CNC mdl

Engraver ver. 01a

User's manual

INTRODUCTION

Engraver is born from the need to avoid the purchase of expansive and complicated CAM software that generate a tool path and the corresponding G-Code, thus allowing us to make our machine do what we please.
Initially thought for simply convert the measure of the X-axis by bringing them back to the rotation axis A, it is afterwards been improved until it is become a complete and easy to use software that can be a good option to the expansive software in commerce.
The improved features allow to:

- save the works in an archive;
- recover from the archive the saved works;
- export and recover archives on external files;
- various views of the work;
- convert and import text files in “xyz” format from other programs.

With the purchase of the software is granted **the right to use and free access to future upgrades that can bring improvements to the software. It is not in any way transferred the ownership of the program or any other right on it.**

It is clarified that the program can be installed just on a single computer: to each PC corresponds a unique activation code that can be used only on that PC.
Other programs are in study which will improve the features of this software (automatic preparation of the incision surface and final adjustment of the surface engraved), for which it will be recognized a special price to the ones already owners of the Engraver’s use license.

The Engraver’s team thanks you for the purchase of this software.

PROGRAM'S FEATURES

Engraver is an application designed to engraving operations of writings, drawings etc., but does not do automatically the tool compensation. It automatically generates the tool path and the corresponding G-code to be loaded on the CNC machine tools.
As we will see later, although the software is designed exclusively for incisions, you can perform other operations very special and fun. Engraver performs works of engraving on both flat and cylindrical surfaces, also working on the rotary axis "A".
The intuitive graphic interface is as simple as possible to allow for quick and easy data entry and obtain the desired result without any complications.
It is able to handle up to 1000 instructions, repeatable automatically up to 25 times to allow precise incision to the desired depth, in order to generate an ideal tool path for each type of incision.
This software allows you to set:

1. the depth of incision to step; however, the software will calculate the ideal depth step to reach the set engraving depth in a precise manner;
2. for the xyza works ("on cylinder"), the number of repetitions of the incision on the cylinder circumference. There is no limit to the number of repetitions;
3. the offset of the origin of the axes “X”, “Y”, “Z”, “A”, independently on each axis, to allow the incision in the desired coordinates;
4. a scale factor on the axes “X”, “Y”, “Z”, “A”, independently on each axis , in order to perform the incision by adapting it to the workpiece measures or according to the desired measurements;
5. the speed of penetration (axis “Z”) and shear (axes “X”, “Y”, “A”) according to the type of material being cut;
6. the clearance between the tool and work piece to allow adequate space between the tool and the workpiece in the phases of translation between an incision and the other.

In order to facilitate data entry jobs, through a simple automated procedure, you can import coordinates from a text file in the format “xyz” generated by other programs.
You can save up to 1,000 jobs on flat surfaces (“XYZ”) plus an additional 1,000 jobs on the circular surfaces (“xyza”). The archives of the works can be exported to an external file archives and, when appropriate, be recovered.
Both while typing the coordinates or when the work is finished, it is possible to graphically view the work and verify the measurements.
In the display of the work on axes xyza (on cylinder), you can put on view the job compared to the development of the entire cylinder circumference in the process, with the view of both the previous and next incision.
This version operates by linear interpolation of the coordinates. The execution of arcs and circles is possible by tracing very closely spaced dots on the arc or circle to engrave.
The software runs on Excel platform and **requires the activation of the macros; the macros are strictly necessary and essential to the proper functioning of the software. It is declared that the macros in this software do not contain malicious code or virus and therefore are believed to be safe. This software does not automatically transmit any data related to the user's computer.**

HOW TO INSTALL

ATTENTION:
Each PC provides its own code. It is therefore necessary to perform the "Generate Installation Code" from the PC where the software will be installed.

Photographic Documentation: Appendix A

1. Run the “Generate Installation Code” file and follow the screen instructions;
2. make the payment of the software and indicate the “Installation Code” in the reason of the payment;
3. fill the ORDER FORM in his every part and send it at the e-mail address or fax number indicated;
4. wait until the sending of the program file and the corresponding activation code.

The file containing the software is in the format Win-Zip Self Extractor, follow these steps:

1. Place the file in the desired folder (e.g. "Documents");
2. Click on the file, automatically will start the decompression procedure and installation. It will be generate a folder (CNC mdl - Engraver) containing the software and a subfolder "Archives" containing the file "CNC mdl - Engraver Ver. 01a - Archive".

Done this, go to the “CNC mdl – Engraver” folder.

It is important to make sure you have activated the Excel macros, because the software needs these activations.

To carry out the necessary activations of the macros please refer to the guidance related to your version of Excel.

At this point you can launch the program “CNC mdl – Engraver Ver. 01a”.

Given the scale and complexity of the formulas present, the boot process may be slow. This, however, is proportional to the speed of your PC.
Wait for the following screen:

Photographic Documentation: Appendix B

1. Click on the button “Register”.
2. Follow the screen instructions.
3. Read carefully the use conditions.
4. Click on the button “Accept Terms and Conditions”.

At this point, a screen appears with reported the “Installation Code”, exactly the same as the one supplied by the software “Generate Installation Code”, and a green field to fill in with the “Unlock Code”.
Type the “Unlock Code” you received, **without dots or spaces** and click enter; then click on the button below.
It will appear a field with the words “Registered version”.
In event of error of typing of the “Unlock Code” and therefore with the message “Not registered version”; try again by clicking on the button “Retype the Unlock Code” and pay close attention in the code typing.
When you have done this, click on the button go ahead.
It will appear a screen that shows the codes and the terms and conditions of use.
Please read carefully and print a copy.
Accept the terms and conditions of use to go ahead or close the program.

Make sure that the words “Registered Version” appears. In event of troubles contact the supplier.
At this point the software is registered and running.

Given the computational complexity and the importance of getting a G-Code file in highly reliable, so as not to damage the CNC equipment used and to preserve the safety of the operators, we kindly ask you not groped operations to crack the software. If the software is modified in its original codes, it may provide machine instructions not conform to required specifications.

To exit click on “Save and Close the program” button. The program will be saved and closed.

To continue the work, press “Click on the button to continue”.

In order to use the machine tools is essential the employ of the personal protective equipment and is important to pay close attention and vigilance.

TIPS

You can adapt the various screenshots of the software at the resolution of your screen or according to your need:

1. at each screenshot click the buttons “Ctrl + Scroll” (click the “Ctrl” button and rotate the “mouse’s wheel”), you will get the desired screen zoom;
2. save and exit the program;
3. open the program again.

Now the view of the screens will adapt at your settings.

When the registration is done, at each opening of the program will appear the first level menu “Home”, which verifies the software registration.

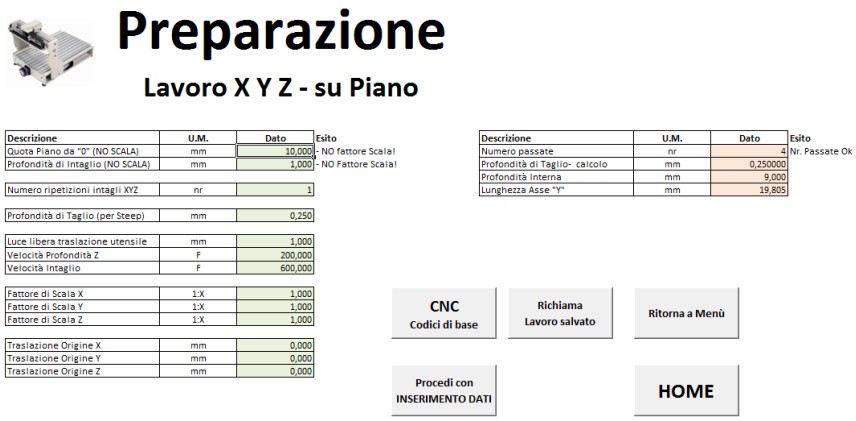
Click on the button “Click on the button to continue” and the second level menu will appear.

The page contains 4 options (Second level menu):

- 3 axes XYZ
- 4 axes XYZA
- Service
- Back to Home

3 Axes XYZ

This button brings the program at the work in plane section, in order to engrave a plane using the 3 axes of the machine tool: plane X “transversal”, plane Y “longitudinal” and plane Z “height / depth”.



In addition to the input fields (input fields in green), tab buttons are:

- CNC Basic codes
- Continue with data input (entering coordinates)
- Recover saved work
- Back to the menu (brings you to the previous menu)
- Home (brings you to the main menu to save and close the program)

First set the initial parameters of the work to be performed:

1. Quote of the plan to engrave (e.g., thickness of the workpiece);
   a. if you set quota to “0” or the share of incision will be higher than the share of the plan to engrave, the program will warn you to pay attention to “bottom quota” in order to avoid incision of the plane of the cutter!
      WARNING: the program proceeds to the incision anyway!
2. The notch depth (how deep the incision should be, compared to the portion of the plan to engrave previously set);
3. The number of skim cuts of the same drawing.
   a. Note: For operations in plane is sufficient to leave nr. 1
4. The notch depth for each pass (step);
5. The clearance between the tool and the portion of the plan to be engraved;
   a. the clearance refers to the space between the tool tip and the “Quote of the plan to be engraved” that is generated during the traslation of the tool between one side and the other of the workpiece (such as when you stop the incision of a line and the tool moves to the next line to be engraved).
6. The speed feed of axis “Z” (drilling speed on axis “Z”)
7. Feed speed of notch (traslation speed of the axes “X” and “Y”)
8. The scale factors of the axes “X”, “Y” and “Z”;
   a. The scale factors are equal to "1" when there is no reduction or enlargement.
9. The traslations values with respect to the origins of the “point zero”.
   a. Leave the values at “zero” for not having traslations.

At this point, the software calculates mainly the number of “passes” (steps) required to the incision (max 25 steps) and adjusts its value in order to reach accurately the “notch depth”.
In case they are needed more than 25 steps, the program will give an “Error”.

Solution 1): slightly increase the notch depth for each pass (step);
Solution 2): reduce the total depth of carving.

Button “CNC Basic Codes”

Then you go to the configuration of the CNC: click on the button "CNC Basic Codes."

You find:

1. Initial settings:
   consists of 10 lines where you will enter the initial configuration commands of the machine (input fields in green).
   Example:
   on the first line write: G0 G49 G40 G17 G80 G50 G90
   Where:
   G0 = rapid positioning;
   G49 = deletes tool length offsets;
   G40 = deletes the compensation of the radius of the blade;
   G17 = selects the work plane axes “XY”
   G80 = deletion of the movement (including the programmed cycles)
   G50 = resets all the scale factors to “1”
   G90 = sets absolute distance mode.

   In the second line we can write: G20 (Unit inch) or G21 (Unit mm).
   Pay attention that some software, even with G21, distorts the values of millimeters.

   Other primary command: M3 followed by the command S “Spindle Speed”.
   Sample line:
   M3 S20000
   M3 indicates ignition and spindle rotation (engraver) clockwise;
   S20000 indicates the spindle speed to 20,000 rpm.

2. Final Settings:
   is composed of 10 lines (input fields in green) where you will enter the final configuration commands of the machine.
   Example:
   in the penultimate line write: M5
   in the last line write: M30
   Where:
   M5 = turns off the spindle;
   M30 = indicates the “End of program”.

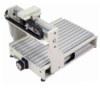
The setting is essential before any operation since the machine tool may have configurations, loaded with previously codes, do not comply.
The informations provided above are simply an example.
The user is responsible to properly configure the software to suit his needs!

From the current tab, you can call through the appropriate buttons, a short illustrative list of codes "G" and "M".

Codici G

Ritorna

Codice	Descrizione
G0	Posizionamento rapido
G1	Interpolazione lineare
G2	Circolazione in senso orario/interpolazione elicoidale
G3	Circolazione in senso antiorario/interpolazione elicoidale
G4	Pausa
G10	Impostazione d'origine del sistema di coordinate
G12	Pezzo circolare orario
G13	Pezzo circolare antiorario
G15	Movimenti polari delle coordinate in G0 e G1
G16	Movimenti polari delle coordinate in G0 e G1
G17	Selezione del Piano XY
G18	Selezione del Piano XZ
G19	Selezione del Piano YZ
G20	Unità pollici
G21	Unità millimetri
G28	Return a Home
G28.1	Assi di riferimento
G30	Return a Home
G31	Indagine diretta



Codici M

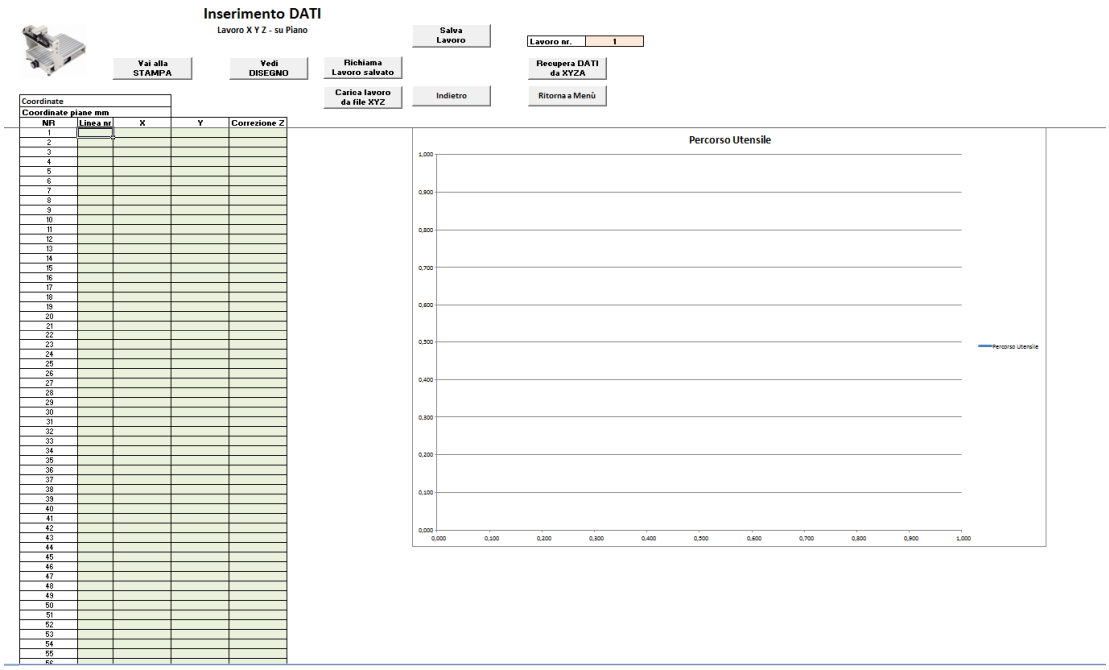
Ritorna

Codice	Descrizione
M0	Interruzione programma
M1	Interruzione programma opzionale
M2	Fine programma
M3	Rotazione del mandrino in senso orario
M4	Rotazione del mandrino in senso antiorario
M5	Interruzione rotazione mandrino
M6	Cambiamento di strumento (per due macro)
M7	Accensione nebbia refrigerante
M8	Accensione flusso refrigerante
M9	Tutti i refrigeranti spenti
M30	Fine programma e Rewind
M47	Ripetere programma dalla prima riga
M48	Abilitare l'override velocità e feed
M49	Disabilitare l'override velocità e feed
M98	Chiamata di subroutine
M99	Ritorno dalla subroutine/ripetere

After the configuration, you can proceed input the coordinates through the "Next" button or return to the previous menu by pressing the "Back" button.

By pressing "Next" button will appear the screen for coordinates input.

Mask "Data Entry"



In addition to the fields of input of the coordinates (in green), in the tab you find:
- go to print (to generate the G-code);
- see drawing (you will get a screen with the design work on);
- call saved job (to retrieve a previously saved work);
- upload work from XYZ file (to load a work from file TXT XYZ);
- save job (to save your work in the archive);
- recover data from XYZA (to retrieve a work set in section XYZA);

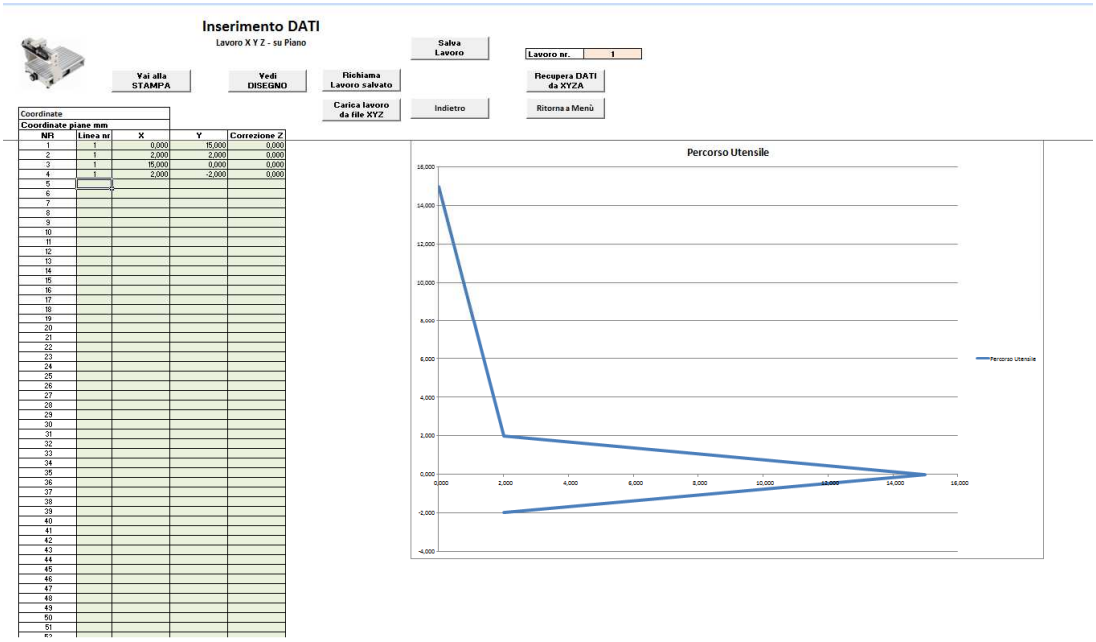
- back (to return to the previous screen);
- return to menu (to return to the menu).

Input of coordinates.
In the screen we find the following columns:
- line no.
- X
- Y
- correction Z

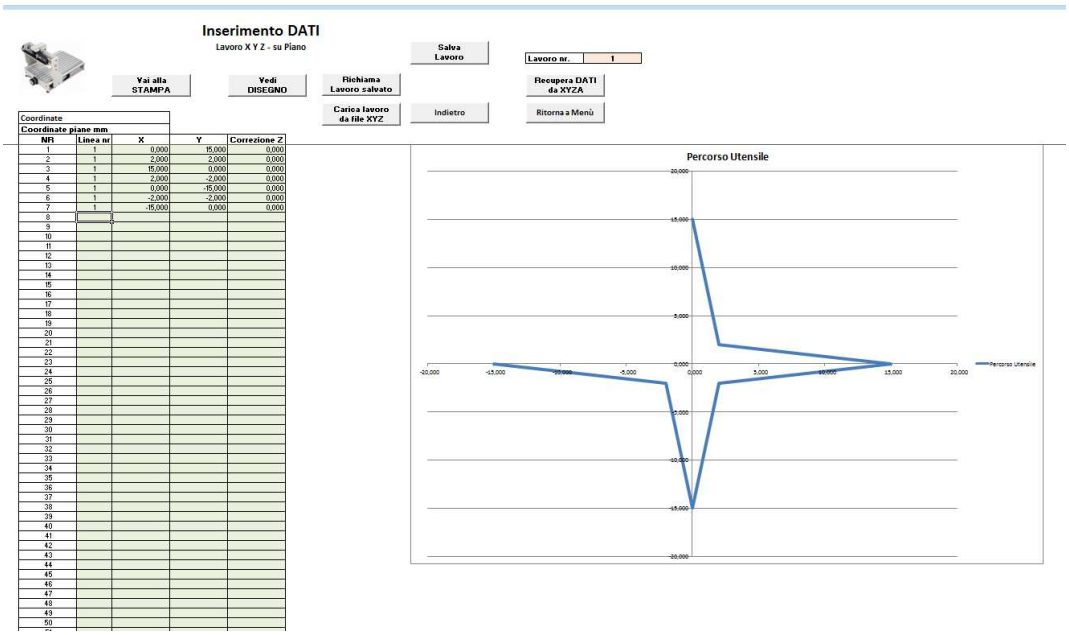
In the columns “X” and “Y” will be entered the coordinates of the various points to engrave. All points which bear the same numbering as the “Line No.” will be inscribed with a line which, starting from the first point, connects all points with the same “Line No.” until the last point with the same numbering. Change the value of “Line No.” causes an interruption of the incision up to the next point.
In the case in which each line of coordinates had a different “Line No.”, the tool engraves a single point for each coordinate set and will move to the next step while maintaining a free height from the work table as set in the initial parameters.
WARNING!
Be careful when entering the value “correction Z”.
For a work of flat engraving, without variations in height of the incision itself, the value “Correction Z” must be set to “0” (zero).
The column “Correction Z” NOT directly determines the depth of the incision (already set up earlier), but only serves to rectify, but on occasions it was the need, the depth of the incision in certain parts or in all of the work.
For example, we could draw a star by setting a “correction Z” = -1 in the middle.
The software will calculate the linear interpolation between the previous point or rather, the tip with “Correction Z” = 0 and the inner part with “Correction Z” = -1, then you will find that the central part of the star will have a depth of engraving higher than at the tips equal to the value of the “correction Z” set.
After the input of coordinate data, it will be able to show the design, save the work and register files into G code (Go to print).

Data entry: details of example.

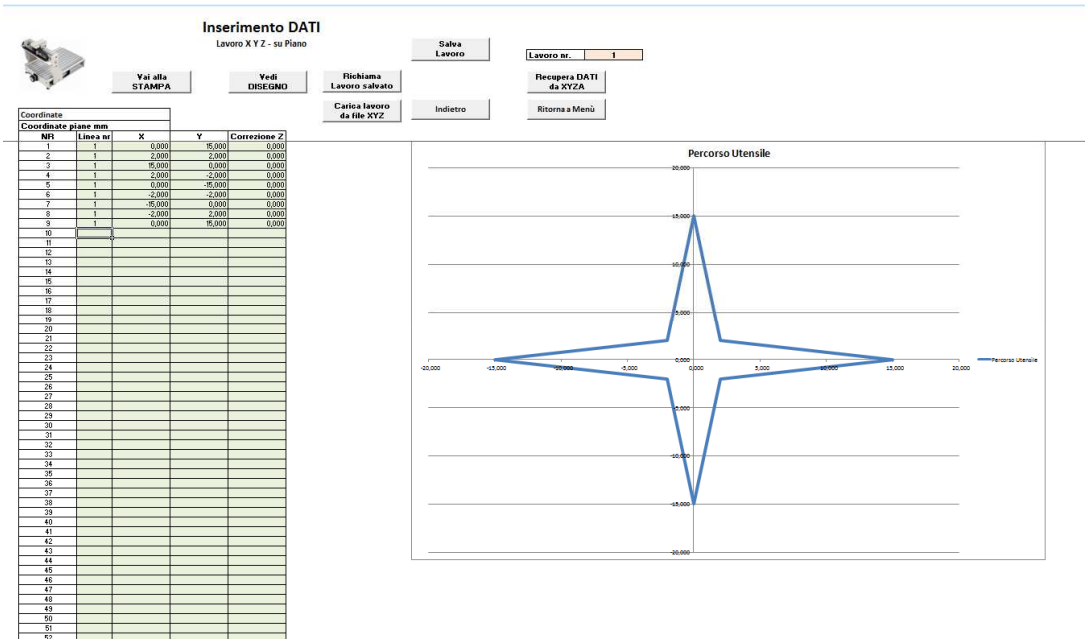
Picture 01



Picture 02

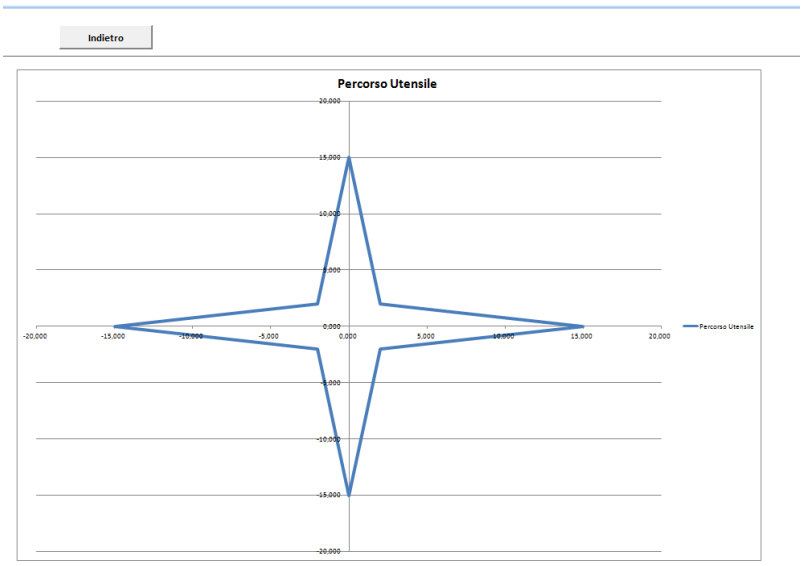


Picture 03



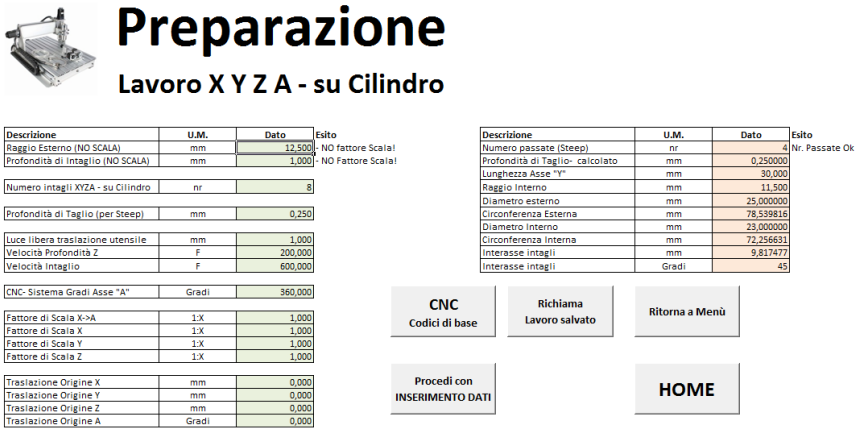
NR	Linea nr	X	Y	Correzione Z
1	1	0,000	15,000	0,000
2	1	2,000	2,000	0,000
3	1	15,000	0,000	0,000
4	1	2,000	-2,000	0,000
5	1	0,000	-15,000	0,000
6	1	-2,000	-2,000	0,000
7	1	-15,000	0,000	0,000
8	1	-2,000	2,000	0,000
9	1	0,000	15,000	0,000

See drawing: details of example.
Picture 01



4 Axes XYZA

The button brings the program in the section of work “on cylinder”, or else, it allows to engrave a surface using the 4 axes of the machine tool: Plan X “Transversal” Plan Y “Longitudinal” Plan Z “Height / depth” and axis A “Rotation of 4th Axis – Spindle”.



Descrizione	U.M.	Dato	Esito
Raggio Esterno (NO SCALA)	mm	12,500	- NO fattore Scala!
Profondità di Intaglio (NO SCALA)	mm	1,000	- NO Fattore Scala!
Numero intagli XYZA - su Cilindro	nr	8	
Profondità di Taglio (per Steep)	mm	0,250	
Luce libera traslazione utensile	mm	1,000	
Velocità Profondità Z	F	200,000	
Velocità Intaglio	F	600,000	
CNC- Sistema Gradi Asse "A"	Gradi	360,000	
Fattore di Scala X->A	1:X	1,000	
Fattore di Scala X	1:X	1,000	
Fattore di Scala Y	1:X	1,000	
Fattore di Scala Z	1:X	1,000	
Traslazione Origine X	mm	0,000	
Traslazione Origine Y	mm	0,000	
Traslazione Origine Z	mm	0,000	
Traslazione Origine A	Gradi	0,000	

Descrizione	U.M.	Dato	Esito
Numero passate (Steep)	nr	4	Nr. Passate Ok
Profondità di Taglio- calcolato	mm	0,250000	
Lunghezza Asse "Y"	mm	30,000	
Raggio Interno	mm	11,500	
Diametro esterno	mm	25,000000	
Circonferenza Esterna	mm	78,539816	
Diametro Interno	mm	23,000000	
Circonferenza Interna	mm	72,256631	
Interasse intagli	mm	9,817477	
Interasse intagli	Gradi	45	

In addition to the input fields (input fields in green), tab buttons are:

- CNC Basic codes
- Continue with data input (entering coordinates)
- Recover saved work
- Back to the menu (brings you to the previous menu)
- Home (brings you to the main menu to save and close the program)

First set the initial parameters of the work to be performed:

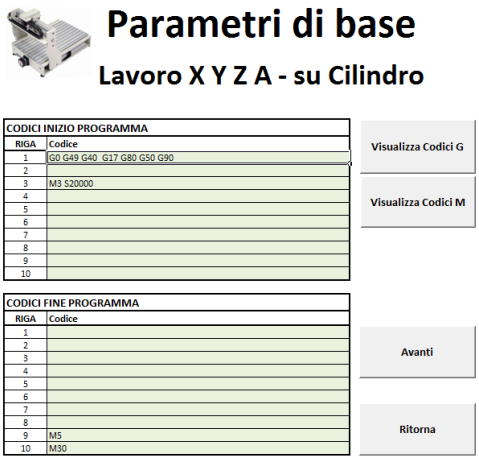
1. outer radius of the piece to be worked;
   a. if we set the depth of incision to a measure greater than the outer radius of the workpiece the program will warn us to pay attention with the written "Check the engraving depth", this to avoid engraving over the center of the piece.
   WARNING: the program proceeds to the incision anyway!
2. The notch depth (how deep the incision should be, compared to the outer radius of the workpiece).
3. The number of carvings to be performed on the circular surface, or else, how many times the drawing is repeated on the circular surface.
   a. Note: the software automatically calculates the rotation of the axis “A” for each drawing to engrave according to the number of incisions to do on the workpiece.
   b. Note: there are no limitations at the number of incisions, but pay attention to the possible overlap of the engraved drawings.
4. The notch depth for each pass (step).
5. The clearance between the tool and the workpiece;
   a. the clearance refers to the space between the tool tip and the surface to engrave, that is generated during the traslation of the tool between one side and the other of the workpiece (such as when you stop the incision of a line and the tool moves to the next line to be engraved).
6. The speed feed of axis “Z” (drilling speed on axis “Z”).
7. Feed speed of notch (traslation speed of the axes “X” and “Y”).
8. The value of the degrees of complete rotation (round angle) of the axis “A” set on the machine tool.
   a. Note: the system can be at 360° or at 400°; verify your machine tool.
9. The scale factors of the axes “X→A”, “X”, “Y” and “Z”.
   a. The scale factors are equal to “1” when there is no reduction or enlargement.
   b. The axis “X→A” is the axis “X” (plan) of the drawing converted into axis “A” (of rotation)

10. The traslations values with respect to the origins of the “point zero”.
    a. Leave the values at “zero” for not having traslations.

At this point, the software calculates mainly the number of “passes” (steps) required to the incision (max 25 steps) and adjusts its value in order to reach accurately the “notch depth”.
In case they are needed more than 25 steps, the program will give an “Error:
solution 1): slightly increase the notch depth for each pass (step);
solution 2): reduce the total depth of carving.

Button “CNC Basic Codes”

Then you go to the configuration of the CNC: click on the button “CNC Basic Codes”.



You find:

1. Initial settings:
   consists of 10 lines where you will enter the initial configuration commands of the machine (input fields in green).
   Example:
   on the first line write: *G0 G49 G40 G17 G80 G50 G90*
   Where:
   G0 = rapid positioning;
   G49 = deletes tool length offsets;
   G40 = deletes the compensation of the radius of the blade;
   G17 = selects the work plane axes “XY”
   G80 = deletion of the movement (including the programmed cycles)
   G50 = resets all the scale factors to “1”
   G90 = sets absolute distance mode.

   In the second line we can write: G20 (Unit inch) or G21 (Unit mm).
   Pay attention that some software, even with G21, distort the values of millimeters.

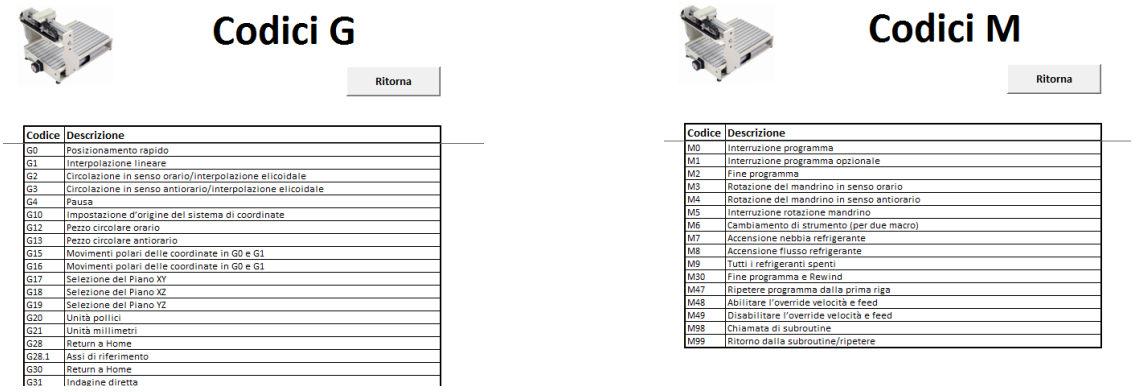
   Other primary command: M3 followed by the command S “Spindle Speed”.
   Sample line:
   M3 S20000
   M3 indicates ignition and spindle rotation (engraver) clockwise;
   S20000 indicates the spindle speed to 20,000 rpm.

2. Final settings:
   is composed of 10 lines (input fields in green) where you will enter the final configuration commands of the machine.
   Example:

in the penultimate line write: *M5*
in the last line write: *M30*
Where:
M5 = turns off the spindle;
M30 = indicates the “End of program”.

The setting is essential before any operation since the machine tool may have configurations, loaded with previously codes, do not comply.
The informations provided above are simply an example.
The user is responsible to properly configure the software to suit his needs!

From the current tab, you can call through the appropriate buttons, a short illustrative list of codes “G” and “M”.



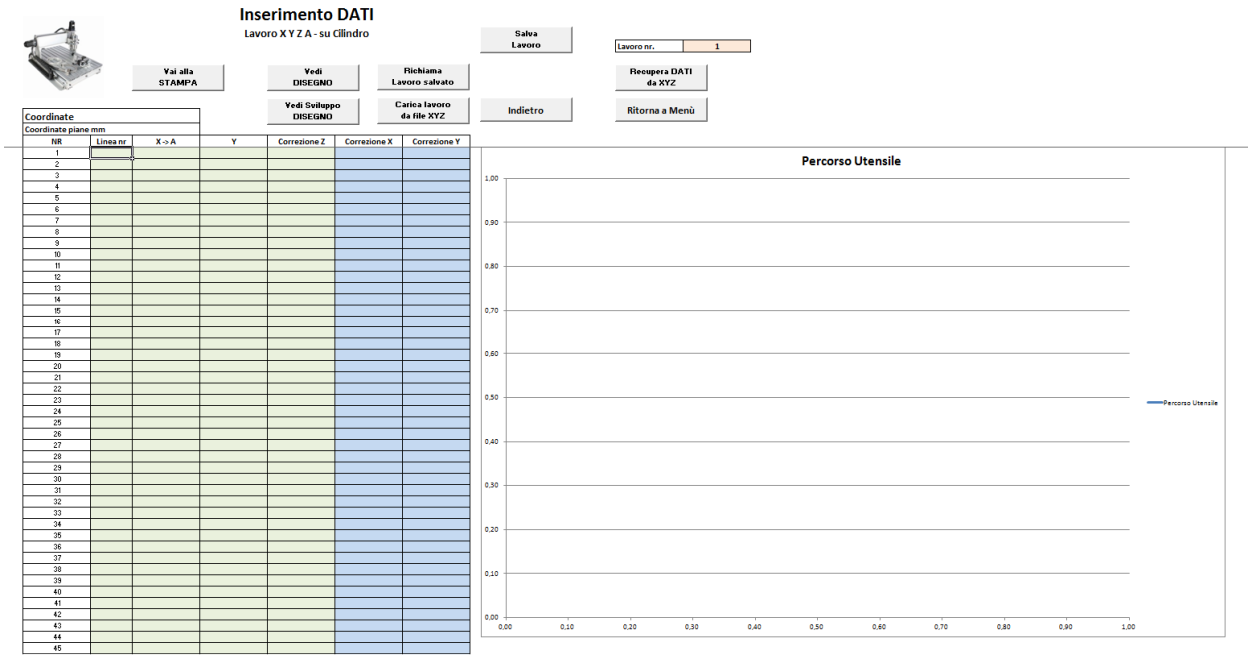
Codice	Descrizione
G0	Posizionamento rapido
G1	Interpolazione lineare
G2	Circolazione in senso orario/interpolazione elicoidale
G3	Circolazione in senso antiorario/interpolazione elicoidale
G4	Pausa
G10	Impostazione d'origine del sistema di coordinate
G12	Pezzo circolare orario
G13	Pezzo circolare antiorario
G15	Movimenti polari delle coordinate in G0 e G1
G16	Movimenti polari delle coordinate in G0 e G1
G17	Selezione del Piano XY
G18	Selezione del Piano XZ
G19	Selezione del Piano YZ
G20	Unità pollici
G21	Unità millimetri
G28	Return a Home
G28.1	Assi di riferimento
G30	Return a Home
G31	Indagine diretta

Codice	Descrizione
M0	Interruzione programma
M1	Interruzione programma opzionale
M2	Fine programma
M3	Rotazione del mandrino in senso orario
M4	Rotazione del mandrino in senso antiorario
M5	Interruzione rotazione mandrino
M6	Cambiamento di strumento (per due macro)
M7	Accensione nebbia refrigerante
M8	Accensione flusso refrigerante
M9	Tutti i refrigeranti spenti
M30	Fine programma e Rewind
M47	Ripetere programma dalla prima riga
M48	Abilitare l'override velocità e feed
M49	Disabilitare l'override velocità e feed
M98	Chiamata di subroutine
M99	Ritorno dalla subroutine/ripetere

After the configuration, you can proceed input the coordinates through the “Next” button or return to the previous menu by pressing the “Back” button.

By pressing “Next” button will appear the screen for coordinates input.

Mask “Data Entry”



In addition to the fields of input of the coordinates (in green), in the tab you find:

- go to print (to generate the G-code);
- see drawing (you will get a screen with the design work on);
- see development design (this will open a screen with the drawing of the work on, in addition with the next pattern and the last in the queue, including the related coordinates);
- call saved job (to retrieve a previously saved work);
- upload work from XYZ file (to load a work from fie TXT XYZ);
- save job (to save your work in the archive);
- recover data from XYZ (to retrieve a work set in section XYZ);
- back (to return to the previous screen);
- return to menu (to return to the menu).

Input of coordinates.

In the screen we find the following columns:

- line no.
- X→A
- Y
- correction Z
- correction X
- correction Y

NOTE: the coordinate value "X → A" will be converted into the rotation of the axis "A" dividing value on the arc of the circumference of the cylinder to be engraved (it is like wrapping a cylinder with a sheet containing the design to be engraved). For circumference refers to the "outer radius" of the cylinder.

In the column "X → A" and "Y" will be entered the coordinates of the various points to engrave. All points which bear the same numbering as the "Line No." will be inscribed with a line which, starting from the first point, connects all points with the same "Line No." until the last point with the same numbering. Change the value of "Line No." causes an interruption of the incision up to the next point.

In the case in which each line of coordinates had a different "Line No.", the tool engraves a single point for each coordinate set and will move to the next step while maintaining a free height from the work table as set in the initial parameters.

WARNING!

Be careful when entering the value "Correction Z".

To perform a job of engraving without variations of penetration, the value "Correction Z" must be set to "0" (zero).

The column "Correction Z" NOT directly determines the depth of the incision (already set up earlier), but only serves to rectify, but on occasions it was the need, the depth of the incision in certain parts or in all of the work.

For example, we could draw a star by setting a "Correction Z" = -1 in the middle.

The software will calculate the linear interpolation between the previous point or rather, the tip with "Correction Z" = 0 and the inner part with "Correction Z" = -1, then you will find that the central part of the star will have a depth of engraving higher than at the tips equal to the value of the "Correction Z" set.

The "Correction X" causes the traslation of the axis X for the indicated value (not axis A!).

The "Correction Y" causes the traslation of the axis X for the indicated value.

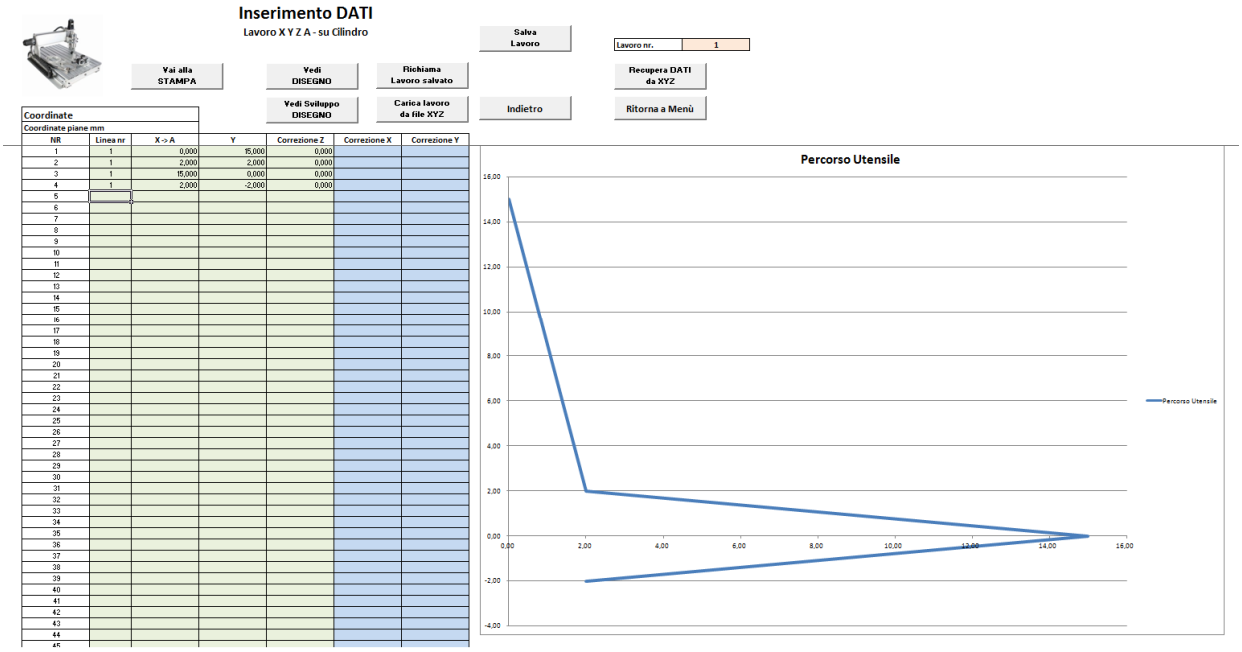
NOTE: Values of "correction" get their full value in the point "correct" while remain unchanged, if not "correct" the previous point and the next. The cut line will be interpolated in the correct coordinates.

The “Correction X” and “Correction Y” are suitable when you want to change the tool path for using special mills or for realizing particular engrave.

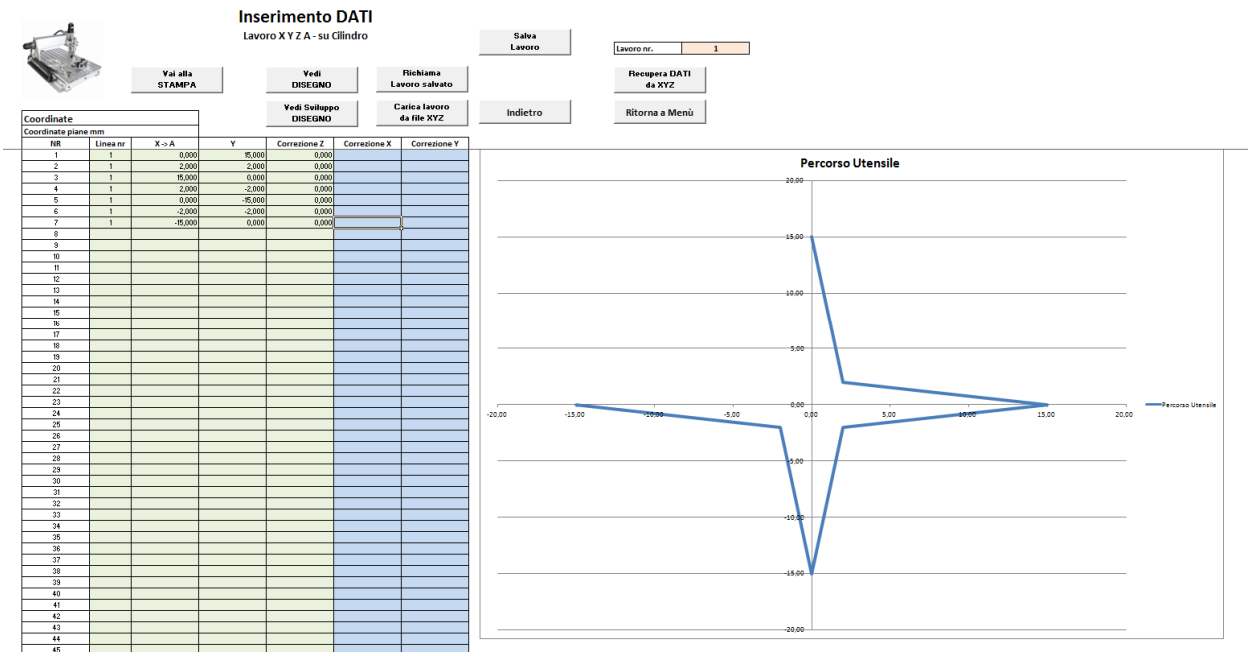
After the input of coordinate’s data, it will be able to show the design, save the work and register files into G code (Go to print).

Data entry: details of example.

Picture 01



Picture 02



Picture 03

Inserimento DATI

Lavoro X Y Z A - su Cilindro

Salva Lavoro

Lavoro nr. 1

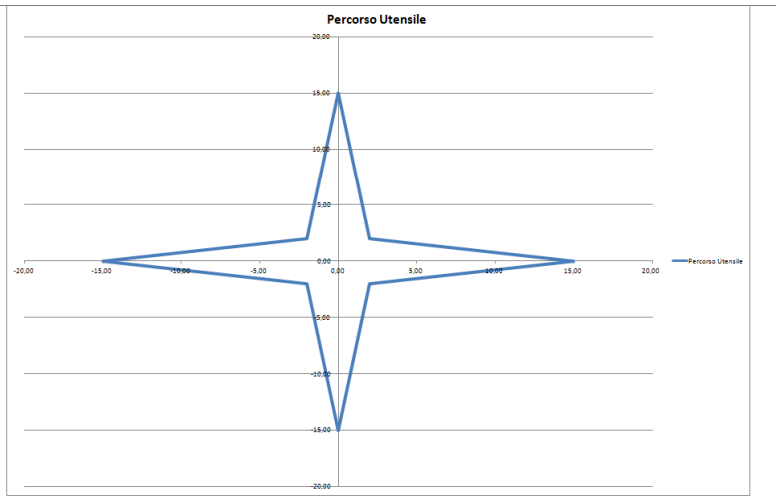
Vai alla STAMPA | Vedi DISEGNO | Richiama Lavoro salvato | Recupera DATI da XYZ

Vedi Sviluppo DISEGNO | Carica lavoro da file XYZ | Indietro | Ritorna a Menù

Coordinate

Coordinate piane mm

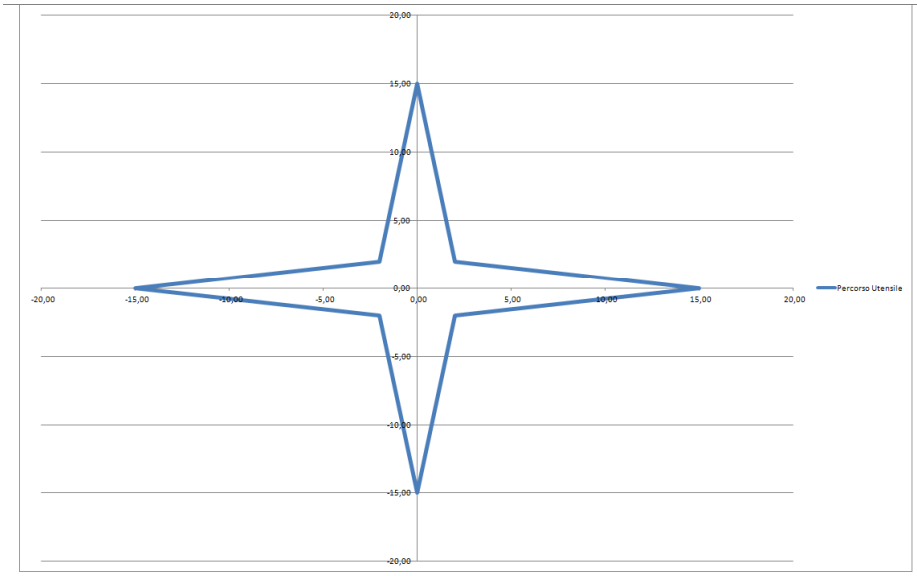
NR	Linea nr	X -> A	Y	Correzione Z	Correzione X	Correzione Y
1	1	0,000	15,000	0,000		
2	1	2,000	2,000	0,000		
3	1	15,000	0,000	0,000		
4	1	2,000	-2,000	0,000		
5	1	0,000	-15,000	0,000		
6	1	-2,000	-2,000	0,000		
7	1	-15,000	0,000	0,000		
8	1	-2,000	2,000	0,000		
9	1	0,000	15,000	0,000		
10						
11						
12						
13						
14						
15						
16						
17						
18						
19						
20						
21						
22						
23						
24						
25						
26						
27						
28						
29						
30						
31						
32						
33						
34						
35						
36						
37						
38						
39						
40						
41						
42						
43						
44						
45						



See drawing: details example.

Picture 01

Indietro



Save work.

By pressing the button “Save work” the following data will be saved in the inner archive of the software:

- preparation data;
- basic codes (codes beginning and end of program);
- all coordinates related to the work.

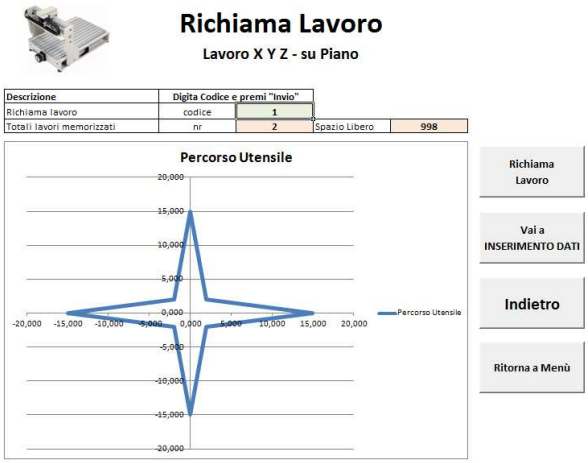
Every work recalled and changed, when saved anew, will get a new “work code”.
It is possible to save up to 1,000 works on plane surfaces (“XYZ”) and up to 1,000 works on circular surfaces (“XYZA”). The archives of the works can be exported on an external archive file.

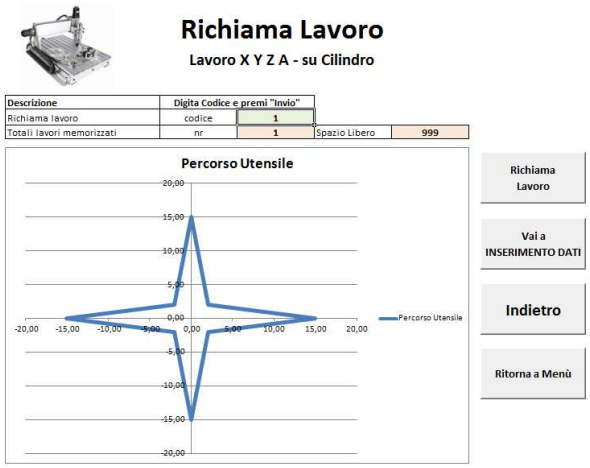
Recall saved work.

By pressing the button “Recall saved work” the following data will be retrieved from the inner archive of the software:

- preparation data;
- basic codes (codes beginning and end of program);
- all coordinates related to the work.

The procedures are the same for both the XYZ format and the XYZA format:





Procedure:

- type the code of the work previously saved and press enter;
- click on “Recall work”;
- the work will be directly transferred to the related software.

It is possible to change the recalled work, make a new save or regenerate the file in G-code.

Load XYZ file.

The recallable files have to be in text format and have the following “tabulation”:

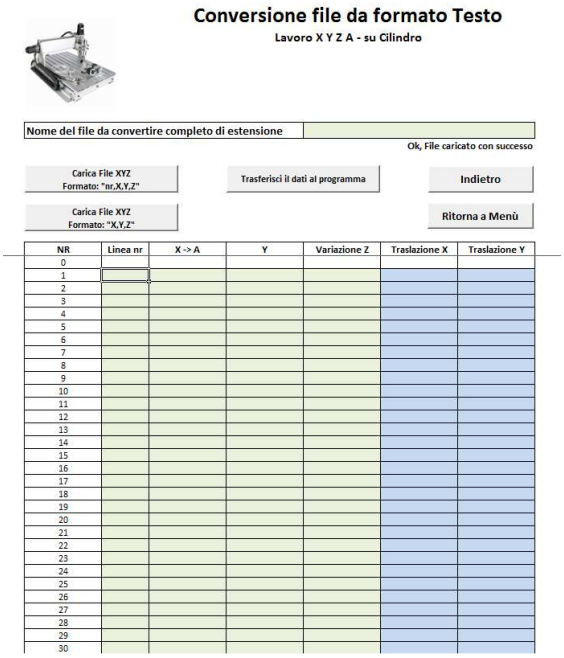
- format: “no.,X,Y,Z”:
    - No = number of line or point;
    - X = coordinated “X”
    - Y = coordinated “Y”
    - Z = coordinated “Z”

  The separator must be a “,” (comma).

- format: “X,Y,Z”:
    - X = coordinated “X”
    - Y = coordinated “Y”
    - Z = coordinated “Z”

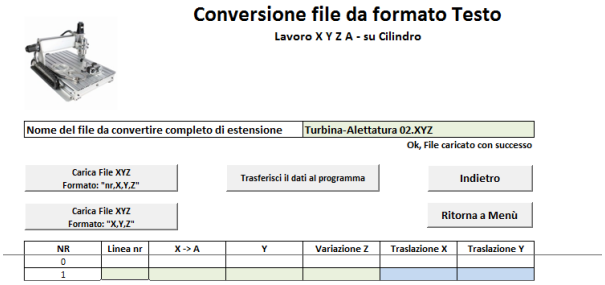
  The separator must be a “,” (comma).
  In the case of file with format “X, Y, Z”, therefore without the “Line no.”, the user must input the “Line no.” manually, as without a number, the coordinates of the line will be not considered.

WARNING!
Pay a lot of attention at the value of the coordinates “Z”, transferred to “Correction Z” (it is suggested to read all the coordinates and verify the content, if necessary bringing it to zero). Many CAD designers play the drawings “at view”, or rather, by tracing the designs on the “axis X” and “axis Y” (like they were drawing on a 2D plan) without caring too much on the “axis Z” (axis for the 3D): so, for experience, you could find works of “axis Z” totally busted (for instance share 10,000 or -10,000). These values of the “Z”, converted by the software into G-code, will generate a series of instruction to the machine tool with vary bad consequences!
It is underlined that the column “Correction Z” DOES NOT directly determines the depth of the incision (already set earlier), but is used exclusively for rectify, if it is necessary, the incision’s depth in certain parts or in all the work.

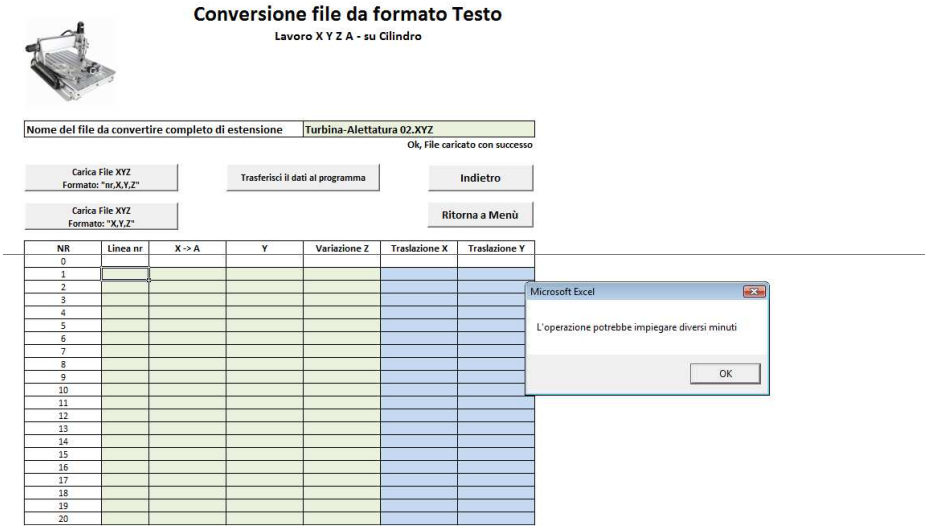


The file to be converted must be placed in the folder “File XYZ” generated by the software when it was installed.

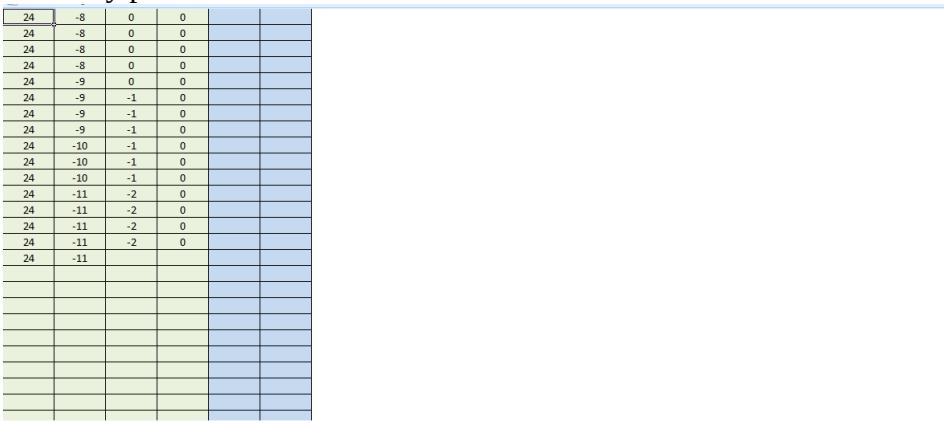
Input the “file name” with his extension (i.e.: TXT, XYZ) and press enter.



Click on the appropriate button (Load XYZ file) and wait for the execution.



Recovery phase



24	-8	0	0		
24	-8	0	0		
24	-8	0	0		
24	-8	0	0		
24	-9	0	0		
24	-9	-1	0		
24	-9	-1	0		
24	-9	-1	0		
24	-10	-1	0		
24	-10	-1	0		
24	-10	-1	0		
24	-11	-2	0		
24	-11	-2	0		
24	-11	-2	0		
24	-11	-2	0		
24	-11				

Recovery phase ended.

Conversione file da formato Testo

Lavoro X Y Z A - su Cilindro

Nome del file da convertire completo di estensione	Turbina-Alettatura 02.XYZ

Ok, File caricato con successo

Carica File XYZ Formato: "nr,X,Y,Z" | Trasferisci il dati al programma | Indietro

Carica File XYZ Formato: "X,Y,Z" | Ritorna a Menù

NR	Linea nr	X -> A	Y	Variazione Z	Traslazione X	Traslazione Y
31	24	-13	-6	0		
32	24	-12	-7	0		
33	24	-12	-7	0		
34	24	-12	-7	0		
35	24	-12	-8	0		
36	24	-12	-8	0		
37	24	-12	-8	0		
38	24	-12	-9	0		
39	24	-12	-9	0		
40	24	-11	-9	0		
41	24	-11	-10	0		
42	24	-11	-10	0		
43	24	-10	-11	0		
44	24	-10	-11	0		
45	24	-9	-11	0		
46	24	-9	-12	0		
47	24	-8	-12	0		
48	24	-8	-12	0		
49	24	-7	-13	0		
50	24	-7	-13	0		
51	24	-6	-13	0		
52	24	-6	-14	0		
53	24	-5	-14	0		
54	24	-4	-14	0		
55	24	-4	-15	0		
56	24	-3	-15	0		
57	24	-3	-15	0		
58	24	-2	-16	0		
59	24	-1	-16	0		
60	24	-1	-16	0		
61	24	0	-17	0		

Click “Transfer data to the program” to import them in the work mask.
The software will return automatically to “Data entry”.

Transfer executed.

Inserimento DATI

Lavoro X Y Z A - su Cilindro

Salva Lavoro | Lavoro nr. 2

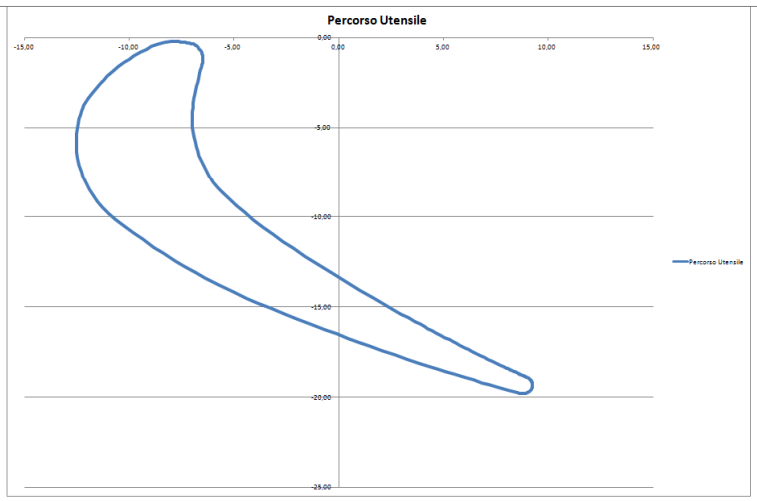
Vai alla STAMPA | Vedi DISEGNO | Richiama Lavoro salvato | Recupera DATI da XYZ

Vedi Sviluppo DISEGNO | Carica lavoro da file XYZ | Indietro | Ritorna a Menù

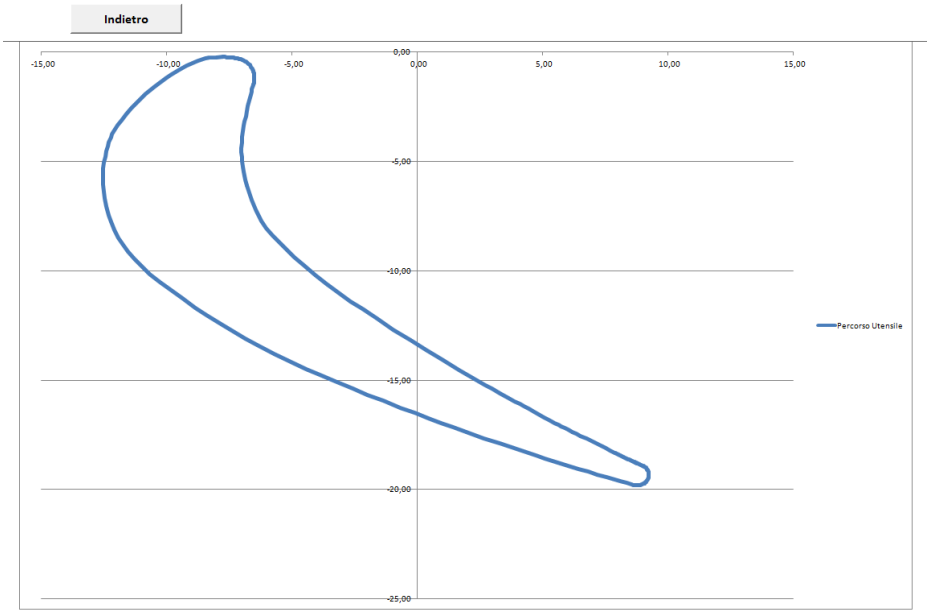
Coordinate

Coordinate piane mm

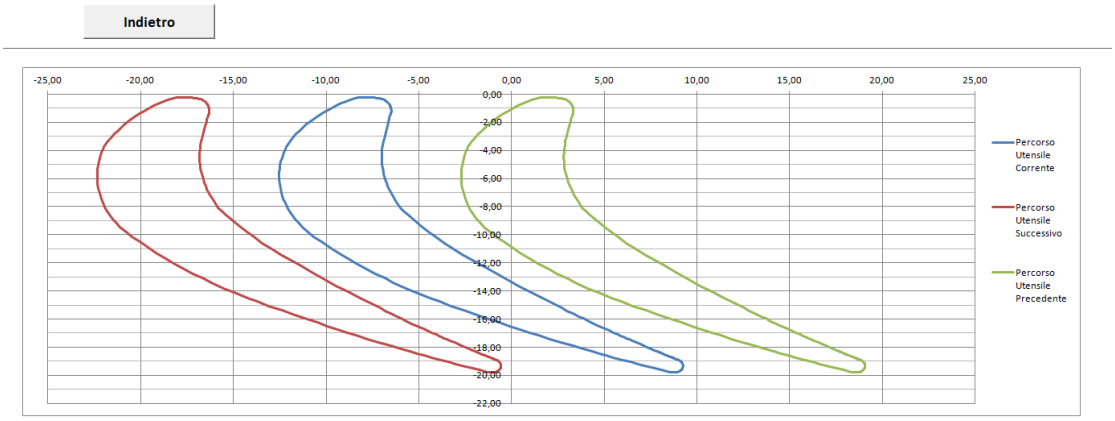
NR	Linea nr	X -> A	Y	Correzione Z	Correzione X	Correzione Y
1	24	-7,761	-0,226	0,000		
2	24	-8,034	-0,233	0,000		
3	24	-8,282	-0,261	0,000		
4	24	-8,447	-0,292	0,000		
5	24	-8,733	-0,384	0,000		
6	24	-8,955	-0,501	0,000		
7	24	-9,193	-0,637	0,000		
8	24	-9,467	-0,806	0,000		
9	24	-9,784	-1,023	0,000		
10	24	-10,004	-1,186	0,000		
11	24	-10,249	-1,383	0,000		
12	24	-10,514	-1,615	0,000		
13	24	-10,816	-1,905	0,000		
14	24	-11,046	-2,147	0,000		
15	24	-11,223	-2,348	0,000		
16	24	-11,407	-2,573	0,000		
17	24	-11,605	-2,835	0,000		
18	24	-11,796	-3,110	0,000		
19	24	-11,951	-3,352	0,000		
20	24	-12,155	-3,744	0,000		
21	24	-12,204	-3,879	0,000		
22	24	-12,281	-4,118	0,000		
23	24	-12,330	-4,290	0,000		
24	24	-12,384	-4,508	0,000		
25	24	-12,436	-4,763	0,000		
26	24	-12,480	-5,047	0,000		
27	24	-12,507	-5,289	0,000		
28	24	-12,522	-5,495	0,000		
29	24	-12,530	-5,679	0,000		
30	24	-12,526	-6,070	0,000		
31	24	-12,503	-6,398	0,000		
32	24	-12,465	-6,702	0,000		
33	24	-12,397	-7,074	0,000		
34	24	-12,303	-7,450	0,000		
35	24	-12,201	-7,772	0,000		
36	24	-12,064	-8,126	0,000		
37	24	-11,896	-8,490	0,000		
38	24	-11,680	-8,881	0,000		
39	24	-11,517	-9,135	0,000		
40	24	-11,284	-9,457	0,000		
41	24	-10,965	-9,834	0,000		
42	24	-10,664	-10,138	0,000		
43	24	-10,234	-10,517	0,000		
44	24	-9,801	-10,893	0,000		
45	24	-9,346	-11,273	0,000		



Picture “See drawing”.



Picture “See drawing development”.



Service

In the service tab, accessible from the 2° level menu, you will find the following buttons:

- Delete archive XYZ
- Delete archive XYZA
- Save archives
- Restore archives
- Show software's codes
- Return

Delete archive XYZ:

it deletes definitively, all data from the archive works XYZ.

Delete archive XYZA:

it deletes definitively, all data from the archive works XYZA

Save archives:

exports data of the works saved from the archives "XYZ" and "XYZA" into the external file named "CNC mdl – Engraver Ver. 01a – Archive", placed in the folder "Archives".
WARNING! All data present in the external archive will be overwritten.

Restore archives:

recovers the works data, previously exported, from the file "CNC mdl – Engraver Ver. 01a – Archive", placed in the folder "Archives", and it copies them in the archives of the software used.
WARNING! All data present in the external archive will be overwritten.

Show software's codes:

shows the installation code and the activation code of the software and checks the registration.

Return:

it returns to the 2° level menu.

NOTE: before performing the saving in the external archive, please generate a copy of the archive itself: the procedure overwrites all the data present in it.

EXIT THE PROGRAM

When all the operations are finished, in order to not lose data, it is necessary perform the procedure using the main menu.

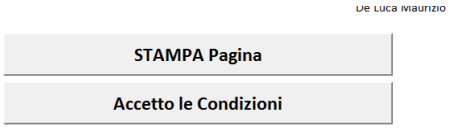
When the “macro” safety screen appears, please remind to confirm the saving.

Enclosure A
Generate activation code

Step 1

Step 2

Step 3



Step 4

CNC - mdl

Engraver Rel. 01a

Codice installazione	[redacted]

STAMPA Pagina

Salva e Chiudi programma

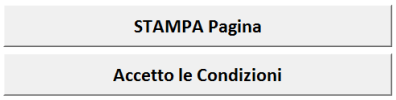
Enclosure B
Software registration

Step 1

Step 2

Step 3

Step 4



Step 5

CNC - mdl

Engraver Rel. 01a

Codice installazione	[redacted]
Codice di sblocco	

Digita il codice e clicca sul pulsante

Step 6

CNC - mdl

Engraver Rel. 01a

Codice installazione	[redacted]
Codice di sblocco	560636412346576

Digita il codice e clicca sul pulsante

Step 7

CNC - mdl

Engraver Rel. 01a

Codice installazione	[redacted]
Codice di sblocco	560.636.412.346.576

Digita il codice e clicca sul pulsante

Step 8

CNC - mdl

Engraver Rel. 01a

Codice di sblocco	560.636.412.346.576

Versione Registrata

Clicca sul pulsante per proseguire

Ridigita il Codice di Sblocco

Step 9

CNC - mdl

Engraver Rel. 01a

Codice installazione	[redacted]
Codice di sblocco	560.636.412.346.576

Leggere attentamente prima dell'utilizzo

Step 10

STAMPA Pagina

Accetto le Condizioni

Step 11

CNC - mdl

Engraver Rel. 01a

Registra

Versione Registrata

Clicca sul pulsante per proseguire

Salva e Chiudi programma