

NX SERIES MACHINE PROGRAM INSTRUCTION USER MANUAL

New, functional and meet with Improved NOKTA CNC Interface. Marble, iron and metal so on. Manipulate 3D products is very easy, increase your speed with Easy Computer Software Interface and Macros Library. To simplify the use of many common components are collected in a single screen, MDI (Manual Mode Command Input) and so on. a single window access to the Command Reference to get a macro automatic control, etc. Progress and cutting speed control. Enjoy the benefits of having a lot of features in hosting NOKTA CNC interface. must be one step ahead of your competitors is speed and practicality, you'll find all of them this interface.

In case of trying to stop running and step etc with the ability to continue the work correctly. Get ready to meet a very important aspect of hosting NOKTA CNC interface.

The screenshot displays the NOKTA CNC interface with the following components:

- Program Editor:** A list of G-code commands from N1 to N16. N1 is highlighted in green. The commands include G90, G17, G40, G80, G49, G21, G64, P0.1, Q0.0, G0, Z10, M3, S18000, G04, P3, G0, X492.0353, Y310.0003, Z10, G1, Z-5.0000, F500, G2, X377.0353, Y195.0003, I-115.0000, J0.0, G2, X262.0353, Y310.0003, I0.0000, J115.0000, G2, X377.0353, Y425.0003, I115.0000, J-0.0, G2, X492.0353, Y310.0003, I-0.0000, J-115.0000, G1, Z-10.0000, F500, G2, X377.0353, Y195.0003, I-115.0000, J0.0, G2, X262.0353, Y310.0003, I0.0000, J115.0000, G2, X377.0353, Y425.0003, I115.0000, J-0.0, G2, X492.0353, Y310.0003, I-0.0000, J-115.0000, G1, Z-15.0000, F500.
- MDI:** A text input field for manual command entry.
- Navigation:** Buttons for Ana Sayfa, Dosya, Ayarlar, Offsetler, Eklentiler, Prob, and Durum.
- Control Panels:**
 - PROGRAMI ÇALIŞTIR:** Buttons for ADIMLA, SPINDLE MOTOR, DURAKLAT, DUR, REFERANS AL (checked), and RESET.
 - CALISMA OFFSETI:** Fields for X, Y, Z, and A offsets, all set to 0.000. DTG values are -0.000, 0.000, 0.000, and 0.000. REF buttons are available for X, Y, Z, and A.
 - ADIM:** Step size selection buttons for .010, .100, 1.00, and 10.0. A progress bar shows 2% completion. JOG CONT STEP button is present.
 - SOGUTMA:** SOGUTMA button, G28-HOME, and a field set to 0.0. HIZ MM label.
 - VAKUM:** VAKUM button, Z-PROB (with a downward arrow), and a field set to 0. DEVIR RPM label.
 - TAKIM BOYU:** T0 button, TAKIM BOYU field set to 0.000, and a directional pad.
 - FREZE - CNC:** FREZE - CNC VER. 6.00 label.
 - VEL:** Velocity control buttons for HIZ (100%), RPM (100%), and VEL (100%) with corresponding progress bars.
- 2D Diagram:** A technical drawing of a part with dimensions: 920.0 (height), 925.0 (width), -5.0 (depth), 1960.0 (total width), and 1955.0 (width of the main body).
- Footer:** DURUM: G54 G90 G21 G80 G40 G94 G97 G99 G64. NOKTA CNC logo with the tagline "Akıllı Makineler Üretilir." and a KAPAT button.

Technical Features and Innovations:

- Workspace Identification Hardwar and Software Limit
- Spindle Motor Forward-Reverse Tapping Cycle Support
- User Defined Work Offset (Workspaces) & Tool Dimension and Lenght Support
- Auto Tool Changer (ATC) Automatic Tool Change Support (This Function is a Optional.)
- Program were followed by Start-Stop and Tool Change Macro
- Transfer File ower Network or USB Load G-Code.
- So I left work to continue from the desired line (with Pause-Feed Hold and Program Run Button)
- JOG Mode selection and stepping strategy (Flight Mode: G64 and Stop Mode: G61 to make the switch automatically.)
- Cooling Fluid Support (M7-M8-M9 Command Support)
- Spindle Motor Speed Control RPM Line bar (increase and decrease reduction while working.)
- The G01 and G00 while cutting speed and load increase or decrease the walking speed control and many more innovations on line bar.

Turning the Machine On and Ready to Operate:

- 1 - Green Lamp located on the CNC Control Cabinet "Power Ok" by pressing the button to open the "Power On" Control Unit.
 - 2 - Points after opening the CNC interface screen, Flashing the "RESET" button to remove the counter from the reset alarm. Located on to energize and drive the machine and all axes will be locked in their current positions.
 - 3 - The first to be done Reference All of reference is to send the order. For this; "REF X" button is pressed before and after waiting to take the reference of the X axis, "REF Y" button and the "REF Z", "REF A" as well as with all axis are sent by reference. Sent to the reference axis, reference is received successfully will notify users that it is ready to work with Green LED warning light.
- Warning! All axes can not be reference without installing the G-code is sent to the referral. Reference to get the first rule.
- In -Keyboard right, and left arrow keys for X and Y axes of the "Page Up" and "Page Down" buttons it is possible to manually control the Z-axis.
- Just In time "MDI" command line can be checked by entering the G-code CNC system. (Ex. "M3 S12000" to run the spindle motor and speed to 12,000 rev / min. Will be appointed.)

Program G-Code Load & Run:

- 1 - "Dosya" file menu screen by pressing the button toggles the program Load Screen
 - 2 - G-Code Flash Memory installed plugs into the USB slot. Located in the middle of the display window USB Memory brand, model or name will appear. USB Memory entering into G-code to be loaded program file is selected, then the "USB-COPY To" G-Code, which is selected by pressing the button control unit Permanent memory CNC PC Hard Disk (among other files in the left window) is installed.
 - 3 - From the window in the left (control unit disk) overlying selected G-code which will be called file name double click or "G-CODE DOWNLOAD" button is pressed. In this way, which will be operated the program will be called to run.
 - 4 - Called up to operate the G-Code "Ana Menu" main menu screen in "PROGRAMI ÇALIŞTIR" program start button is pressed.
- Warning! Use CNC machine tools by following the operating instructions. (Using Your Machine Operation and Maintenance reading the instructions.)



COMPUTER SOFTWARE INTERFACE BUTTON CONTROL FUNCTIONS:

PROGRAMI ÇALIŞTIR: Run the Loaded G-Code Program File

DUR: Operation stops being the G-Code programs and takes the lead. (Used for permanent stoppages.)

REFERANS AL: All Axis Reference (Home) to the position in the order. (Sequence A-Z-Y-X d.) Reference (Home Sensors - Switches bench It must be connected. At least two axes at the start of each job (X and Y and Z and of the possible axes) must be taken of reference. Power outages, stop or pause with the STOP program, Extreme breaking, manual stop, etc. Emergency Emergency Stop. stance in The least accurate of the three axes in order to continue the work place thick (X-Y-Z's) job can be restarted by sending the reference point. Software reference purchase or cancellation of the respective axis "Settings" interface in the "Home Sensors Off" tab marked counter reference sensor will be revoked. But still it can not run the program from receiving referral. Only Axes References to References will be taken before it is sent. Mandatory state unless the circuit to get this setting.

ADIMLA: The current installed G-Code program is used for the line-line execution move. Stepping mode is activated, "PROGRAMI ÇALIŞTIR" button With the installed executable program line by line. To run each line of succession "PROGRAMI ÇALIŞTIR" button is pressed repeatedly.

Flight Mode: Velocity (Speed Mode) and Stop (Episodic Mode) will switch between modes. (See. G61, G64 etc. G64 P0.1 Q0.1) a corner bench in flight mode, which will be exposed to deviations in the rotation radius. This shift value F (Feed Rate) is linked. Precision bearings and so on. programs in the lathe machine with G61 mode will be performed in the molding process.
NOTE! Flight Mode or stopping the program file must be selected before the execution or G-Code must be specified in the initial header line.

Starting a program from a specific line: Main Screen installed on the G-Code 's hovering with the mouse right click on the desired line in the menu "Başlangıç Satırı Yap" will be marked by selecting the relevant line as the starting line. Then "PROGRAMI ÇALIŞTIR" button to start the program.

NOTE! 3D or in a long line of program files, this process can take some time, the program will run after the wait.

STATUS: CNC Software under the window in this field indicates warning or error message contains user-defined information.

Jog Mode: Machine JOG (Manual) control mode when the keyboard arrow keys, control or by handwheel provides continuous or incremental fashion transition. In continuous mode, while axis can move uncontrollably, and move in incremental mode with step increments until the entered value is provided.

* Step Mode is activated stepping speed (the speed axis) STEP changed with the multiplier value.

* While progress is made in continuous mode while holding down the SHIFT key pace-FEEDRATE G00 will be 100% temporarily.

SPINDLE MOTOR: This Button is Open and Close Spindle Motor. MDI command line parameters S Spindle with Speed Control. This process is entirely possible to do with the MDI command line. (See. M3 S24000)

STEP by STEP Load G-Code and Run CNC Router

1.Step: Click the "RESET" Button

2. Step: Click the "REFERANS AL" Button

3. Step: Load The G-Code File on the "DOSYA" File Menu

4. Step: Select on the cnc start point over keyboard **Left, Right, Page Up or Page Down keys (Note! Page-UP key is Z Axis + , Page Down = Z- move)**

5. Step: Click the "Zero-X, Zero-Y, Zero-Z, Zero-A" Buttons (All coordinates to be stored zero point to start job.)

6. Step: Click the "PROGRAMI ÇALIŞTIR" button Run the opened CNC Program.

-After Finishing all moves and stop the program, clear powder and push new part to cutting.

Note! The graphic display can be turned off by writing the command per G-Code (**AXIS, hide**) to close the Graph view screen and speed up the loading of the file with >100.000 Rows and above in files with too many high G-Codes.