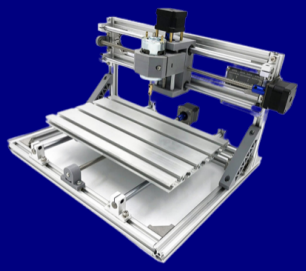
USER MANUAL

Guide to Operating the 3018 CNC with Ultrasonic Transducer

(English)

Instructions for use
Required commands
Software installation



Biomedical Engineering Research Laboratory

University of Costa Rica

2023

LIIB

Laboratorio de
Investigación en
Ingeniería Biomédica

Contents

1 Introduction

This guide aims to explain how to operate a CNC machine model 3018 to control the position of an ultrasonic transducer. It will provide instructions on the installation process of the required software, the programming language used by the machine, the use of a graphical interface, and general instructions for using the system.

This guide consists of several sections covering different aspects of this process:

1. **Software Installation:** Before getting started, you will learn how to install MATLAB and the necessary drivers for the CNC 3018. We will provide you with step-by-step instructions to ensure a successful setup.
2. **Creating MATLAB Code:** You will learn how to write MATLAB code that facilitates communication between the CNC 3018 and the ultrasonic transducer. We will explain key functions and commands required for programming.
3. **Graphical User Interface (GUI):** We will introduce you to the GUI we have developed in MATLAB to simplify operation. This interface allows you to control the machine intuitively, even if you have no prior programming experience.
4. **Usage Instructions:** Here, you will find a detailed description of how to use the CNC 3018 machine adapted for the ultrasonic transducer. We cover everything from starting to finishing a typical project.
5. **Troubleshooting:** If you encounter issues during operation, this section provides solutions to common problems you may face.

We are confident that this guide will equip you with the necessary tools to make the most of this setup and efficiently carry out your projects. Let's get started!

2 Software Installation

2.1 MATLAB Installation

Obtain a MATLAB License

MATLAB is commercial software, so you will need a valid license to install and use it. You can obtain a MATLAB license through the official MathWorks website.

System Requirements

Before installation, make sure your system meets the minimum system requirements for MATLAB. This includes your system's capacity, operating system, RAM, disk space, etc. Check the specific requirements for the version of MATLAB you want to install on the MathWorks website.

Download MATLAB

Log in to your MathWorks account and go to the MATLAB download page. Select the MATLAB version you want to download (e.g., MATLAB R2023a) and choose your operating system (Windows, macOS, or Linux).

You will be provided with an installation file. Download it to your computer.

Run the Installer

Once the download is complete, run the MATLAB installation file you downloaded. This will initiate the installation process.

Follow the Installation Instructions

The MATLAB installer will guide you through the installation process. Be sure to follow the detailed instructions provided by the installer.

During installation, you will be prompted to enter the MATLAB license key you obtained in Step 1.

You will also have the option to select which MATLAB components you want to install. If you're unsure, the default configuration is usually suitable for most users.

License Activation

Once the installation is complete, you will need to activate your MATLAB license. Follow the provided instructions to activate MATLAB online.

Launch MATLAB

After activating the license, you can launch MATLAB from the Start menu (on Windows) or from the terminal (on macOS or Linux). MATLAB will provide a graphical user interface (GUI) where you can begin your work.

That's it! You should now have MATLAB installed and ready for use on your system.

3 Creating MATLAB Code

A G-Code file is a text file with the extension .gcode, used by 3D printers or CNC machines, as is the case here. This type of file contains the instructions that the machine will follow to produce its movement in specified directions.

When opening the MATLAB file of this CNC control system, you will have the option to execute movements on the machine, carried out automatically through this language. Alternatively, you can directly input commands in the G-Code language. In the appendices, you can find the most useful and commonly used commands.

Examples of G Code Usage

G00: Rapid Positioning

Example: `G00 X100 Y20 Z5`

- **G00** is the G code for rapid positioning, which moves the tool quickly to a location without cutting material.
- **X100** indicates rapid movement along the X-axis to position 100 units.
- **Y20** indicates rapid movement along the Y-axis to position 20 units.
- **Z5** indicates rapid movement upward or downward along the Z-axis to position 5 units.

This code is used to move the tool quickly to the specified location before performing any cutting operation. It is useful for efficient travel between points without cutting material.

G01: Linear Interpolation

Example: `G01 X50 Y30`

- **G01** is the G code for linear interpolation, which moves the tool in a straight path from its current position to the new position.
- **X50** indicates linear movement along the X-axis to position 50 units.
- **Y30** indicates linear movement along the Y-axis to position 30 units.

This code is used to cut material along a linear path. The tool will move in a straight line from its current position to the specified new position.

G02: Clockwise Circular Interpolation

Example: `G02 X60 Y40 I10 J0`

- **G02** is the G code for clockwise circular interpolation, which moves the tool in a circular path clockwise.

- **X60** indicates the X-coordinate of the center of the circle.
- **Y40** indicates the Y-coordinate of the center of the circle.
- **I10** is the distance in the X-axis from the current position to the center of the circle.
- **J0** is the distance in the Y-axis from the current position to the center of the circle.

This code is used to perform tool movements in a clockwise circular path around the specified center.

G03: Counterclockwise Circular Interpolation

Example: `G03 X80 Y60 I10 J0`

- **G03** is the G code for counterclockwise circular interpolation, which moves the tool in a counterclockwise circular path.
- **X80** indicates the X-coordinate of the center of the circle.
- **Y60** indicates the Y-coordinate of the center of the circle.
- **I10** is the distance in the X-axis from the current position to the center of the circle.
- **J0** is the distance in the Y-axis from the current position to the center of the circle.

This code is used to perform tool movements in a counterclockwise circular path around the specified center.

4 Using the Graphical User Interface

Once MATLAB is installed and open, you should open the file *Interfaz_CNC.m*. This file contains the script in which the graphical interface was developed to control the machine's position. The next step is to press the RUN button in the EDITOR tab.

The first step is to choose the language; in this case, click on *Spanish*. This will open the main menu, as shown in Figure 1. In this menu, the box on the left allows you to view the G-code that will be executed on the machine. This code will be created based on the selected operation or by the user. On the right side, buttons are presented for the available options to control the machine.

Below is a description of the buttons available in the main menu.

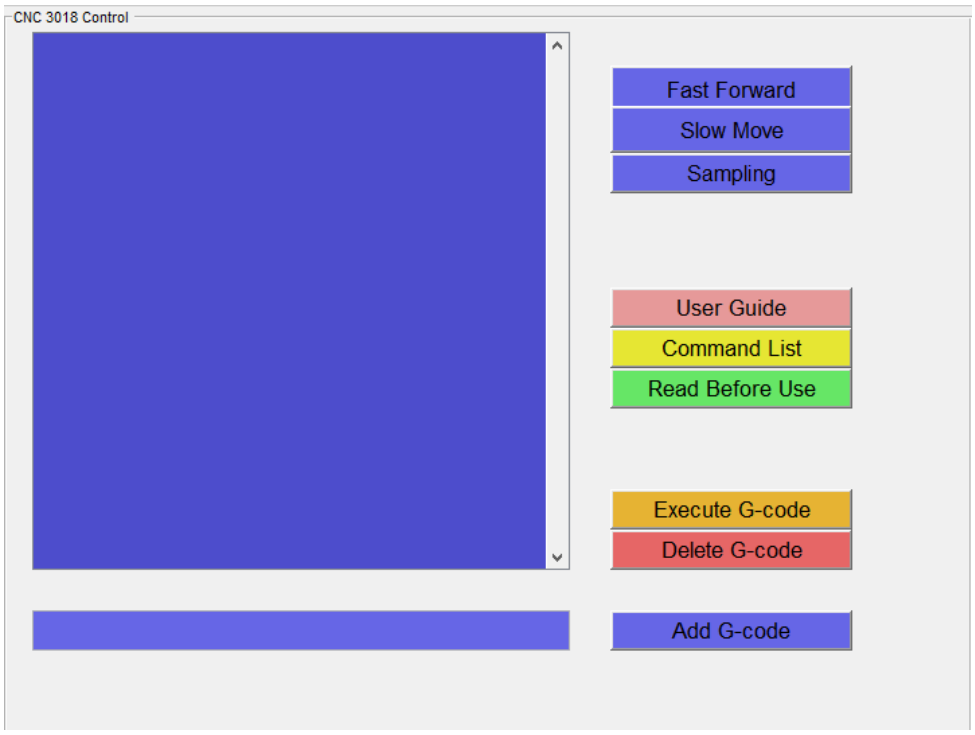


Figure 1: View of the main menu of the grapgical interface.

4.1 Fast Movement and Slow Movement

These correspond to the two options for automatic movement, whose buttons are located at the top right of the menu and are colored blue. When one of these two is selected, the interface shown in Figure 2 will open. Here, four directional buttons can be seen, where the vertical ones correspond to movement along the Y-axis and the horizontal ones correspond to movement along the X-axis.

To change the machine's pointer position, you must specify the number of steps to move in the *Distance* box and indicate how long you want the movement to be executed in the *Time* box. The distance should be given in millimeters. Finally, click on one of the directional buttons, depending on the required movement.

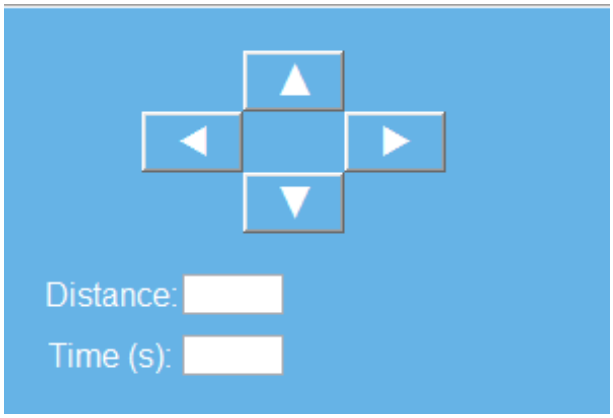


Figure 2: View of the fast/slow movement menu.

4.2 Sampling

Just below the movement buttons, you will find the *Sampling* button, which will open the menu shown in Figure 3. Here, it is only necessary to enter the required sampling length in millimeters and the waiting time before starting the sampling.

From the current position, the machine will move the transducer half of the travel distance in the negative direction, and there the sampling will begin, followed by a return to the initial position. The X-axis will be the most commonly used in this case, due to the configuration and operation of the transducer, although there is also the possibility of performing this function along the Y-axis.

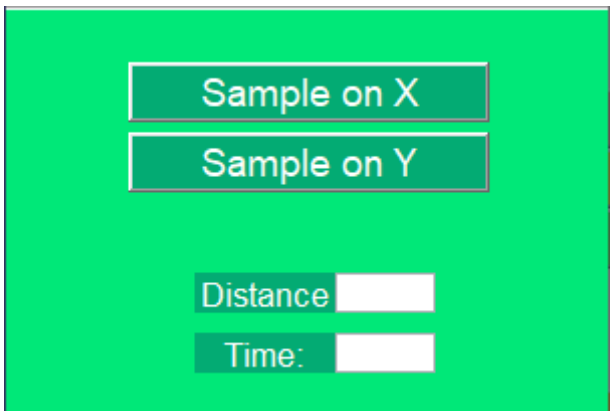


Figure 3: View of the sampling menu.

4.3 Run G-code

Once the required movements are selected, click the *Run G-code* button, which will open the original CNC application. Wait for the ”Ready” indication to appear in the top right, then click ”send” to start the movement.

4.4 Add G-code

If desired, it is also possible to explicitly enter a code in the programming language that the machine accepts by clicking on the bottom left bar. Once the required code is entered, press the *Add G-code* button, and the command will appear in the command panel.

4.5 Delete G-code

In case of an error or a change in the selection of commands or required movement, you can select the corresponding command from the command panel and click the *Delete G-code* button.

5 General Usage Instructions

To use this system, follow these steps:

1. Place the required transducer in the holder and secure it using the two velcro straps.
2. Connect the CNC machine to the computer using a USB cable. This cable will be connected to one of the ports on the circuit board at the back of the machine. The port name on the machine must match the name of the original machine application, called *.
3. Once the machine is connected, it should be turned on by lowering the switch shown in Figure 4. A red LED light will indicate that the machine is powered on.

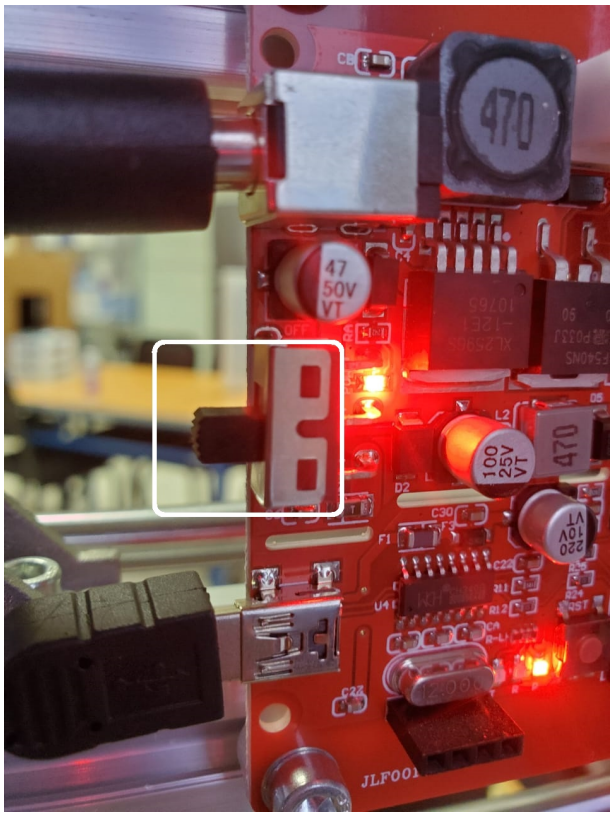


Figure 4: CNC machine switch

4. Now, the next step is to open the file *Interfaz_CNC.m* in MATLAB and run it to use the graphical interface created and explained in the previous section. This will allow you to control the position of the transducer and perform various functions.

6 Troubleshooting

Maximum Displacements

Although the MATLAB code takes into account the displacement limits of the machine and prevents the execution of functions that do not respect these limits, there may be cases where this prevention is insufficient. If the machine's position exceeds its possible ranges, press the *Pause* button on the original CNC machine application.

Origin Calibration

Through the use of the graphical interface, the machine's position will always return to the initial point if the code $G28$ is entered at the end of each movement. This point is referred to as the geometric center of the three axes, and the movement restrictions were generated from this point. However, if this point is lost, you can calibrate the origin.

This can be achieved by moving the machine's position to the start of each axis and executing the code $G00\ X242\ Y135\ Z35$.

Software

It is important to be careful not to alter the code of the .m file that controls the CNC machine and contains the graphical interface for simple use. A small change could result in syntax or compilation errors.

Appendice

G Codes

G00 Rapid positioning

G01 Linear interpolation (feed move)

G02 Clockwise circular interpolation

G03 Counterclockwise circular interpolation

G04 Dwell (wait for spindle)

G09 Exact stop

G10 Set program zero

G11 Cancel program zero

G12 Fixed cycle CW (turning)

G13 Fixed cycle CCW (turning)

G15 Switch to XY plane

G16 Switch to ZX plane

G17 Select XY plane

G18 Select ZX plane

G19 Select YZ plane

G20 Input in inches

G21 Input in millimeters

G22 Check machine zero or reference position (home)

G23 Go to machine reference position (home)

G27 Return to reference point

G28 Return to secondary reference point

G29 Skip function

G30 Thread cutting

G31 Cornering circular interpolation

G32 Cancel cutter compensation

G33 Rigid tapping

G34 Left-hand threading

G35 Right-hand threading

G36 Multiple threading left

G37 Multiple threading right

G38 Cancel cornering offset

G39 Cancel cutter length offset

G40 Cutter compensation off, left

G41 Cutter compensation left

G42 Cutter compensation right

G43 Tool length offset +

G44 Tool length offset -

G49 Cancel tool length offset

G50 Cancel scale factor

G51 Scale factor

G52 Local coordinates

G53 Machine coordinates

G54 Work coordinate system 1

G55 Work coordinate system 2

G56 Work coordinate system 3

G57 Work coordinate system 4

G58 Work coordinate system 5

G59 Work coordinate system 6

G60 Single direction positioning

G61 Exact stop mode

G62 Automatic corner override mode

G63 Thread cutting mode

G64 Cutting mode

G65 Call macro

G66 Wait for signal

G67 Wait for signal cancellation

G68 Coordinate rotation

G69 Cancel coordinate rotation

G70 Rapid feed in drilling cycle

G71 Cutting feed in drilling cycle

G72 Facing cycle in X-axis

G73 Facing cycle in Z-axis

G74 Grooving cycle

G80 Cancel cycle

G81 Simple drilling cycle

G82 Drilling cycle with dwell at the bottom

G83 Peck drilling cycle

G84 Tapping cycle

G85 Boring cycle, feed-out

G86 Boring cycle, spindle stop

G87 Boring cycle, backboring

G88 Boring cycle, feed-in

G89 Boring cycle, dwell

G90 Absolute coordinates

G91 Incremental coordinates

G92 Set position

G94 Feedrate per minute (F in mm/min)

G95 Feedrate per revolution (F in rev/min)

G98 Return to initial level

G99 Return to R plane

The G codes are commands used to define motion and operation functions. Below, you will find a list of common G codes and their functions.

M Codes

M00 Program stop

M01 Optional program stop

M02 Program end

M03 Spindle on, clockwise

M04 Spindle on, counterclockwise

M05 Spindle stop

M06 Automatic tool change

M07 Coolant Mist On

M08 Coolant Flood On

M09 Coolant Off

M10 Pallet clamp (in)

M11 Pallet clamp (out)

M13 Spindle on, clockwise with coolant on

M14 Spindle on, counterclockwise with coolant on

M15 Turn spindle on clockwise and wait

M16 Rotate spindle to orient

M19 Tool change with pallet change

M20 Tool change with no pallet change

M21 Tool change with drawbar check

M22 Tool change without drawbar check

M23 Unclamp the pallet and start a move to pallet change position

M24 Unclamp the pallet and start a move to pallet change position

M27 Selects program to run

M28 Start spindle and coolant

M29 Stop spindle and coolant

M30 End of program

M31 Wait for input signal

M37 Release the tool or pallet clamp

M38 Tool or pallet clamp (on)

M39 Tool or pallet clamp (off)

M40 Turn the spindle and coolant off

M41 Turn on spindle load monitor

M43 Turn on output 1

M44 Turn on output 2

M45 Turn off output 1

M48 Cancel M48

M49 Turn off output 2

M50 Wait for output 1 to turn on

M51 Wait for output 2 to turn on

M52 Move robot to home position

M53 Mirror image in X

M54 Mirror image in Y

M56 Mirror image in IV

M57 Wait for output 1 to turn off

M62 Wait for output 2 to turn off

M63 Mirror image in X off

M64 Mirror image in Y off

M65 Mirror image in IV off

M66 Call subroutine

M67 End of subroutine

M68 Mirror image in X on

M69 Mirror image in X off

M70 Wait for input signal 1 to turn on

M71 Wait for input signal 2 to turn on

M73 Turn off mirror image in X

M76 Call subroutine

M77 End of subroutine

M80 Mirror image in X on

M81 Mirror image in X off

M83 Mirror image in Y on

M98 Wait for input signal 1 to turn on

M99 Wait for input signal 2 to turn on

The M codes are commands used to control auxiliary functions of the CNC machine.