DrufelCNC

G-code manual

DrufelCNC, 2020

Content

Annotation 4

G00 - Accelerated movement 5

G01 - Linear interpolation 5

G02 - Circular interpolation 9

G03 - Circular interpolation 9

G04 - Pause 15

G10 - Enable data entry mode 15

G17 - XY plane selection 17

G18 - XZ plane selection 17

G19 -YZ plane selection 17

G20 - Input inch data 21

G21 - Enter metric data 21

G31 - Movement of axes to a given position 21

G38.n - Straight probe 23

G50 - Scale mode 23

G51 - Scale mode 23

G53 - Disabling the offset of the origin of the coordinate system 25

G68 - Rotate coordinate system 25

G69 - Cancel Rotation 26

G80 - Cancel Canned Cycle 27

G81 - Simple Drilling Cycle 27

G82 - Drilling Cycle with Dwell 28

G90 - Absolute positioning mode. 29

G90.1 - Absolute distance mode for I , J and K offsets. 29

G91 - Relative positioning mode. 30

G91.1 - Incremental distance mode for I, J and K offsets 30

G98 - Canned Cycle Return or Feed Rate Modes 31

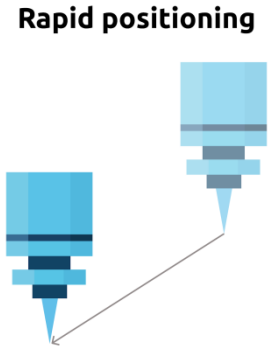
G99 - Canned Cycle Return or Feed Rate Modes 31

Annotation

This document is a G code user guide. These G-codes can be used in the DrufelCNC software. G-code is a programming language for numerically controlled machines and machine tools.

G00 - Accelerated movement.

Code G00 is used for faster movement. Accelerated movement is necessary for quick movement.
The speed of the tool is too high and inconsistent!
The use of the G00 code reduces the overall processing time.



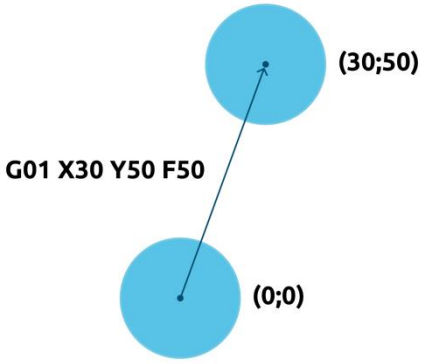
G01 - Linear interpolation

Code G01 is designed to perform tool movement in a straight line at a given speed. The main difference between the G01 code and the G00 is that with linear interpolation, the tool moves at a given speed. Movement speed is indicated by the F command. Conventionally, a frame for linear interpolation is written as follows:

G01 Xn.n Yn.n Zn.n F n.n
G01 Xn.n Yn.n Zn.n An.n Bn.n Cn.n Fn.n

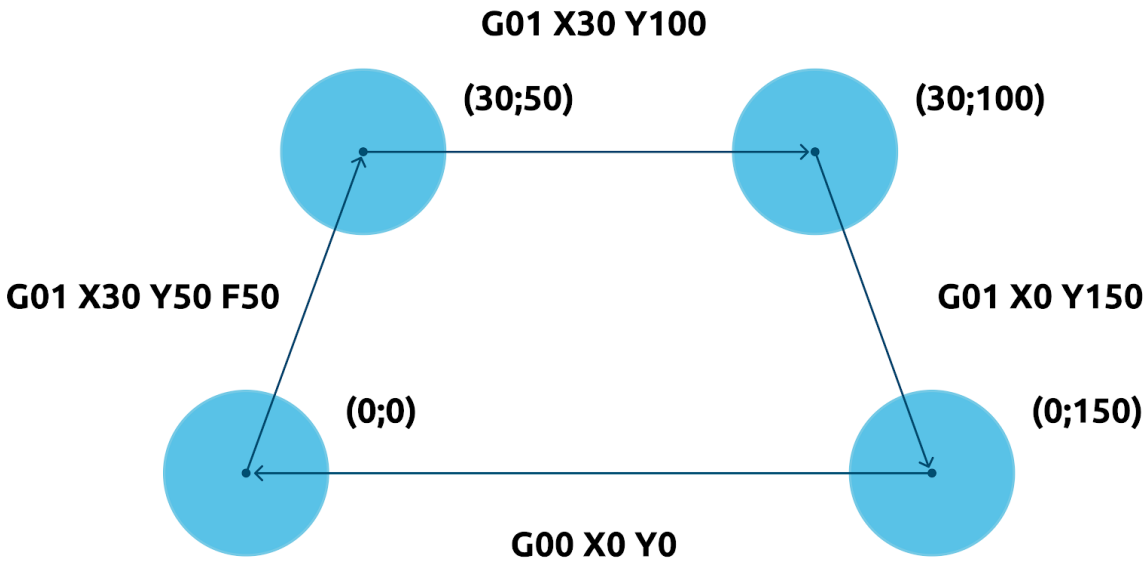
Consider the following simple program:

X0Y0 - zero position
G00X0Y0Z-5 - lowering the cutter along the z axis by -5mm
G01X30Y50F50 - moving the tool to a point (30; 50) with a working feed speed of 50 mm / min
G00X30Y50Z10 - raising the cutter along the z axis by 10mm



Let's complicate the program a bit. Let's write a program cutting a polygon.

G00Z10 ;raise the cutter to a safe position from the workpiece
G00X0Y0 ;zero position
G01X0Y0Z-5F50 ;lowering the cutter along the axis on z by -5 with a working feed speed of 50 mm / min
G01X30Y50F50 ; moving the tool to a point (30; 50) with a working feed speed of 50 mm / min
G01X30Y100 ;moving the tool to a point (30; 100)
G01X0Y150 ;moving the tool to a point (0; 150)
G00X0Y150Z10 ;raising the cutter along the axis by z by 10
G00X0Y0 ;move to zero position



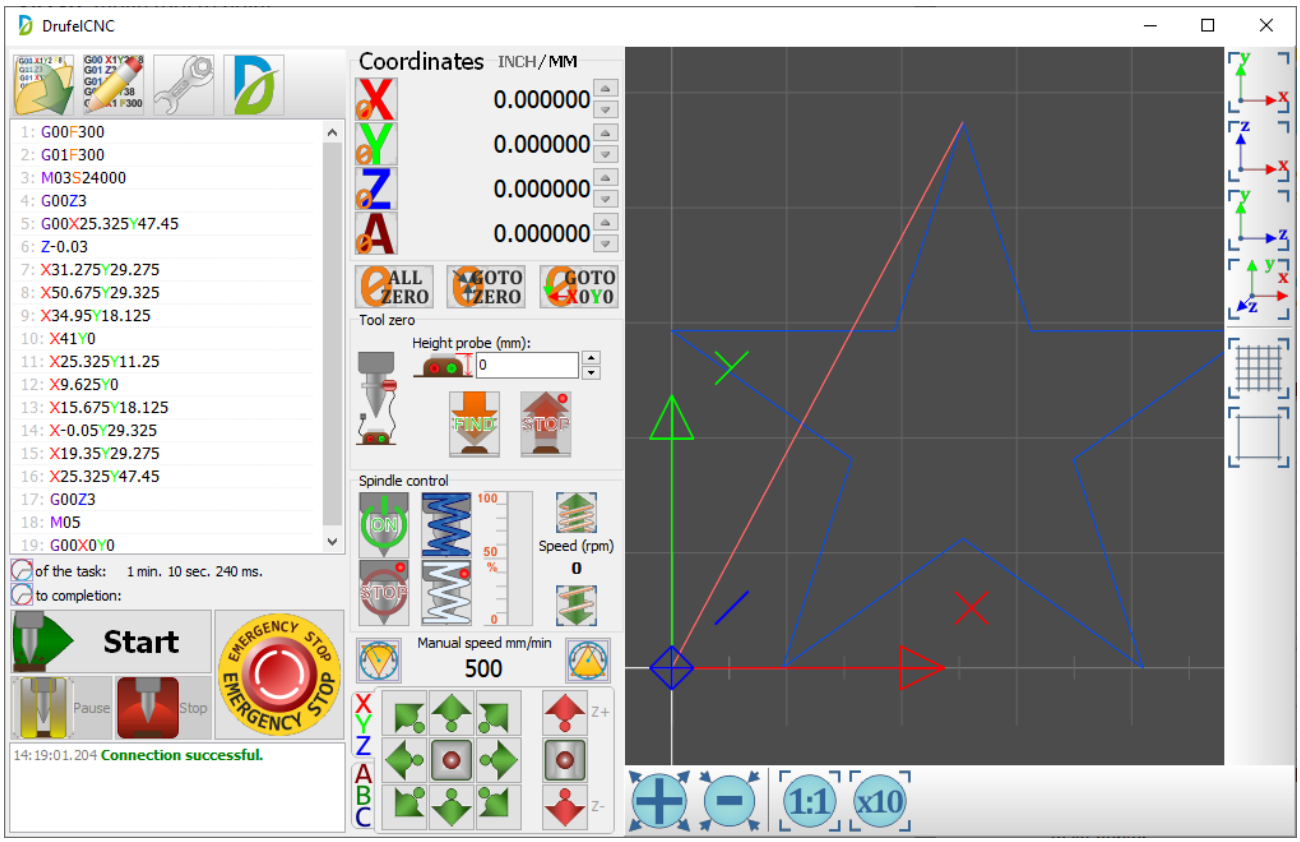
The speed value can be set once. You can also shorten your code. Short code will look like this:

G00Z10
G00X0Y0
G01Z-5F50

G01X30Y50
G01Y100
G01X0Y150
G00Z10
G00X0Y0

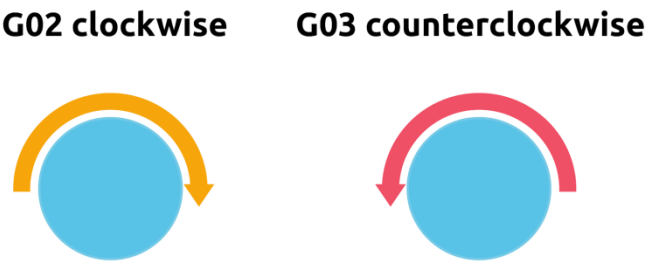
Let's try to write a program for a more complex figure:

G00F300 ;for G00 commands we set the speed to 300
G01F300 ;for G01 commands we set the speed to 300
M03S24000 ;turn on the spindle at a speed of 24000 rpm
G00Z3 ;raise the cutter to a safe position
G00X25.325Y47.45 ;moving the tool to a point (25.325; 47.45)
Z-0.03 ;axis milling cutter axis z -0.03
X31.275Y29.275 ;move tool to point
X50.675Y29.325 ;moving the tool to a point
X34.95Y18.125 ;move the tool to a point
X41Y0 ;move tool to point
X25.325Y11.25 ;move tool to point
X9.625Y0 ;move tool to point
X15.675Y18.125 ;move tool to point
X-0.05Y29.325 ;move the tool to a point
X19.35Y29.275 ;moving the tool to a point
X25.325Y47.45 ;move the tool to a point
G00Z3 ;raising the cutter along the axis by z by 3
M05 ;spindle shutdown
G00X0Y0 ;move to zero position


DrufelCNC
Coordinates
INCH / MM
0.000000
0.000000
0.000000
0.000000
ALL ZERO
GOTO Z ZERO
GOTO X0Y0
Tool zero
Height probe (mm):
FIND
STOP
Spindle control
ON
STOP
Speed (rpm)
Manual speed mm/min
500
Start
Pause
Stop
EMERGENCY STOP
1:1
x10
14:19:01.204 Connection successful.

G02, G03 - Circular interpolation

Codes G02 and G03 are designed to perform circular interpolation. G02 command is used to move in an arc clockwise, and G03 - counterclockwise.



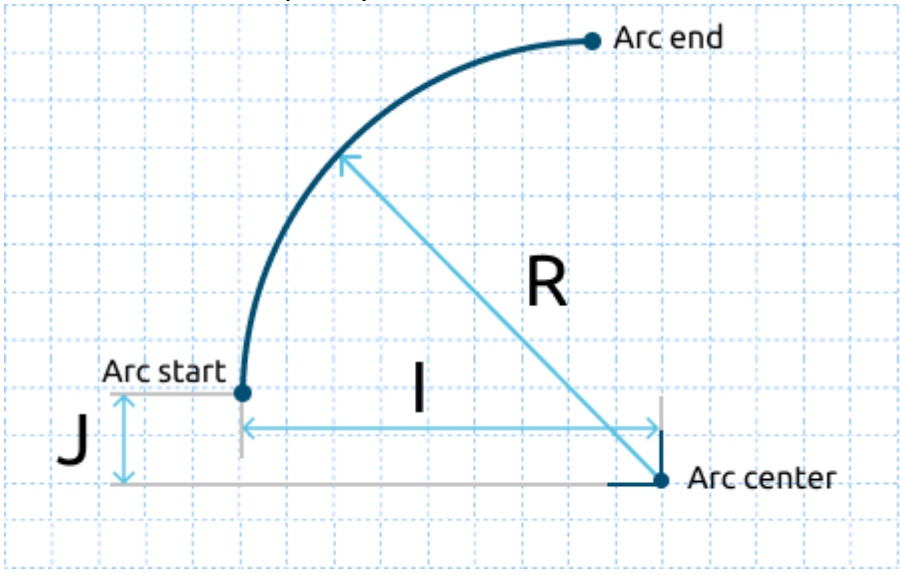
The direction of movement is determined when we look at the tool from the spindle side, in the negative direction of the Z axis. As with linear interpolation, in the circular interpolation block, you must specify the feedrate F.

There are two ways to form a circular interpolation frame:

- setting the center of the circle using I, J, K;
- setting the radius of the circle with R.

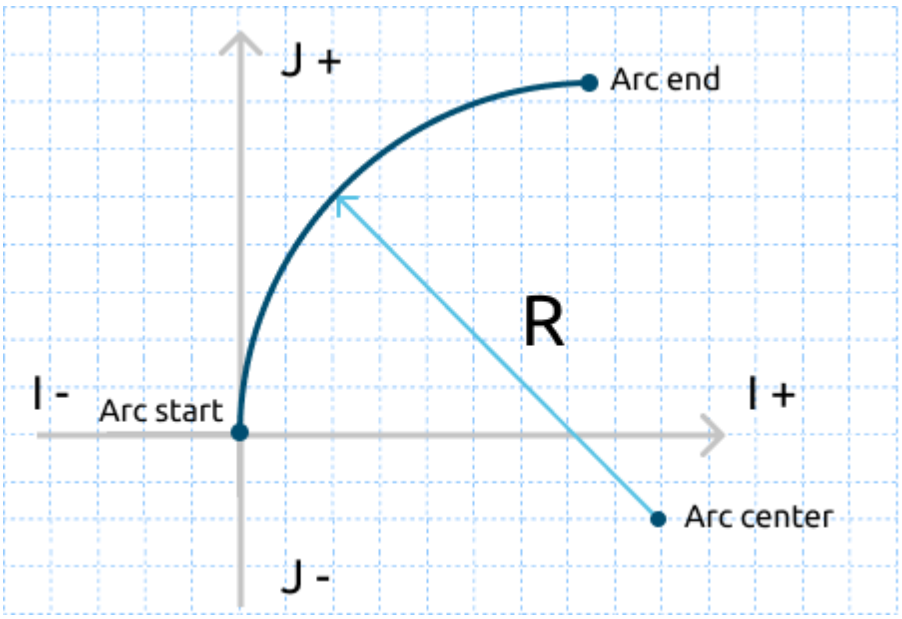
Arc with I, J, K

For a complete description of the arc, it is not enough to specify only the coordinates of its end point. You must also specify the coordinates of the center.

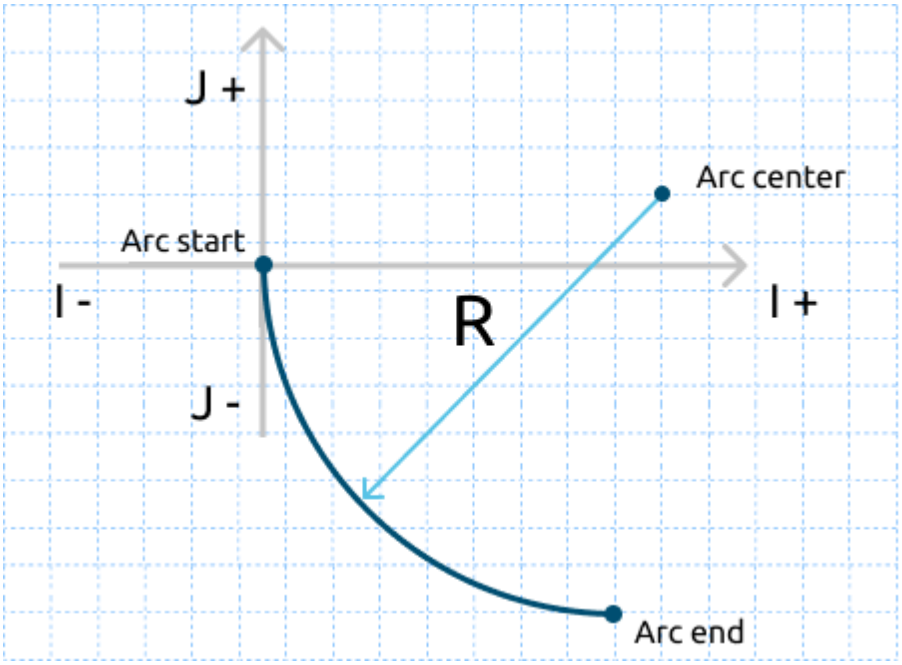


I, J, K are used to determine the center of the arc

Using I, J, and K, you specify the relative (incremental) distances from the starting point of the arc to its center.



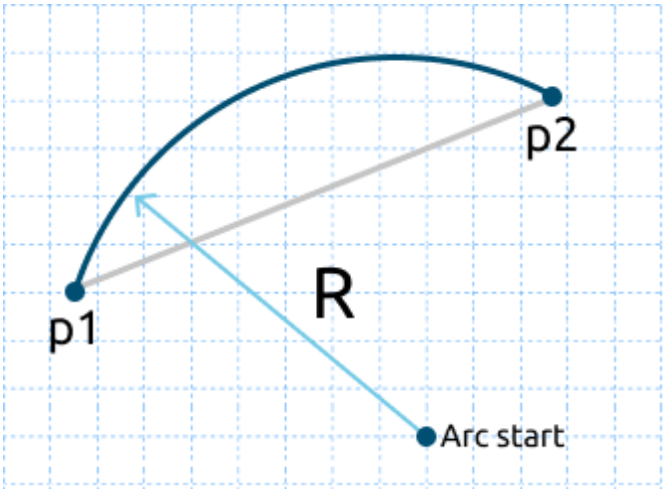
You must specify a positive value for I and a negative value for J



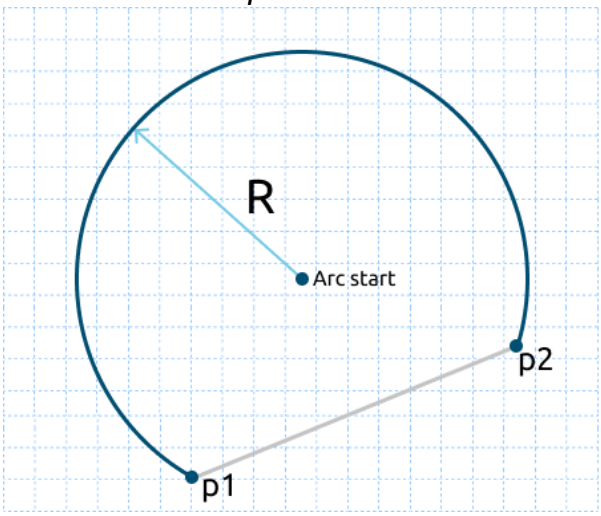
You must specify a positive value for I and a positive value for J

Arc with R

A simpler way to specify the center of the arc is based on applying the address R (radius). To unambiguously determine the shape of the arc, you must specify the corresponding sign in front of the numerical value of the radius R. For an arc that is greater than 180 °, the value of R will be negative. For an arc that is less than 180 °, the R value will be positive.



Since the arc is less than 180 ° (its center is located outside the chord), then R will have a positive value



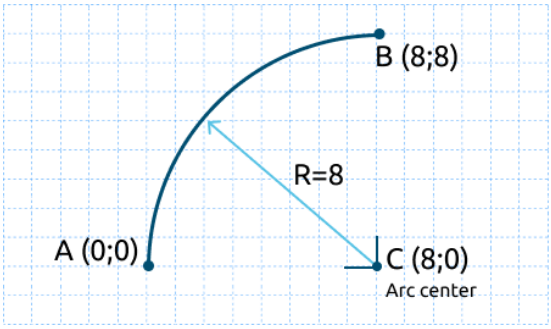
Since the arc is greater than 180 ° (its center is located inside the chord), then R will have a negative value

Using G02 and G03

Let's see how circular interpolation works, using an example. The following fragment of the control program moves the tool along an arc with a radius of 8 mm from point A (0; 0) to point B (8; 8) with a working feed speed of 80 mm / min.

G01X0Y0
G02 X8.0 Y8.0 I8.0 J0.0 F80

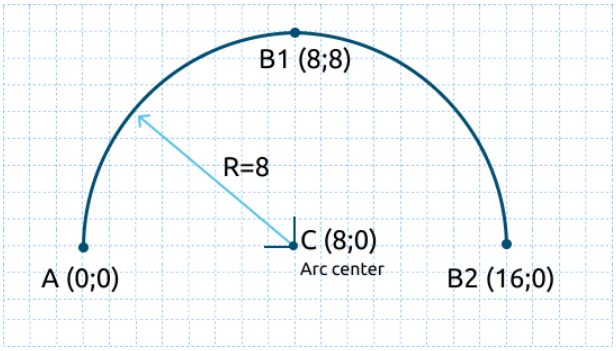
Since the center of the arc is at a distance of 8 mm along the X axis and 0 mm along the Y axis relative to the starting point A, then I will be 8.0, and J is 0. The resulting arc is only a quarter of the full circle. Let's try to describe the whole circle gradually.



Moving along an arc with R = 8 from point A (0; 0) to point B (8; 8)

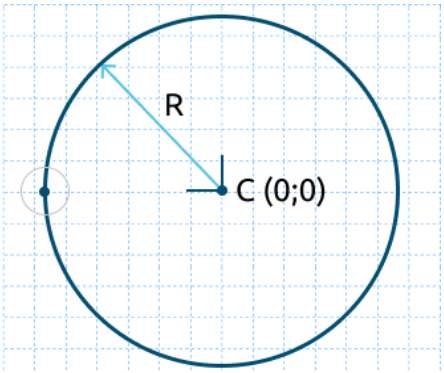
The next frame moves the tool from point B (B1) to point B2. Since the feedrate does not change, there is no need to re-specify the data F-word.
Since the center of the arc is at a distance of 0 mm along the X axis and 8 mm along the Y axis relative to point B, then I will be 0, and J will be -8. Thus, we were able to create a displacement along an arc from point A to point B2 using two frames.
Currently, most CNC systems allow you to perform an operation to describe the full circle in two or even one frame. Therefore, the movement from point A to point C can be written as follows:

G01X0Y0
G02 X8.0 Y8.0 I8.0 J0.0 F80
G02 X16.0 Y0.0 I-8.0 J0.0



Modern CNC systems allow the description of such an arc in one block

G01X-8Y0
G02 X-8.0 Y0.0 I8.0 J0.0



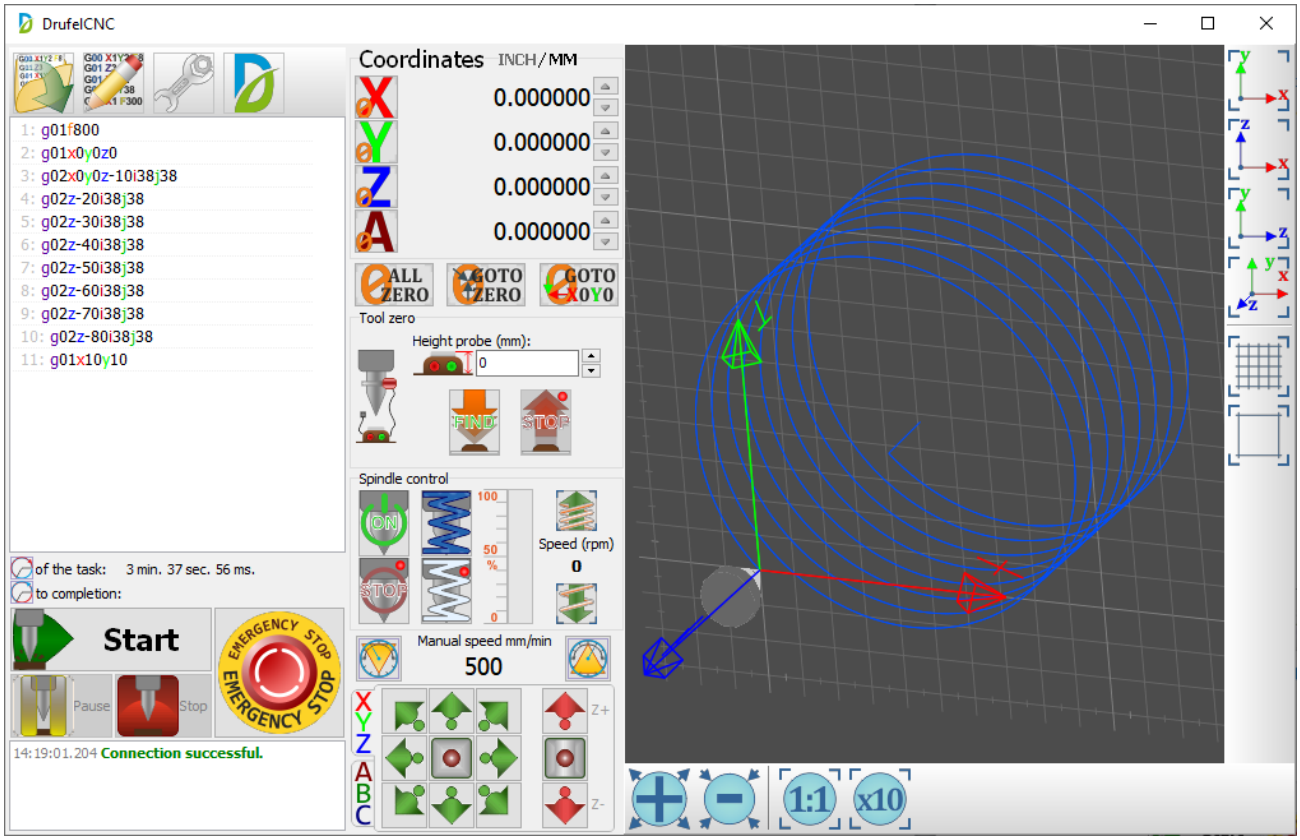
Full circle description in one frame is also possible.

Spiral.

If the XY plane (G17) is activated and the Z word is programmed in the circular interpolation block, then a spiral forms in the XY plane. The direction of the arc or spiral in the XY plane can be determined visually.

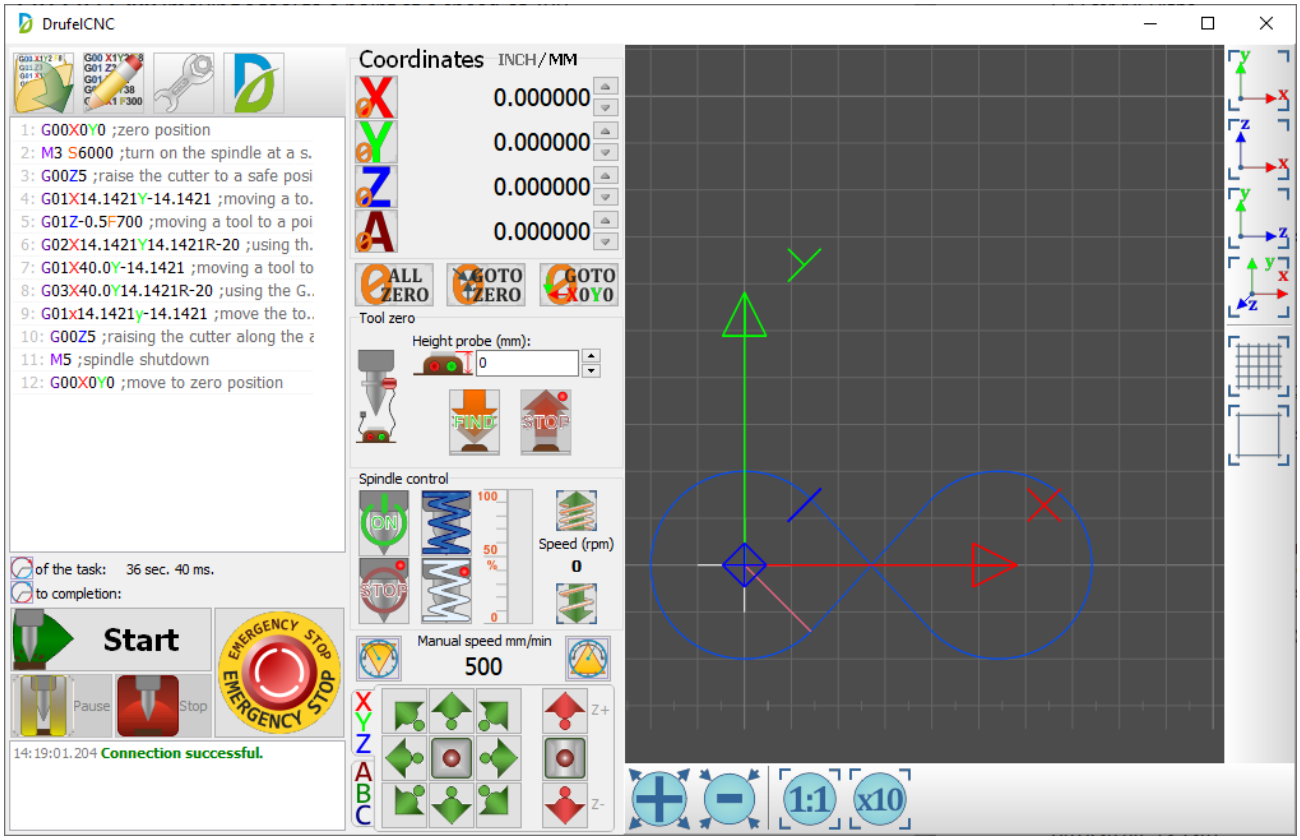
An example of a spiral:

G01F800
G01X0Y0Z0
G02X0Y0Z-10I38J38
G02Z-20I38J38
G02Z-30I38J38
G02Z-40I38J38
G02Z-50I38J38
G02Z-60I38J38
G02Z-70I38J38
G02Z-80I38J38
G01X10Y10



An example of a finished program:

G00X0Y0 ;zero position
M3 S6000 ;turn on the spindle at a speed of 6000 rpm
G00Z5 ;raise the cutter to a safe position
G01X14.1421Y-14.1421 ;moving a tool to a point
G01Z-0.5F700 ;moving a tool to a point at a speed of 700
G02X14.1421Y14.1421R-20 ;using the G02 command using R
G01X40.0Y-14.1421 ;moving a tool to a point
G03X40.0Y14.1421R-20 ;using the G02 command using R
G01x14.1421y-14.1421 ;move the tool to a point
G00Z5 ;raising the cutter along the axis by z by 3
M5 ;spindle shutdown
G00X0Y0 ;move to zero position



G04 - Pause

Command G04 is used to delay control program execution or pause. This modeless code is used in conjunction with the S- or P-address, which indicates the length of the exposure time. Usually this time is from 0.001 to 99999.999 seconds.

Code G04, S- or P-address are programmed together in a single block that does not contain any movements.

If P is used to determine the holding time, then the decimal point cannot be programmed. Address P determines the holding time in milliseconds, and S in seconds.

Example:
G04 S1.5 ;shutter speed 1.5 seconds;
G04 P200 ;shutter speed 200 milliseconds.

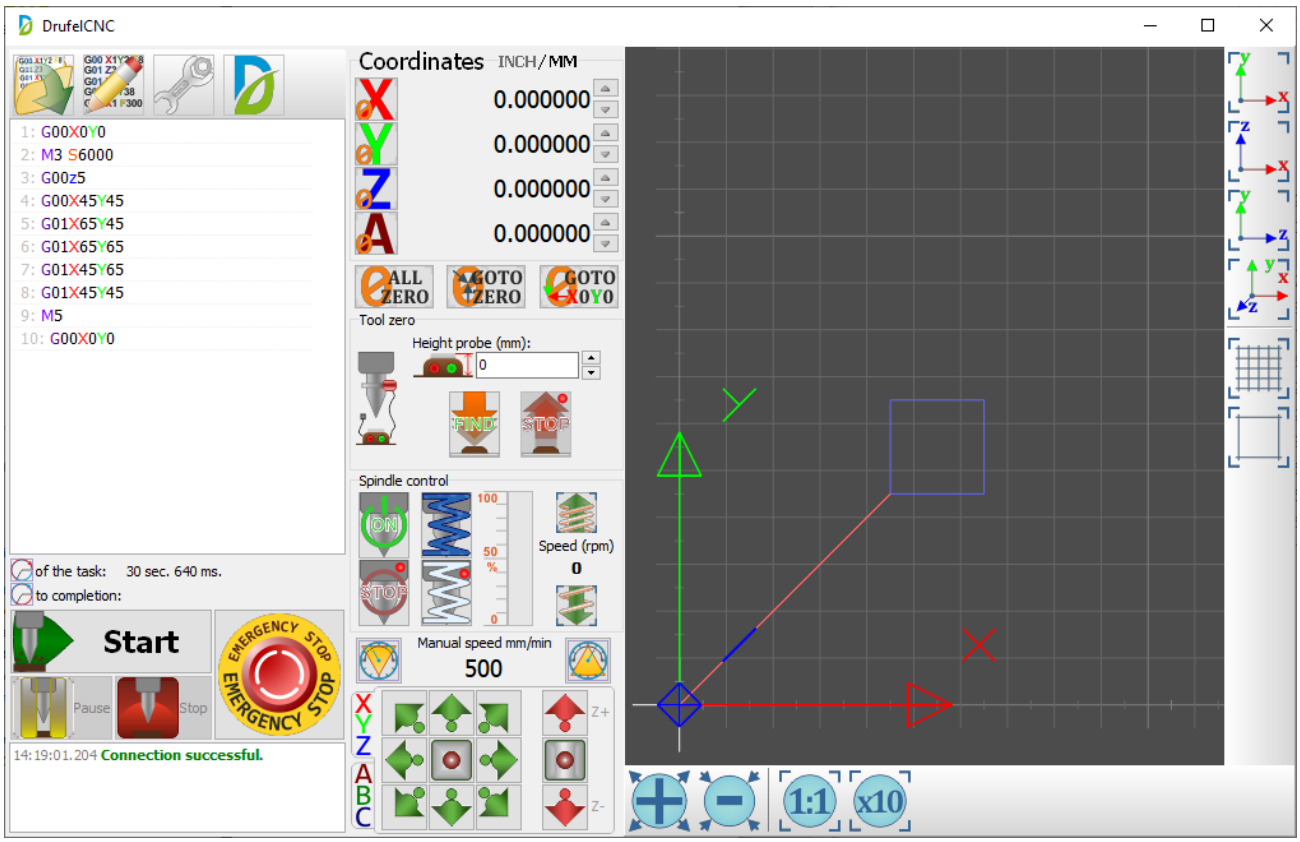
G10 - Enable data entry mode.

Set new coordinates for the origin

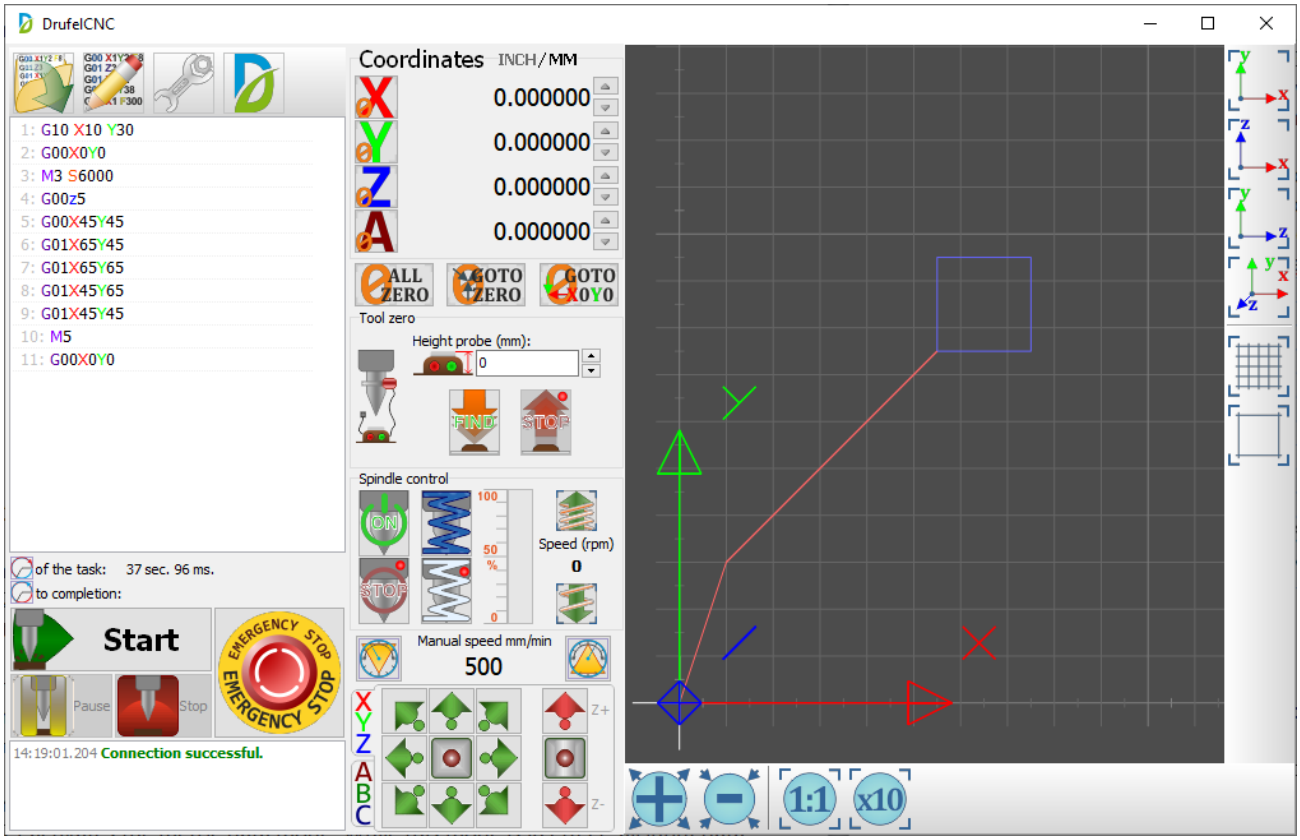
Format: G10 X10 Y10 Z10 A B C

The preparatory function G10 is modal and remains active until it is canceled by code G11, G53.

Let's look at an example:
G00X0Y0
M3 S6000
G00Z5
G00X45Y45
G01X65Y45
G01X65Y65
G01X45Y65
G01X45Y45
M5
G00X0Y0



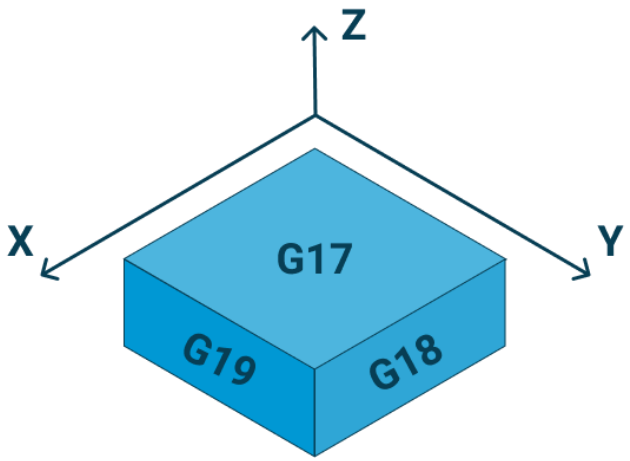
Add a line: **G10 X10 Y30**



G17, G18, G19 - Plane selection

In CNC programming, there are 3 G-codes for selecting a plane during NC programming, which are used to define two axes: X, Y or Z. The plane selection is modal and is valid for everyone until you enter a different circular plane command.

The 3 Plane selection G-Codes are:
G17 for XY Plane
G18 for XZ Plane
G19 for YZ Plane



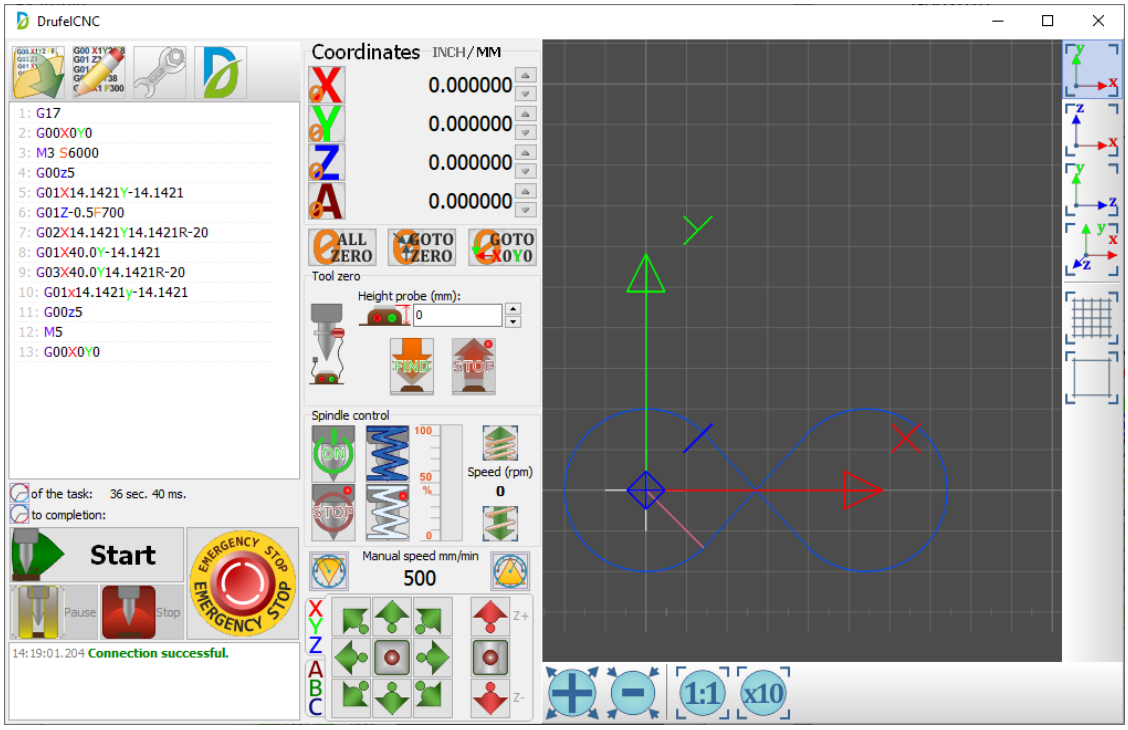
XY plane selection with G17 code.

The XY G17 plane selection code is set by default and sets the plane to the circular interpolation mode G02 and G03.

In the circular interpolation blocks, the words X, Y, Z, I and J. are valid. The word K is not valid. If the Z word is programmed in a circular interpolation block, then a spiral forms in the XY plane. The direction of the arc or spiral in the XY plane can be determined visually: Positive direction X - to the right side, positive direction Y - up. The XY plane has a right-handed coordinate system. In G17, the endpoint of the arc is defined in the block by the words X and Y. The center point of the arc is defined in the block by the words I and J.
G17 is activated by default. Code G17 is canceled by codes G18 and G19.

Example:
G17
G00X0Y0
M3 S6000
G00Z5
G01X14.1421Y-14.1421

G01Z-0.5F700
G02X14.1421Y14.1421R-20
G01X40.0Y-14.1421
G03X40.0Y14.1421R-20
G01X14.1421Y-14.1421
G00Z5
M5
G00X0Y0

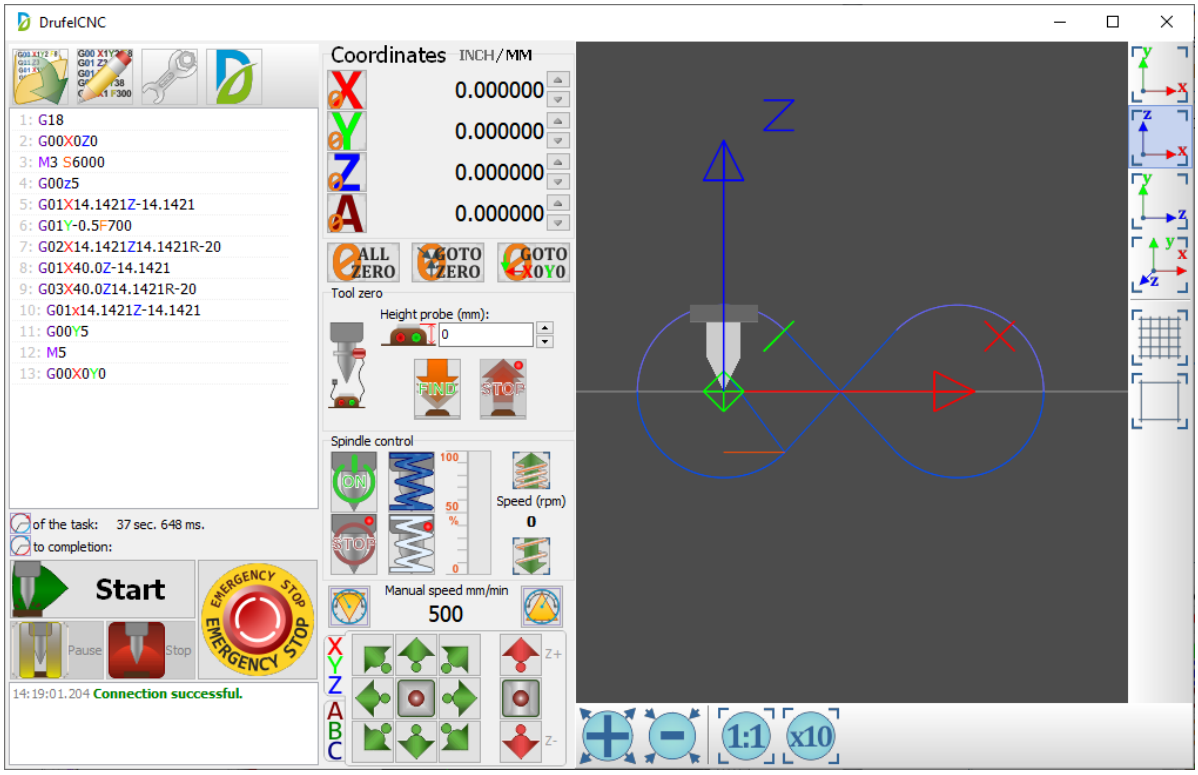


XZ plane selection. Code G18

XZ plane selection code G18 sets the plane to circular interpolation mode G02 and G03. In the circular interpolation blocks, the words X, Y, Z, I and J. are valid. The word J is not valid. If the word Y is programmed in a circular interpolation block, a spiral forms in the XZ plane. The direction of the arc or spiral in the XZ plane can be determined visually: The positive X direction is to the right, the positive Z direction is up. The XZ plane has a right-handed coordinate system.
Code G18 is canceled by codes G17 and G19.

Example:
G18
G00X0Z0
M3 S6000
G00z5
G01X14.1421Z-14.1421

G01Y-0.5F700
G02X14.1421Z14.1421R-20
G01X40.0Z-14.1421
G03X40.0Z14.1421R-20
G01x14.1421Z-14.1421
G00Y5
M5
G00X0Y0

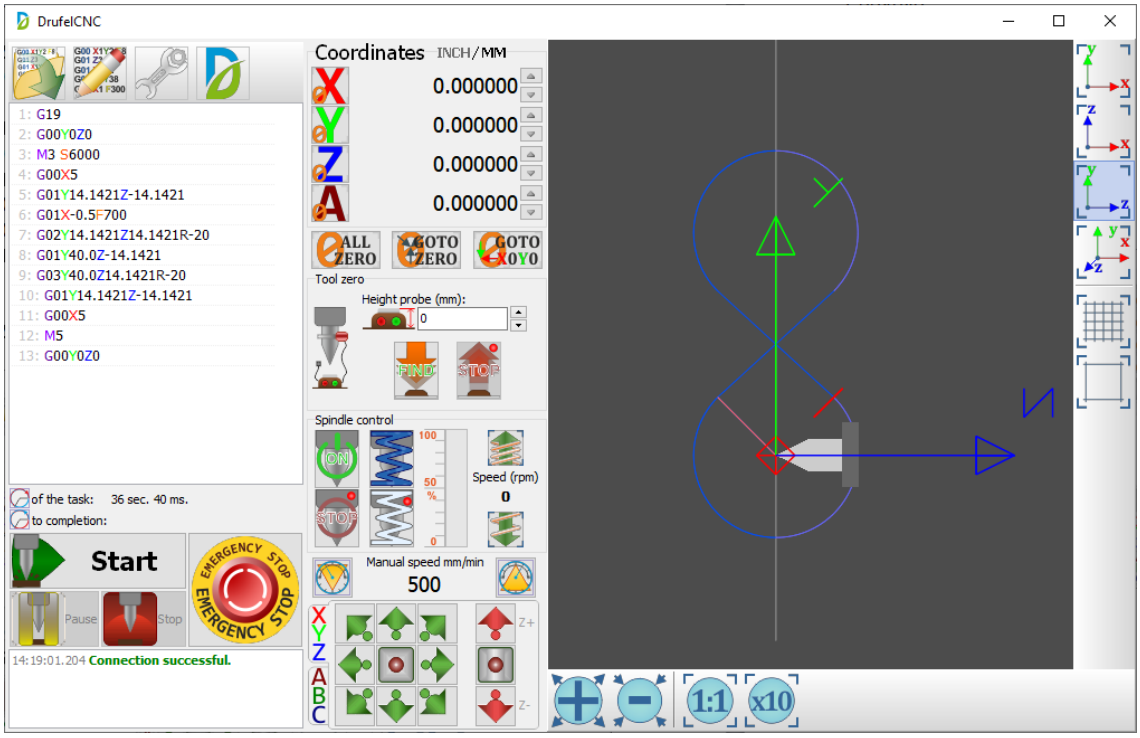


YZ plane selection. Code G19

The YZ plane selection code G19 sets the plane to circular interpolation mode G02 and G03. In circular interpolation blocks, the words X, Y, Z, I, and K are valid. Word I is not valid. If the word X is programmed in a circular interpolation block, then a spiral forms in the YZ plane. The direction of the arc or spiral in the YZ plane can be determined visually: The positive direction Y is to the right, the positive direction Z is up. The YZ plane has a right-handed coordinate system. In G19, the end point of the arc is defined in the block by the words Y and Z. The center point of the arc is defined in the block by the words J and K. Code G19 is canceled by codes G17 and G18.

Example:
G19
G00Y0Z0
M3 S6000

G00X5
G01Y14.1421Z-14.1421
G01X-0.5F700
G02Y14.1421Z14.1421R-20
G01Y40.0Z-14.1421
G03Y40.0Z14.1421R-20
G01Y14.1421Z-14.1421
G00X5
M5
G00Y0Z0



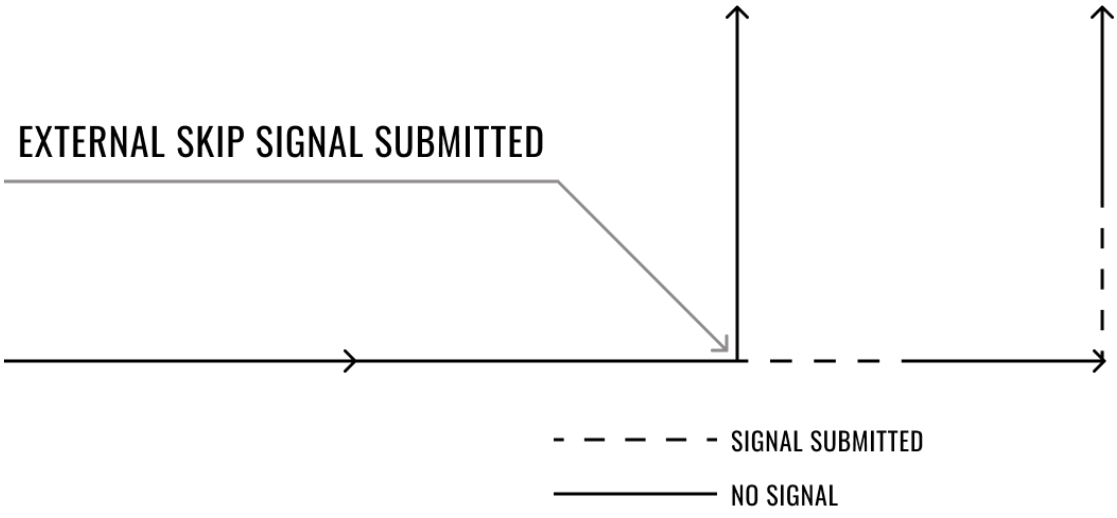
G20 G21

G20 - Input inch data
Code G20 activates the mode of working with inch data. While this mode is in effect, all input data is perceived as inch. It is recommended that in all programs that are written in inch sizes, put the G20 command at the beginning of the program, so that if the program that was running before, the metric mode was in effect, ensure that the correct format is selected.
The command is modal and is valid until canceled by the G21 command.

G21 - Enter metric data
Code G21 activates the metric data mode. While this mode is in effect, all input data is perceived as metric. It is recommended that in all programs that are written in metric sizes, put the G21 command at the beginning of the program, so that if the inch program was in effect before, ensure that the correct format is selected.

G31 - Movement of axes to a given position

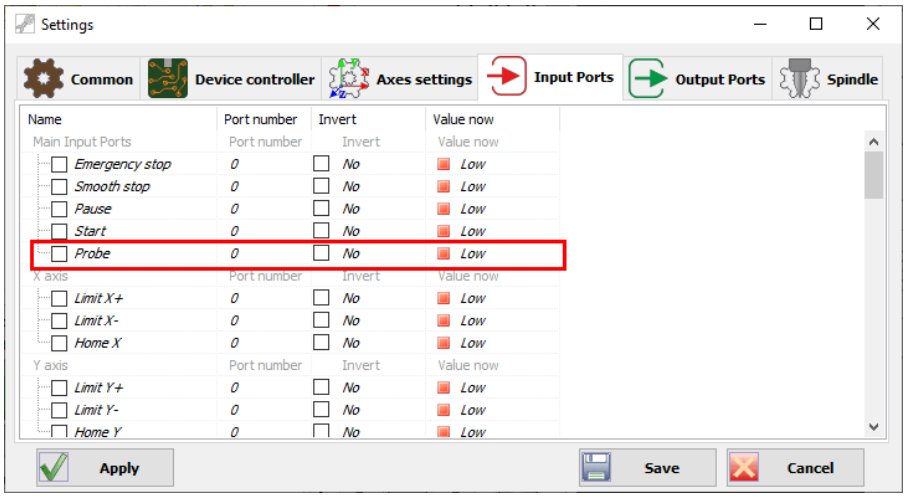
The G31 or G31.x command allows you to move the axes to the specified position. Using command G31 G31 or G31.x requires an encoder.
Movement by command G31 or G31.x is performed either until the specified position is reached, or until the signal of the additionally installed encoder (skip signal) is received.

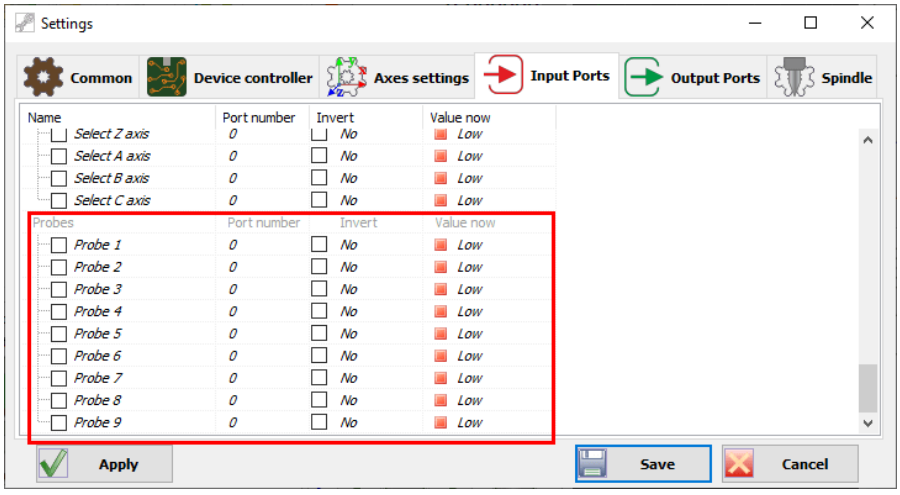


You can use the X, Y, Z or A axis and specify the end point for that axis. You can assign any feedrate you want, otherwise the last commanded feedrate will be used.

The probing command will be executed until the probe hits or a certain distance is covered.
In DrufelCNC, you can use G31, G31.0, G31.1, G31.2, G31.3, G31.4, G31.5, G31.6, G31.7, G31.8, G31.9 (provided that these sensors are enabled and configured).

G31 - Probe
G31.1 - Probe 1
G31.2 - Probe 2
G31.3 - Probe 3
G31.4 - Probe 4
G31.5 - Probe 5
G31.6 - Probe 6
G31.7 - Probe 7
G31.8 - Probe 8
G31.9 - Probe 9

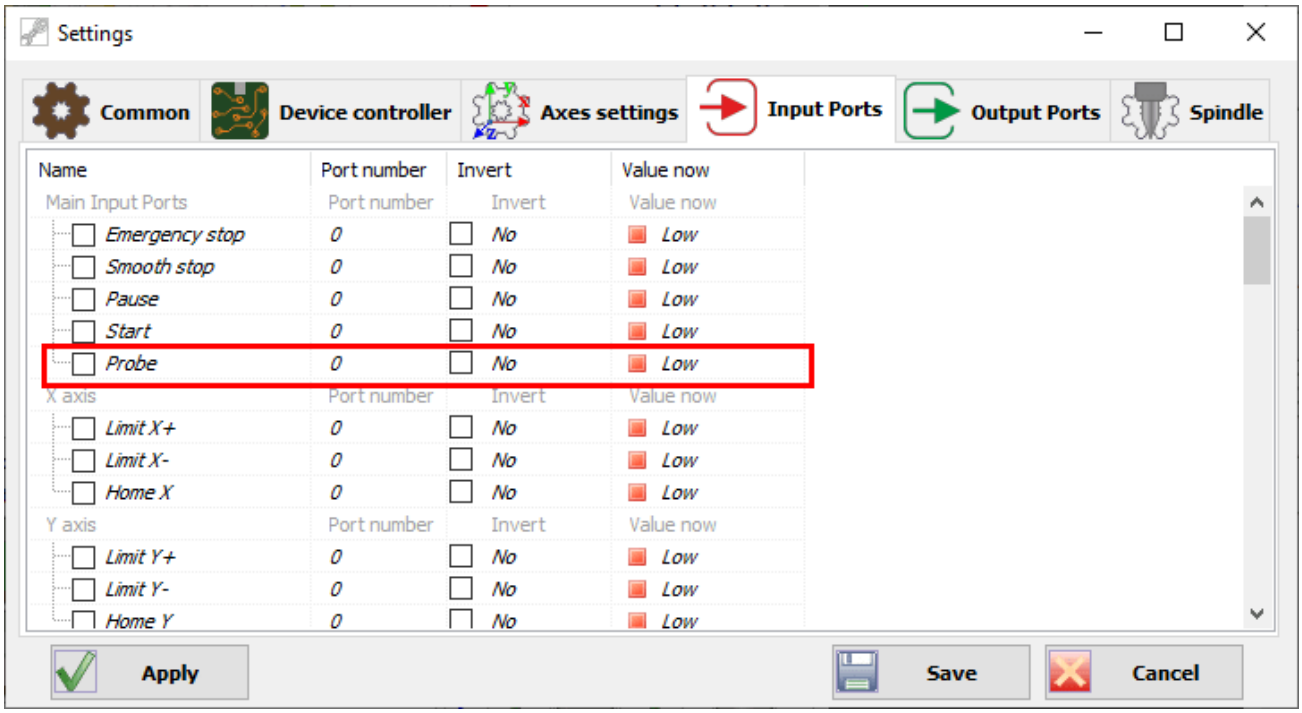




G38.n - Straight probe

G38.n Straight probe

G38.2 - Probe towards the workpiece, contact stop, signal error in case of malfunction
G38.3 - probe towards the workpiece, stop on contact
G38.4 - removal of the probe from the workpiece, stop on loss of contact, error in case of failure
G38.5 - probe from the workpiece, stop on loss of contact



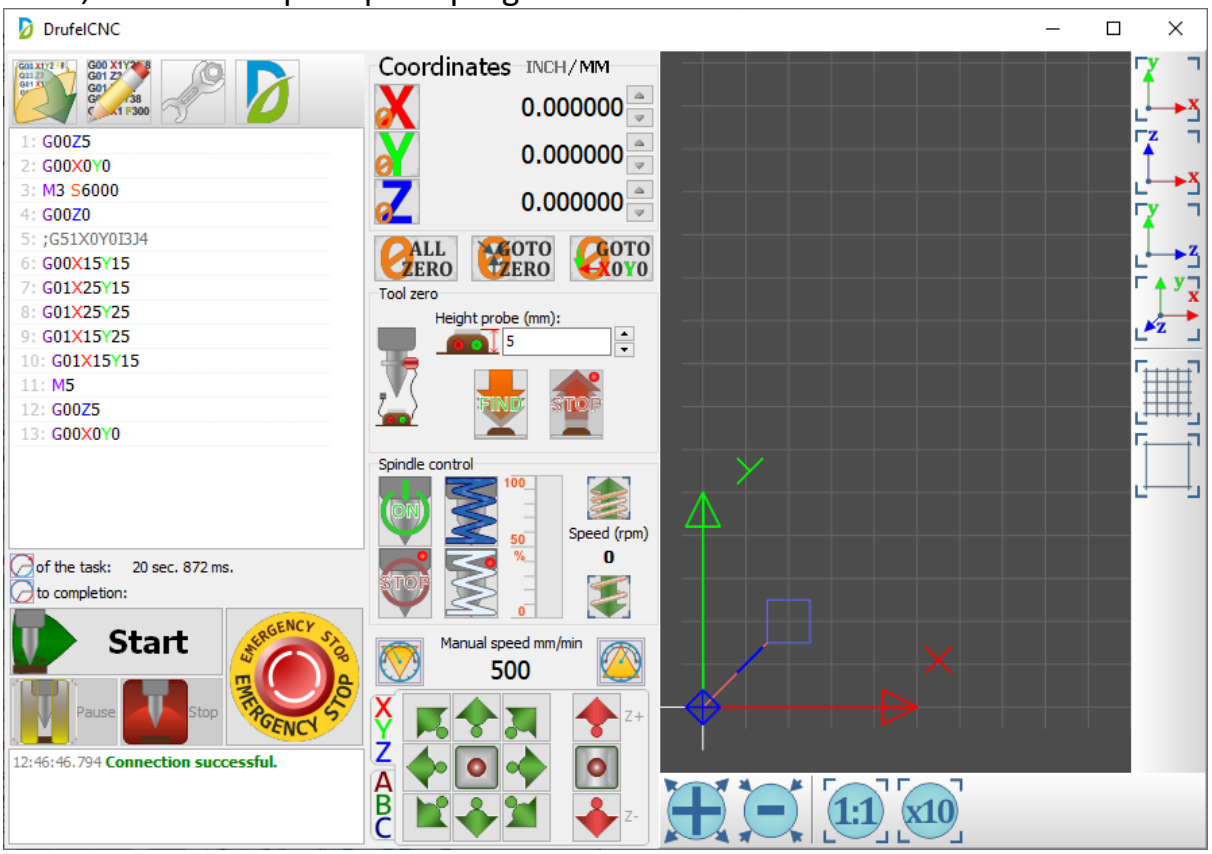
G50, G51 - Scale mode

The G51 command allows you to use the scale mode in numerical control systems. For each axis, it is possible to specify a scaling factor.
Command format: G51 X Y Z I J K
I - scale for X
J - scale for Y
K - scale for Z
X Y Z - coordinates of the center of scaling

Command format: G51 X Y Z A B C P
P - scale for axes X Y Z A B C
X Y Z A B C - coordinates of the center of scaling

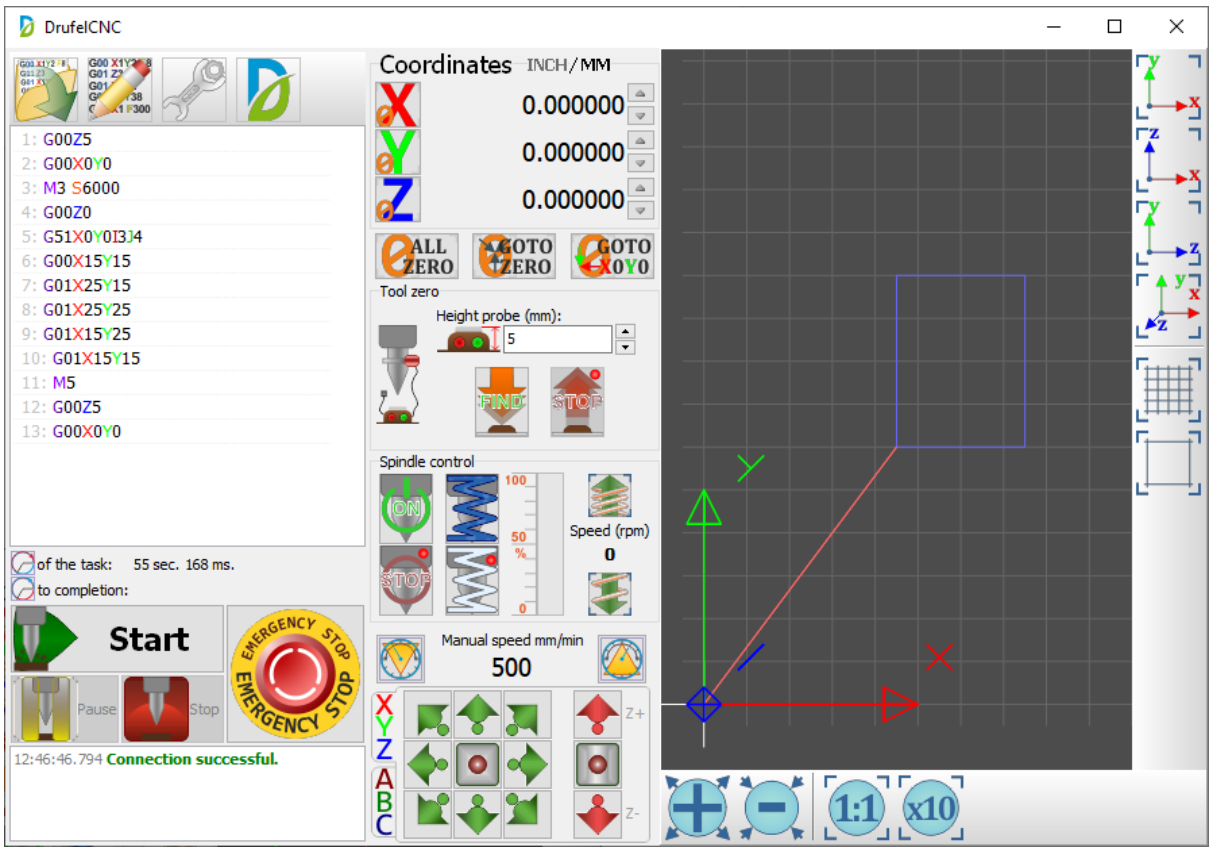
The G50 command allows you to cancel the scaling mode with numerical control systems (CNC) of machines.

Consider an example of using the G50 and G51 commands below:
First, create a simple square program.



Scale the shape by adding the line G51X0Y0I2I2. Total result:

G00Z5
G00X0Y0
M3 S6000
G00Z0
G51X0Y0I3J4
G00X15Y15
G01X25Y15
G01X25Y25
G01X15Y25
G01X15Y15
M5
G00Z5
G00X0Y0



G53 - Disabling the offset of the origin of the coordinate system

The G53 command is used to disable the offset of the origin of the coordinate system.
When the G53 command is executed, the working coordinate offsets are temporarily canceled and the machine coordinate system is used.
The G53 command is a safe way to return to the initial position of the machine.
The offset values of the origin of the coordinate system are stored in tables.
The G53 command is not modal, acting only in the frame in which it is assigned.

G68 - Rotate coordinate system

The G68 command is used to perform rotation (rotation) of the coordinate system by a certain angle.
Command format: G68 X__ Y__ Z__ R

The choice of coordinates depends on the selected plane (G17, G18, G19). X and Y and Z are the coordinates of the point relative to which the coordinate system will be

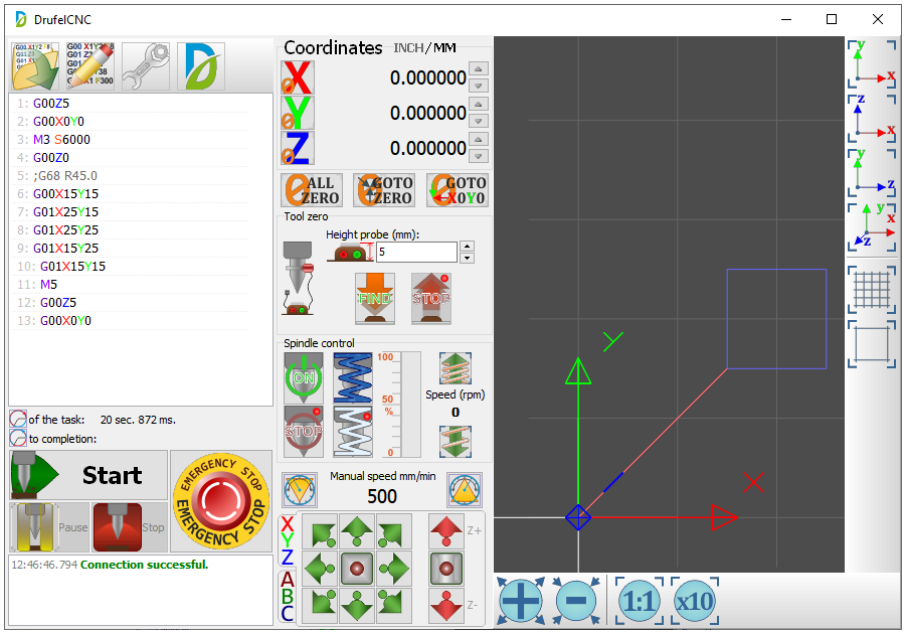
rotated, and R - determines the value of the angle of rotation. If X Y is not specified, then X Y Z are equal to the previous position.

G69 - Cancel Rotation

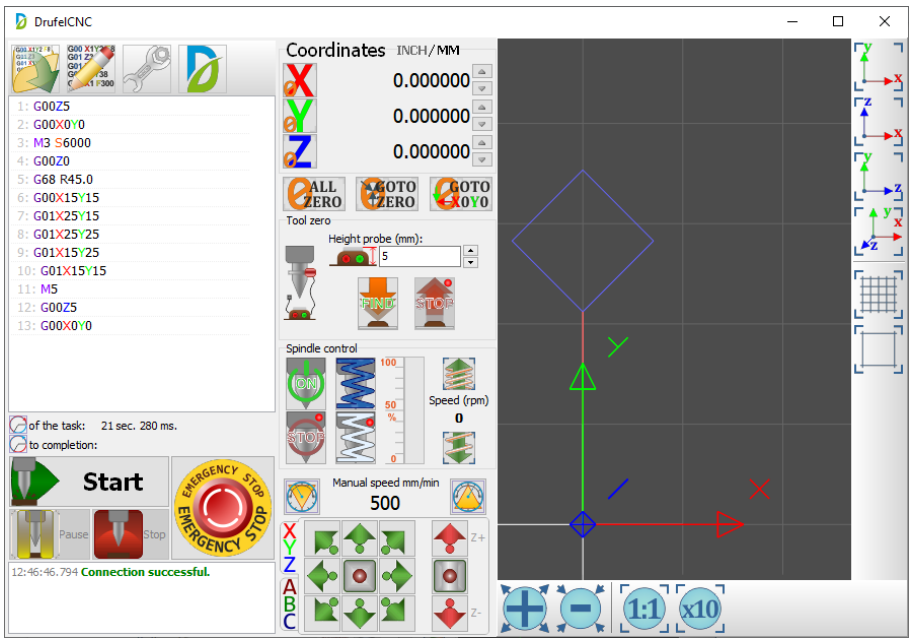
G69 command is used to cancel the rotation of the coordinate system by a certain angle.

Example using the G68 and G69 commands:
First, create a simple square program.

G00Z5
G00X0Y0
M3 S6000
G00Z0
G68 R45.0
G00X15Y15
G01X25Y15
G01X25Y25
G01X15Y25
G01X15Y15
M5
G00Z5
G00X0Y0



Rotate the shape 45 degrees, add the line G68 R45.0. Total result:



G80 - Cancel Canned Cycle

G80 - code cancel of the constant cycle.

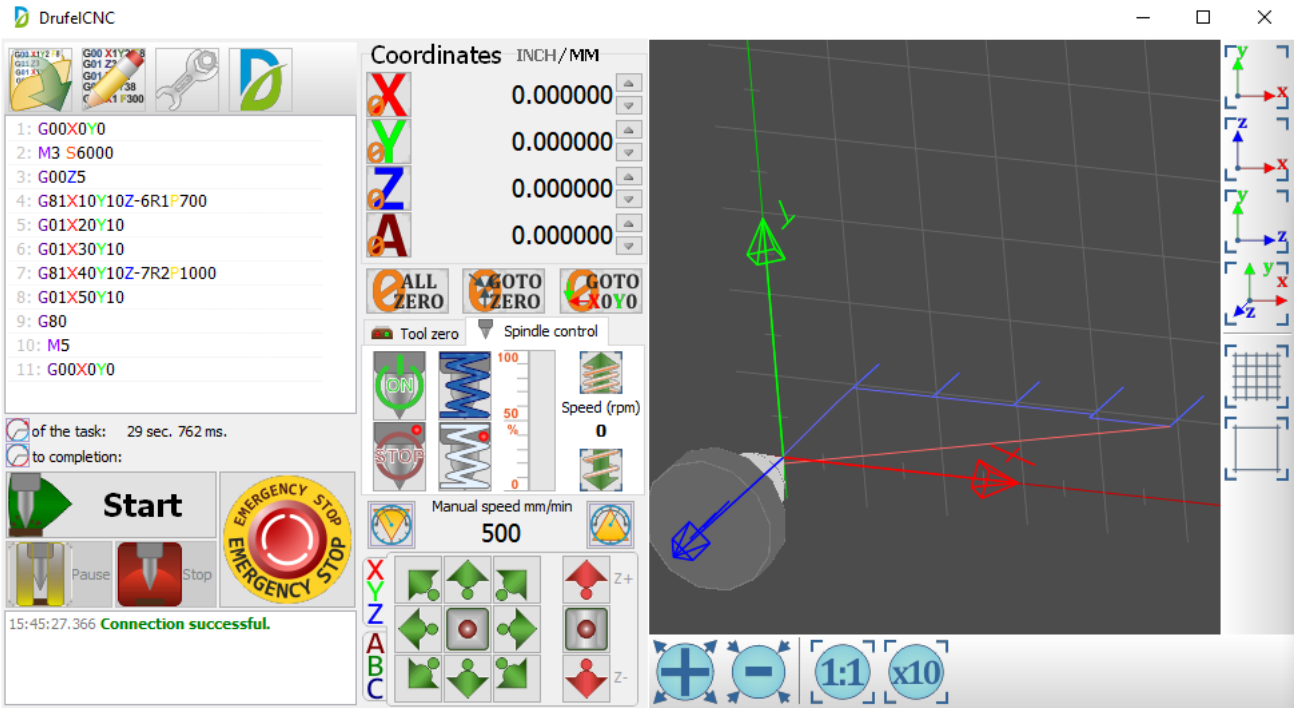
G81 - Simple Drilling Cycle

Code G81 is for calling a standard drilling cycle. The following frame shows a typical format for this loop:

G81 X10.0 Y15.3 Z-3.0 R0.5 F50.

X and Y determine the coordinates of the holes to be machined. Z indicates the final drilling depth, and R is used to establish the retraction plane.

G00X0Y0
M3 S6000
G00Z5
G81X10Y10Z-6R1
G01X20Y10
G01X30Y10
G81X40Y10Z-7R2
G01X50Y10
G80
M5
G00X0Y0



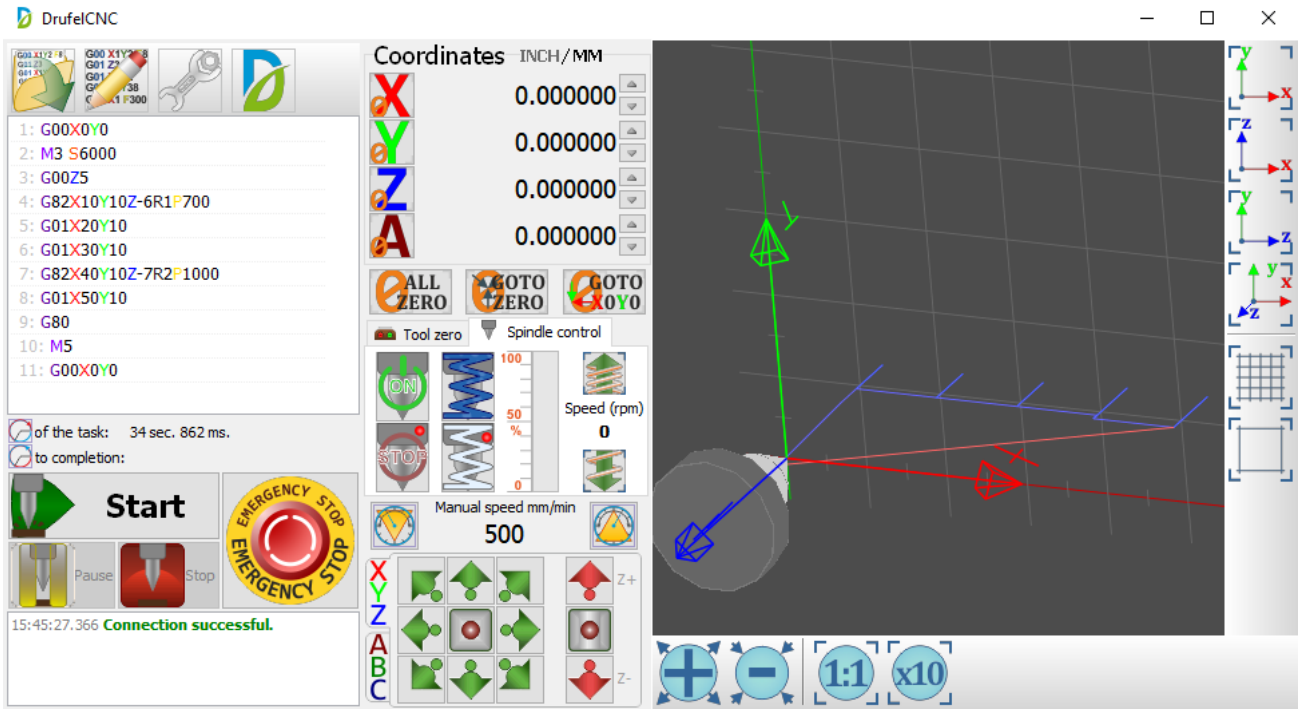
G82 - Drilling Cycle with Dwell

Code G82 is for calling a pause drilling cycle. The following frame shows a typical format for this loop:

G82 X10.0 Y15.3 Z-3.0 R0.5 P1000 F50.

X and Y determine the coordinates of the production holes. Z indicates the time of the drilling depth, the address of the waiting time at the bottom of the hole. The exposure time is indicated in milliseconds.
The G82 command is often used to drill blind holes, since the programmed waiting time provides better chip removal from the bottom of the hole.

G00X0Y0
M3 S6000
G00Z5
G82X10Y10Z-6R1P700
G01X20Y10
G01X30Y10
G82X40Y10Z-7R2P1000
G01X50Y10
G80
M5
G00X0Y0



G90 - Absolute positioning mode.

In the absolute positioning mode G90, the actuators are moved relative to the zero point of the working coordinate system G54-G59 (it is programmed where the tool should move). The G90 code is canceled using the relative positioning code G91.

G90.1 - Absolute distance mode for I , J and K offsets.

G90.1 is the absolute distance mode for offsets I, J & K. When G90.1, I and J are valid, both must be indicated with G2 / 3 for the XY plane or J and K for the XZ plane, otherwise this is an error.

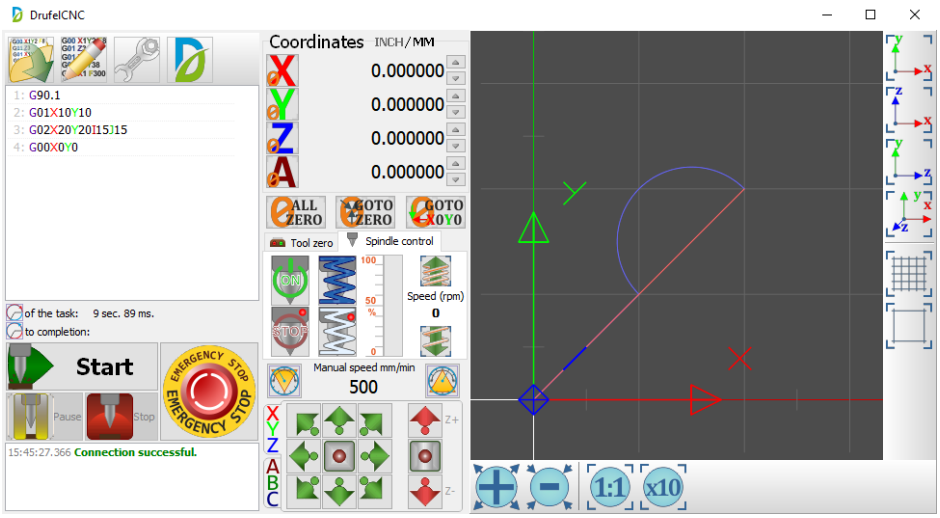
Example:

G90.1

G01X10Y10

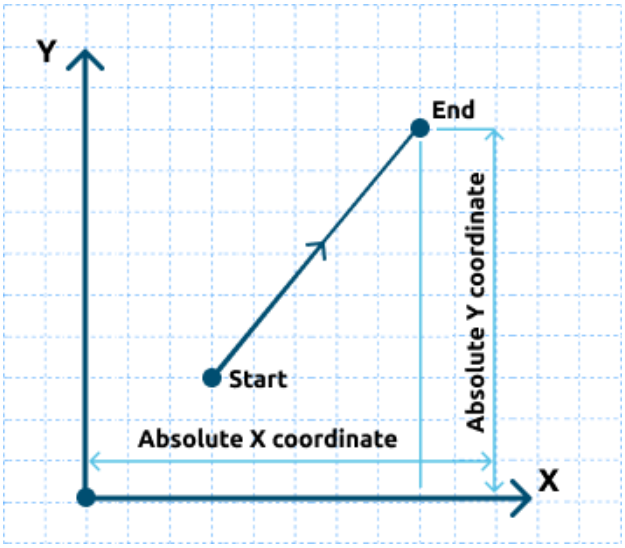
G02X20Y20I15J15

G00X0Y0



G91 - Relative positioning mode.

In the relative (incremental) positioning mode G91, the position of the actuator, which it occupied before moving to the next reference point, is each time taken as the zero position (it is programmed how long the tool should move). Code G91 is canceled by absolute position code G90.



G91.1 - Incremental distance mode for I, J and K offsets

G91.1 - distance increment mode for offsets I, J & K. G91.1 Returns I, J & K to their default behavior. G91.1 is active by default.

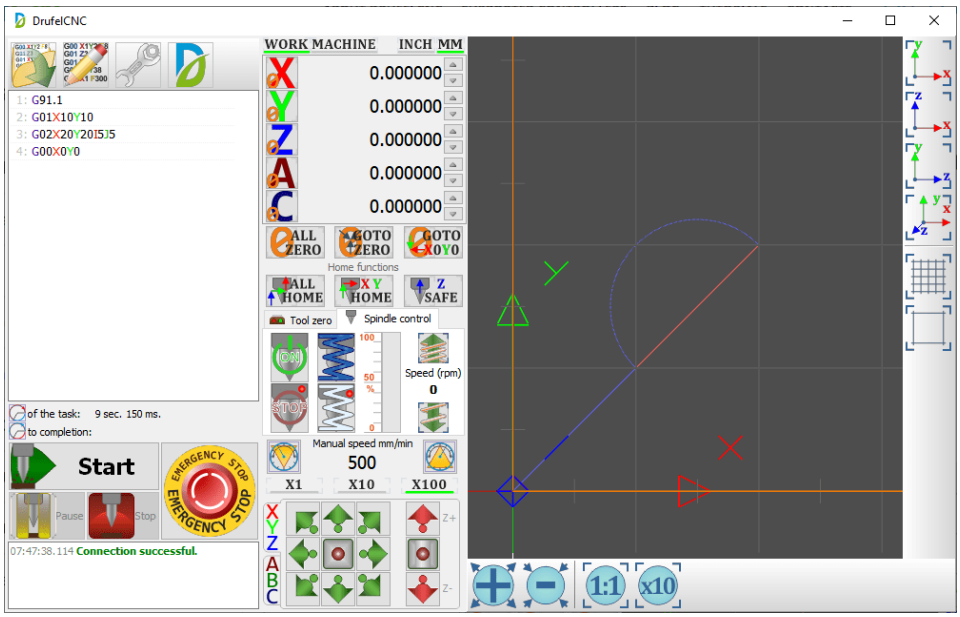
Example:

G91.1

G01X10Y10

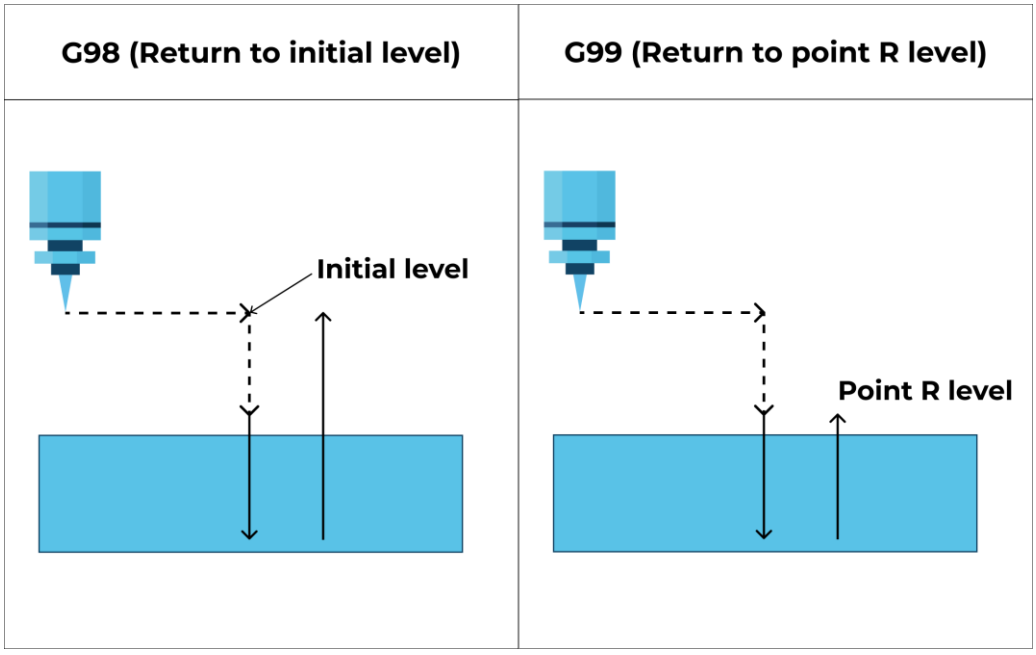
G02X20Y20I5J5

G00X0Y0



G98, G99 - Canned Cycle Return or Feed Rate Modes

G98 - take back to the position in which the axis was located immediately before the start of this series of one or more continuous continuous cycles.

G98 specifies a reset. G98 causes the tool to move to the position where it started the cycle.

G99 specifies the return to the reference point or safe retraction zone. **G99** causes the tool to move to the point defined by the R parameter in the block with the hard cycle function.