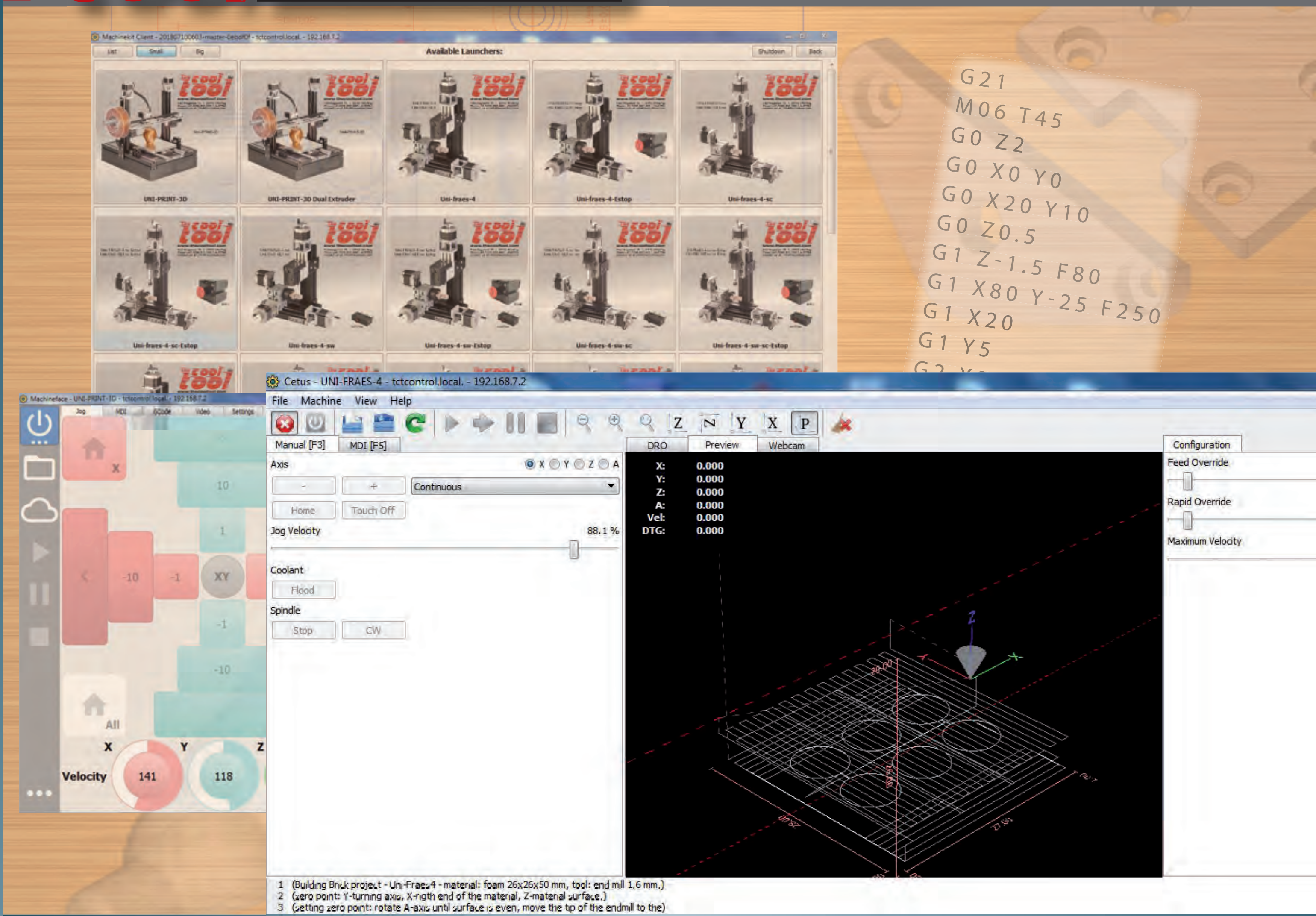
VSTVETCNC1_E-01.03.2019_1

Education



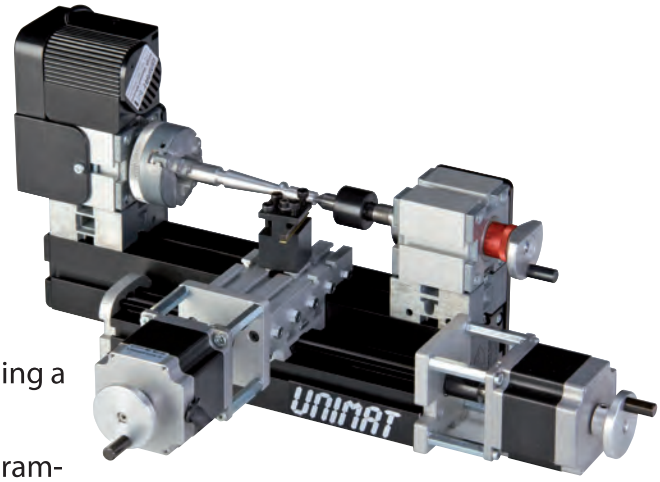
[TVET-CNC-1]

QUALITY PRODUCT MADE IN AUSTRIA

4.0 Unimat in TVET – CNC

A Practical Approach to G-Code Programming and CNC Machining

From precision turned parts to machine engine blocks – production of such items would not be possible without automated production technologies. Computer numerical controlled (CNC) machine tools are utilized to manufacture identical parts, with high accuracy in large quantities.



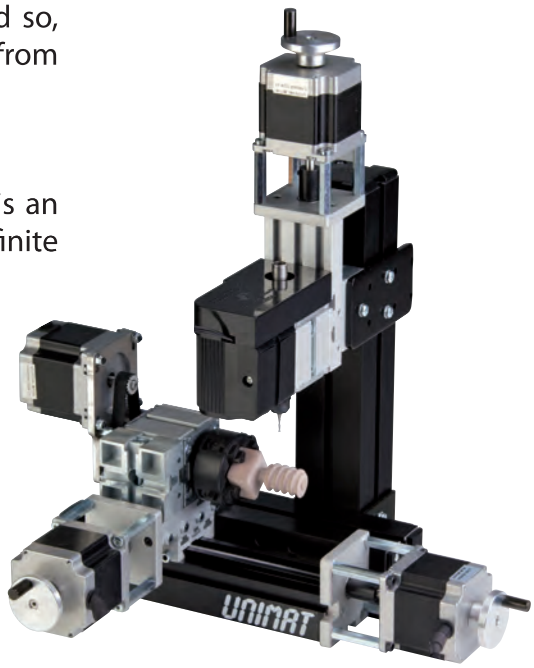
How do we control machines? How do we instruct them where to move, along which path and at what speed?

Such instructions can be written in a widely used NC programming language, commonly called G-code. A syntax of G-codes, M-codes and letter addresses determines motion and actions of a machine.

Why learning to program machine code instead of simply using a CAM software to generate G-code automatically?

By first learning at least the very fundamentals of this programming language one will be able to write the instructions for simple geometries (and thus understand the code a CAM software generates), make small changes to the program, detect errors and so, most importantly, use this knowledge across different CNC systems from various suppliers.

Especially for innovative education in higher grades Unimat CNC is an interesting option. Due to the modular construction an almost infinite number of application possibilities arise.



Some of the characteristics are:

- Cross Slide: travel 50 mm (also available in 80 mm)
- Longitudinal Slide: travel 145 mm (also available in 445 mm)
- Accuracy: max. 0.07 mm
- CNC-Mill – up to 4 axes

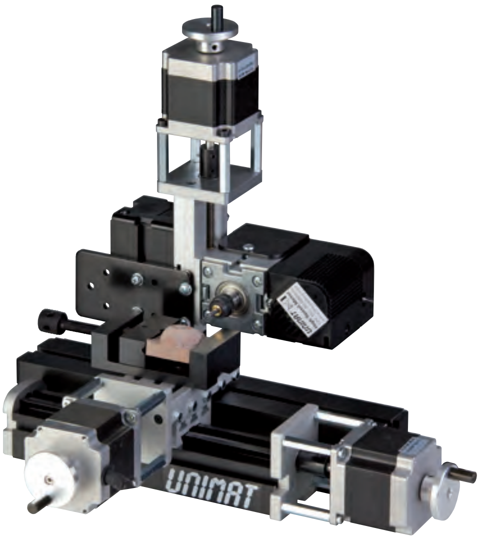
The gap between practical profound training and permanently more restricted budgets, can be bridged with Unimat CNC. Developed for prototyping and model making, this system also provides a lot of advantages for the educational sector. While purchasing industrial machines is expensive and ongoing expenditures for maintenance etc. are high, Unimat CNC allows for an entire machine park at only a part of the cost. Students can now use these machines actively and autonomously.

Opening up new possibilities of using CNC machinery in education

Course of Action

1. Introducing different machine types and illustrating the coordinate systems

The coordinate system of a CNC machine is classified as X, Y, Z for the linear and A, B, C for the rotational axes. In most cases a rectangular Cartesian coordinate system applies.



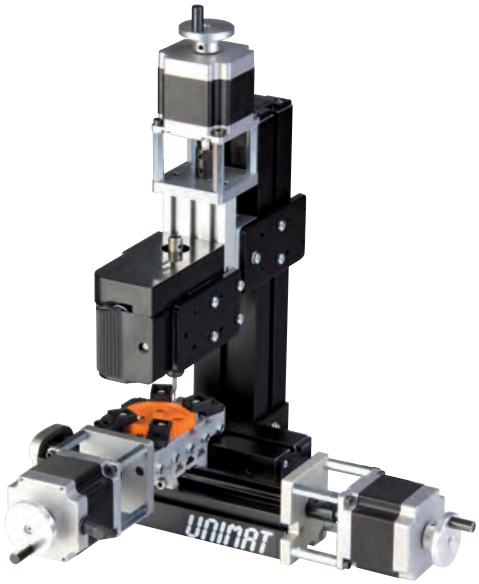
2. Practicing some simple G-Code commands and syntax

With a set simple commands like G0 and G1 for linear motion, G2 for clockwise circular arcs and

F for feed rate quite some outlines of simple geometries can be programmed.

3. Iteratively writing the code for a simple geometry and simulating it in the CNC software

To be loaded, displayed and simulated in the CNC software the code might have to be adapted, certain commands added.

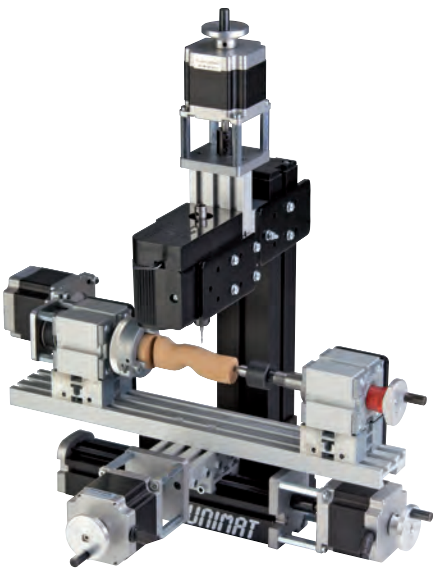


4. Operating the CNC machine and processing the first workpiece

Once the simulation has been executed error free and certain parameters, like safety height, feed rate etc, are in line with machine, tool and material features and the zero point has been set correctly the program can be executed on the machine.

5. Using simple drawing, CAD and CAM

Finally, it‘s time for some more advanced designs, 2D drawings or 3D models, for which the machine code will be generated by use of Computer Aided Manufacturing (CAM) applications.



„One only apprehends what one can make himself, and one only conceives, what he can create himself.“

Johann Wolfgang von Goethe, free translation

Opening up new possibilities of using CNC machinery in education

• Profound expert knowledge

Everything learned on these machines can also be applied on big ones. There are no specific details used here that would be invalid on larger projects.

• Easy handling, commissioning and maintenance

The simple and modular set up enables a fast insight into the functionality of the machines, thus utilizing them does not require much prior knowledge. Service can be done easily by anyone on-site.

• Use is location-independent, no special environment or furniture needed.

Due to the small weights and sizes, the machines (supplied in a handy transport container) can be handled, set-up and stored easily.

• 2 year guarantee period

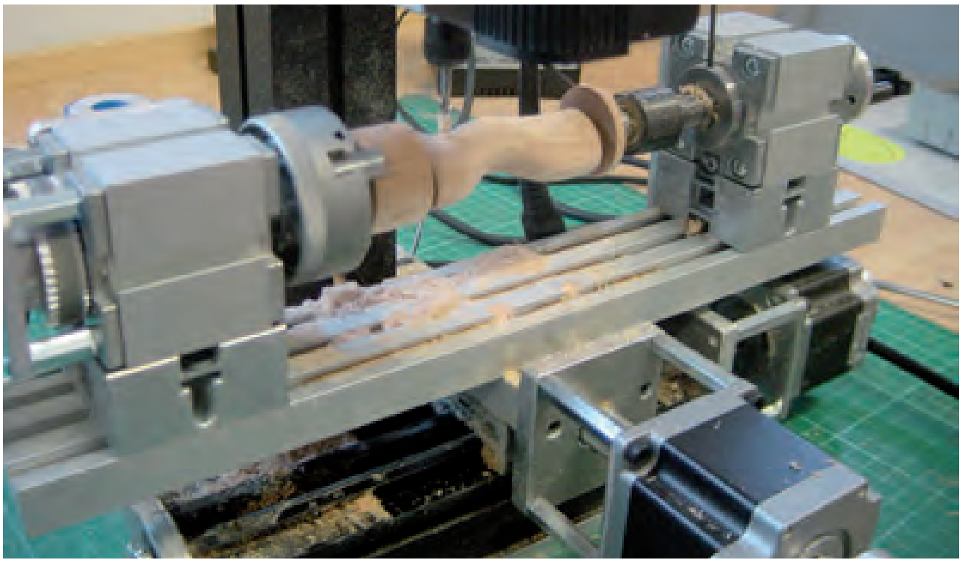
The machines are Austrian quality products with a 2 year guarantee period (wearing parts excluded).

• Professional controller software - without licence fees

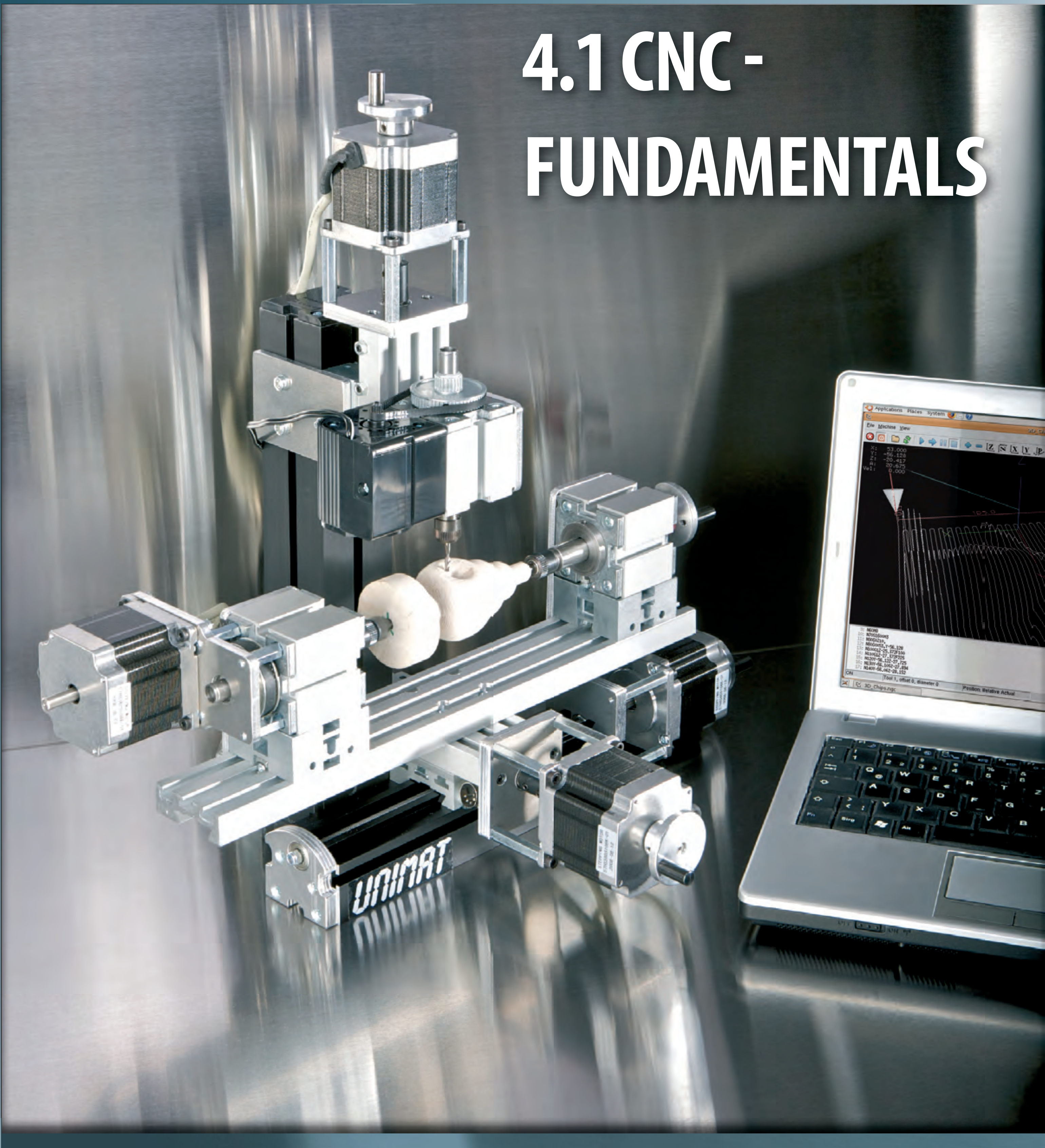
Machinekit (a fork of LinuxCNC), pre-installed on TCTControl, is an open source software system for computer control of machine tools such as milling machines, lathes and many others using ISO standard (RS-274NGC) gcode as input. Upgrades will be available free of charge.

Project: furniture construction, "ergonomic handle"

Mag. Radatz, workshop for interior design at HTL Mödling
AutoCAD design, G-code generated with CAM software Deskproto.
Workpiece beech wood, 30mm dia. x 95mm, Unimat CNC 4-axis mill.



4.1 CNC - FUNDAMENTALS



QUALITY PRODUCT MADE IN AUSTRIA

Advantage and challenges of automation 4.1.1

Advantages

- Automatic reproducibility, repeatability
- Productivity increase
- Flexibility of production
- Reduce tool wear (constant conditions)
- Constant manufacturing cycles (Production planning)
- CNC machines can be connected with each other (manufacturing systems)
- Great diversity of processing options
- Members can control several machines simultaneously reduced labor costs
- Complex parts easily duplicate, one-time programming
- Automation improvements via robot integration, conveyor belts etc.

Challenges

- High investment (software, machines and tooling)
- CNC specialists for maintenance and running of the machines are required
- Development requirements for planning department (CAD/CAM/CIM)
- Costly individual parts due to programming, set-up

CNC – Computerized Numerical Control

In CNC, machines are operated via numerical control, wherein a software program is designated to control an object. The language behind CNC machining is alternately referred to as G-code, and it's written to control the various behaviors of a corresponding machine, such as the speed, feed rate and coordination.

CNC machining makes it possible to program parameters like speed and position of machine tool functions and have a software executing them in repetitive, predictable cycles. Due to these capabilities, the process has been widely adopted in manufacturing industry and is now indispensable in many production areas.

First, a 2D drawing or 3D model is constructed in a CAD software. In a next step a CAM software generates tools paths and other information which is then translated (with the help of a post processor) to machine code for the CNC controller to execute. Once program has been loaded, simulations and a trials can be run to test the code before finally processing it on the machine.

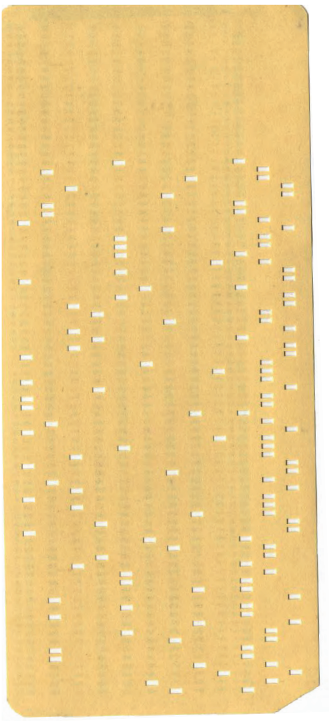
Examples of CNC machines:

- Lathes for turning and cylindrical contouring operations.
- Milling machines - from simple 3-axis machines up to 6 axis machines
- Routers and engravers
- Laser Cutters for cutting sheet material

History of CNC -> NC – Numerical Control

The first NC machines were built in the 1940s and 1950s, based on existing tools that were modified with motors that moved the controls to follow points fed into the system on punched tape or cards. These early servomechanisms were rapidly augmented with analog and digital computers, creating the modern CNC machine tools that have revolutionized machining processes.

punch card

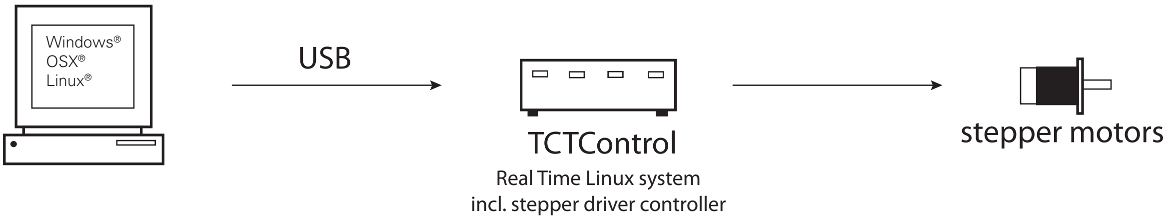


punch card stamping press

The cool tool CNC control systems 4.1.3

TCTControl - CNC machines •
SandyBox - stepper driver controller - CNC machines

TCTControl - [current solution]



Functions of PC:

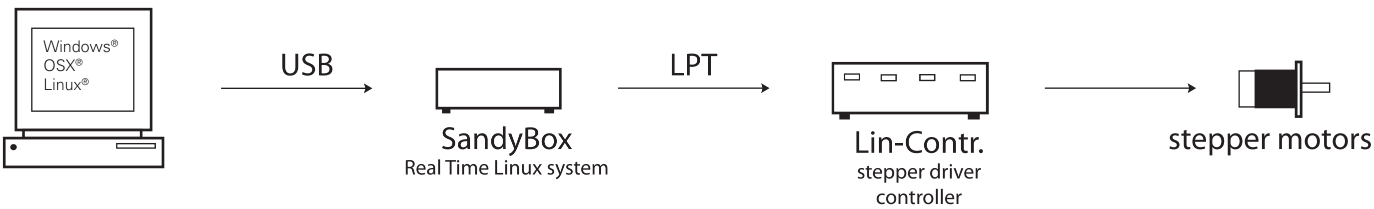
1) User interface (type your own G-Code or open G-Code files)

2) „Manual" control of the machine

TCTControl box:

Control electronics including motion control and pre-installed CNC software
Technical Specifications: Client and drivers included for
Win (7, 8, 10), OSX, Linux
Number of Axes controllable: 4 (upgradeable to 6)
IOs: 6x input for (limit/end) switches, 1x E-stop, 2x relays, 0-10V control signal
Board: 1GHz CPU, 512MB DDR3 RAM, 4GB storage
Power supply: External, 110-240 V, 24 V, 5 A
Connectivity: USB, Ethernet, WiFi (optional)
File types: ISO G-Code RS-274, DIN 66025
Tool path simulation mode: available

SandyBoy - Lin-Contr. - [previous solution]



Functions of PC:

1) User interface (type your own G-Code or open G-Code files)

2) „Manual" control of the machine

SandyBox:

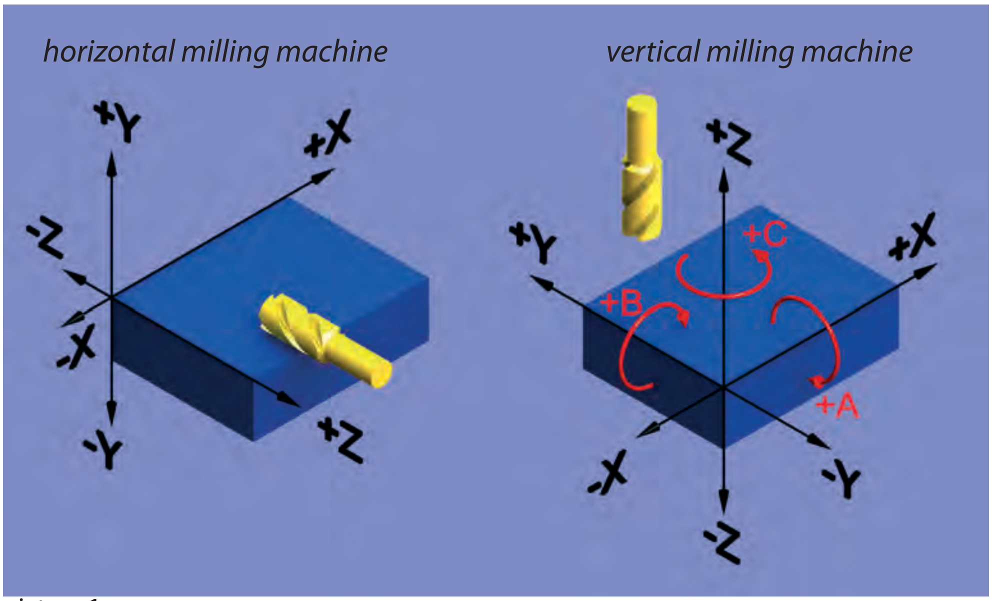
Motion control with pre-installed CNC software, Technical Specifications:
Client and drivers included for Win (7, 8, 10), OSX, Linux
Board: 1GHz CPU, 512MB DDR3 RAM, 4GB storage

Lin-Contr.:

Number of Axes controllable: 3 or 5
IOs: 5x input, 1x relays, 0-10V control signal
Power supply: External, 110-240 V, 24 V, 2 A
Connectivity: USB, Ethernet, WiFi (optional)
File types: ISO G-Code RS-274, DIN 66025
Tool path simulation mode: available

Coordinate systems of NC machines

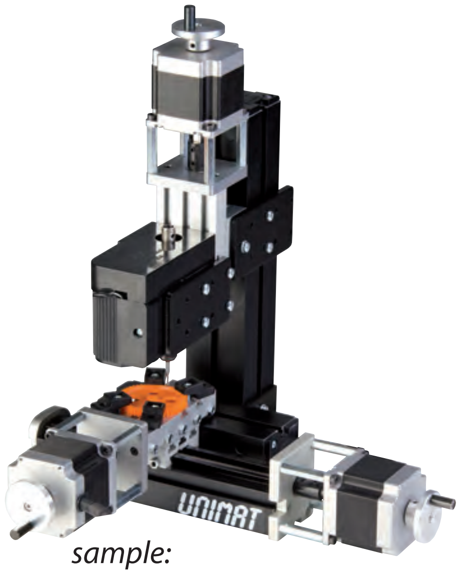
The Cartesian coordinate system refers to the work piece. The axis of the head spindle is Z.
Thereby we get different coordinate systems for Vertical-, Horizontal milling machines and turning machines.
Here you can see the coordinate system of a vertical- and horizontal milling machine.



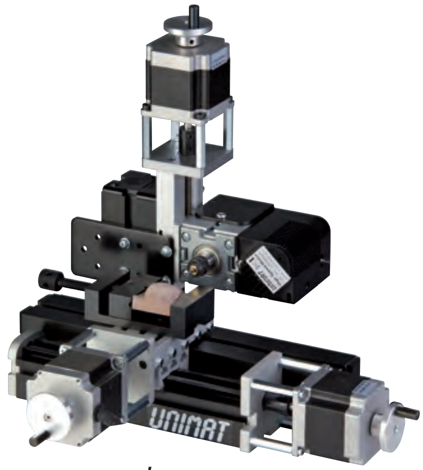
picture 1

If the machine can execute a rotation around an axis, then the name of this rotation axis is A, B or C (pic. 1).
Our 4 axes milling machine can turn the work piece around the x axis, thus the name of this rotation axis is A.

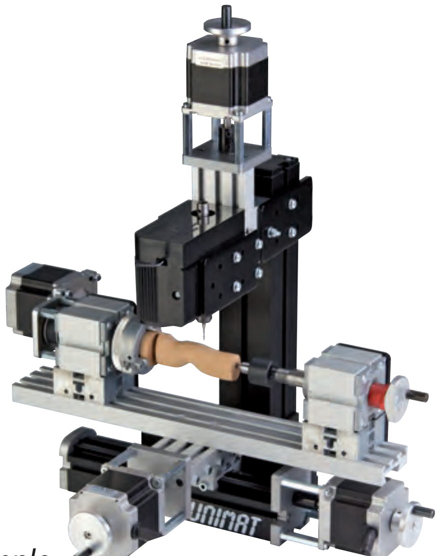
It is important to know that it is following defined G-Code programming -> the work piece is moving. Because of this we can use every norm conformable G-Code on every NC machine – the machine will move to the right direction.



sample:
vertical milling machine



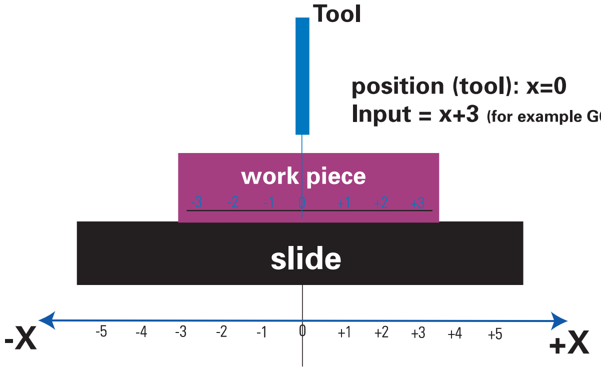
sample:
horizontal milling machine

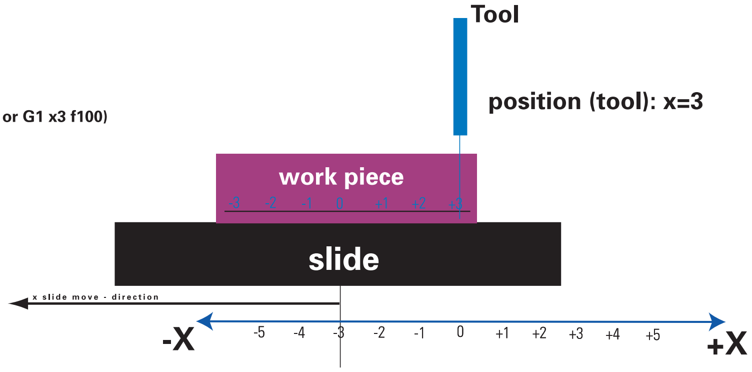


sample:
vertical milling machine with A -axis

Explanation:

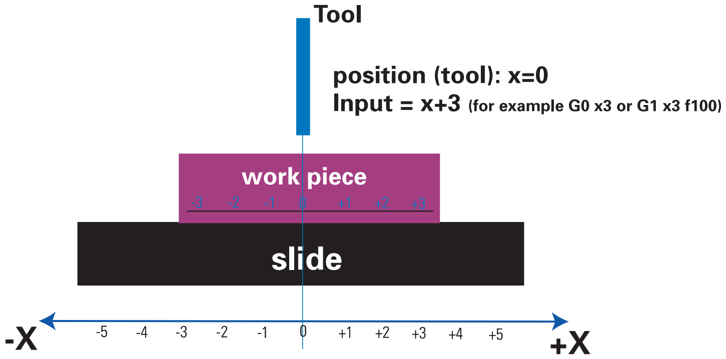
Machines where the work piece is moving (x/y) for example Uni-Fraes-V3 and Uni-Fräs4 – If you want to move the x axis from the zero position to x+3, then the slide goes –3.

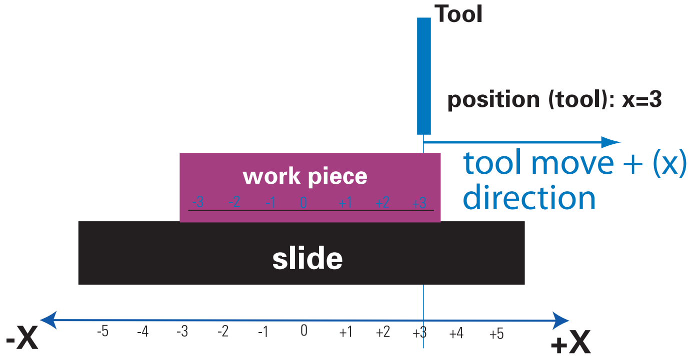




Now the position of the tool is +3 relative to the work piece.

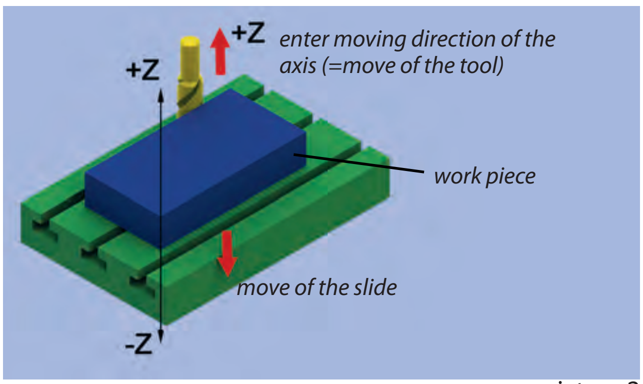
Machines where the tool is moving (gantry mills - like Step490) - If you want to move the x axis from the zero position to x+3, then the X slide goes with the tool +3.





Now the position of the tool is +3 relative to the work piece.

If you have a machine where the slide with the work piece is moving in Z direction, the same principle applies (pic. 2).

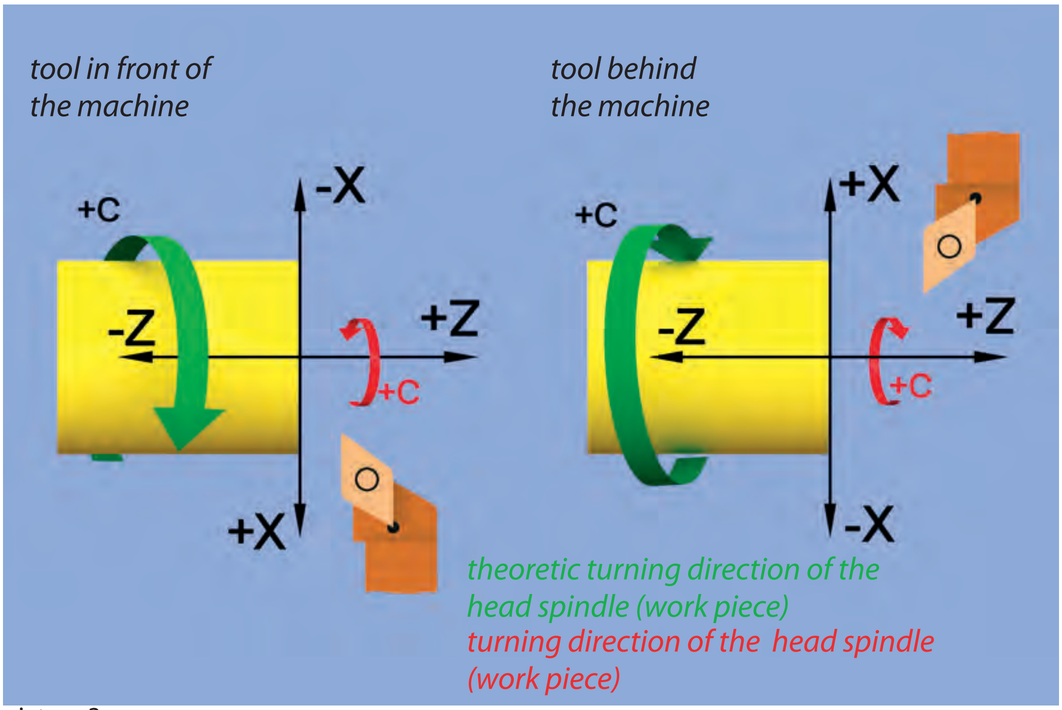


picture 2

If you want to move the Z axis into minus direction (z-5 for example), the slide with the work piece must travel into the plus direction.

Turning machines have 3 axes X, Z and C. The C axis is the head spindle (rotation axis).

The Z axis is the linear traveling of the head spindle. This is similar to the milling machines.

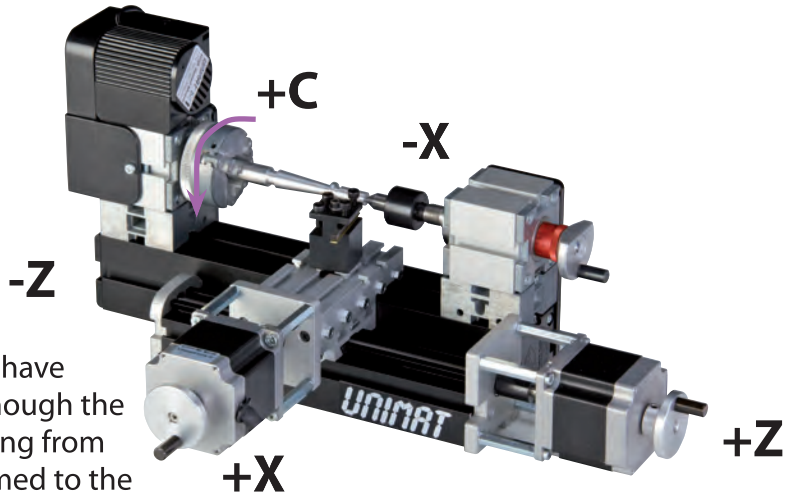


picture 3

All movements towards the work piece have a negative direction.

On picture 3 we can see that we have different coordinate systems, depending on whether the tool is positioned in front or behind the work piece.

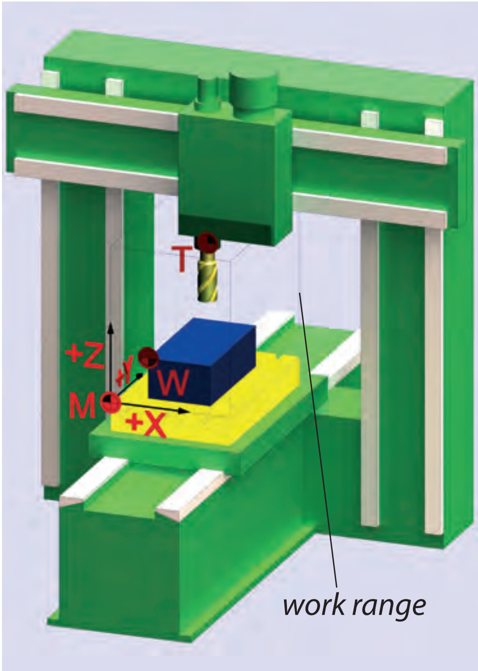
Our Unimat CNC (Uni-Dreh) turning machine has the following coordinate system.



Most of the NC turning machines have the tool behind the machine. Although the Unimat machine has the tool sliding from the front, the G-Code is programmed to the standard command (tool behind the machine)!

Machine zero point (M) (pic. 4):

This point associates the zero positions of all machine axes. It is not possible to change the position of this point. The linear encoder correlates with this point.



picture 4

Reference point (R) (pic. 5):

The reference point and the machine zero point are normally the same point. Only machines (for example a lathe) that cannot travel with the slides to the machine zero point, is the reference point at another position. If you travel with the slides to the reference point, the controller system (software) know the position of the machine's zero point. Many machines have switches for setting the reference point.

Reference point of the tool holder (T) (pic. 5):

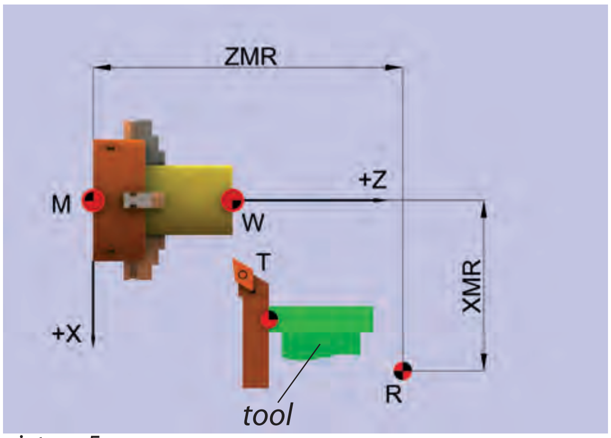
This is the point of intersection of the tool axis and the mounting surface of the tool.

Work piece zero point (W) (pic. 6 and 7):

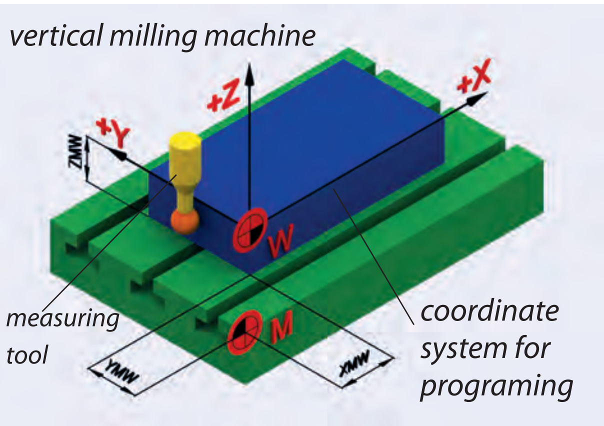
This point is very important. Without a work piece zero point you must make the programming of the design (work piece) set out from the machine zero point. This is not very useful. It is hard to design the program and it is not possible to use the same G-Code (program) on different machines.

Therefore it is state of the art that all coordinates of the work piece (G-Code program) refer to the work piece zero point.

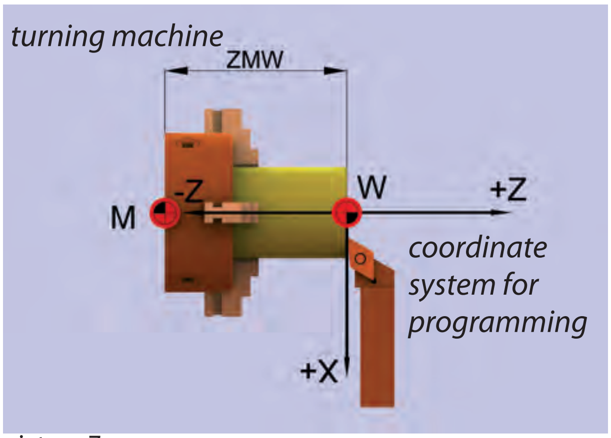
Because of this you can use the same G-Code on different machines.



picture 5

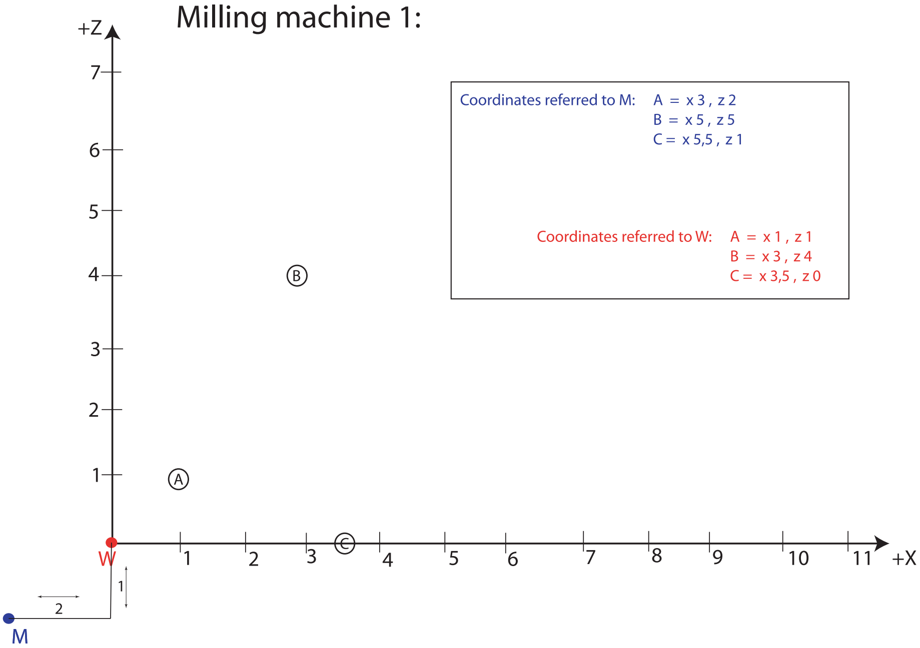


picture 6

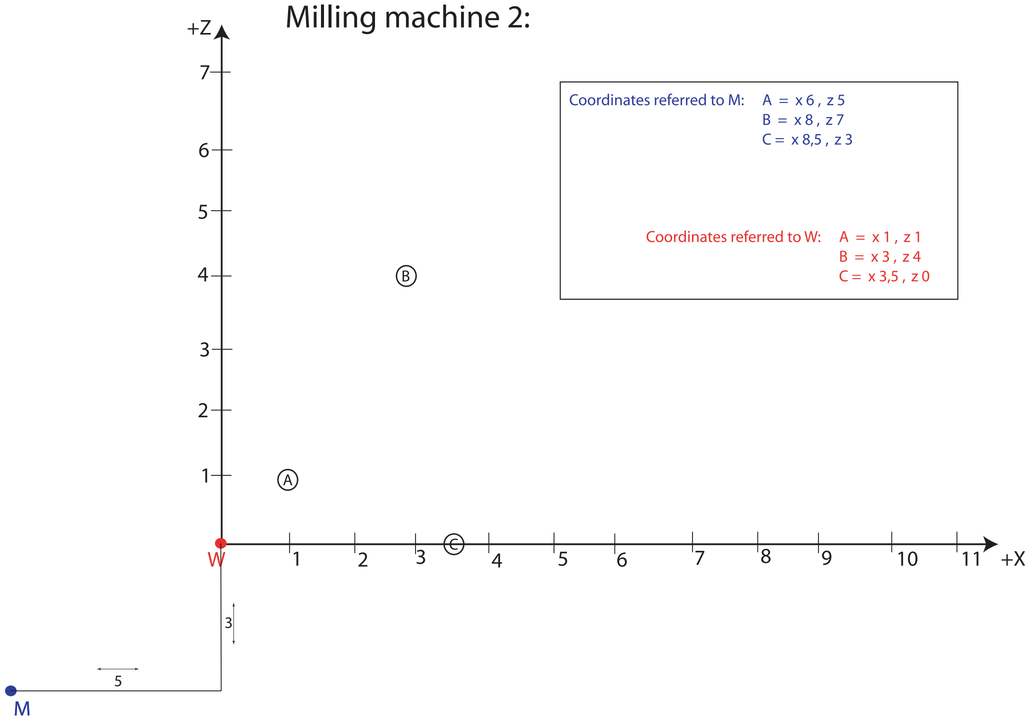


picture 7

Milling machine 1:



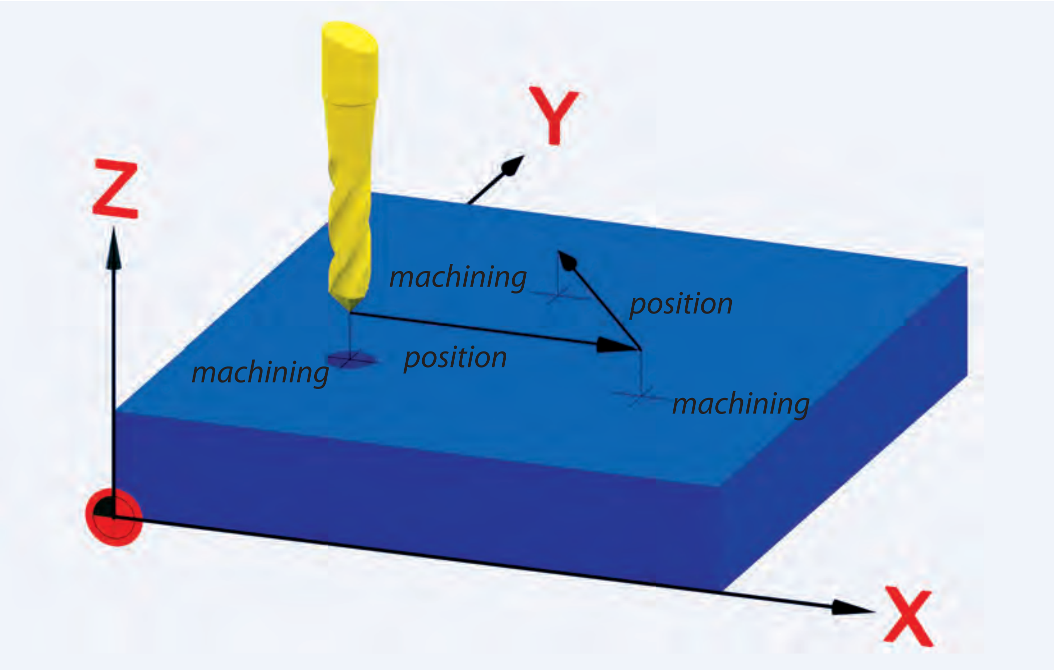
Milling machine 2:



It is possible to use the machine zero point and reference point on the Unimat system. Normally we use only the work piece zero point (for machineswithout reference switches).
Every G-Code program (work piece design) is referred to the work piece zero point.
Therefore we must only set the work piece zero point. The position of the work piece zero point referred to the work piece (left corner or center, etc.), is inserted as commanded by the G-Code file. Later we will see some sample files.

Point drive systems

This system is used for drilling-, punching- and spot welding machines. This controller can drive only one or two axes at the same time, from working point to working point (pic. 8). Only the end position (working point) is important, not the route (way).

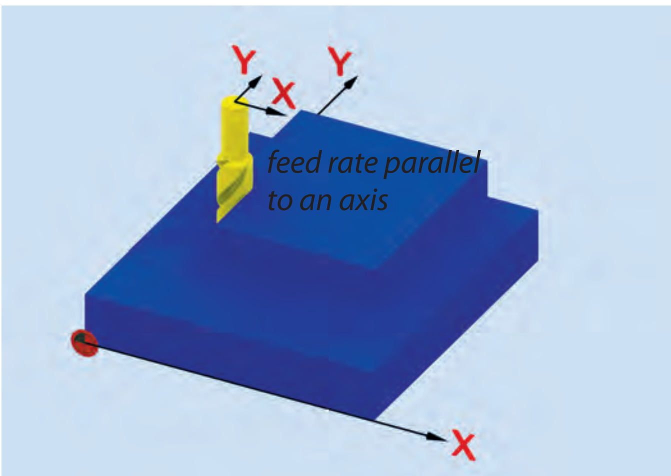


picture 8

The drilling bit goes to point 1, the z-axis goes down and up, then the drilling bit goes to point 2, z-axis goes down and so on, there is no control of the route from point 1 to 2!!!!

Distance drive system:

The machine can only travel parallel to the axis. For example you can make boxes, squares, but not circle or triangles (pic. 9).



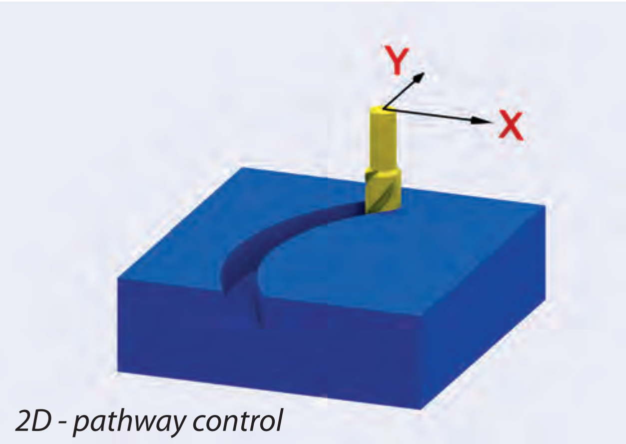
picture 9

Pathway drive system:

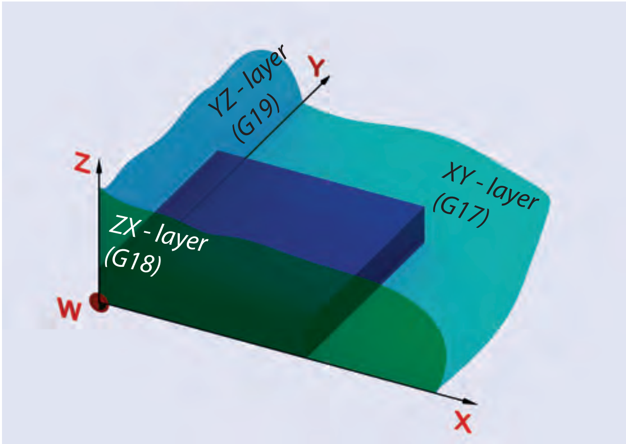
This system can drive 2 or more axes at the same time. The controller interpolates the speed of the axes – the tool follows exactly the programmed path. Here, the path is equally important as the endpoint of the movement.

2D pathway control

2 axis can drive interpolated at the same time. This control system has only 1 layer (X/Y) /pic. 10).



picture 10



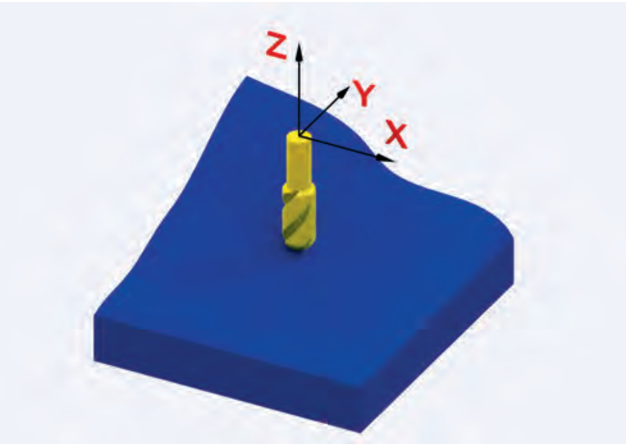
picture 11

2 ½ D pathway control

Here it's the same, 2 axes can drive at the same time, but it is possible to change between 2 or 3 layers (X/Y, Z/X, Y/Z) (pic. 11). You can select the layer with the following G command.

3D pathway control

This control system can drive (interpolates) 3 or more axes at the same time (pic. 12). Most of the modern machine centers have 5 axes.



picture 12

Machinekit® is a 3D pathway control system!
Up to 9 axes can be interpolated.

a) Manually in any text editor or user interface on a CNC machine.

b) Automatically – use any CAM software to convert the CAD drawing into the NC program

Today the normal way is to generate the G -Code with a CAM software (we recommend **Fusion 360** and **Inkscape**).

Basics:

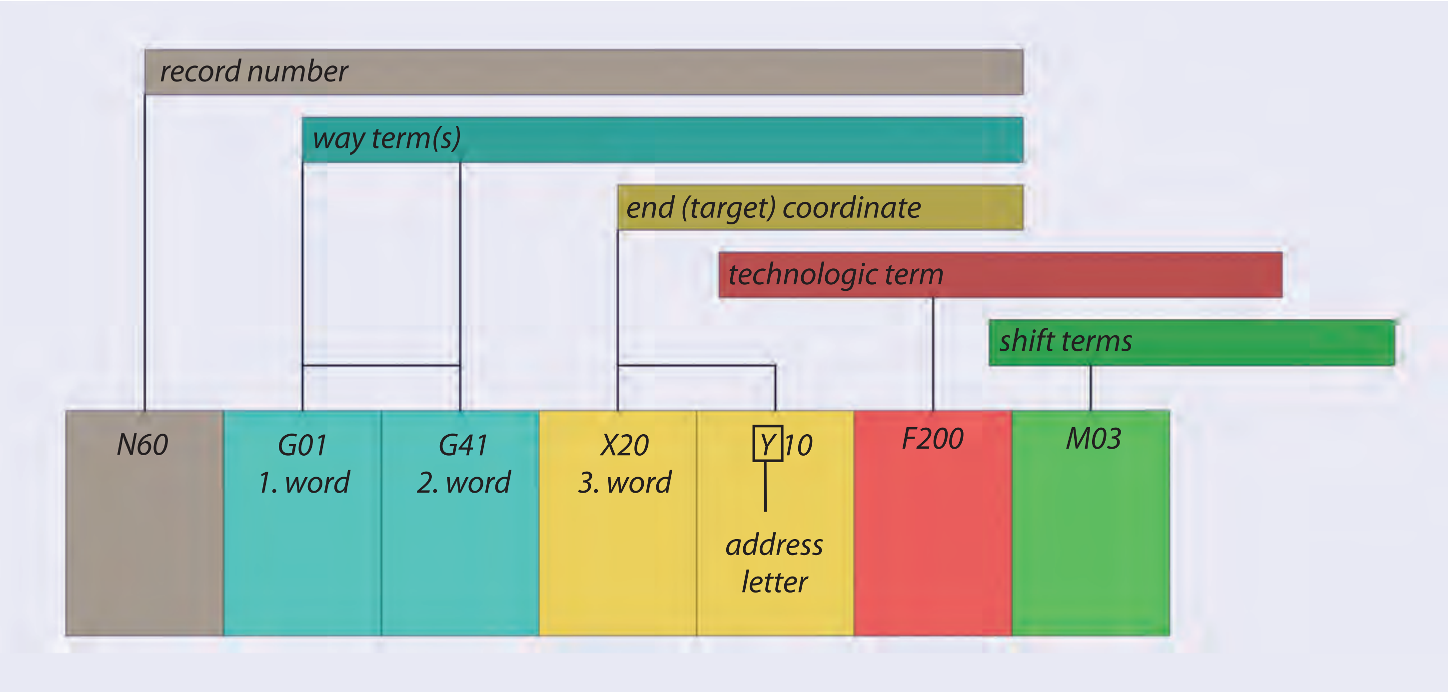
N …. Record number

Gxx …. way terms (linear or circle interpolated and so on)

X, Y, …. geometric terms

F,S,T …. technologic terms (feed rate, speed, tool)

M …. shift terms (motor on/off and so on)



Index of the useable comands in CoolCNC Linux

Code	Description	Parameters
G0	Coordinated Straight Motion Rapid	-
G1	Coordinated Straight Motion Feed Rate	-
G2, G3	Coordinated Helical Motion Feed Rate	I, J, K oder R
G4	Dwell	P
G7, G8	Diameter or Radius Mode	-

Code	Description	Parameters
G10 L1	Set Tool Table Entry	P, R, X, Z
G10 L2	Coordinate System Origin Setting	P, X, Y, Z, A, B, C
G17	XY plane	-
G18	ZX plane	-
G19	YZ plane	-
G20	Inches	-
G21	Millimeter	-
G28	Return To	-
G30	Return To	-
G33	Spindle Synchronized Motion	K
G33.1	Rigid Tapping	K
G38.2 - .5	Probing	-
G40	Cancel Cutter Compensation	-
G41	Cutter Compensation left	-
G42	Cutter Compensation rigth	-
G41.1	Cutter Compensation Transient	D, L
G42.1	Cutter Compensation Transient	D. L
G43	Use Tool Length Offset from Tool Table	H
G49	Cancel Tool Length Offset	I, K
G53	Motion in Machine Coordinate System	-
G54 - G59	Select Coordinate System	-
G59.1 - .3	Select Coordinate System	-
G61	Path Mode	-
G61.1	Path Mode exact stop	-
G64	Continuous Mode with Optional Tolerance	P
G73	Drilling Cycle with Chip Breaking	R, L, Q
G76	Multipass Threading Cycle (Lathe)	P, Z, I, J, R, K, Q, H, L, E
G80	Cancel Motion Modes	
G81	Canned Drilling Cycle	R, L, P
G82 - G89	Other Canned Cycles	R, L, P, Q
G90	Absolut Distance Mode	-
G91	Interpolate Distance Mode	-
G92	Offset Coordinate Systems & Set Parameters	X, Y, Z, A, B, C
G92.1 - .2	Cancel Offsets	-
G92.3	Apply Parameters to Offset Coordinate Systems	-
G93	Feed Modes	-
G94	Feed Modes (mm/min)	-
G95	Feed Modes (mm/U)	-

Code	Description	Parameters
G96	Constant Surface Speed	D, S
G97	RPM Mode	-
G98, G99	Canned Cycle Z Retract Mode	-
F	Feed	-
S	Speed	-
M0	Pause	-
M1	Pause	-
M2	end of program	-
M3	Spindle on clockwise	S
M4	Spindle on counterclockwise	S
M5	Spindle off	-
M6	Change Tool T=Tool Number	T
M7	Mist on	-
M8	Flood on	-
M9	Turn all coolant off	-
M30, M60	Pallet Shuttle	-
M48 - M53	Overrides	P
M61	Set Current Tool Number	Q
M62 - 65	Output Control	P
M66	Input Control	P, E, L, Q
M100 - 199	User Defined M-Codes	P, Q
O	O Codes	-

More details about how to use the codes you can find at www.linuxcnc.org.

G0 / G00

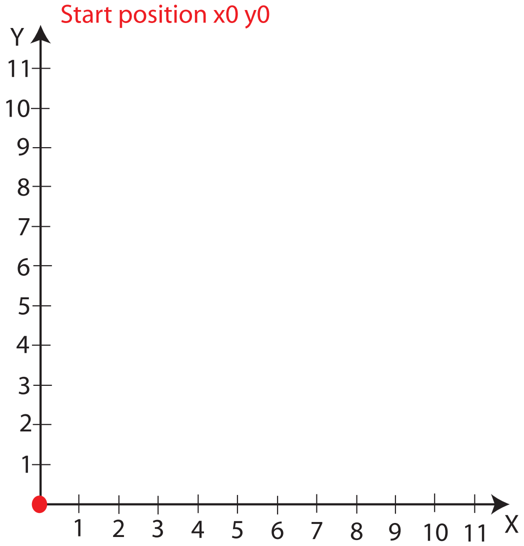
Rapid linear mode – the machine moves at a maximum speed.

G1 / G01:

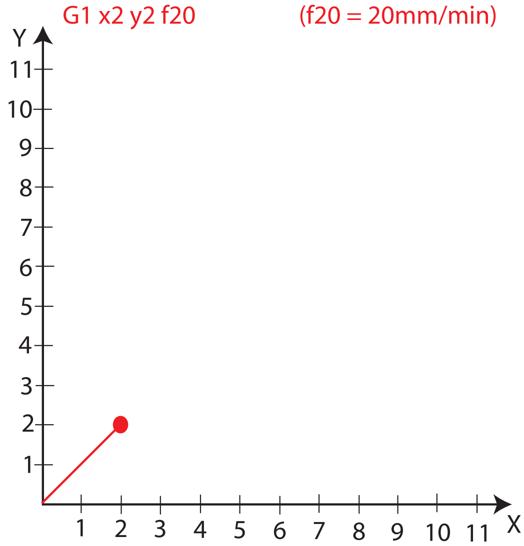
Linear interpolated movement. The feed rate [F] must be set.

Sample 1:

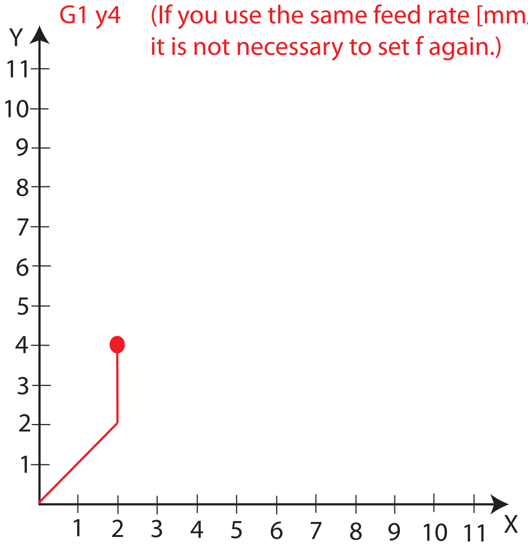
Start position x0 y0



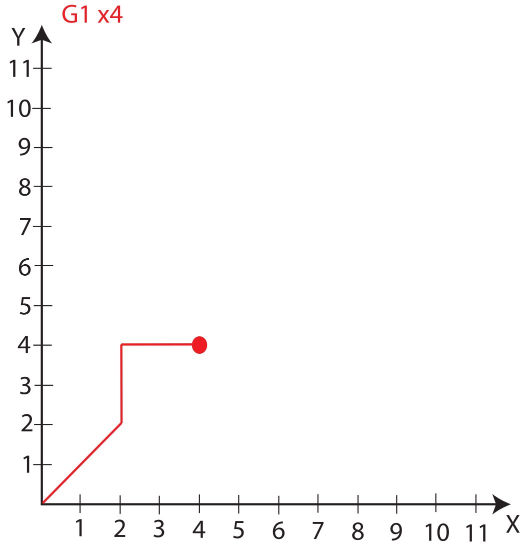
G1 x2 y2 f20 (f20 = 20mm/min)



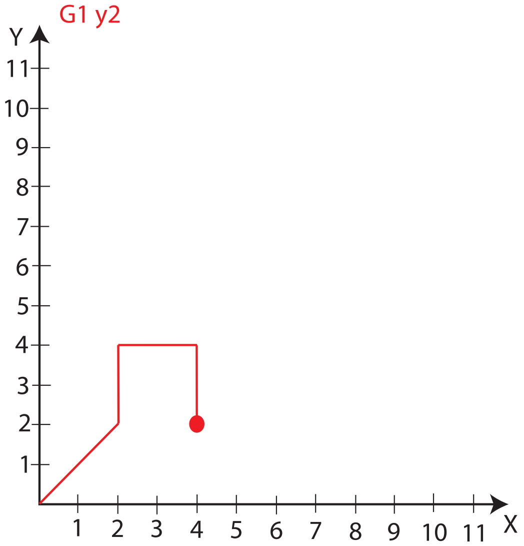
G1 y4 (If you use the same feed rate [mm/min], it is not necessary to set f again.)



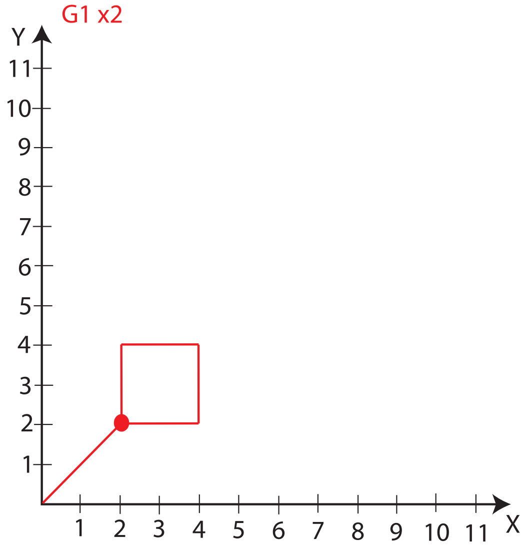
G1 x4



G1 y2

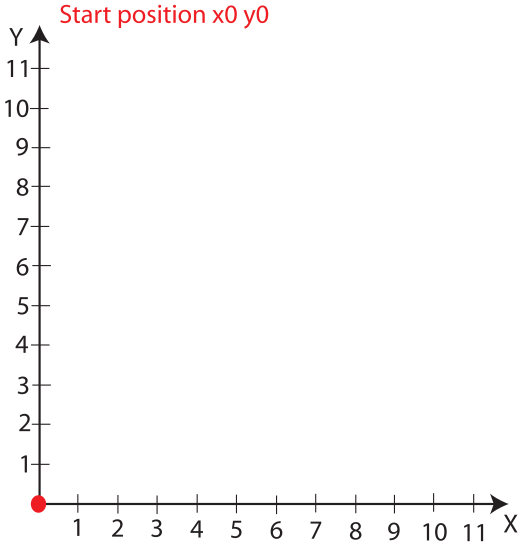


G1 x2

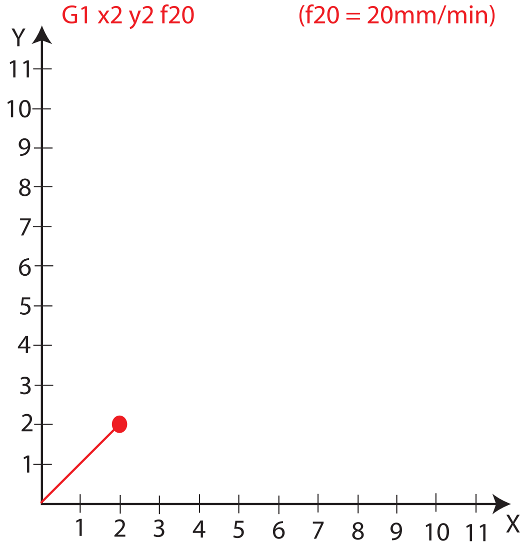


Sample 2:

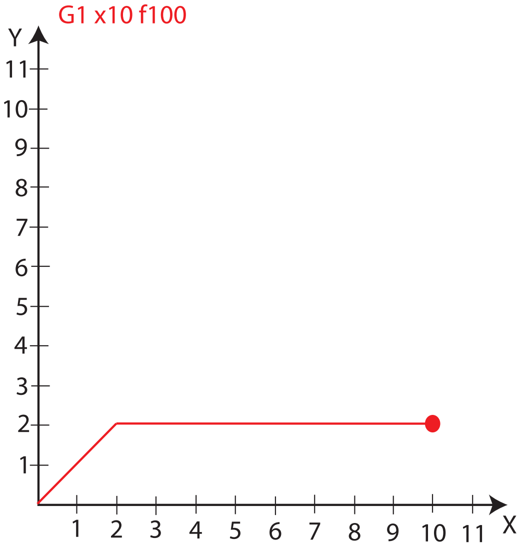
Start position x0 y0



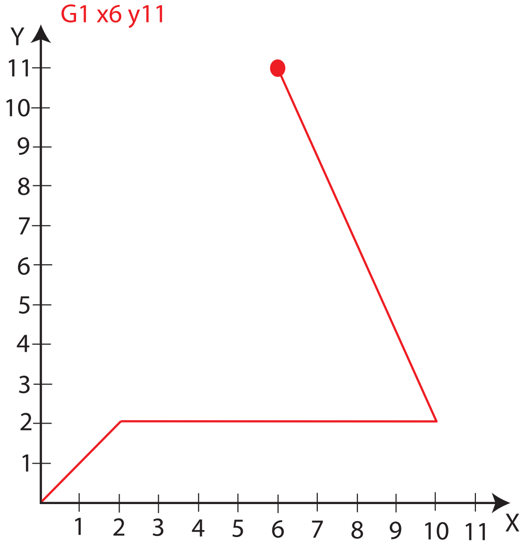
G1 x2 y2 f20 (f20 = 20mm/min)



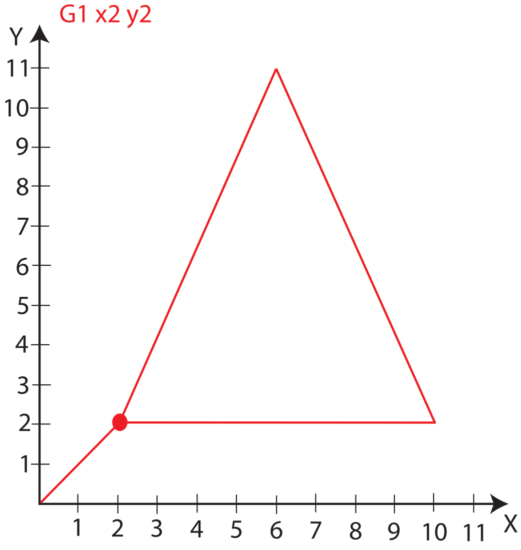
G1 x10 f100



G1 x6 y11



G1 x2 y2



G2 und G3 / G02 und G03

Circle interpolated movement. The feed rate [F] must be set.

G2/G02 is clockwise circular movement.

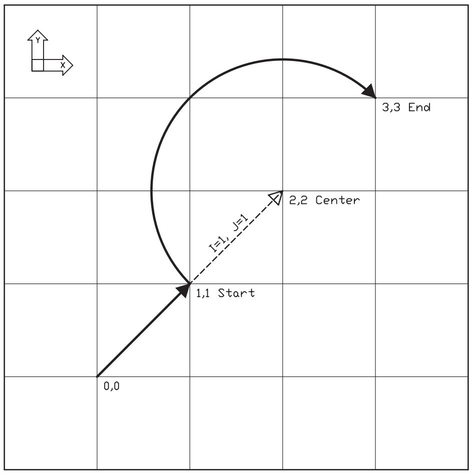
G3/G03 is counter-clockwise circular movement.

The G02 command moves the tool in a clockwise path from the starting point (the current tool position) to the designated ending point in the currently selected plane (see G17-G19). The I , J, and K parameters represent the incremental X, Y, and Z distances (respectively) from the starting point of the arc to the center point of the arc (pic. 1).

Example:

G1 x1 y1 f3
G2 x3 y3 i1 J1

Moves the tool to program coordinates X=1, Y=1 at a feedrate of 3 mm/min. Moves the tool using clockwise circular interpolation to program coordinates X=3, Y=3 with a center point of X=2, Y=2 at a feedrate of 3 mm/min



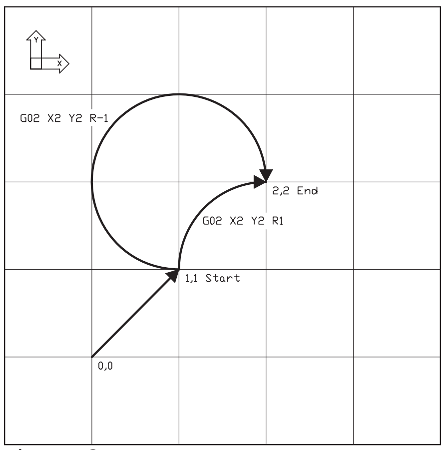
picture 1

An alternative way to specify the distance to the center point is to specify the radius, using the R parameter (pic. 2). This is usually easier than determining the correct I, J and K values. For any given radius, there are usually two possible arcs: one that sweeps an angle less than 180 degrees, and one that sweeps an angle greater than 180 degrees (see diagram below). To specify an angle less than 180 degrees, make R a positive number; to specify an angle greater than 180 degrees, make R a negative number.

G1 x1 y1 f3

G2 x2 y2 r1 (or r-1)

Moves the tool to program coordinates X=1, Y=1 at a feedrate of 3 mm/min. Moves the tool using clockwise circular interpolation to program coordinates X=2, Y=2



picture 2

When using the R word, please note:

• If the arc sweeps a 180 degree angle, it doesn't matter whether R is negative or positive.

• If the end point is the same as the starting point, CoolCNC will ignore the command, since the center point cannot be determined.

You can use the G02 command to specify a helical move (helical interpolation). During a helical move, the circular motion described above is combined with linear motion that is perpendicular to the plane containing the arc. For example, circular motion in the XY plane is combined with linear motion along the Z axis to form a helix.

To specify helical motion, add an X, Y or Z parameter to the command, wich indicates the ending point of the linear motion. In the following example, a Z parameter has been added to the G02 command for an arc in the XY plane.

G1 x1 y1 z1 f3

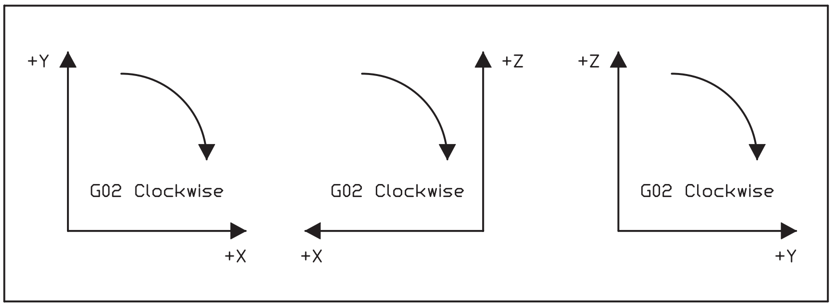
G2 x3 y3 z2 i1 j1

Moves the tool to program coordinates X=1, Y=1, Z=1 at a feedrate of 3 mm/min. Moves the tool using clockwise circular interpolation to program coordinates X=3, Y=3 with a center point of X=2, Y=2, while simultaneously moving the tool in a straight line along the Z axis to Z=2

The accuracy of the helical move, relative to a theoretically perfect helix, is controlled by the Helical Interpolation Accuracy setting on the G-Code panel of the Configuration dialog box (see "G-Code Settings" in the Initial Setup section).

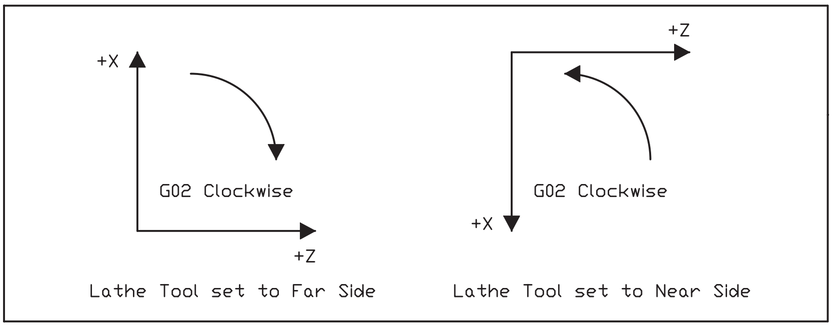
When using G02, there are several things to keep in mind:

• This is a modal command, meaning that all successive moves will be treated as clockwise circular feedrate moves until another modal move command (G00, G01 or G03) occurs.
• The interpretation of the X, Y and Z coordinates depends on the G90/G91 command in effect. The I, J and K values are unaffected by G90/G91.
• The tool will move at the current feedrate set by the last F command.
• Only XY arcs can be cut when G17 is active, only XZ arcs can be cut when G18 is active, and only YZ arcs can be cut when G19 is active. Arcs cannot be specified for the A or W axes.
• The clockwise direction of rotation is as viewed from the positive end of the unused axis (the axis not in the plane of motion). For example, a G02 arc move in the XY plane is clockwise as viewed from the positive end of the Z axis (i.e. from above). The following diagram (pic. 3) illustrates this behavior for all three arc planes:



picture 3

In some cases, a clockwise arc as defined above will be displayed counter clockwise in a viewport. For example, in lathe applications, the Lathe Tool setting (on the Machine Tool panel of the Configuration dialog box) determines whether the X positive direction is up or down in the ZX viewport. When Lathe Tool is set to Near Side, the X positive direction is down and G02 arcs are displayed counter-clockwise on the screen (pic. 4).



picture 4

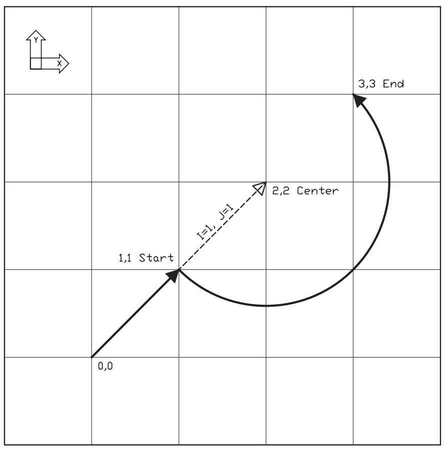
The G03 command is identical to the G02 command, but it moves the tool in a counter-clockwise arc instead of a clockwise arc (pic. 5).

Example:

G1 x1 y1 f3

G3 x3 y3 i1 j1

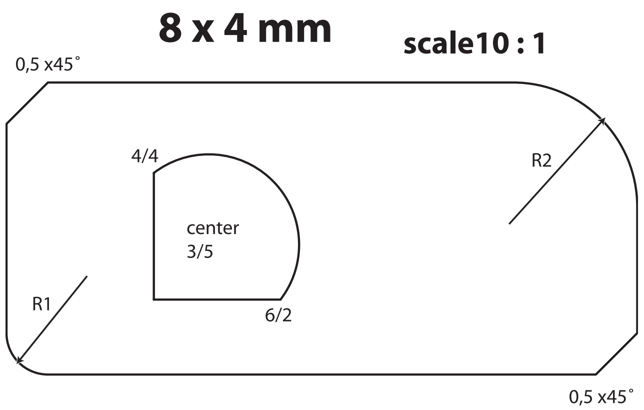
Moves the tool to program coordinates X=1, Y=1 at a feedrate of 3 mm/min. Moves the tool using counter-clockwise circular interpolation to program coordinates X=3, Y=3 with a center point of X=2, Y=2 at a feed rate of 3 mm/min



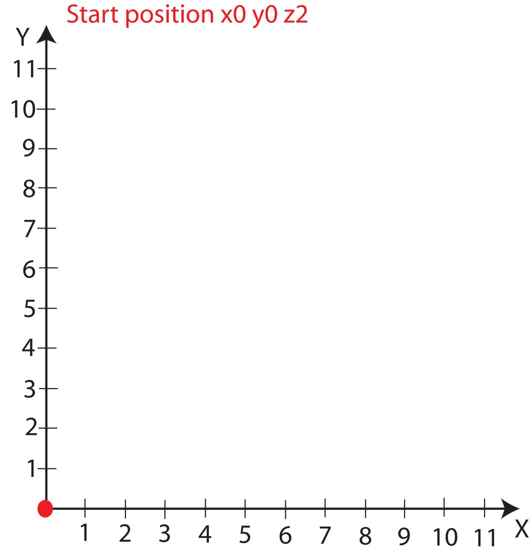
picture 5

Machinekit® can interpret both version (I,J,K or R) !!!

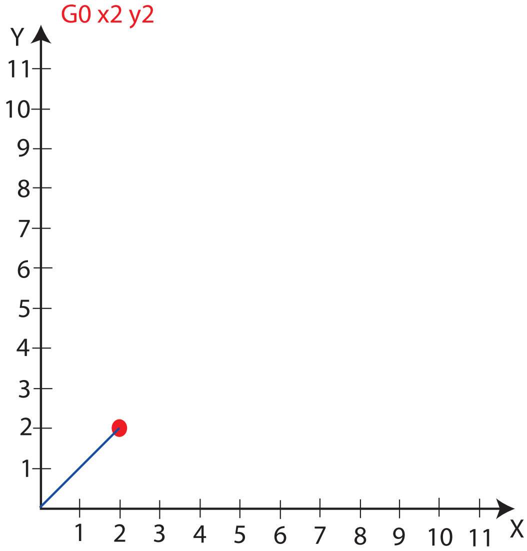
Sample 3:



Start position x0 y0 z2

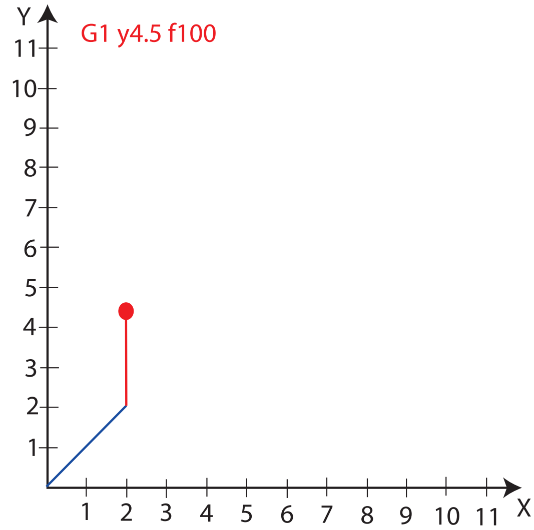


G0 x2 y2

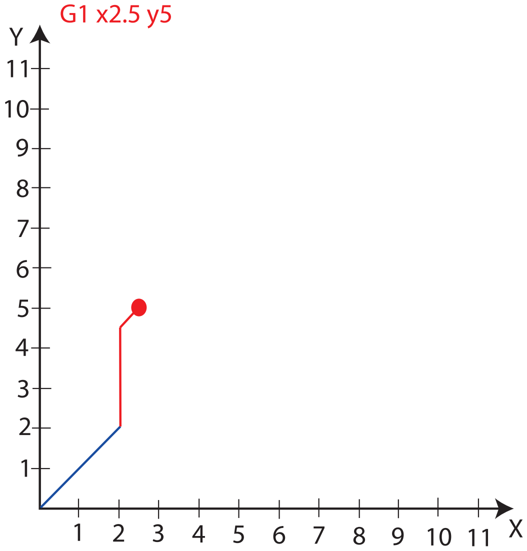


G1 z-1 f20 (the z axis move from z2 to z-1, with 20mm/min)

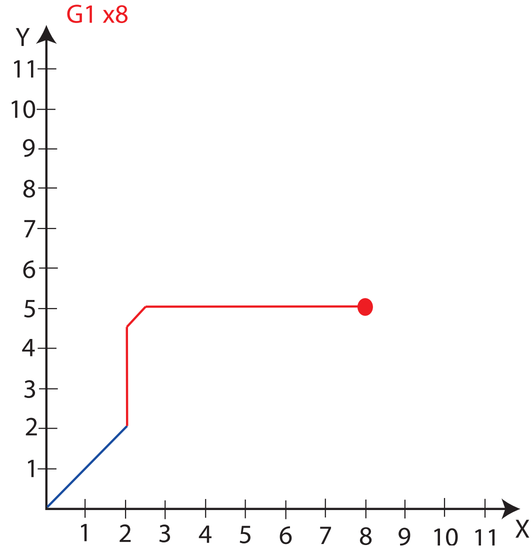
G1 y4.5 f100

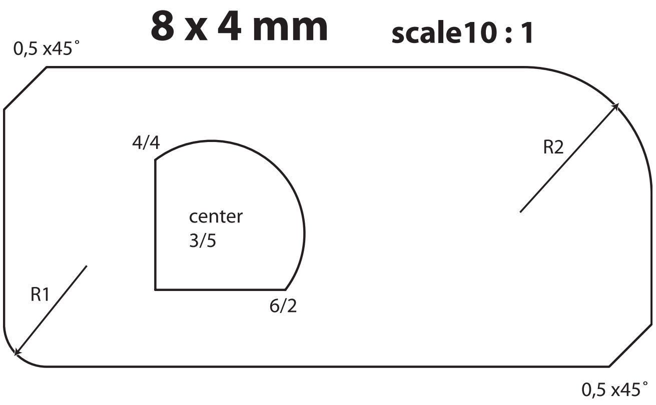


G1 x2.5 y5

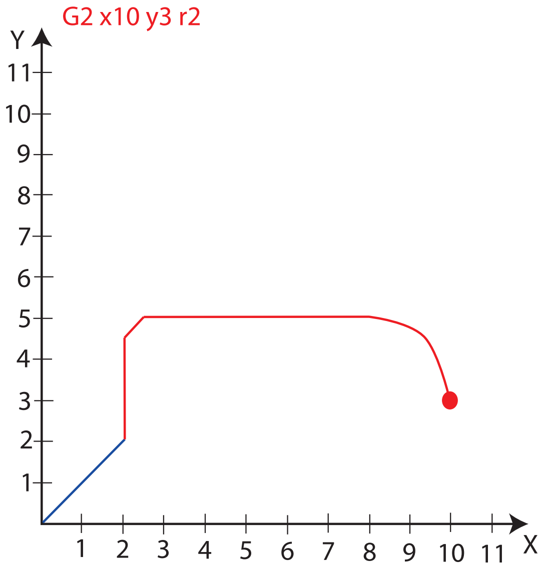


G1 x8

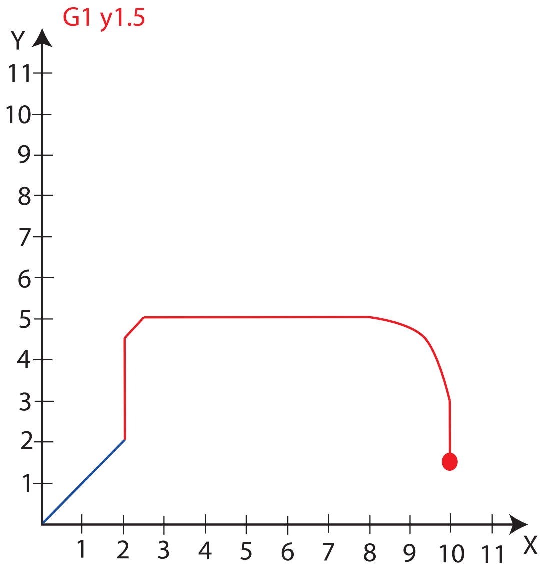




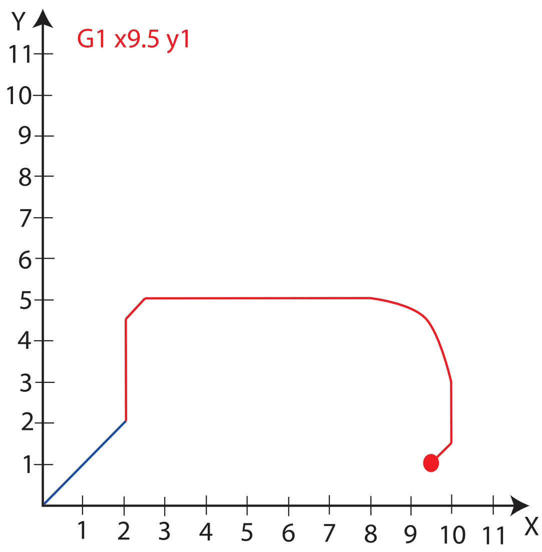
8 x 4 mm
scale10 : 1
0,5 x45˚
4/4
R2
center
3/5
R1
6/2
0,5 x45˚



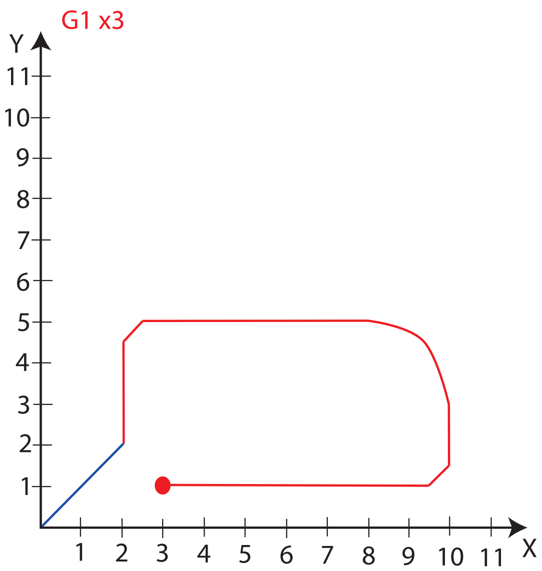
G2 x10 y3 r2



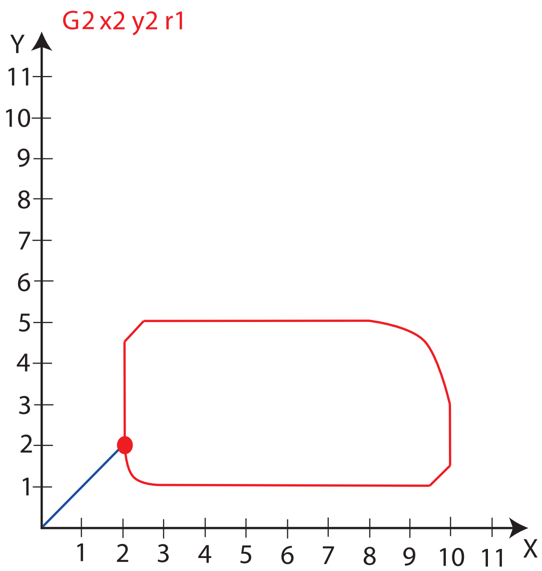
G1 y1.5



G1 x9.5 y1

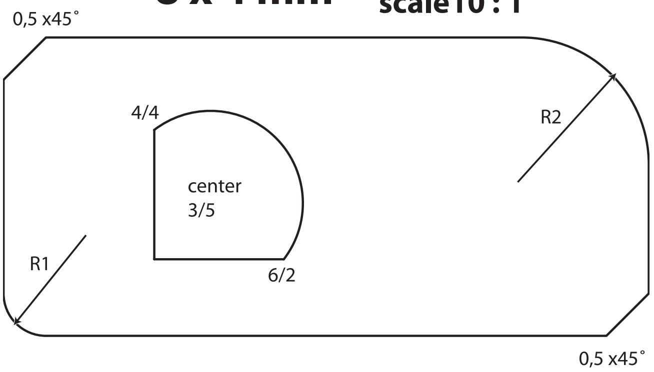


G1 x3



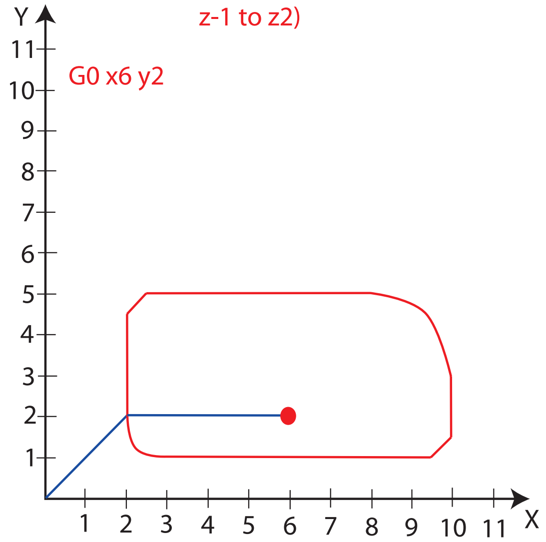
G2 x2 y2 r1

8 x 4 mm **scale10 : 1**



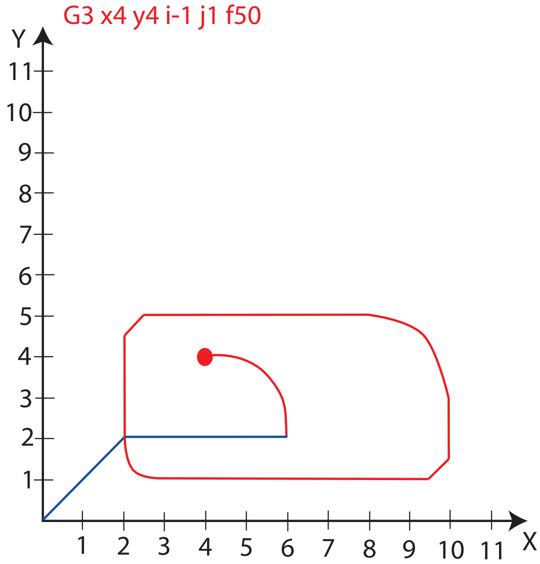
G0 z2 (z axis goes in rapid mode from z-1 to z2)

G0 x6 y2

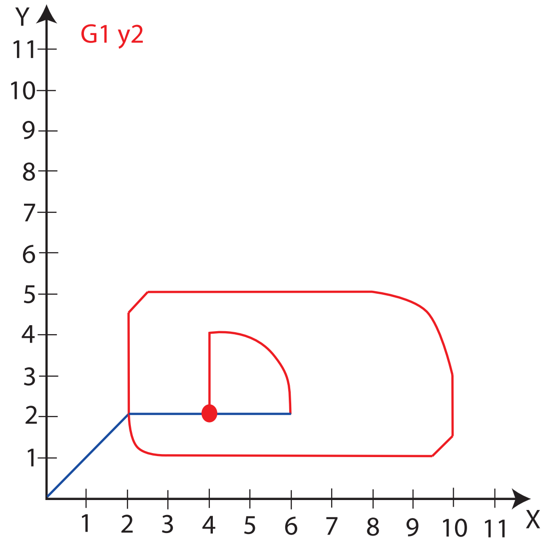


G1 z-1 f20 (z move from z2 to z-1 with 20mm/min)

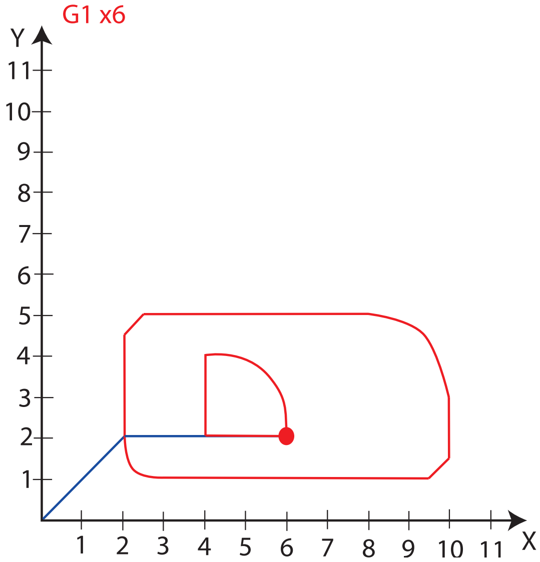
G3 x4 y4 i-1 j1 f50



G1 y2

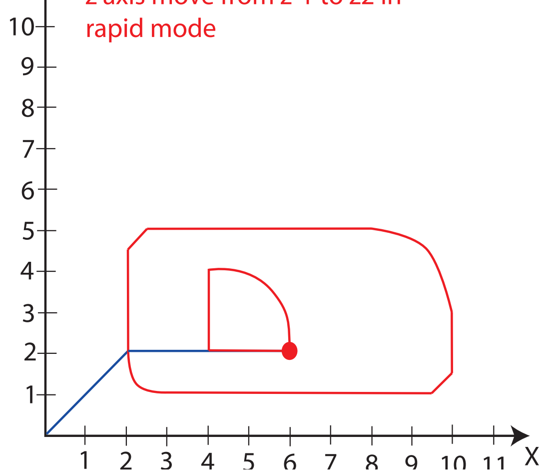


G1 x6



G0 z2

z axis move from z-1 to z2 in rapid mode



General advice on how to handle the TCTControl box

The TCTControl boxes are available in 4 or 6 axes option. Both versions come with stepper motor outputs and: 1x USB client, 1x USB host, 1x 10/100 ethernet, 1x E-stop input, 6x Signal input (i.e. for limit switches), 1x spindel control ouput, 2x relais output, 1x UniPrint3D port and 1x Power supply connector (AC adapter included in delivery).

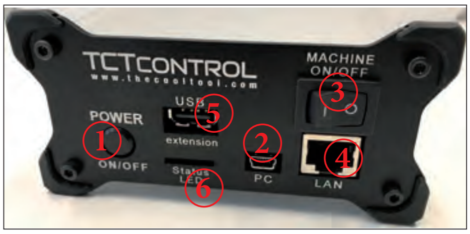
Note the following:

1) Never plug or unplug the step motors during operation.
The "MACHINE" [3] must be switched off first. This also applies to all other inputs and outputs on the rear panel.

3) Use the box in a dry environment. The ambient temperature should be between -10 and +35 degrees celsius (14 and 95 degrees Fahren-heit).

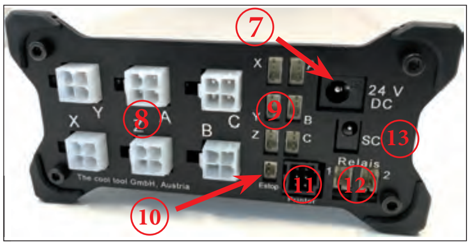
4) Only use the step motors supplied by us.

5) Turn on "MACHINE" only after the control software machinekit® (Cetus/machineface®) has been started. To simulate projects on the screen turn off "MACHINE".



1 ... boot/shut down button (software part)
2 ... connection to the PC (USB)
3 ... ON/OFF (stepper motor driver part)
The stepper motor drivers part is independent from the software part and must be switched on/off separately.
4 ... network connection
5 ... USB port to connect WLan adapter, flash drives, ...
6 ... status LEDs:

Blue LED shine
TCTControl software part is running
Blue and red LED shine
CNC application is running
software status -> machine power on
(ON/OFF button without function)



7 .... power connector (24 V / 5 A adapter)
8 .... machine axes (stepper motor)
9 .... input eg. reference switches
10 ... E-stop (art.no. 164 425 CNC)
11 ... connection to UniPrint3D
12 ... relais outputs 1 + 2 (signal 24 V)
13 ... spindle control output (0 - 10 V)

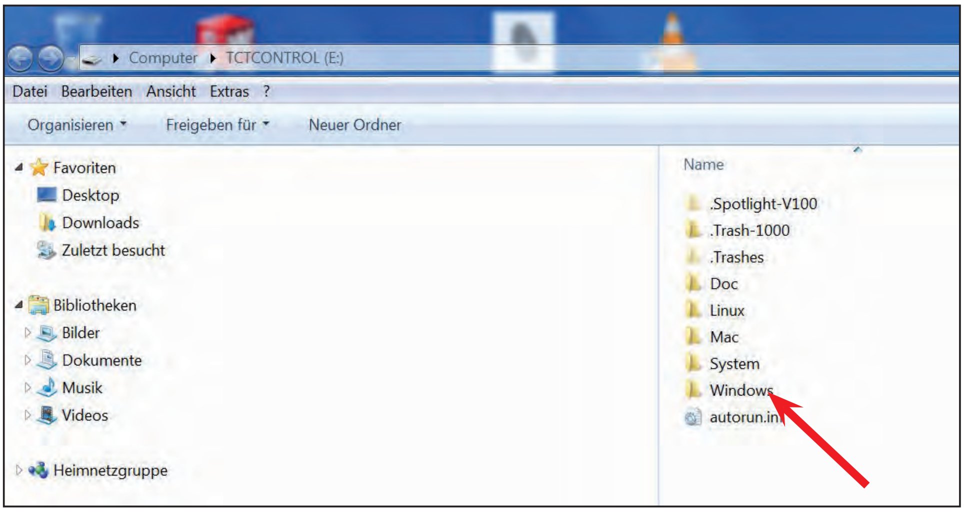
Use the supplied USB cable to connect TCTControl to a free USB port.

Wait until it shows under Windows.

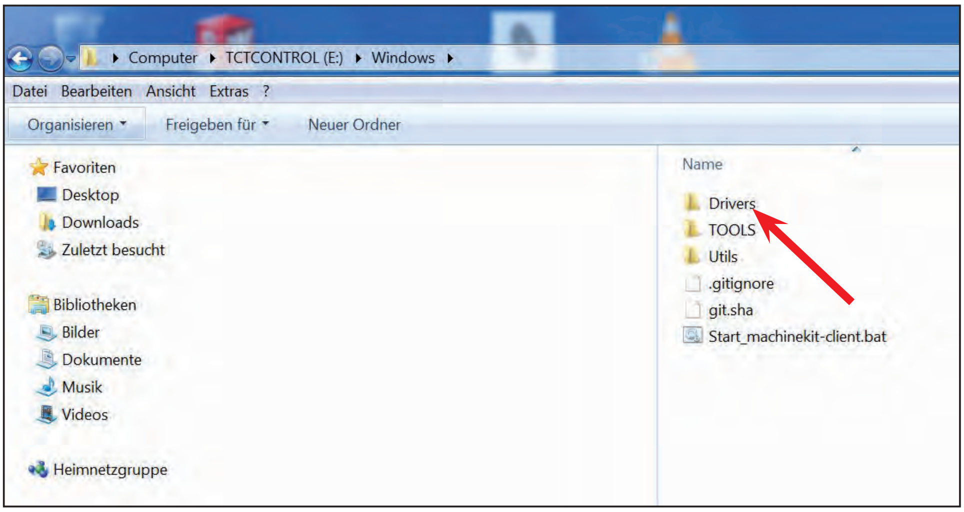
(PC with Windows 7 or higher)

For Win8 and Win10 user:
a) disable the driver signature verification before you continue.
You can find many tutorials on the internet.
or
b) follow the video **https://youtu.be/y-rFVQplUWs**
(In this case, continue with „**Start CNC application 4.2.2**".)

open folder „Windows“
on TCTControl

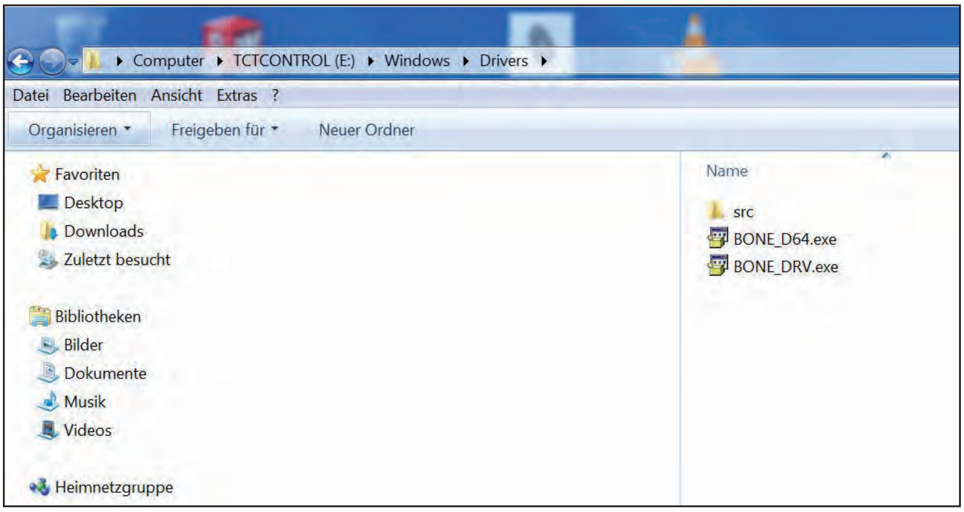


go to „Drivers“



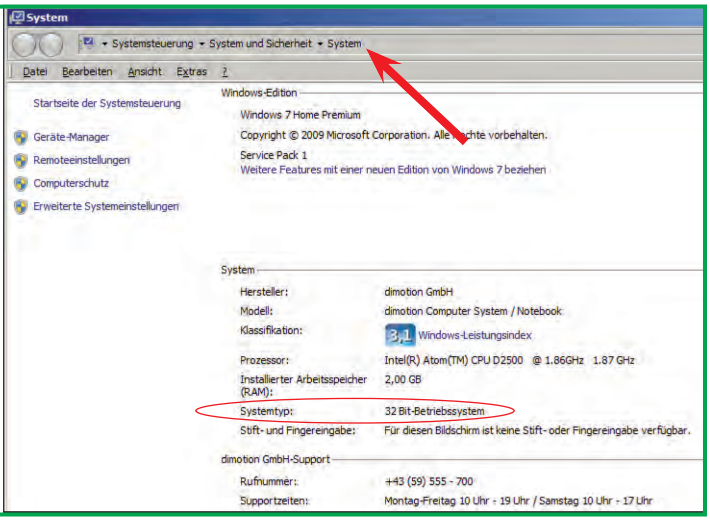
start „BONE_DRV“ for 32bit OS

start „BONE_D64“ for 64bit OS

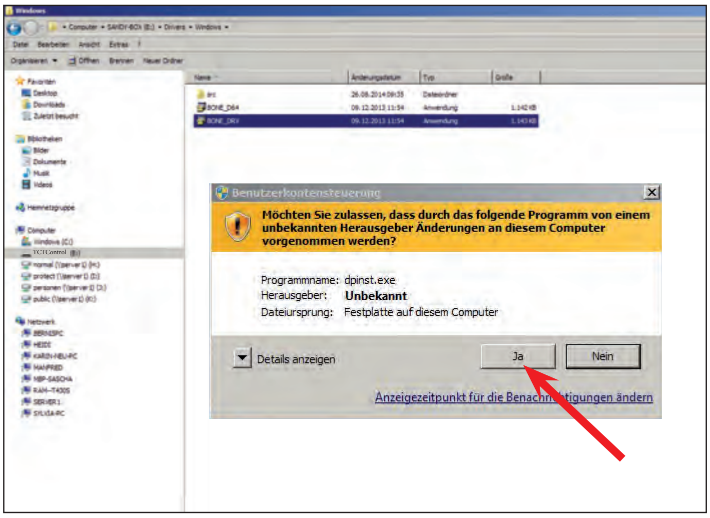


OPTIONAL

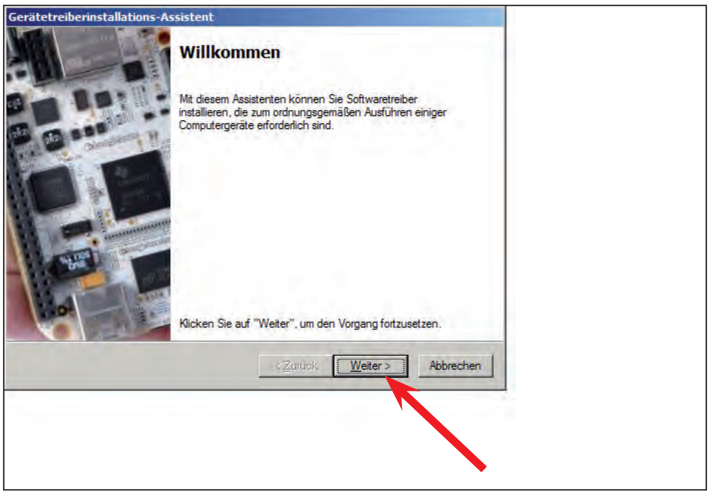
Under „Systeminformation“ you can find the information.



click YES



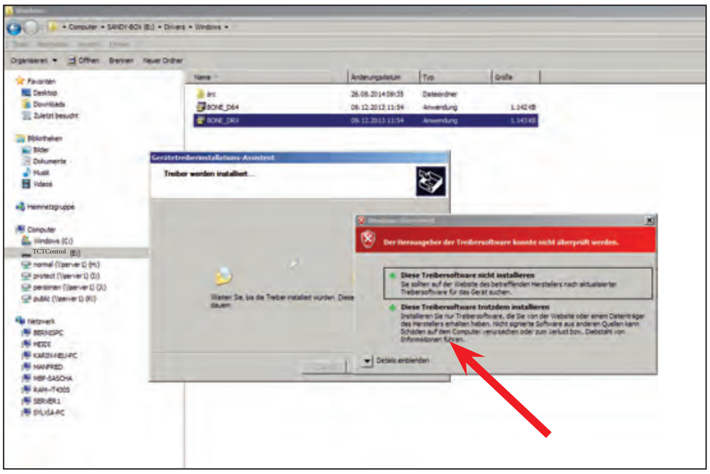
click CONTINUE



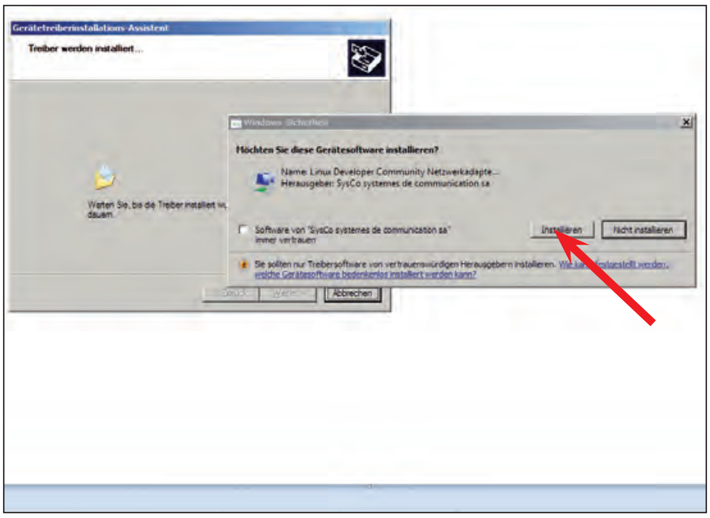
YES install this driver!

(This needs to be continued several times.)

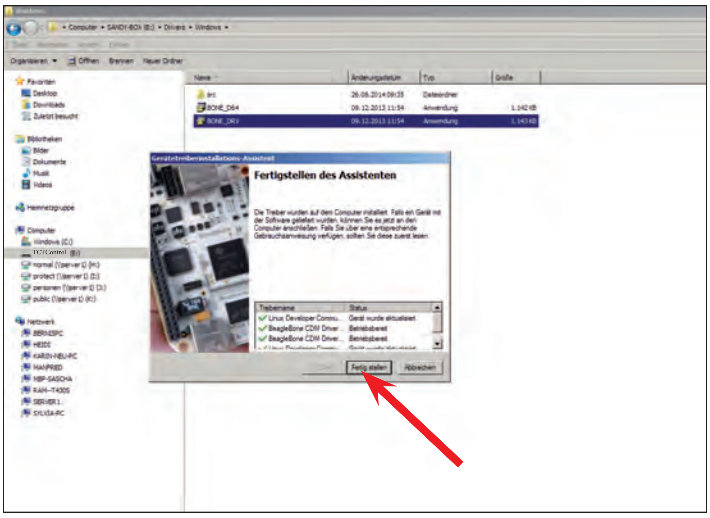
Also confirm all firewall and virus scanner messages concerning TCTControl.



click INSTALL

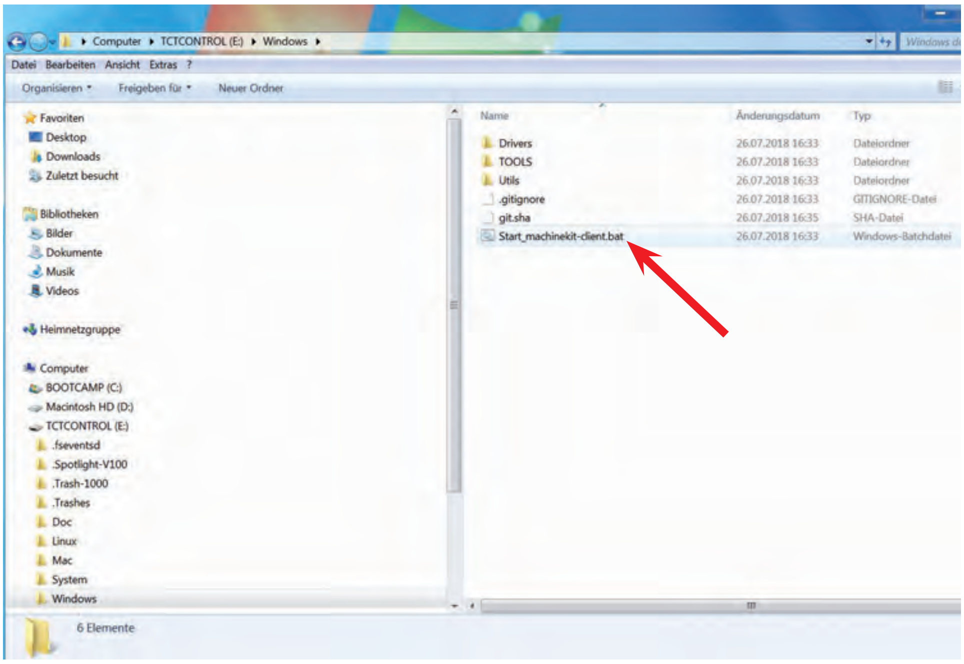


click COMPLETE

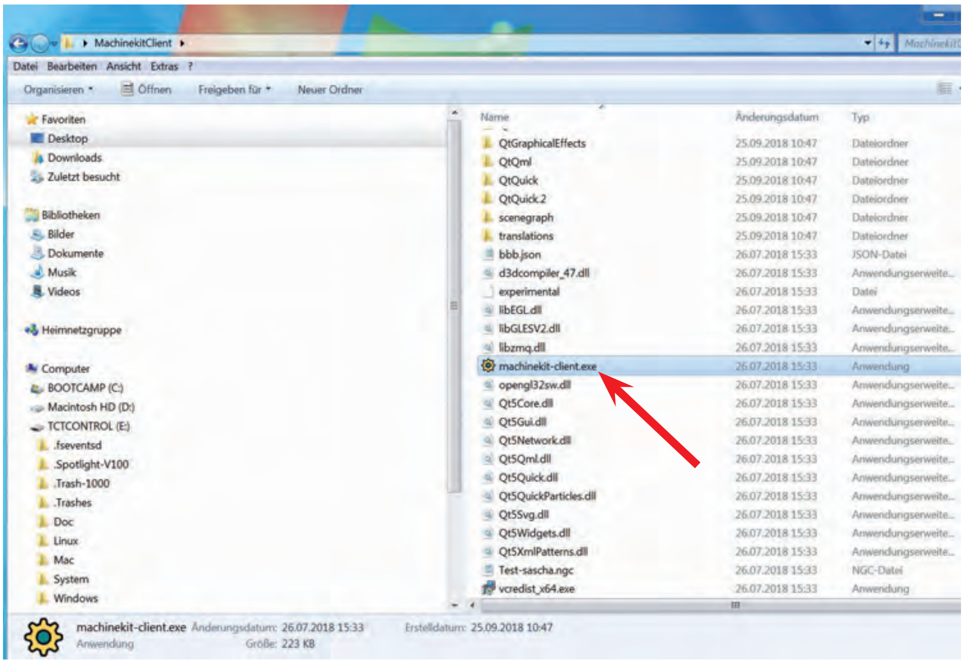


Start CNC application 4.2.2

Variant A:
Run the machinekit-client directly from TCTControl.
Go to
TCTControl\Windows\
start “Start_machinekit-client.bat”



Variant B:
Run the machinekit-client from your Windows system.
Go to
TCTControl\Windows\Utils\
copy the folder “MachinekitClient “ to you system (eg.: to the Desktop),
Open the folder and start
“machinekit-client.exe”

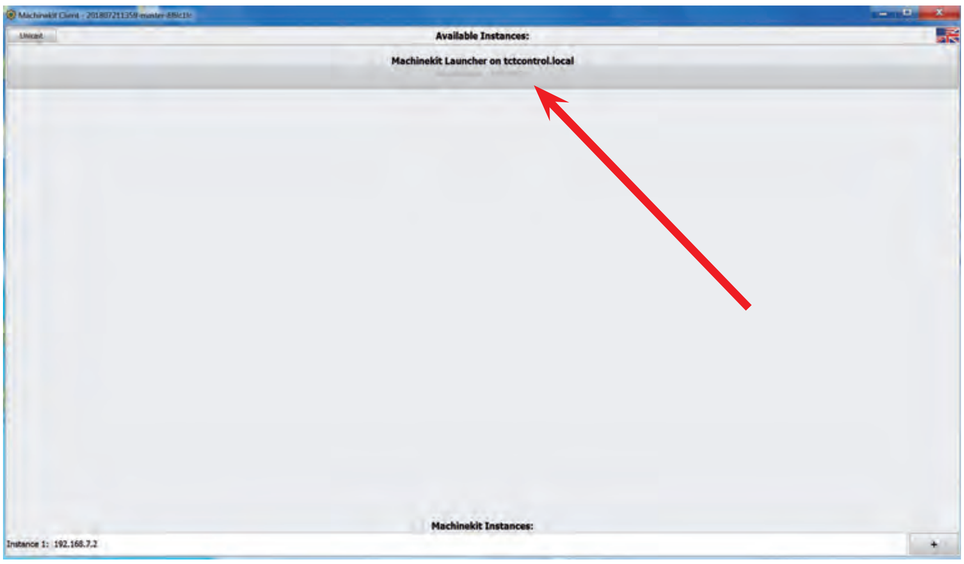


Machinekit-client starts

If you used variant B, than you must enter the IP of the TCTControl, at the first start.
Click at "+"

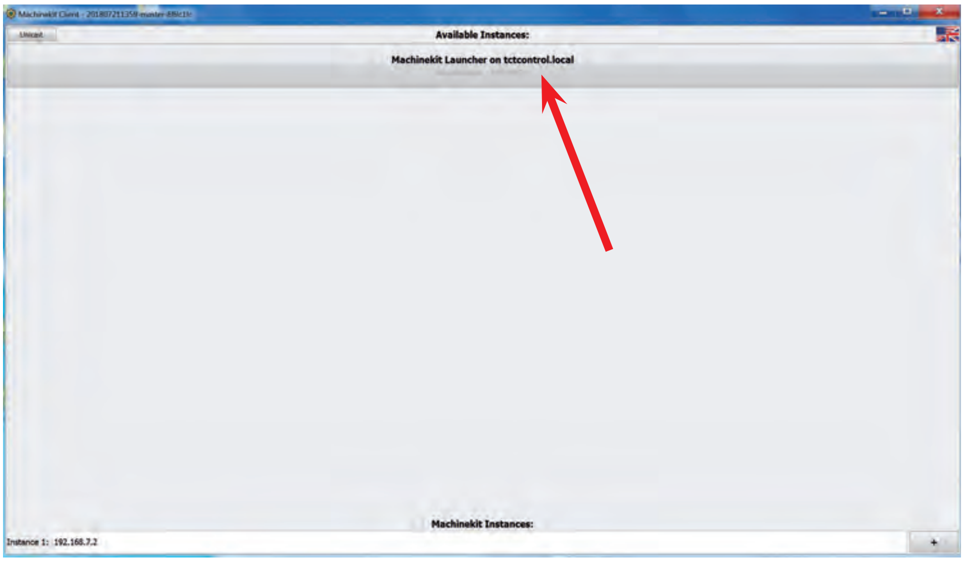
The standard IP of the TCTControl is: **192.168.7.2**

Few seconds later, the machinekit-client shows the connected TCTControl.

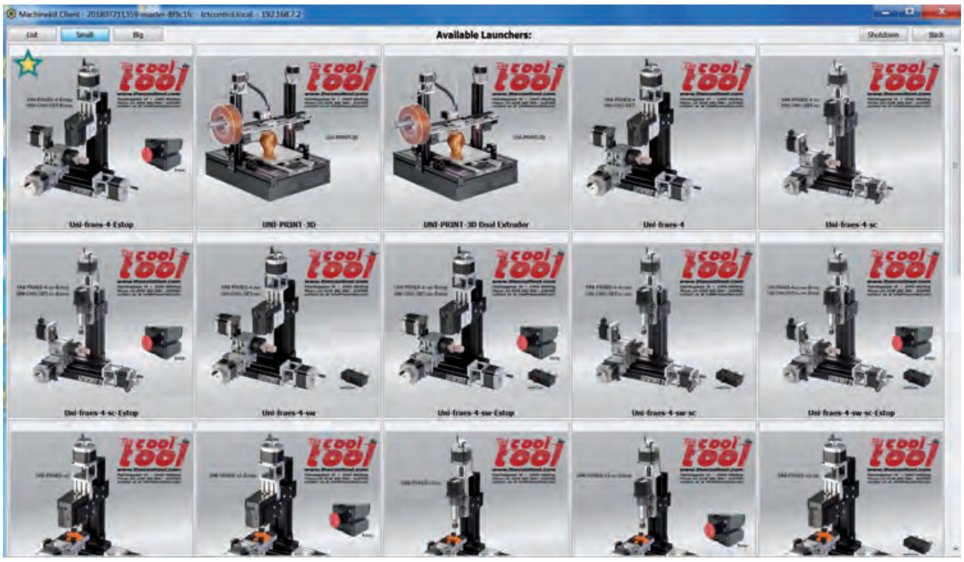


To change the language, click on the flag and select your preferred language.

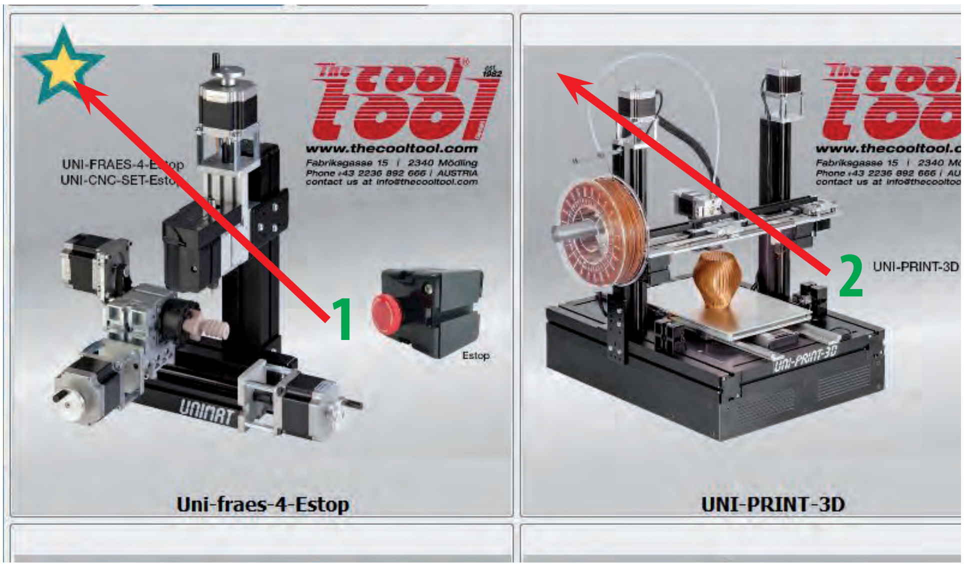
Go to the machine menu.
Click on
“Machinekit Launcher ......”



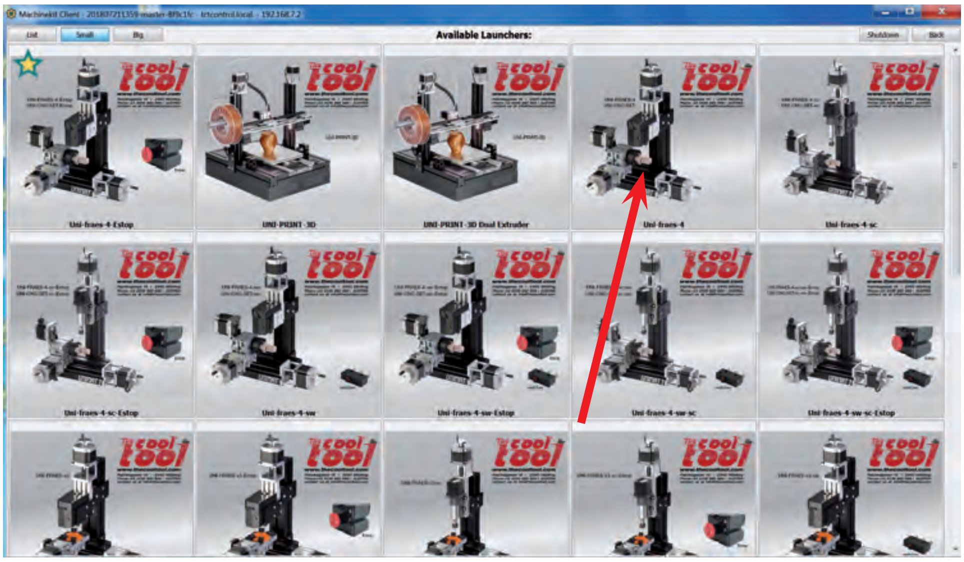
All available machine configurations are now being displayed.



Use the STAR to rank machines up.

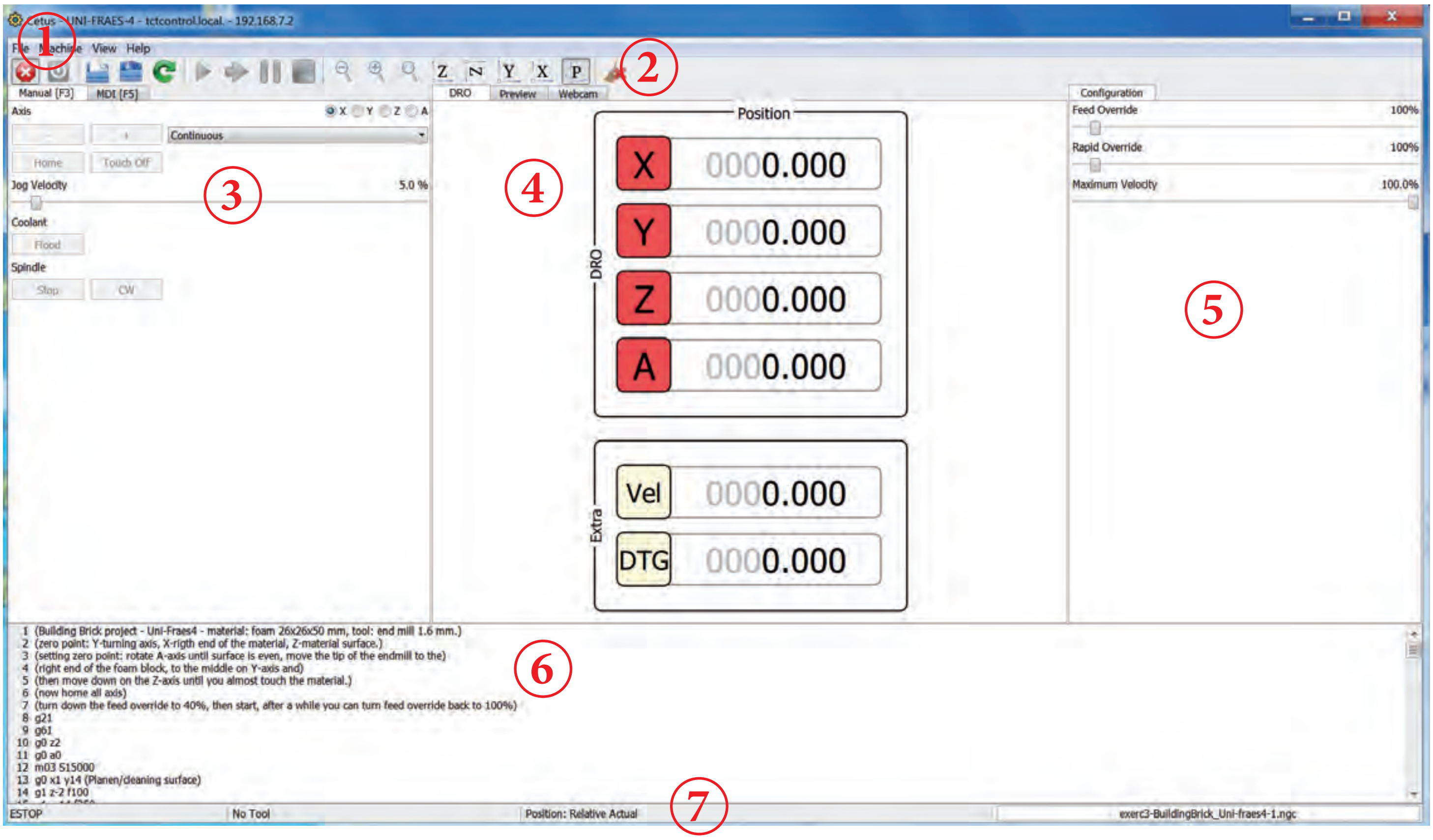


To start a machine configuration click on it.



The starting process takes some seconds.

Cetus - machinekit-client interface
[milling • cutting • turning]



1 .... menu bar

A1) File -> Open File --- To open file from PC or network and upload it to the TCTControl
A2) File -> Open File from Machine ---- To open file that already stored on TCTControl
A3) File -> Edit File with System Editor --- To edit the currently loaded G-Code file.
A4) File -> Reopen file --- After editing an open file, reopen it to update.
A5) File -> Edit Tool Table --- Add and remove tools.
A6) File-> Disconnect from Session --- Go back to the Launcher, without shutting down the running machine configuration.
A7) File-> Shutdown Session --- Shut down the machine configuration and go back to launcher.
A8) File-> Exit User Interface --- Closes the machinekit-client application, the machine configuration remains running.
B) Machine --- Add and remove tools. --- For more details take a look at point 2 „icon bar“
C) View --- Add and remove tools. --- Setup the user interface

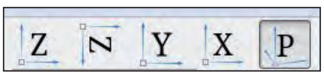
2 .... icon bar

 E-Stop ON/OFF
 machine ON/OFF
 open file from PC

 open file from TCTControl
 reload open file
 start working process

 process only next line
 working process pause
 stop working process


zoom function for preview



change preview view

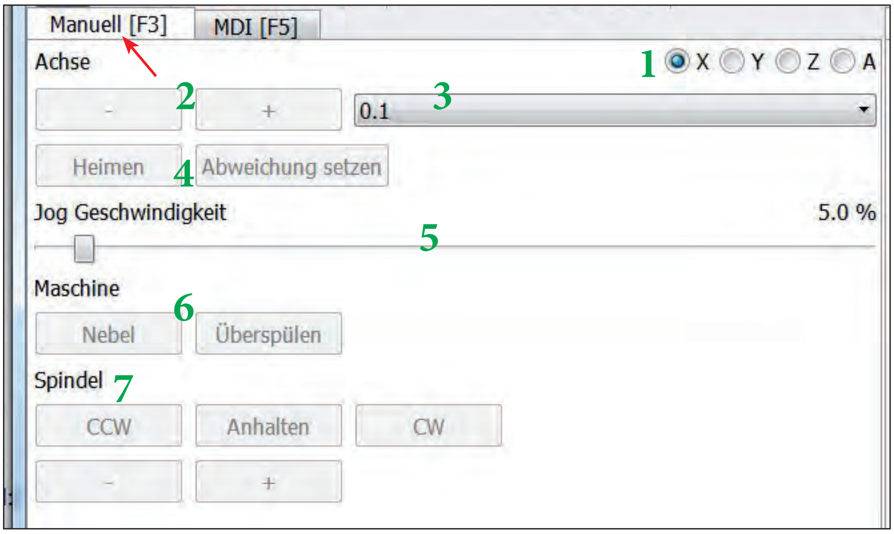
 reset live plot

The tool Education

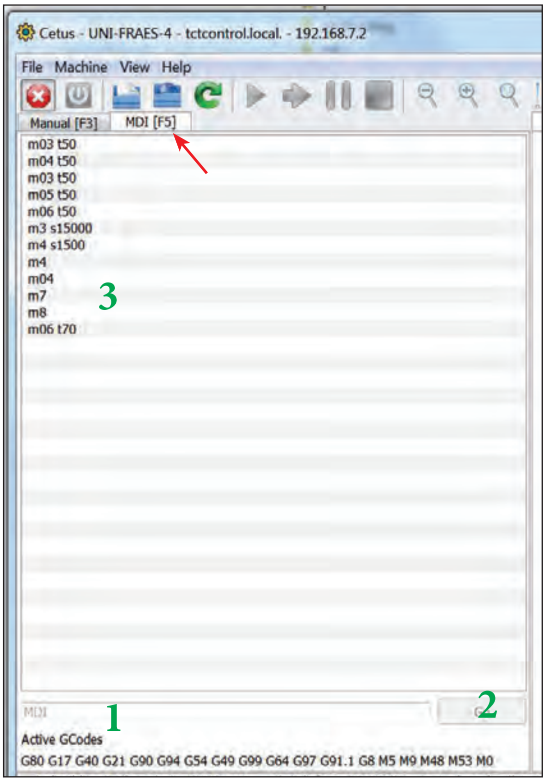
3 .... control area
Can only be used if E-Stop is OFF and machine is ON.

A) Manual

May slightly vary depending on the machine configuration.



1 ...... axis selection
2 ...... direction of movement
3 ...... distance of movement (0.001 up to continuous)
4 ...... referencing the axis (e.g. set zero point, ...)
5 ...... speed of the movement (feed rate)
6 ...... relais (cooling, dust extraction, ...)
7 ...... headspindel (ON/OFF, rpm, direction of rotation
available functions depend on the motor used.



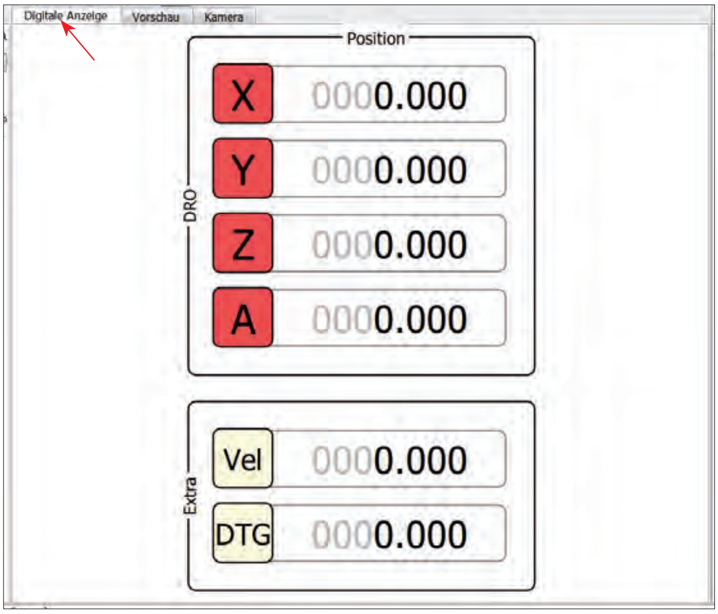
B) MDI (manual data input)
Can only be used if all axes are referenced.

1 ...... enter one command line (e.g. G0 X10 y10)
2 ...... to execute inputted command, click here
3 ...... history of entered command lines

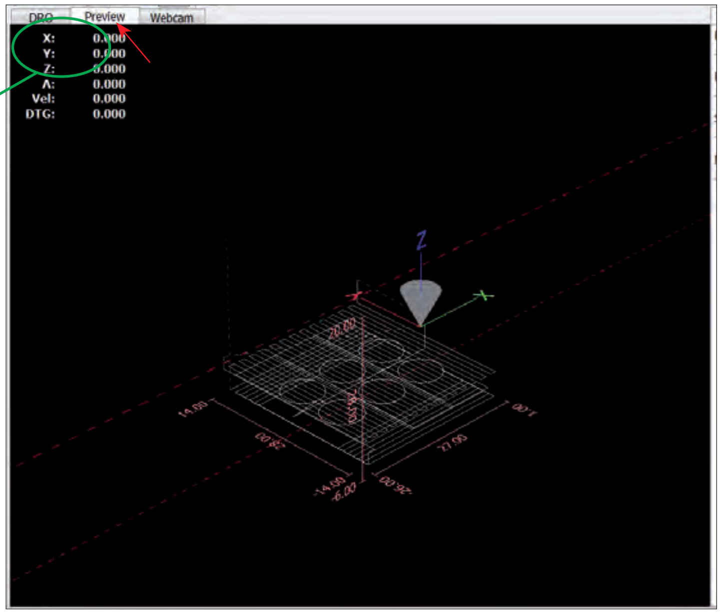
If you click on a line in the history, the line will be entered again (1).

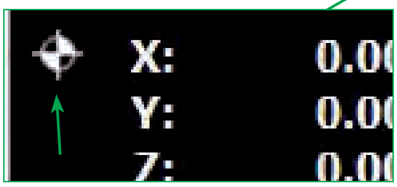
4 .... information area

DRO (digital readout)
X, Y, ... (depends on the machine used) show the actual positions of the axes. If the background is red, the axis is not referenced (homed). If the background is green, the axis is homed.
Vel .... actual speed of the machine movement.
DTG .... distance to go



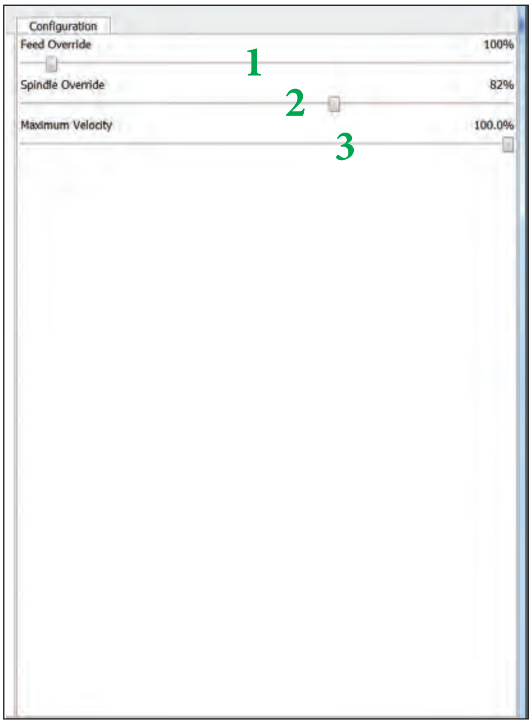
Preview
shows additionally all tool paths of the open G-Code files





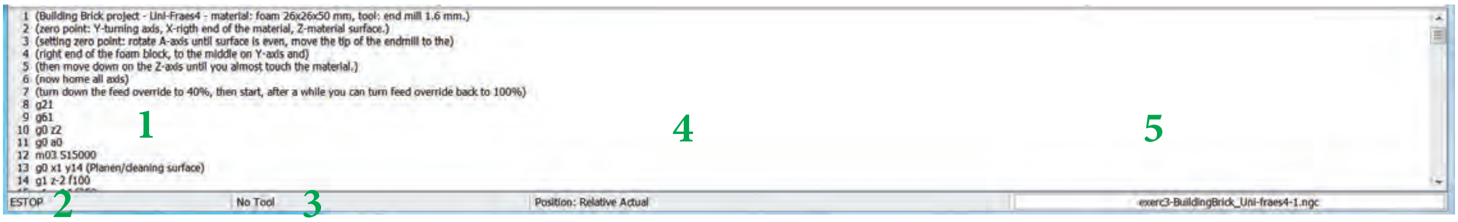
marks referenced (homed) axis.

5 .... Configuration area



1 ...... Feed Override (override programmed feed rate [F] of loaded G-Code file)
2 ...... Spindle Override (override programmed spindle speed [S])
3 ...... Maximum Velocity (override the maximal speed of the machine)

6 .... loaded G-Code file and 7 .... info line



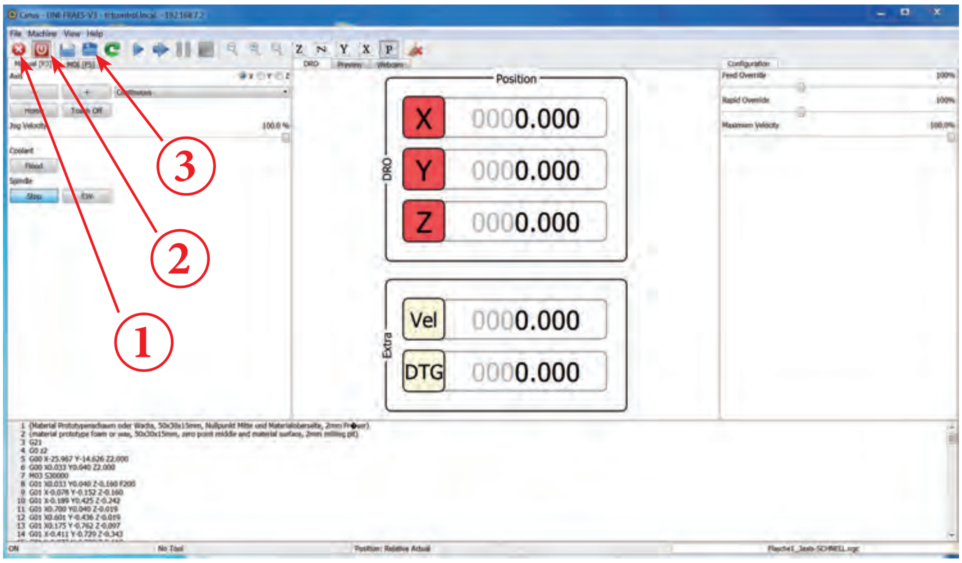
1 ...... Currently loaded G-Code file
2 ...... Machine status ON/OFF
3 ...... Currently loaded tool
4 ...... Position mode
5 ...... File name of the currently loaded G-Code file. Also progress display for running G-Code.

Cetus - start a sample file
[milling • cutting • turning]

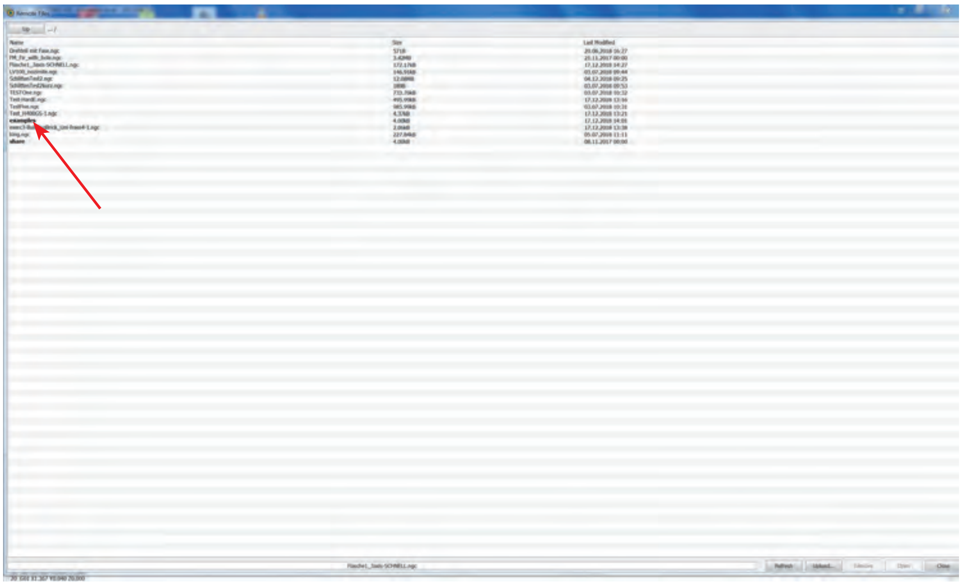
Machine: UNI-FRAES-V3
File: Sample_M1.ngc
Raw material: Special colored Acrylic, 50x50x3 mm[art.no.: 166PLEXS]
Tool: 1.6 mm end mill

•) Mark the center of the square (raw material)
•) Use the clamping jaws to fix the raw material. Protect the machine table by placing a second plate (plywood or acrylic, thickness min. 3 mm) beneath the raw material.

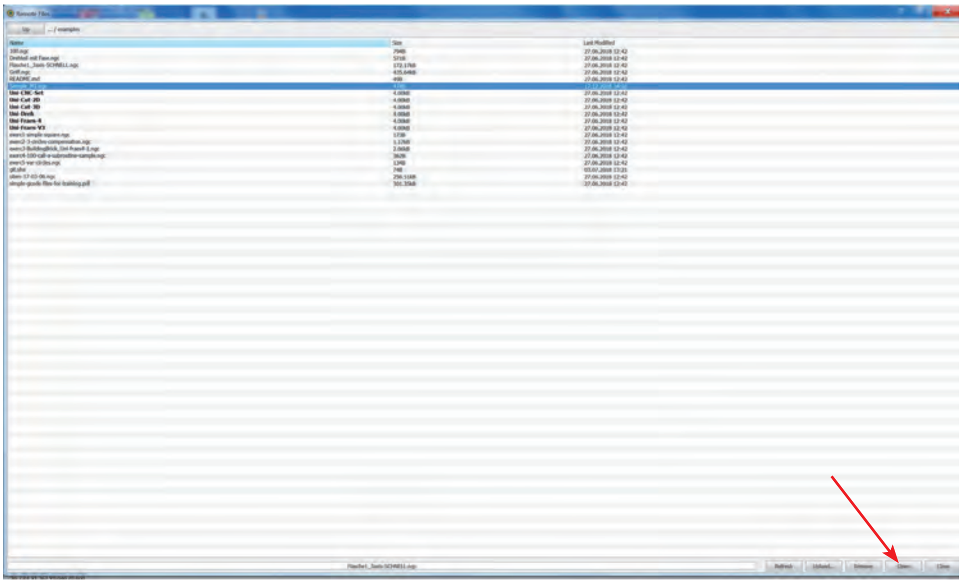
•) Start UNI-FRAES-V3 (machinekit-client)
1) Deactivate the E-Stop
2) Machine power ON
3) Open G-Code file stored on machine



double click on EXAMPLES



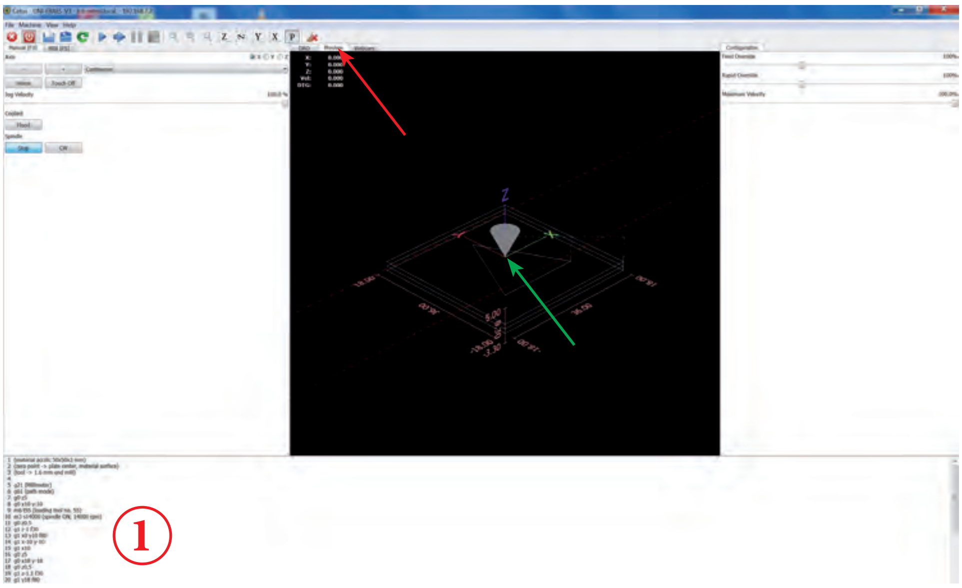
1) select the sample file
[Sample_M1.ngc]
2) click on OPEN



switch to PREVIEW

Origin of the coordinate system = workpiece zero point

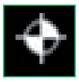
1 ... Currently loaded G-Code
Comments can be written in brackets ().

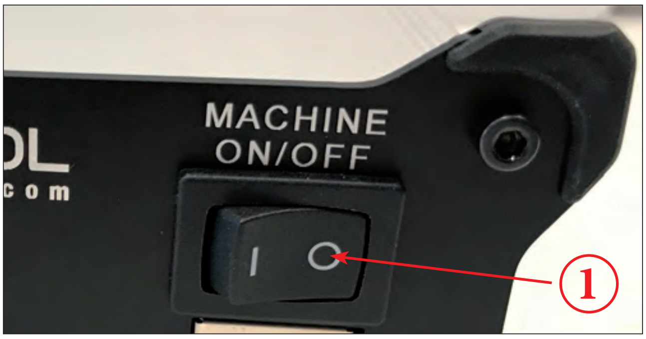


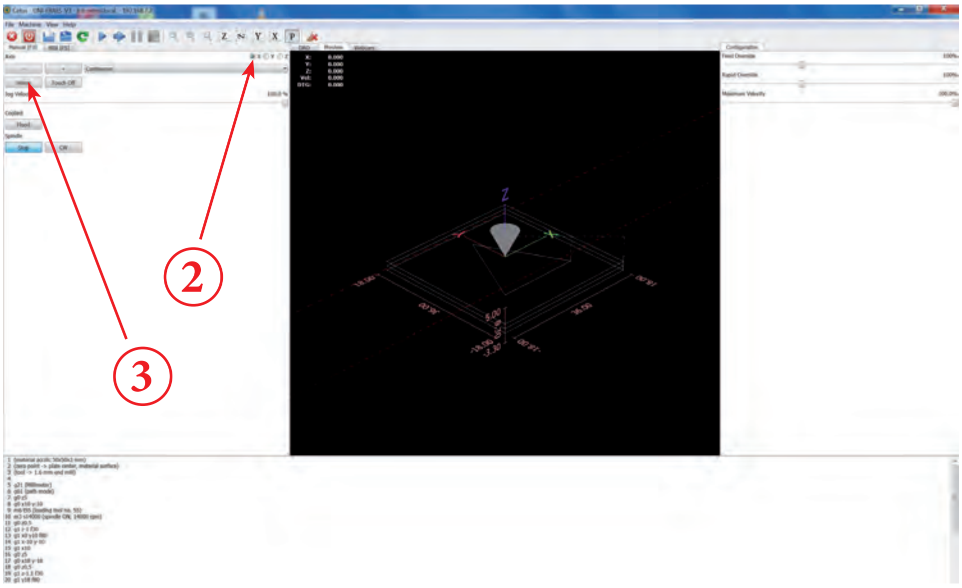
•) Simulate the G-Code

1 ... Turn MACHINE OFF!
2 ... Select the X axis
3 ... click HOME

Repeat the process with Y and Z axis.

Now all 3 axes are homed. 

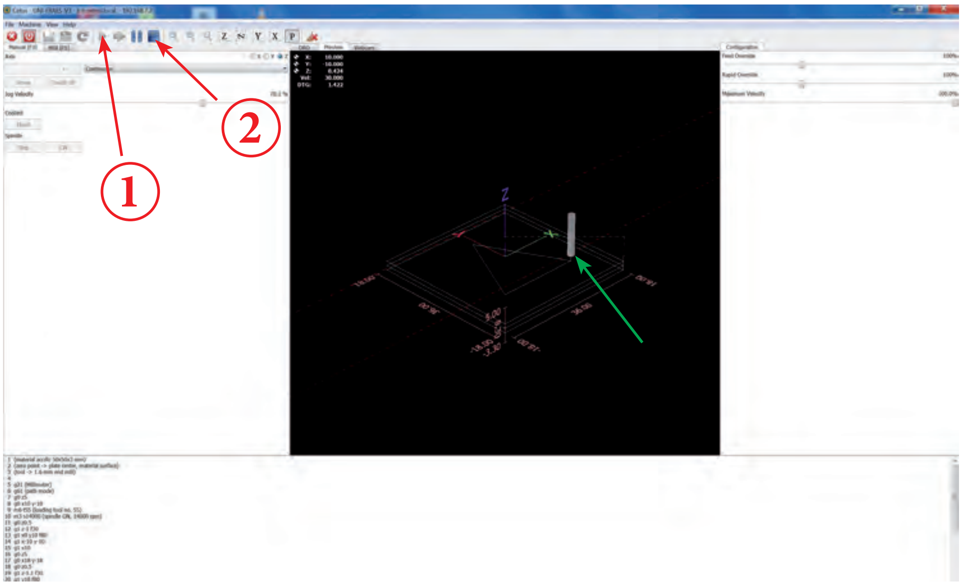




1) click on „PLAY“

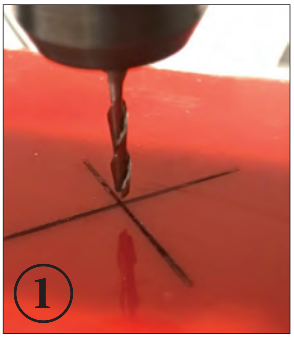
The simulation is running.

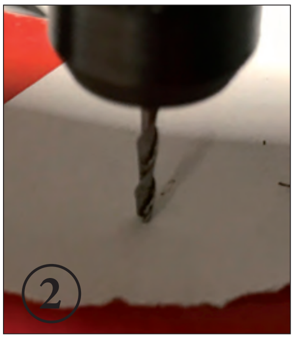
2) click on „STOP“ to stop the simulation

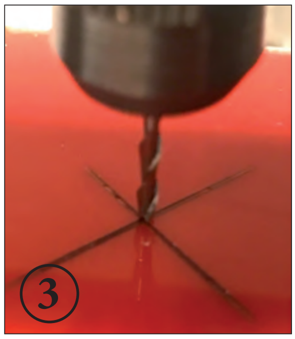


Cetus - start a sample file 4.2.2.2
[milling • cutting • turning]

•) Run the G-Code at the milling machine



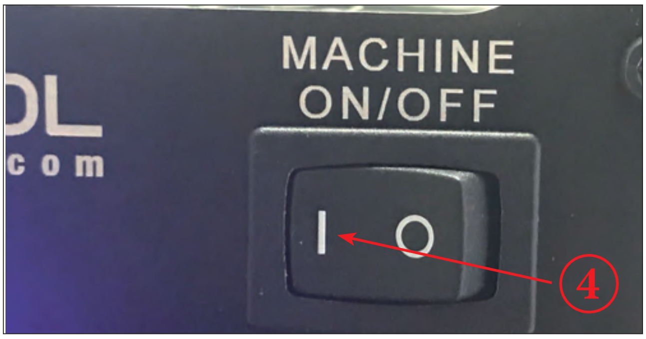




1 ... move the milling head (X/Y axis) to the zero position at the raw material.
2 ... put a thin paper between raw material end milling head. Go down with the Z axis and stop if the paper get stuck.
3 ... now the milling head stay at the zero position.

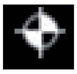
For that work you can move the axes manual (using the hand wheels - MACHINE at the TCTControl must be OFF!
or
Use the Cetus software interface to move the axes - MACHINE at the TCTControl must be ON!

4 ... turn MACHINE ON at the TCTControl

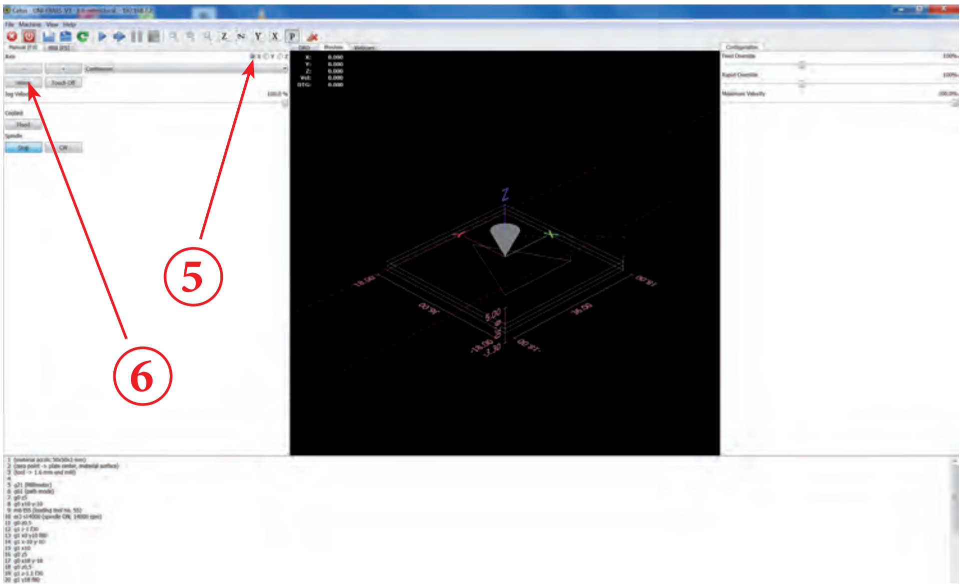


5 ... select the X axis
6 ... click HOME

Repeat the process with Y and Z axis.

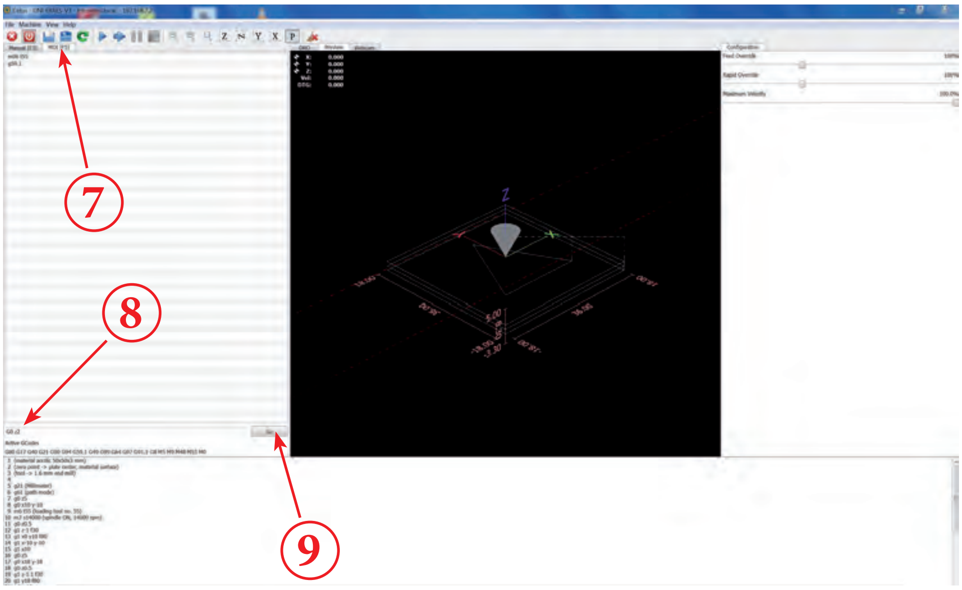
Now all 3 axes are homed. 

If a axis is already homed, no problem, home it again.



Cetus - start a sample file 4.2.2.2
[milling • cutting • turning]

7 ... switch to MDI
8 ... enter: G0 z2
9 ... click GO

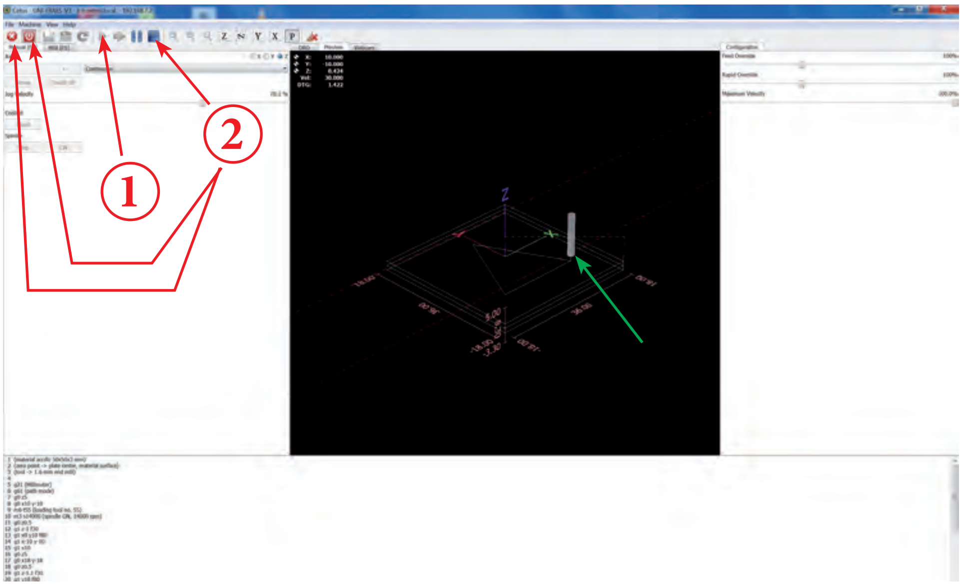


10 ... turn the milling motor ON

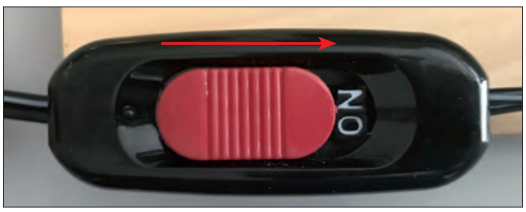
1) click on „PLAY“

The milling process starts.

2) to stop the machine (milling process) click on one of the three icons
a) E-stop
b) Machine power
c) Stop



When the milling operation is finished turn the milling motor OFF!

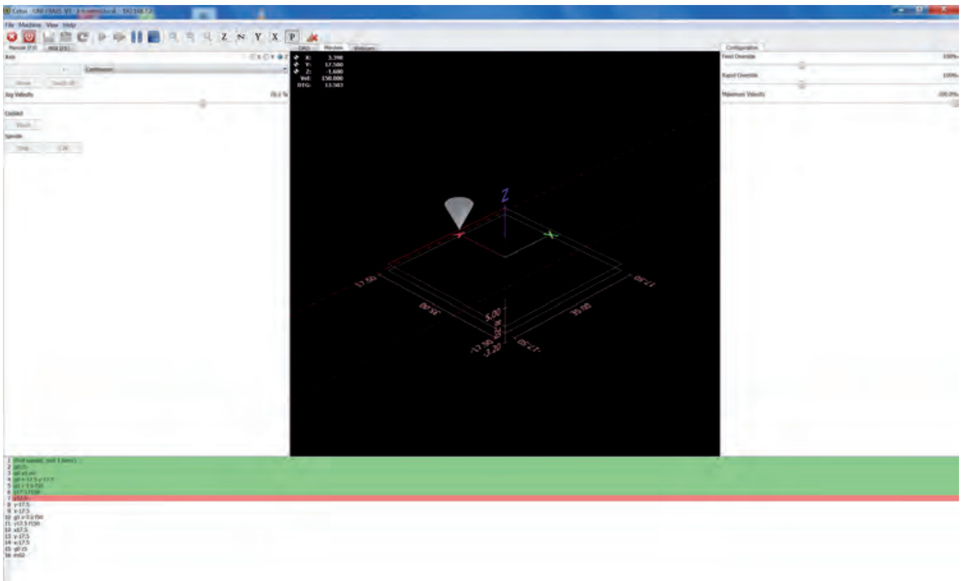


Exercise 1

Machine: UNI-FRAES-V3
File: .../examples/exerc1-simple-square.ngc
Raw material: Special colored Acrylic, 50x50x3 mm[art.no.: 166PLEXS]
Tool: 1.6 mm end mill

•) Mark the center of the square (raw material)
•) Use the clamping jaws to fix the raw material. Protect the machine table by placing a second plate (plywood or acrylic, thickness min. 3 mm) beneath the raw material.

It is similar to the Sample_M1.ngc file



Exercise 2

Machine: UNI-FRAES-V3
File: .../examples/exerc2-3circlrs-g41_g42.ngc
Raw material: Special colored Acrylic, 50x50x3 mm[art.no.: 166PLEXS]
Tool: 1.6 mm end mill

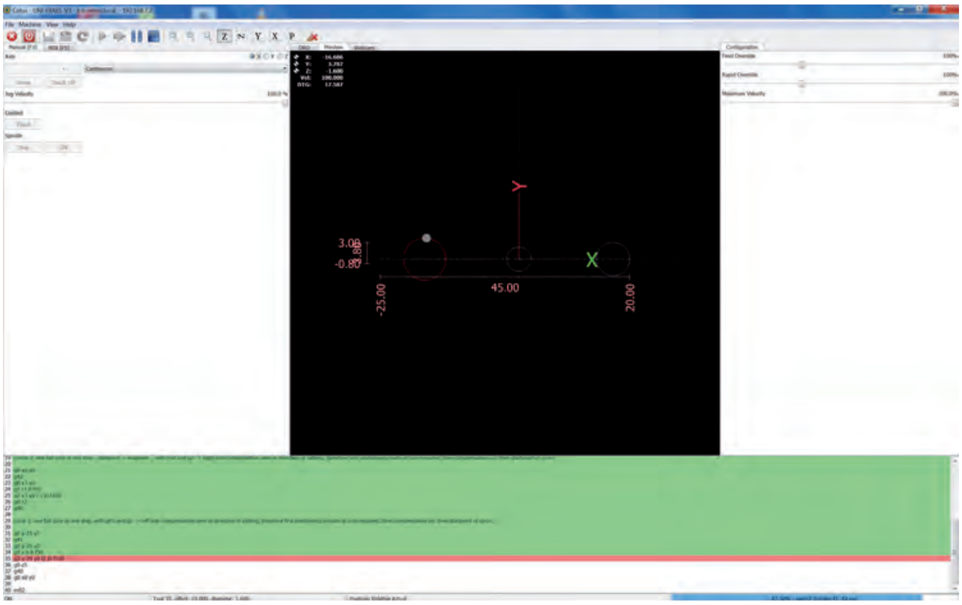
•) Mark the center of the square (raw material)
•) Use the clamping jaws to fix the raw material. Protect the machine table by placing a second plate (plywood or acrylic, thickness min. 3 mm) beneath the raw material.

Programmed diameter of the circles is 6 mm.

1st circle (on right possition):
It is without tool diameter compensation. The diameter of the cut out part is 6 - 1.6 = 4.4 mm
The diameter of the cuted hole at the raw material is 6 + 1.6 = 7.6 mm

2nd circle (in the middle):
With tool diameter compensation. The diameter of the cut out part is 6 - 1.6 - 1.6 = 2.8 mm
The diameter of the cut hole in the material is 6 mm

3rd circle (on the left possition):
It is with tool diameter compensation. **The diameter of the cut out part is 6 mm.**
The diameter of the cut hole in the material is 6 + 1.6 + 1.6 = 9.2 mm

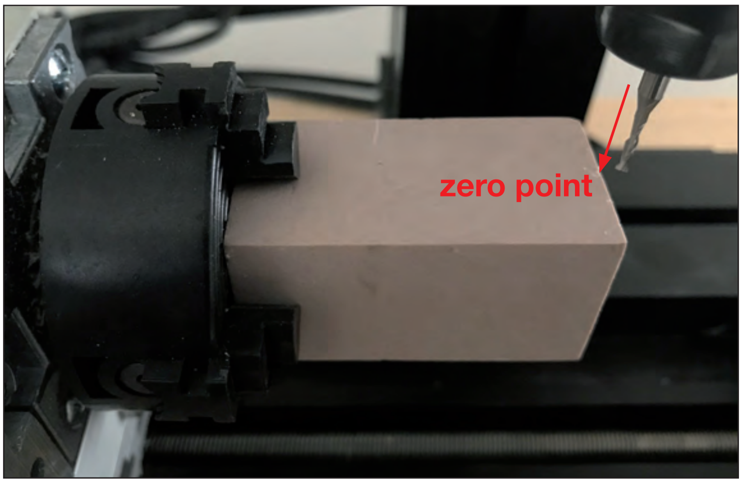


Experiment
Now replace the milling head at your milling machine - use for example a 1.2 mm end mill.
For tool number take a look at your tool table (t50 = 1.2 mm end mill)
Now „File =>Edit file with System Editor", change the tool number from T55 to T50
Save as => exerc2-3circlrs-g41_g42-A.ngc ==> on the Desktop ==> open the new files in CETUS
Use a new raw material and cut the circles again (don't forget to update the zero point).
The diameters of the cuted hole (circle 2) and the cut out part (circle 3) will be also 6 mm.
It was only one small change at the G-Code file necessary!

Exercise 3

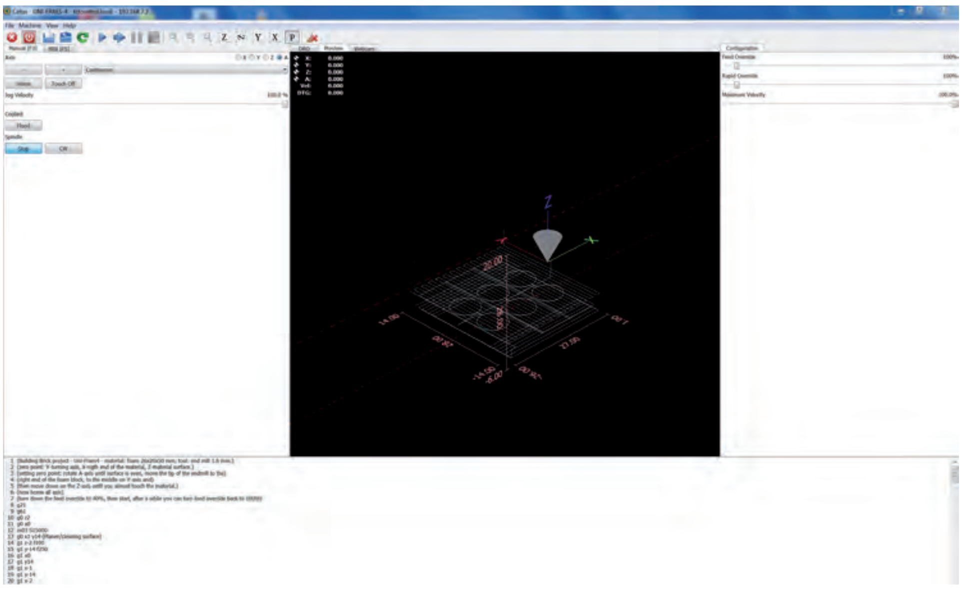
Machine: UNI-FRAES-4
File: .../examples/exerc3-BuildingBrick_Uni-fraes4-1.ngc
Raw material: Special milling foam, 50x25x25 mm[art.no.: 166FOAM S]
Tool: 1.6 mm end mill

•) Mark the zero point (raw material)
•) Use the 4 jaw chuck to fix the raw material.



After successfully milling this project the first time, try the following:

a) Go to „MDI“
b) Enter
G0 a180 => „GO“
c) Go to „MANUAL“
d) Home A axis again!
e) Now you can start the milling file again.

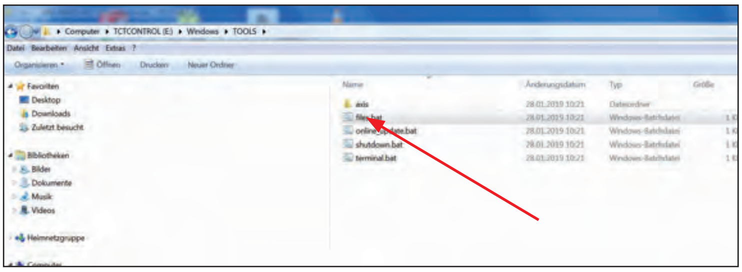


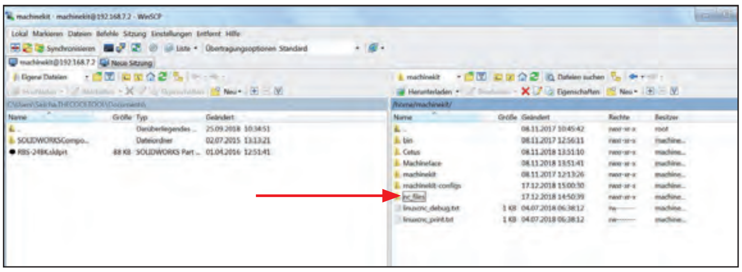
Exercise 4

Machine: UNI-FRAES-V3
File: .../examples/exerc4-100-call-a-subroutine-sample.ngc
Raw material: plywood or acrylic, dimensions depending on the selected parameters
Tool: 1.6 mm end mill

Preparation

•) Start the files.bat file at TCTControl/Windows/TOOLS/
•) When asked for a password use: machinekit
The left part of the „WinSCP" window is the file system of your PC. The right part is the file system of the TCTControl.
•) Go to nc_files/examples
•) Copy following files to the folder nc_files:
100.ngc
exerc4-100-call-a-subroutine-sample.ngc

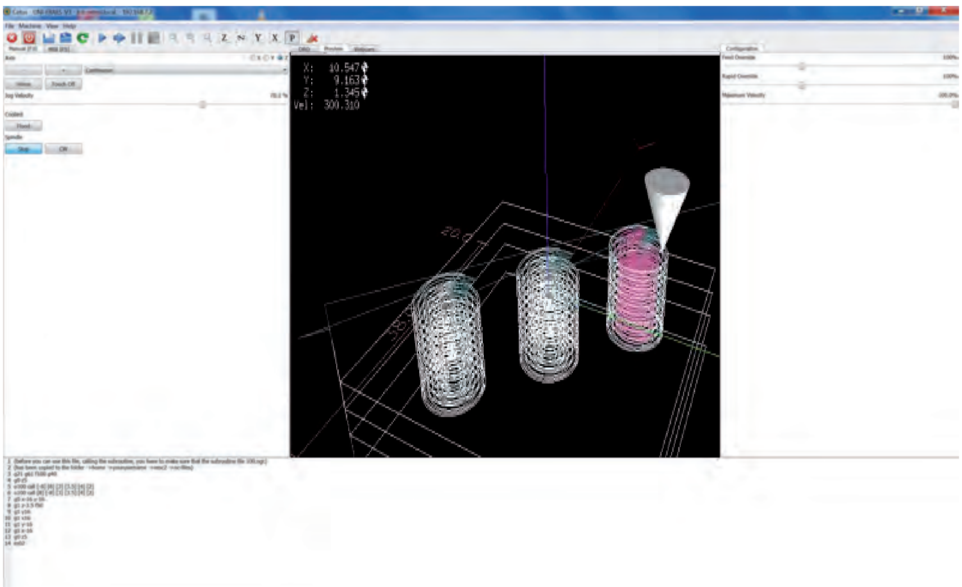




•) Open the exerc4-100-call-a-subroutine-sample.ngc file in CETUS

See comments in file exerc4-100-call-a-subroutine-sample.ngc !!!
To learn what the parameters like [-8] [8] [3] [3.5] [4] [2] mean, please read the comments in 100.ngc.

Experiment with different parameters.

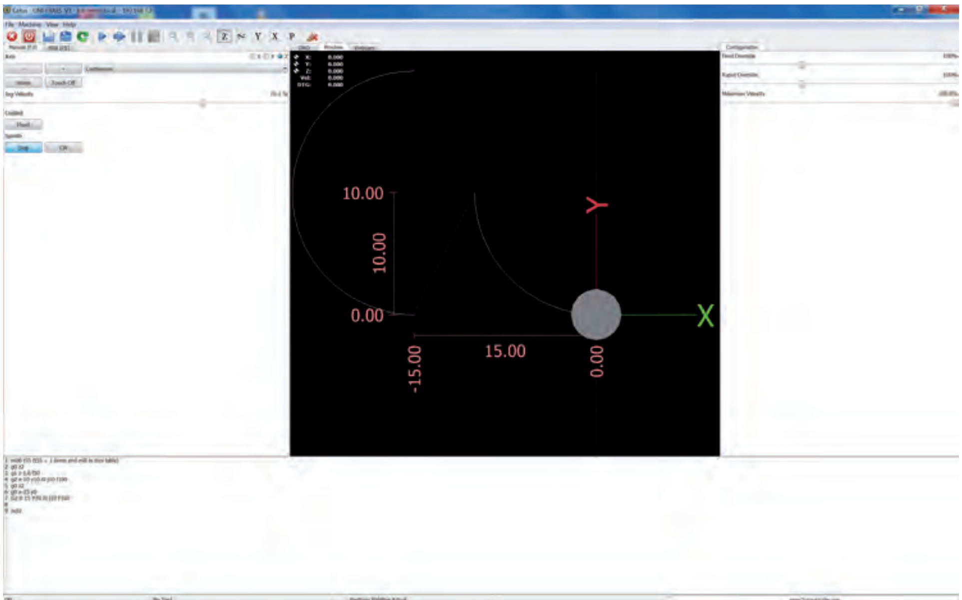


Exercise 5

Machine: UNI-FRAES-V3
File: .../examples/exerc5-var-cicles.ngc
Raw material: Special colored Acrylic, 50x50x3 mm[art.no.: 166PLEXS]
Tool: 1.6 mm end mill

•) Mark the center of the square (raw material)
•) Use the clamping jaws to fix the raw material. Protect the machine table by placing a second plate (plywood or acrylic, thickness min. 3 mm) beneath the raw material.

Showing circle segments with G2.

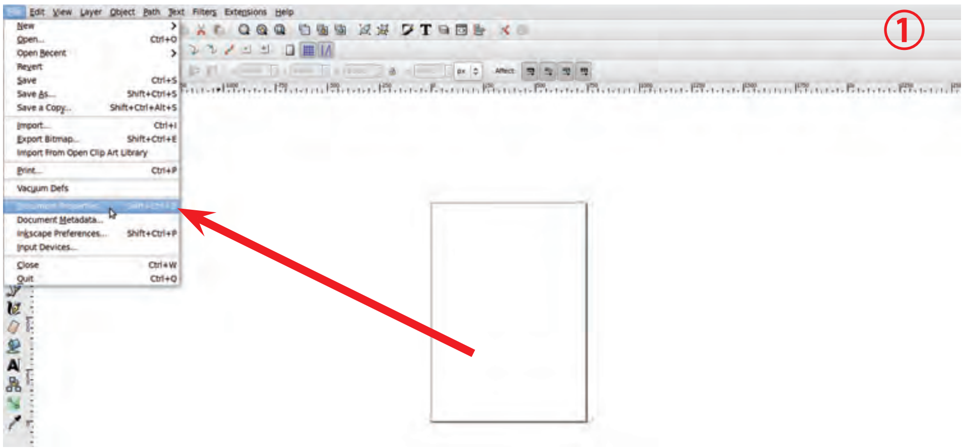


https://inkscape.org

Sample 1 - engraving into acrylic

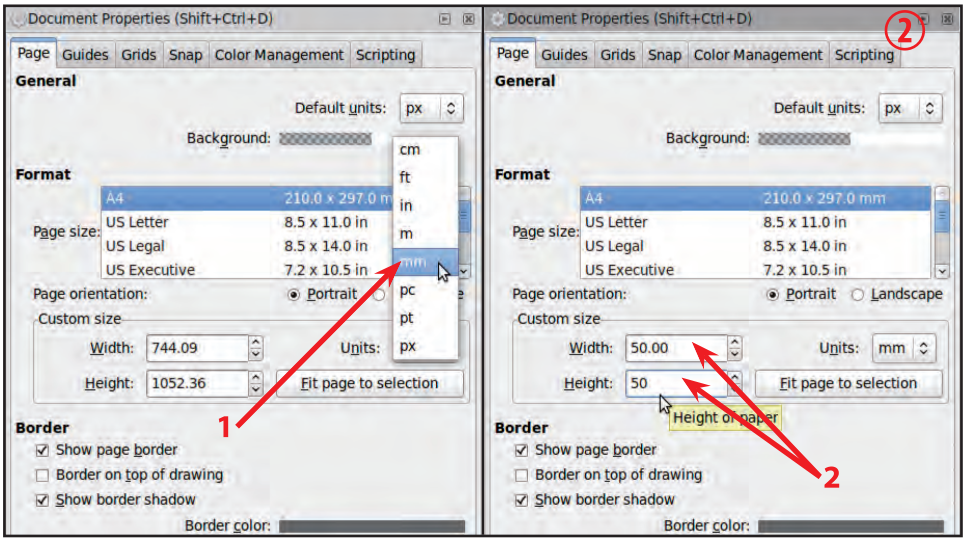
raw material: acrylic plate ~ 50 x 50 x 3 mm
tool: end mill ø1.0 mm
optional: end mill ø1.6 mm

1) open "Inkscape®" then click on "File" --> "Document Properties...".

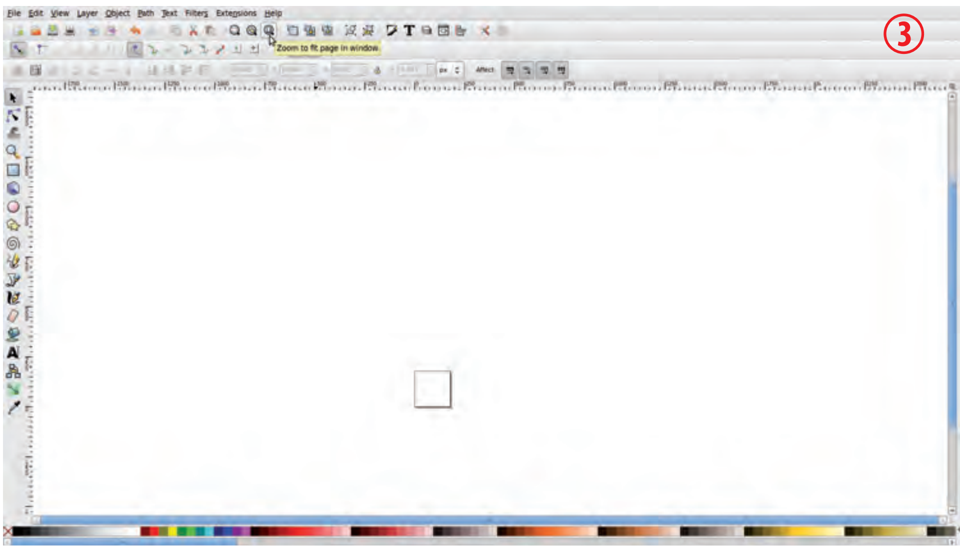


2) select at "Custom size" --> "Units" --> "mm".

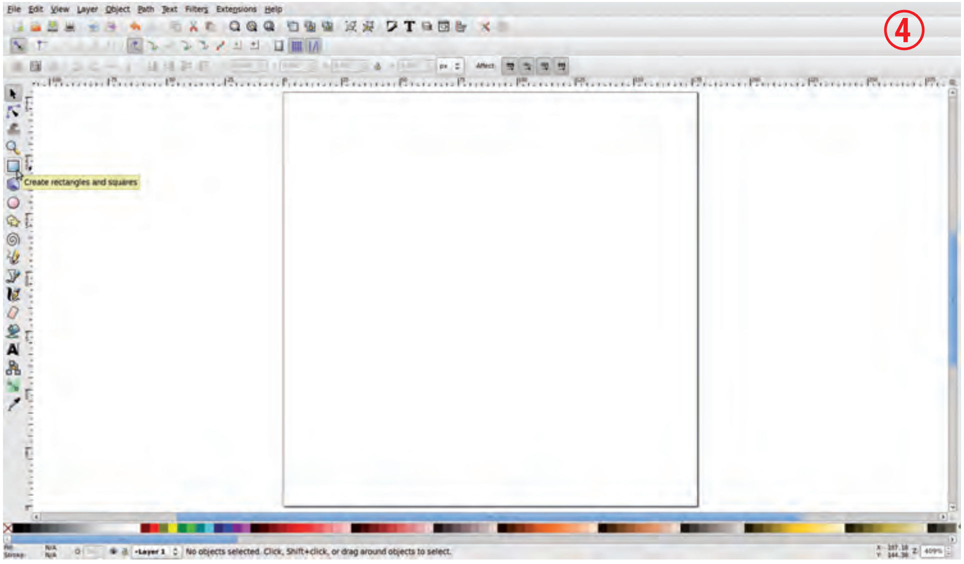
enter for both "Width" and "Height" 50 after that close the window.



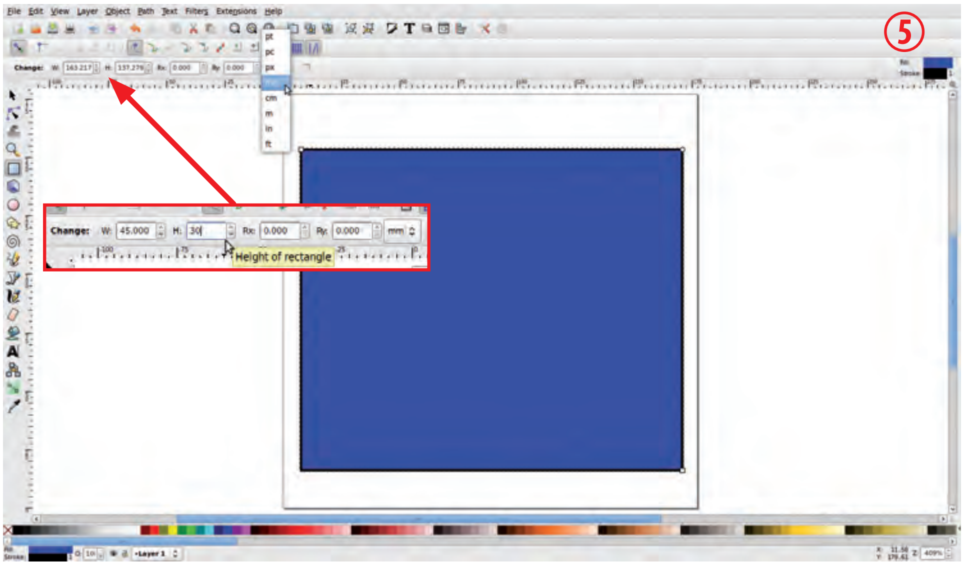
3) click on the magnifier icon "Zoom to fit page in window".



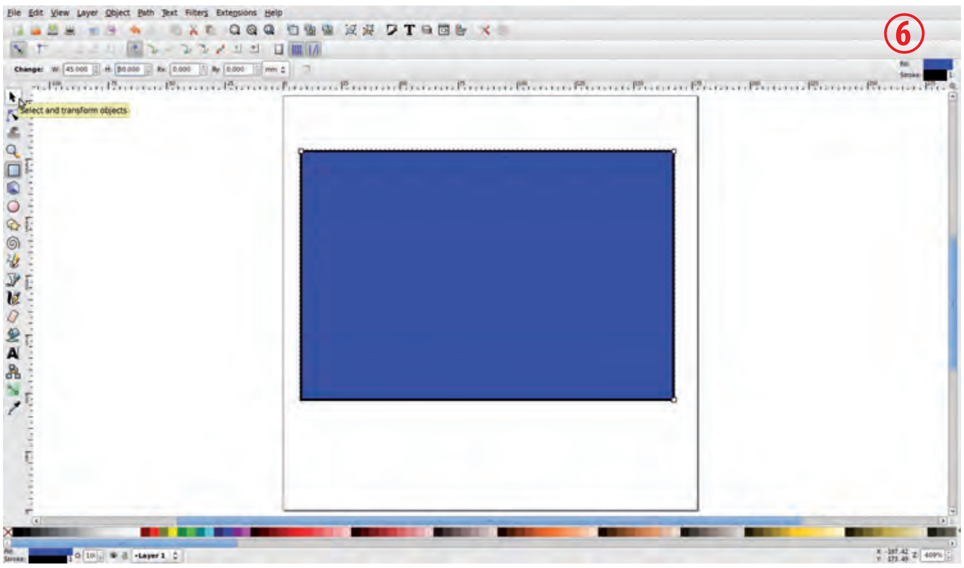
4) click on the square icon "Create rectangles and squares".



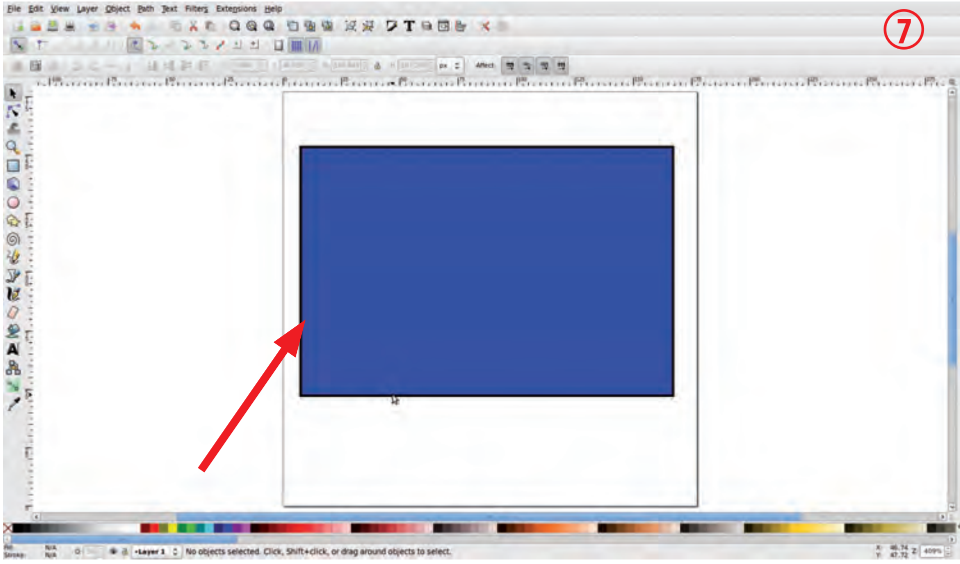
5) at "Change" select "mm" then enter for "W" (width) 45 and for "H" (height) 30.



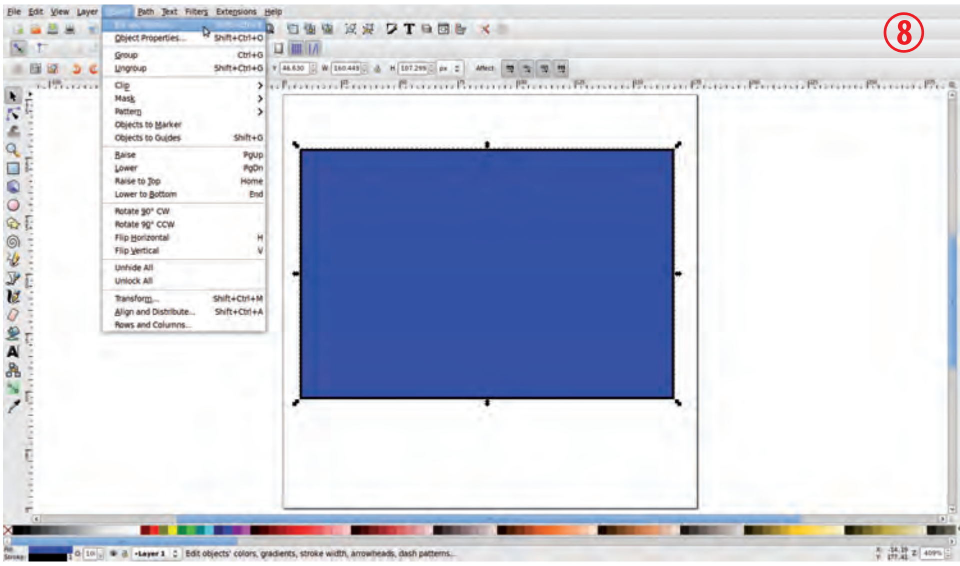
6) click on the arrow "Select and transform objects".



7) select the rectangle (click on it).

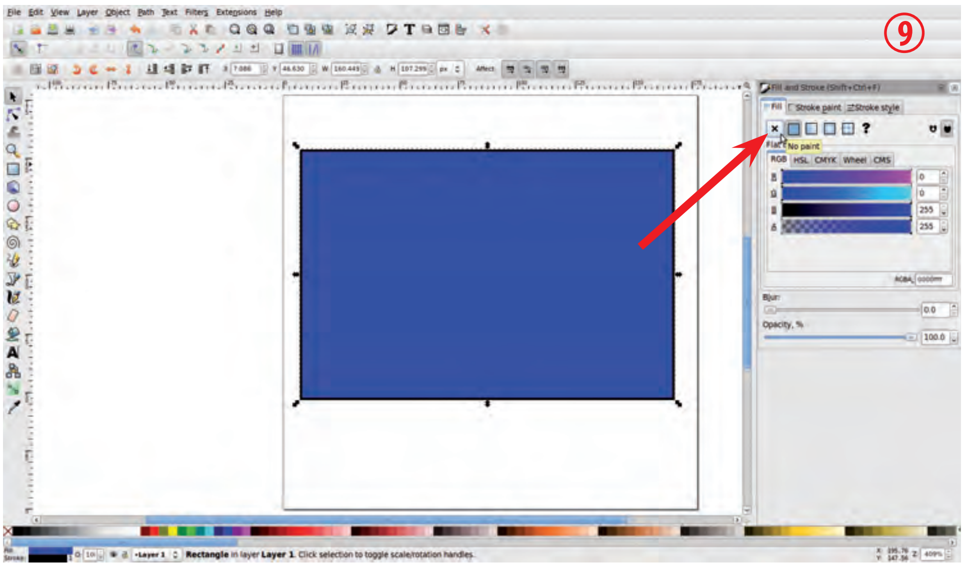


8) Now that the rectangle is marked click on "Object" --> "Fill and Stroke".



9) in the window "Fill and Stroke" select "Fill" and click on "X" (No paint).

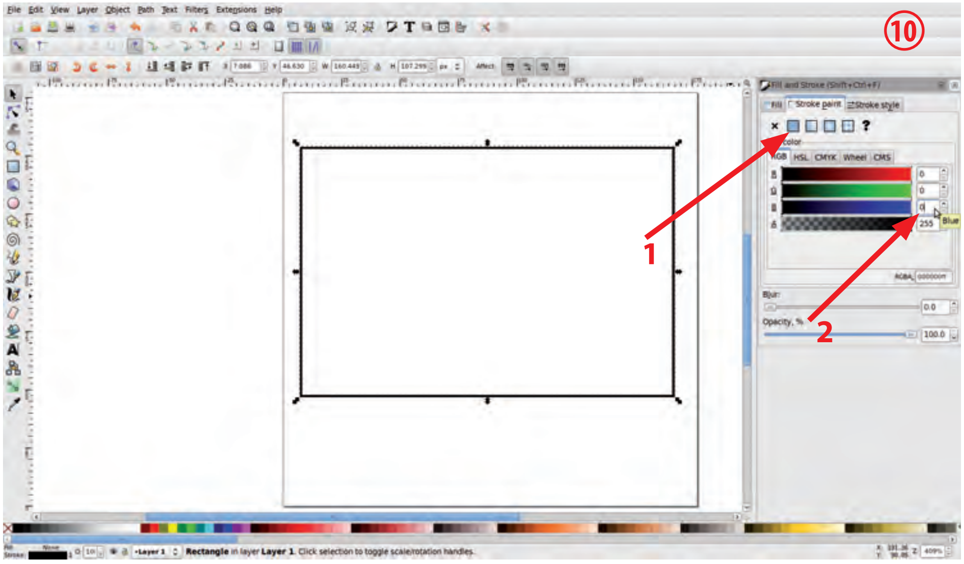
Attention:
The rectangle must be selected (marked).



https://inkscape.org

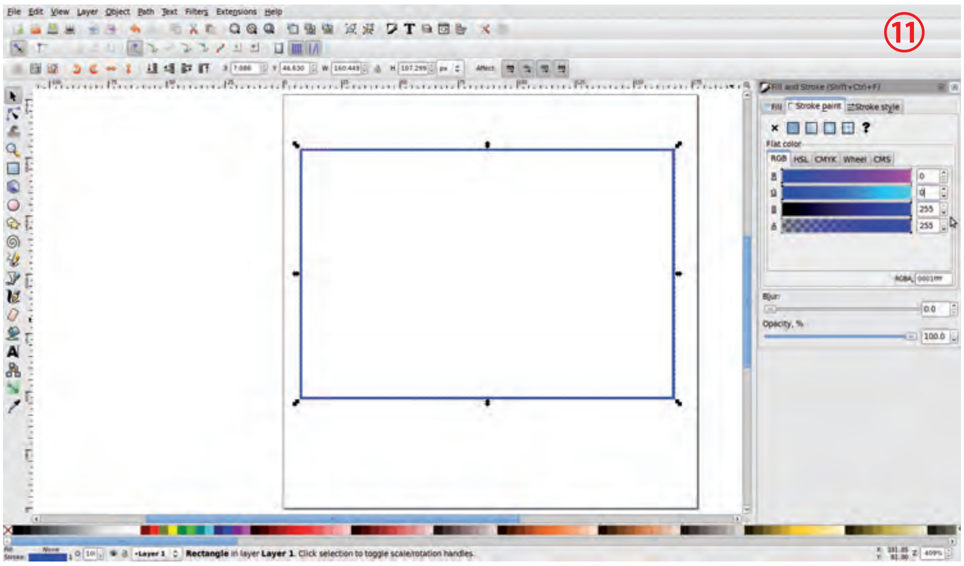
10) switch to “Stroke paint” and click on “Flat color” (1), then click “Blue” (2).

Attention:
The rectangle must be selected (marked).



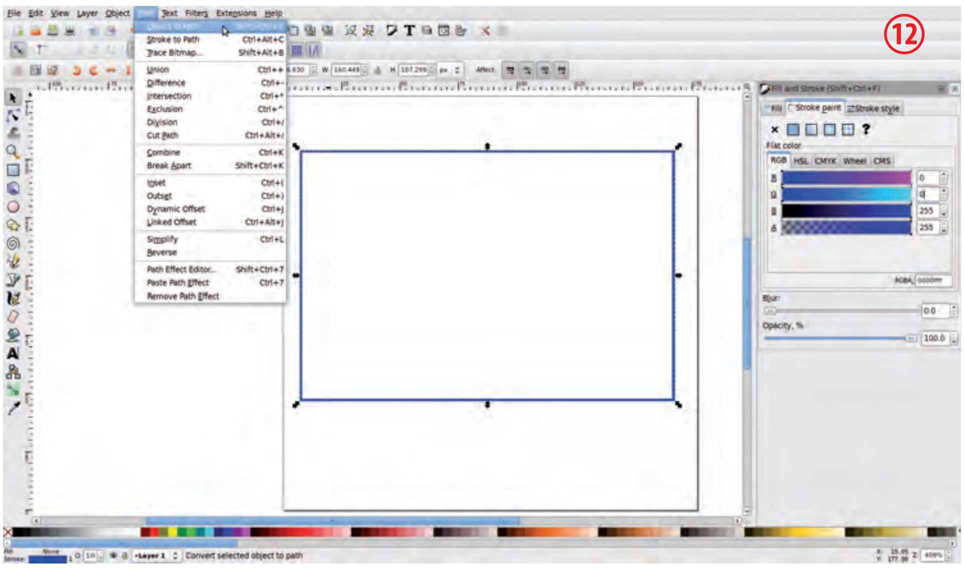
11) write “255” and press “Enter”. The contour color of the rectangle is now blue.

Attention:
The rectangle must be selected (marked).



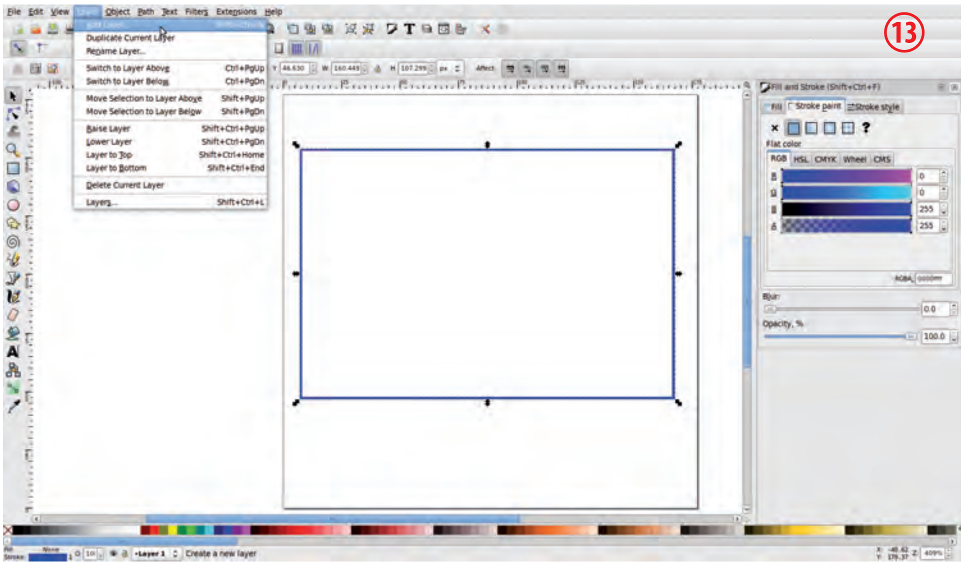
12) click on “Path” --> “Object to Path” (vector). Now the rectangle is a vector graphic.

Attention:
The rectangle must be selected (marked).

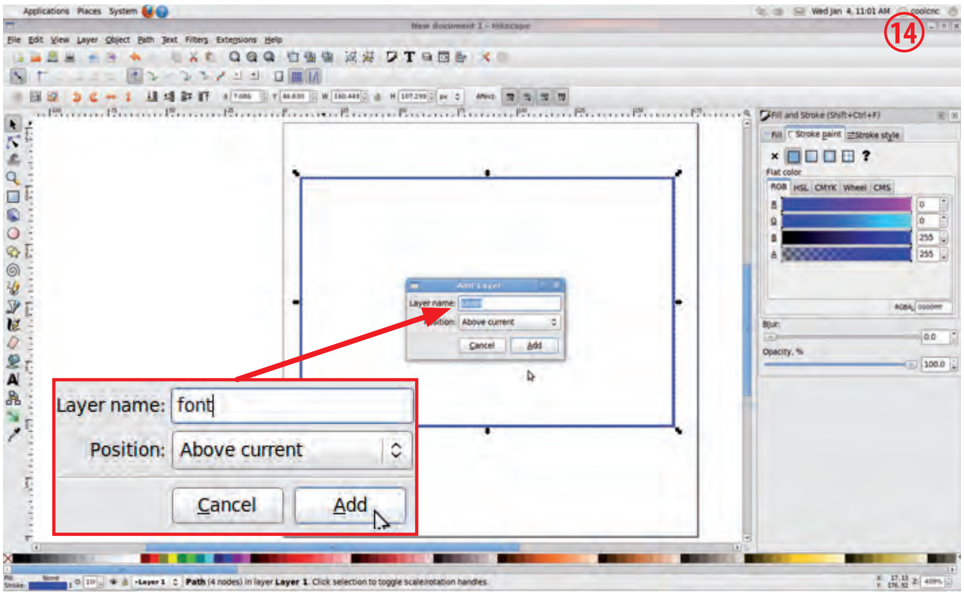


Create a new layer for the labeling:

13) click on “Layer” --> “Add Layer”.

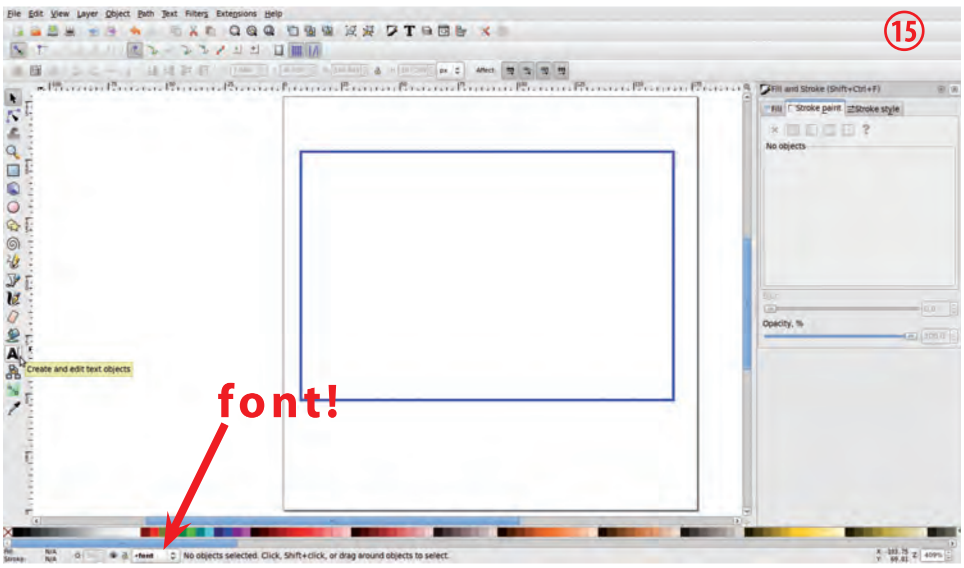


14) enter following name for the layer “font”, then click on “Add”.



15) click on “A” icon “Create and edit text objects”.

Attention:
Layer “font” must be activated.



The tool Education

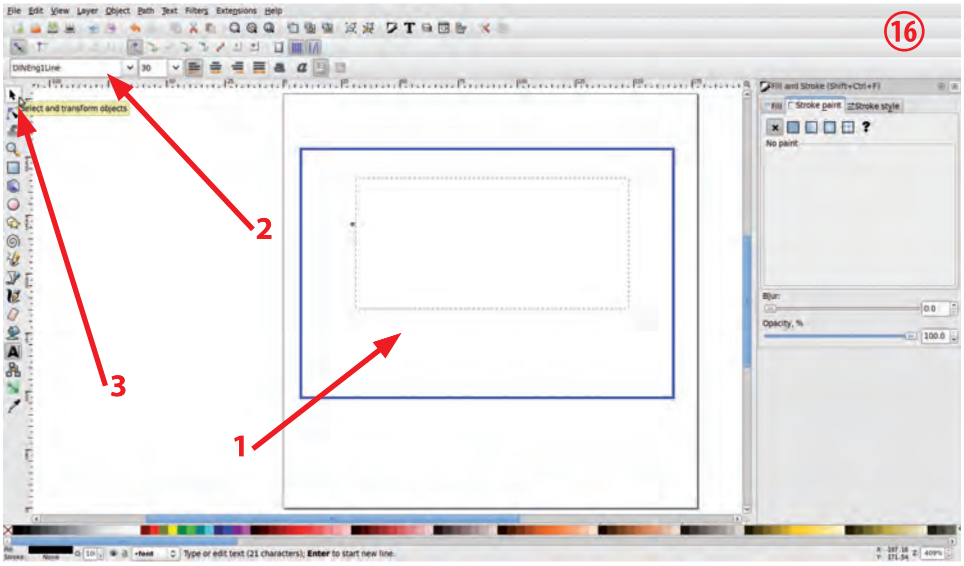
https://inkscape.org

16) click into the blue rectangle, then select a (Single Line Fonts) - for example "DINEng1Line". Write your text - for example:

The Cool Tool

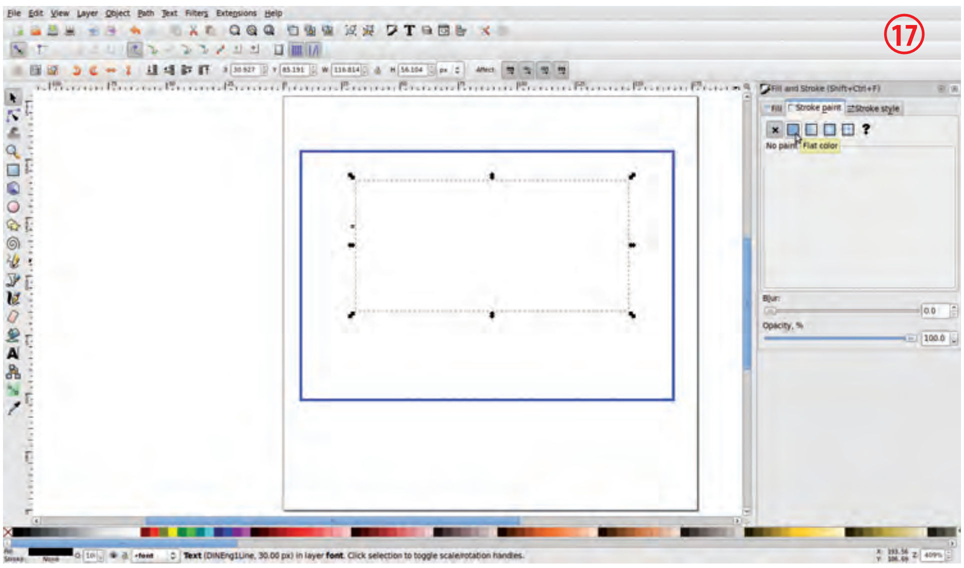
CoolCNC

After that click on the arrow icon "Select and transform objects".

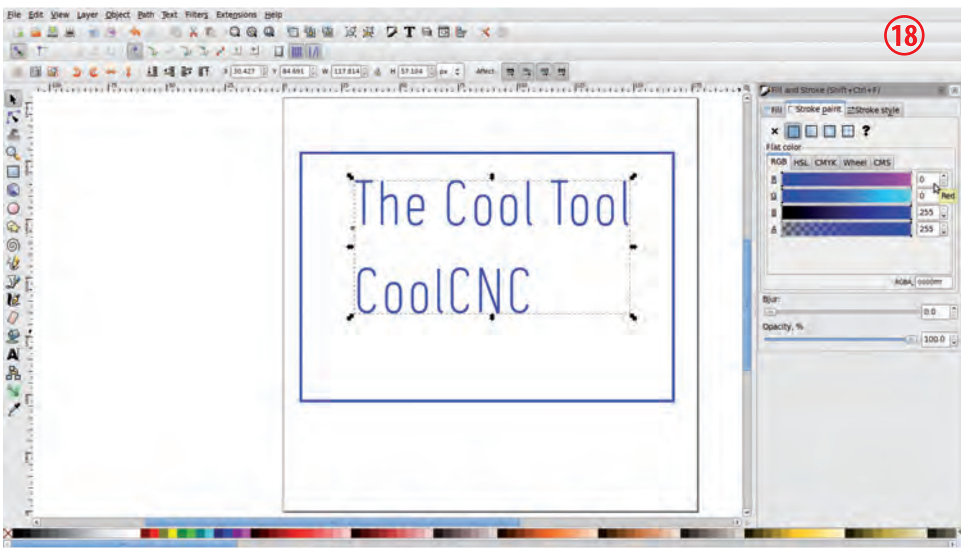


17) click on "Stroke paint" --> "Flat color".

Attention:
The text field must be activated (marked).



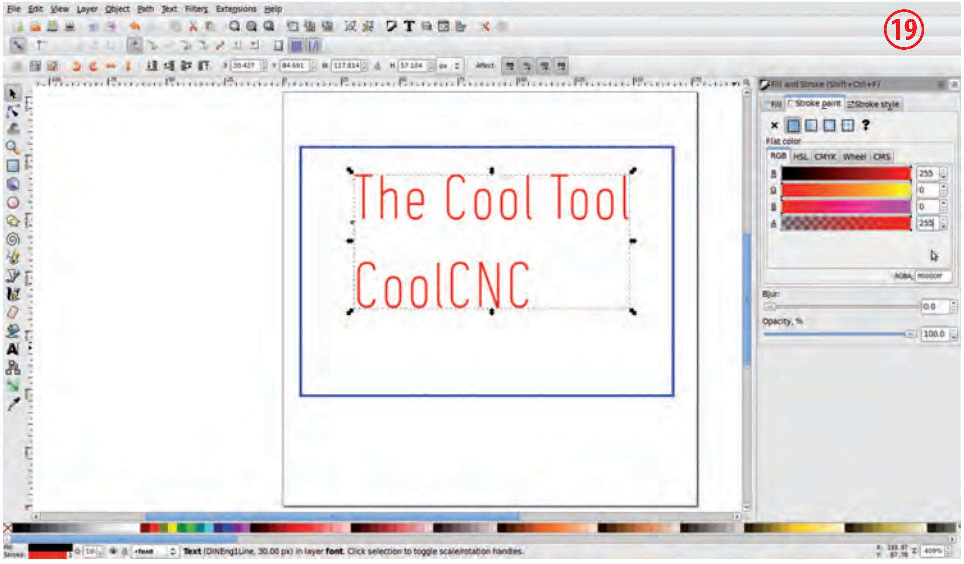
18) click on "red"



https://inkscape.org

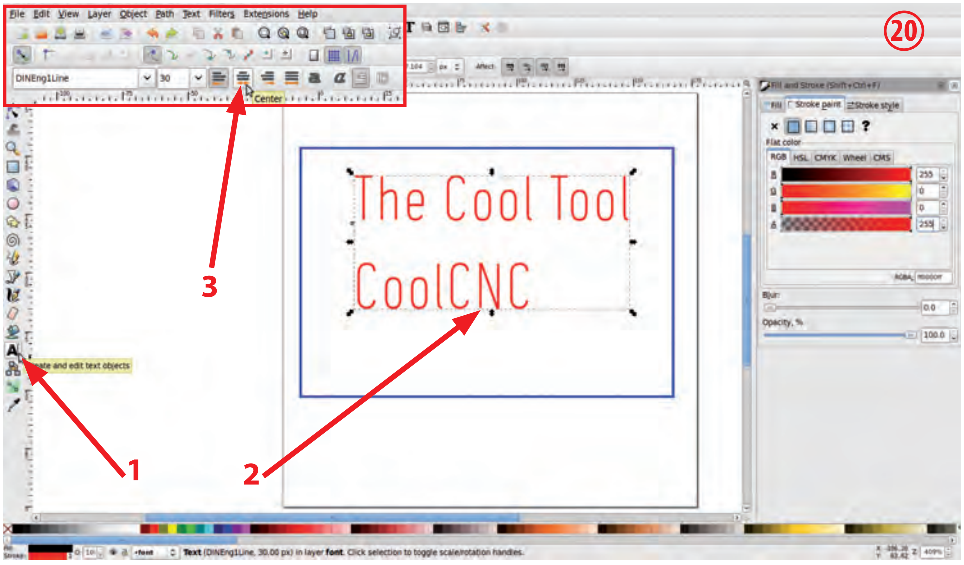
19) enter “255” then click on “blue” and change value to “0” and press “Enter”.

Attention:
The text field must be activated (marked).

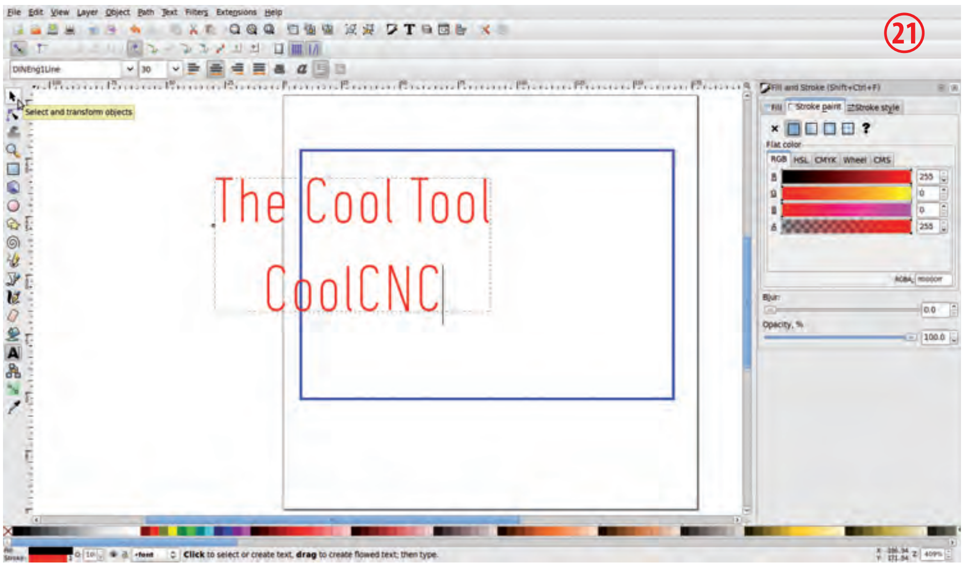


20) click on “A” icon “Create and edit text objects”, then click into the text field (The Cool Tool ....).

After that click on “Center”.



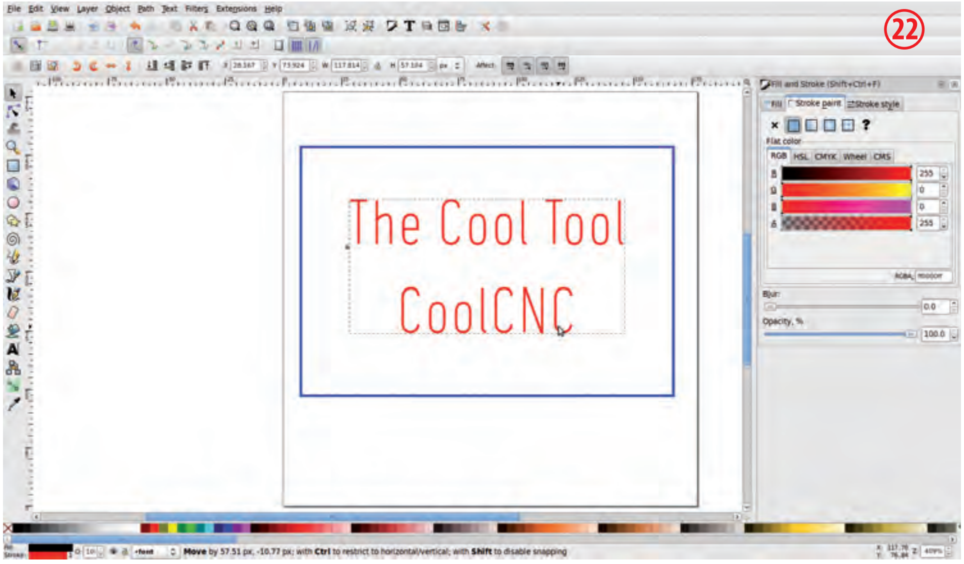
21) click on the arrow icon “Select and transform objects”.



https://inkscape.org

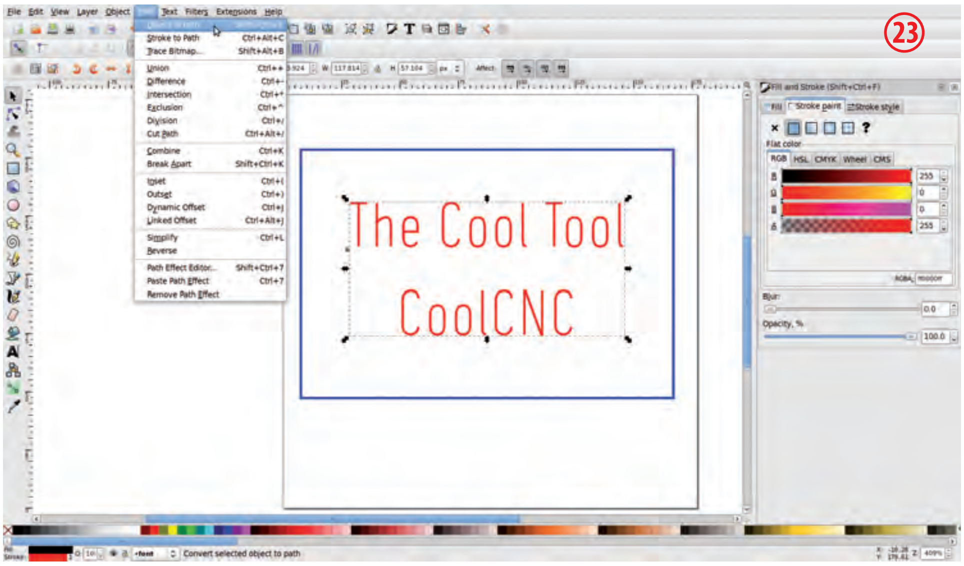
22) select the text field and place it in the middle of the blue rectangle.

Attention:
Please take care that the blue rectangle is at the center of the raw material. If it is not at the center, select the blue rectangle and the text field and place it again.



23) click on "Path" --> "Object to Path" (vector). Now the font is converted to a vector graphic.

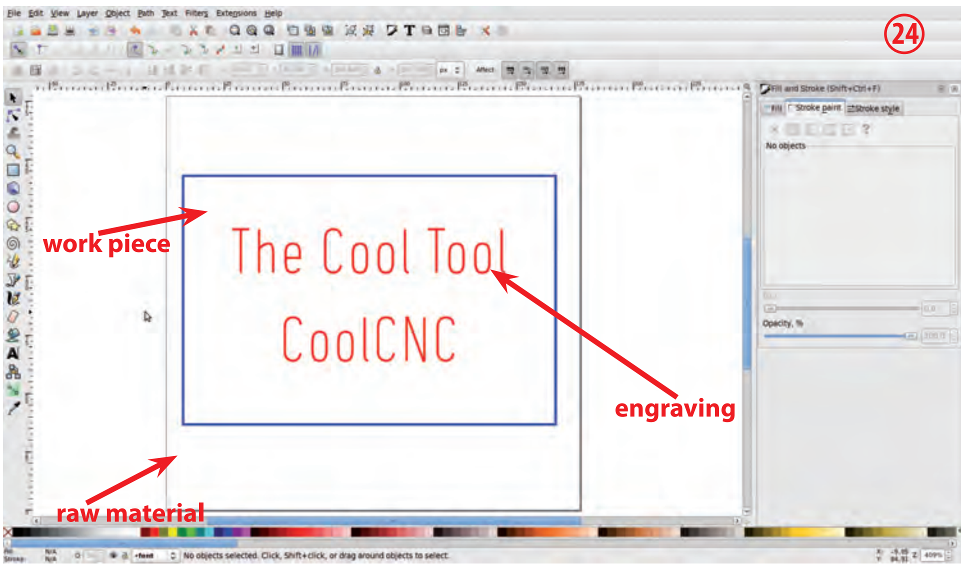
Attention:
The text field must be activated (marked).



24) Why two layers?
For each cutting depth you need a separate layer. Here we have two different depths:

1) red text: 1.0 mm (engraving)

2) blue rectangle: 4.5 mm
(raw material 4.0 mm - acrylic)



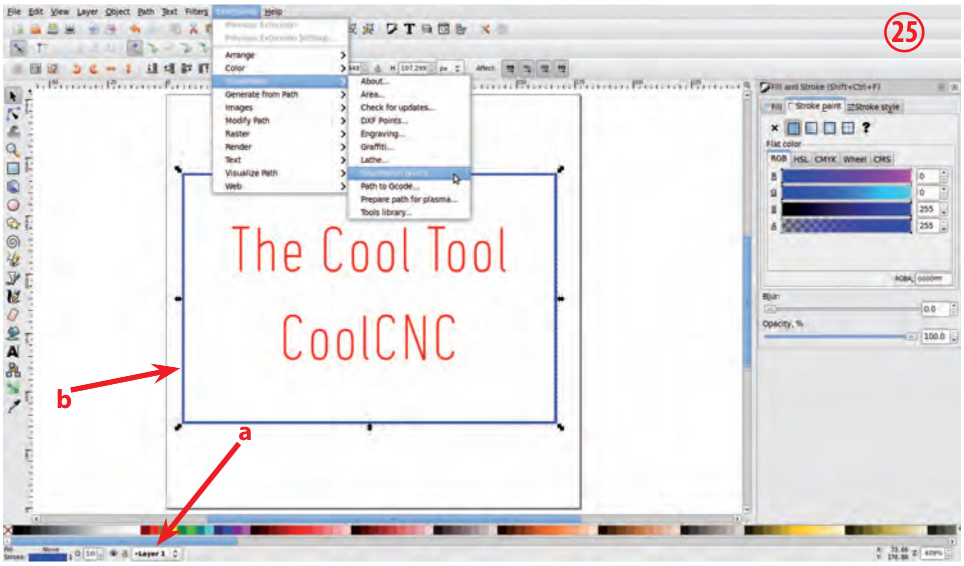
https://inkscape.org

Gcode Tools®

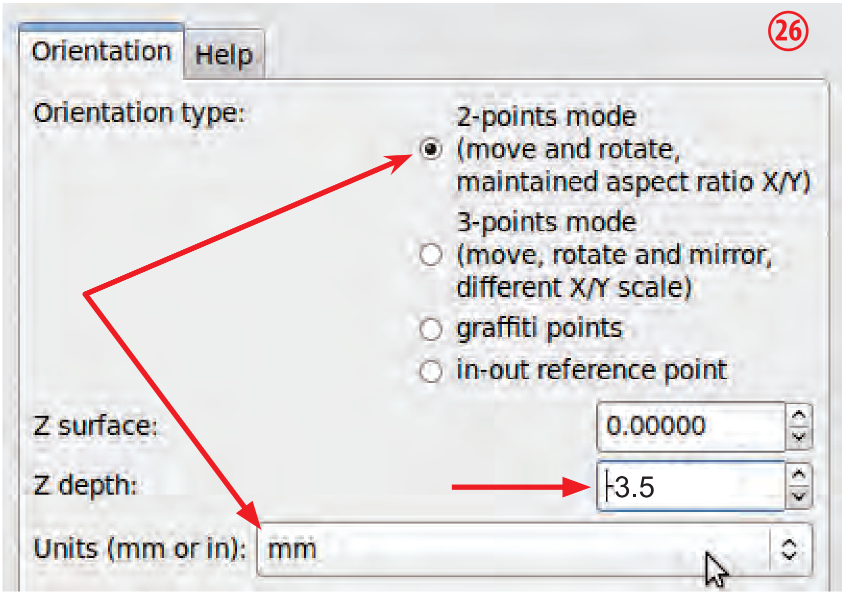
25) Orientation points
(zero point for the blue rectangle)

click on "Extensions" --> "Gcodetools" --> "Orientation points ..."

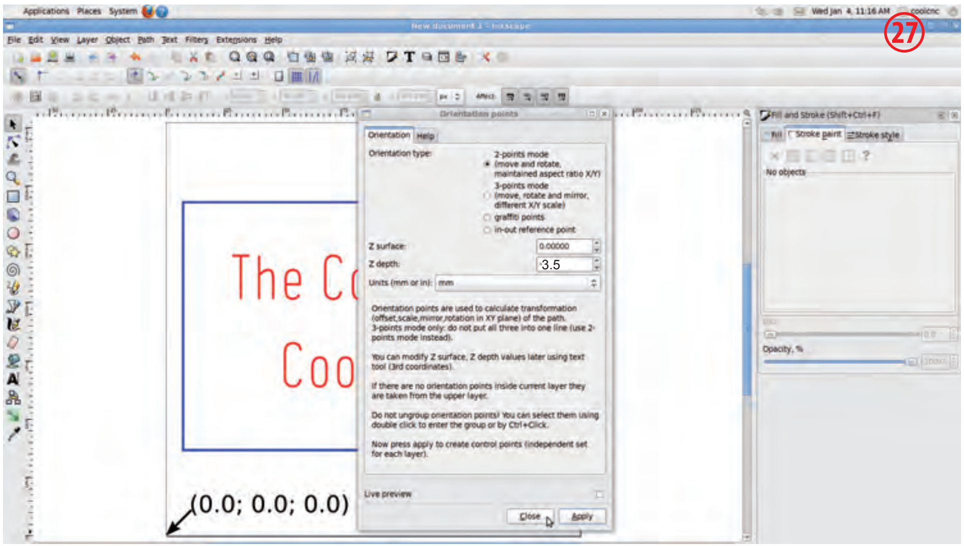
Attention:
a) layer "Layer 1" must be activated.
b) blue rectangle must be selected.



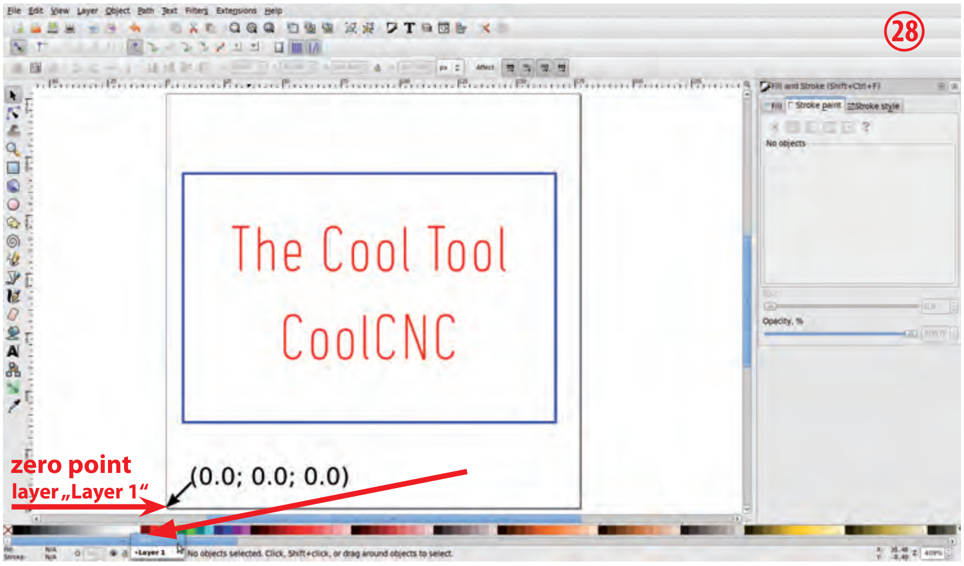
26) select "2-points mode" and "mm". enter at "z depth" "-3.5".
Attention:
"z surface" remains "0.0".



27) click on "Apply" and then "Close"



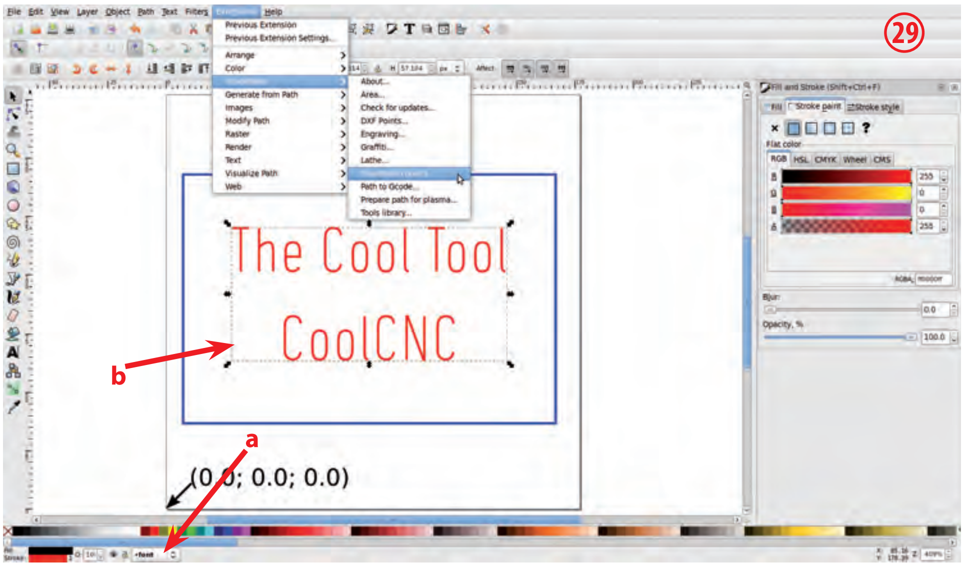
28) select the layer “font”



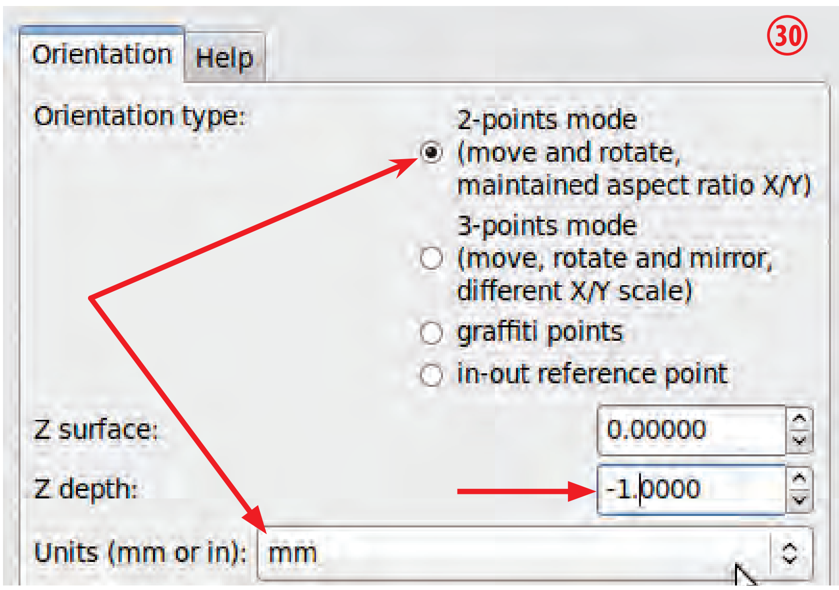
29) Orientation points
(zero point for the red text)

click on “Extensions” --> “Gco-detools” --> “Orientation points ...”

Attention:
a) layer “font” must be activated.
b) red text must be selected.

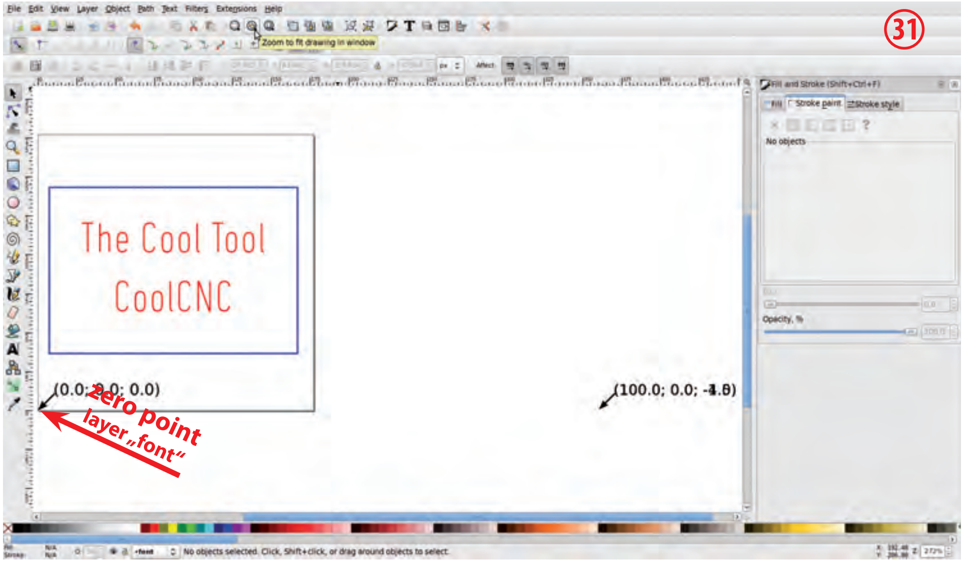


30) select “2-points mode” and “mm”. enter at “z depth” “-1.0”.
Attention:
“z surface” remains “0.0”.

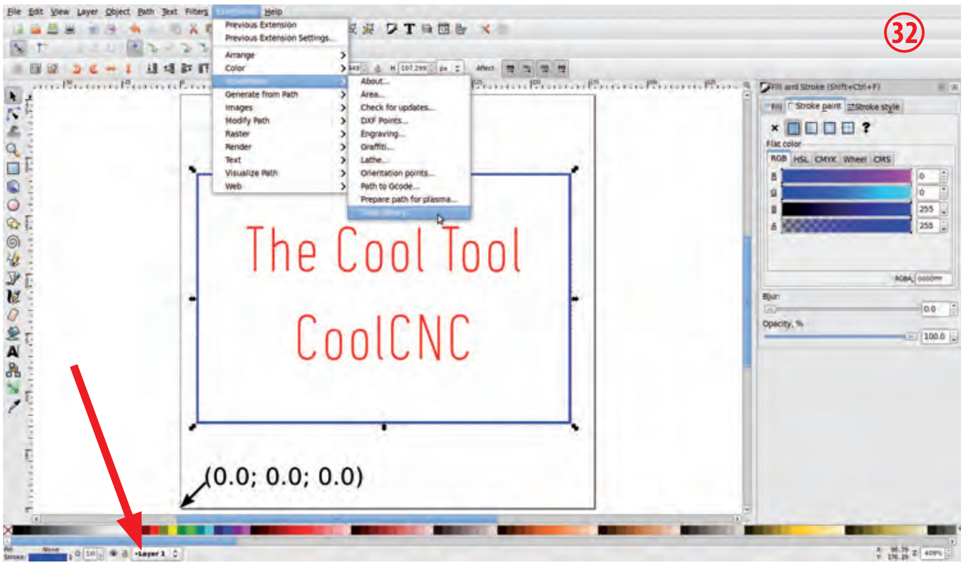


31) The position of the zero point for layer "Layer 1" and layer "font" is the same (congruent).

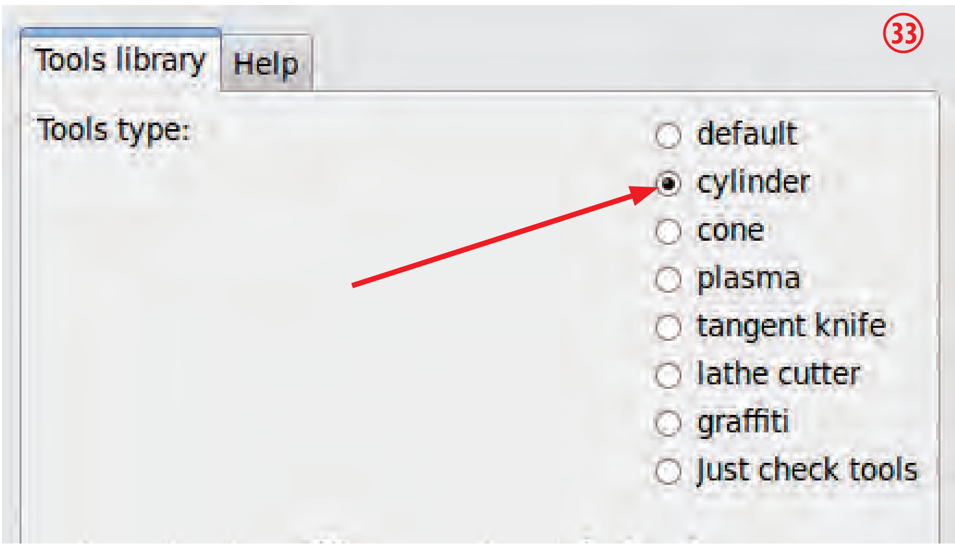
Attention:
Do not move this point!



32) Tools library
activate "Layer 1" and select the blue rectangle (marked), then click on "Extensions" --> "Gcodetools" --> "Tools library"

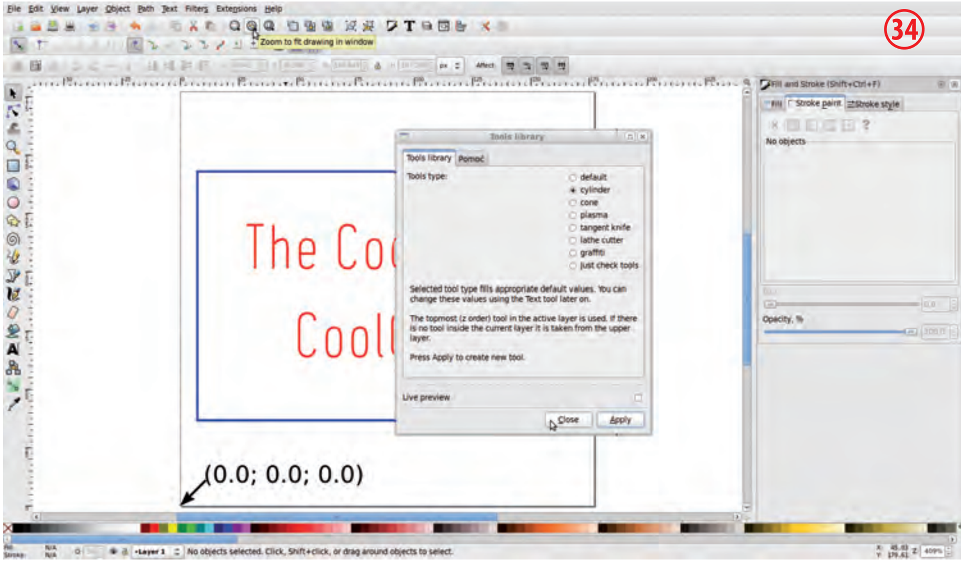


33) select "cylinder"

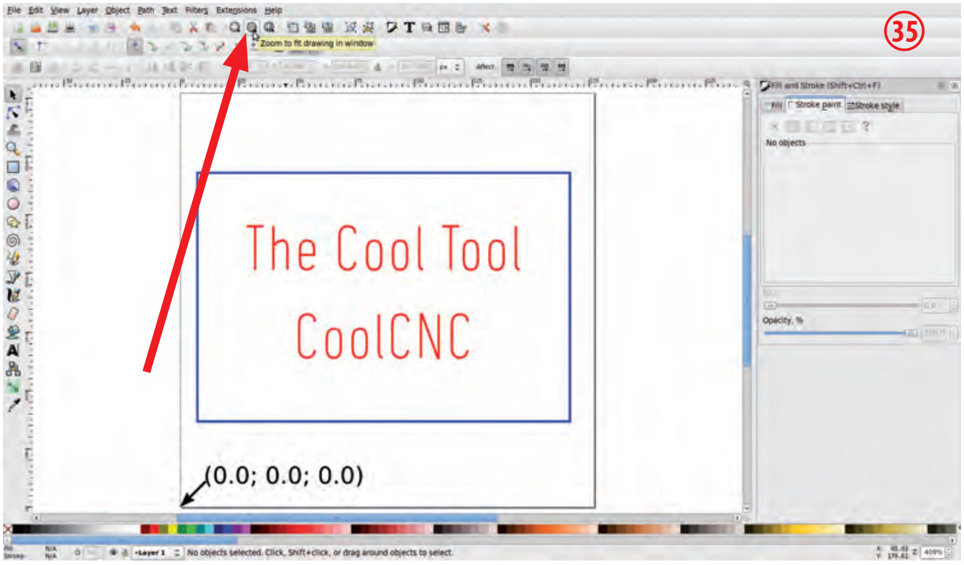


https://inkscape.org

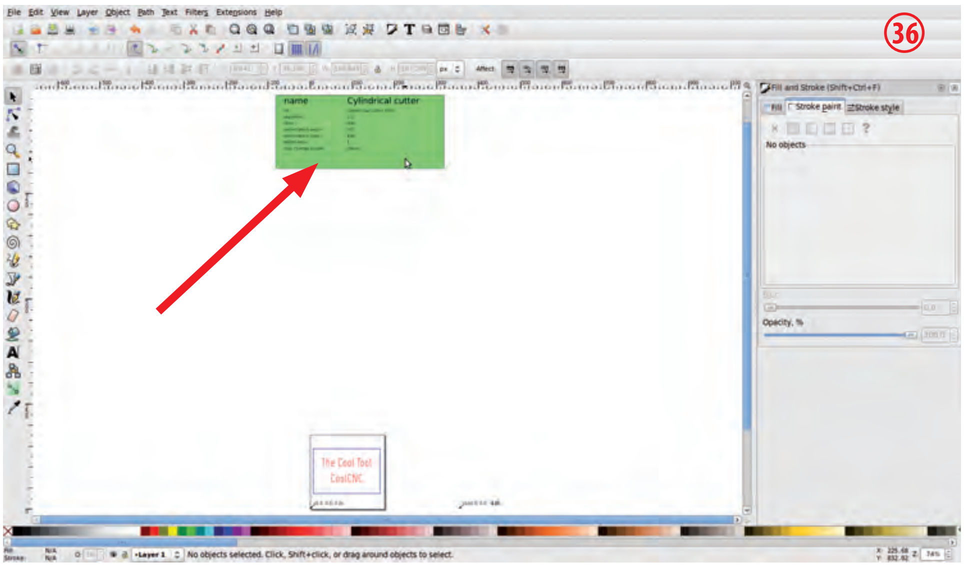
4) click on "Apply" then close the window - click on "Close"



2) click on the magnifier icon "Zoom to fit drawing in window"



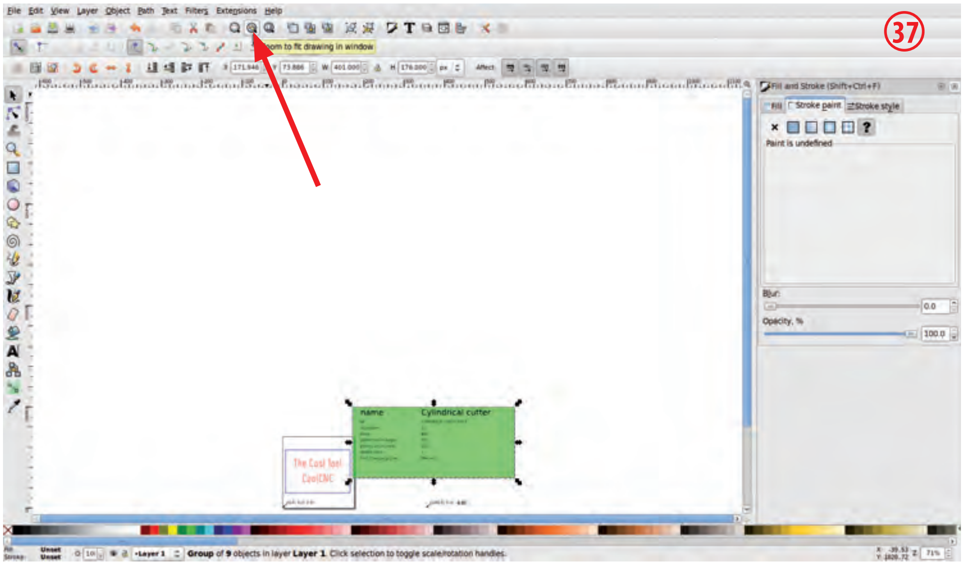
36) now you can see the "Tool window" for "Layer 1".



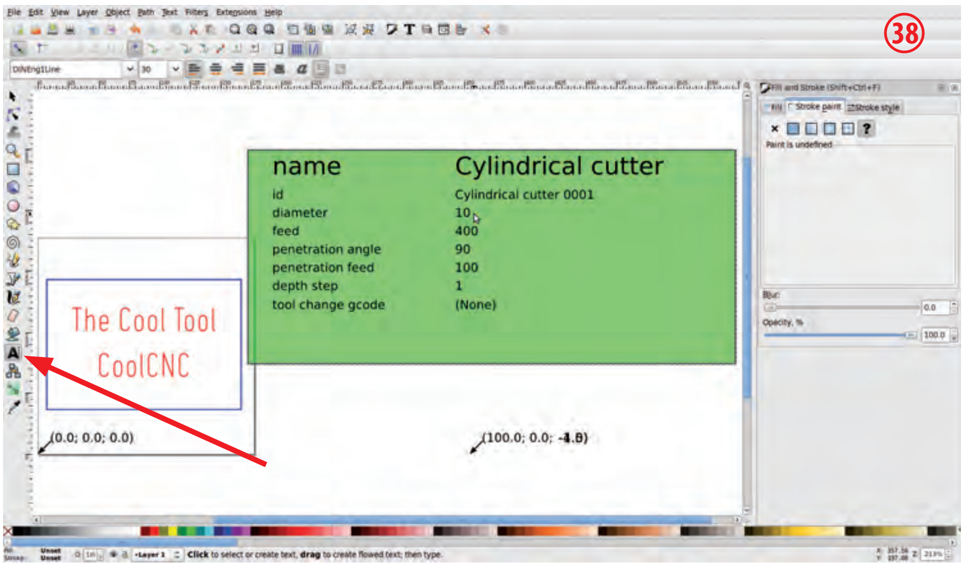
The cool tool Education

https://inkscape.org

37) place the "Tool window" near the blue rectangle, then click on the magnifier icon (Zoom to fit drawing in window).

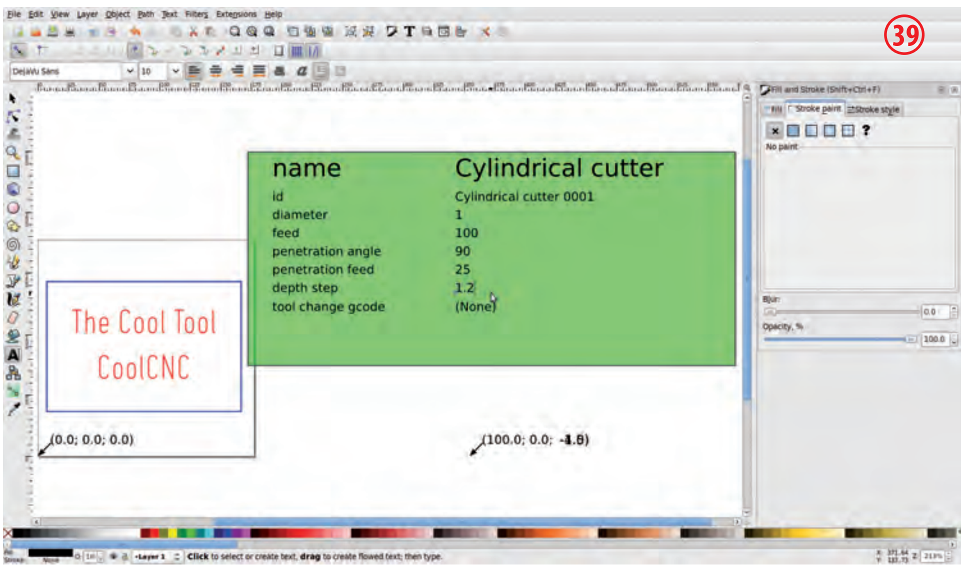


38) click on the "A" icon (Create and edit text objects)

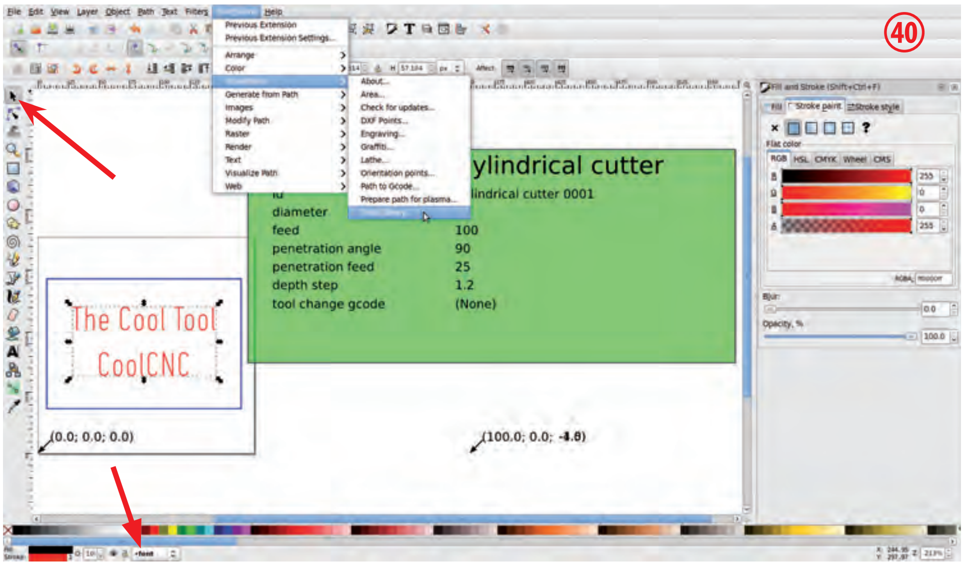


39) edit the following parameters in the "tool window:

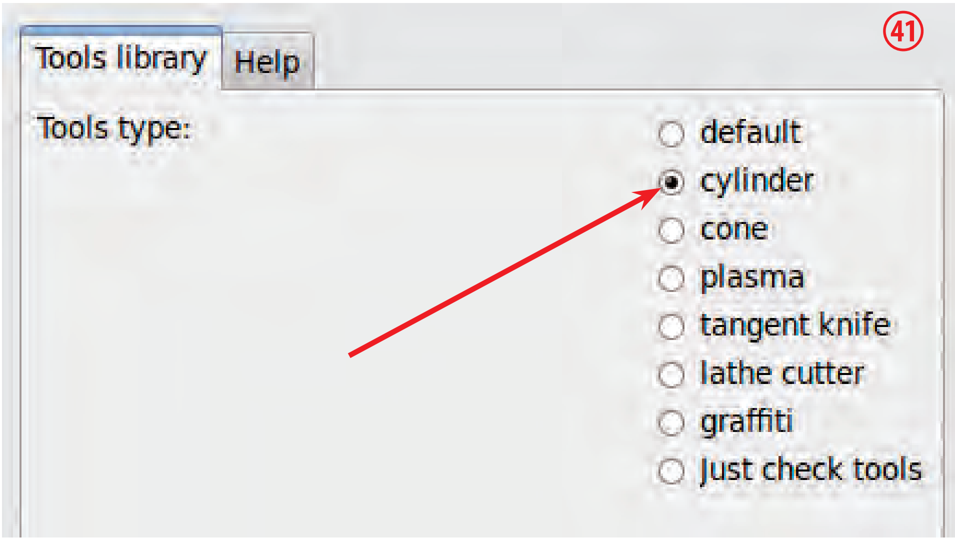
diameter = 1
feed = 100
penetration feed = 25
depth step = 1.2



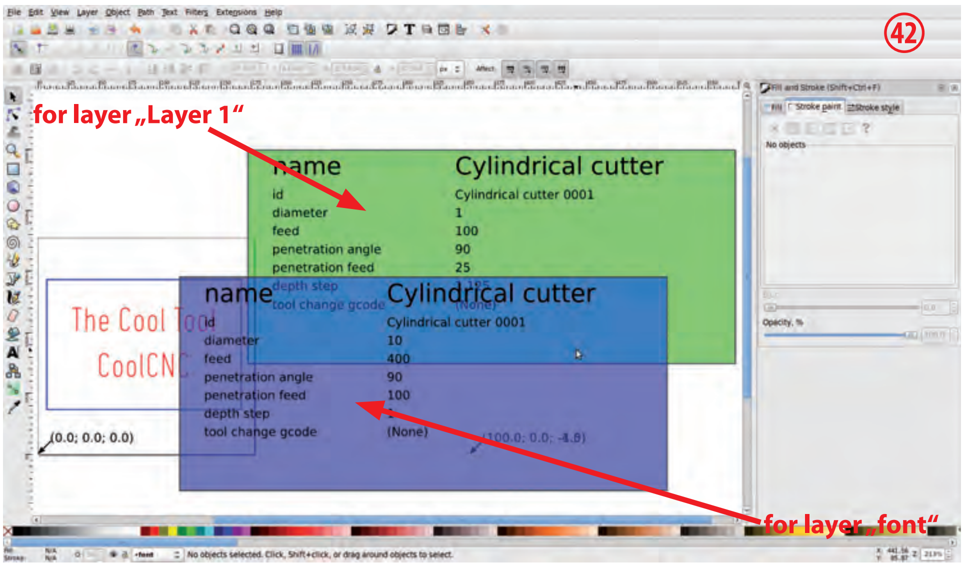
40) activate layer “font” and select the red text (marked), then click on “Extensions” --> “Gcodetools” --> “Tools library”



41) select “cylinder” and click on “Apply” then close the window - click on “Close”.

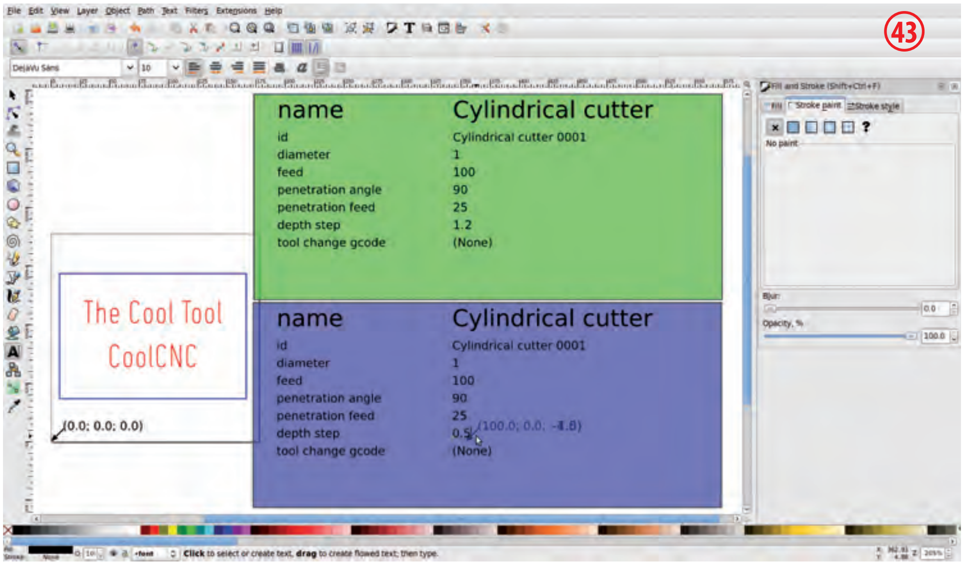


42) the new “Tool window” (window for layer “font”) is magenta.

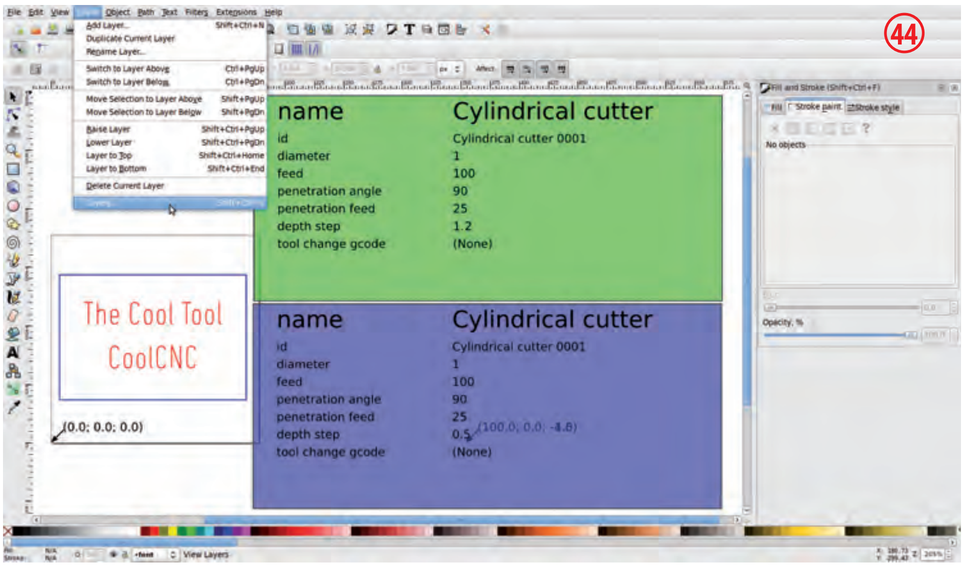


43) click on the "A" icon (Create and edit text objects) and edit following parameters:

diameter = 1
feed = 100
penetration feed = 25
depth step = 0.5

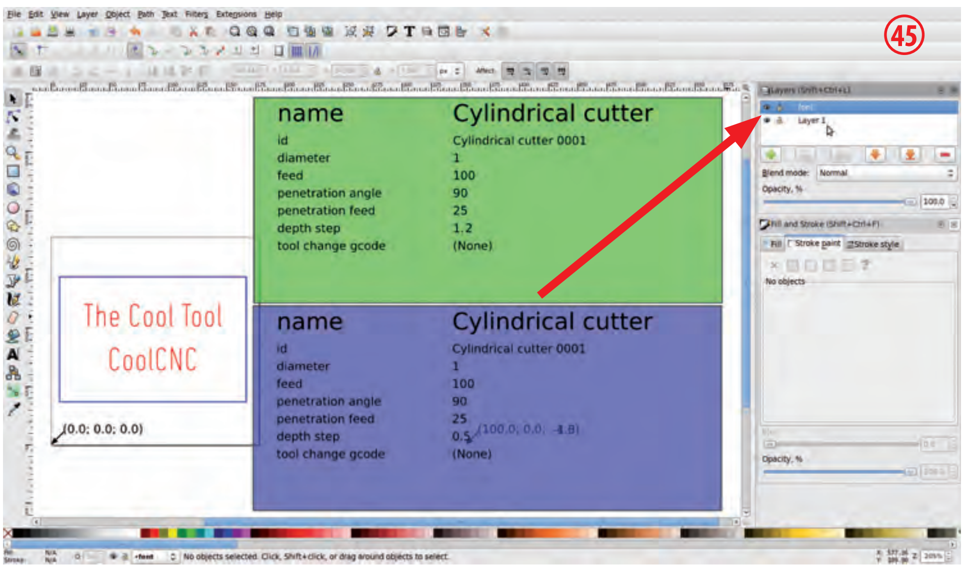


44) check the order of the layers.
click on "Layer" --> "Layers ...".



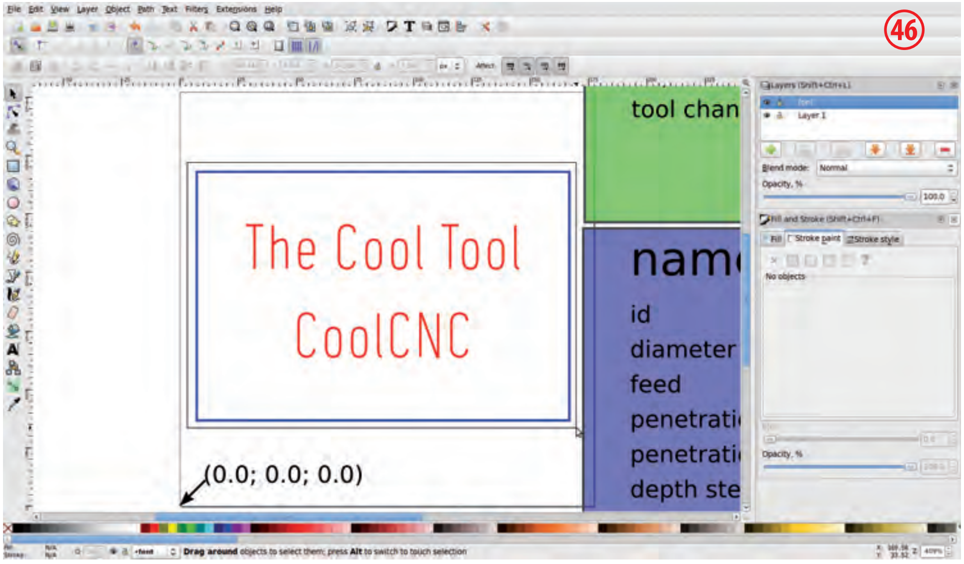
45) correct order: 1) font
2) Layer 1

If the order isn't correct, you can change the order. Select one of the two layers and then move it (with the arrows) up or down.

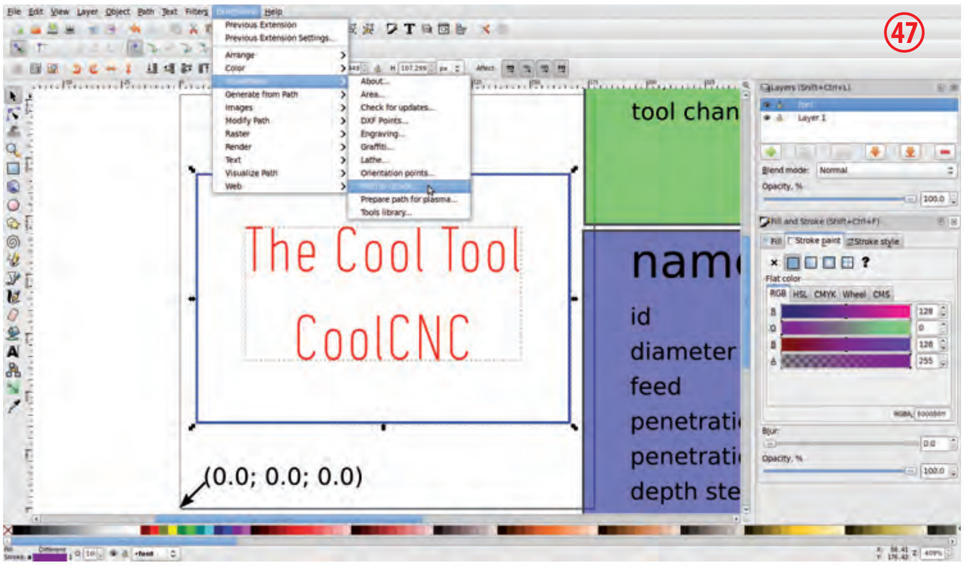


https://inkscape.org

46) click on the arrow icon "Select and transform objects", then select the engraving (red text) and the work piece (blue rectangle).



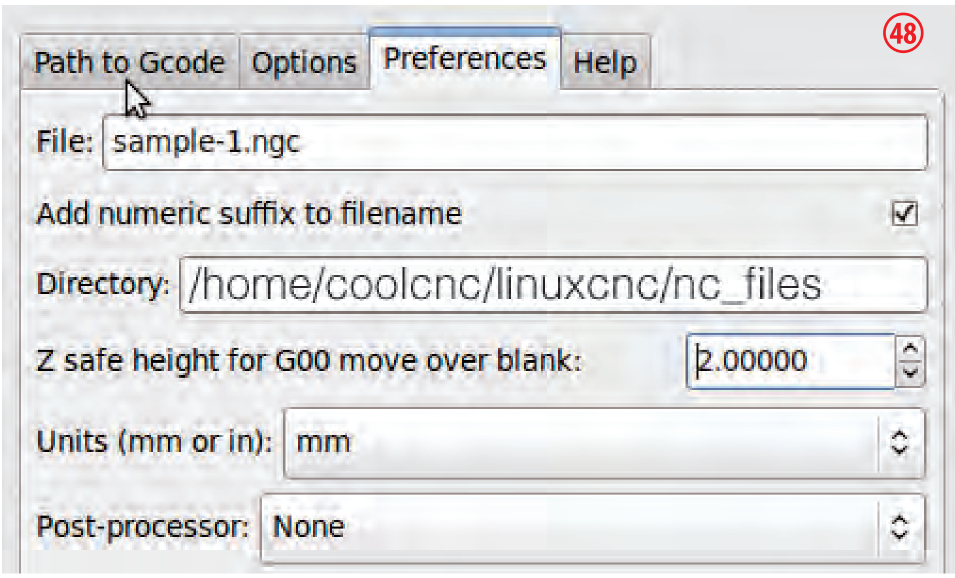
47) click on "Extensions" --> "Gcodetools" --> "Path to Gcode ...".



48) click on "Preferences" and enter following parameters:

File: **sample-1.ngc**
Add numeric..... **"activate"** ☑
Directory: **/home/coolcnc/linuxcnc/nc_files**
("coolcnc" = username)
Z safe height **2.00**
Units **mm**
Post-processor **None**

click on "Path to Gcode"!

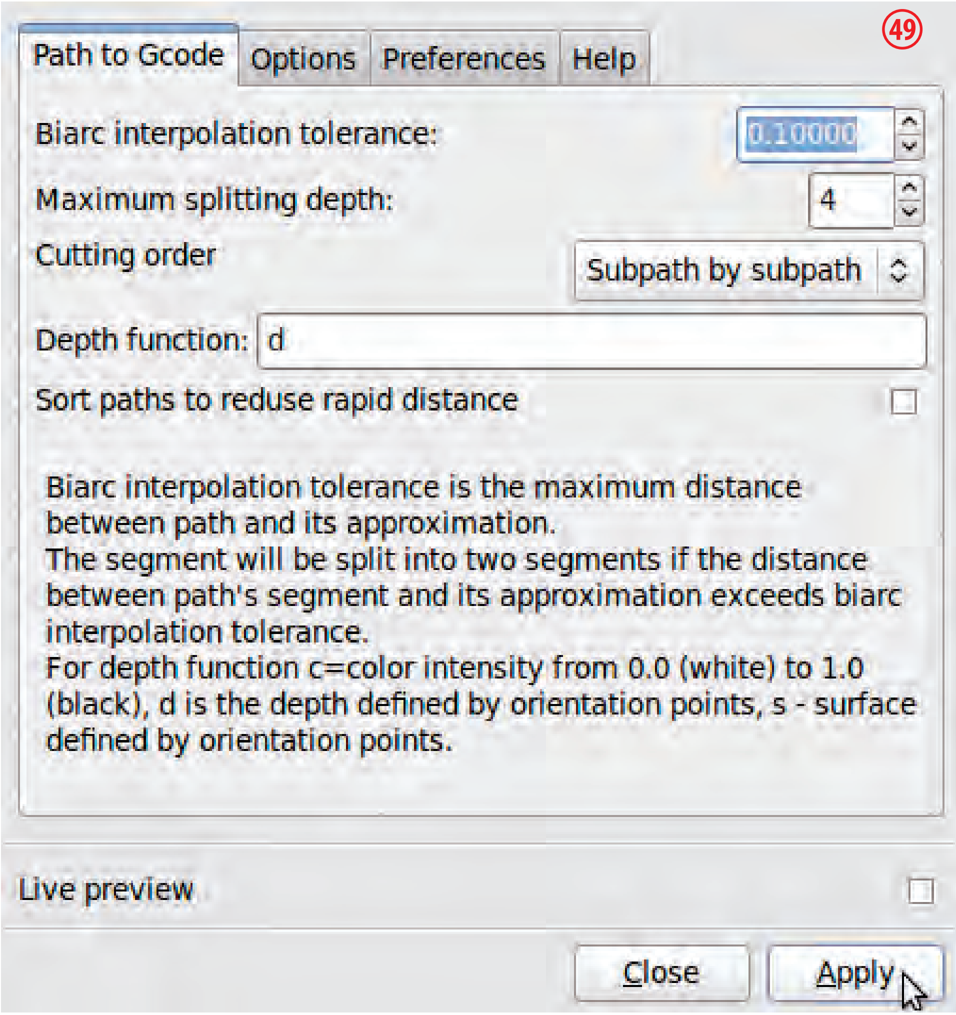


The cool tool Education

49) check the parameters:

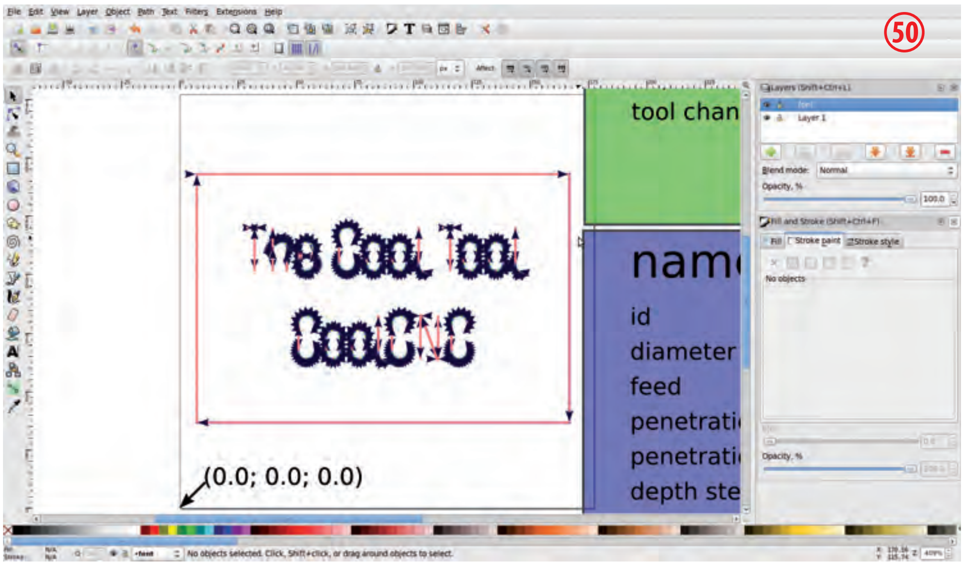
Biarc inter..... **0.100**
Maximum splitting **4**
Cutting order
subpath by subpath
Depth funktion: **d**

click on “Apply” and then close the window- click on “Close”!



50) finished!

Now you can save the Inkscape file (click on “File” --> “Save”) , after that you can close Inkscape®.



51) Open the generated G-Code file in CETUS (UNI-FRAES-V3).

The tool tool Education

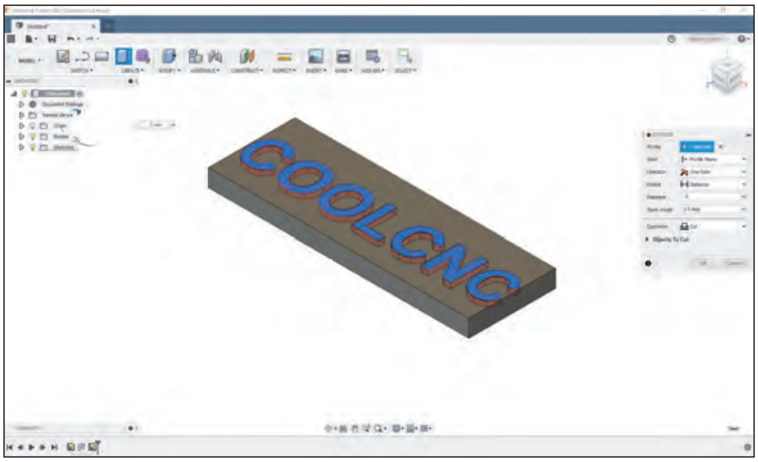
Working with Autodesk® Fusion 360®

https://www.autodesk.com/products/fusion-360/students-teachers-educators

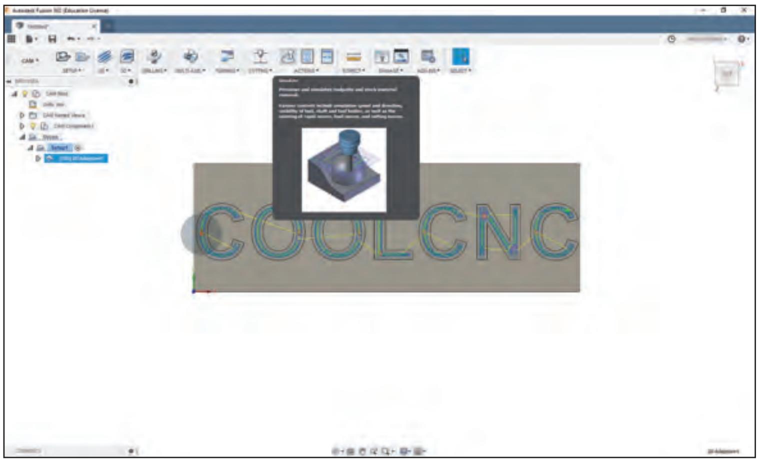
Fusion 360 is a professional 3D CAD/CAM software for which AUTODESK offers free licenses for education.

Advanced designs, 2.5D drawings or 3D models can be constructed, the CAM functionality allows for generating G-code files.

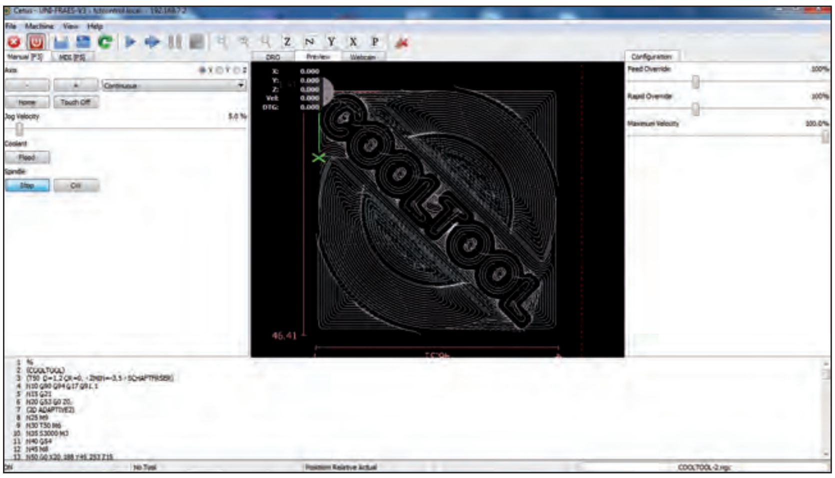
Suitable post-processors and tool-tables make it compatible to the Machinekit CNC software.



Fusion 360 - CAD
(design process)



Fusion 360 - CAM
(set machining parameters)



Cetus - machinekit
(loaded G-Code)

Autodesk® Inventor® can also be used.

The cool tool est. 1982 Education