much granted from the license, no part of this publication can be reproduced, saved in a recording system or transmitted in whatever form or with any means, electronic, mechanical or recording system or otherwise without Promax srl authorization.
Any reference to names of society or products have only demonstrative scope and it does not allude to some real organization.

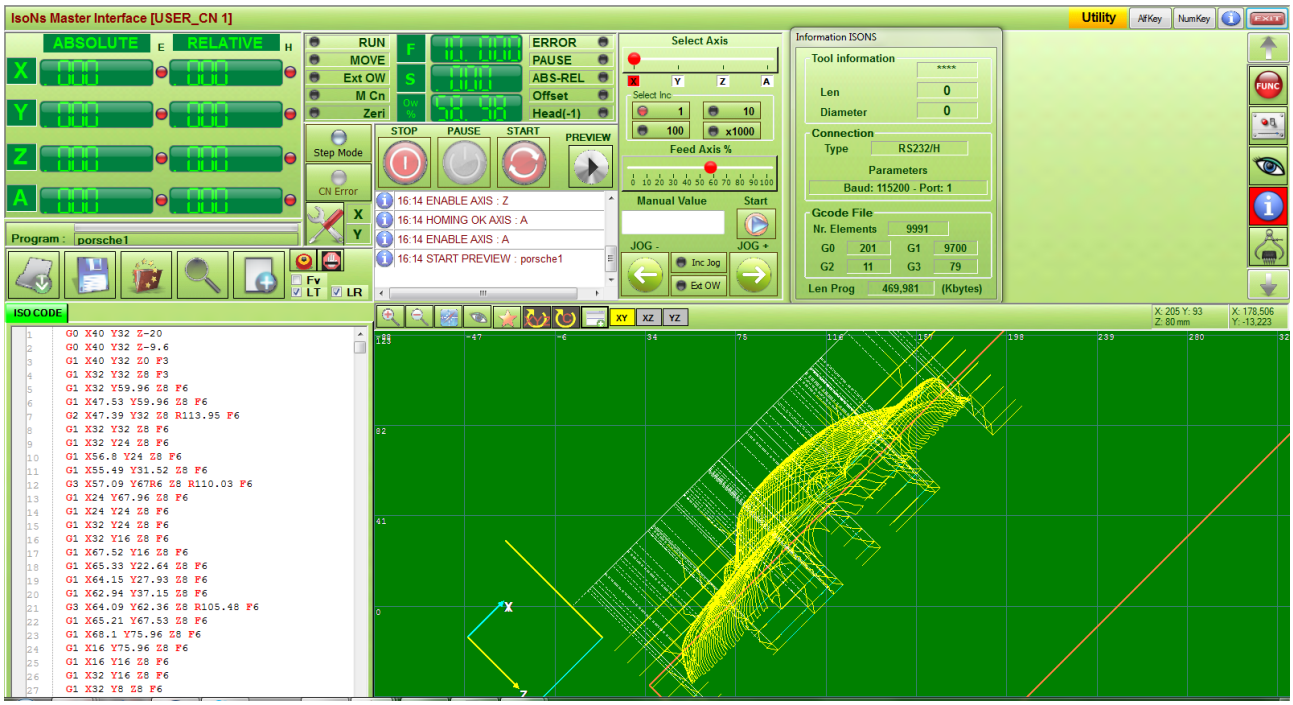
Rev. 3.0.0

1 PREFACE

This manual explains the use of operator interface IsoNs proposed by Promax Ltd.
The interface can be customized using development environments such as Visual Studio and compatible so its appearance may vary. Through the interface, you can use all the functions of IsoNs available for the control of the machine. The interface standard provides for the control of machines IsoNs general and therefore may lack some specific features.

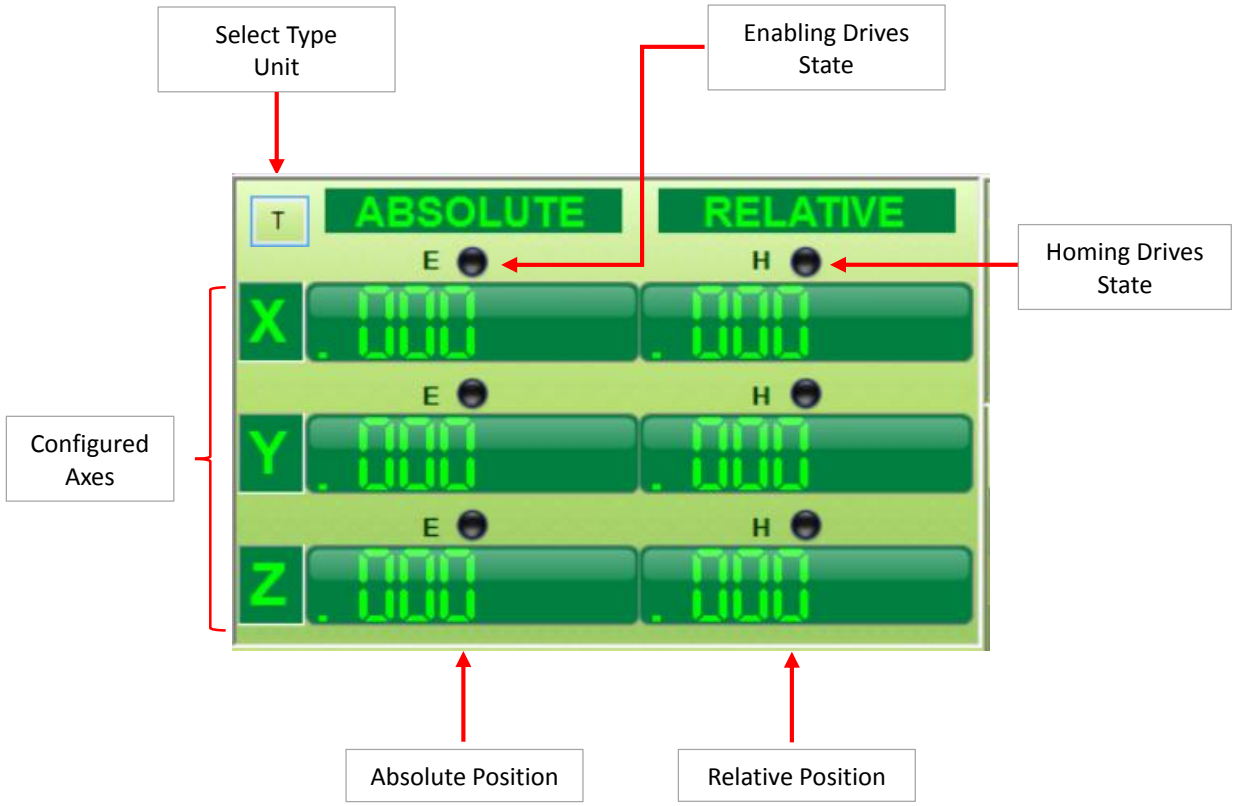
2 General

The interface looks IsoNs collected in a single form and thus facilitates the use
Depending on the resolution of the monitor can display more or less information at once.
The interface is divided into well-defined fields that we analyze later.



2.1 Axis Units

This is all information about the current share of the axes and the status of these. The format and the representation of values is defined in the configuration file IsoNs.Cfg



Axis Configured
Indicates the name of the axes that have been configured

Enable Drives State
The LED ON indicates that the device driver is enabled

Homing Axis State
The LED ON indicates that its has made the Homing

Absolute Position
Absolute position from **Machine Origin**
For disable or enable the Displayed click on LABEL **ABSOLUTE**

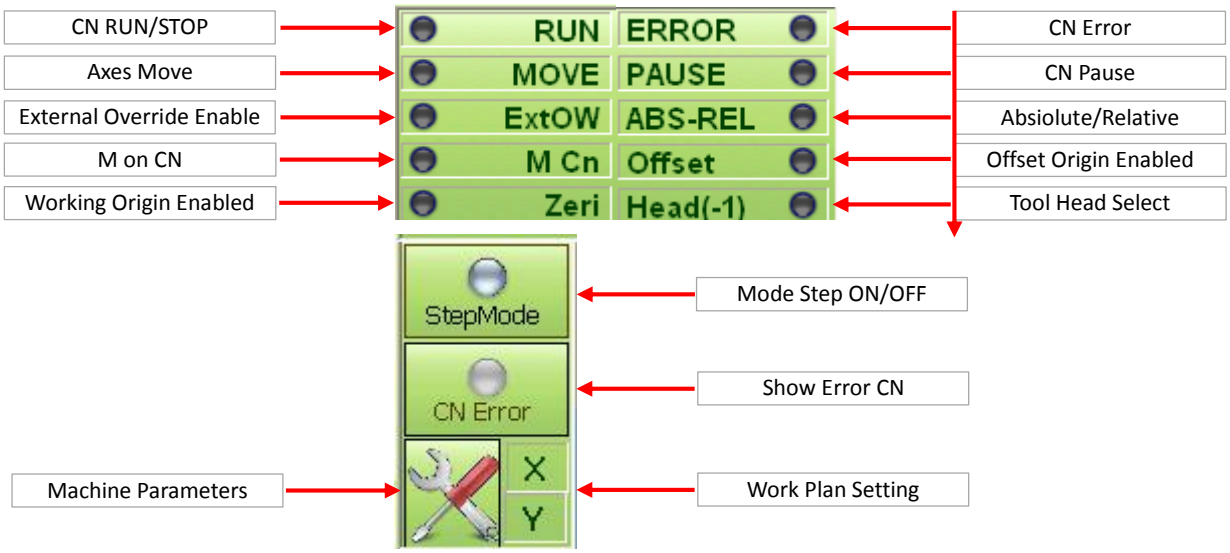
Relative Position
Relative position from **Working Origin**
For disable or enable the Displayed click on LABEL **RELATIVE**

Select Type Units
This button is enabled by configuration and allows you to view the following parameters:
Demand Position
Real Position
Follwing Error

Depending on the number of axes configured to dispose the components internal field can also change in size. The unit of measurement and the display format of the shares is determined by the configuration of axes IsoNs.

2.2 CN STATUS

In this field displays information about the state of NC. The information is immediately visible from the state of its LED. Some buttons access to special functions.



Enable/Disable mode STEP execution program

Pressing this button enables or disables the execution of the Gcode to STEP.
When the Gcode is being run by that each LOCK STEP is paused. To execute the next block, press the START BUTTON.

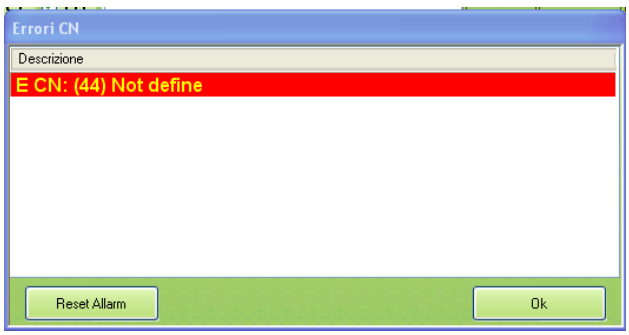
Machine parameters

Browser machine parameters (see chapter **MANAGEMENT MACHINE PARAMETERS**)

Displayed CN errors

This button is enabled when the CN have errors.
An error on the NC blocks the current Gcode in RUN.

Cn errors window

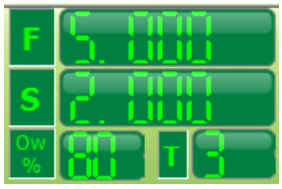


The window displays all the errors of the NC.
Reset Alarm button can sometimes restore errors.

WARNING
Not all alarms can be restored.
Must still verify the cause of the ALARM.

2.3 AXES FEED

Displayed the Feed Axis information , Spindle and ovveride



F Actual feed
S Spindle
% Percentage actual speed (select by virtual potentiometer or external analog input)
T Tools select

The reference values of the parameters F, S and% are set in the configuration IsoNs.cfg

2.4 BUTTONS CONTROL

Use the following buttons for complete Gcode control

STOP
Block current implementation of the Gcode in progress, or the boards by manual handling.
If the set **M STOP**, before the termination of the program is turned on M which generally serves to manage the users of the machine.
The STOP button can also be remote from the outside physical button.

WARNING
STOP button does not make a emergency stop. Therefore it is necessary that the machine is prepared by all safety regulations.

PAUSE
Send paused at the current implementation of the Gcode.
When activated the **M PAUSE**, this is done. During the pause is still possible the manual handling or jog the axes running scripts ISO (machine functions).
From the state of PAUSE STOP interrupting you can exit via the execution of the current Gcode, or START recovering from the exact point of interruption the Gcode. In this last case if you have configured the **M STOP** from shooting, this is done.
The PAUSE button can also be remote from the outside physical button.

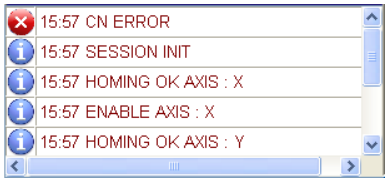
START
Execution Gcode loaded into the ISO WINDOW.
The execution can be interrupted by STOP or PAUSE

PREVIEW
Preview Gcode loaded into the ISO WINDOW.
(See chapter **3D PREVIEW)**

2.5 INFORMATION WINDOW

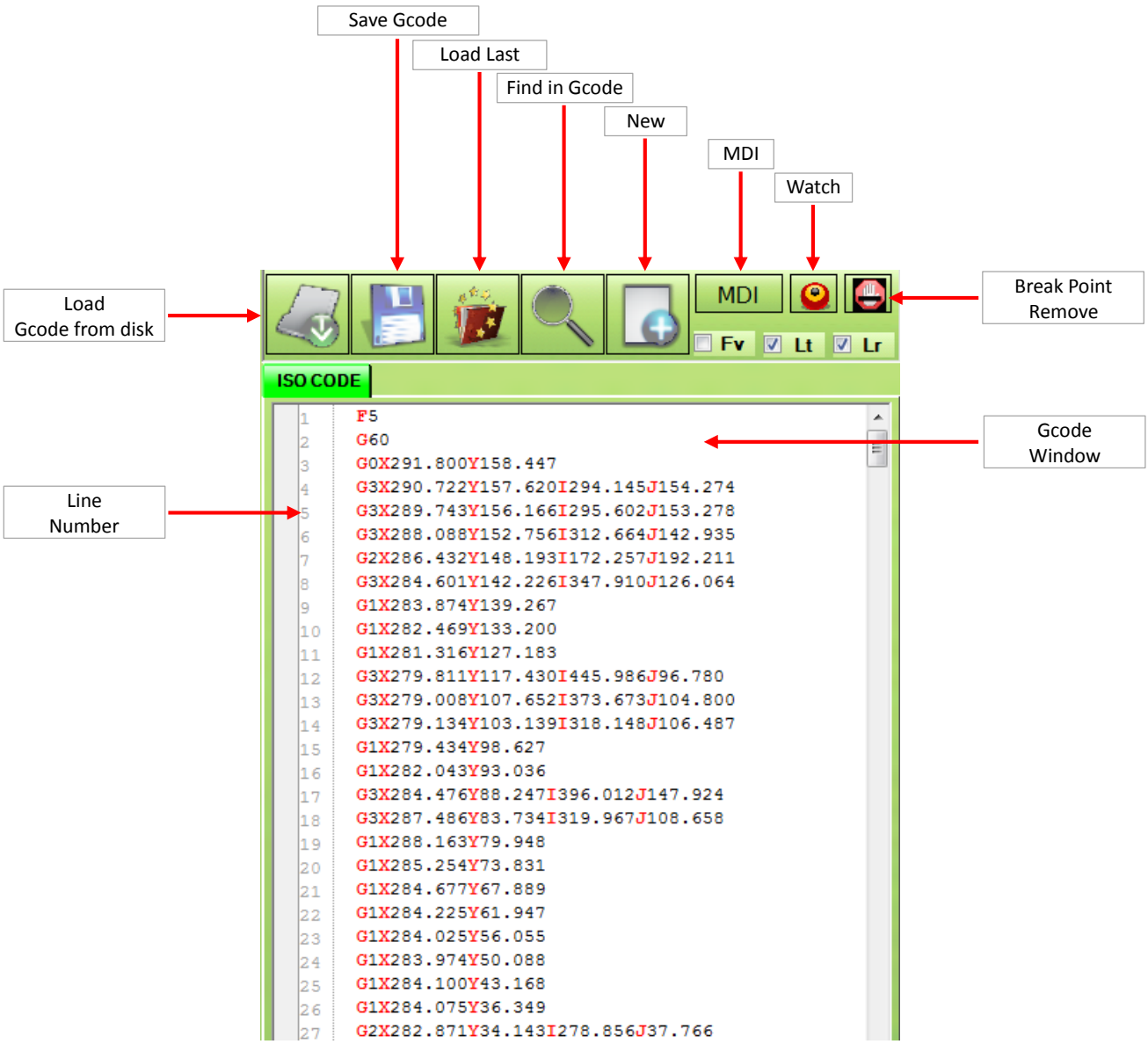
Displays information on all major transactions.

This window contains the latest information 50. It will still create a log (ISONS.LOG) file that records permanently all operations.



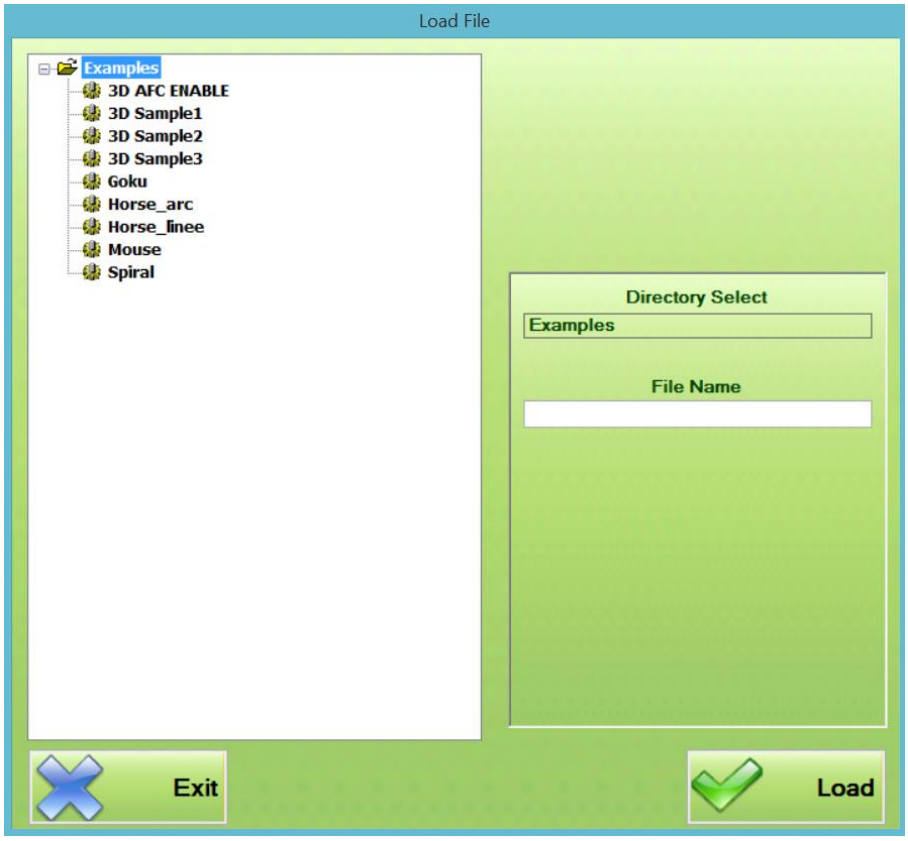
2.6 GCODE ISO

Handles editor, saving, loading programs from disk ISO.



2.6.1 LOAD GCODE FROM DISK

Allows you to load a program from disk.
All NC programs are collected in specific folders. Just loaded, the program is automatically populated with immediate Ceck OF ERRORS (see ERRORS ON GCODE).
The browser load management program is presented as follows:

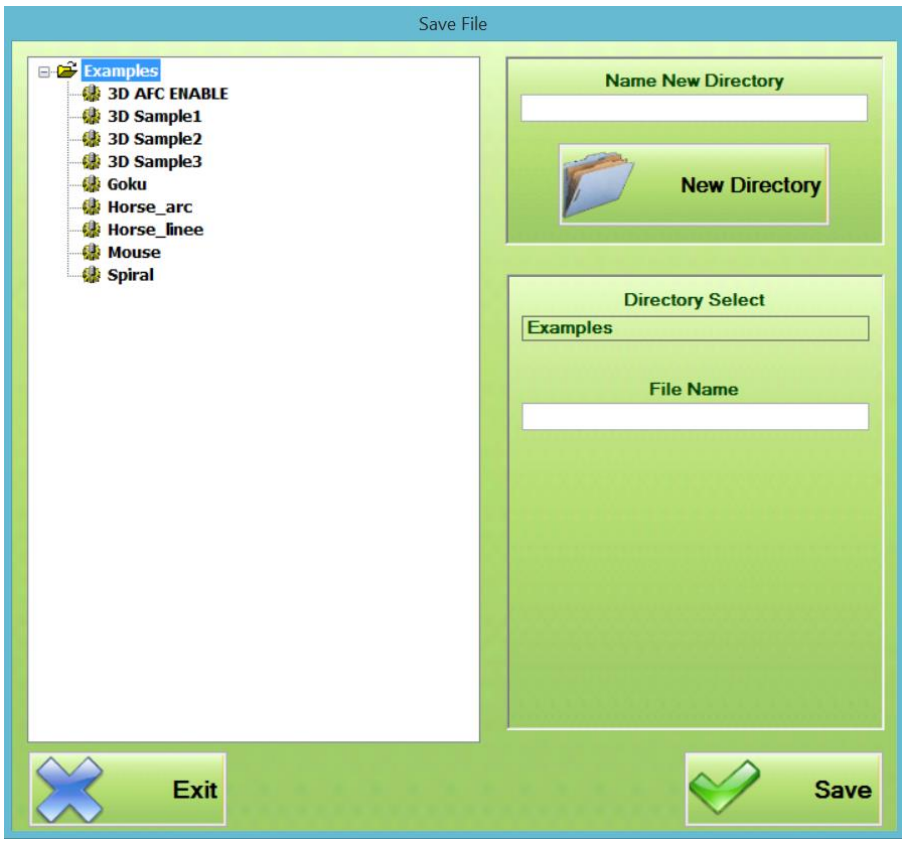


To select the file to load, follow these steps:
Make double click on the desired file, or select the desired file with a single click and press the Load button.
If you need to write the file name in the File Name field.
In this final way you can make reeds deleting a file, instead of the LOAD button, press the Delete key on the keyboard.
The programs can be collected in order to promote research folders.

WARNING
When the program has been erased, it is permanently lost.
It will still be asked to confirm the actual cancellation.

2.6.2 SAVE GCODE TO DISK

Allows saving the ISO Gcode content in the window. Following the press of the button displays the Browser rescue.



Before you save the file you need to choosing the save folder, clicking the mouse on the + symbol or the name of the folders in the folder.
you can save the file instead of an other simply by double clicking on the name of that file and confirm saving, Writing a file name in the FILE NAME box to save, and then pressing the SAVE button, you can save the file with a new name (the allowed characters are filtered from the same box).

you can still create a new folder by entering the name in the new name FOLDER and then pushing the New Folder button. In the file tree show the name of the new folder ready to save the file.

WARNING
Overwriting a file, is permanently deletes the old file with the consequent loss of all its data.

2.6.3 MODE DISPLAY GCODE

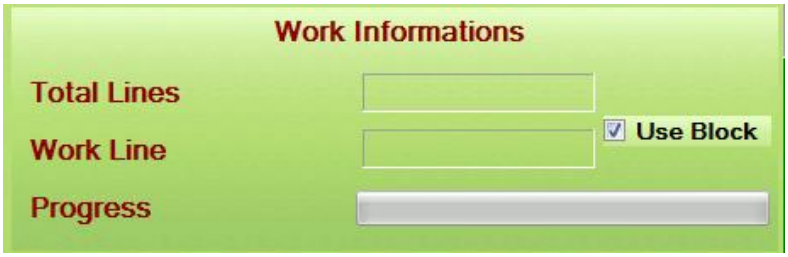
In IsoNs you can configure the display of the status of the Gcode: This is done with the following options:

Fv ***Fast Visua Speed.***

Enabling this option is used to display speed of the Gcode. This in fact is not loaded in the editor window, but only displays the progress of running. This option is useful when you work Gcode consisting of several blocks (Gcode> 10 Mb). In this situation, take up resources on your PC slowing down the loading of the Gcode.

Use Block

Enabling this option makes further processing of the Gcode block.
This type of machining parts program is recommended for more than 60 Mb.
This option uses the minimum PC resources



With the option enabled Fv previous window appears which indicates the total number of phone lines, running and current status of progress.

LT ***Normal Line Working***

This option allows you to view or not the line worked on (with yellow background) The line can be made with respect to the real line.

LR ***Real Line***

This option allows you to view the line or not processed by the CNC Real (blue outline),
This line can be located back from the normal line of a number of buffer blocks dependent on the set LOOK HEAD CNC

2.6.4 BROWSER CONFIGURATION MANAGEMENT PROGRAMS

The browser for saving and loading program can be configured via the file in the folder where you installed **BrowserNs.cfg** IsoNs

in this file are present the following parameters:

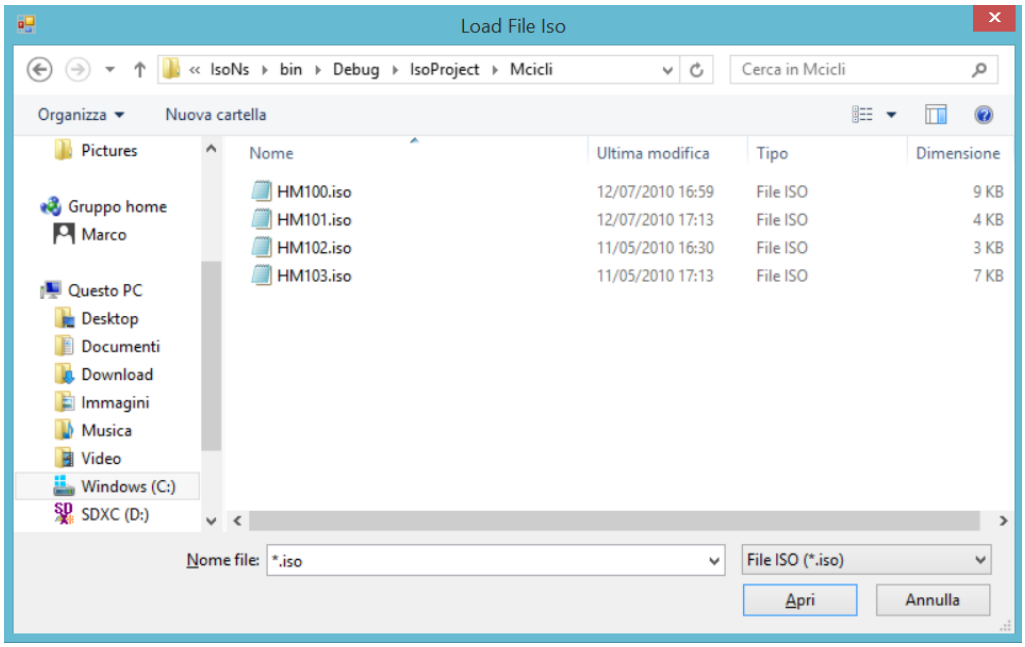
BROWSER=
FILEEXT=

Default Value:
BROWSER=0
FILEEXT=*.iso

BROWSER	Select the browser interface	
Value	**0**	Select ISONS browser
Value	**1**	Select standard Windows® Browser

FILEEXT Gcode extension file
Value Insert the extension es:
***.iso** (default)
***.cn**
ecc.

STANDARD WINDOWS® BROWSER

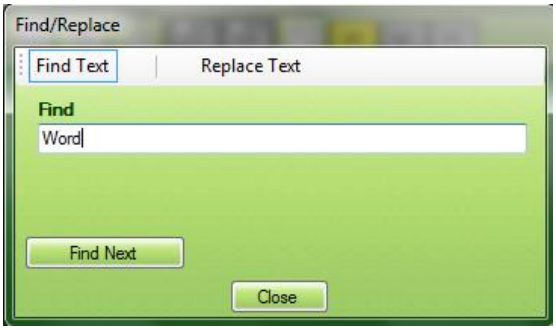


2.6.5 FIND IN FILE

Allows you to search the loaded program, parts of the text.
You can also make the substitution of words.

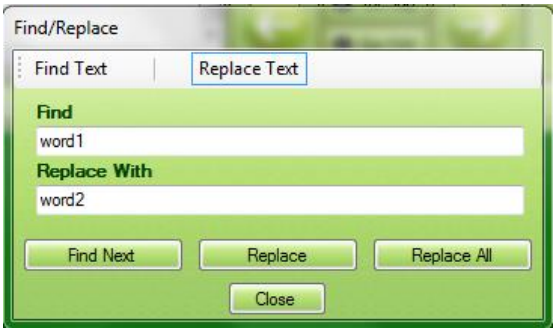
Searching text

1) Make sure the button is highlighted text search, or click on this.
2) Enter the text in the Find box to search
3) Press the Find Next button. The found text is highlighted in the ISO FINEST
4) To continue the search of the same text again, press FIND NEXT



2.6.6 Replace Text

1) Make sure that the part REPLAC TEXT button, otherwise click on this.
2) Enter the text in the Find box to replace
3) Enter in the box replaced with the new text
4) Press FIND NEXT and then replaced
5) REPLACE ALL carries a full replacement of the entire program



2.6.7 LOAD LAST

Allow the load last file used

2.6.8 NEW

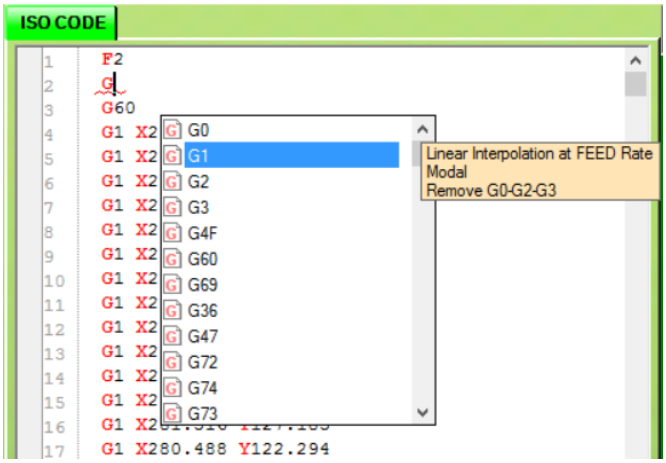
New file in windows ISO

WARNING
Function clears the window NEW iso file in use, so if this did not save all the data will be lost

2.6.9 ISO WINDOW

ISO is still in the window can write a program from scratch.
IsoNs provides an EDITOR to advanced features of modern technologies of word processing programs. RealTime write errors are highlighted in order to make an immediate correction. An on-line Help is always suggested helping the Operator in the choice of the function to use.

The input error is immediately displayed with a red underline.
The on-line help function provides on the basis of initial released.
To enter the function from the Help is just double click on the desired function in the window list instructions.
A click with the right button will make the window reappear HELP



Error window Gcode
In addition to the underscore of RED error, displays a more detailed description IsoNs in a special window that appears when there are one or more errors in the Gcode

Line	Description
7	AXIS NOT CONFIGURED...
7	ILLEGAL INSTRUCTION.... 358.995
8	AXIS NOT CONFIGURED...

Double-clicking on the row field of error, the cursor automatically IsoNs line of the Gcode that generated the error.

IsoNs using an editor that highlights keywords in different colors in order to make more understandable the program.
The loop LOOP and IF END_LOOP END_IF can be reduced so as not to occupy the field of view.

```
3   G1X0Y0
4   IF  $VAR1=10
5       $_PARM_1=0
6   ELSE
7       $_PARM_1=1
8   END_IF
9   G0
```

To minimize, simply click on the symbol **-**

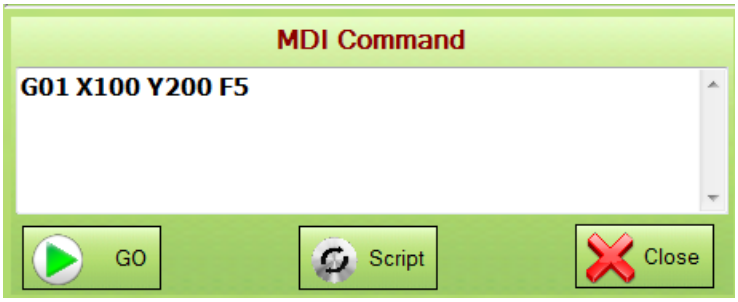
Click on the **+** symbol to expand

```
3   G1X0Y0
4   IF  $VAR1=10...
9   G0
```

IsoNs also use the comments for the program and are placed as follows:
// THIS IS A COMMENT
The comment will not be considered at the end of the execution of the program

2.6.10 MDI WINDOW

This allows you to enter direct commands and send them running ISO. The Gcode loaded into the ISO WINDOW , is not altered.



Can be inserted more lines of commands, and these can be performed in 2 ways:

GO

Performs the commands lines in RUN MODE NORMAL, but every M predefined (MSTART,MEND, MGOBLOCK, etc.) are disable. The commands can be contain all ISO G CODE instructions.
This mode can be used only when the CNC is in STOP.

Script

Performs the commands lines in SCRIPT MODE. The commands must contain only a limited ISO G CODE:
G0,G1, F, S and all M which not used ISO CODE different from these.
This mode can be used even when the CNC is in PAUSE.

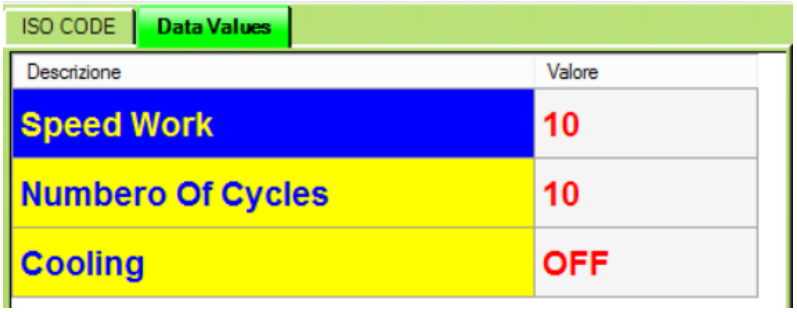
2.6.11 DATA INPUT MASK

IsoNs to facilitate the data entry Gcode, may be associated with each file type ISO, one or more DATA INPUT MASKS.

This allows the data entry program in conversational mode, facilitating parametric programming file.
The data input are presented in an understandable form to user-led and with minimum and maximum values already controlled by the mask to minimize data entry errors.

The data is passed to the Gcode before running this.

To associate a MASK DATA INPUT to an ISO file to see the manual "MANAGEMENT MASKS".



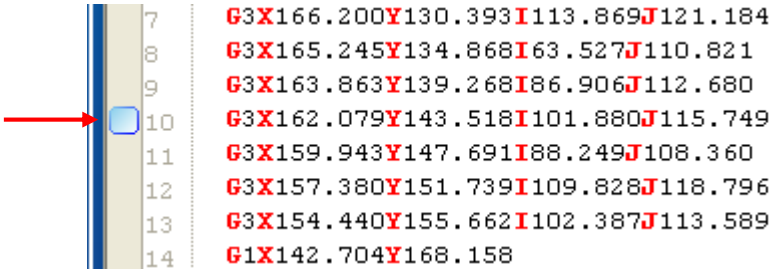
3 ADVANCE GCODE FUNCTIONS

The features described below, can be disabled. These are decided by the configuration of IsoNs.

3.1 Break Points insert

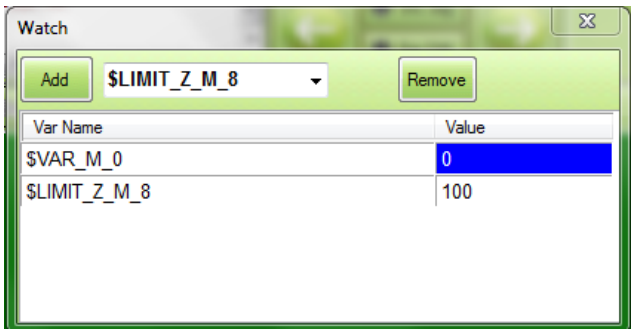
The breakpoints used to interrupt the Gcode in a particular spot.
This allows for debugging to make checks on the performance of the Gcode. Once intercepted, the breakpoint, the Gcode is in a state of PAUSE. From this you can start with the START or STOP to stop permanently. When intercepted the breakpoint, it is also possible to switch mode STEP execution of Gcodes.
To insert a breakpoint, simply click your mouse on the gray bar of the window to the ISO reference line number where you want the program to be interrupted.

Click the mouse to insert the breakpoint. 'S happened insertion is identified by the active marker. The removal of the breakpoint is achieved by clicking on the marker again.



3.2 VARIABLES WATCH

This feature can be activated by the button WATCH need to monitor the status of variables IsoNs. It is possible to read and write these in order to help the iso operator in debug programs.



From the **ComboBox** to choose the variables to monitor and press the ADD button to add them to the list.
During the execution of the Gcode and can see the value assumed by the monitored variables. Double-clicking on the field value of the variable, you can force a value to leisure.
The REMOVE button removes the variable from the list selected by the blue line.

For variable insert to absolute address:

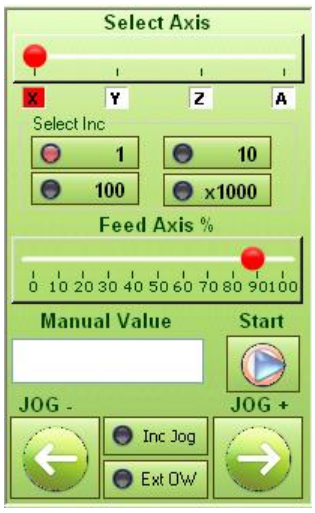
#numberVar ex: #100

3.3 BREAK POINT REMOVE

Remove all break Points insert

4 MANUAL AXIS JOG

Allows the manual handling of axes and speed management override.



4.1 Select Axis

Select axis for manual JOG
Move the cursor on desired axis

4.2 Select Inc

Select the amount of increase to be made by the buttons + and JOG JOG function is activated when-INCREMENTAL JOG (Jog button Inc active)

4.3 Feed Axis %

If disabled the override potentiometer (Ow Ext button disabled) adjusts the speed rate of movement in both axes is MANUAL at run Gcode automatically.
The manual speed of each axis is defined in machine parameters IsoNs.

4.4 Manual Value

Allows you to move an axis at a precise value.
In the box enter the value of the share where to take the Axis selected by the selector and then press the Select Start Control Panel Manual

If it is activated INCREMENTAL JOG (Jog button Inc is active), the axis moves in a relative way, or absolutely.
The FEED can be adjusted by OVERRIDE (external or internal).
STOP BUTTON CONTROL ends of the any motion in progress

4.5 JOG-/JOG+

Holding down this button, the axis moves set to jog in both directions.
Releasing the button stops the movement in progress.
If the feature is activated INCREMENTAL JOG (Jog button Inc is active), the axis with each press of the button moves + JOG-/JOG the amount indicated by the selectors Select Inc (x1, x10, x100, x1000) corresponding to 1 millisecond, 1 cent, 1 tenth, 1 mm.

4.6 Inc Jog

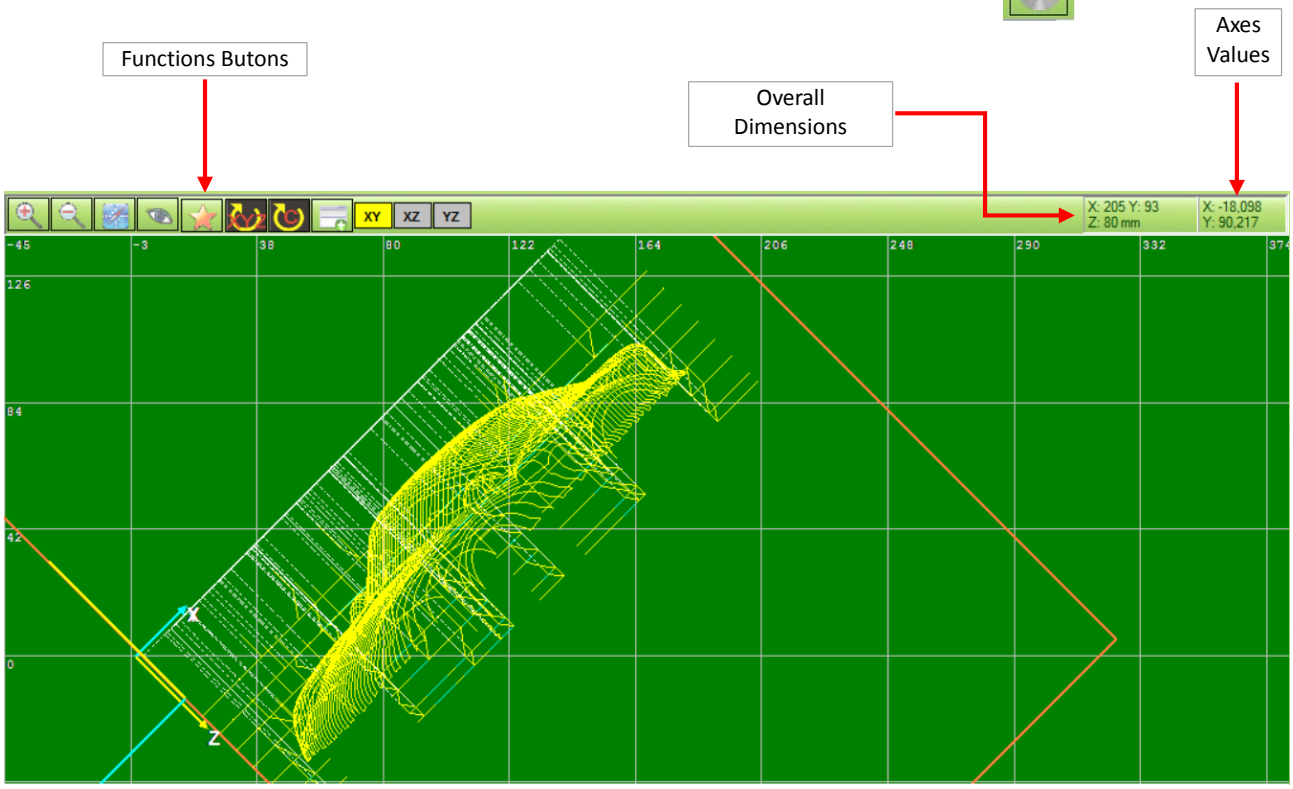
Eable or Disable INCREMETAL JOG

4.7 Ext OW

Enable or Disable External override (analog input)

5 3D PREVIEW

It allows previewing the tool path and the trace REALTIME during processing.
This function is very useful because it can detect errors before programming the tool path. Through the simulation is also possible to search Gcode graphic block. The simulation start from the Preview Button.



5.1 Overall dimensions piece

View the measures X, Y, Z dimensions of the piece. The measures are calculated with the movement G1, G2, G3. Any movement in G0 are not considered in order to piece if these dimensions are not followed by other movements in G1, G2, G3.
If the attack on the piece is made in G1, this is considered in the bulk of the piece.

5.2 Axis Value

Moving the mouse pointer in the area of simulation, you see the dimension X, Y

5.3 Functions Buttons

Enable the 3D function

5.3.1 ZOOM IN/ZOOM OUT

Allows you to enlarge or reduce the visual area of the piece.
Press the desired button to activate (or deactivate) the function (it turns red).
Then click with the mouse at the point you want to zoom in or out. The function remains active until you press the button again.

5.3.2 ZOOM WITH MOUSE - PINCH TO ZOOM

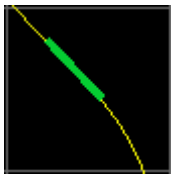
If you use a mouse with center wheel, the zoom can be done through this.
Aim the pointer on the affected area and rotate the mouse wheel to enlarge or reduce the portion of the screen.
For touch screen System ***MULTITOUCH*** can be use the **"Pinch to Zoom".**

5.3.3 PAN AREA

Allows you to move the visual area of the workpiece in all directions.
Press the button to enable (or disable) function. Then click somewhere on the altar of the piece and drag the mouse to the desired direction of movement. When you release the viewing area will be moved in the direction of the amount of dragging the mouse (diagonal movements are also considered). The function remains active until you press the button again.

5.3.4 SEARCH THE LINE GCODE NUMBER

Allows you to search the block number of a graphic ISO or vice versa.
Press the button to enable (or disable) function. Then move the mouse in the vicinity of the right to the desired item is highlighted with a different color, the window will display the ISO generated the line element.



```
12  G3X157.380Y151.739I109.828J118.796
13  G3X154.440Y155.662I102.387J113.589
14  G1X142.704Y168.158
15  G1X130.943Y180.528
16  G2X128.003Y184.350I175.575J217.902
```

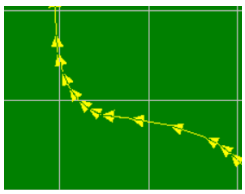
At this point you can move with the ↑ ↓ keys on the keyboard of the highlight bar YELLOW WINDOW ISO to get the reference graph of the highlighted block (you get the same effect as clicking the UP and DOWN line number with the mouse any).
Once you select a section are also highlighted the following information:

F: 7.992
X: 79.713
Y: 245.484
Z: 60
A: 34.596
VT: 0.353
LEN3D: 1.862
LEN2D: 1.756
SGLP: 3570
SGL3D_X: 54
SGL3D_Y: 6
SGL3D_Z: 0
SGL3D_A: 259
AFC_X: 437
AFC_Y: 48
AFC_Z: 0
AFC_A: 2071

F	→	Segment Feed
AccR	→	Centrifugal acceleration for G3 G3
X,Y..	→	Axis Value
VT	→	FEED tangential AXIS (if present)
LEN3D	→	Tridimensional length
LEN2D	→	Bdimensional length (only axis to Work Plan)
SGLP	→	Threshold 2D edge
SGL3D_	→	Threshold 3D Edge
AFC_	→	AFC Parameters (adaptive feed control)

5.3.5 TOOL DIRECTION

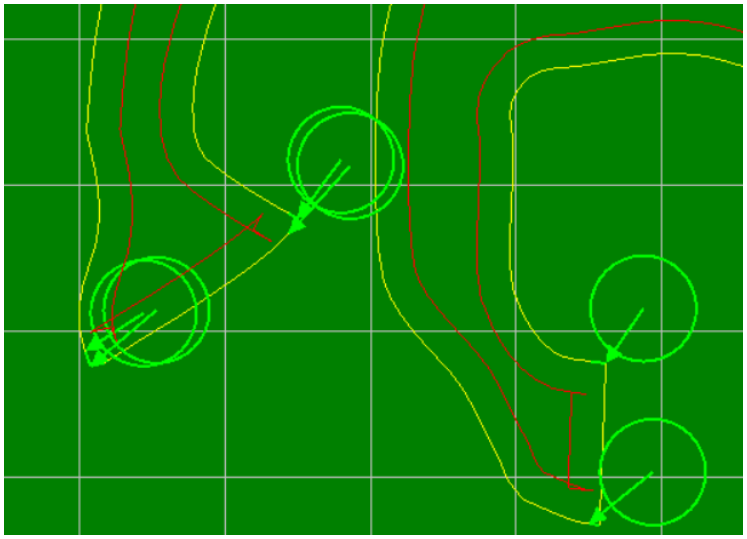
Allows to displayed the tool direction. The Arrows indicate the direction.



5.3.6 TOOL OFFSET

If is enabled the TOOL OFFSET (G41-G42), this is
Se abilitata la COMPENSAZIONE OFFSET UTENSILE (G41-G42) , This is displayed in the simulation with a trace of different color deviated from the profile set with the tool radius set by instruction **D** or **Tn**.
Using this button you can display or hide the track offset tool.
The simulation as well as view track offsets tool also allows you to highlight parts of the tool path that because of the tool diameter generate a HIGH PROFILE error on the journey.
This can alert the operator and then decide accordingly whether or not the processing to be performed.

In this example, the simulation showed the line that caused the error distance of the profile. it is evident that this error is due to the use of a tool with a diameter too large to make the kind of work.
To go back to the line number is sufficient to activate the ISO SEARCH THE NUMBER OF PART OF THE LINE PROGRAM and highlight the line that generated in GREEN's error.



5.3.7 SLIDING ON THE PATH OF SECTION

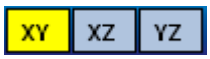
When activated NUMBER OF LINE SEARCH OF PART OF THE PROGRAM, you can cycle through the features by following these steps:
Click the mouse on the highlighted section.
Scroll down the path with the two buttons SLIDING TRACT.

5.3.8 CENTER PIECE ON SCREEN

This feature allows you to center the design of the piece using a visual area in ZOOM optimal for its representation.

5.3.9 CENTER DIMENSION

This feature allows you to center the entire WORK PLAN in visual area allowing you to view the layout of the floor inside the workpiece

5.3.10 VIEW Work Plan 

These buttons allow you to view only the graphics of the work selected

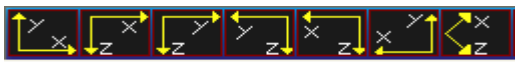
5.3.11 ROTATE X,Y,Z

By activating this function is possible using the mouse to rotate the view of the profile on the three axes X, Y, Z. Once activated, press the left mouse button and drag in the direction you want to show the right angle of the drawing.

5.3.12 ROTATE TO CENTER

By activating this function, you can rotate the view using the mouse on the center of the profile. Once activated, press the left mouse button and drag in the direction you want to show the right angle of the drawing.

5.3.13 EXTENDED MENU

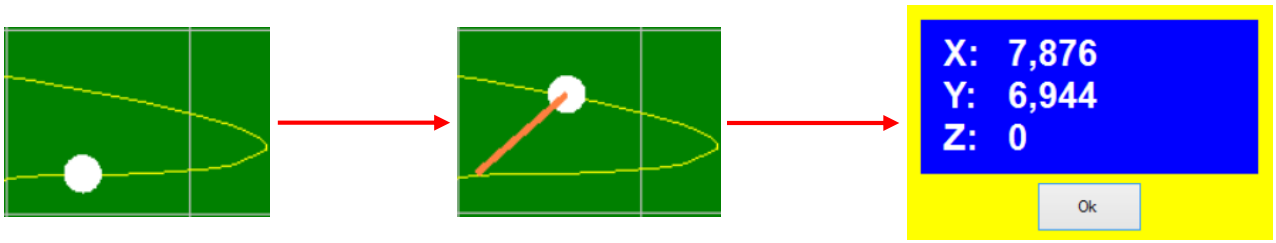


5.3.14 PRESET VIEWS

X,Y (Up) - X,Z (front) - Y,Z (left) - Z,Y (right) - X,Z (rear) - X,Y (down) - Isometric

5.3.15 MEASURES

It allows you to make measurements on the drawing. The measurements are performed on the three dimensions X, Y, Z always starting from the point of beginning of an entity G0 (G0 VIEW option only if activated)-G1-G2-G3. The start and end point of measurement is highlighted with a circle.
Click on the start point and then measuring the end point.



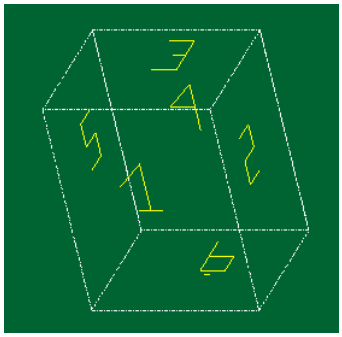
5.3.16 OPTION

This menu contains some of the simulation settings permanent. You can also configure the various colors of panel simulation.

5.3.16.1 Move Only Dimension

In cases in which the design is particularly complex, or the PC does not have sufficient data processing speed, the phase rotation and displacement can be difficult, because it is constantly redesigned the profile in use. By activating this option, during these phases is represented only the 'space of the profile with the 6 faces numbered waves speed up these procedures.

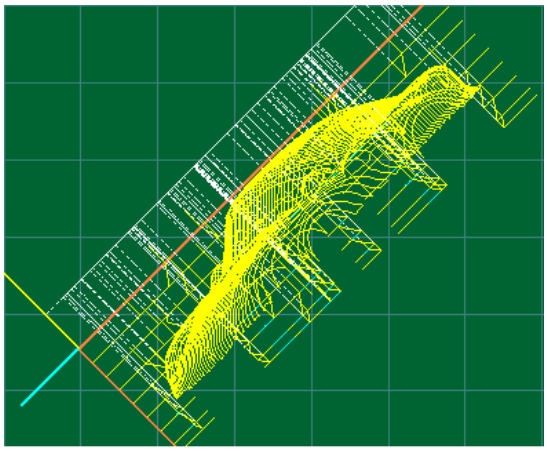
1 Frontal
2 right
3 Up
4 Rear
5 Left
6 Down



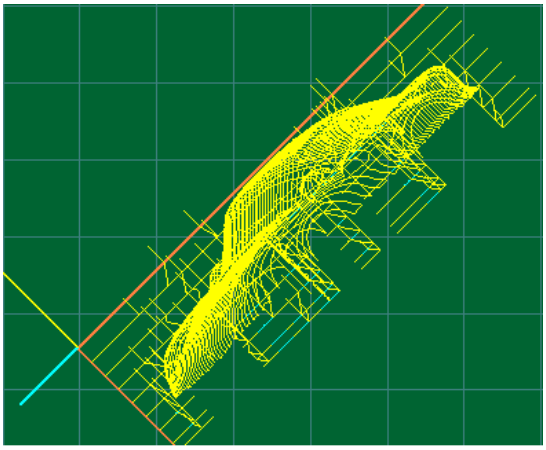
5.3.16.2 View G0

Enables or disables the display of fast movements in G0. This also affects the block or in the research measures, since if View G0 is not activated, you can not find the blocks G0 or take measurements from them. The displacements are represented in the G0 lines with hatch style.

Enable



Disable

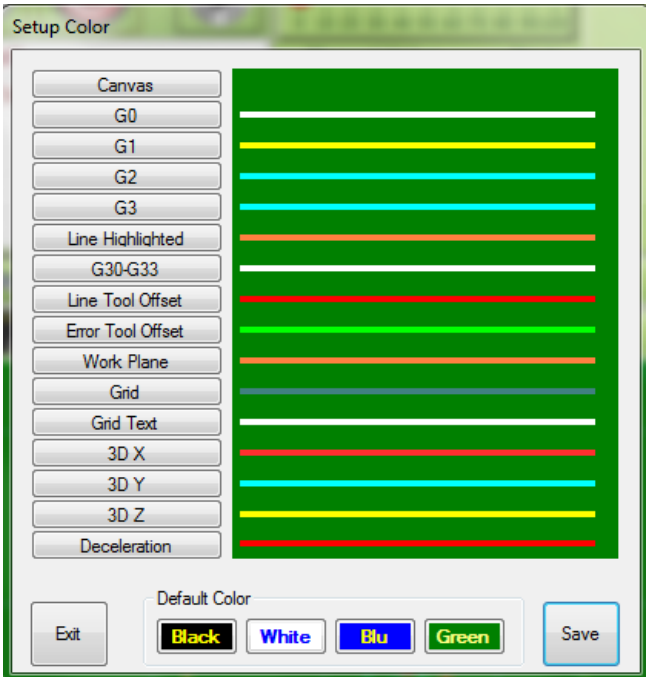


5.3.16.3 Grid On/Off

Enable or Disable Grid

5.3.16.4 Colors

Colors configuration



Canvas
Canvas Back color

G0
G0 color

G1
G1 color

G2
G2 color

G3
G3 color

Line Highlighted
highlights lines

G30 G33
G30 G33 color

Line Tool Offset
G41 G42 color

Error Tool Offset
Tool Offset errors

Work Plane
Work Plan

Grid
Grid color

Grid Text
Grid Text

3DX
3D pointer X

3DY
3D pointer Y

3DZ
3D pointer Z

Default Color
Default colors

Button Save
Save option

5.3.16.5 Filter Fast Preview

Reduces the number of strokes in the preview display to speed up the graphics. This is necessary when we Gcode relatively long (> 10000 lines) or with PC graphics are not particularly efficient. The filter allows you to set the following parameters:

Nr Activation
The filter operates only if the Gcode has a number of elements (G1 only) greater than or equal to this parameter (set according to the PC - typically 10000)

Len Min (mm)
This value expressed in mm (example: 10,000 = 1 cm) defines the action of the filter with the following algorithm:

- Are added all the micro lines in G1
- Only when the total length of microtratti reaches or exceeds the set value appears suddenly

Nr Max Elements
Specifies the maximum number of items to display in the simulation, well are not displayed.
Serves to limit the simulation when we are in the presence of files that are too large or otherwise composed of LOOP.
The default value is 100,000

5.3.16.6 Automatic change plane On/Off

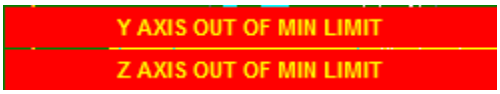
If this function is activated during processing is automatically changed the work plan in order to display the correct simulation REALTIME.

5.3.16.7 View SGLP-SGL3D

Highlight the stops on edge thresholds SGLP o SGL3D.

5.4 Alarms related to piece dimensions

If the piece comes from the size of the work plan set, a warning is displayed to alert the operator.

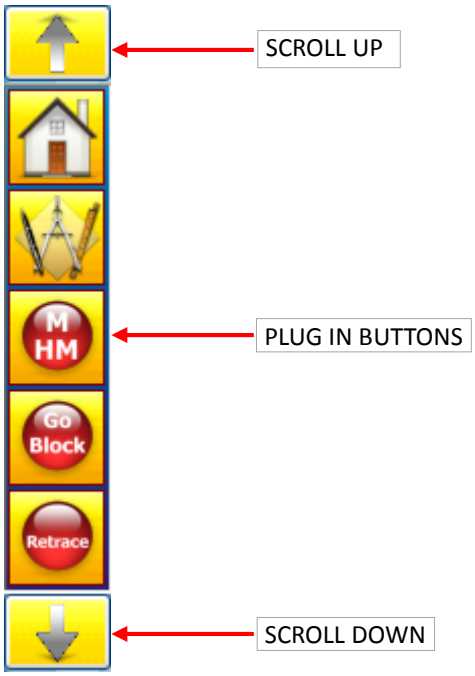


The display of the alarm can be removed by clicking on the RED LABEL . From this time the alarm will be reactivated only to START a new simulation.

6 PLUG IN

Plugins are the I function which are activated by push button or activated auitorun. These functions can be of various types, because the PlugIn were expanded according to the requirements of the machine. Described below are some plugins that can be loaded in the interface.

6.1 PLUG IN MENU' BAR



7 HOMING AXIS

Functions that manage the enables of drivers and homing of the axes.
This is necessary to boot the machine, as if there were no operations and HOMING AXIS ENABLE DRIVER, you can not use the machine in the works.
The initialization axes proceeds as follows:

1) Enable Axis "**ENABLE SINGLE**" or "**ENABLE ALL**"
2) To enable all axes been on a search for HOMING through the "**HOMING ALL**" button

7.1 ENABLE SINGLE

Pressing the button for ENABLED THE DRIVER ON AXIS.
If the driver is enabled, this is DISABLED.

7.2 ENABLE ALL

Enable all Drives.

7.3 HOME SINGLE

Pressing the button starts the homing of its axis.
The homing depends on the type of board installed and the configuration IsoNs. The **STOP** button stops the homing sequence.

7.4 HOMING ALL

Homing all Axis. The sequences are set in IsoNs.cfg

7.5 STOP HOMING

The button STOP break The Homing procedure

WARNING
STOP button does not make a emergency stop. Therefore it is necessary that the machine is prepared by all safety regulations

7.6 INDEX ENCODER OFFSET

Indicates the phase shift of the index compared to the encoder ZERO MICRO.
This is expressed unit measure.. Therefore a value 1000 in unit 0.001mm is equal to 1 mm. If the motor has a 2 mm per revolution, in this case, the index is centered in 180 degrees.
This phase is only present if you use the search type of HOME-mark of ZERO RZERO_MODE parameter to a value of 2-3-6-7 (see Programming Manual IsoNs).
The zero mark must be located at 'an angle of internal security (IDEAL 180 degrees). This avoids the fact that any error of zero switch can determine the mark search with an error of 1 encoder revolution

WARNING
This feature is not always available
It depending on sistem driver used

7.7 HOMING AND ENABLE AXES By EXTERNAL INPUTS

With the Homing PlugIn rev. 3.0.0.0 or later is possible use a external digital inputs to Enable Axes and Home Axes. For enabled this function see below:
Create by WordPad, NotePad (or a Text Editor) a text file named **Homing.cfg**, with the following data:

Nr. Digital input for enable all Axes
Nr. Digital input for Homing all Axes

Ex:
The CNC Digital input Nr. 6 is connected to button **Enable All Axes**, and Digital input Nr. 7 is connected to Button **Homing All Axes**, you must write:

6
7

Save this file in the same folder of IsoNs.exe
You must modify the file **IsoNs.cfg** (in the same folder of IsoNs.exe) in the following mode:
Edit by WordPad, NotePad (or a Text Editor) the **IsoNs.cfg.** Find the section READ_INPUT and add the digital inputs inserted in the file **Homing.cfg**

Ex:
READ_INPUT,6,7
or

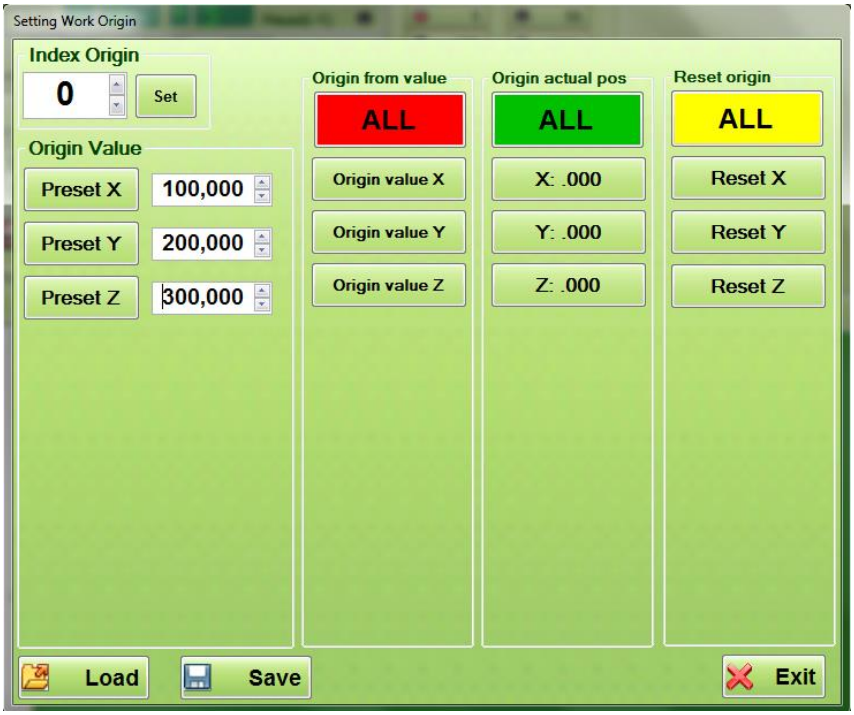
READ_INPUT,3,2,6,7

Save the file IsoNs.cfg

WARNING
Before to Modify IsoNs.cfg, BackUp this file

8 MANUAL WORKING ORIGIN

IsoNs manages up to 256 Working Origin, callable from Gcode by instruction **USER_ZERO n**, where **n** is the index of the work origin. PLUG-IN function but allows free to manage the manually origin.
This is done by carrying axes and activating the desired zero.
The zero part is automatically saved to a file and made available to the DEFAULT instructions G54, G55, G56, G57, G58, G59



8.1 Index Origin

Sets the index of the workpiece zero (0 to 255). The value is automatically reported in the appropriate fields.

8.2 ENABLE ACTUAL INDEX

The Button SET enable actual index select by index oprigin

8.3 Origin at Actual Position - ALL

This button set origin at actual axis position. The Index is set in **"Index Origin"**

8.4 Origin From Value - ALL

This Button set the origin by value set in the fields **"Orign Value"**

8.5 Reset Origins - ALL

The Button **"Reset All Origins"** remove all axis origins at actual Index set in **"Index Origin"**

8.6 Preset axis value

The Buttons **PRESET X,Y,Z** ecc. Preset the axis value at **"Origin Value"**
If the **"Origin Value"** is 100,00 the axis value is 100.000

8.7 Origin Value

This fields allows to set axis origins manually.
The Origin value refers to MACHINE ZERO

8.8 Origin single axis to value

The Buttons **"Origin Value X, Origin Value Y ecc."** sets the axis origin value from the fields **"Origin Value"**

8.9 Origin single axis to actual position

The Buttons **"X:.000, Y:.000 ecc."** sets the axis origin value from **actual position**

8.10 Reset Single Origin

The Buttons **"Reset X, Reset Y ecc"**. reset the single axis origin

Example to setted working origin in two index
Move axis to first origin

1) **Move axes to first origin**
2) **Select index 0**
3) **Press button SET**
4) **Move axes to second origin**
5) **Select index 1**
6) **Press button SET**
7) **for enable the first index, select 0 "Select Index"**
8) **Premere Set Index**
9) **Exit per uscire**

In this example, two points were made part zero, index 0 and 1.
The start is made with index 0. To change from the Gcode using index USER_ZERO N (where N is the index from 0 to 255).
To disable the part zero to 'index selected by the Gcode using G98 (suspends zero axis) or G94 X0Y0Z0ecc. (for all axes configured)

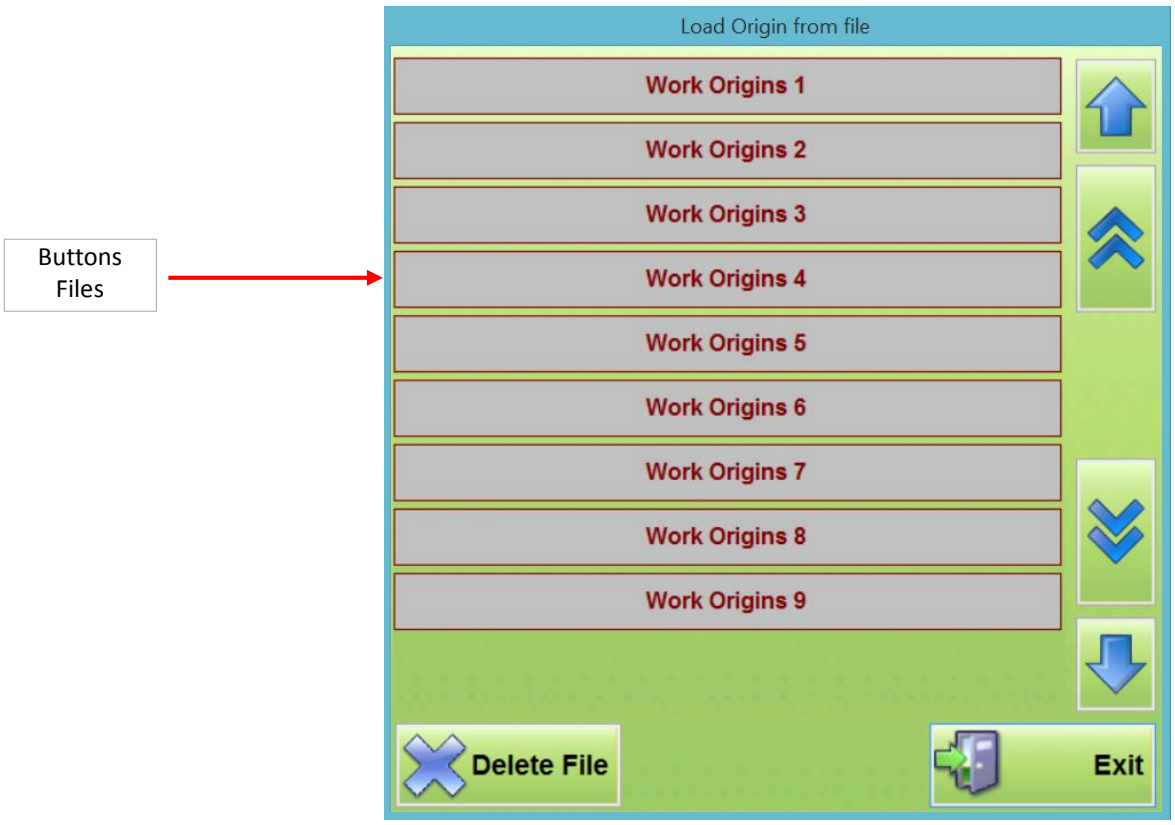
8.11 Save Actual Origin setting

With **SAVE** button you can save in the files the ACTUAL ORIGIN SETTING.
This allows using the LOAD button, to reload the previously saved ORIGINS

Insert the file name and press Ok to Save

8.12 Load the origins from file

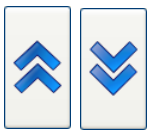
Pressing the **LOAD** button you can load a previously saved ORIGIN file . The session is loaded with zeros displayed in the fields QUOTE AXIS imposed, but is not enabled. To enable it press **"Origin From Value"**, or use the instructions: G54, G55, G56, G57, G58, G59 of Gcode



The list of the saved file is presented as a button, so to load the related files, just press the button.

8.12.1 SCROLL UP DOWN

Scroll 1 position



Scroll 1 page

8.12.2 Delete file

Using the DELETE FILES button, you can delete a file saved part zero.
After pressing the button, all files are displayed with a red background.

Now press on the related files to delete. (the file is not completely eliminated, but only moved to the Recycle Bin again to be recovered).

If you wish to exit the delete button again so that the DELETE FILE 'file list returns to its original color.

8.13 Enable PassWord for Work origins

Is possible in the PlugIn Rev. 3.0.0.0 or later, use the PassWord code to enable the Work Origin setting.
For use this function you must create a empty text file (by WordPad,NotePad etc.) named **ZeroPezzo.cfg** and save it in the same folder of IsoNs.exe. If this file is found by ZeroPezzo plug in, the commands are enabled only if the PassWord is inserted in correct mode.

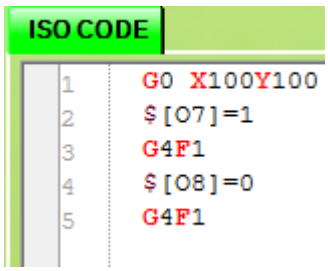
Insert the code in the RED field, and press Button Key

9 COMPILE M or HM to PC

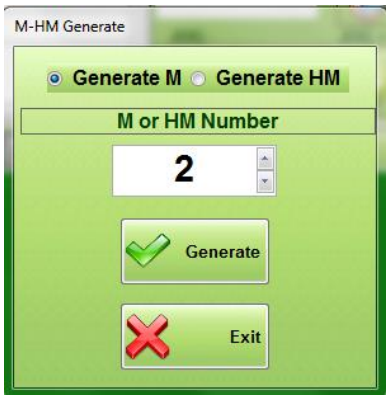
IsoNs manages functions M or HM (Extended M) as a special Gcode resident in a particular folder on your PC. The function M or HM is derived from a standard ISO program, with the simple difference that must be generated by this function. Once generated the function M or HM This is recalled in other parts program. You can also call functions M from other functions M is resident in the PC is resident in the CNC. IsoNs check first if the function M is invoked on a PC if the negative is called the M CN

9.1 Create M - HM

1) Must first write the ISO code is used to manage the M or HM



2) Then activate the plugin MHM



3) Select the Generate M button to create a function M
Select the Generate HM button to create a function HM

4) Select the M HM Number

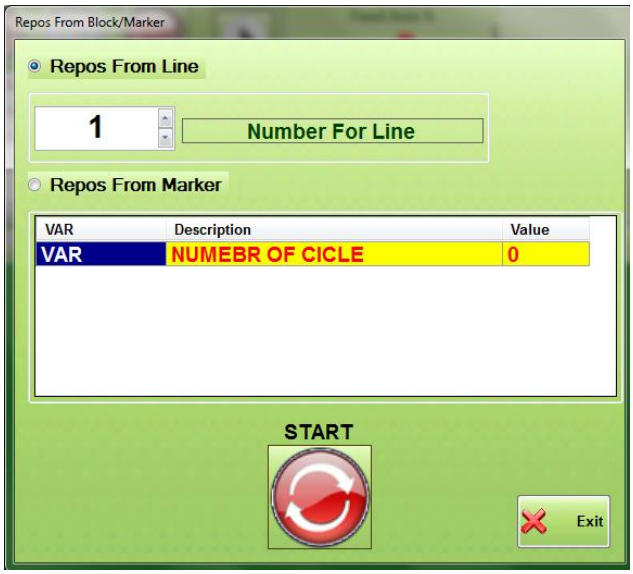
5) Press Button Generate

Now the M or HM (example M 2) is available in Gcode with **Mnumber**
In the plugin FuncMHM version 2.0.0.0 or higher the source code is automatically saved in the folder:
_Source_M and ***_Source_HM***
The name is :

Ns_M+(M number) **Ex: Ns_M10**
Ns_HM+(HM number) **Ex: Ns_HM10**

10 REPOS FROM LINE or MARKER

This is a feature that allows you to run the Gcode from a number of any line or the value of a marker.
The line number can be selected graphically using the 3D simulation. The block includes the restart of the execution of the Gcode in the particular mode until it is encountered or the number of line or restart VALUE OF MARKER. Once this is achieved, it performs the any **M GOBLOCK** configured. This usually arranges for the work. The axes are moved in G0 mode to block select and the process begins after the execution of **M GO BLOCK**.



Repos From Line
Select the check **"Repos From Line"** and enter the line number for recovery from block.
If the simulation is activated NUMBER OF LINE SEARCH OF PART OF THE PROGRAM, automatically writes the line number highlighted by the function.

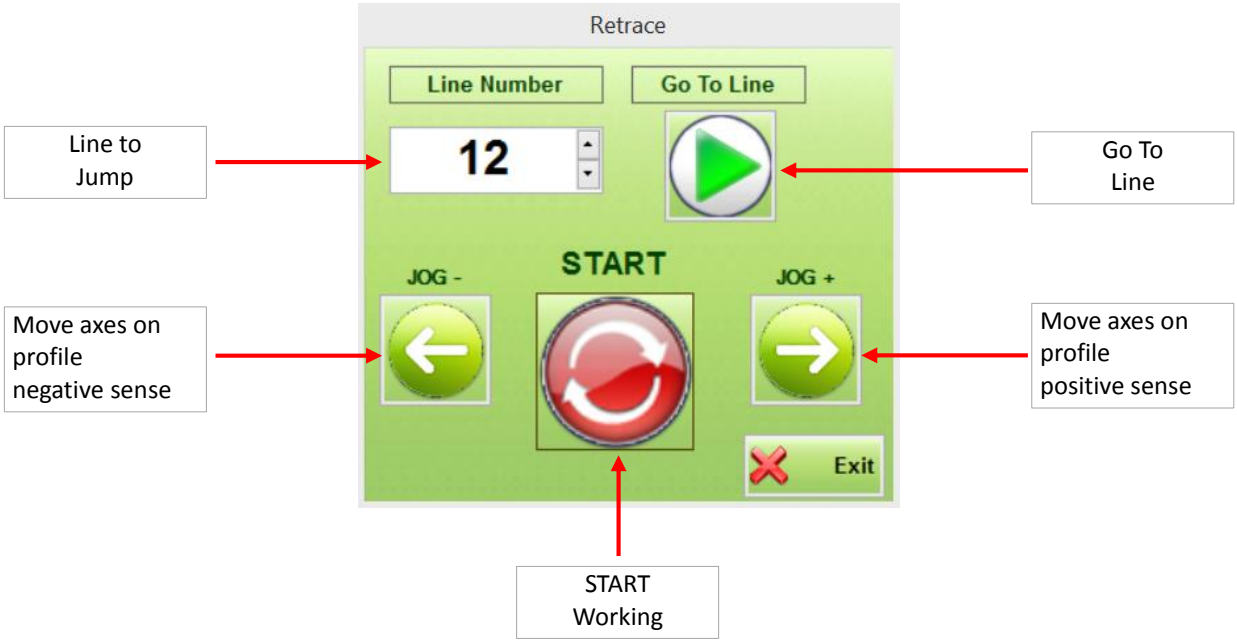
Repos From Marker
Select the check "**Repos From Marker"** and enter the value that must be satisfied for the restart
The marker is a special variable isons.
processing resumes when the value of the marker is set

ISO CODE

```
MARKER $VAR NUMBER OF CICLES
G91
LOOP 100
        G1X10Y10
        $VAR=$VAR+1
END_LOOP
```

11 RETRACE

Very special function that fits a certain type of machine.
This allows you to run the profile actually moving the JOG MODE.
Contrary to the **"Repos From Line or Marker"** allows for a restart from any point of the stretch of a block (as opposed to **"Repos From Line or Marker"** allowing only start from 'the beginning of the stroke). The axes are really moved in order to facilitate the operator to precisely locate the point of restart on the piece.
At the START process is executed if configured **M GORETRACE** (prepares the tool to work):



11.1 Line To Jump

Nr. Line to initial JUMP (avoid scrolling jog profile)

11.2 Go to line Jump

Move the Axes at line to jump.

11.3 JOG -

Scrolls the profile in a negative way. By holding down the axles moving in the negative words 'to the' beginning of the ISO Gcode covering all the strokes in the opposite direction at the speed of the single trait. Releasing the button stops the axes at the point where they are.

11.4 JOG +

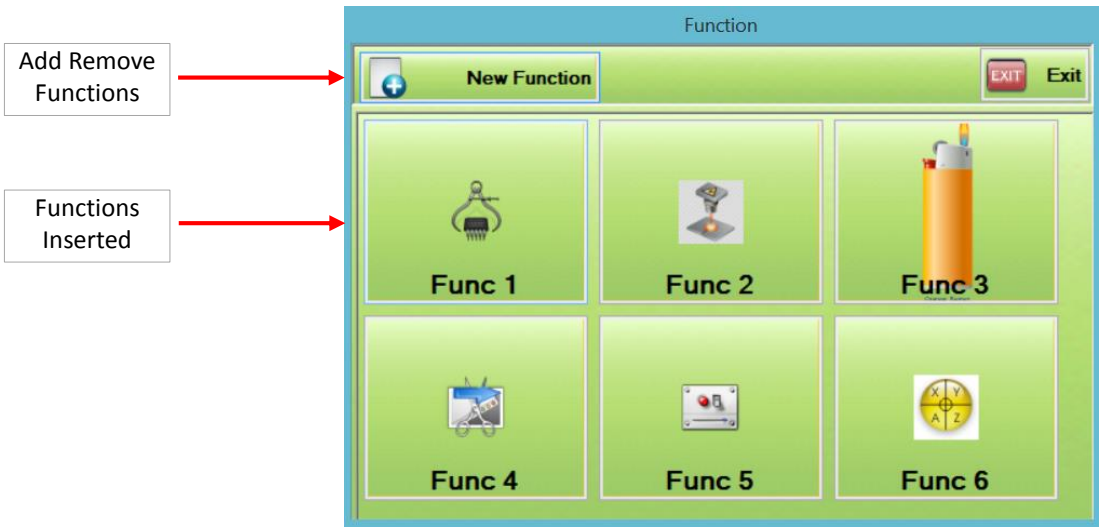
Scrolls the profile in a positive way. By holding down the axles moving in a positive sense that 'towards the end of the ISO Gcode covering all the strokes in the right direction at the speed of the single trait. Releasing the button stops the axes at the point where they are.

11.5 START

Start working from the current point where they are aces.
Is performed whether the M GORETRACE configured.

12 SPECIAL FUNCTIONS

This PlugIn contains special functions.
The special features are Gcode of ISO (ISO Script) that are launched by pressing a button.
For the management of special functions using the special utility MANAGEMENT FUNCTIONS
Unlike a normal Gcode, scripts can be executed even when the CPU is in PAUSE.
This allows you to launch the service functions for the 'user (Manual tool change, cleaning tool, etc..) Script run all the code ISO ISO limited to only linear interpolation (thus missing G2 G3 G41 G42 and consequently refer to the manual programming to see the limitations of the Script ISO code).

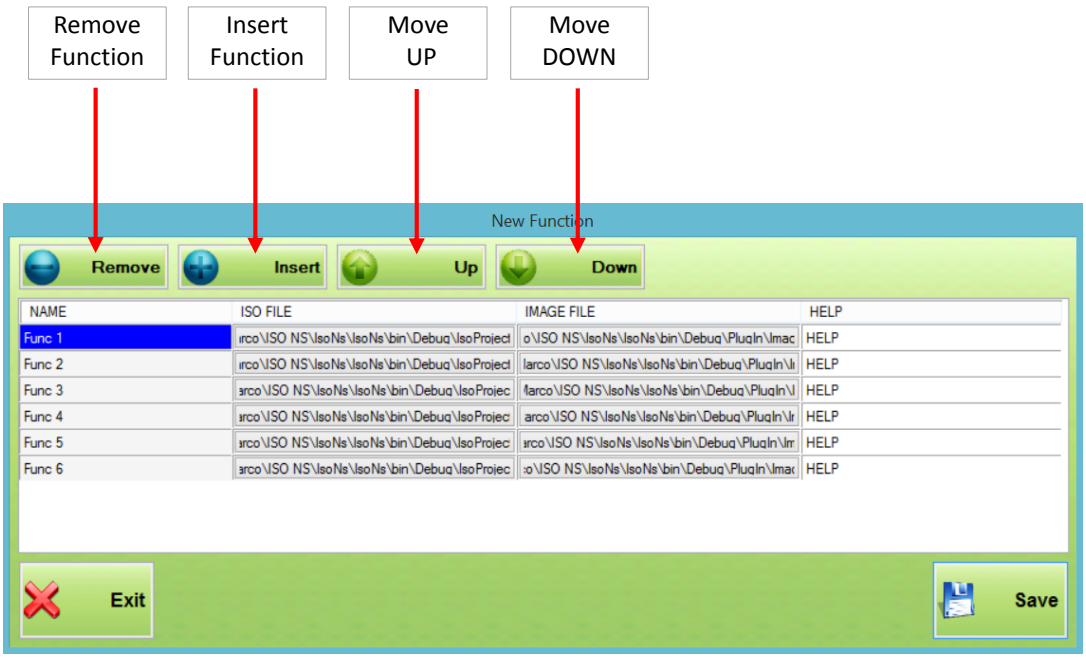


12.1 ACTIVATED FUNCTION

The activation function is performed by pressing the appropriate button.
The button always allows general STOP to interrupt the function USE

12.2 ADD REMOVE FUNCTIONS

Insert the new functions



12.2.1 Remove Function
Delete the selected function (blue bar) from the list of functions

12.2.2 Insert Function
Insert the new function

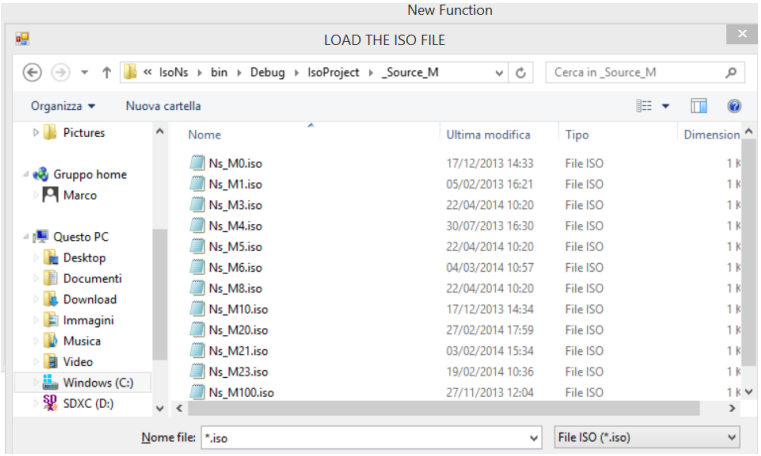
12.2.3 Move Up
Move Up the selected function

12.2.4 Move Down
Move Down the selected function.

12.2.5 Save
Save the current configuration of the functions (WARNING the old configuration is permanently lost)

12.2.6 Assign or modify the function name
To assign or change the name of a function, simply double click on the **Name field** and enter the new name. The name is then represented on the button so you need to pay attention to the string length.

12.2.7 Assign or modify ISO SCRIPT
To assign or change the ISO Script code to be executed by the function and press the button for ISO FILE. Below you open the browser standard for managing Windows files.
Look for the right file in the folder and press OPEN.
Please note, at this stage is saved the full path of the file, then copying the configuration scripts on another computer that the files must be in the same folder.



12.2.8 Assign or modify image file
The image files need to have a graphical representation of the function to display the button.
This is optional.
Operate in the same way as **Assign or modify** the Script ISO code by uploading your image file (JPG, PNG, BMP, etc.).
The size of this must be adapted to the button (96 dpi is the maximum recommended).

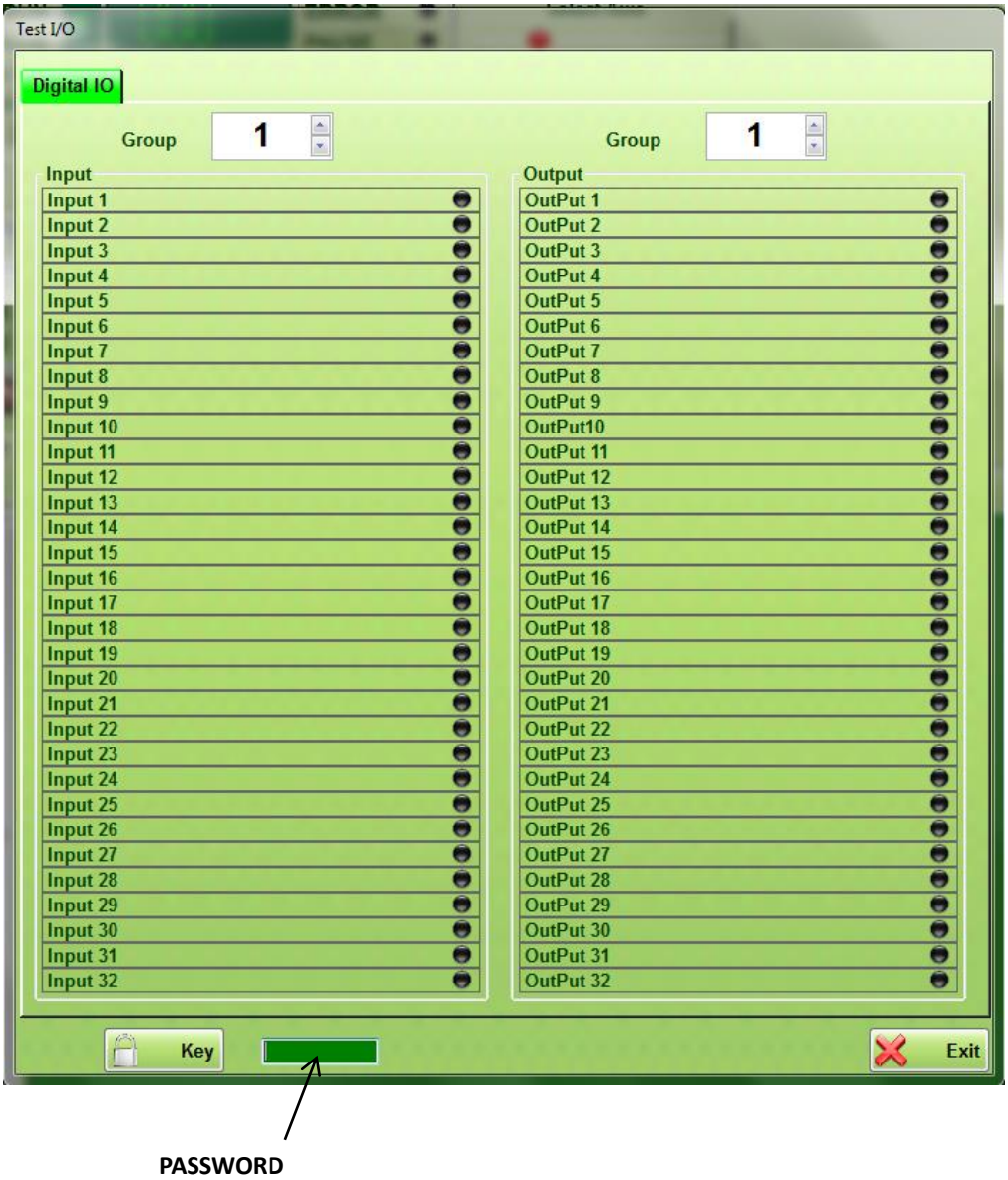
12.2.9 Assign or modify Help
The help is represented as a balloon when the mouse is over the button. The help is not a required field but has a help function is recommended.
To assign or change the Help of a function, simply double click on the HELP field and enter the new text

13 TEST I/O

Allows the digital I/O of CN.
The use of this function must be made by an expert in that interacts with the machine in an uncontrolled manner.

13.1 Test Digital I/O

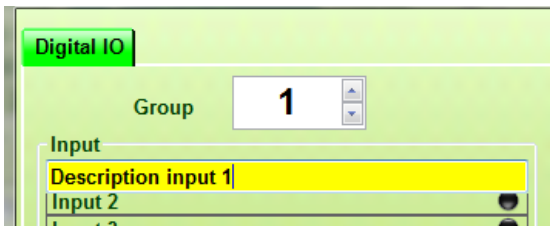
Displays status of all inputs and digital outputs. It also allows the SET and RESET of the digital outputs. The name of the I/O is in the file **"NsIO.cfg"**



PASSWORD

The Group selects a range of 8 groups of 32 available I / O (up to 256 digital I / O)
The corresponding LED indicates the status of input / output.
The SET / RESET output is controlled by clicking a mouse on the name of output.
The Output SET/RESET can be used only after have inserted the Password in the field and pressed the button KEY. The PLC cycle to internal control can prevent CN SET / RESET outputs of

Is possible change the I/O description by the PlugIn Testio
Is need the correct PassWord
Procedure:
Press the right mouse button in the desired description to change, insert the new description in the Yellow field and press the CR key of KeyBoard for complete the insertion.



Manual management NsIO.cfg file
Is also possible change the I/O description in the file NsIO.cfg by text editor (ex: notes)
The NsIO.cfg is present in the same folder of IsoNs

NsIO.cfg Format
For added a I/O description insert:
I/O Number=Description
Ex:
23=Input 23
<u>It is not necessary to maintain a chronological order of the I / O</u>

[DIGITAL_INPUT] → ***identifier digital inputs section***
1=Input 1 → ***Input Number and Description***
2=Input 2
3=Input 3
4=Input 4
.
.
38=Input 38
[END_DIGITAL_INPUT] → ***End digital inputs section***
[DIGITAL_OUT] → ***identifier digital outputs section***
1=OutPut 1 → ***Output Number and Description***
2=OutPut 2
3=OutPut 3
4=OutPut 4
5=OutPut 5
6=OutPut 6
7=OutPut 7
10=OutPut10
[END_DIGITAL_OUT] → ***End digital outputs section***

14 NsKeyJog.dll

This PlugIn if installed, allow axis manual motion from PC KeyBoard. This plugin is activated in AUTORUN.

14.1 Default keys config

NsKeyJog configures automatically the Keys of KeyBoard. Is possible personalize the configuration from keys sequence :

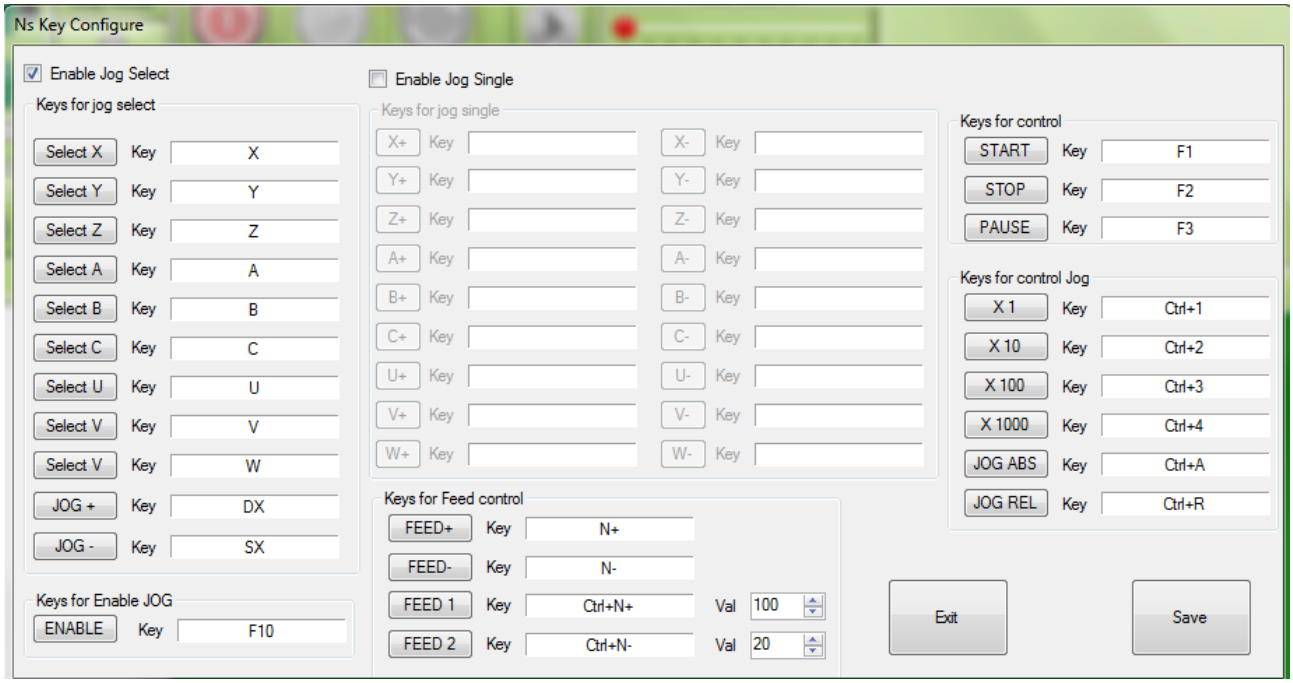
Ctrl *and* ***Shift*** *and* ***F12***

If the Keys functions is not actived, need to press the defined key **"FUNCTIONS ACTIVATED"**

KEY	FUNCTION
X	Select X axis for JOG
Y	Select Y axis for JOG
Z	Select Z axis for JOG
A	Select A axis for JOG
B	Select B axis for JOG
C	Select C axis for JOG
U	Select U axis for JOG
V	Select V axis for JOG
W	Select W axis for JOG
RIGHT ARROW	JOG + Axis selected
LEFT ARROW	JOG - Axis selected
F1	START PROGRAM
F2	STOP PROGRAM
F3	PAUSE PROGRAM
F10	FUNCTIONS ACTIVATED
KEY PAD +	Increase FEED override
KEY PAD -	Decrease FEED override
Ctrl , KEY PAD +	FEED 100%
Ctrl , KEY PAD -	FEED 20%
Ctrl , A	Absolute JOG
Ctrl , B	Incremental JOG
Ctrl , 1	Incremental JOG x1
Ctrl , 2	Incremental JOG x10
Ctrl , 3	Incremental JOG x100
Ctrl , 4	Incremental JOG x1000

14.2 Keys configure

With the following combination of keys, activate the configuration menu
Ctrl** e **Shift** e **F12

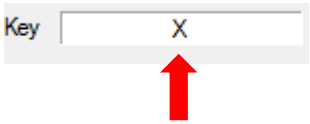


Enable Jog Select
If this is select, is actived the JOG axis from selector (Keys for jog select)

Enable Jog Single
If this is select, is actived the SINGLE JOG axis (Keys for jog single)

Keys configuration
Pressing the key (activation red) starts on the configuration of the function. At this point, simply press the key combination of Ctrl, Alt, Shit and / or the key you want to associate. Pressing the button a second time (when it is red) the selection function is disabled

Deleted key
To delete a keyboard shortcut associated with it, simply double click on it eg:



Select X,Y,Z,A,B,C,U,V,W
Allow to configures Keys for Axis Selector (if Enable Jog Select is activated)

X+,Y+,Z+ etc.
Allow to configures Keys for JOG single Axis (if Enable Jog Single is activated)

Enablel
Configures key **FUNCTIONS ACTIVATED**

FEED+ FEED-
Configures keys for increase or decrease FEED

FEED 1 FEED 2
Configure keys for predefined FEED.
Val containes the percentage of FEED

START STOP PAUSE
Configures keys for START,STOP,PAUSE PROGRAM

X1 x10 x100 x1000
Configures keys for increment JOG

JOG ABS JOG REL
Configures keys ABSOLUTE or RELATIVE JOG

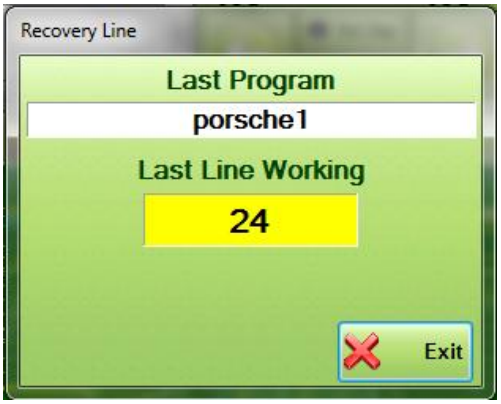
15 Recovery

This function manages the recovery of the last line processed after a program interruption.
The termination of the program, can be caused by the following conditions:

1) ***Button Stop***
2) ***Allarm***
3) ***Black Out***

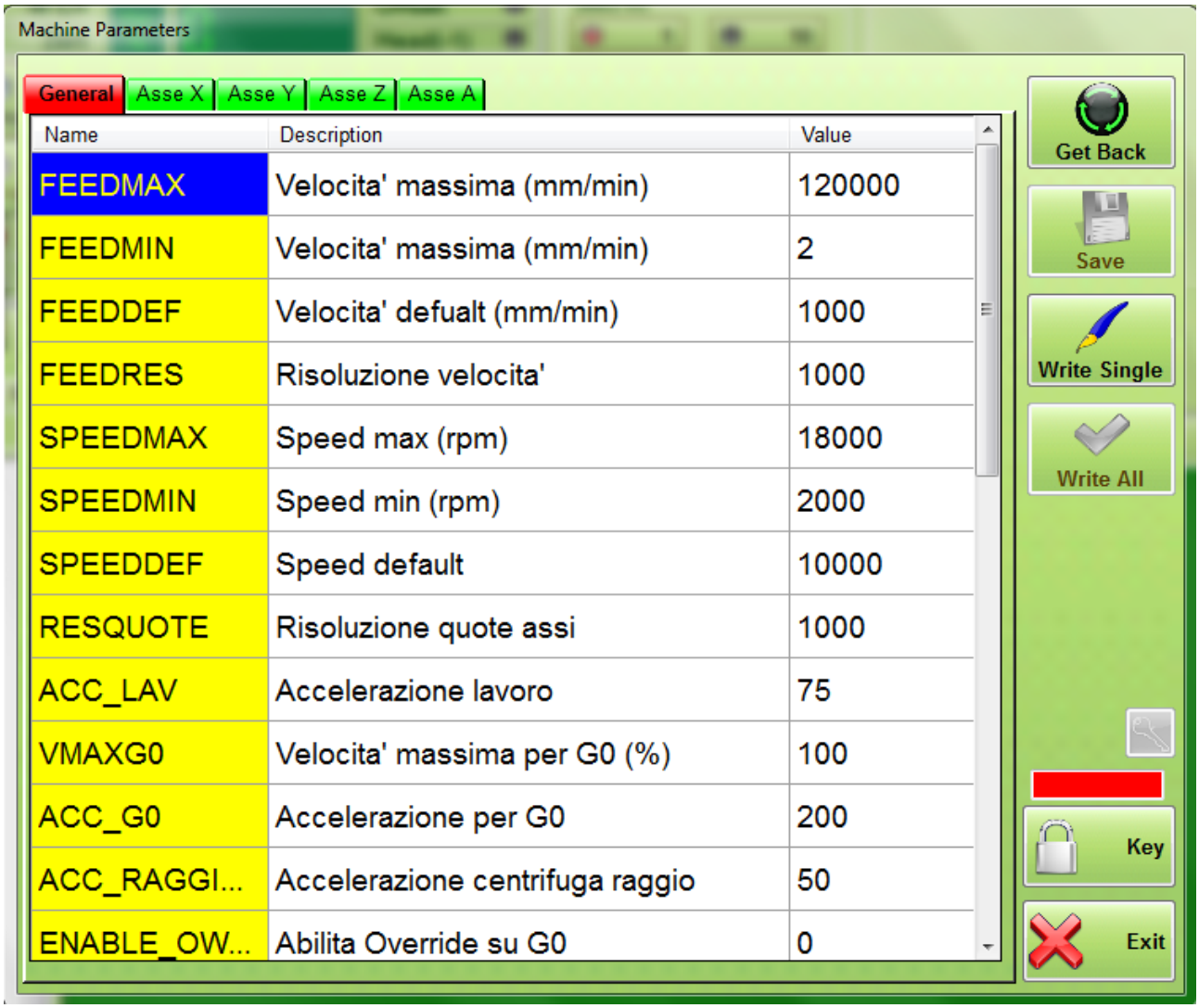
The Recovery feature shows the last line in the works.
In this case it is not, however, said that this line is completely processed.

If the Gcode is interrupted for the condition 3, a message flashes to alert you of a
ABNORMAL EXIT PROGRAM. This means that a fault has occurred due to external causes (eg sudden power failure).



16 MANAGEMENT MACHINE PARAMETERS

Through this form you can manage the parameters of the CNC machine in use and ISONS
The writing of the machine parameters is password protected, since this' should only be performed by authorized personnel. Changing the machine parameters can affect the operation of the machine also causing damage to it.
For a description of the machine parameters refer to the PROGRAMMING MANUAL.
The machine parameters are still displayed even if no password is entered.
All parameters are divided into groups according to the type of these, Parameters have a name, a description and a value.
The value parameter is always an integer and therefore contains no decimal part.
To enable the management form the machine parameters, press the following button:



16.1 GET BACK

This button restores the previous version of the parameters used.
All parameters will be reported to the state before the rescue.

16.2 SAVE

Save on disk

16.3 WRITE SINGLE

Writes the selected parameter in the CNC

16.4 WRITE ALL

Writes all Parameters in the CNC

16.5 PASSWORD CHANGE

This button is enabled when you enter the correct password. you can change the current password, enter a new password in the Password field and press this button.

WARNING
Remember the new password inserted.
If this is forgotten contact SERVICE

16.6 KEY

This button activates the ability to write parameters.
If the password in the Password is correct, will be enabled to write the parameters.

PassWord Field

Default PassWord=684620

Index

1 PREFACE 3
2 General 3
2.1 Axis Units 4
2.2 CN STATUS 5
2.3 AXES FEED 6
2.4 BUTTONS CONTROL 6
2.5 INFORMATION WINDOW 7
2.6 GCODE ISO 8
2.6.1 LOAD GCODE FROM DISK 9
2.6.2 SAVE GCODE TO DISK 10
2.6.3 MODE DISPLAY GCODE 11
2.6.4 BROWSER CONFIGURATION MANAGEMENT PROGRAMS 12
2.6.5 FIND IN FILE 13
2.6.6 Replace Text 13
2.6.7 LOAD LAST 13
2.6.8 NEW 13
2.6.9 ISO WINDOW 14
2.6.10 MDI WINDOW 15
2.6.11 DATA INPUT MASK 16
3 ADVANCE GCODE FUNCTIONS 17
3.1 Break Points insert 17
3.2 VARIABLES WATCH 17
3.3 BREAK POINT REMOVE 17
4 MANUAL AXIS JOG 18
4.1 Select Axis 18
4.2 Select Inc 18
4.3 Feed Axis % 18
4.4 Manual Value 18
4.5 JOG-/JOG+ 18
4.6 Inc Jog 18
4.7 Ext OW 18
5 3D PREVIEW 19
5.1 Overall dimensions piece 19
5.2 Axis Value 19
5.3 Functions Buttons 20

5.3.1 ZOOM IN/ZOOM OUT 20
5.3.2 ZOOM WITH MOUSE - PINCH TO ZOOM 20
5.3.3 PAN AREA 20
5.3.4 SEARCH THE LINE GCODE NUMBER 20
5.3.5 TOOL DIRECTION 21
5.3.6 TOOL OFFSET 21
5.3.7 SLIDING ON THE PATH OF SECTION 21
5.3.8 CENTER PIECE ON SCREEN 21
5.3.9 CENTER DIMENSION 21
5.3.10 VIEW Work Plan 22
5.3.11 ROTATE X,Y,Z 22
5.3.12 ROTATE TO CENTER 22
5.3.13 EXTENDED MENU 22
5.3.14 PRESET VIEWS 22
5.3.15 MEASURES 22
5.3.16 OPTION 23
5.4 Alarms related to piece dimensions 25
6 PLUG IN 26
6.1 PLUG IN MENU' BAR 26
7 HOMING AXIS 27
7.1 ENABLE SINGLE 27
7.2 ENABLE ALL 27
7.3 HOME SINGLE 27
7.4 HOMING ALL 27
7.5 STOP HOMING 27
7.6 INDEX ENCODER OFFSET 28
7.7 HOMING AND ENABLE AXES By EXTERNAL INPUTS 28
8 MANUAL WORKING ORIGIN 29
8.1 Index Origin 29
8.2 ENABLE ACTUAL INDEX 29
8.3 Origin at Actual Position - ALL 29
8.4 Origin From Value - ALL 29
8.5 Reset Origins - ALL 29
8.6 Preset axis value 29
8.7 Origin Value 29
8.8 Origin single axis to value 30
8.9 Origin single axis to actual position 30

8.10 Reset Single Origin 30
8.11 Save Actual Origin setting 30
8.12 Load the origins from file 31
8.12.1 SCROLL UP DOWN 31
8.12.2 Delete file 32
8.13 Enable PassWord for Work origins 32
9 COMPILE M or HM to PC 33
9.1 Create M - HM 33
10 REPOS FROM LINE or MARKER 34
11 RETRACE 35
11.1 Line To Jump 35
11.2 Go to line Jump 35
11.3 JOG - 35
11.4 JOG + ... 35
11.5 START 35
12 SPECIAL FUNCTIONS 36
12.1 ACTIVATED FUNCTION 36
12.2 ADD REMOVE FUNCTIONS 36
12.2.1 Remove Function 37
12.2.2 Insert Function 37
12.2.3 Move Up 37
12.2.4 Move Down 37
12.2.5 Save 37
12.2.6 Assign or modify the function name 37
12.2.7 Assign or modify ISO SCRIPT 37
12.2.8 Assign or modify image file 37
12.2.9 Assign or modify Help 37
13 TEST I/O 38
13.1 Test Digital I/O 38
14 NsKeyJog.dll 40
14.1 Default keys config 40
14.2 Keys configure 41
15 Recovery 43
16 MANAGEMENT MACHINE PARAMETERS 44
16.1 GET BACK 45
16.2 SAVE 45
16.3 WRITE SINGLE 45

16.4 WRITE ALL 45
16.5 PASSWORD CHANGE 45
16.6 KEY 45