

# CNC Routers

## XMD series

User's Manual

ver. 1.08



Manual last updated in April 2013

## Table of Contents

INTRODUCTION.....	5
TECH SPECS AND ACCESSORIES .....	6
BASIC TERMS AND PARAMTERS.....	8
THE WIRED REMOTE CONTROL.....	10
DRAWINGS' PREPARATION BASICS.....	13
Machine home position vs. the project's starting position.....	13
2D Projects.....	13
DXF files.....	14
Drilling.....	14
Engraving, single-line font.....	14
Gcod Postprocesor.....	17
MegaCut (2D) software .....	19
Machine start-up procedure.....	19
Software settings.....	19
Language version.....	19
Control Window - enlarged.....	20
Main window.....	21
Opening a drawing.....	22
Simulation.....	22
Configuration.....	23
Import tab.....	23
Sequence tab.....	25
Directions tab .....	26
CNC tab.....	26
Drilling tab.....	28
Ploter tab.....	29
General tab.....	30

Tool measurement.....	30
Automatic tool length measurement.....	31
Semi-automatic tool length measurement.....	32
Cutting.....	33
Multi-layer drawings.....	33
Lining – cutting out the insides of shapes.....	37
Manual Control window.....	38
Homing process.....	40
Project's starting location.....	40
Mote to and Move by.....	41
Network connection (ethernet).....	41
Router calibration.....	43
MegaCut3D software.....	44
Machine start-up procedure.....	44
First run - activation.....	44
Software settings.....	45
Language version.....	45
Control Window - enlarged.....	46
Main Window.....	47
Typical tooling sequence.....	48
Loading the 3D model.....	49
Loading a bmp file.....	50
Material.....	51
Location of the model in the material.....	52
Location of the model in the material vs. bmp heights.....	53
Tools library.....	54
Operations.....	55
Output files.....	58
Tool measurement.....	59
Automatic tool length measurement.....	61
Semi-automatic tool length measurement.....	62

Cutting.....	63
Manual Control.....	64
Homing process.....	65
Project's starting location.....	66
Move to and Move by.....	66
Configuration.....	67
Network connection (ethernet).....	68
Router calibration.....	69
REQUIREMENTS AND LIMITATIONS .....	70
ALGORITHHYMS.....	71
Lining algorithms.....	71
Overlapping shapes algorithms.....	73
MAINTENANCE RECOMMENDATIONS .....	74

## INTRODUCTION

The XMD CNC routers are computer-controlled machines capable of tooling such materials as wood, plywood, PVC, XPS, EPS, dibond or „soft” metals such as aluminum, brass or copper. They are not suitable for working with e.g. steel, glass or stone.

Thanks to modern electronics, advanced software and cast iron frame they are suitable for both 2D and 3D projects.

All XMD routers come standard with automatic tool length measuring system, automatic homing process based on proximity switches and spindle-cooling system. Optional accessories include tool-cooling system and a wired remote control.

The proprietary software provided with the machine is user-friendly, yet at the same time fully featured, enabling full control over the working process. Along with the machine, two pieces of software are provided:

- [MegaCut](#) is suitable for all sorts of 2D projects where the customer has full control over the cutting or engraving process by preparing a suitable drawing and setting required parameters in MegaCut configuration.
- [MegaCut3D](#) is a full 3D application which generated the tool paths based on a three-dimensional model and the parameters the operator sets for each operation. Thanks to a user-friendly interface and advanced algorithms, working with MegaCut 3D requires only basic CNC knowledge and spatial imagination.

The working speed as well as spindle rotations can be modified during the cutting process. Additionally, the cutting process can be stopped at any time and then continued (e.g. when the cutting tool requires a clean up). Some of the important parameters, such as tool length or the project's starting point are saved in the electronic controller, thanks to which it is possible to restart the cutting process from any location on a 3D project. It is especially useful when e.g. a long tooling process is interrupted by a power cut-out. Each time the machine is turned on it automatically homes in relation to the proximity switches, thanks to which it is ready to continue a previously started job.

## TECH SPECS AND ACCESSORIES

Available models:

	<b>XMD32</b>	<b>XMD46</b>	<b>XMD48</b>
Working area	700 x 800 x 90 [mm]	1200 x 1800 x 90 [mm]	1250 x 2500 x 90 [mm]
Spindle power	2,2 [kW]	2,2 [kW]	3,0 [kW]
Spindle rotations	24 000 [obr/min]	24 000 [obr/min]	24 000 [obr/min]
Max. speed	5 000 [mm/min]	5 000 [mm/min]	5 000 [mm/min]
Weight	180 [kg]	380 [kg]	500 [kg]
External dimensions	110x130x145 [cm]	162,5x230x145 [cm]	170x299x145 [cm]

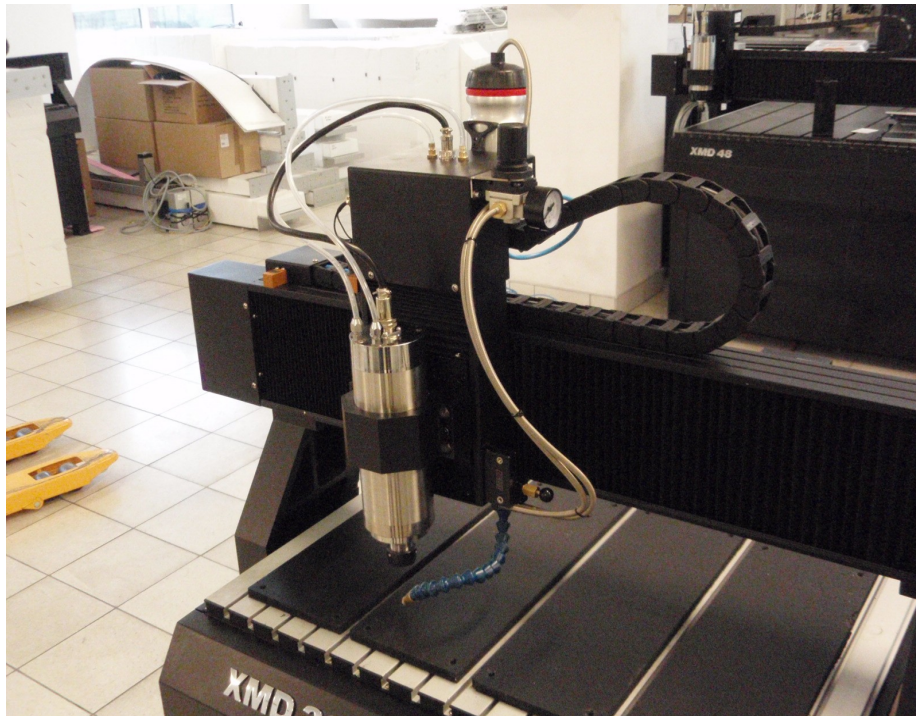
Working environment: 0 – 40° C

Standard package includes:

- CNC router
- electronic controller
- spindle cooling system
- automatic tool length measuring system
- control software - MegaCut and MegaCut 3D
- material holders
- two types of tool holders (ER20 type, diameter: 3 and 6 mm)
- sample tooling bits
- training at MegaPlot factory

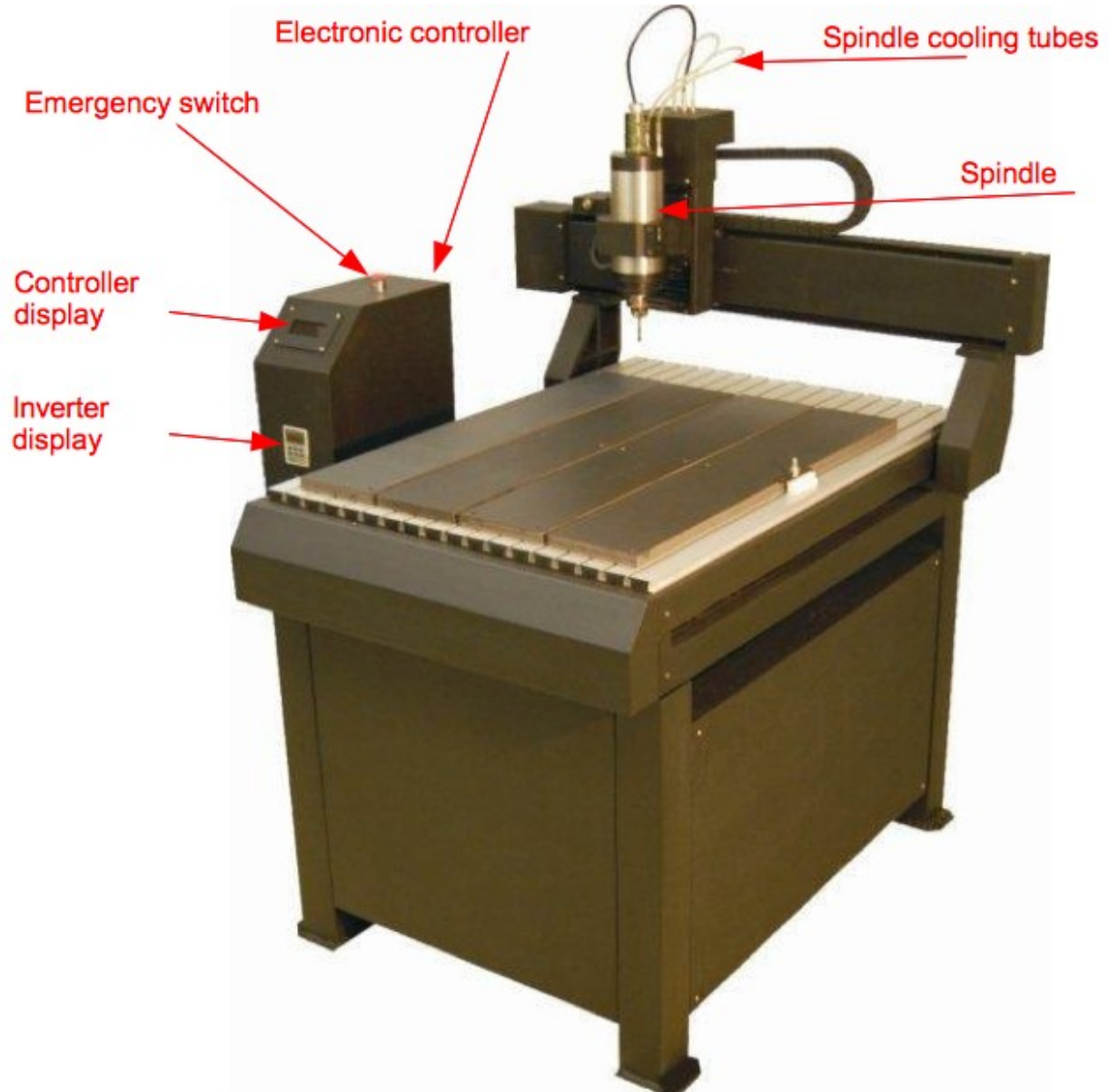
Optional equipment:

- wired remote control (a useful controlling device on a long cable, mostly used to set a starting position, move the machine in any axis or adjust the working parameters during the cutting process)
- tool cooling system, necessary when working with metals. Consists of a cooling liquid container, a regulator and connecting tubes. This tool cooling system needs to be connected to an air compressor which is not included.



## BASIC TERMS AND PARAMTERS

Before you begin working with the router, please familiarize yourself with the following basic terms:



The router needs to be connected to a single-phase ~220 line. It is of utmost importance that the power line is properly grounded. Before you plug the machine to the power source, please connect all the labeled plugs between the machine and the controller as well as the USB cable between the controller and the machine. All the plugs and cables are suitably labeled.

To turn the machine on, please turn the main switch clockwise. To turn the machine off, press this switch all the way down.



The controller's LCD screen shows various types of information re. the router. E.g. during the start up procedure, it shows the machine model. Once the controller connects properly to the PC, the display will show *Connected*. If there is a problem with the controller-PC connection, the display will read *Disconnected*.

Below the controller's main display, there is the spindle inverter's display. Both the display and the small keyboard below are used for diagnostics and servicing only and the customer should not be making any adjustments unless clearly instructed by a MegaPlot technician.

## THE WIRED REMOTE CONTROL

The wired remote control is an optional equipment. If you initially ordered the machine without one, you can order one any time in the future as the electronic controller always features a suitable connector. From the customer's perspective, this is a plug'n'play device.

The wire remote control can be used to manually control the machine in each axis, turn the spindle rotations on and off or set the project's starting position. It can also be used to adjust the spindle rotations and cutting speed during the cutting process. The remote control features a display which shows the current XYZ co-ordinates (in relation to the project's starting position, not the machine absolute co-ordinates – please see following sections for details), current spindle rotations as well as speed.



The remote control buttons' descriptions:



Turn the spindle rotations on



Turn the spindle rotations off



Decrease spindle rotations (3000-24000 rpm)



Increase spindle rotations (3000-24000 rpm)



Move the spindle up (Z axis)



Move the spindle down (Z axis)



Move the machine in X and Y axes



Decrease the speed incrementally



Increase the speed incrementally



Set the speed to 100mm/min



Set the speed to 5.000mm/min



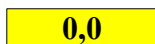
(press and hold)

Start the cutting process of the drawing currently opened in the software



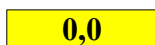
(short press)

Continue the cutting after it had been stopped



(press and hold)

Set the current location as the project's starting position



(short press)

Go back to a previously set project's starting position

**STOP**

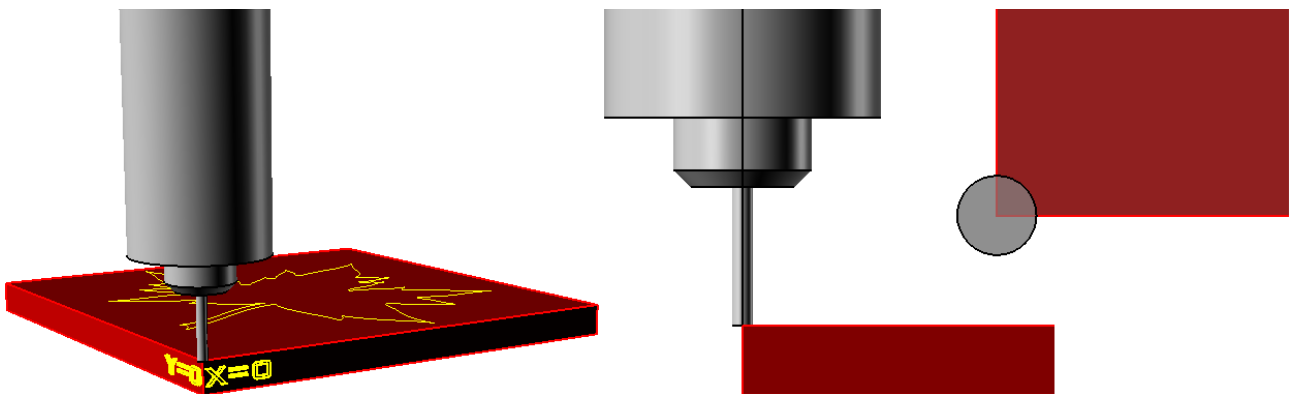
Stop the cutting process

## DRAWINGS' PREPARATION BASICS

### Machine home position vs. the project's starting position

Before you begin working with the machine, it is very important you understand the difference between the machine's home position and the current project's starting position.

Every time the machine is turned on, it automatically moves to its home position which is determined by the location of the proximity sensors. This is a constant location which the customer's cannot change. However, the customer sets the starting location for a given process anywhere on the working area of the machine. It corresponds to the lower left corner of the drawing.



The customer can set a number of different projects' starting locations anywhere on the machine. It is especially useful when the machine is used to produce a set of different elements and the material is always placed in the same spot on the machine. In this case, the customer can switch between the previously saved starting locations as needed.

If the machine is turned off and back on, the last saved project's starting location is remembered. Note the starting location is common to both MegaCut and MegaCut 3D, i.e. the same piece of material may be e.g. tooled in 3D first with MegaCut 3D and then MegaCut 2D can be used to engrave a pattern on one of the surfaces.

The project's starting location applies to X and Y only, and does not include the Z axis value.

### 2D Projects

2D projects can be saved in the following formats: PLT, DXF or NC (Gcod). In vector drawings (plf, dxf) it is important to avoid overlapping lines and shapes.

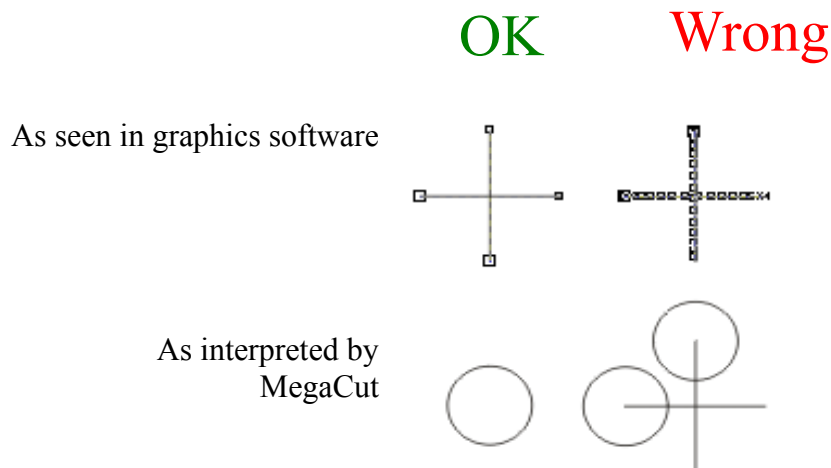
## DXF files

- all the elements of the drawing should be located on the same, zero layer
- do not use blocks, all the shapes should be drawn with polylines, curves, etc.
- text (letters) need to be converted to curves. If you use e.g. AutoCaD, you can use the Express tools for this: `express \ text \ explode \ explode text`.

## Drilling

The locations where you wish to drill a hole in a 2D drawings need to be marked with a cross. The cross needs to be drawn with two lines crossing at 90 degree angle (one vertical and one horizontal). Each of these two lines should consist of two nodes only. Only such crosses will be interpreted as the drilling locations by the software, and only if the „cross means point” setting is on in the Import tab in the software. Any other similar shapes prepared differently will be interpreted as standard curves to be cut.

The drawing below illustrates the way in which the software will interpret similar shapes which were prepared differently. The left one shows a proper drilling cross and in the software is visualized as a circle with a given diameter. The right one is not correct, as it was drawn as a Bezier curve and consists of a large number of nodes. As a result, the software interprets it and previews it as a set of two lines to be cut, not a single drilling location.



## Engraving, single-line font

When using an engraving font, please keep in mind that although it appears to be a single-line font, in fact it is built as any other type of font, but with two layers of overlapping lines. That is why all the letters will be engraved twice. To avoid this, ungroup the shape and delete the overlapping lines.

## 3D models

3D models can be prepared in one of the following formats: RAW, DXF 3D, BMP (grayscale).

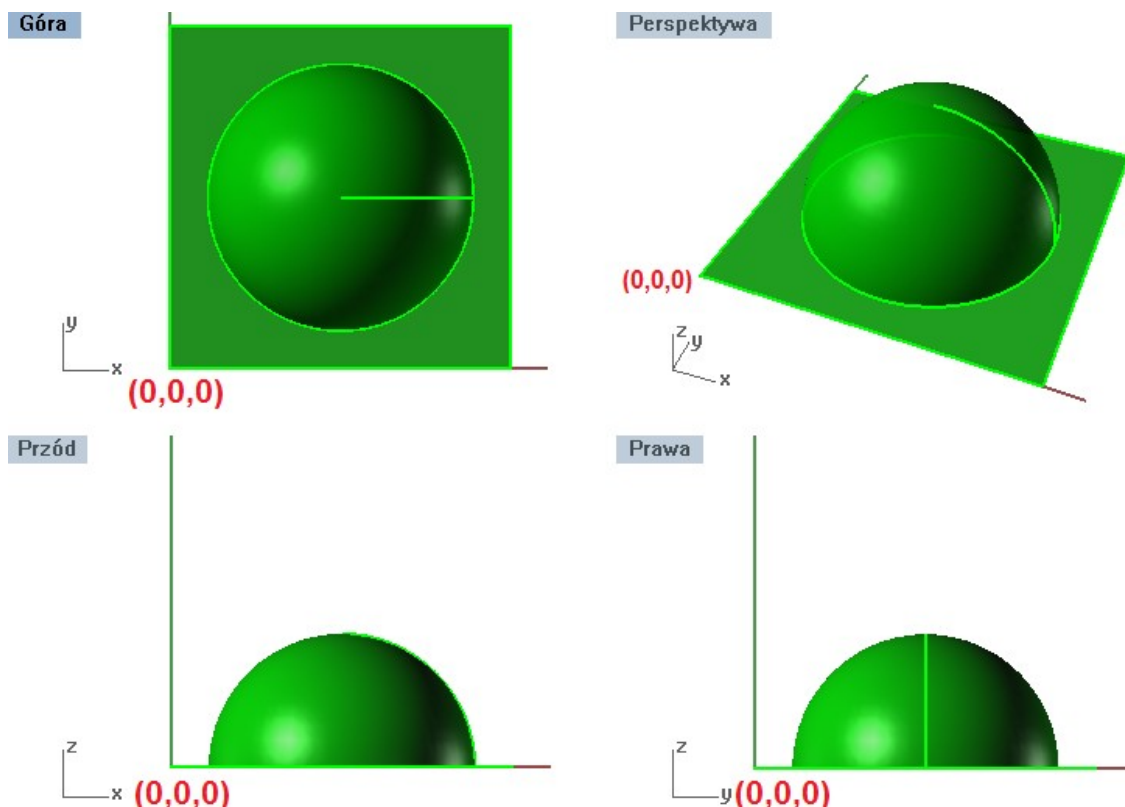
**RAW triangles** (a 3D format generated by e.g. Rhinoceros 3D or Blender)

This is a text file consisting of the co-ordinates of the triangles making up the surfaces. Please do not confuse this file format with the RAW files used in digital photography, they do not have anything in common.

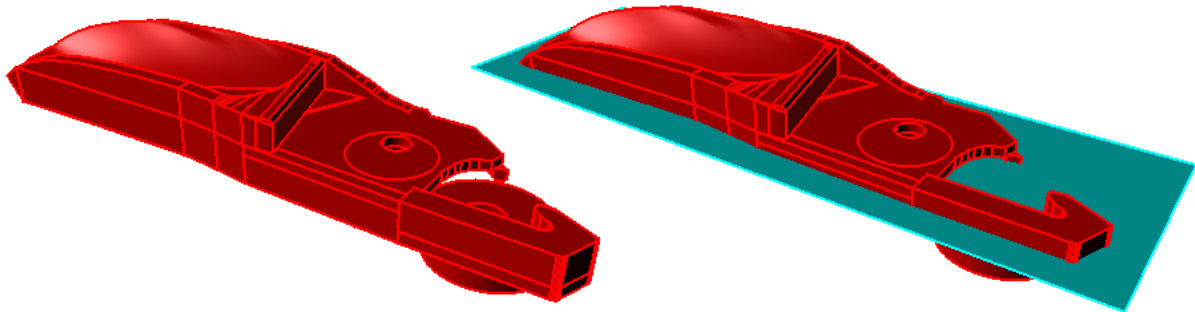
**DXF 3D** (a 3D format generated by e.g. Rhinoceros 3D or AutoCad)

To prepare a 3D dxf file suitable for MegaCut 3D software, please draw your shape using such objects as 3D Face or 3D Polyline (polyface mesh – surface mesh). All 3DSolid objects as well as other objects not representing surfaces (Arc, Line, Circle) will be ignored.

When designing a 3D model, make sure to place it in the first quarter of the co-ordinates system, as close to the 0,0 position as possible and above the Z axis 0 position. The 3D model should have a continuous surface with no holes in it in order to prevent tooling between two such unlinked surfaces.



The part of the model below the Z axis 0 position will not be tooled. If you wish to tool a certain layer of an object only and your 3D software doesn't allow for an easy separation of the required part, you may wish to cover the part which you do not wish to cut with a masking layer as shown below.



When designing your model, bear in mind the machine parameters (Z axis max. height, working area) as well as the available tools (the working length of the bits).

Please try to keep the model dimensions within 50x50 cm area, as it represents the typical max. calculating capacity of your PC. For any 3D work in MegaCut, it is recommended your PC has at least 4GB of RAM and a fast CPU. A slow configuration PC will result in very long time it will take to calculate the tool paths.

### **BMP** (grayscale bitmaps)

The software will calculate the depth of tooling based on the depth values you set for the white and black colors. It is important to keep all color changes smooth. Note your BMP may require graphics tuning prior to using it in MegaCut in order to ensure good effects. If the grayscale is not smooth, your 3D model may consist of a lot of spikes instead of a smooth surface. Your bmp files should be saved in grayscale with the max. number of colors (e.g. 24-bit). A good example of a suitable file is shown below.





## Gcod Postprocesor

The MegaCut reads from the Gcod information re. shape size and dimensions, including tool paths. The Gcod files should have the NC (\*.nc) extension. They can be prepared in any text file editor or CAM software.

Supported Gcod commands:

G20	inch co-ordinates
G21	mm co-ordinates
G0	Set movement speed (read from MegaCut Configuration)
G1	Set tooling speed as set in F parameter
G2	Set clockwise curve speed
G3	Set anticlockwise curve speed
X, Y	x and y co-ordinates
I, J, K	Circle center I=x, J=y, K=z (absolute adressing)
F	Working speed (limited by machine type)
S	Spindle rotation (limited by spindle type)

All other commands are ignored.

Working speed and spindle rotations are set in the Gcod file, while the movement speed is set in

MegaCut Configuration. The spindle rotation is turned on at the beginning of the tooling process and switched off at the end. Commands related to spindle rotation settings (on, off, rpm) are ignored. Absolute addressing..

When preparing a Gcod file, please keep in mind the above instructions. You can use both the full and short terminology:

Full	Short
N10 G90 G21	N10 G90 G21
N20 G0 X15 Y15 Z5	N20 G0 X15 Y15 Z5
N30 G1 X15 Y15 Z-1	N30 G1 Z-1
N30 G1 X20 Y15 Z-1	N30 X20
N40 G1 X20 Y20 Z-1	N40 Y20
N50 G1 X20 Y20 Z5	N50 Z5
N60 G0 X0 Y0 Z5	N60 G0 X0 Y0 Z5

Groups of contradicting commands in a single line:

```
N10 G0 G1 X15 Y15 F1000
```

In the above example, both G0 and G1 are set. The software will only change the G1 to F1000.

However, in the following example:

```
N10 G1 G0 X15 Y15 F1000
```

G0 will be set.

Both absolute and incremental addressing can be used in a single NC file:

```
N10 G90 G00 X10 Y10 Z5 S24000 (absolute)
N20 G1 Z-1
N30 X25 G91 Y15 (mixed: absolute and incremental)
```

In the N30 line, the target location will have the absolute co-ordinates of (25,25,-1).

## MegaCut (2D) software

The MegaCut (2D) software is suitable for precise tooling of any 2D shapes. It also supports multi-layer drawings for tooling at different depths. It allows for both through cuts as well as surface engraving. When cutting through a material, it is possible to use a spiral cut and a polishing operation. Thanks to a lining operation, it is possible to cut out the insides of a shape. The cutting tool diameter can be calculated in the tooling process and the tool can go on the inside of a shape, on the outside or along the center of the cutting line.

Thanks to Gcod support, MegaCut 2D allows for tooling in 3 axes as well.

The preparation of a drawing for MegaCut 2D is done in a graphics software. MegaCut 2D accepts the following formats: PLT (CorelDraw), DXF (AutoCad) or NC (postprocessor – Gcod). Once a drawing is loaded in the MegaCut software it is possible to scale it to the required size. Thanks to the simulation feature, it is possible to preview on the PC screen the exact tooling sequence prior to starting the actual cut.

### Machine start-up procedure

When turning on the machine, please always follow this sequence of steps:

- turn the electronic controller on (turn the red main switch clockwise)
- turn on the spindle cooling pump
- turn on the PC
- ensure the proper controller-PC connection (controller display should read *Ready, USB connected*; if it says *USB disconnected* it means the PC failed to recognise the machine properly)
- start MegaCut software

Possible problems in controller-PC communication are explained in the following section: [Błąd: Nie znaleziono źródła odwołania.](#)

### Software settings

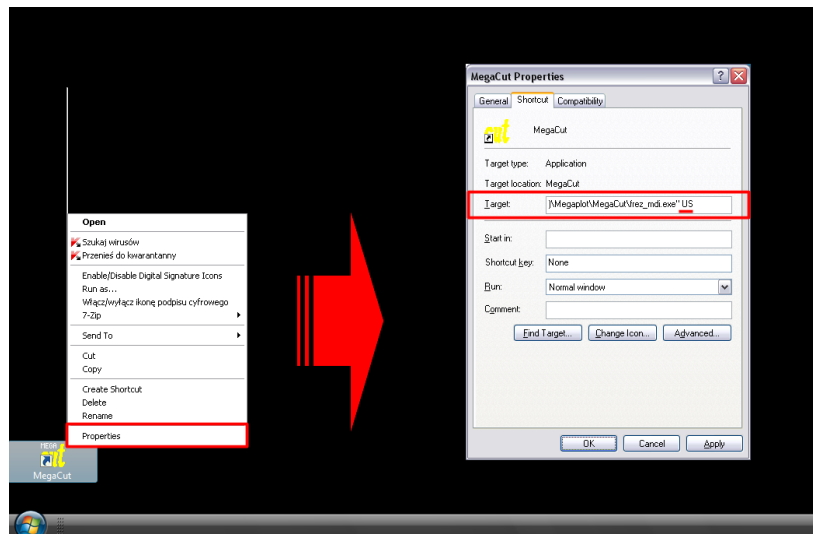
#### Language version

At the moment, MegaCut 2D is available in the following languages: Polish, English, Italian and Slovakian. The application always attempts to start up in the same language version as your PC OS. If not available, it defaults to English.

Should you require MegaCut software in any other language, please do not hesitate to contact

## MegaPlot.

To force the MegaCut application to start in a different language (provided it is available), you need to modify the MegaCut shortcut icon located on your PC desktop.



At the end of the *Target element line*, add a space and the required language code (US, PL, IT, SL, etc.). When done, the line should look as follows:

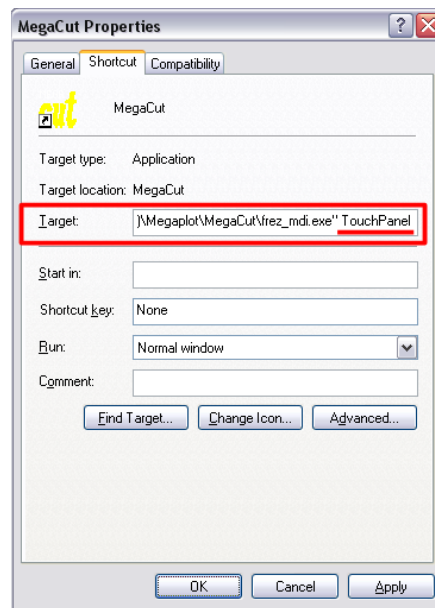
```
"C:\Program Files\Megaplot\MegaCut\frez_mdi.exe" PL
```

### Control Window - enlarged



The control window can be switched to a special touch-screen friendly mode (See [Manual Control window](#)). While the functionality remains the same, the modified look with larger buttons, etc. is better suited for operation with fingers, instead of a mouse.

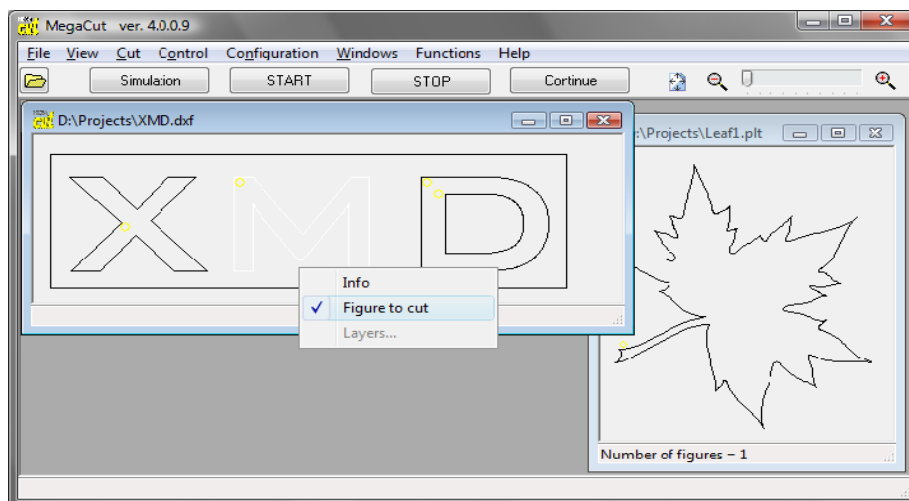
To force the modified view, you will need to modify the software shortcut again (as explained above when changing the language version). After a space, add *TouchPanel* so the line looks like this:

```
"C:\Program Files\Megaplot\MegaCut\frez_mdi.exe" touchPanel
```



## Main window

Once MegaCut is started, you will see the main window which may display a number of different files simultaneously. Under the top menu you will see a number of shortcut for the most often commands, e.g. open a file , tooling process commands, project zoom . There is also a status bar at the bottom showing current information on the machine status.




Each opened file has a handy pop-up window (available under a right-button mouse click) with the following options:

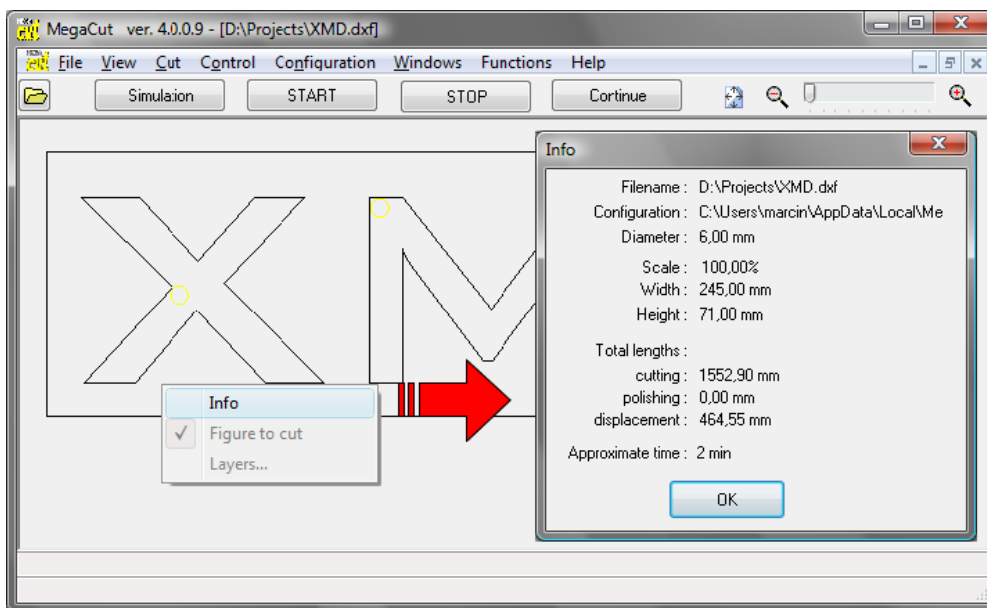
- *Info* - shows: project's dimensions, estimated tooling time, scale setting, tool diameter

- *Shapes to cut* – shows whether a selected shape is set to be cut or not. Inactive shapes are displayed with dotted lines.
- *Layers* – only active when a drawing has been opened as a multi-layer one (see [Multi-layer drawings](#)).

## Opening a drawing

To open a file, go to File \ Open [F3] or click .

Once a drawing is opened, check its dimensions by going to File \ Info or by right-clicking on the drawing and selecting Info from the window which pops up.



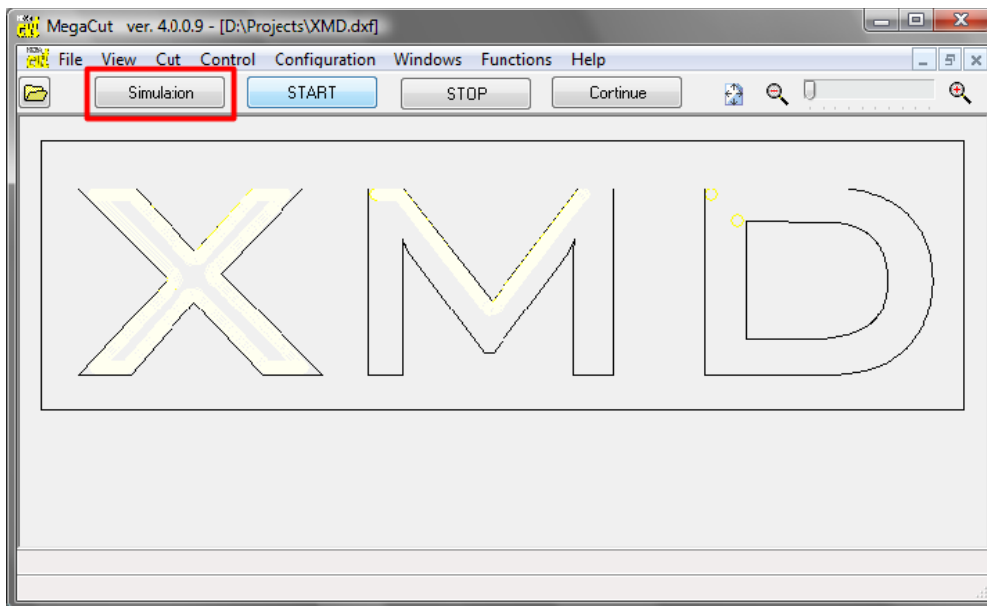
If the dimensions do not match the desired ones, it can be caused by e.g.:


- using a different length unit in the graphics software (e.g. inch instead of mm)
- errors in the export procedure in the graphics software - e.g. when exporting to plt from CorelDraw 11 (in this particular case, setting the scale to 101,6% fixes the problem. Alternatively, you can re-export from CorelDraw 11 to dxf which will have no such error).

The estimated cutting time shown in this window is based on the cutting parameters from Configuration.

## Simulation

Prior to starting a cut, it is recommended you go through the Simulation to ensure the tooling sequence will match the expected one. To simulate the cutting process, go to File \ Simulation or press F1.



The simulation is not available when a project has been zoomed. To go back to the original view of the entire shape which will bring back the simulation, please press .

Cutting direction and cutting sequence can be set in Configuration. Also in Configuration it is possible to set whether the tool should move on the inside or the outside of a shape, or move along the center of the line

## Configuration

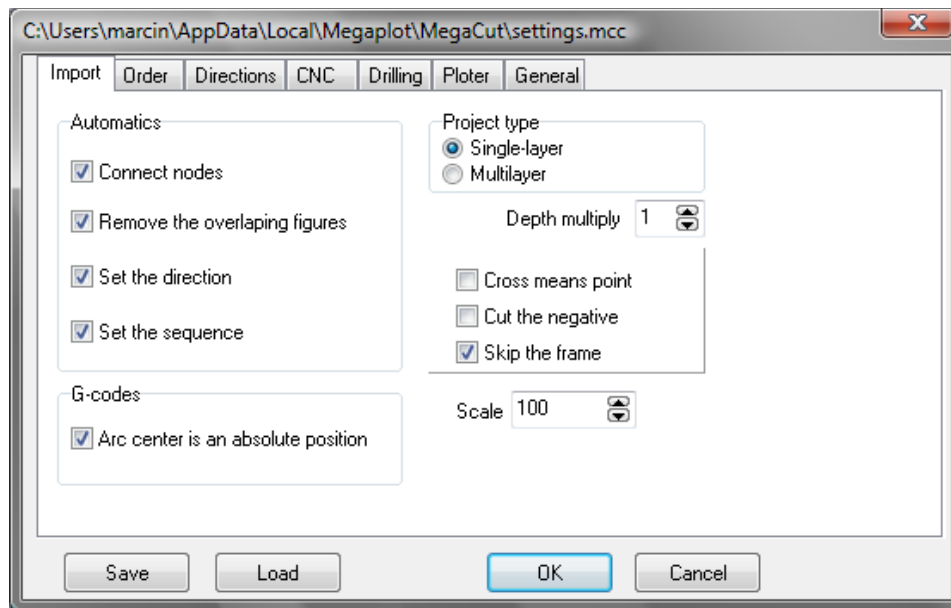
To enter the Configuration window, go to Configuration \ Main. The various available parameters are available under different tabs and should be adjusted and saved prior to opening a file.

The title bar of the Configuration window shows the location of the configuration file. You can save a number of different configuration settings and then open them as needed.

If you have a problem with cutting a particular shape, we may sometimes request you send us the configuration file. In this case, please send us the \*.mcc file, the name and location of which is shown in the title bar.

### ***Import tab***

### *Automaty*



*Connect the nodes* – the software will automatically connect all overlapping nodes in order to turn the lines into closed shapes

*Delete overlapping shapes* – the software will delete all overlapping shapes in order to avoid double-passes. For best performance, however, it is recommended you avoid having overlapping lines so delete them prior to exporting your shape to plt or dxf.

*Set cutting direction* – the software will determine the cutting direction

*Set cutting sequence* - the software will determine the cutting sequence

### *Project type*

*Single-layer* – all tooling will be done on the same depth

*Multi-layer* - the tooling will be done on different depths/layers. Layers are indicated by different colors and each layer/color has a corresponding tooling depth  
For more details, please go to: [Multi-layer drawings](#).

*Depth multiplier* – this only applies to multi-layer drawings in the plt format. The depth of each layer is automatically preset as the *pen width\*depth multiplier*. Prior to starting a cut, please double check or adjust the cutting depths values for each layer. This parameter is ignored for all other types of drawings.

*Cross means point* – the machine will drill a hole at the intersection of two lines crossing at 90 degree angle. For more details, please see [Drilling](#).

*Cut the negative* – the negative of a shape will be cut. This parameter is closely related to two



other parameters: *Skip the frame* and *Allow tool diameter*.

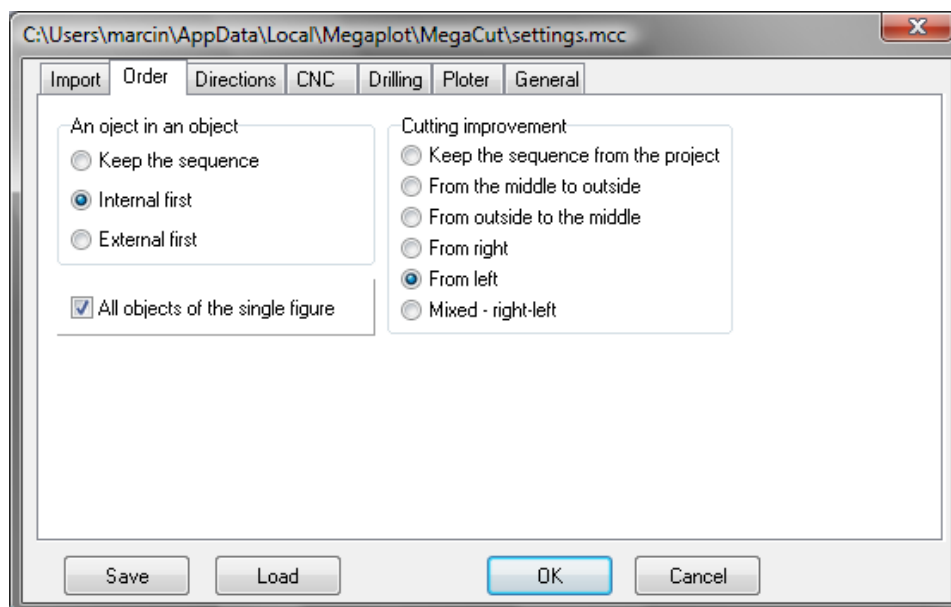
*Skip the frame* – when selected, the frame will not be cut. The frame is often drawn when creating a shape in order to position the shapes in relation to the starting position of a project. The frame is the largest closed shape in the drawing encompassing all other shapes within it.

**Scale** – allows to scale up or down a drawing. Keep in mind, scaling up a plt with small objects may result in losing the smoothness of curves. The drawing's dimensions can be checked by going to File \ Info.

### G-codes

*Absolute co-ordinates of arch centers* - (I, J) will be interpreted as absolute co-ordinates in relation to the project's starting location. When off, relative addressing will apply (in relation to the last location).

### Sequence tab



*Shape within shape* – set the sequence of cutting of shapes consisting of smaller shapes inside (e.g. letters *O* or *P*).

*Keep original sequence* – keep the sequence from the drawing

*Insides first* – most commonly used, the small insides are cut first

*Outsides first* – less common, the outside outline is cut first.

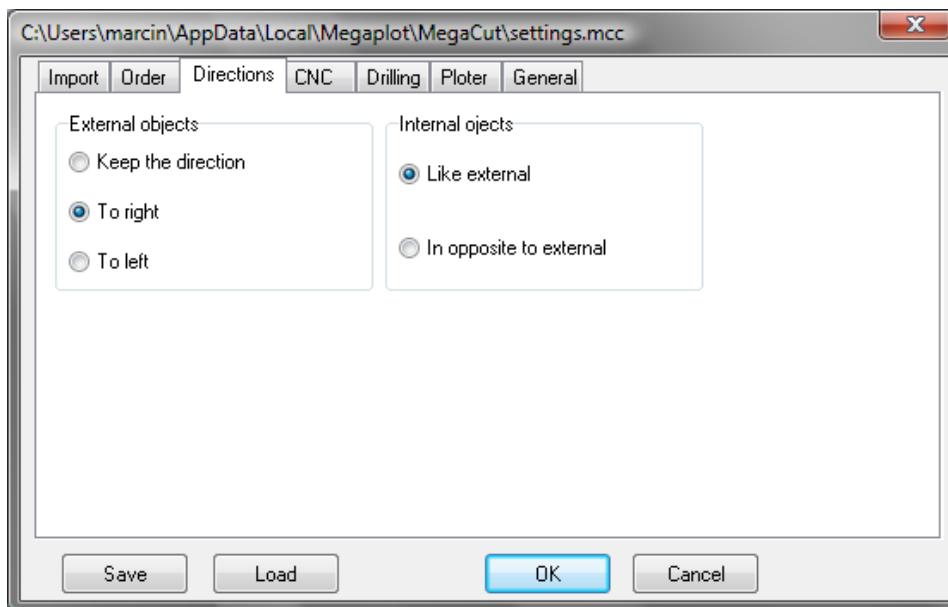
*All elements of a single shape* – when on, all elements of a shape (e.g. letter *O*) will be cut before the machine moves to another shape. If off, first all inside elements

will be cut, and then all outside ones.

*Optimize* – select the most suitable optimization for a given project

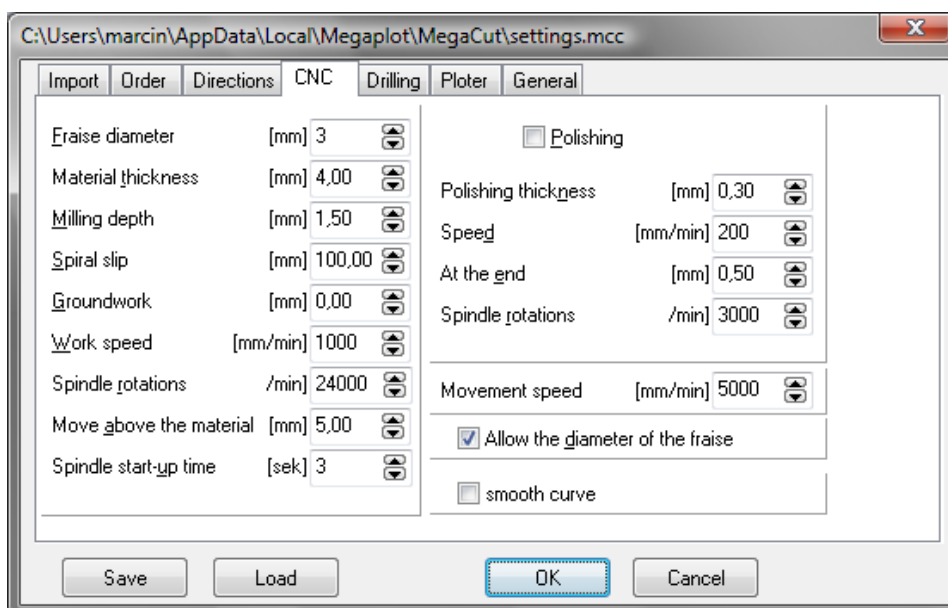
### Directions tab

The direction of tooling is a very important parameter which may have an impact on the quality of the cut. .



### CNC tab

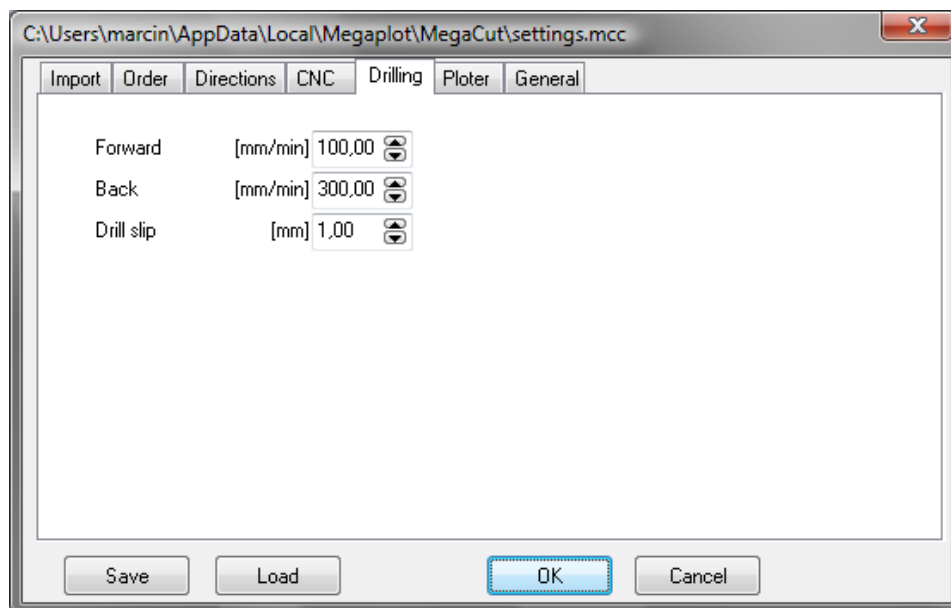
This tab consists of the most important tooling parameters.



<i>Tool diameter -</i>	set the diameter of the tool. If <i>Allow for tool diameter</i> is on, this parameter will be used to determine the movement of the tool path in relation to the actual drawing (outline size).
<i>Material thickness -</i>	insert the material thickness when the tool length measurement is performed on the table level. If the tool length is done on top of the material, set the material thickness to 0, but make sure to insert the correct <i>Milling depth</i> .
<i>Milling depth -</i>	set the depth of the cut. When cutting deep, or when cutting hard materials, make sure to set the correct <i>Spiral slip</i> .
<i>Spiral slip -</i>	if cutting soft or thin materials when can be cut with a single pass, set the spiral slip to 100. With thicker material, incremental cutting depth is used. In this case, insert the depth the tool should reach when cutting a single outline. Example: if the milling depth is 5 mm and the spiral slip is 1 mm, the machine will go over the shape 5 times, going 1 mm deeper on each pass. There will be one additional pass at the end to smooth the surface.
<i>Pad -</i>	depth of the pad placed under the material
<i>Cutting speed -</i>	the cutting speed needs to be set properly, based on the material being tooled as well as the tool. It can be adjusted during a cut via the Manual Control window or the remote control. The set speed may not be reached when cutting small objects or curves.
<i>Spindle rotations -</i>	any value between 3000 and 24000 rpm. It can be adjusted during a cut via the Manual Control window or the remote control.
<i>Move above material-</i>	the height over the material at which the machine moves from one object to another.
<i>Spindle delay -</i>	time needed for the spindle to reach required rpm.
<i>Movement speed -</i>	the speed at which the machine moves above the material from one object to another. It can be adjusted during a cut via the Manual Control window or the remote control.
<i>Allow for tool diameter-</i>	if on, the software will adjust the cutting path in order to maintain the object size (based on the tool diameter set in configuration). It will move the actual path by half of the diameter of the tool off the drawing. It applies to closed shapes only, in all open shapes (lines), the tool will follow the center of the lines. If off, the tool will move along the center of the lines also in closed shapes. The following parameters are closely related: <i>cut the negative</i> and <i>skip the frame</i> as they determine which side of the line the tool will move on (outside/inside).

- Curves smoothing* - hardware improvement of the quality of the curves which may lead to slightly modified paths. When on, the max. speed is limited to 500mm/min. The visual representation of smoothed curves may take a long time to load for complicated drawings.
- Polishing* - this is used for spiral cuts to improve surface quality. When on, the software leaves a thin layer of the material (polishing depth) for the polishing operation. The tools moves along the spiral and before it reaches the required milling depth, it goes back to the actual cutting path. The *before end* setting determines the depth at which this polishing operation begins.
- Polishing depth*- the distance the cutting path is offset in relation to the drawing during the spiral movement, i.e. the amount of material which will be tooled right before reaching the required milling death
- Polishing speed*- speed during the polishing operation
- Before end* - depth at which the polishing operation begins
- Polishing spindle rpm* - spindle rotation when polishing

### **Drilling tab**

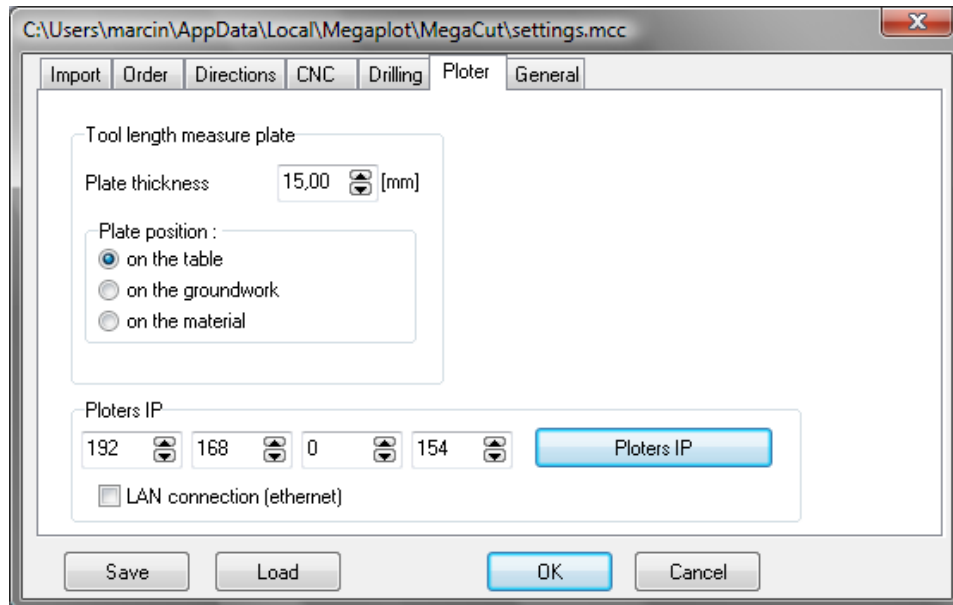


- Down* - speed at which the tool drills the hole
- Up* - speed when going up
- Drilling step* - depth for each step of drilling. If, for example, the milling depth is set to 5mm and the drilling step to 1 mm, the spindle will move down and back up

5 times, each time increasing the drilling depth by 1 mm. The up movements ensure proper material cleaning out of the drilled hole.

## **Ploter tab**

Different settings for the measurement of the tool length (for details, go to: [Tool measurement](#)).



### *Tool measurement pad*

*Pad thickness* – set the thickness of the pad used to determine the tool length

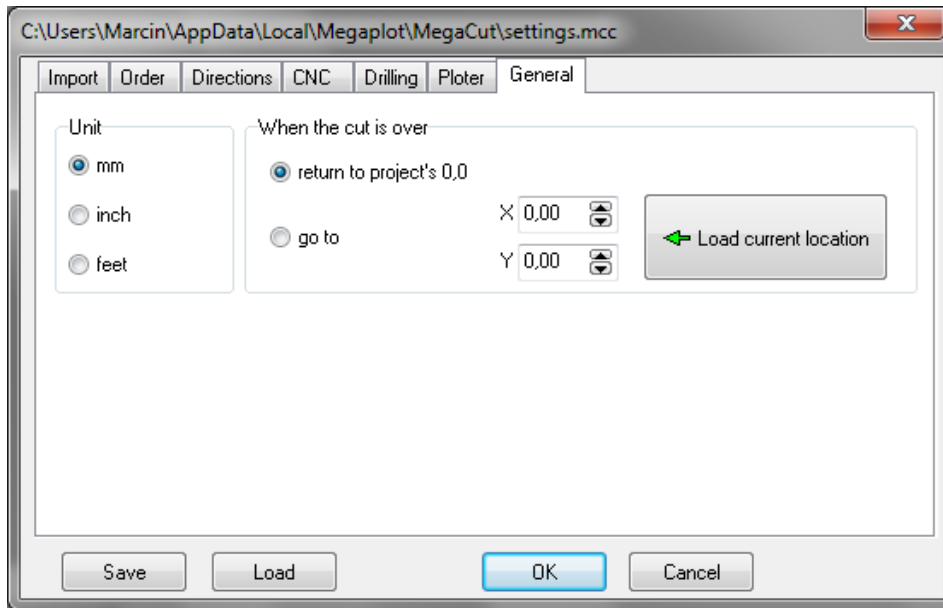
*Pad location* – set the location of the pad during the measurement procedure. Material depth and pad thickness need to be set prior to the measurement procedure. When measuring on the top of the material, the material thickness may be set to 0.

*Ploter IP* – hidden by default, as the ethernet module is an optional custom component. It becomes visible only when the ethernet parameter is properly set (details: [Network connection \(ethernet\)](#)).

*Ploter IP (button)*- sets the router's IP in the controller to the address shown on the left. For this to work, the controller needs to be connected to the software via USB.

*Network connection (Ethernet)*-activates the ethernet connection

## General tab



*Units* - set the units used in MegaCut software

*When the cut is over* - specifies the place where the spindle is supposed to move after the cutting finishes

*return to project's 0,0* - when the operation ends the spindle moves back to the original project's zero location. It will stay at the height specified in the Configuration in the CNC Tab: *Movement above the material*.

*go to XY* - when the operation ends the spindle goes all the way up and moves to the specified X and Y values IMPORTANT: if you use high material holders, there is a danger of the spindle hitting those holders!

*Load current location* - a button to set the X and Y values to the current spindle location

## Tool measurement

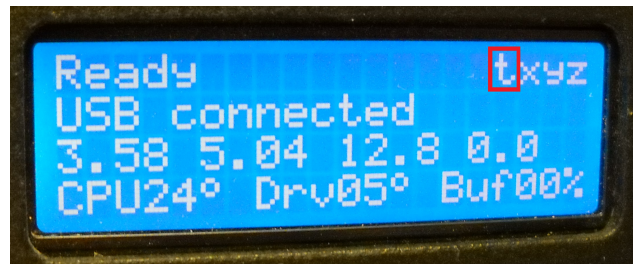
The tool length needs to be measured every time a new tool is installed. It determines the distance between the tool and the machine table.

The tool length measurement may be done either on the machine table, the pad under the material or on top of the material. The chosen method needs to be set in Configuration in the Ploter tab.

Before you measure the tool length:

- make sure correct pad thickness is set in Configuration (Configuration \ Ploter \ Pad thickness) [this applies to automatic measurement].

- make sure the down movement of the spindle at the given location will be possible and the tool will be able to reach the plate without problems.
- Make sure the tool will be able to reach the lowest location of your project. If, e.g. the spindle is installed high and the tool is short, it may not be possible for the tool to reach the measuring plate or the table, in which case the spindle may reach its lowest position and attempt – without success – further down movement. In this case, stop the measuring process, home the machine, correct the problem (lower the spindle or use a longer tool) and repeat the process.
- Ensure the measuring plate works properly [applies to automatic measurement].

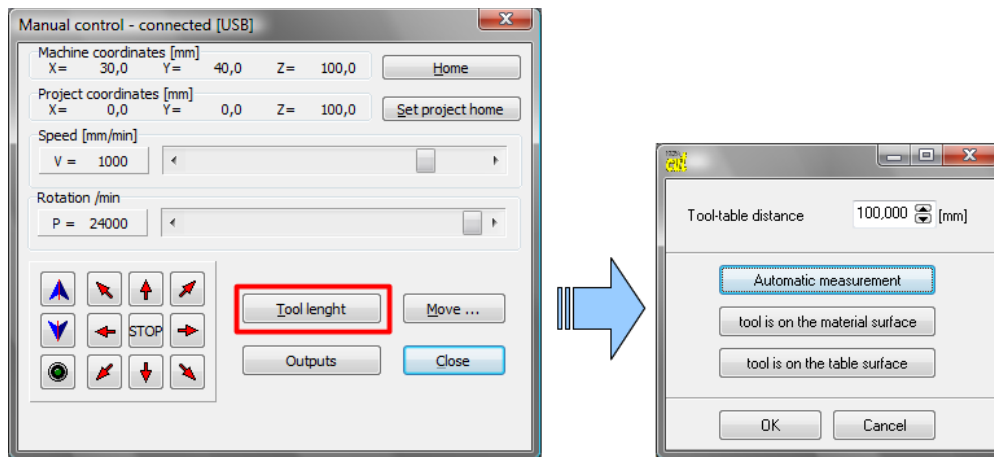


Touch the spindle with the measuring plate. If the small letter *t* turns into a capital *T*, it means the automatic tool length measurement is working properly.

### Automatic tool length measurement

The automatic tool length measurement is done with the measuring plate described above. The spindle will slowly move down until its tool reaches the plate. Once done, the tool length is automatically saved and the spindle will move back up.

To ensure the measurement is precise, make sure the measuring plate is clean, the surface under the plate is clean and flat and the pad thickness is specified properly in the software (Configuration \ Ploter \ Pad thickness).



To measure the tool length on the machine table:

- go to Configuration, Ploter tab and select the location of the plate (table, pad, material)
- move the spindle above the plate
- place the plate on the surface
- *go to Manual Control \ Tool length*
- *press Automatic measurement*
- wait till the tool touches the plate and the spindle moves back up
- put away the measuring plate

If you put the measuring plate directly on the aluminum table of the router, any small movement of the plate may prematurely end the measuring process. To avoid this, put a thin piece of isolation on the bottom of the measuring plate. Make sure you remeasure the plate thickness and insert the correct value in the configuration.

**CAUTION!** Always place the plate with the insulated side down. Otherwise, the measuring process will not work properly and may result in breaking the tool, damaging the router or injury.

### **Semi-automatic tool length measurement**

Alternatively, you can measure the tool length in the following way:

- move the spindle to the location of the measurement
- open *Manual Control* and *Tool Length*
- insert *999* for *tool length* (to allow for unlimited movement down)



- move the spindle down very slowly until the tool reaches the table (make sure to do it very slowly to avoid breaking or damaging the tool)
- *open Manual Control and Tool Length*
- press “Tool on table level” or “on material level”
- move the spindle up

If the new tool is shorter than the previous one and you use the semi-automatic way to measure it, the spindle will not reach the table level as it is impossible to move the spindle into the negative co-ordinates. In this case, enter the 999 values as described above.

When the router is switched off and back on again, the last tool measurement is remembered. If the tool has not been replaced in the meantime, there is no need to measure it again.

## Cutting

Before you start the cutting process, please double check all parameters in configuration, load the drawing and perform the simulation procedure. When you are ready to start, press START or F2.

The spindle will first move to the project's starting position and then the spindle rotation will turn on. Spindle rotation and cutting speed may be adjusted during the cutting process.

You can stop the cutting process at any time by clicking STOP. If it is necessary to clean the tool, you can move the spindle to any location on the machine and once the tool is clean, press CONTINUE or Shift +F2 and the router will move back to the location where the cutting process was stopped and will continue from this spot.

After you press START the cutting process will start from the beginning as shown on the Simulation. However, it is possible to start the cutting from any given shape in the drawing. If needed, select a shape and click Cut \ *Start from selected shape*. You can also cut a single selected shape, if required.

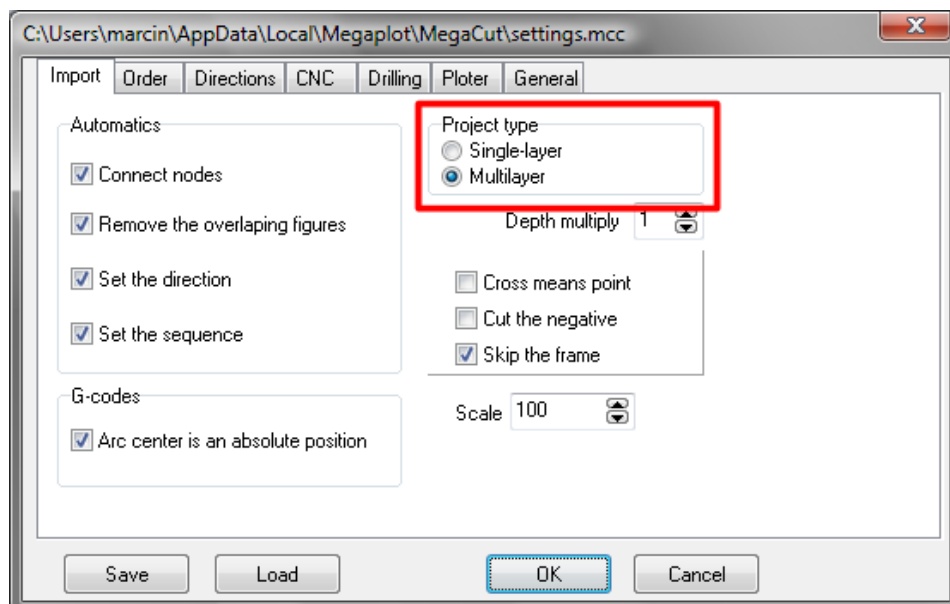
## Multi-layer drawings

When using multi-layer drawings, it is possible to engrave and cut from a single operation. In this case, the drawing will consist of a number of colors and for each color the operator can select the required cutting depth.

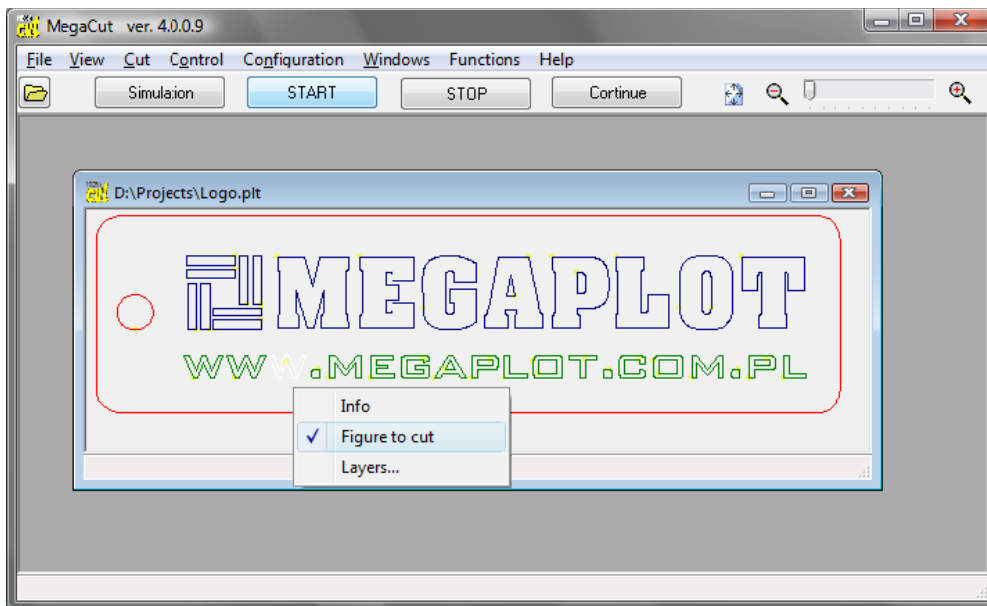


The multi-layer drawings can be prepared in either plt or dxf. When preparing your drawing, color code the different layers. Later on you will set the cutting depth for each color in the MegaCut software.

Before you open a multi-layer drawing, go to Configuration and in the Import tab select the project type to Multi-layer. Once selected, the milling depth setting in the CNC tab will be deactivated as instead of one milling depth, you will now need to specify the depth for each layer separately.

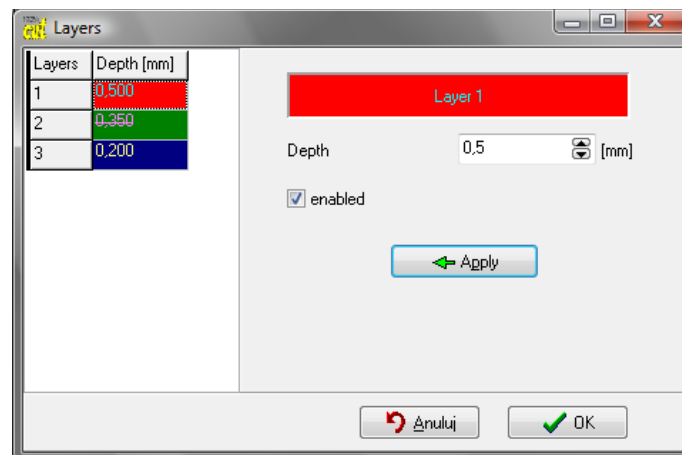


To set the parameters for each layer, open the Layers window, by either right clicking on the drawing or going to Configuration – Layers.



On the left you will see all the colors in the drawing. Once you select a color, you can modify its parameters on the right. Adjust as needed and click Apply.

Apart from selecting the depth of each layer, you can also use this Layers window to deactivate a layer so it is not cut at all (all the elements of a given color will be skipped). Deactivated layers will be crossed out (in the example below, the green layer is inactive).

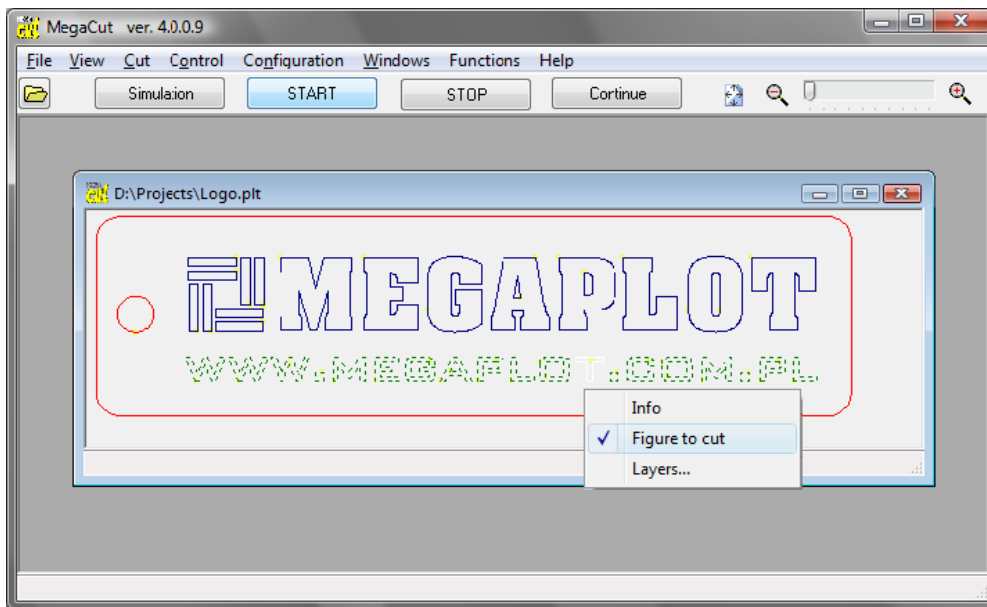


The layers which have been turned off will be displayed with dotted lines. Even after you deactivate an entire layer, it is still possible to turn back off selected elements of a layer. Simply select an object, right click on it and activate it.

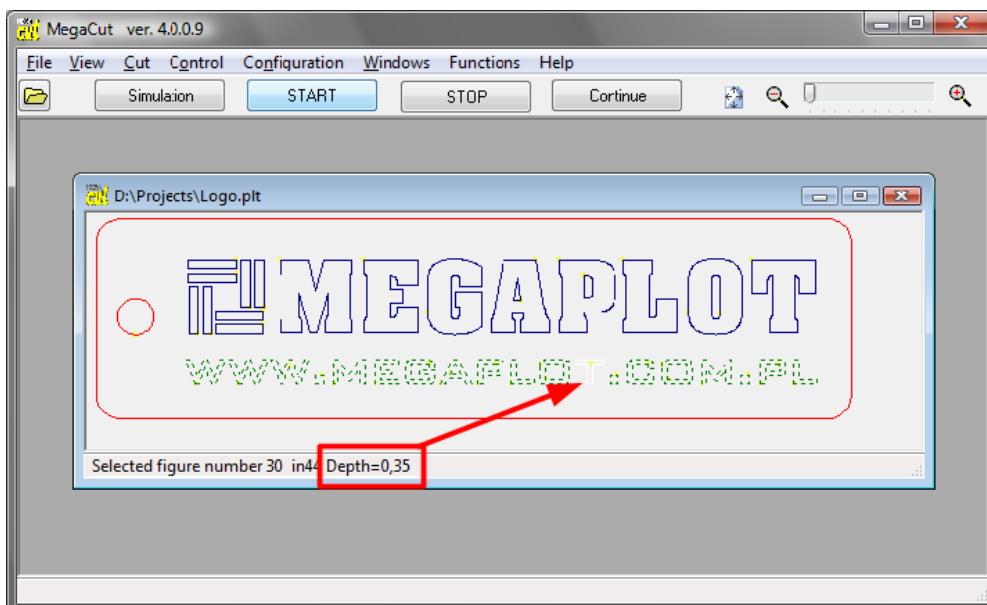
Once a multi-layer drawing is opened, the milling depths are automatically set, depending on the

type of drawing. **Make sure to correct these values as needed before starting the cut.**

- PLT the initial depths are based on the pen width multiplied by the depth multiplier from Configuration. The pen width can be specified in the CorelDraw plt export window.
- DXF the initial depths are based on the milling depth setting from the CNC tab in Configuration



When back in the main window, you can select any shape you wish and confirm the settings by checking the milling depth as displayed on the bottom status bar.



### Multi-layer drawings requirements:

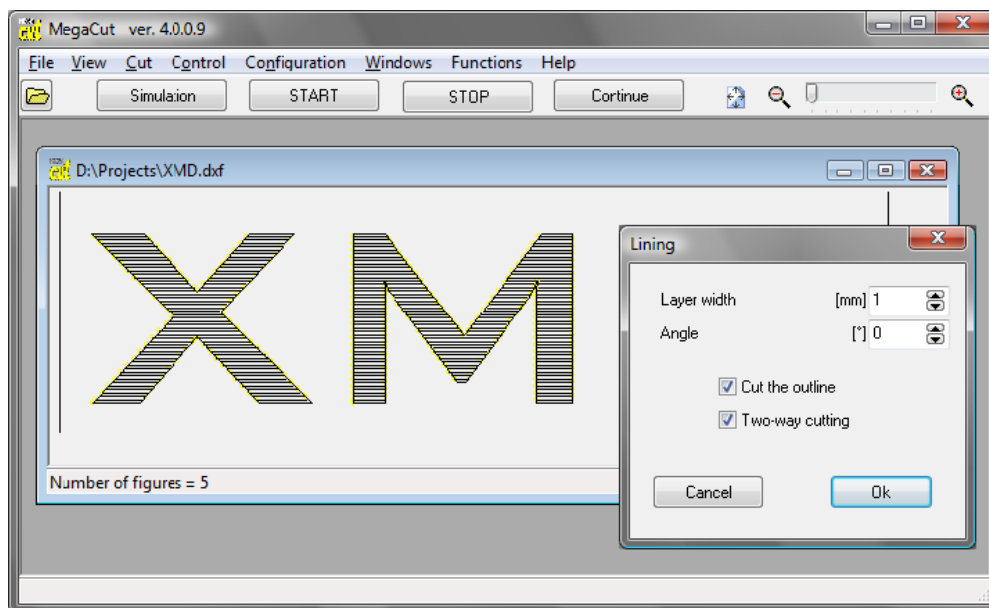
- use standard palette colors
- DXF:
  - all elements should be on a single, zero layer
  - do not use BLOCKS, all elements should be drawn with polylines, curves, etc.
  - each element needs to be colored (layer color is ignored)

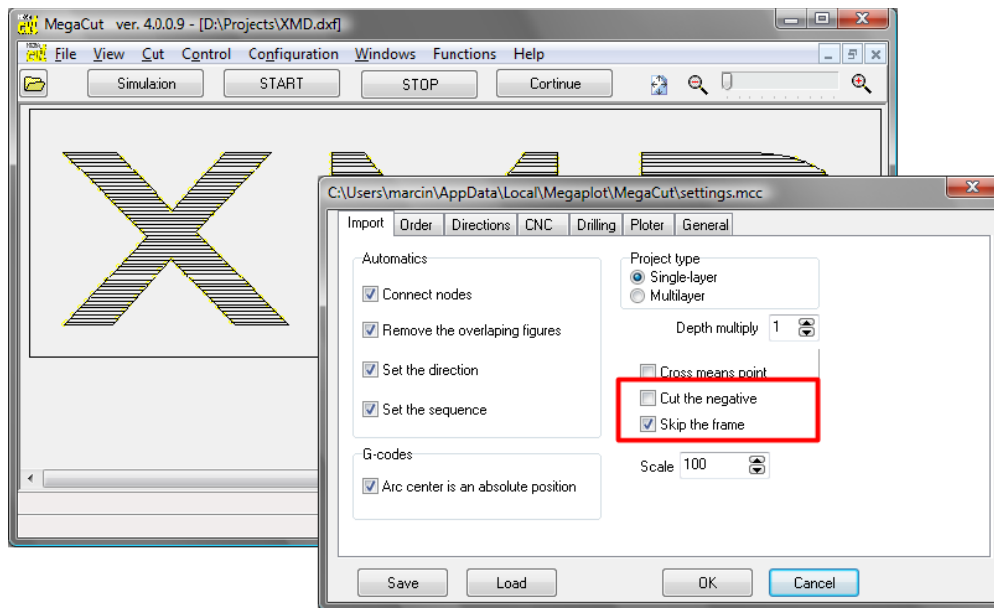
## Lining – cutting out the insides of shapes

In order to clear the inside of a closed shape, go to Functions \ Lining. You can modify the lining operation by specifying the distance between the passes as well as the required angle.

Prior to performing the lining operation, it is important to select – as required – the *Cut the negative* and *Skip the frame* parameters as they influence the lining operation (insides vs. outsides). Select *Allow for tool diameter* if required.

The following examples show the lining operation alternatives, depending on the settings.



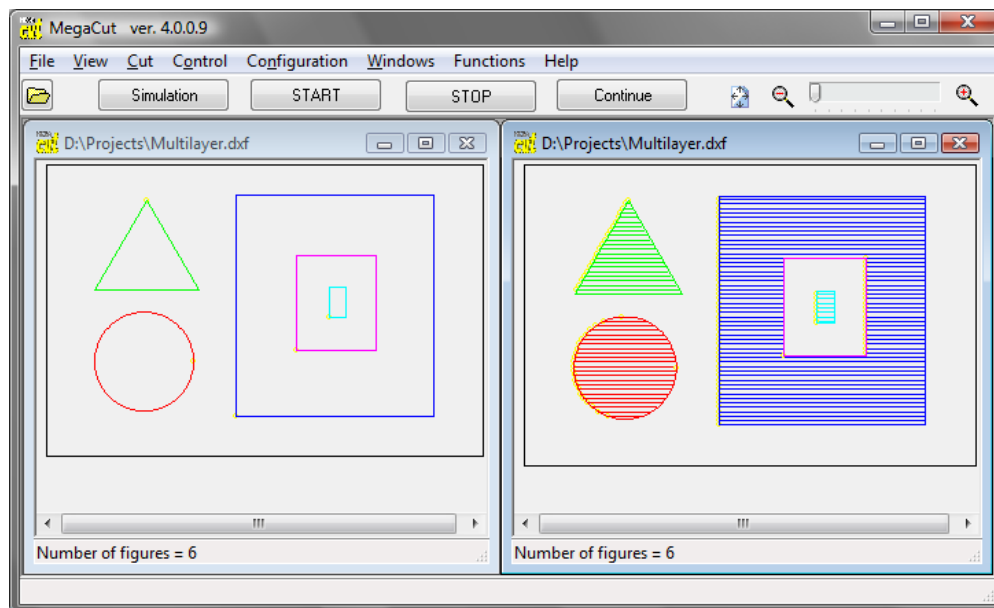


When *Spiral slip* is active, this setting also applies to the lining operation.

The lining operation can be performed at different depths if combines with the multi-layer drawings described above.

## Manual Control window

The manual control window allows for a number of manual operations, including the control of the machine in each axis, as well as:

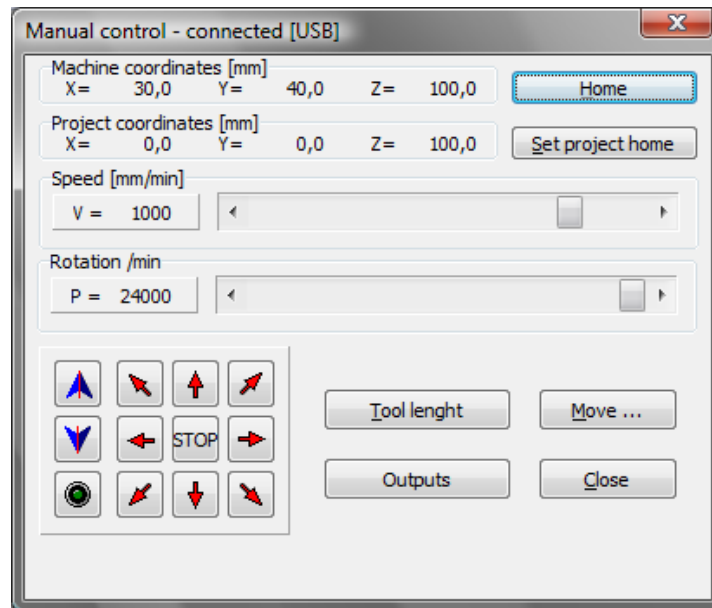


- activating and deactivating spindle rotations – **always make sure the spindle cooling**

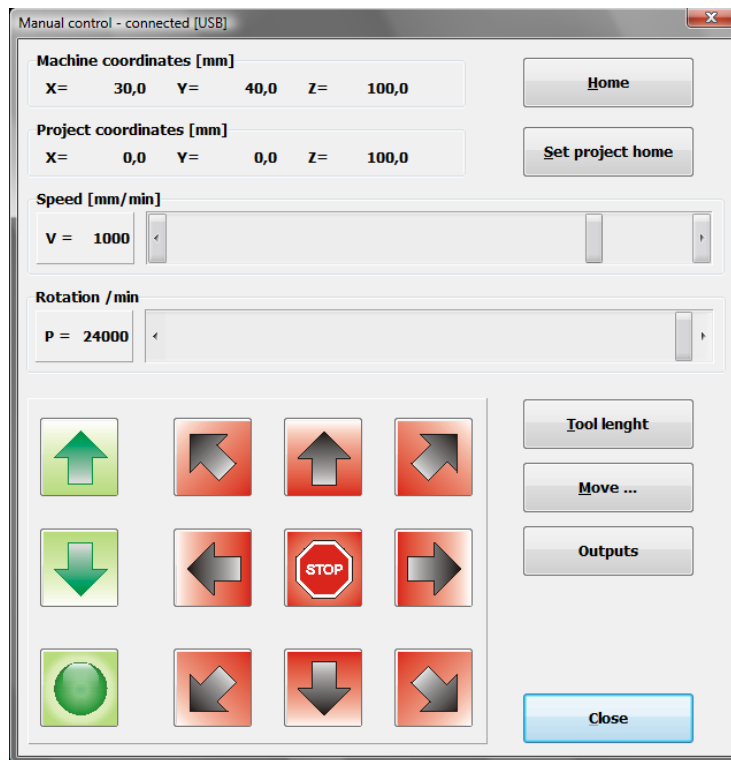
**system is on before you turn the spindle rotations on,**

- homing procedure,
- setting the project's starting position,
- moving in any axis by a given distance to to absolute co-ordinates

The control window shows the machine co-ordinates (in relation to its absolute home position) as well as the project's co-ordinates (in relation to its starting location). Note the remote control display only shows the project's co-ordinates.



This window can display in the alternative, touch-friendly mode as described earlier ([Control Window - enlarged](#)).



## Homing process

Homing (Manual Control – Home button) is used to bring the machine back to its original home position by calibrating it in relation to the proximity sensors. It is recommended to perform the homing process when the machine was blocked while moving or the tool got stuck inside of material because of wrong cutting parameters. If the machine gets stuck in the material or by hitting an object on its table, you will hear the sound of the motors trying to overcome the obstacle. In this case, home the machine to bring it back to its original home position.

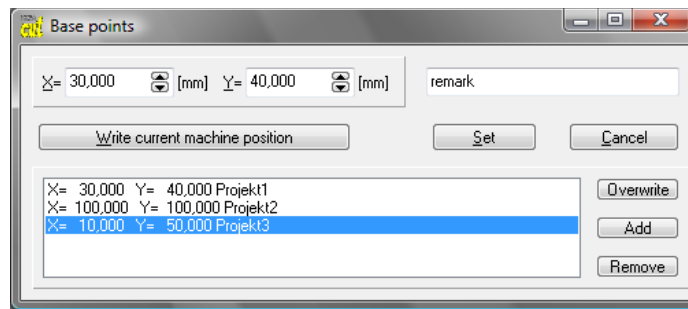
To ensure the homing process is performed properly and precisely, ensure the proximity switches as well as the lead screw covers are clean and allow for free movement (see [MAINTENANCE RECOMMENDATIONS](#) ).

## Project's starting location

The difference between the machine home position and the project's starting locations was described in [Machine home position vs. the project's starting position](#).

To set a new starting position for a project, move the spindle to the required location, go to Manual Control and select *Set project's starting location*.



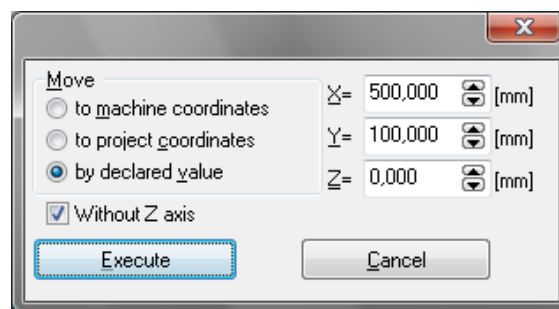


Next, press the *Insert current machine location*. Note this location will be remembered even after the controller has been turned off and back on.

You can save any number of project's starting locations, name them and go back to them in the future as required. Just select the one you wish to set and click Set.

### Mote to and Move by

Apart from manual movement in the Manual Control window, you can also move the machine precisely in each axis by using the Move... window. There are three choices available as shown below:



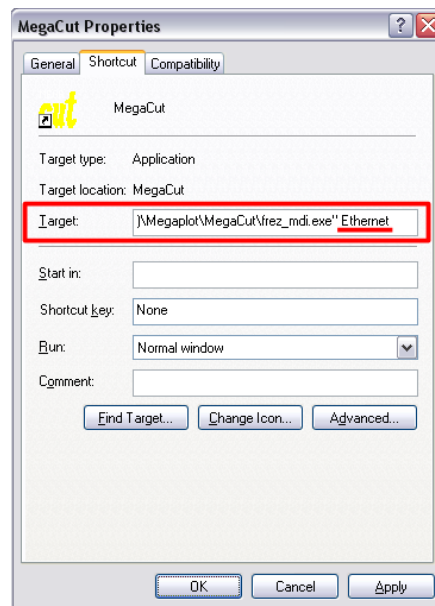
Before you execute the movement, please make sure to bring the spindle up so the tool does not run into the material or the material holders.

You can use this to check the location of last project's starting location. Simply select *To project's co-ordinates* and insert 0 for all three axes.

## Network connection (ethernet)

The standard controller-PC is done via USB connection, yet ethernet communication is available as an optional upgrade. The applicable options will be visible in the configuration only after the correct parameter is set in the MegaCut icon shortcut properties.

In order to control the machine via ethernet connection:



- connect the controller to the PC via standard USB connection
- right click on the MegaCut shortcut icon on PC desktop and modify the path by adding a space at the end and adding the parameter *ethernet*. The target path should now look as follows:

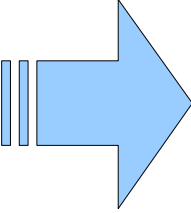
"C:\Program Files\Megaplot\MegaCut\frez\_mdi.exe" ethernet

- start MegaCut
- go to Configuration and the Plotter tab
- *enter the plotter IP* for the machine (it needs to be unique on your network)
- press the plotter IP button. The IP should now show up on the controller's display
- activate the network connection (ethernet) setting
- confirm by clicking OK
- close MegaCut
- disconnect the USB cable from PC and the controller
- connect the controller to your ethernet network
- open MegaCut, if the connection works properly, "*Ethernet connected*" will display in the bottom status bar as we as in the top bar of the Manual Control window

## Router calibration

The calibration feature is used when the machine moves in one axis incorrectly, i.e. too much or too little, and if the error is proportional along the entire length of the axis. You can set the calibration adjustment for each axis separately.

In order to calibrate the machine in this way, create a text file named *kalibracja.txt*. The file should have three lines, and each line should have the calibration value for the given axis: X, Y and Z respectively:

Kalibracja.txt		
1.003		increases movement by 3mm in X axis
0.999		decreases movement by 2mm in Y axis
1.000		no calibration in Z axis

For axis not requiring calibration, set the value to 1.0. This calibration file should be placed in the main MegaCut folder where the *frez\_mdi.exe* is located.

The calibration must be within the range  $\langle 0.99; 1.01 \rangle$  so it can correct errors of  $\pm 10$ mm.

The calibration value can be calculated in the following way:

$$\text{calibration value} = \frac{\text{actual distance}}{\text{required distance}}$$

## MegaCut3D software

MegaCut3D is used to cut 3D models out of RAW or DXF 3D files. It can also create a 3D model out of a suitable grayscale bitmap file. You can create the model from scratch in any compatible 3D graphics software (e.g. Rhinoceros, AutoCad), or by adopting one of the many free 3D models available on the internet. It can also be obtained in the process of 3D scanning. The model preparation is done entirely outside and independently of MegaCut 3D software.

Once a 3D model is loaded into MegaCut 3D, the operator sets the parameters and material dimensions, and MegaCut 3D software prepares the output tool paths.

### Machine start-up procedure

When turning on the machine, please always follow this sequence of steps:

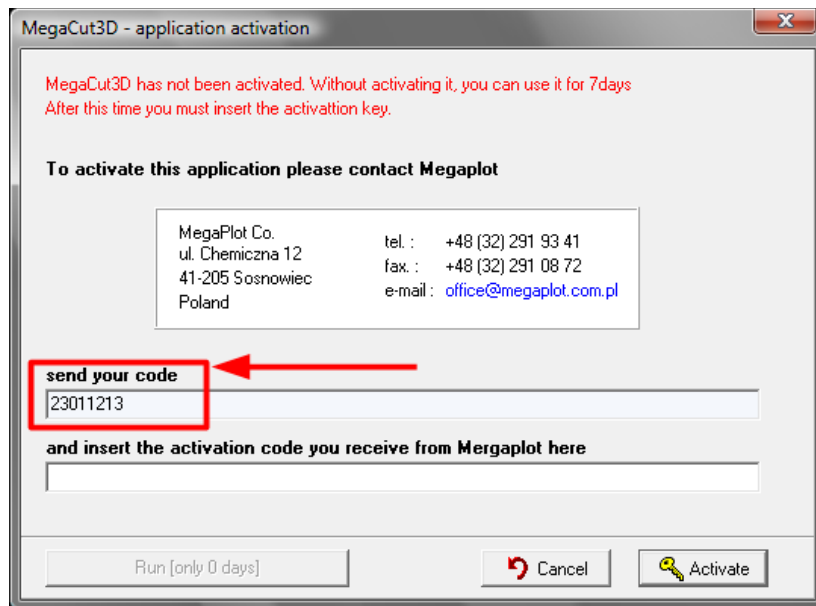
- turn the electronic controller on (turn the red main switch clockwise)
- turn on the spindle cooling pump
- turn on the PC
- ensure the proper controller-PC connection (controller display should read *Ready, USB connected*; if it says *USB disconnected* it means the PC failed to recognise the machine properly)
- start MegaCut 3D software

Possible problems in controller-PC communication are explained in the following section: [Błąd: Nie znaleziono źródła odwołania.](#)

### First run - activation

MegaCut 3D is supplied with each XMD router and is included in the price (including future upgrades). However, in order to use it you need to activate it. Without the activation, the software will run fully functional for 7 days only.

In order to activate MegaCut 3D, first follow the steps explained above in [Machine start-up procedure](#). When the software starts for the first time, an activation window will be displayed. In order to receive the activation code, please contact MegaPlot as advised on this screen (your individual code needed for the activation will only display if you connect the machine to the PC). Once activated, the software will not require any further activations in the future.



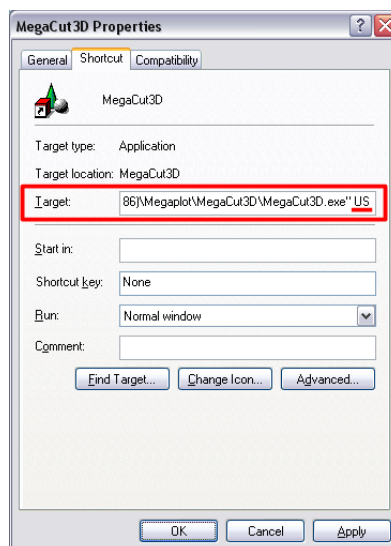
## Software settings

### Language version

At the moment, MegaCut 3D is available in the following languages: Polish and English. The application always attempts to start up in the same language version as your PC OS. If not available, it defaults to English.

Should you require MegaCut software in any other language, please do not hesitate to contact MegaPlot.

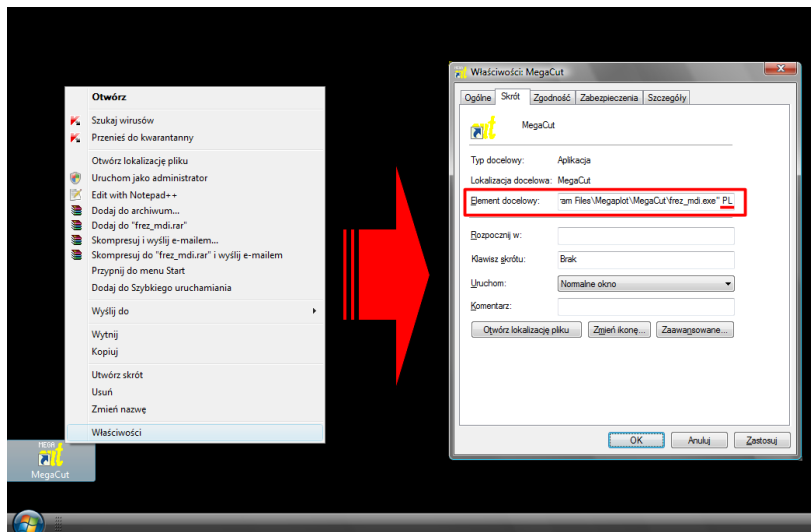
To force the MegaCut application to start in a different language (provided it is available), you need to modify the MegaCut shortcut icon located on your PC desktop.



At the end of the *Target element line*, add a space and the required language code (US, PL,

etc.). When done, the line should look as follows:

"C:\Program Files\Megaplot\MegaCut3D\MegaCut3D.exe" PL

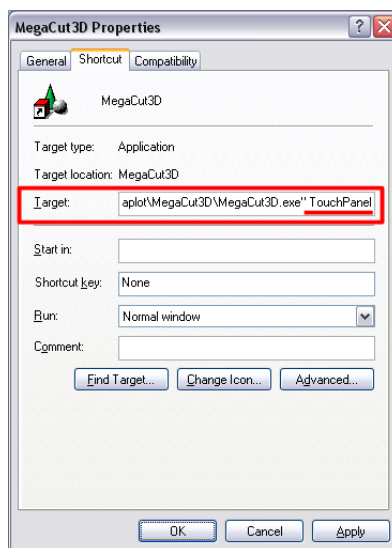


## Control Window - enlarged

The control window can be switched to a special touch-screen friendly mode (See [Manual Control window](#)). While the functionality remains the same, the modified look with larger buttons, etc. is better suited for operation with fingers, instead of a mouse.

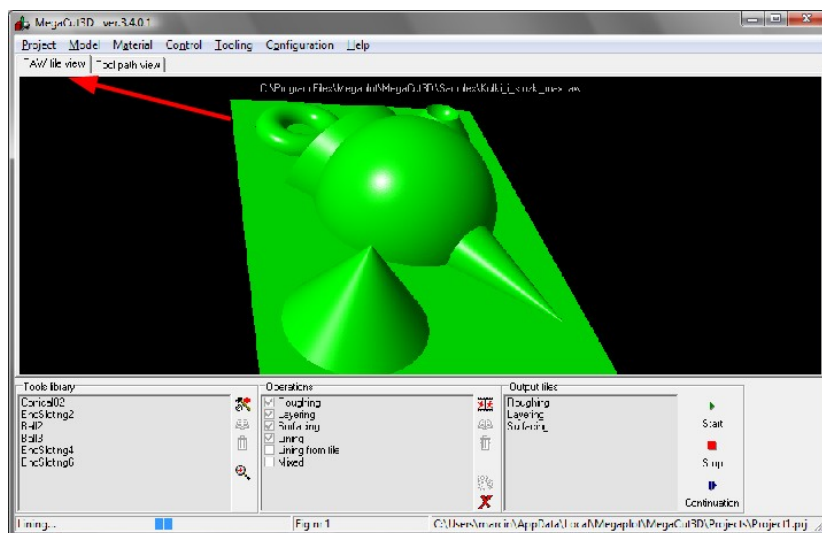
To force the modified view, you will need to modify the software shortcut again (as explained above when changing the language version). After a space, add *TouchPanel* so the line looks like this:

"C:\Program Files\Megaplot\MegaCut3D\MegaCut3D.exe" touchPanel

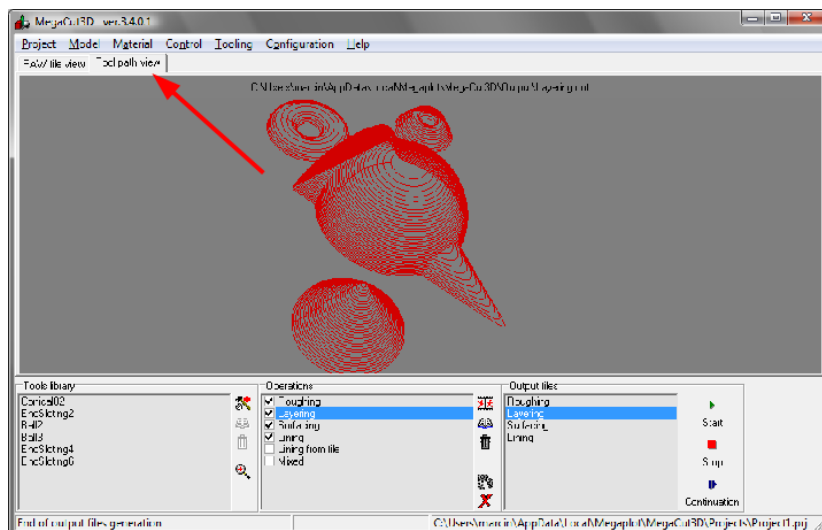


## Main Window

When you open MegaCut 3D, the main window shows two tabs: one is used to preview the 3D model and the other tab is used to preview the output files/tool paths. Below the model, you will see three lists: tool library, operations list and output files list.



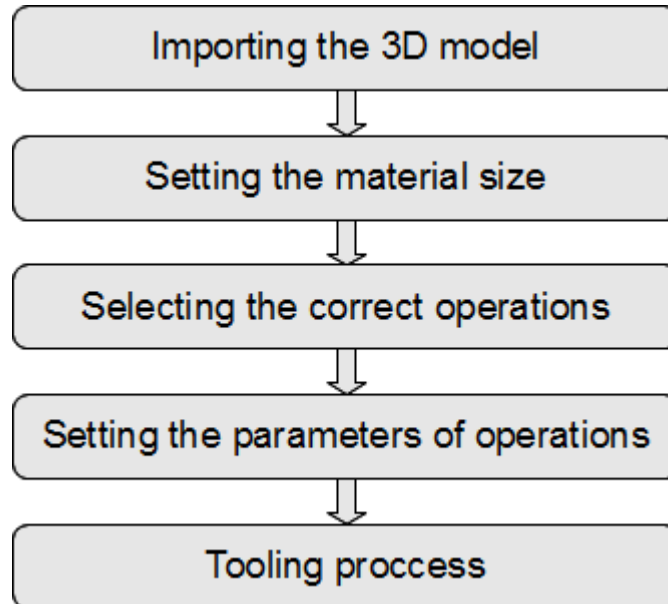
The bottom bar displays current information as well as a progress bar indicating the process of generating output files. On the right side, the name of the project is displayed.



The top part of the window shows the main menu.

## Typical tooling sequence

Here is the outline of the standard, most common 3D tooling process:



Here is the more detailed way which one should follow for best results:

1. Turn the machine one and wait till the controller's display says Ready
  - a) fix the material on the machine table
  - b) set the project's starting position (Manual Control – Set Project's starting postion)
2. Open MegaCut 3D
3. Create a new project – Project – New [CTRL+N]
4. Open a previously prepared 3D model (*Model \ Open [CTR+O]*)
5. Set material size (Menu: *Material*)
6. Define the tools you will use for given operations (Tool library)
7. Define the operations you will perform and set their parameters (*Operations*)
8. Select the operations you will perform
9. Generate the output files for your operations (button *Generate output files* on the right side of the operations list or go to Tooling – Generate output files F4)



10. Select one of the output files – you should always start with the roughing operation
11. Install the tool which had been defined for this operation
12. Measure the tool length (Manual Control – Tool Length)
13. Start the operation - START
14. Repeat steps 10-13 for all operations to achieve the required effect. For most projects, the combination of the roughing operation and the lining operation is sufficient.

## Loading the 3D model

MegaCut 3D is not used to prepare 3D models. These should be prepared in suitable 3D graphics software. For more information on supported format and drawing requirements, please see the following sections.

There are three ways to open a 3D model:

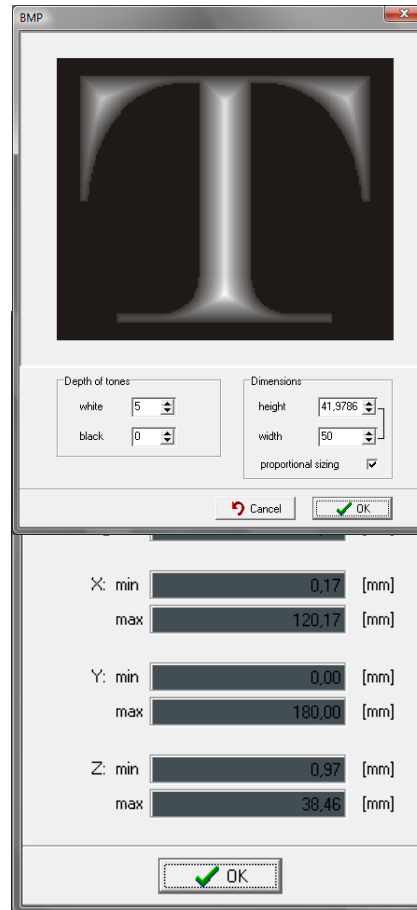
- select Model – Open [*CTRL+O*]
- double click on the black preview section
- *CTRL+O*

Depending on the model dimensions and resolution (file size), the file should open within 5-15 seconds. Once it loads, you can view it in various ways: rotate it with the right click, move with the left click, zoom in and out (mouse roll or CTRL + right click). To go to the initial view, select Model – Restore original view.

To preview information about the model, such as dimensions and file size, you can go to Model – Properties or you can double click on the model path name right above the model. Before you start any cutting, make sure the model size corresponds to the material dimensions. Also, double check whether any parts of the model are not below the Z axis, as such parts will not be cut. If any part of the model extends downwards below the Z axis, the Z axis min. value on the properties window will be negative.

## Loading a bmp file

MegCut 3D can also load a grayscale bmp file and turn it into a three dimensional model. More details on the preparation of bmp files is provided in other sections. To open a bmp file, go to Model – Open *CTRL+O* or double click on the preview screen.



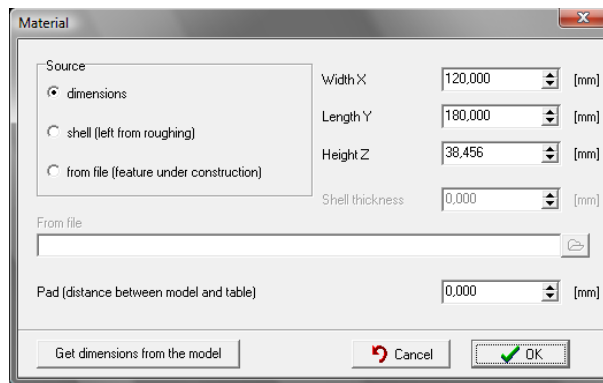
When opening a bmp file you need to specify the height values for the black and white colors. And the software will automatically determine the heights for all grays in between. If, for example, you set 5 for white and 0 for black, the lighter shades will be above the darker ones (and the letter T from the example above will be convex).

At this point, you can also scale your bmp file, both length and height

## Material

Material size and the pad thickness are specified in the Material menu.

*Source:*



*dimensions* – in this case you can insert the exact material dimension in all 3 axes.

*surface (left from roughing operation)* – the material size can also be automatically determined as the shell left from the roughing operation. This option is useful when you plan to continue to work on an object which has been earlier prepared with the roughing operation.

*Pad (distance between material and table)* – here you can insert the actual thickness of the pad. You can also play with the *Z height and pad thickness* settings in order to force the tooling at the required height of the material (when e.g. the material is much higher than the model). For more details, see the following sections.

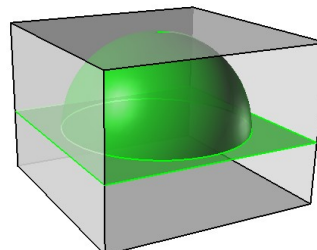
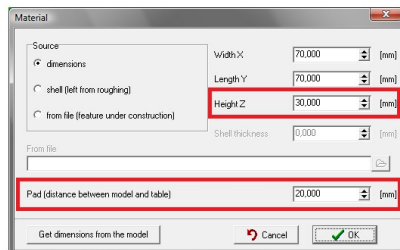
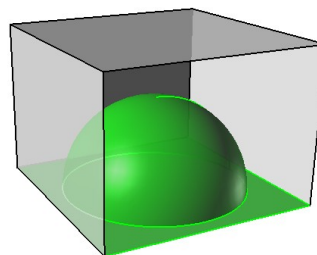
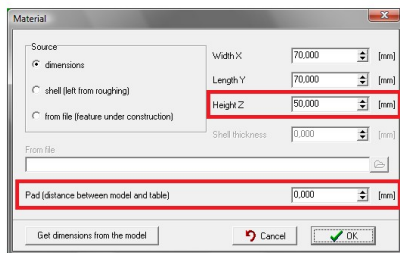
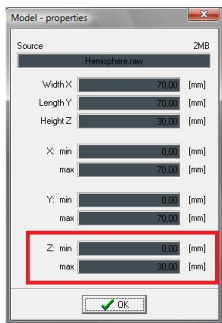
If the material is much larger than your model, selecting the right project's starting location will enable you to control the location of the model in your material (see [Machine home position vs. the project's starting position](#)).

### Location of the model in the material

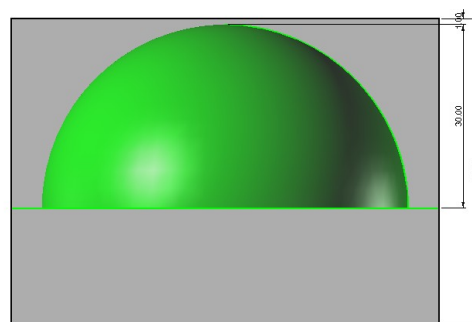
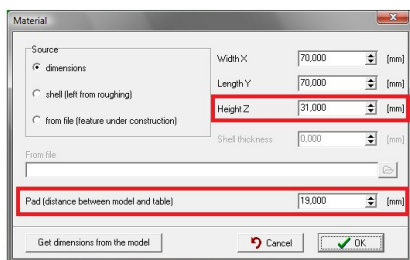
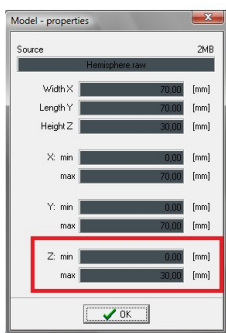
Should your material be much higher than the height of your model, you can control the height at which the model will be cut:

- By entering the total material height as its Z height, the model will be cut at the bottom of your material. In this case, the roughing operation will tool off all of the excess material off the top.
- In order to have the model cut at the top section of your piece of material, break the total height of your material into two values: material height and pad thickness. Insert the model height as your material height, and the remaining part of the actual height of the material insert as the pad thickness. It is important the combined height of the pad and material do match the actual height of your material.

Example: in this case, the model is 30 mm high, and the material is 50 mm high.

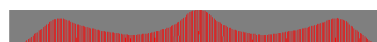
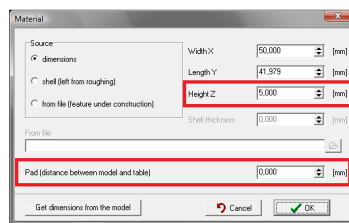
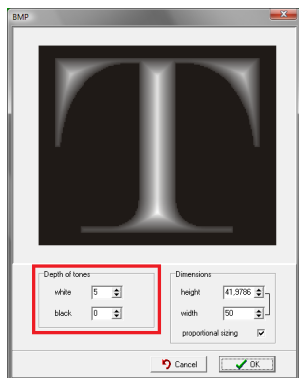


The second option is more optimal in most cases. It is recommended a 1 mm layer is left on top.



### Location of the model in the material vs. bmp heights

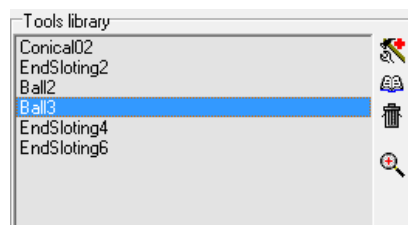
The bmp files give you even more choices as you can additionally modify the values for black and white.





## Tools library

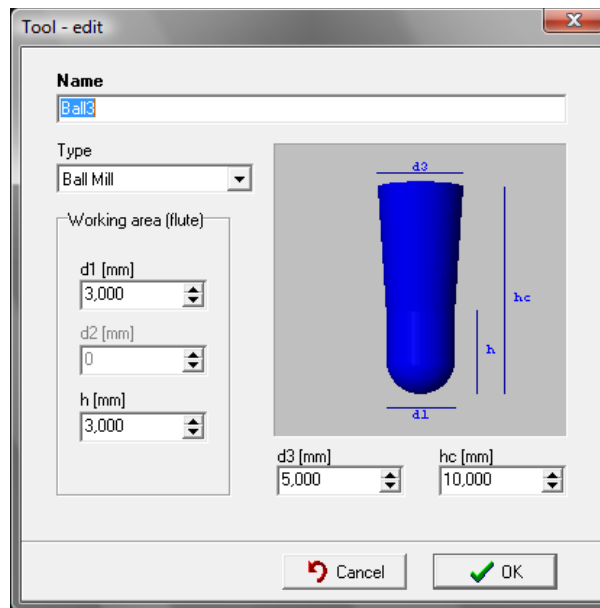
The tools library is the collection of tools used by the machine for given operations. Note the tools are sorted by their diameter, not name.



There is no limit on the number of tools you can save, giving each of them a name you can easily recognize.

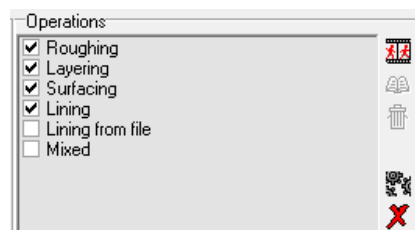
You can define three different types of tools: flat, ball and conical. Flat tools are mostly used for the roughing operations as well as for surfacing operations. Ball tools are most often used for the finishing operations and the conical tools are mostly used for very fine details only.

On the right side of the tool library there are buttons which allow to add, edit and delete tools from the library. When adding a new tool, make sure you specify all of its parameters. Note the number and types of parameters that need to be specified depend on the type of the tool.



## Operations

3D tooling requires a number of operations: almost always the roughing operation and at least one finishing operation. You can define an unlimited number of operations in the software under a name of your choice. When ready to generate the output files, select the required operations and proceed.



There are buttons on the right side of the operations panel which allow you to add, edit and delete operations. The generation of the output files will be discussed later on.

The software allows you to define 6 different types of operations:

- **roughing** – always used as the first operation to get rid of excess material as quickly as possible. Typically, after this operation you will have a thin layer of material left for the following finishing operations.
- **lining** – a finishing operation in which the tool moves in parallel lines over the 3D model. Distance between the lines as well as the required angle is set in parameters of this operation. The output file covers the entire surface of the model (as opposed to *lining from file*).
- **lining from file** – a finishing operation in which the tool moves in parallel lines over specified areas of the model specified by the user. It is most useful for spending extra

time over specific fine details of a model, without wasting time over less important, or simpler, area. For this operation the user has to prepare a vector plt file consisting of a frame of the same equal to the size of the model and lines over the areas be tooled. The distance between the lines needs to correspond to the tool diameter.

- **layer** – a finishing operation in which the layers are worked on from the top to the bottom. It gives best results for steep walls, not so much for wide area convex areas. That is why sometimes a follow up lining operation may be required. This operation skips flat surfaces, if the model consists of such, the surface operation may be more suitable.
- **surface** – a finishing operation for surfaces, often used to compliment the layering operation. The algorithms qualify surfaces to be tooled based on their area in relation to the area of the entire model. Hence small surfaces of a large model may be skipped in this operation.
- **mixed** – a finishing operation, similar to the layer operation as it moves from top to the bottom. However, here the tooling path is based on spirals and not layers. As opposed to the layer operation, the mixed operation also includes the surfaces.

The choice of operations always depends on the model. The most common combination is the roughing plus lining one.

The number and type of available parameters of an operation depend on its type.

Parameters description based on the roughing operation:

- *Tool* – select a tool from the tools library (see [Tools library](#))

- *Layer distance* – distance between two neighboring passes in the X/Y layer
- *Layer depth* – depth of consecutive layers (Z axis)
- *Soft start* – applies to the roughing operation only, used when tooling hard materials requiring slow entry into the material
- *Leave for next operation* – the thickness of material to be left over the model for the following finishing operation. In the roughing operation this is usually set to 0.5 mm, while in the finishing operations it is most often set to 0 mm.
- *Speed* – the speed at which the tool moves in the material. This settings needs to take into account: type of material, tool type, thickness and depth of layers. This setting can be adjusted during the cutting process in Manual Control or with the remote control.
- *Movement speed* – speed at which the machine moves over the material, can usually be st to the max. machine speed
- *Drilling speed* – speed at which the tool enters the material
- *Rotations* – spindle rotations, any value in the range: 3000-24000 rpm. This setting can be adjusted during the cutting process in Manual Control or with the remote control
- *Z axis lower limit* – limits the tooling below a certain height. The default zero setting means there is no limit. But e.g. 3 mm will mean the operation will skip the tooling at the lowest 3 mm of the project. This features is inactive for lining operations.




- *Z axis upper limit Z* – limits the tooling above a certain height. Default setting of 170 mm (higher than material size) means no limit. But e.g. a 25 mm setting means the top 25 mm of the material will not be tooled. This features is inactive for lining operations.

Other operation-specific parameters:

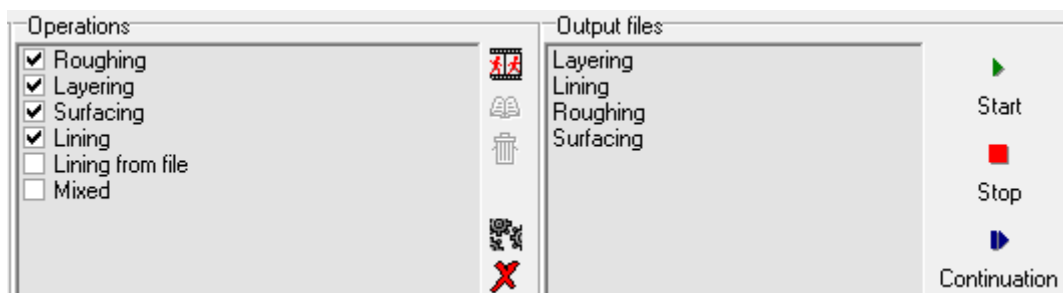
- *Connected layers* – in the layer and mixed operation, this setting gets rid of unnecessary up movement when moving from one layer to another
- *Angle* – in the lining operation, this settings enables the operator to set any required angle for the lines in the xy axes
- *Connected ends*– in the lining operation it allows for two-way operation. When activated, the machines moves along the lines in both directions. When off, the lines are cut in one direction only, which means after each line the machine goes up and moves over the material to the beginning of another line. There are materials which can only be tooled one way.

## Output files

Output files are the tool paths generated by the software based on the operations. Once an operation (or a number of operations) is defined, click *Generated output files*  or *select Tooling \ Generate output files F4*.

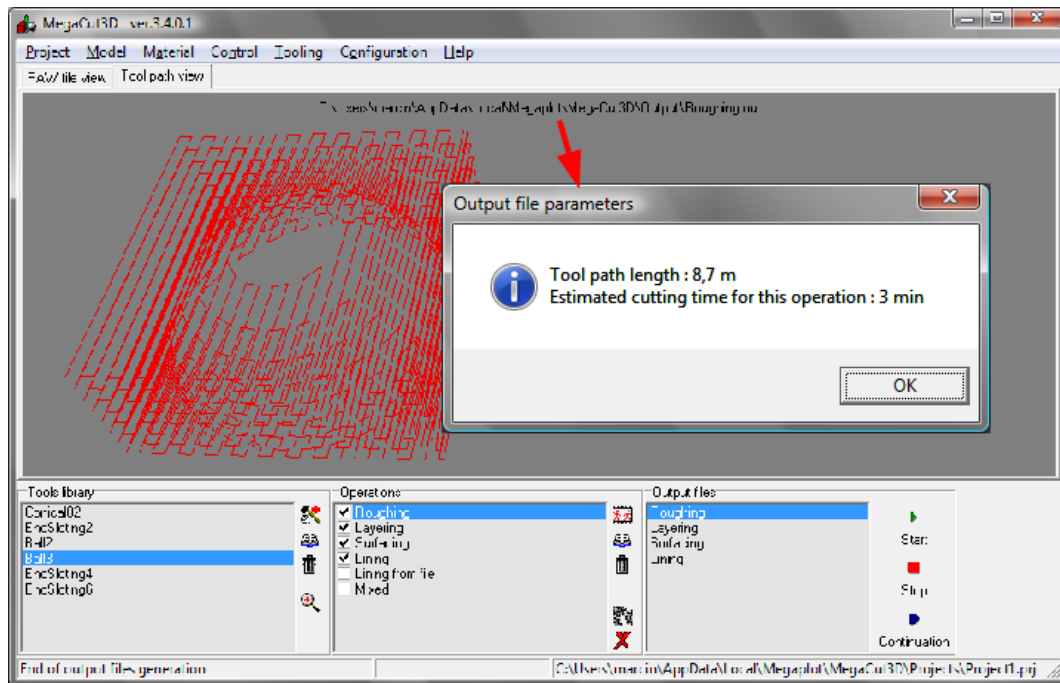
When generating new output files, all previously generated output files will be lost. Since the output files generation can be a time consuming process, there are two good solutions to this problem:

- you can generate the files one at a time; once generated, start the cut and start generating a new one.
- generate all output files at the same time and then start cutting them one after another.



The time it takes to generate an output file depends on the size of the 3D model, the resolution set in configuration as well as the PC hardware. Limitations re. 3D models as well as PC requirements are listed here: [REQUIREMENTS AND LIMITATIONS](#) . Increasing the resolution from the default resolution of 5 points/mm to a higher one will significantly increase the time needed to generate the files.

During the output files generation process, the status bar shows the name of the currently processed file as well as the progress bar. As the process completes, new output files will show up one after another on the output files list.



A double click on the output file path shows such parameters of the output file as total path length and estimated cutting time.

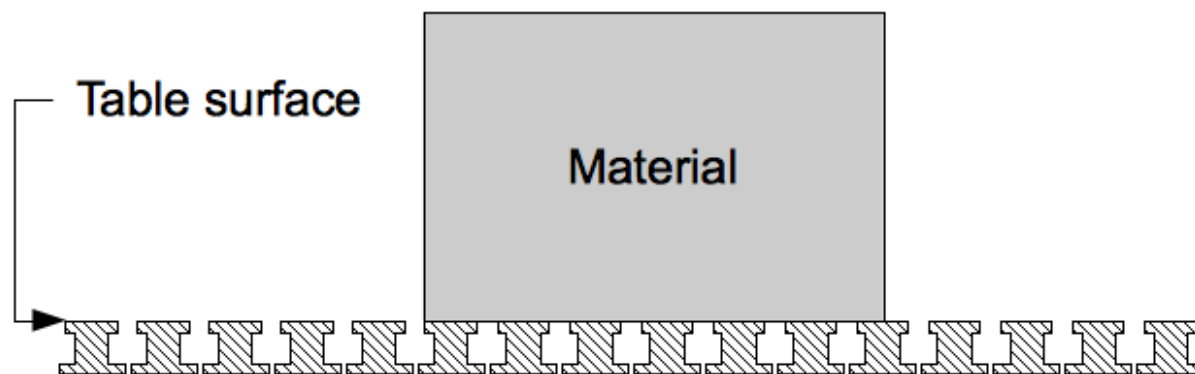
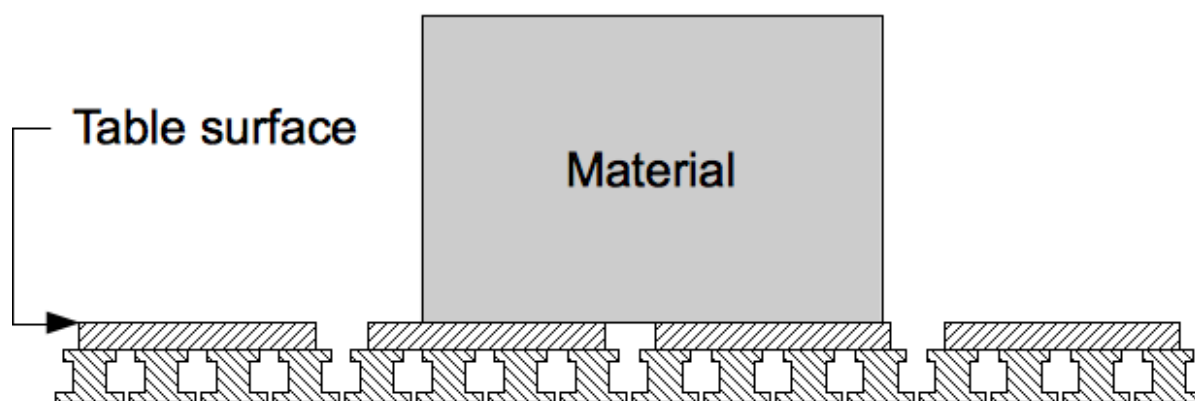
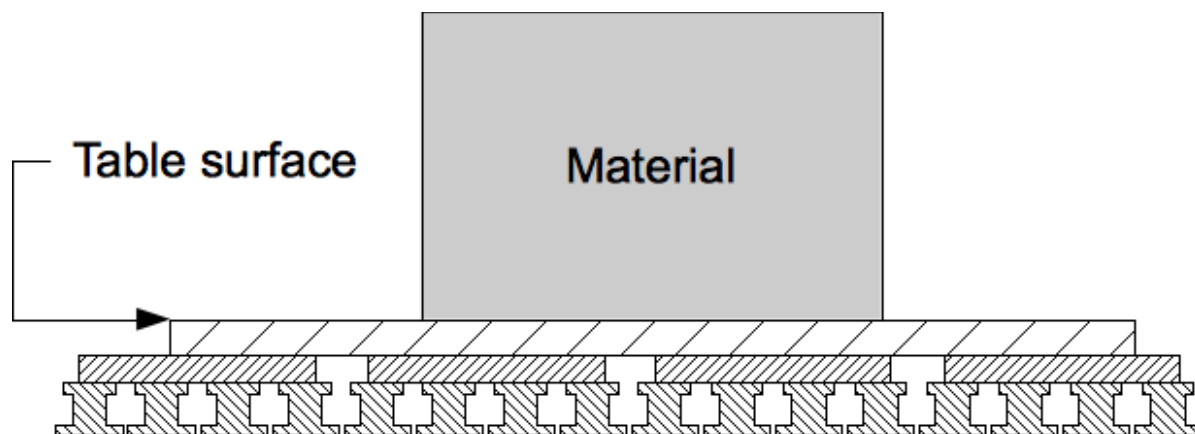
## Tool measurement

The tool length needs to be measured every time a new tool is installed. It determines the distance between the tool and the machine table.

For 3D tooling it is recommended the tool length is measured on the table level.

Measuring the tool length on the material is not a good idea as after the first operation the top layer of the material will be taken off and there will be no reference for subsequent measurements.

Note the table level here is an abstract idea, as it simply refers to the surface to which the material is fixed. So if the material is fixed to a pad, then this pad can be used as the level on which the tool is measured.

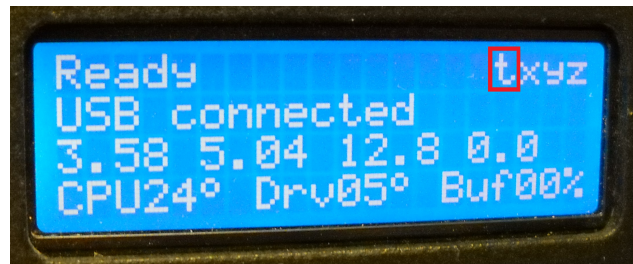


Before you measure the tool length:

- make sure correct pad thickness is set in Configuration (Configuration \ Ploter \ Pad

thickness) [this applies to automatic measurement].

- make sure the down movement of the spindle at the given location will be possible and the tool will be able to reach the plate without problems.
- Make sure the tool will be able to reach the lowest location of your project. If, e.g. the spindle is installed high and the tool is short, it may not be possible for the tool to reach the measuring plate or the table, in which case the spindle may reach its lowest position and attempt – without success – further down movement. In this case, stop the measuring process, home the machine, correct the problem (lower the spindle or use a longer tool) and repeat the process.
- Ensure the measuring plate works properly [apples to automatic measurement].



Touch the spindle with the measuring plate. If the small letter *t* turns into a capital *T*, it means the automatic tool length measurement is working properly.

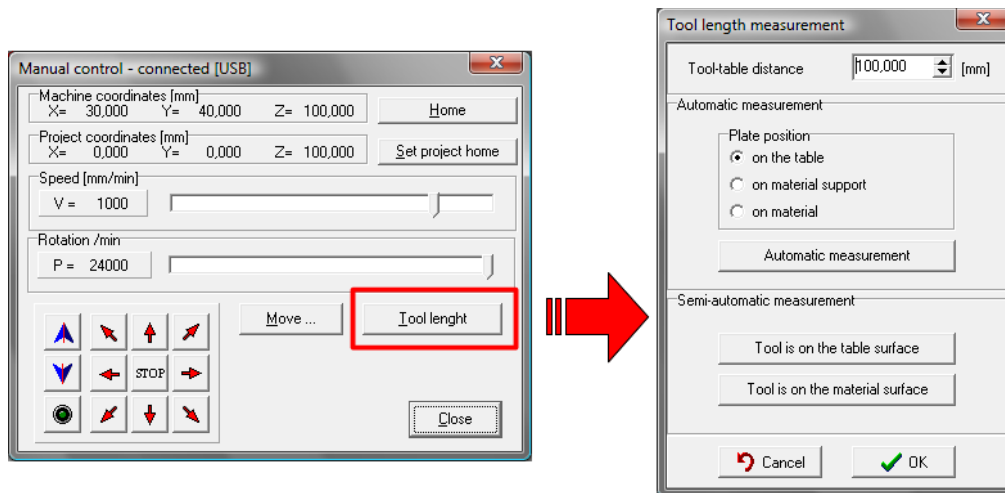
## Automatic tool length measurement

The automatic tool length measurement is done with the measuring plate described above. The spindle will slowly move down until it tool reaches the plate. Once done, the tool length is automatically saved and the spindle will move back up.

To ensure the measurement is precise, make sure the measuring plate is clean, the surface under the plate is clean and flat and the pad thickness is specified properly in the software (Configuration \ Ploter \ Pad thickness).

To measure the tool length on the machine table:

- move the spindle to a desired location
- prepare the measuring plate



- go to *Manual Control \ Tool length*
- press *Automatic measurement*
- message will show up: “*Place the pad under the tool*”
- place the plate under the tool
- confirm by clicking OK in the software
- wait till the tool touches the plate and the spindle moves back up
- put away the measuring plate
- confirm by clicking OK in the software

If you put the measuring plate directly on the aluminum table of the router, any small movement of the plate may prematurely end the measuring process. To avoid this, put a thin piece of isolation on the bottom of the measuring plate. Make sure you remeasure the plate thickness and insert the correct value in the configuration.

**CAUTION!** Always place the plate with the insulated side down. Otherwise, the measuring process will not work properly and may result in breaking the tool, damaging the router or injury.

### **Semi-automatic tool length measurement**

Alternatively, you can measure the tool length in the following way:

- move the spindle to the location of the measurement
- open *Manual Control* and *Tool Length*
- insert 999 for *tool length* (to allow for unlimited movement down)
- move the spindle down very slowly until the tool reaches the table (make sure to do it

very slowly to avoid breaking or damaging the tool

- *open Manual Control and Tool Length*
- press “Tool on table level” or “on material level”
- move the spindle up

If the new tool is shorter than the previous one and you use the semi-automatic way to measure it, the spindle will not reach the table level as it is impossible to move the spindle into the negative co-ordinates. In this case, enter the 999 values as described above.

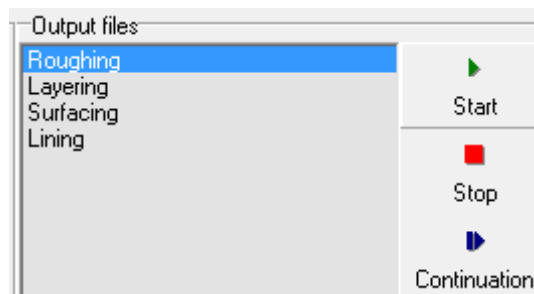
When the router is switched off and back on again, the last tool measurement is remembered. If the tool has not been replaced in the meantime, there is no need to measure it again.

## Cutting

Before starting a cut:

- make sure the spindle cooling system works properly and is turned on,
- move the spindle all the way up, as the moment you press Start the spindle will move to the project's starting position so it needs to be able to move above the material.

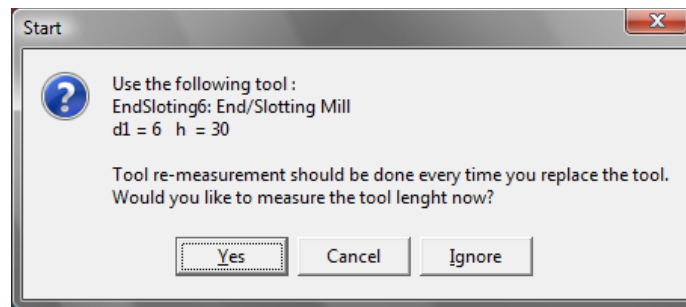
Once you set the project's starting location, set the material size, measure the tool length and generate the output files, you are ready to start. Select one of the output files and click Start.



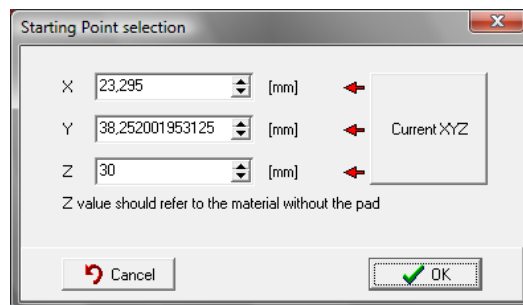
Always start with the roughing operation followed by one or more finishing operations (the lining operation usually proves most useful).

Usually, you will need to replace the tool between operations. Once you replace it, make sure to perform the tool measurement procedure. You will get a warning message before the next operation starts to make sure you do not forget to change and measure the tool.

You can stop the cutting process at any time by clicking STOP. If it is necessary to clean the tool, you can move the spindle to any location on the machine and once the tool is clean, press CONTINUE or Shift +F2 and the router will move back to the location where the cutting process was stopped and will continue from this spot.



In the case of a power line problem, you will not be able to use the Continue feature. Instead it will be possible to restart the cut from any particular location on the 3D model by selecting Tooling - Start from a selected point CTRL + F2. All you have to do is select the correct output file and move the tool as close as possible to the place you wish to start. the software will automatically find the closest matching node and will continue from this location.



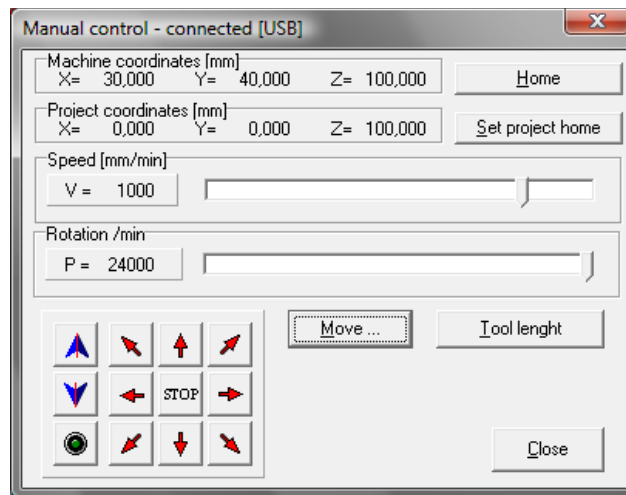
The tooling process is based on the parameters set in Configuration but some of them - such as spindle rotation and cutting speed - can be adjusted any time during a cut by means of sliders in Manual Control or with the wired remote control. (see [THE WIRED REMOTE CONTROL](#)).

## Manual Control

The manual control window allows for a number of manual operations, including the control of the machine in each axis, as well as:

- activating and deactivating spindle rotations – **always make sure the spindle cooling system is on before you turn the spindle rotations on,**
- homing procedure,
- setting the project's starting position,
- moving in any axis by a given distance to absolute co-ordinates

The control window shows the machine co-ordinates (in relation to its absolute home position) as well as the project's co-ordinates (in relation to its starting location). Note the remote control display only shows the project's co-ordinates.



This window can display in the alternative, touch-friendly mode as described earlier ([Control Window - enlarged](#)).



## Homing process

Homing (Manual Control – Home button) is used to bring the machine back to its original home position by calibrating it in relation to the proximity sensors. It is recommended to perform the homing process when the machine was blocked while moving or the tool got stuck inside of material because of wrong cutting parameters. If the machine gets stuck in the material or by hitting an object on its table, you will hear the sound of the motors trying to overcome the obstacle. In this case, home the machine to bring it back to its original home position.

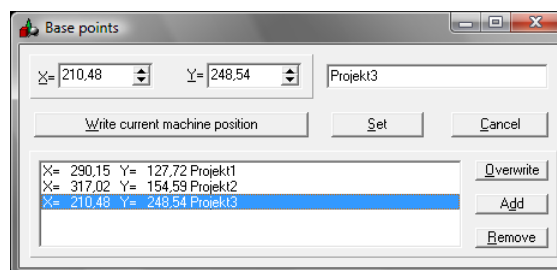


To ensure the homing process is performed properly and precisely, ensure the proximity switches as well as the lead screw covers are clean and allow for free movement (see [MAINTENANCE RECOMMENDATIONS](#) ).

### Project's starting location

The difference between the machine home position and the project's starting locations was described in [Machine home position vs. the project's starting position](#).

To set a new starting position for a project, move the spindle to the required location, go to Manual Control and select *Set project's starting location*.

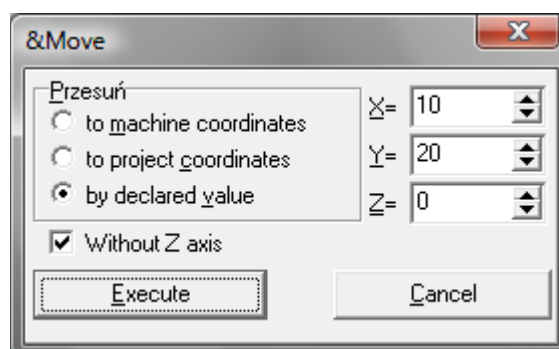


Next, press the *Insert current machine location*. Note this location will be remembered even after the controller has been turned off and back on.

You can save any number of project's starting locations, name them and go back to them in the future as required. Just select the one you wish to set and click Set.

### Move to and Move by

Apart from manual movement in the Manual Control window, you can also move the machine precisely in each axis by using the Move... window. There are three choices available as shown below:



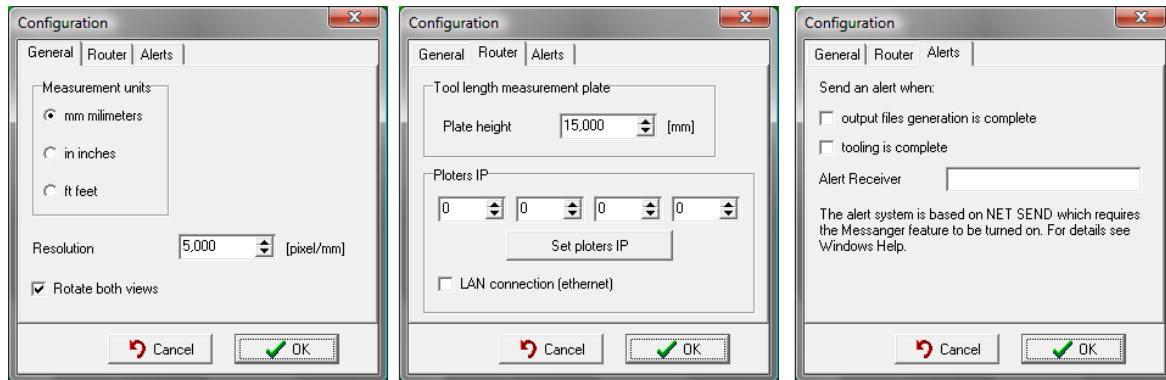
Before you execute the movement, please make sure to bring the spindle up so the tool does not run into the material or the material holders.

You can use this to check the location of last project's starting location. Simply select *To*

project's co-ordinates and insert 0 for all three axes.

## Configuration

All the main parameters are set in the three tabs in the Configuration window.



### General tag

*Resolution* - preciseness in points/mm at which the algorithms calculate the output files. The default value is 5 points/mm i.e. the accuracy will be 0.2 mm. Increasing this value will significantly slower the output files generation.

*Connected preview windows* - when on, rotating the 3D model will result in identical rotation of the output files.

### Ploter tab

*Tool Lenthg Measurement Plate - Pad Thickness* - real thickness of the measuring plater

*Set ploter IP* - saves the inserted IP into the electronic controller. USB connection is required for this operation.

*Network connection (ethernet)* - activates the network connection

### Notifications tab

MegaCut3D can send a notification to another PC on the same network at the following events:

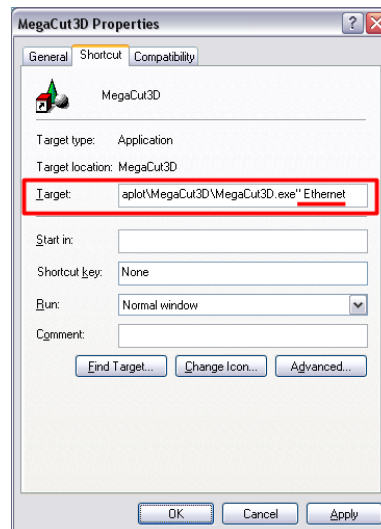
- output files generation completion
- tooling process end.

These notifications are only available in Windows XP as they use the messenger service.

Recipient - name of the user or PC on the local network

## Network connection (ethernet)

The standard controller-PC is done via USB connection, yet ethernet communication is available as an optional upgrade. The applicable options will be visible in the configuration only after the correct parameter is set in the MegaCut icon shortcut properties.



In order to control the machine via ethernet connection:

- connect the controller to the PC via standard USB connection
- right click on the MegaCut shortcut icon on PC desktop and modify the path by adding a space at the end and adding the parameter *ethernet*. The target path should now look as follows:
- "C:\Program Files\Megaplot\MegaCut\frez\_mdi.exe" ethernet
- start MegaCut
- go to Configuration and the Ploter tab
- *enter the ploter IP* for the machine (it needs to be unique on your network)
- press the ploter IP button. The IP should now show up on the controller's display
- activate the network connection (ethernet) setting
- confirm by clicking OK
- close MegaCut

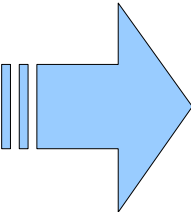
- disconnect the USB cable from PC and the controller
- connect the controller to your ethernet network
- open MegaCut, if the connection works properly, "*Ethernet connected*" will display in the bottom status bar as we as in the top bar of the Manual Control window

## Router calibration

The calibration feature is used when the machine moves in one axis incorrectly, i.e. too much or to little, and if the error is proportional along the entire length of the axis. You can set the calibration adjustment for each axis separately.

In order to calibrate the machine in this way, create a text file named *kalibracja.txt*. The file should have three lines, and each line should have the calibration value for the given axis: X, Y and Z np.:

Kalibracja.txt	
1.003	increases movement by 3mm in X axis
0.998	decreases movement by 2mm in Y axis
1.000	no calibration in X axis



For axis not requiring calibration, set the value to 1.0. This calibration files should be placed in the main MegaCut folder where the *frez\_mdi.exe* is located.

The calibration must be within the range <0.99; 1.01> so it can correct errors of +/-10mm.

The calibration value can be calculated in the following way:

$$\text{calibration value} = \frac{\text{actual distance}}{\text{required distance}}$$

## REQUIREMENTS AND LIMITATIONS

The router must be placed on hard and perfectly level surface.

The PC and the electronic controller must be connected to the same electric phase. Both should be plugged into the same surge protector. The power line must be properly grounded.

Both MegaCut and MegaCut 3D can be installed on the following Windows OS: 2000, XP, Vista or 7.

In order to avoid interference in the constant PC-controller connection, please turn off the following in your PC:

- all power saving modes,
- screen saver

Max. drawing/model size

- 2D - the only limitation is the working are of the machine
- 3D - limited by the PC hardware (memory plus CPU). If the PC is sufficiently strong and fast, 3D models of up to 50x50 cm should work fine (sometimes up to 1x1 meter). It is highly recommended to equip the PC in as much RAM as possible (at least 4GB) when working with 3D models.

The supplied software is only used to control the machine, you will still need your own graphics software to prepare the shapes/drawings/models.

You cannot start but MegaCut and MegaCut 3D at the same time, only one should be opened at any given time.

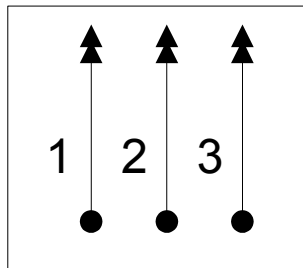
# ALGORITHMS

## Lining algorithms

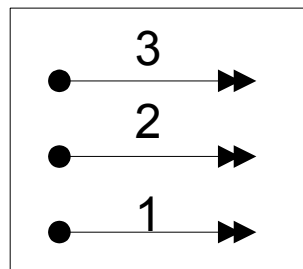
This description applies to the lining operation available under Functions - Lining in MegaCut.

1. There are four basic types of lining:

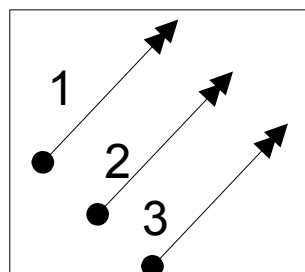
- vertical (HPATTERN\_VERTICAL)



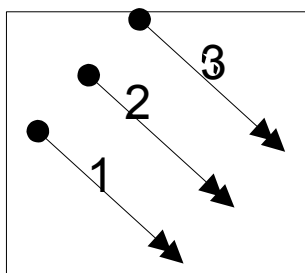
- horizontal (HPATTERN\_HORIZONTAL)



- forward slash (HPATTERN\_FORWARD\_SLASH)

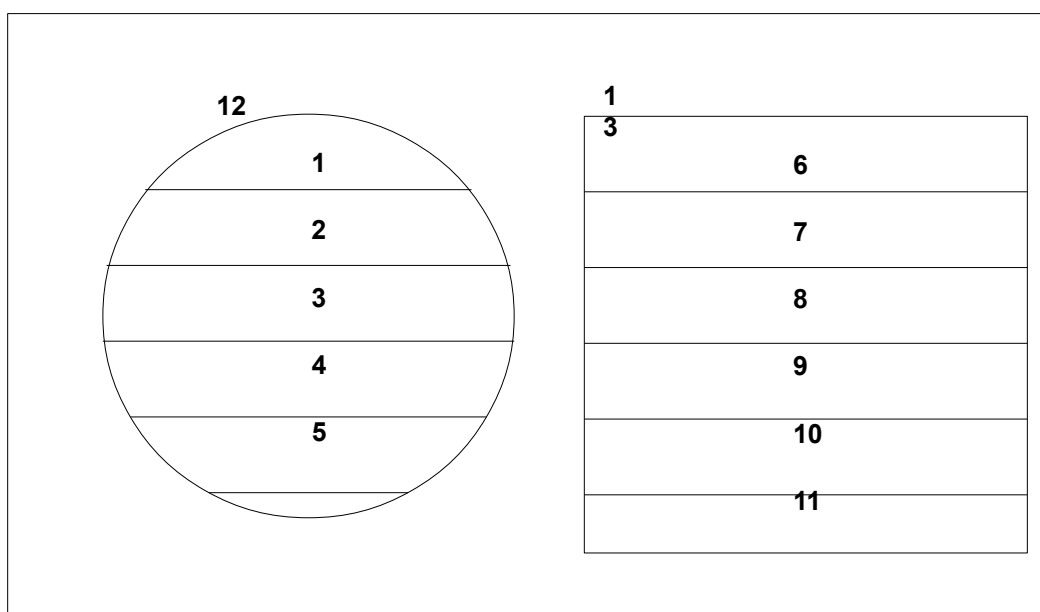


- back slash (HPATTERN\_BACK\_SLASH)

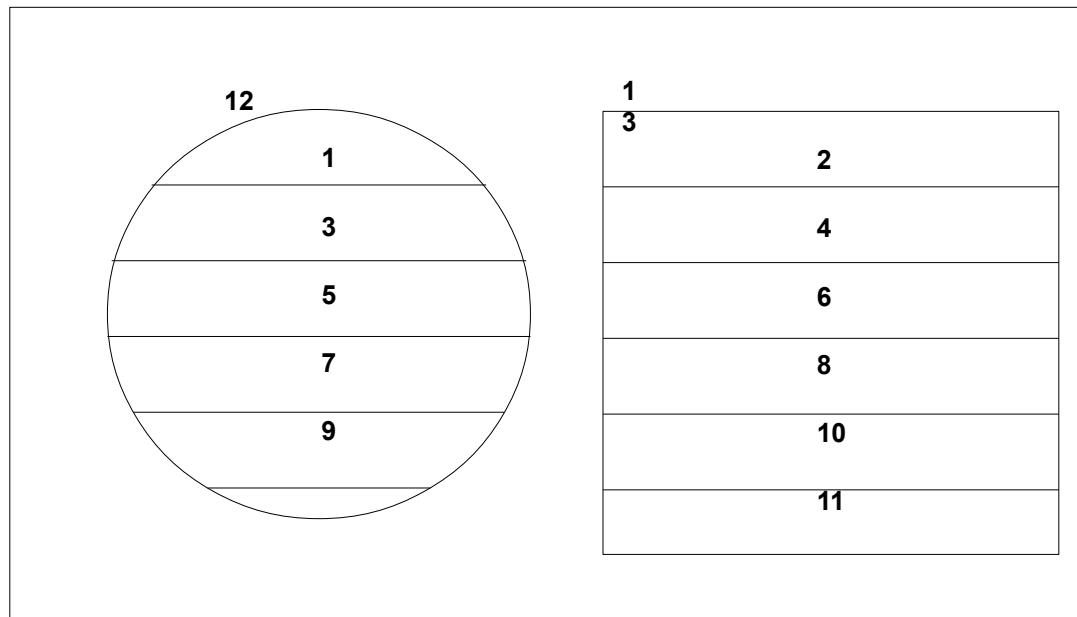


2. Sequence of cutting of lines and shapes:

If *All elements of a shape* is active, first the machine will cut the lines and then the outlines.



If this feature is off, first all the lines in the entire drawing will be cut and next the outlines as set in the Cutting Direction setting.



Should you wish to first cut all the lines in a drawing first and then all the elements of a given shape:

- cut only the lines, no outlines, and deactivate “All elements of a shape”,
- cut the outlines with “All elements of a shape” activated

## Overlapping shapes algorithms

You can have all overlapping lines deleted when loading a drawing in MegaCut by selecting “Delete overlapping lines” in the settings. For a shape to be considered an overlapping one and deleted, it has to have the exact same number of nodes with the exact same values for each node. They need to have the starting and ending nodes in the same spots.

Note that if the *Connect the nodes* feature is on, the software will connect all the nodes prior to attempting the overlapping lines deleting. This means all open shapes will turn into closed ones and as a result they will not be deleted. This may result in certain shapes to be cut twice. To avoid this, turn off the nodes linking procedure or delete all overlapping lines back in your graphics software.

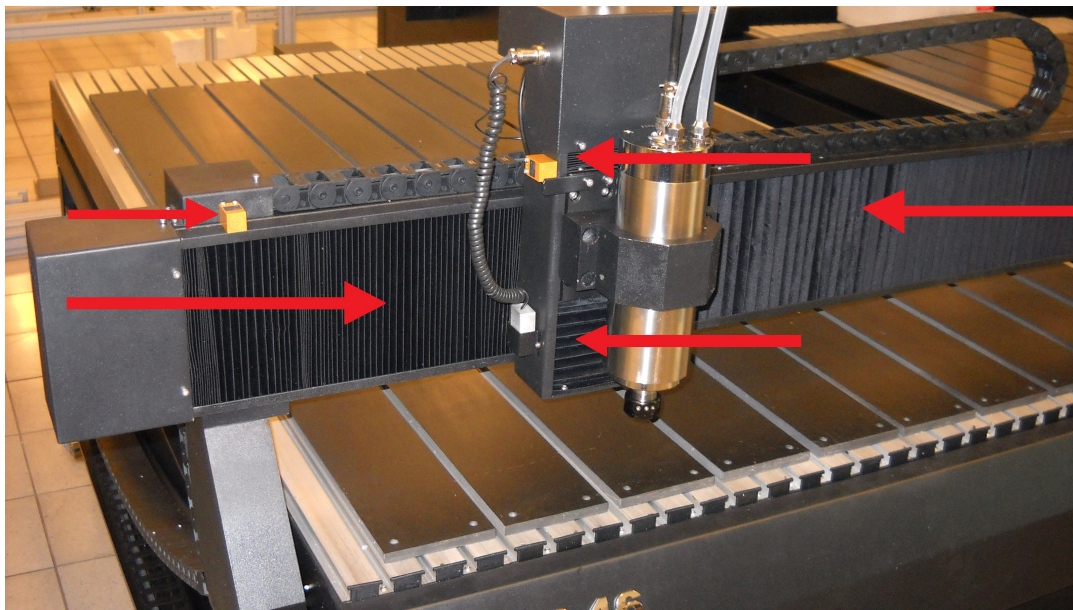


## MAINTENANCE RECOMMENDATIONS

Always observe all applicable safety laws.

Keep the ball screws and linear guides clean. Dust them off and grease on a regular basis.

Keep the proximity sensors and screw covers clear. Dirty screw covers may prevent the machine from homing properly as the portal may be blocked before it reaches the proximity sensors.



Keep the machine clean of any material chips. It is recommend it to clean the machine off with compressed air on regular basis. Do not use a vacuum during the router's operation as the electrostatics may interrupt machine operation.

Use distilled water for the water cooling system. The water pump should be completely underwater at all times. Make sure none of the pipes feeding the water are bent or damaged.

To avoid overloading the spindle, always use tools suitable for given material. Set the important parameters such as speed, rotations, width and depth of passes bearing in mind the material properties and the used tool.