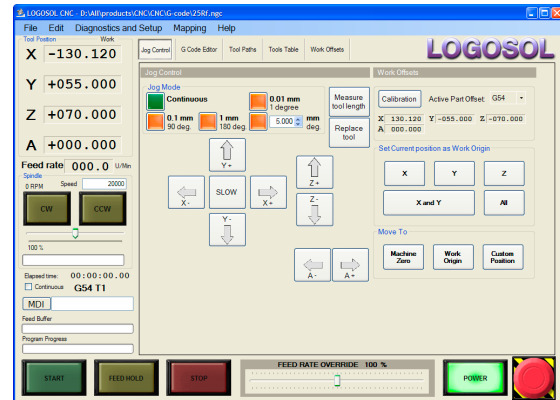
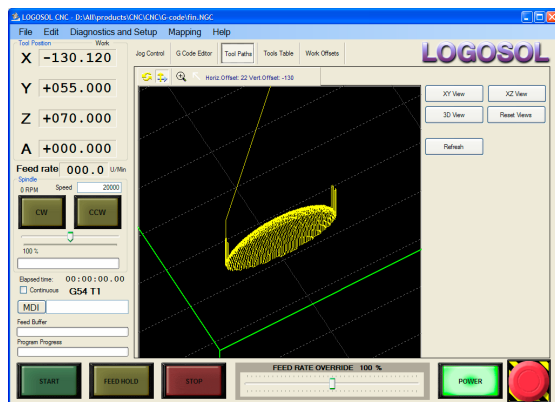


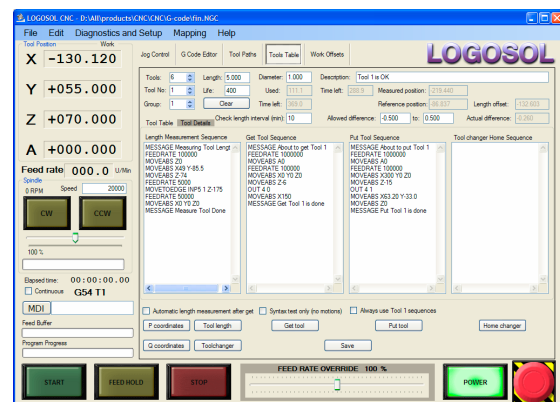
G Code Editor and Batch Jobs – page 2



Joystick Control – page 5



Tool Paths – page 9



Tools Management – page 10

Appendix A. Tool Length Offset – page 12

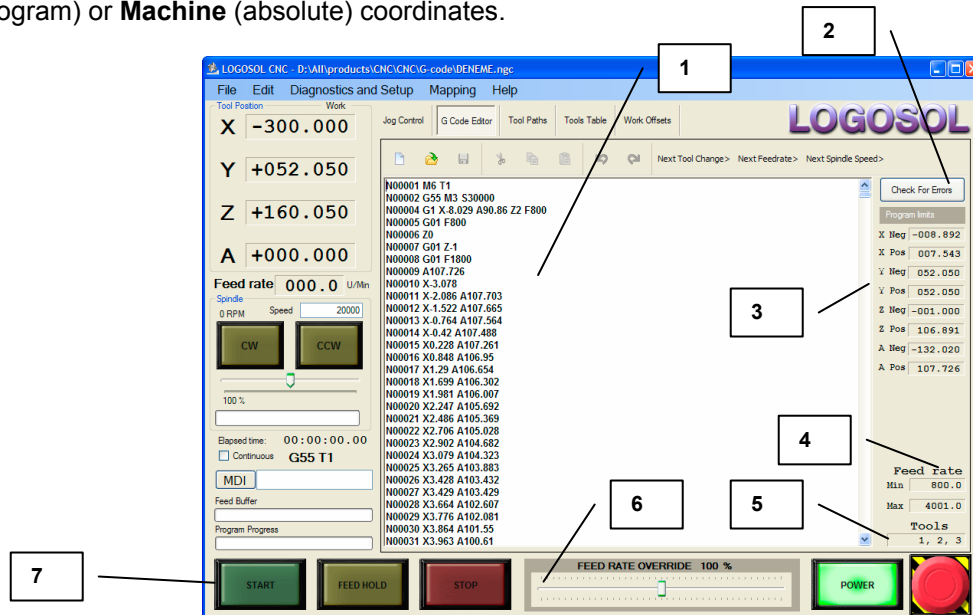
Appendix B. Cutter Radius Compensation – page 13

Appendix C. Supported G codes – page 14

Appendix D. Supported M codes – page 23

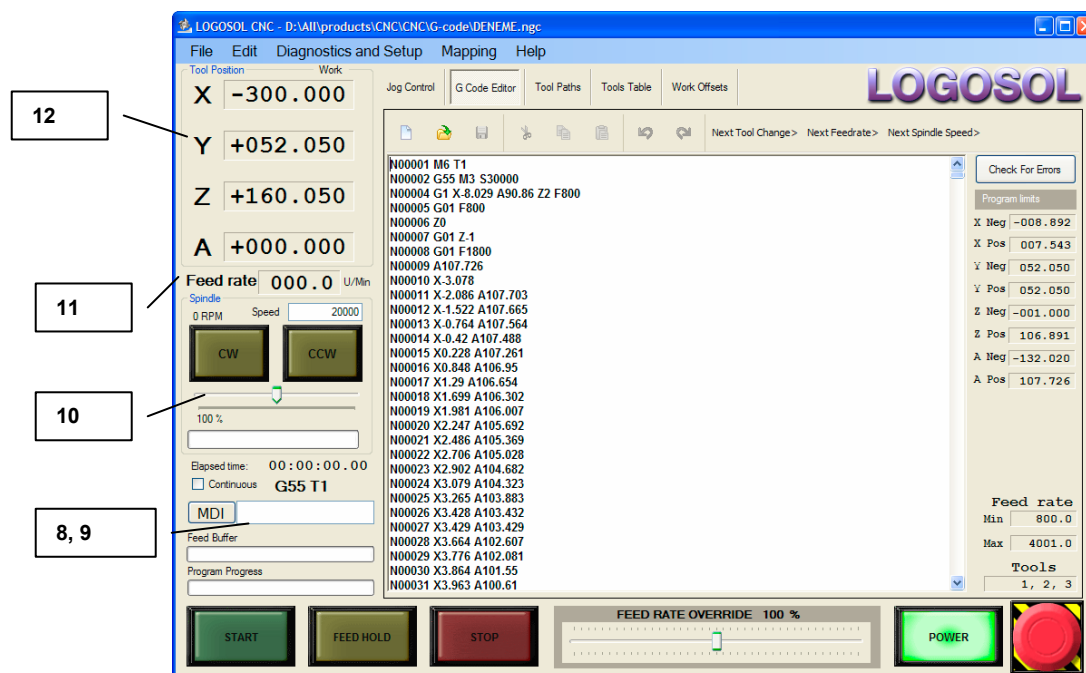
G Code Editor

G Code Editor allows loading, editing, saving and running G code file. The tool position is displayed in **Work** (program) or **Machine** (absolute) coordinates.



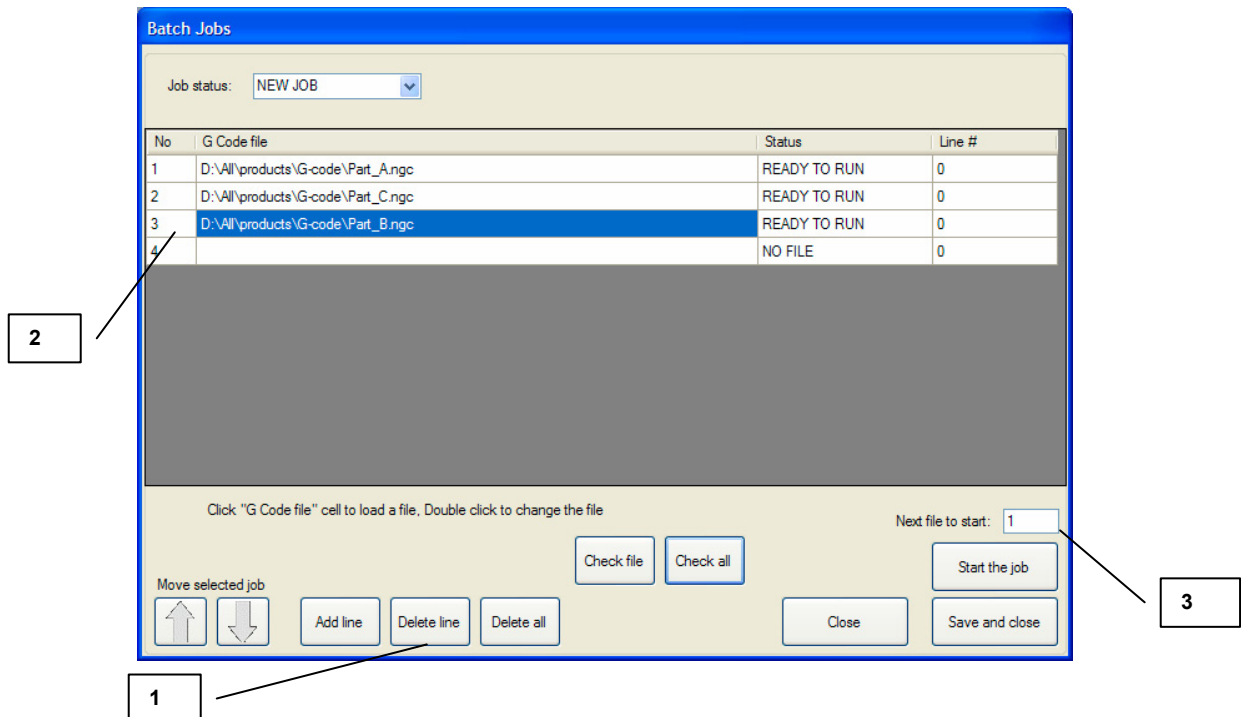
1. After loading G code file (program) is automatically checked for errors and machine limits violation. The file could be edited using standard PC compatible keyboard. The screen allows scrolling the G code file, searching for specified data and positioning the cursor at **Next Tool Change**, **Next Feedrate**, or **Next Spindle Speed**. Basic editor functions such as copy, paste, etc. are implemented. The current row executed by the CNC is marked during the file execution.
2. To check for errors the loaded G code file Click **Check for Errors** button.
3. Travel distances for each axis are displayed in this section. The program coordinates are converted into machine (absolute) coordinate using **Active Part Offset** and **Tool Table** data. To switch between program and machine tool travel limits click **Tool position - Work/Machine**. Error message appears if the converted program limits are outside of the machine work zone. Refer to **Jog Control** section for **Active Part Offset** detailed description.
4. Minimal and maximal linear speed programmed.
5. Tools used in the current program.
6. **FEED RATE OVERRIDE 100%** (0 – 200%). The speed manual correction - from 0 to two times programmed speed.
7. Program control buttons.
 - **START** – starts the program from the beginning or from the selected row, or **only when MDI bar is green** – starts the program row entered in **MDI** field.
 There are three G code file execution modes:
 - **Multi line mode** –execution is controlled according to the buffer state;
 - **Single line mode** – single row is loaded and executed into the buffer;
 - **Single line mode with stop** – after each row and the execution goes in **FEED HOLD** state and waits until the next **FEED HOLD** button click.
 To select one of the modes go to **Diagnostic and Setup** menu.
 - **FEED HOLD** – stops the current motion without stopping the program. Press **FEED HOLD** again to continue.
 - **STOP** – stops the current program.

8. Manual data input. Entering data in **MDI** field activates Manual Data Input Mode (**MDI** bar becomes green). If the commands are entered already - click on **MDI** bar to switch the mode to manual data input. **MDI** command field allows single command row to be entered and executed without affecting the program data. To execute the entered row clock start. MDI mode becomes inactive after the commands entered are executed.
9. Current and modal data display.
 - **Program Progress** - the estimate of the remaining program to be executed.
 - **Elapsed time** – time from the moment when the program was started. The counter will stop when **STOP** is clicked or at program end. **FEED HOLD** will not affect the counter.
 - **Active Zero offset** (G55 is shown).
 - **Tool number** in use (T1 is shown).
 - **Reference tool number**. By default blank. When reference tool mode is used - reference tool number.
 - **Continuous** – if checked the program will be restarted automatically after executing the last row.
10. Spindle manual control. Spindle **CW** (clockwise) and **CCW** (counterclockwise). Clicking one of the buttons will start the spindle in the selected direction with the default speed (the speed set in **Diagnostic and Setup / Spindle menu**). Second click or clicking the opposite direction will stop the spindle.
11. **Feed rate**. Actual linear speed as a result of the axes movement.
12. Current tool position could be displayed in **Machine** or **Work** (program coordinates). This displays the tool position relatively to **Machine Zero** or **Work Zero**.



Batch Jobs

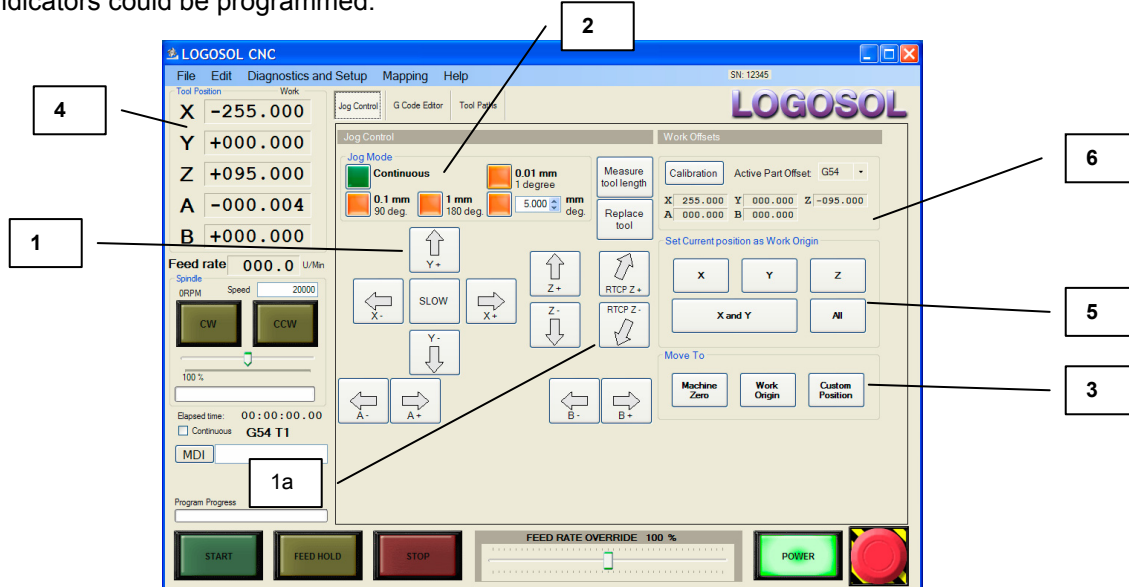
Batch Jobs screen allows sequence of multiple G code files to be executed. To enter Batch mode select **Batch Jobs** from **File** menu. Enter the files you want to execute. Check the files for errors by clicking **Check file** or **Check all**. To start the batch job click **Start the job** button. The G-code files will be executed in the order displayed. **STOP** button (main screen) will interrupt the program execution. To restart the batch job click **Start the job** button. The job will restart from the point where the program is interrupted. To start the G code file currently loaded click **START** button on main screen. Using **Next file to start** and Line # =0 the operation could be restarted from any file on the screen or from the beginning of the current G code file. All G code files *Status* must be **READY TO RUN**, **FINISHED** or **INTERRUPTED** to start the batch job.



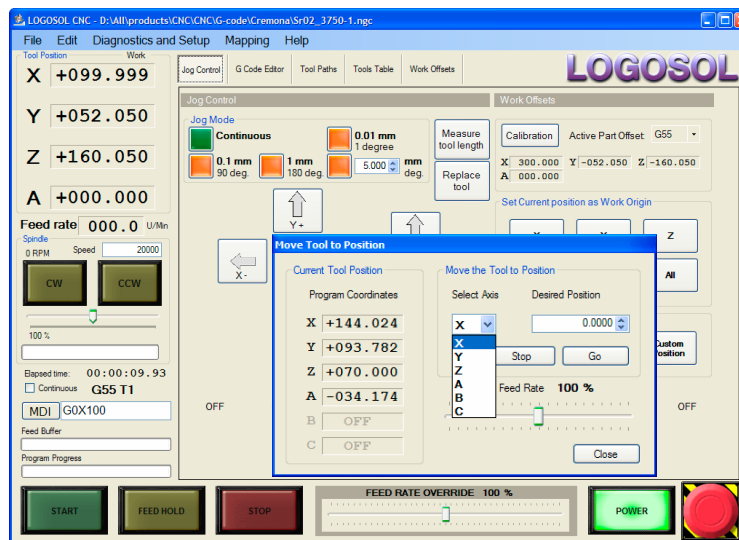
1. Add or delete lines.
2. Add files by single click on empty line or replace existing file by double click.
3. Select the file you want to start with.

Jog Control

The screen allows movements in **Jog Control** mode, setting the work coordinate system offsets, custom control buttons, and (if the proper hardware is installed) tool measurement. Up to eight custom buttons / indicators could be programmed.



1. Axis and direction select in **Jog Control** mode. **SLOW / FAST** button selects between **Jog speed** and **Rapid speed**.
- 1a. When **RTCP** (Rotating Tool Center Point) is active, the machine could be moved parallel to the tool axial line. When the rotation angle is 0 deg, the movement is parallel to the Z axis. For all other - the motion is distributed between X and Z axes according to rotation angle.
2. **Jog Mode:**
 - **Continuous** – Select the axis and direction, click and hold the mouse pointer. The axis will move until the mouse button is released;
 - **Step mode** - **0.01mm / 1 degree**, **0.10mm / 90 degree**, **1.00mm / 180 degree** and custom – the axis will make a motion equal to the selected step. Depending on the axis (linear or rotary) the motion will be in mm (inch) or deg.
3. Movements to **Machine Zero**, **Work Origin** or **Custom Position**. **Move To - Machine Zero** or **Work Origin** moves the machine to the point with **Machine / Work** coordinates 0 for all axes. To move to a different position select **Custom position**. The desired position should be entered in work coordinates.

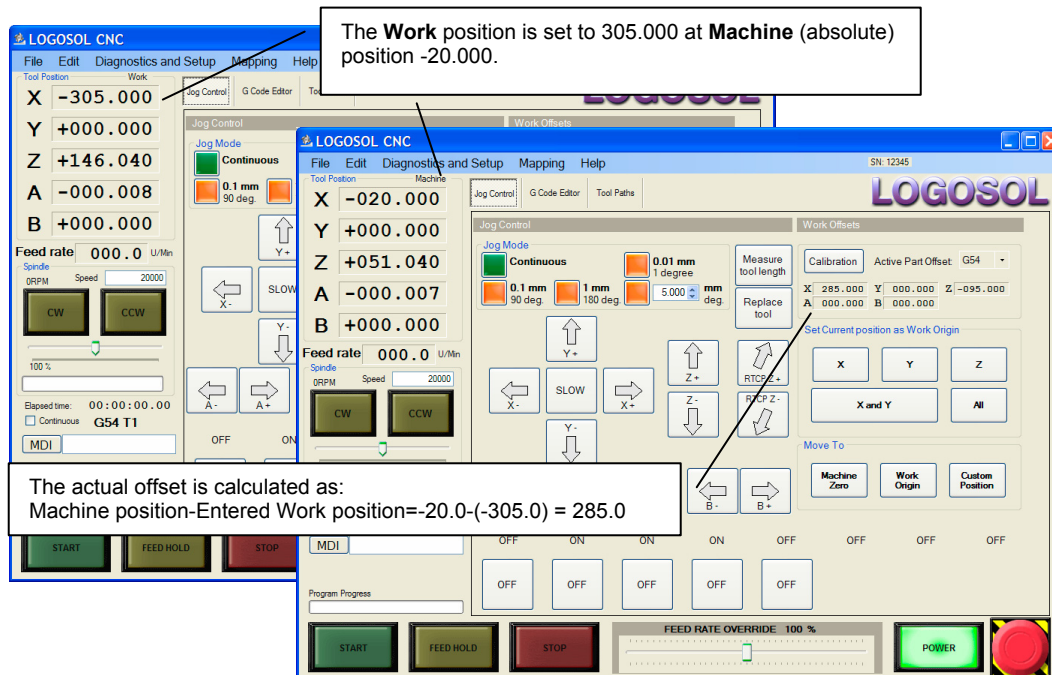
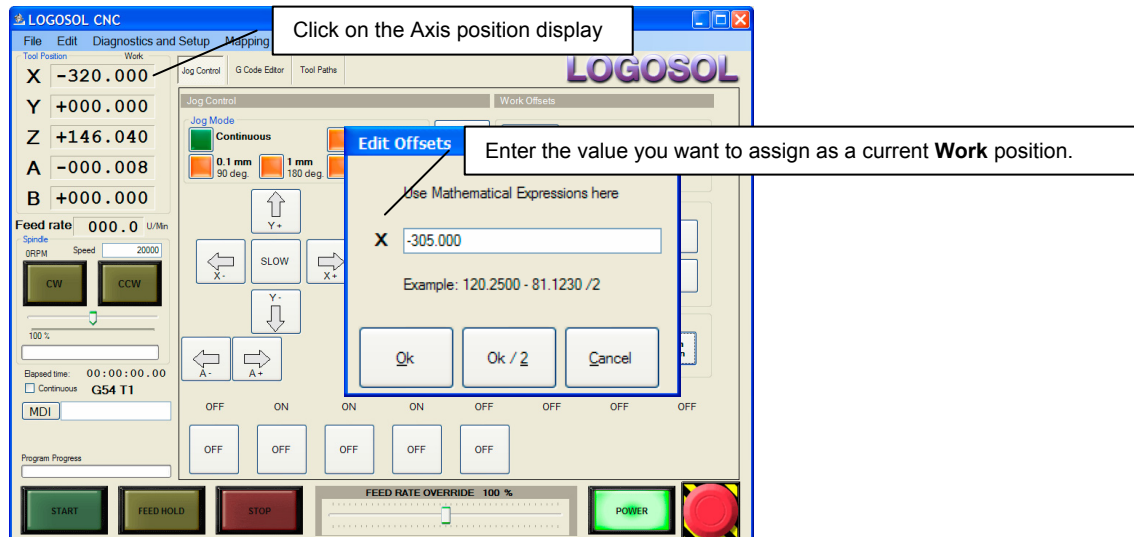


Custom Position screen

- Displays current **Work** (program) or **Machine** (absolute) tool position. To switch between click on **Work** (**Machine**) label at the right upper corner of the field.
The field could be used to edit **Active Part Offset**.

Method I:

Click on **X** (**Y** or **Z**) coordinates display in **Tool Position** section.



Method II:

To set the current positions as origin move the machine to the target position and use the buttons in **Set Current position as Origin**.

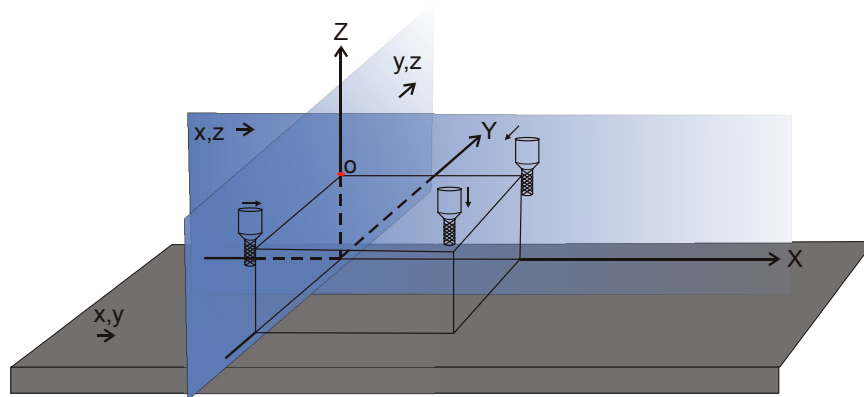
The first screenshot shows the LOGOSOL CNC control interface. The 'Tool Position' section displays X: +255.000, Y: +000.000, Z: +095.000, A: +000.003, and B: +000.000. The 'Jog Control' section includes buttons for 'Continuous', '0.1 mm', '1 mm', and '5.000 mm' jog modes, along with directional arrows and a 'SLOW' button. The 'Work Offsets' section shows 'X: 000.000', 'Y: 000.000', 'Z: -095.000', 'A: 000.000', and 'B: 000.000'. A callout box points to the 'Set Current position as Work Origin' button in the 'Work Offsets' section.

The second screenshot shows the same interface after the 'Set Current position as Work Origin' button has been pressed. The 'Tool Position' section now displays X: +000.000, Y: +000.000, Z: +095.000, A: +000.003, and B: +000.000. A callout box points to the 'X: +000.000' value, stating 'The Work Tool Position is set to 0 (X=0)'.

The third screenshot shows the same interface after the 'Machine Zero' button has been pressed. The 'Tool Position' section now displays X: +255.000, Y: +000.000, Z: +000.000, A: +000.003, and B: +000.000. A callout box points to the 'Active Part Offset: G54' dropdown menu, stating 'The active Part Offset is set equal to the Machine current Tool Position'.

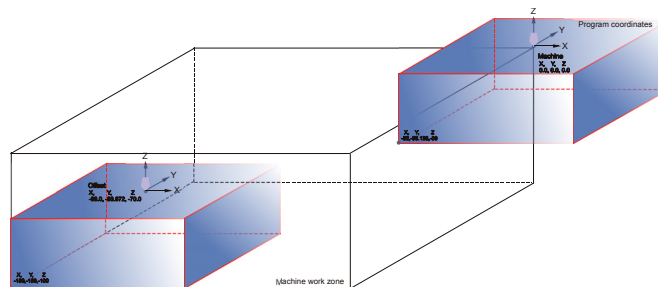
5. To place **Work Origin** (O) at a specific point (far upper corner of the block on the picture below).

- Move the tool parallel to **X** axis to touch **y, z** plane of the block and click **X** from "Set Current position as Work Origin".
- Move the tool parallel to **Y** axis to touch **x, z** plane of the block and click **Y** from "Set Current as position Work Origin".
- Move the tool parallel to **Z** axis to touch the top of the block and click **Z** from "Set Current position as Work Origin".



6. **Active Part Offset - G54 to G59.3** allows different origins to be selected. Machine (absolute) tool position is calculated by the software, using **Active Part Offset** according to the formula:

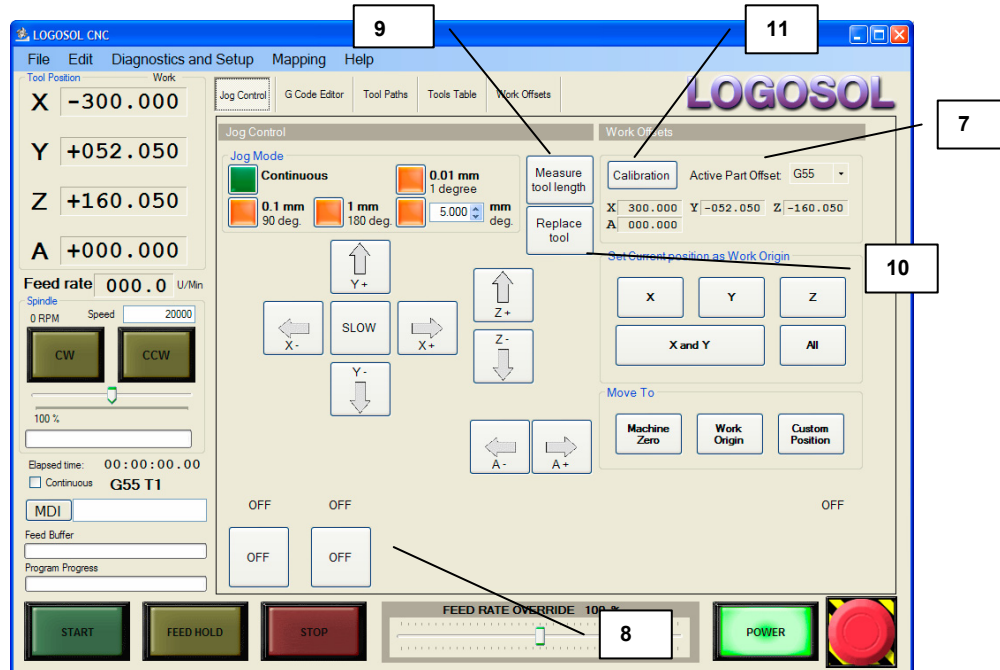
$$\text{Work}(X,Y,Z,A,B,C) + \text{Offset}(X,Y,Z,A,B,C) = \text{Machine}(X,Y,Z,A,B,C)$$



Depending on the selected **Active Part Offset** the program coordinates are transformed into machine (absolute) coordinates during the program execution. The transformed coordinates are controlled for limits (machine and software) violations.

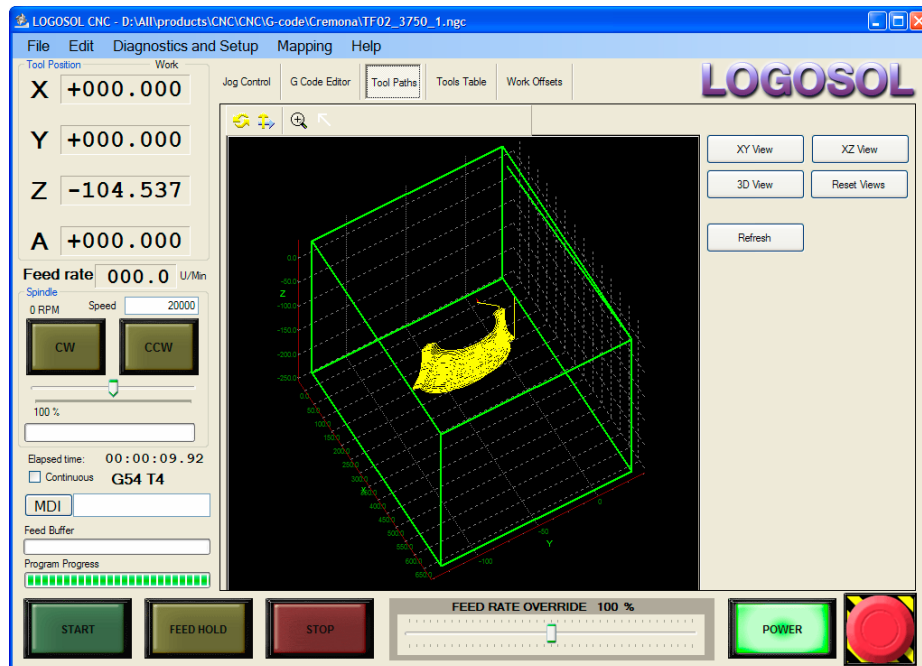
- Buttons and indicators (LED) assigned in **Mapping** menu are displayed in this section of the screen. Up to 8 buttons / LED's could be active.
- Measure tool length** is applicable when tool measurement hardware is installed. Contact the machine manufacturer for information about tool measurement devices. For information how to setup the software when tool measurement hardware is installed refer to **Logosol CNC Setup Guide**. The measured Z value is added as offset during the G code execution and will not affect the **Active Part Offset**.
- Click at **Replace tool** to set the tool **Life** data. The corresponding values should be entered by machine manufacturer or other authorized technician. Entering the tool data will reset the time counter for this tool.

10. **Calibration** procedures are G code programs developed by the machine manufacturer, targeted for specific machine applications. Refer to machine manufacturer for detailed information.



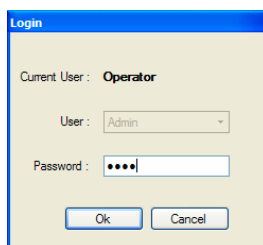
Tool Paths

Tool Pats screen provides the 2D or 3D image of the tool path. Using the buttons at the left upper corner the image could be moved, rotated or zoomed.



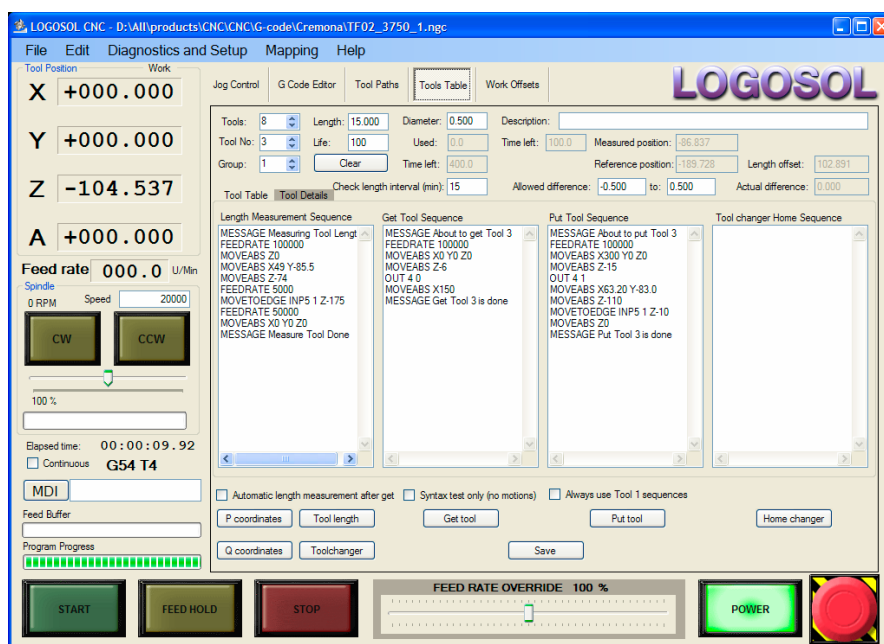
Tools management

To make **Tools Table** and **Work Offsets** available for editing select **Login** from the **File** menu. Type the password "1234" and click OK.



Tool life management and Tools Table

Tools Table screen is developed to provide access to tool change control sequences and tool data. The sequences should be edited by an experienced service technician with knowledge of the CNC software and the particular machine. Usually all sequences are entered and tested during the production cycle of the machine. Refer to **Logosol CNC Setup Guide** for more information about integrated tool control capabilities.



Life – the limit of the tool use (min).

Used – accumulated time when the tool is in the spindle and spindle is turning (**Used**=0 after tool replacement).

Time left – the difference between **Life** and **Used**. When **Time left** is 0 for one of the tools the tool automatically is replaced with tool of the same group. When the time expires for all tools of the group a message is displayed when program is checked for errors if any of the tools in this group are used.

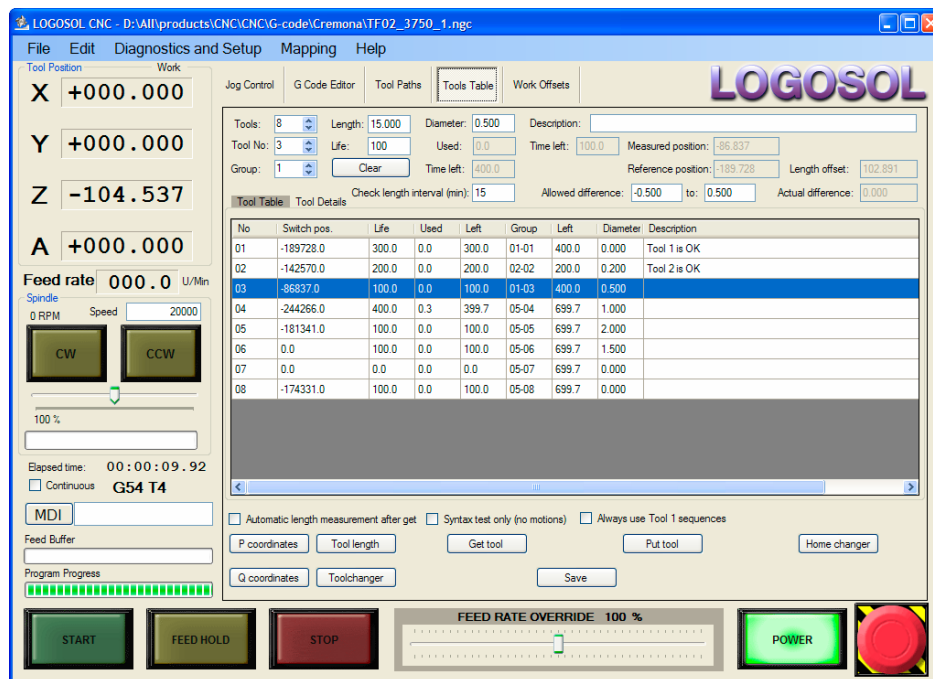
To replace the tool when **Time Left** is 0 go to **Jog Control** screen, start **Replace tool** procedure and follow the instructions on the screen. To turn off the **Tool Life** management set **Life** =0.

Length – used when the **Length Measurement Sequence** is section is empty. Not applicable if the **Length Measurement Sequence** and the proper equipment are implemented. Two length compensation modes are available:

- **Standard** option will apply the tool length compensation only to Z axis.
- **RTCP (Rotating Tool Center Point)** could be used when the tool (spindle) is attached to one of the rotary axes. According to the angle position of the tool the length difference will be transformed into vertical and horizontal axis offset.

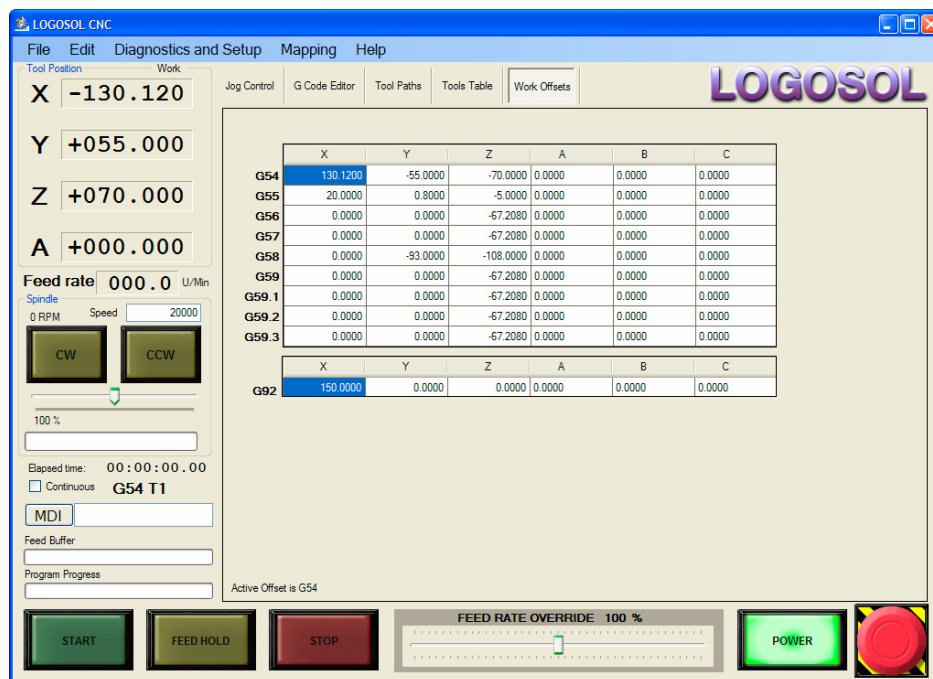
To select the proper mode contact the machine manufacturer.

All tools data could be displayed in **Tool Table** screen.



Work Offsets

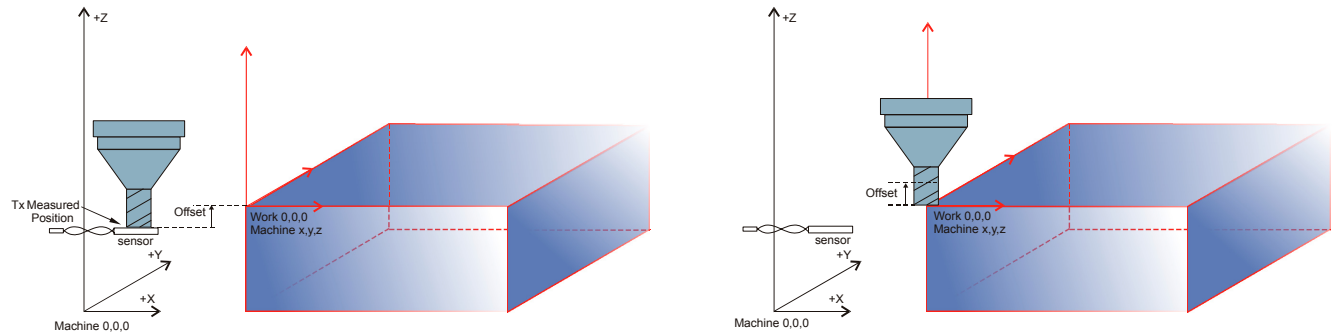
All work coordinate system offsets are displayed. Usually the active work offset is set automatically when the tool is measured. The offsets could be edited using this screen. To avoid errors consider current active tool data, reference tool data, detail dimension, detail position and machine work zone. If any changes are necessary they should be done by an experienced CNC programmer.



Appendix A. Tool Length Offset

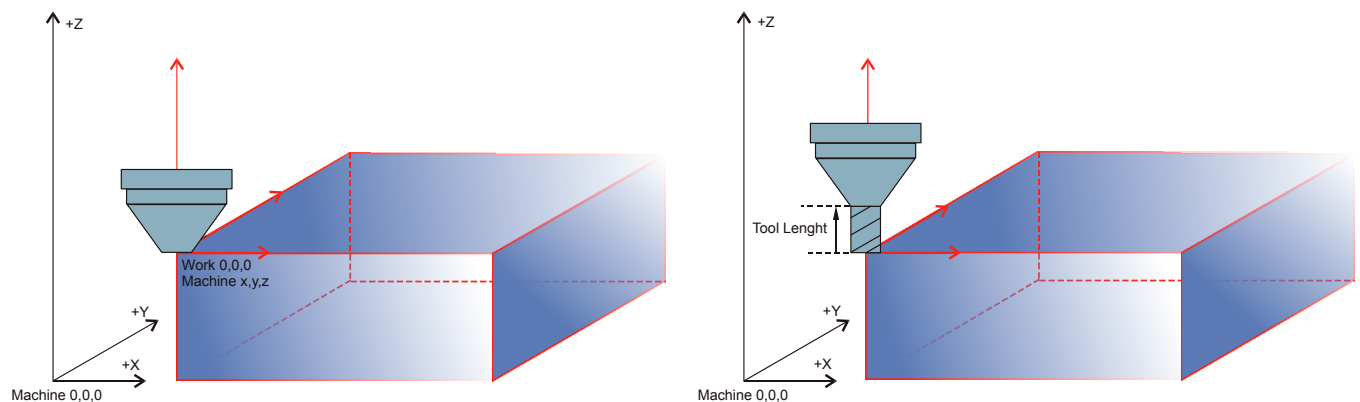
Tool Offset with Automatic Length Measurement

After setting the work coordinates system each tool is measured by touching the sensor. The measured offset is added to the current work coordinate offset (G54 to G59.3) during program execution.



Tool Offset without Automatic Length Measurement

If **Automatic Length Measurement after Get** is not checked and **Length Measurement Sequence** is empty, the data entered as tool **Length** are used by software to calculate Z offset.



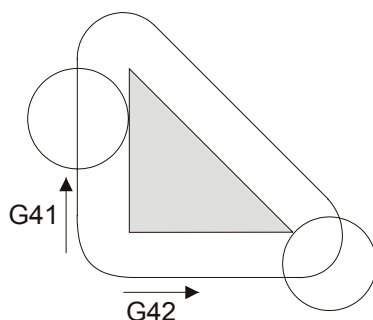
Appendix B. Cutter Radius Compensation

Introduction

The cutter radius compensation capabilities enable the programmer to specify that a cutter should travel to the right or left of an open or closed contour in the XY-plane composed of arcs and straight line segments.

Cutter radius compensation is performed only with the XY-plane active. All the figures in this appendix, therefore, show projections on the XY-plane.

When the Interpreter performs cutter radius compensation, the tool path is rounded at the corners, as shown on the figure below.



Z-axis motion may take place while the contour is being followed in the XY-plane. Portions of the contour may be skipped by retracting the Z-axis above the part, following the contour to the next point at which machining should be done, and re-extending the Z-axis. These skip motions may be performed at feed rate (G1) or at traverse rate (G0). The Z motion will not interfere with the XY path following. In actual programs, include Z-axis motion wherever you want it.

Rotational axis motions (A, B, and C axes) are allowed with cutter radius compensation, but using them would be very unusual.

Programming Instructions

Turning Cutter Radius Compensation On - To start cutter radius compensation keeping the tool to the left of the contour, program G41. To start cutter radius compensation keeping the tool to the right of the contour, program G42. In the figure above, for example, if G41 were programmed, the tool would move clockwise around the triangle, so that the tool is always to the left of the triangle when facing in the direction of travel. If G42 were programmed, the tool would stay right of the triangle and move counterclockwise around the triangle.

Turning Cutter Radius Compensation Off - To stop cutter radius compensation, program G40. It is OK to turn compensation off when it is already off.

Sequencing - If G40, G41, or G42 is programmed on the same line as tool motion, cutter compensation will be turned on or off before the motion is made. To make the motion come first, the motion must be programmed on a separate, previous line of code.

Use of D Number (G41 Dx) - Programming a D word with G41 or G42 is optional.

If a D number is programmed, it must be a non-negative integer. It represents the slot number of the tool whose radius (half the diameter given in the tool table) will be used, or it may be zero (which is not a slot number). If it is zero, the value of the radius will also be zero. Any slot in the tool table may be selected. The D number does not have to be the same as the slot number of the tool in the spindle, although it is rarely useful for it not to be.

If a D number is not programmed, the slot number of the tool in the spindle will be used as the D number.

Material Edge Contour - When the contour is the edge of the material, the outline of the edge is described in the NC program. For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

Appendix C. Supported G codes

G code	DESCRIPTION	NOTES
G0	RAPID POSITIONING	G0 X0 Y10.0 Z10.0 A0... Rapid speed is parameter from Machine Settings Y10 = Y10.0 10mm (inch))
G1	LINEAR INTERPOLATION	G1 X0 Y0 Z10 A10... F350 Move with speed 350mm/min
G2	CIRCULAR/HELICAL INTERPOLATION (CLOCKWISE)	G2 X50 Y0 I25 J0 F250 End point (X,Y or Z) Center Offset (I,J or K)
G3	CIRCULAR/HELICAL INTERPOLATION (COUNTERCLOCKWISE)	G3 X50 Y0 R50 F250 End point (X,Y or Z) Radius R
G4	DWELL	G4 P3 3 seconds delay
G10	COORDINATE SYSTEM ORIGIN SETTING	G10 L2 P1 X10 Y10 Z20... Work coordinate system defined with G54 – G59.4 is replaced with the values in G10 P1 – P9 defines which one of the G54 – G59.4 is affected
G17	XY-PLANE SELECTION	G17 G2 X12 Y16.0 I6 J8 F350
G18	XZ-PLANE SELECTION	G18 G3 X16 Z12.0 I8 K6 F350
G19	YZ-PLANE SELECTION	G19 G2 Y12 Z16 J6 K8 F350
G20	INCH SYSTEM SELECTION	Program data are not affected by this command
G21	MILLIMETER SYSTEM SELECTION	Program data are not affected by this command
G26	START RTCP TOOL LENGTH COMPENSATION	Starts RTCP compensation according to the parameters entered in Tool Length window.
G27	CANCEL RTCP TOOL LENGTH COMPENSATION	Cancel RTCP compensation.
G28	RETURN TO HOME	G28 X5 Y5 Z10 A2 Move to position (5,5,10,2) and then return to machine home position
G30	RETURN TO SECONDARY HOME	Same as G28
G40	CANCEL CUTTER RADIUS COMPENSATION	
G41	START CUTTER RADIUS COMPENSATION LEFT	Current tool radius compensation
G42	START CUTTER RADIUS COMPENSATION RIGHT	Current tool radius compensation
G43	TOOL LENGTH OFFSET START	Start tool length offset if it is canceled with G49
G49	CANCEL TOOL LENGTH OFFSET	Cancel tool length offset. The code is valid till next G43, Tool change, or program start
G53	MOTION IN MACHINE COORDINATE SYSTEM	
G54	USE PRESET WORK COORDINATE SYSTEM 1	Work Coordinate system
G55	USE PRESET WORK COORDINATE SYSTEM 2	
G56	USE PRESET WORK COORDINATE SYSTEM 3	
G57	USE PRESET WORK COORDINATE SYSTEM 4	
G58	USE PRESET WORK COORDINATE SYSTEM 5	
G59	USE PRESET WORK COORDINATE SYSTEM 6	
G59.1	USE PRESET WORK COORDINATE SYSTEM 7	
G59.2	USE PRESET WORK COORDINATE SYSTEM 8	
G59.3	USE PRESET WORK COORDINATE SYSTEM 9	
G80	CANCEL MOTION MODE (INCLUDING ANY CANNED CYCLE)	
G81	CANNED CYCLE: DRILLING	G91 G81 G98 X4 Y5 R-2 Z-4.45 L13 F450
G82	CANNED CYCLE: DRILLING WITH DWELL	G91 G82 G99 X4 Y5 R-2 Z-4.45 L13 P3 F450
G83	CANNED CYCLE: PECK DRILLING	G91 G83 G99 X4 Y5 R-2 Z-4.45 Q2 L13 F25
G85	CANNED CYCLE: BORING - NO DWELL/FEED OUT	G91 G85 G99 X4 Y5 R-2 Z-4.45 L5 F250
G86	CANNED CYCLE: BORING - SPINDLE STOP/RAPID OUT	G91 G86 G99 X4 Y5 R-2 Z-20.45 L5 P3 F150
G89	CANNED CYCLE: BORING - DWELL/FEED OUT	G91 G89 G99 X4 Y5 R-2 Z-20.45 L3 P1 F350
G90	ABSOLUTE DISTANCE MODE	
G91	INCREMENTAL DISTANCE MODE	
G92	OFFSET COORDINATE SYSTEMS AND SET PARAMETERS	Coordinate system control
G92.1	CANCEL OFFSET COORDINATE SYSTEMS AND SET PARAMETERS TO ZERO	
G92.2	CANCEL OFFSET COORDINATE SYSTEMS BUT DO NOT RESET PARAMETERS	
G92.3	APPLY PARAMETERS TO OFFSET COORDINATE SYSTEMS	
G93	INVERSE TIME FEED RATE MODE	
G94	UNITS PER MINUTE FEED RATE MODE	
G98	INITIAL LEVEL RETURN IN CANNED CYCLES	
G99	R-POINT LEVEL RETURN IN CANNED CYCLES	

The basic G code format used is: GXX X.. Y.. Z.. I.. J.. K.. P..

Data format:

Metric data format is 123.456 mm.

Inch data format is 12.3456 inch.

Delay format 123.456 sec.

*Note: Once, set the **metric** or **inch** input data format should not be changed during the machine operation. Any changes may affect the machine accuracy and safety. Consult machine manufacturer prior.*

Examples:

G0 X10 Y-10.0 Z0.33 (moves the tool at work position X=10 Y=-10 and Z=0.33)

Z.75 (moves the tool at work position Z=0.75).

G4 P4 (4 sec delay).

G4 P0.4 (0.4 sec delay).

G0 Rapid Travel

G0 moves the machine from one point to another point at the Rapid speed of the machine. G0 is generally used when cutting will not take place when moving from one location to another.

G0 is modal and will remain in effect until it is canceled by the G1, G2 or G3 codes. G0 will not cancel any feed rates used by the interpolation modes. An F word can appear on the same line with a G0 code, however, the F word will only be used when an interpolation code is used.

G1 Linear Interpolation

This code is used for linear interpolation. Linear moves can be made by one, or any combination of, all the active axes.

Example G0 and G1 :

F30. (This F word is modal).

G0 Z.1 (This line will be in rapid travel).

X1.3 Y2.7 (This line will be in rapid travel).

G1 Z-10.245 (The G1 will cancel the G0 and use the F30. from above).

X.5 (This will be at F30.0).

Z.1 G0 (This line will be in rapid travel).

G2 Circular Interpolation Arc Clockwise

G2 is used for CW circular interpolation.

Example center offset:

G2 X50 Y0 I25 J0 F250 (X and Y – end point, I and J – center offset)

Example radius:

G2 X50 Y0 R50 F250 (X and Y – end point, R – radius)

G3 Circular Interpolation Arc Counterclockwise

G3 is used for CCW circular interpolation.

Example center offset:

G3 X50 Z0 I25 J0 F250 (X and Z – end point, I and J – center offset)

Example radius:

G3 X50 YZ0 R50 F250 (X and Z – end point, R – radius)

G4 Dwell

Whenever a pause in the program is required, use the G4 code. A pause may be used to allow coolant to fully turn on after using the M7 or M8 codes.

Example:

G0 X10

EXAMPLE: G90 G0 S10000 M3 X-.45 Y-.2

Z-.3 M8

G4 P1 (This one second dwell allows the coolant to turn on completely).

X3. G1 F80.

- A P word represents time. The time is given in seconds

- P1 = 1 second

- P0.5 = 1/2 second

- P60 = 1 minute

G10 Coordinate system origin setting

To set the coordinate values for the origin of a coordinate system, program G10 L2 P ... X... Y... Z... A... B... C..., where the P number must evaluate to an integer in the range 1 to 9 (corresponding to G54 to G59.3) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

Example:

G10 L2 P1 X3.5 Y17.2

The line above sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

G17, G18, and G19 Plane selection

Circular (G2 and G3) interpolation plane selection. Program G17 to select the XY-plane, G18 to select the XZ-plane, or G19 to select the YZ-plane.

G20 and G21 - Length Units

Program G20 to use inches for length units. Program G21 to use millimeters.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units.

G26 and G27 RTCP tool length compensation mode

G26 activates RTCP (Rotating Tool Center Point) compensation if mode is enabled in Tool Length compensation setup screen.

G27 switches to standard (Z axis mode only) compensation.

Example:

G0 X0 Z30 (Z axis only compensation)

G26

G0 A15 (RTCP is on and the compensation is applied to X and Z or Y and Z depend on Tool length compensation setup)

G1 X50 Y40 F500

G0 A0

G27

G0 X15 Z25 (Z axis only compensation)

G28 and G30 Return to Home position

To return to home position by way of the programmed position, program G28 X... Y... Z... A...B... C... (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

Example: G28 X10 Y.5 Z-14 (move to X=10, Y=0.5 and Z=-14 and after that to Home)

G40 Cutter Compensation Cancel

To stop cutter radius compensation, program G40. It is OK to turn compensation off when it is already off.

This code is used to cancel cutter radius compensation.

G41 and G42 Cutter Compensation

The cutter radius compensation capabilities enable the programmer to specify that a cutter should travel to the right or left of an open or closed contour in the XY-plane composed of arcs and straight line segments.

Cutter radius compensation is performed only with the XY-plane active. All the figures in this appendix, therefore, show projections on the XY-plane.

Z-axis motion may take place while the contour is being followed in the XY-plane. Portions of the contour may be skipped by retracting the Z-axis above the part, following the contour to the next point at which machining should be done, and re-extending the Z-axis. These skip motions may be performed at feed rate (G1) or at traverse rate (G0). The Z motion will not interfere with the XY path following. In actual programs, include Z-axis motion wherever you want it.

To start cutter radius compensation keeping the tool to the left of the contour, program G41. To start cutter radius compensation keeping the tool to the right of the contour, program G42. For example, if G41 were programmed, the tool would move clockwise around the triangle, so that the tool is always to the left of the triangle when facing in the direction of travel. If G42 were programmed, the tool would stay right of the triangle and move counterclockwise around the triangle.

If G40, G41, or G42 is programmed on the same line as tool motion, cutter compensation will be turned on or off before the motion is made. To make the motion come first, the motion must be programmed on a separate, previous line of code.

G41 Dx - Programming a D word with G41 or G42 is optional.

If a D number is programmed, it must be a non-negative integer. It represents the slot number of the tool whose radius (half the diameter given in the tool table) will be used, or it may be zero (which is not a slot number). If it is zero, the value of the radius will also be zero. Any slot in the tool table may be selected. The D number does not have to be the same as the slot number of the tool in the spindle, although it is rarely useful for it not to be.

If a D number is not programmed, the slot number of the tool in the spindle will be used as the D number.

When the contour is the edge of the material, the outline of the edge is described in the NC program. For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

Example (G41):

```
G40                      (Cutter Compensation is off)
G0 X90 Y-10             (move to initial position)
G41 G1 X100 Y-5 F500    (start the Cutter Compensation)
Y10
X150 Y0
X95
G40                      (Cutter Compensation is off)
```

Example (G42):

```
G40
G0 X90 Y-5
G42 G1 X95 Y0 F500
X150
X100 Y10
Y-5
G40
```

G43 and G49 Tool Length Compensation

G43 is used to apply tool length compensation to the Z axis. By default this code is active after power up.

Example:

```
G49
G1 X100 Z20
G43 H1
G1 X25 Z20
```

G53 Move in Absolute Coordinates

For linear motion to a point expressed in absolute coordinates, program G1 G53 X... Y... Z... A... B... C... (or use G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point.

G53 can be used only when G0 or G1 is active.

G53 can not be used while cutter radius compensation is on (G41 or G42).

Example:

```
G53 G0 X145 Y24 (moves the tool to machine position X=145 and Y=24)
G1 X12 Z10.5   (moves the tool to coordinates X=12 and Z=10.5 of the active work coordinate system
selected with G54 to G59.3)
```

G54 to G59.3 Use Preset Coordinate System

To select coordinate system 1, program G54, and similarly for other coordinate systems. The coordinate system can not be changed while cutter radius compensation is on (G41 or G42).

G80 Cancel Modal Motion

G80 will cancel any motion including any canned cycle.

Canned Cycles

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the active coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times". Omitting the L word is equivalent to specifying L=1. When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and in G98 mode), or otherwise to the R position.

Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L. In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

- A straight rapid travel parallel to the XY-plane to the given XY-position;
- A straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

G81 Cycle

The G81 cycle is intended for drilling. Program G81 X... Y... Z... A... B... C... R... L.

Example 1:

Initial position X=1, Y=2 and Z=3

G90 G81 G98 X4 Y5 Z1.5 R2.8

The cycle is G81 (drilling) and will be performed once. The distance mode is absolute (G90) and initial level return (G98) is selected.

The following moves take place:

- A rapid motion parallel to the XY-plane to position X=4 and Y=5;
- A rapid motion of the Z-axis to Z=2.8;
- A feed motion of the Z-axis to Z=1.5 (drilling);
- A rapid motion of the Z-axis to Z=3.

*Example 2:**Initial position X=1, Y=2 and Z=3**G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3*

The cycle is G81 (drilling) and will be performed three times. The distance mode is relative (G91) and initial level return (G98) is selected. The cycle initial positions are X=5 (=1+4), Y=7 (=2+5), and Z=4.8 (=1.8+3). The Z position is 4.2 (=4.8-0.6).

The preliminary motion is a rapid travel along the Z-axis to Z=4.8.

The first repeat consists of 3 moves:

- A rapid motion parallel to the XY-plane to X=5 and Y=7;
- A feed motion of the Z-axis to Z=4.2 (drilling);
- A rapid motion to the Z-axis to Z=4.8.

The second repeat initial position are: X=9 (=5+4) and Y=12 (=7+5).

The second repeat consists of 3 moves:

- A rapid motion parallel to the XY-plane to X=9 and Y=12;
- A feed motion of the Z-axis to Z=4.2 (drilling);
- A rapid motion to the Z-axis to Z=4.8.

The third repeat initial positions are: X=13 (=9+4) and Y=17 (=12+5).

The third repeat consists of 3 moves:

- A rapid motion parallel to the XY-plane to X=13 and Z=17
- A feed motion of the Z-axis to Z=4.2 (drilling);
- A rapid motion to the Z-axis to Z=4.8.

G82 Cycle

The G82 cycle is intended for drilling. Program G82 X... Y... Z... A... B... C... R... L... P...

G82 is similar to G81 and has the same motions sequences. The difference is the delay P seconds after feed motion of Z-axis (drilling)

*Example:**G90 G82 G98 X4 Y5 Z1.5 R2.8 P0.5 (0.5sec delay at Z=1.5)***G83 Cycle**

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program G83 X... Y... Z... A... B... C... R... L... Q... The X and Y-axes behavior is the same as in G81. The Z-axis motion is divided on Q steps. After each step Z-axis moves to its initial position with rapid travel.

*Example:**G91 G83 G98 X4 Y1 R-2 Z-40 Q5*

In the example above the drilling distance is divided into 5 steps until the final Z-axis position is reached. After each step Z-axis returns to its initial position with rapid speed and then moves to the end Z-axis position of the previous step.

G85 Cycle

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling.

Program G85 X... Y... Z... A... B... C... R... L... Preliminary motion is same as described in G81. All other Z-axis motions are at feed rate speed.

G86 Cycle

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program G86 X... Y... Z... A... B... C... R... L... P... Preliminary motion is same as described in G81.

The Z-axis motions in this cycle are:

- Move the Z-axis only at the current feed rate to the Z position;
- Dwell for the P number of seconds;
- Stop the spindle turning;
- Retract the Z-axis at traverse rate to initial Z;
- Restart the spindle in the direction it was going;

The spindle must be turning before this cycle is used.

G89 Cycle

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. Program G89 X... Y... Z... A... B... C... R... L... P... Preliminary motion is same as described in G81.

The Z-axis motions in this cycle are

- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Retract the Z-axis at the current feed rate to initial Z.

G90 and G91 Absolute and incremental distance mode

To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system. To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent increments from the current values of the numbers. I, J and K numbers always represent increments, regardless of the distance mode setting.

G92, G92.1, G92.2, G92.3 - Coordinate System Offsets

G92 will make the current point have the coordinates you want (without motion), program G92 X... Y... Z... A... B... C..., where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. When G92 is executed, the origin of the currently active coordinate system moves. The offset programmed with G92 is stored in Work Offsets table.

G92.1 cancels the offset and clears the G92 offset data in Work Offsets table.

G92.2 cancels the offset but will keep the data in Work Offsets table.

G92.3 activates the offset and applies the offset data stored in Work Offsets table.

Example:

Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3. The axis offsets are always used when motion is specified in absolute distance mode using any of the nine coordinate systems (those designated by G54 - G59.3). Thus all nine coordinate systems are affected by G92. Being in incremental distance mode has no effect on the action of G92. Non-zero offsets may already be in effect when the G92 is called. If this is the case, the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by $[[7-9] + -3]$. You can set axis offsets in one program and use the same offsets in another program.

G93 and G94 Set Feed Rate Mode

Two feed rate modes are recognized: units per minute and inverse time. Program G94 to start the units per minute mode. Program G93 to start the inverse time mode.

In **Units per Minute Feedrate** mode, an F word is interpreted to mean the controlled point should be moved at a certain number of units per minute. The units could be millimeters per minute, inches per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

In **Inverse Time Feedrate** mode, an F word means the move should be completed in (one divided by the F number) minutes. For example, if the F number is 2.0, the move should be completed in half a minute. When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2, or G3 motion, and an F word on a line that does not have G1, G2, or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions.

G98 and G99 - Set Canned Cycle Return Level

When the spindle retracts during canned cycles, there is a choice of how far it retracts:

- retract perpendicular to the selected plane to the position indicated by the R word, or;
- retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

To use the first option program G99. To second option program G98. Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

Appendix D. Supported M codes

M0	PROGRAM STOP
M1	OPTIONAL PROGRAM STOP
M2	PROGRAM END
M3	TURN SPINDLE CLOCKWISE
M4	TURN SPINDLE COUNTERCLOCKWISE
M5	STOP SPINDLE TURNING
M6	TOOL CHANGE
M7	MIST COOLANT ON
M8	FLOOD COOLANT ON
M9	MIST AND FLOOD COOLANT OFF
M30	PROGRAM END AND RESET
M32	M32 Px USE MACHINE PARAMETERS SET NUBER X
M64	M64 Px - SETS OUTPUT x (ON)
M65	M64 Px - CLEARS OUTPUT x (OFF)

M0, M1, M2, M30 Program Stopping

To stop a running program temporarily (regardless of the setting of the optional stop switch), program M0. To stop a running program temporarily (but only if the optional stop switch is on), program M1. It is OK to program M0 and M1 in MDI mode, but the effect will probably not be noticeable, because normal behavior in MDI mode is to stop after each line of input, anyway. If a program is stopped by an M0 or M1 pressing the cycle start button will restart the program at the following line.

To end a program, program M2 or M30. Both of these commands have the following effects:

- Axis offsets are set to zero (like G92.2) and origin offsets are set to the default (like G54).
- Selected plane is set to XY (like G17).
- Distance mode is set to ABSOLUTE (like G90).
- Feed rate mode is set to UNITS PER MINUTE (like G94).
- Cutter compensation is turned off (like G40).
- The spindle is stopped (like M5).
- The current motion mode is set to G1.
- Coolant is turned off (like M9).

Pressing cycle start will start the program back at the beginning of the file.

M3, M4, M5 Spindle Control

To start the spindle turning clockwise at the currently programmed speed, program M3. To start the spindle turning counterclockwise at the currently programmed speed, program M4. To stop the spindle from turning, program M5.

It is OK to use M3 or M4 if the spindle speed is set to zero. If this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle will start turning. It is OK to use M3 or M4 when the spindle is already turning or to use M5 when the spindle is already stopped.

M6 Tool Change

To change a tool in the spindle from the tool currently in the spindle to the tool most recently selected (using a T word), program M6 Tx. The tool change may include axis motion while it is in progress. Refer to *Logosol CNC setup guide* and machine manufacturer manuals for detailed information.

M7, M8, M9 Coolant Control

To turn mist coolant on, program M7.

To turn flood coolant on, program M8.

To turn all coolant off, program M9.

It is always OK to use any of these commands, regardless of the current coolant state.

M32 Use Machine Parameter Set

To use the specified parameter set program M32 Px. Where x is number (0 to 99) and represents the number of the machine parameter set.

M64 and M65 Set / Clear output

To set the output (ON) program M64 Px. Where x is output number.

To clear the output (OFF) program M65 Px. Where x output number.

The number must be valid output. Refer to *Logosol CNC setup guide* and machine manufacturer manuals for detailed information.