ISBN: 978-93-90655-25-4

DESIGN AND IMPLEMENTATION OF CNC MACHINE FOR PRINTED CIRCUIT BOARD

Mohammed S. Salman 1,
Tahreer Mahmood 2,
Zuhair Sameen Shuker 3

Published by
Novateur Publication
466, Sadashiv Peth, M.S.India-411030

Design and Implementation of CNC Machine for Printed Circuit Board

Authored By

Mohammed S. Salman[1]

Tahreer Mahmood[2]

Zuhair Sameen Shuker[3]

[1,2]Department of Electronic Engineering
College of Engineering
University of Diyala
Iraq

[3]Department of Renewable energy
Baqubah Technical Institute
Middle Technical University
Iraq

[1]E-mail: mohselman@uodiyala.edu.iq

[2]E-mail: tahreer_mahmood_eng@uodiyala.edu.iq (Corresponding author)

[3] Email: zuhairalsagar@mtu.edu.iq

Acknowledgments

First of all, our deepest gratitude goes to ALLAH, the Compassionate, the Merciful for everything. We would like to express our sincere gratitude and thanks to research group for the continuous support of this work and study. their patience and guidance helped us in all my time of research and the writing of this book. It was a great privilege to work and study under their guidance. Our sincere thanks also go to all those helped and supported me during this work. Without their support it would not be possible to conduct this research book.

We extremely grateful to our mothers for her prayers, love and sacrifices. We very much thankful to our wife's and sons for their love, understanding, prayers and continuing support to complete this work. Also, we express thanks to our friends for their support, without them we could have never completed this book. We express our special thanks to the department of electronic engineering university of Diyala and department of renewable energy Middle Technical university for their honest support throughout this research work.

Last but not the least, we would especially like to thank our country for their continuous support, represented by our sponsors, the universities of Diyala & Middle Technical.

Abstract

Printed Circuit Board (PCB) Mill is a device that etches out a pattern on a copper clad board such that it makes a PCB. PCBs are used everywhere in the field of electrical engineering to connect electrical components to one another. This research book presents the design and implementation of low-cost mini-PCB commuter numerical control machine based on microcontroller. A stepper motors were used to compile the movement of the three axes coordinates. Moreover, the generated Gerber file was analyzed using MATLAB platform where the analyzed file is converted to C-code and transmitted to the milling and drilling machine through the serial communication port using RS232 protocol. In addition, a path planning technique based on sorting drilling algorithm was also proposed. The simulated and implemented results were coincided .

Contents

Chapter One

Introduction

Chapter Two

Material and Methodology

Chapter Three

Construction of CNC

Chapter Four

A First Look at EAGLE

Chapter Five

Conclusion and Suggestions for Future Work

Subject Page No.

Chapter One

Introduction

1.1 General Introduction

A printed circuit board (PCB) can be defined as a mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate. Components (e.g., capacitors, resistors or active devices) are generally soldered on the PCB. Advanced PCBs may contain components embedded in the substrate. PCBs can be single sided (one copper layer), double sided (two copper layers) or multi-layer (outer and inner layers). Conductors on different layers are connected with vias. Multi-layer PCBs allow for much higher component density. PCBs may be classified in many different ways according to their various attributes. One fundamental structure common to all of them is that they must provide electrical conductor paths that interconnect components to be mounted on them. There are two basic ways to form these conductors:

i. Subtractive: where the unwanted portion of the copper foil on the base substrate is etched away, leaving the desired conductor pattern in place.
ii. Additive: where the formation of the conductor pattern is accomplished by adding copper to a bare (no copper foil) substrate in the pattern and places desired. This can be done by plating copper, screening conductive paste, or laying down insulating wire onto the substrate on the predetermined conductor paths [1].

Computer Numeric Control (CNC) is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium (computer command module, usually located on the device) as opposed to controlled manually by hand wheels or levers, or mechanically automated by cams alone. Most NC today is computer (or computerized) numerical control (CNC),in which computers play an integral part of the control. In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs [2].

CNC Machine is a generally used in the manufacturing sector that involves the use of computers to control machine tools. Tools that can be controlled in this manner include milling, and drilling. The CNC stands for Computer Numerical Control. From this CNC technology, the biggest change in the world of digital electronics & Micro-controller, we are presenting here an idea of CNC bit plotter using ARDUINO UNO [3].

1.2 What is the CNC?

CNC machining is a manufacturing process in which pre-programmed computer software dictates the movement of factory tools and machinery. The process can be used to control a range of complex machinery, from grinders and lathes to mills and routers. With CNC machining, three-dimensional cutting tasks can be accomplished in a single set of prompts [1].

Short for “computer numerical control,” the CNC process runs in contrast to — and thereby supersedes — the limitations of manual control, where live operators are needed to prompt and guide the commands of machining tools via levers, buttons and wheels. To the onlooker, a CNC system might resemble a regular set of computer components, but the software programs and consoles employed in CNC machining distinguish it from all other forms of computation [2].

When a CNC system is activated, the desired cuts are programmed into the software and dictated to corresponding tools and machinery, which carry out the dimensional tasks as specified, much like a robot.

In CNC programming, the code generator within the numerical system will often assume mechanisms are flawless, despite the possibility of errors, which is greater whenever a CNC machine is directed to cut in more than one direction simultaneously. The placement of a tool in a numerical control system is outlined by a series of inputs known as the part program.

With a numerical control machine, programs are inputted via punch cards. By contrast, the programs for CNC machines are fed to computers though small keyboards. CNC programming is retained in a computer’s memory. The code itself is written and edited by programmers. Therefore, CNC systems offer far more expansive computational capacity. Best of all, CNC systems are by no means static, since newer prompts can be added to pre-existing programs through revised code the figure below shows actual design CNC [1].

Fig.1: Actual design of CNC

1.3 Historical Developments in CNC Machines

Computer numerical control is a modern concept in the manufacturing and production industries. However, the concept of CNC harkens back to the basic idea of NC, or numerical control [3].

In 1947, the U.S. Air Force found that the complex designs and shapes of aircraft parts such as helicopter rotor blades and missile components were causing problems for manufacturers, who could not keep up to projected production schedules. At that time, John Parsons, of the Parsons Corporation, of Traverse City, Michigan, began experimenting with the idea of making a machine tool generate a “thru-axis curve" by using numerical data to control the machine tool motions. In 1949, the U.S. Air Material Command awarded Parsons a contract to develop NC and in turn speed up production methods. Parsons subcontracted this study to the servomechanism laboratory of the Massachusetts Institute of Technology (MIT), which in 1952 successfully demonstrated a vertical spindle Cincinnati Hydrotel, which made parts through simultaneous three-axis cutting tool movements. In a very short period of time, almost all machine tool manufacturers were producing machines with NC [1].

1.4 Organization of research

This book consists of five chapter , which can be briefly presented as:

Chapter One:

This chapter presents a general introduction and literature review for the work.

Chapter two:

In this chapter we will talk about the electrical and electronic parts made in advance with regard to their parts and almost everything related to them.

Chapter three:

In this chapter we will talk about how to build the mechanical parts and how to install the shutters and how to move the mechanical parts.

Chapter four:

In this chapter we will talk about the programming of electronic parts and what programs are used to manage the machine and where to get it and what results we obtained from the work of the machine.

Chapter five:

In this chapter we will talk about conclusion and future work.

Chapter Two

Material and Methodology

2.1 Introduction

This chapter shows how to build and do experimental the project, method of this project is generally a guiding principle to handle the problem. The project implementation method is discussed briefly focusing on basic components. The framework must be clear to ensure that the project runs smoothly, and project objectives are capable of success. Figure 2.1 shows three subsystems of this CNC machine; Mechanical system design, electronics system, and computer for software tools.

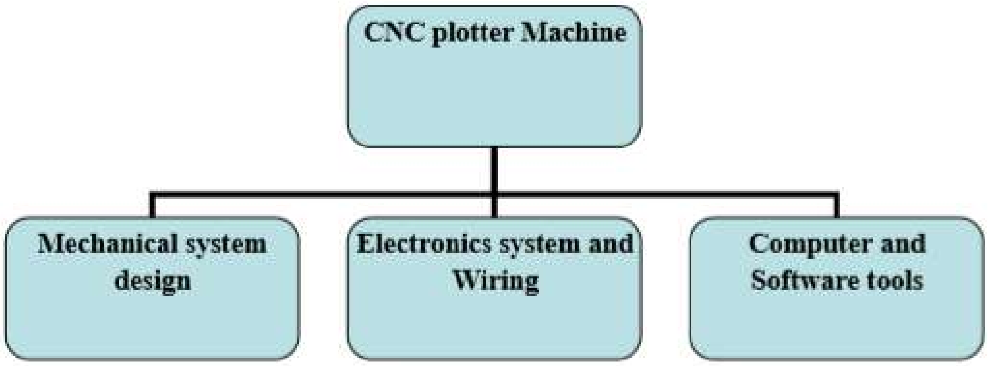


Figure 2.1 subsystem of CNC Machine.

2.2 Electronics System and Wiring

This section will discuss the electronic parts and wiring which is needed for designing and building our CNC machine. So, the main electronic component required are one card CNC, , stepper motors, D.C. power supply, pen holder, some wires and USB to serial adapter [5].

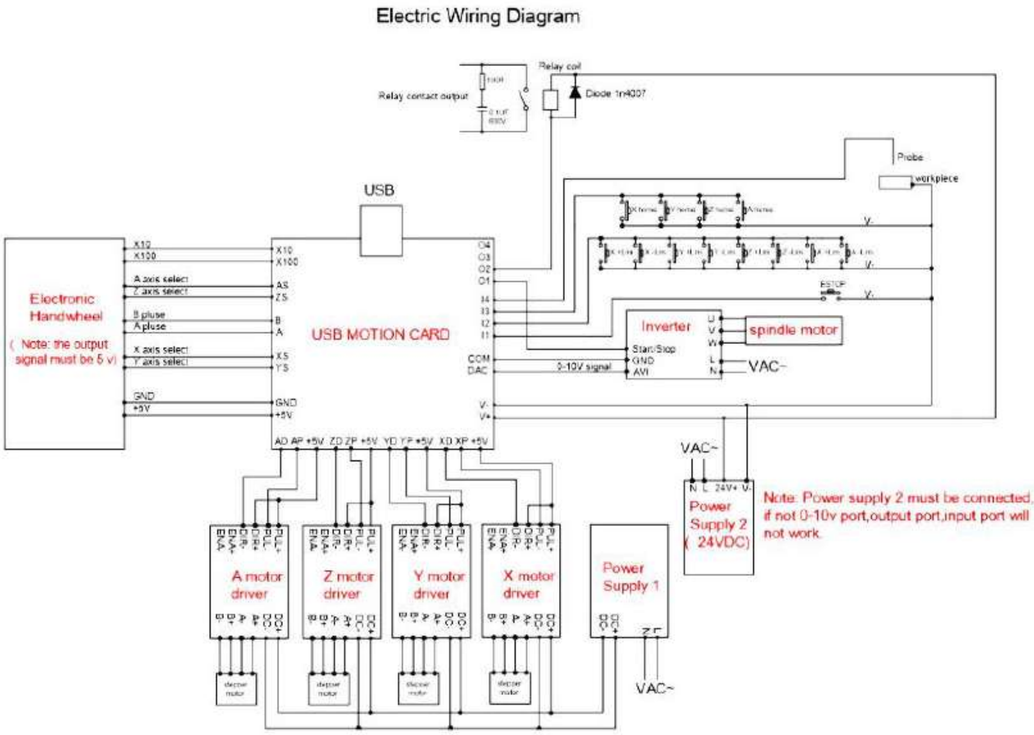


Figure 2.2 Electric wiring diagram

2.2.1 Card CNC

1- **Description**

4-Axis 100KHz USB CNC Controller Card for Mach3 lets you use any Windows Computer to be used as CNC Machine Controller. Previously with LPT Port Based Cards, user were limited to use only older computers which had LPT Port.

With this New CNC Control Board you can even run Mach3 on a Windows Tablet which can give more compact and professional look to your CNC Machine.

2- Technical Specifications

- No of Axis: 4 (you can connect four stepper motor drives or servo drives)

- Maximum step-pulse frequency: 100KHz (suitable for smooth servo or stepper motion)
- Automatic probe tool Support: Yes
- Emergency input: Yes
- Limit switches: Yes (4 general-purpose inputs, you can connect the limit switch, estop switch, probe switch, back to zero)
- Electronic handwheel Support: Yes
- Optical Isolation between USB Side Control and External inputs: External Voltage: 12V/24V
- Spindle Control Output: Yes (0-10V output port)
- Isolated Outputs: 4 general-purpose isolated relay drive output interface (can drive four relays for controlling the spindle starts, forward rotating and reverse rotating, pumps and other device)
- Status LED: 1 Status LED (indicate connection status of the controller board)
- Size: 8×7.7cm /3.15×3.03inch
- Tested with following Operating Systems:
  - Windows XP (direct)
  - Windows 7 64bit /32Bit (direct)
  - Windows 10 (Indirect using Windows XP on VmWare)

3- Setup Instructions

- **Step 1:** Install Mach3: If you already have a Mach3 Installed, skip this step.
- **Step 2:** Download the USB Motion Driver RnRMotion.dll from the link down below and copy and paste the file into your Mach3 Plugins folder Example: “C:/Users/ProgramFiles/Mach3/Plugins
- **Step 3:** Make sure you board is connected to the PC via USB cable. It should automatically recognize the USB device.
- **Step 4:** Open Mach3 and you should be prompted to select a driver. Select the “RnRMotionController” and check “Don’t ask me again”. This will allow your board to communicate with Mach3.
- **Step 5:** Download the XML Mach3 Config File above by right clicking on the link and selecting “save as”. Save the Mach3Mill.xml file in your Mach3 root folder[6].

Figure 2.3 (A) Card CNC

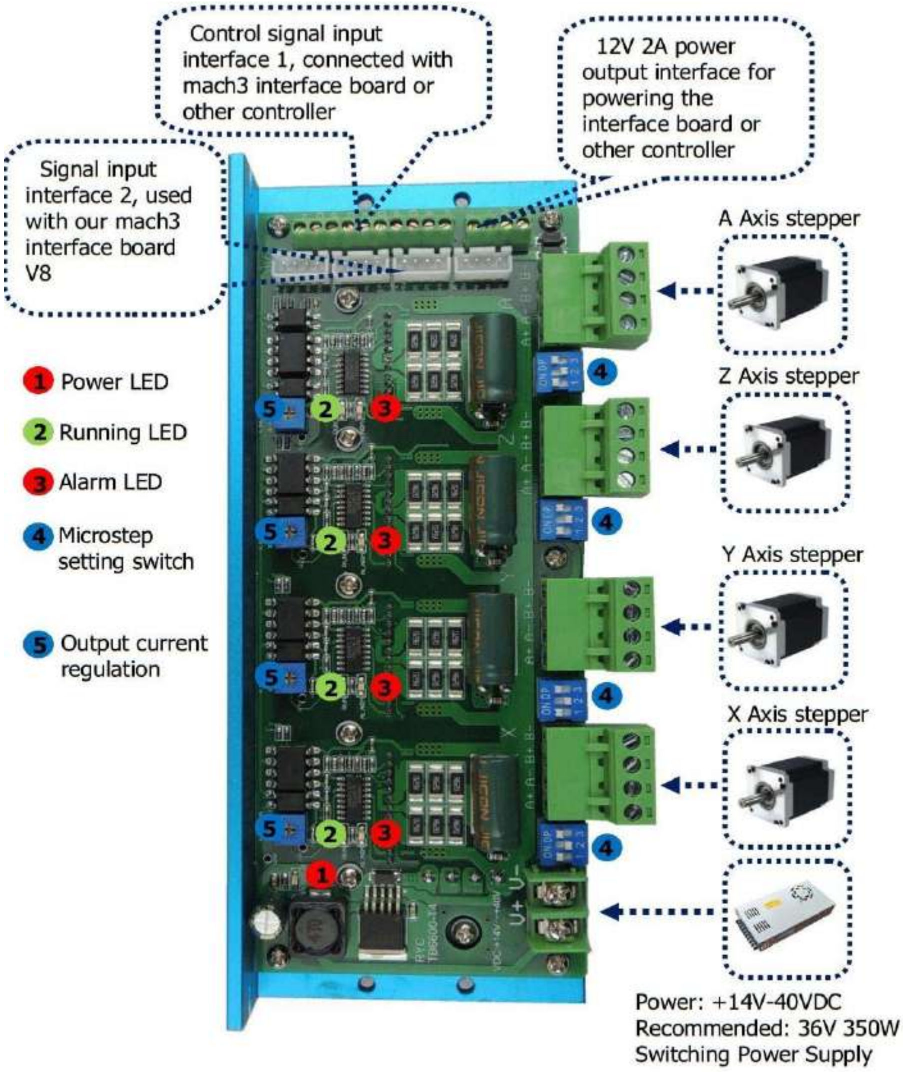


Figure 2.3 (B) Card CNC

2.2.2 Stepper motor

1- What is a Stepper Motor

It is a brushless electromechanical device which converts the train of electric pulses applied at their excitation windings into precisely defined step-by-step mechanical shaft rotation . The shaft of the motor rotates through a fixed angle for each discrete pulse. This rotation can be linear or angular. It gets one step movement for a single pulse input. When a train of pulses is applied, it gets turned through a certain angle. The angle through which the stepper motor shaft turns for each pulse is referred as the step angle, which is generally expressed in degrees.[7]

Figure 2.4 stepper motor

The number of input pulses given to the motor decides the step angle and hence the position of motor shaft is controlled by controlling the number of pulses.

This unique feature makes the stepper motor to be well suitable for open-loop control system wherein the precise position of the shaft is maintained with exact number of pulses without using a feedback sensor. If the step angle is smaller, the greater will be the number of steps per revolutions and higher will be the accuracy of the position obtained. The step angles can be as large as 90 degrees and as small as 0.72 degrees, however, the commonly used step angles are 1.8 degrees, 2.5 degrees, 7.5 degrees and 15 degrees.[7].

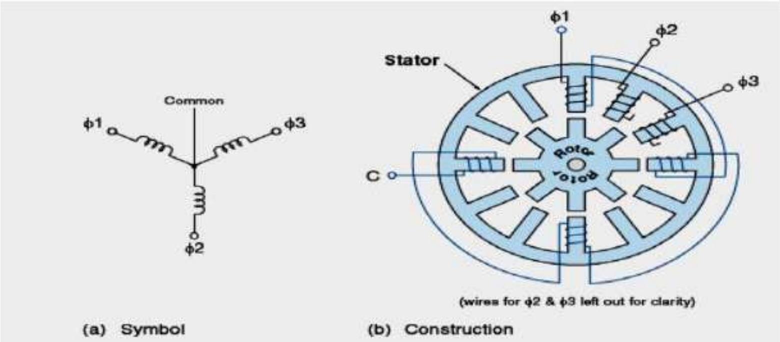


Figure 2.5 Schematic of a stepper motor

The main formula expressing the behavior of the motor is given by equation 1.

$$U = RI + K_T\,\omega\,(t) + L\,\frac{dI}{dt} \qquad (2.1)$$

Were

• RI = Voltage to drive current (resistance times current)

• kT ω(t) = Voltage to compensate the back-EMF1 (kT is the torque constant and ω the speed)

• LdI/dt = Voltage to establish/modify current level

The torque is directly proportional to the current and can be expressed by equation 2.2.

$$M = kT.\ I \qquad (2.2)$$

As a consequence, the current can be deducted from the previous formula as shown in equation 3.

$$I = \frac{U - KT\,\omega\,(t) - L\frac{dI}{dt}}{R} \quad \Longrightarrow \quad I = \frac{U}{R}\,(1\text{-}e^{-}\,\frac{R}{L}) - \frac{KT\,\omega\,(t)}{R} \qquad (2.3)$$

The direction of the shaft rotation depends on the sequence of pulses applied to the stator. The speed of the shaft or the average motor speed is directly proportional to the frequency (the rate of input pulses) of input pulses being applied at excitation windings. Therefore, if the frequency is low, the stepper motor rotates in steps and for high frequency, it continuously rotates like a DC motor due to inertia.

Like all electric motors , it has stator and rotor. The rotor is the movable part which has no windings, brushes and a commutator. Usually, the rotors are either variable reluctance or permanent magnet kind. The stator is often constructed with multipole and multiphase windings, usually of three or four phase windings wound for a required number of poles decided by desired angular displacement per input pulse. Unlike other motors it operates on a programmed discrete control pulses that are applied to the stator windings via an electronic drive. The rotation occurs due to the magnetic interaction between poles of sequentially energized stator winding and poles of the rotor.[8]

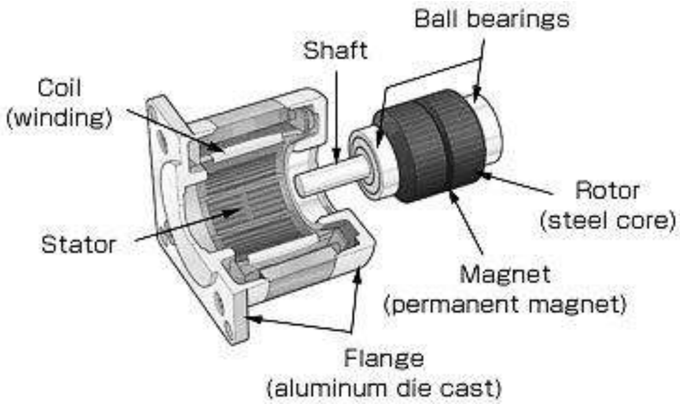


Figure 2.6 construction of stepper motor

There are several types of stepper motors are available in today's market over a wide range of sizes, step count, constructions, wiring, gearing, and other electrical characteristics. As these motors are capable to operate in discrete nature, these are well suitable to interface with digital control devices like computers.

Due to the precise control of speed, rotation, direction, and angular position, these are of particular interest in industrial process control systems, CNC machines, robotics, manufacturing automation systems, and instrumentation [8].

2- Types of Steppers Motors

There are three basic categories of stepper motors, namely permanent magnet stepper motor , variable reluctance stepper motor and hybrid stepper motor . In all these motors excitation windings are employed in stator where the number of windings refer to the number of phases. A DC voltage is applied as an excitation to the coils of windings and each winding terminal is connected to the source through a solid-state switch. Depends on the type of stepper motor, its rotor design is constructed such as soft steel rotor with salient poles, cylindrical permanent magnet rotor and permanent magnet with soft steel teeth. Let us discuss these types in detail [7].

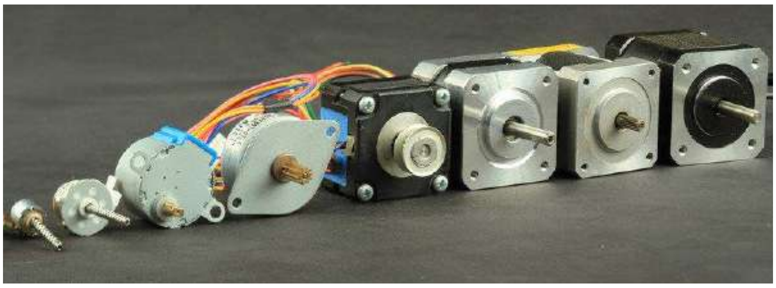


Figure 2.7 types of steppers

A- HY-DIV268N-5A two phase hybrid stepper motor drive

HY-DIV268N-5A subdivision-type two-phase hybrid stepping motor drive using DC 12 ~ 48V power supply, suitable for drive Two-phase hybrid stepping motor dynamic voltage 12 to 48V, the current is less than 5A outer diameter of 35 to 86 mm. This drive using the drive's current loop subdivision control, the motor torque ripple is very small, low-speed running is very smooth, almost no vibration and noise. High-speed torque is much higher than other two-phase drive, high positioning accuracy. Widely used in the engraving machine, CNC machine tools, packaging machinery and other high resolution requirements on the device. The main features 1 average current control, two-phase sinusoidal current drive output 2 DC 12 ~ 48V power supply, the internal integration of 12V and 5V regulator 3 optically isolated signal input / output 4, overvoltage, undervoltage, overcurrent, and white short-circuit protection 5 4 file segmentation and automatic half-streaming capabilities up to 16 segments 4 file output phase current settings 7 high starting speed 8 high-speed torque 一, the electrical parameters Input voltage DC 12 ~ 48V input The input current of 1 to 5 amps, select the drive a stepper motor. Output current of 0.2A ~ 5A Temperature Operating Temperature -10 to 45 ℃; Storage temperature -40 ℃ to 70 ℃ Humidity not condensing, not drops Gas prohibit combustible gas and conductive dust Weight 200 grams.

Control signal interface Figure 1 is a wiring schematic of the drive 1, the definition of control signals PUL +: step pulse signal is input side or the positive pulse signal input positive terminal PUL-: the negative input of the negative input pulse signal or a positive pulse signal DIR +: stepping direction signal input to the positive terminal or negative pulse signal input to the positive terminal DIR -: stepping direction signal input of the negative side or reverse step pulse signal input negative terminal EN +: offline can reset signal input side is EN-: offline can reset signal input negative terminal Offline enable signal is active, reset drive failure to prohibit any pulse, the output of the drive Power component is turned off, the motor holding torque. 2、Control signal connections PC control signal can be high, also can be low effective. When active high, the control signal The negative side together as a signal to active low, positive side of all control signals together as a signal common. For example, open-collector and PNP output interface circuit diagram is as follows:

细分	1	2	3	电流	4	5	6
NG	ON	ON	ON	0.2A	ON	ON	ON
1	OFF	ON	ON	0.6A	OFF	ON	ON
1/2	ON	OFF	ON	1.2A	ON	OFF	ON
1/2	OFF	OFF	ON	1.8A	OFF	OFF	ON
1/4	ON	ON	OFF	2.5A	ON	ON	OFF
1/8	OFF	ON	OFF	3.3A	OFF	ON	OFF
1/16	ON	OFF	OFF	4.2A	ON	OFF	OFF
NG	OFF	OFF	OFF	5A	OFF	OFF	OFF

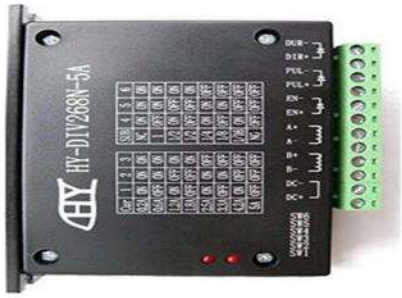


Fig. 2.8 HY-DIV268N-5A two phase hybrid stepper motor

B- Variable Reluctance Stepper Motor

It is the basic type of stepper motor that has been in existence for a long time and it ensures easiest way to understand principle of operation from a structural point of view . As the name suggests , the angular position of the rotor depends on the reluctance of the magnetic circuit formed between the stator poles (teeth) and rotor teeth [8].

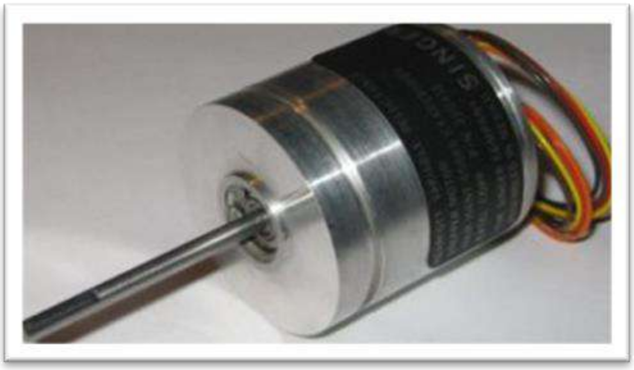


Figure 2.9 Variable Reluctance Stepper Motor

1- Construction of Variable Reluctance Stepper Motor:

It consists of a wound stator and a soft iron multi-tooth rotor . The stator has a stack of silicon steel laminations on which stator windings are wound . Usually , it is wound for three phases which are distributed between the pole pairs . The number of poles on stator thus formed is equal to an even multiple of the number of phases for which windings are wounded on stator . In the figure below , the stator has 12 equally spaced projecting poles where each pole is wound with an exciting coil.

These three phases are energized from of a DC source with the help of solid state switches . The rotor carries no windings and is of salient pole type made entirely of slotted steel laminations . The rotor pole's projected teeth have the same width as that of stator teeth . The number of poles on stator differs to that of rotor poles, which provides the ability to self start and bidirectional rotation of the motor . The relation of rotor poles in terms of stator poles for a three phase stepper motor is given as , Nr = Ns ± (Ns / q). Here Ns = 12 , and q= 3 , and hence Nr = 12 ± (12 / 3) = 16 or 8. An 8-pole construction rotor without any excitation is illustrated below [6].

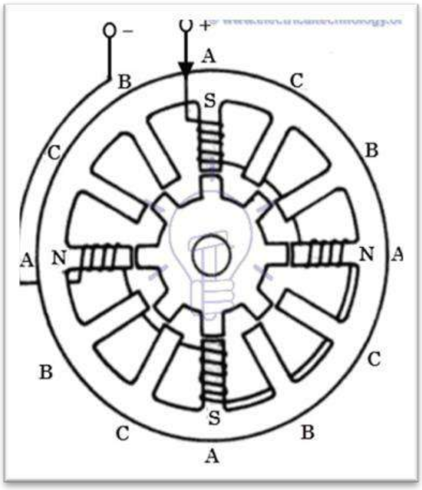


Figure 2.10 Construction of Variable Reluctance Stepper Motor

2- Working Variable Reluctance Stepper Motor:

The *stepper motor works on the principle* that the rotor aligns in a particular position with the teeth of the excitation pole in a magnetic circuit wherein minimum reluctance path exist. Whenever power is applied to the motor and by exciting a particular winding , it produces its (magnetic field) and develops its own magnetic poles. Due to the residual magnetism in the rotor magnet poles , it will cause the rotor to move in such a position so as to achieve minimum reluctance position and hence one set of poles of rotor aligns with the energized set of poles of the stator . At this position, the axis of the stator magnetic field matches with the axis passing through any two magnetic poles of the rotor . When the rotor aligns with stator poles , it has enough magnetic force to hold the shaft from moving to the next position , either in clockwise or counter clockwise direction . Consider the schematic diagram of a 3-phase , 6 stator poles and 4 rotor teeth is shown in figure below . When the phase A-A' is supplied with a DC supply by closing the switch -1, the winding become a magnet which results one tooth become North and other South . So the stator magnetic axis lies along these poles .

Due to the force of attraction, stator coil North Pole attracts nearest rotor tooth of opposite polarity , i.e., South and South Pole attract nearest rotor tooth of opposite polarity , i.e., North. The rotor then adjusts to its minimum reluctance position where the rotor magnetic axis exactly matches with stator magnetic axis.[6]

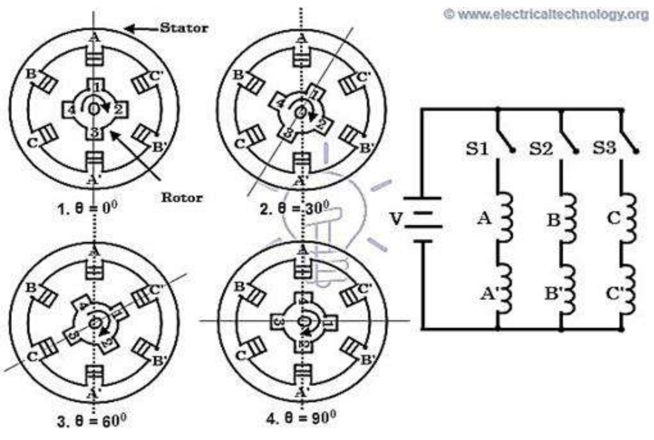


Figure 2.11 rotor magnetic and stator magnetic of stepper

When the phase B-B’ is energized by closing switch -2 keeping phase A-A’ remain de-energized by opening switch-1 , winding B-B’ will produce the magnetic flux and hence the stator magnetic axis shifts along the poles thus formed by it. Hence the rotor shifts to the least reluctance with magnetized stator teeth and rotates through an angle of 30 degrees in the clockwise direction . When the switch-3 is energized after opening switch-2, the phase C-C’ is energized , the rotor teeth align with new position by moving through an additional angle of 30 degrees . By this way , the rotor moves clockwise or counterclockwise direction by successively exciting stator windings in a particular sequence . The step angle of this 3-phase 4-pole rotor teeth stepper motor is expressed as , 360/(4 × 3) = 30 degrees(as step angle = 360/ Nr × q) . [8].

The step angle can be further reduced by increasing the number of poles on the stator and rotor , in such case motors are often wound with additional phase windings .

This can also be achieved by a adopting different construction of stepper motors such as multistack arrangement and reduction gear mechanism . [8].

C- Permanent Magnet Stepper Motor

The permanent magnet design motor is perhaps the most common among several types of stepper motors . As the name implies , it adds permanent magnets to the motor construction . This type of stepper motors is also referred as *can-stack motor* or *tin-can motor*. The main advantage of this motor is its low manufacturing cost . This type of motor has 48-24 steps per revolution.[6]

Figure 2.12 permanent magnet stepper motor

1- Construction Permanent Magnet Stepper Motor

In this motor, the stator is of multipolar and its construction is similar to that of variable reluctance stepper motor as discussed above. It consists of slotted periphery on which stator coils are wound. It has projected poles on the slotted structure where the wound windings can be two or three or four-phase.The end terminals of all these windings are bought out and connected to the DC excitation via solid state switches in the drive circuit. The rotor is made up of a permanent magnet material like a ferrite that can be in the shape of either cylindrical or salient pole, but usually it is of smooth cylindrical type. The rotor designed to have an even number of permanent magnetic poles with alternate North and South polarities.[7].

2- Working of Permanent Magnet Stepper Motor

The operation of this motor works on the principle that unlike poles attract each other and like poles repel each other. When the stator windings are excited with a DC supply, it produces (magnetic flux) and establishes the North and South poles. Due to the force of attraction and repulsion between permanent magnet rotor poles and stator poles, the rotor starts moving up to the position for which pulses are given to the stator. Consider a 2-phase stepper motor with two permanent magnetic rotor poles as shown in the figure below When the phase A is energized with a positive with respect to the A’, the windings establish North and South poles. Due to the force of attraction, the rotor poles align with stator poles such that the magnetic pole axis of rotor adjusts with that of stator as shown in figure .When the excitation is switched to B phase and switching off phase A, the rotor further adjusts to magnetic axis of phase B, and thus rotates through 90 degrees in clockwise direction .Next, if the phase A is energized with a negative current with respect to A’, the formation of stator poles causes the rotor to move through another 90 degrees in clockwise direction .In the same way, if the phase B is excited with negative current by closing phase A switch, the rotor rotates through another 90 degrees in the same direction. Next, if the phase A is excited with positive current, the rotor comes to the original position thus making a 360 degrees complete revolution. This implies that, whenever the stator is excited, the rotor tends to rotate through 90 degrees in clockwise direction. The step angle of this 2-phase 2-pole permanent magnet rotor motor is expressed as, $360/(2 \times 2) = 90$ degrees. The step size can be reduced by energizing two phases simultaneously or a sequence of 1-phase ON and 2-phase ON modes with a proper polarity.[8].

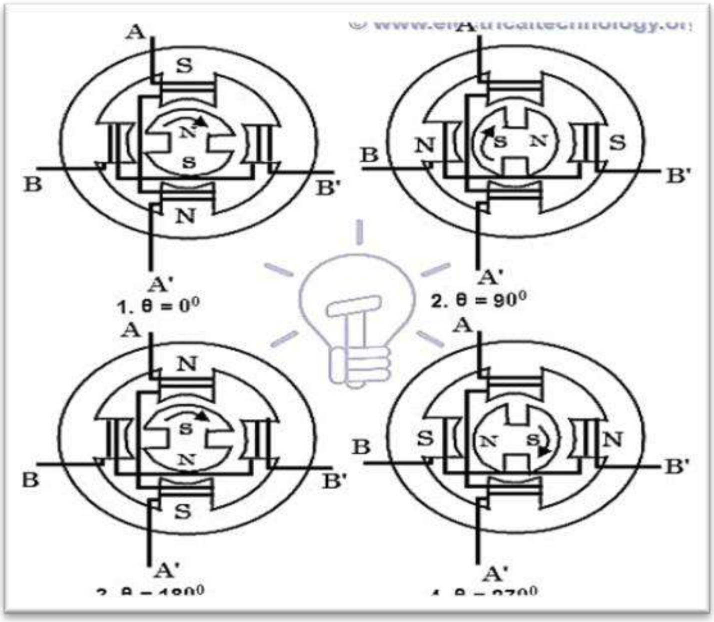


Figure 2.13 Working of Permanent Magnet Stepper Motor

D- Hybrid Stepper Motor

It is the most popular type of stepper motor as it provides better performance than permanent magnet rotor in terms of step resolution, holding torque and speed. However, these motors are more expensive than PM stepper motors. It combines the best features of both variable reluctance and permanent magnet stepper motors. These motors are used in applications that require very small stepping angle such as 1.5, 1.8 and 2.5 degrees.[7].

Figure 2.14 Hybrid Stepper Motor

1- Construction of Hybrid Stepper Motor

The stator of this motor is same as its permanent magnet or reluctance type counterpart. The stator coils are wound on alternate poles. In this, the coils of different phases are wound on each pole, usually two coils at a pole which is referred as a bifilar connection. The rotor consists of a permanent magnet which is magnetized in axial direction to create a pair of magnetic poles (N and S poles). Each pole is covered with uniformly spaced teeth. The teeth are made up of soft steel and two section, of which on each pole are misaligned each other by a half-tooth pitch.[8].

2- Working of Hybrid Stepper Motor:

This motor works similar to that of permanent magnet stepper motor. The figure below shows 2-phase, 4-pole, 6-tooth rotor hybrid stepper motor. When the phase A-A’ is excited with a DC supply, keeping B-B’ unexcited, the rotor aligns such that the south pole of the rotor faces north pole of the stator while north pole of rotor faces south pole of the stator Now, if the phase B-B’ is excited, keeping A-A’ switched off in such a way that upper pole becomes north and lower becomes south, then the rotor will align to a new position by moving through counterclockwise direction. If the phase B-B’ is oppositely excited such that the upper pole becomes south and lower becomes north, then the rotor will turn clockwise direction. By a proper sequence of pulses to the stator, the motor will turn in desired direction. For every excitation, rotor will get locked into new position, and even if excitation is removed motor still maintains its locked condition due to the permanent magnet excitation. The step angle of this 2-phase, 4-pole, 6-tooth rotor motor is given as 360/ (2 × 6) = 30 degrees. In practice, hybrid motors are constructed with more number of rotor poles in order to get high angular resolution.[8].

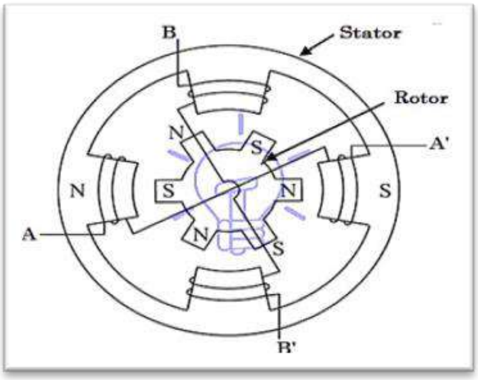


Figure 2.15 Working of Hybrid Stepper Motor

E- A4988 Stepper Motor Driver

A4988 is a complete micro stepping motor driver with built-in translator for easy operation. The product is available in full, half, 1/4, 1 / 8 and 1 /16 step mode to operate bipolar stepper motors of an output drive capability of up to 35 V and ± 2 A. A4988 includes a fixed off-time current regulator, the regulator can operate in slow or mixed decay modes. Converter is easy to implement key for A4988. Just enter a pulse in the "step" input drives the motor to produce micro-step.

Without the need for phase sequence tables, high frequency control lines, or complex interfaces to program. A4988 interface is ideal for complex microprocessor is unavailable or overloaded applications [7].
In the micro-step operation, the chopper control A4988 automatically selects within the current decay mode (slow or mixed). In mixed decay mode, the device is initially set to a fixed part of the rapid decay in downtime and slow decay for the remainder of downtime. Mixed decay current control scheme results in reduced audible motor noise, increased step accuracy, and reduced power consumption. Internal synchronous rectification control circuitry provides to improve the pulse-width modulation (PWM) operation power consumption. Internal circuit protection includes: thermal shutdown with

hysteresis , under voltage lockout (UVLO) and crossover current protection. No need special power sequencing [7] .

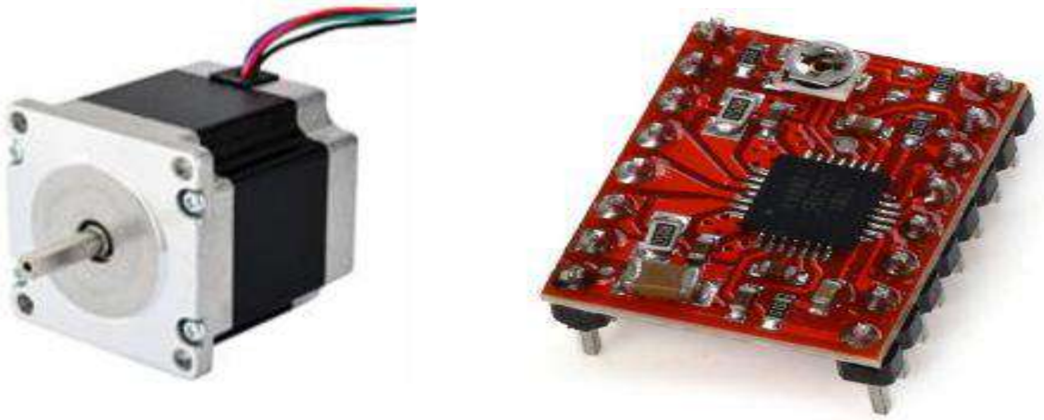


Figure 2.16 Stepper motor

2.2.3 Circuit and Wiring

The wiring of the various components of electronics system Pin out information's:

- ENA = to enable (5Volt), and to disable (0 Volt) the driver
- MS1, MS2, MS3 = this logic gate input to control motor rotations, see the logic/

MS1	MS2	MS3	Resolution
LOW	LOW	LOW	Full Step
HIGH	LOW	LOW	Halft Step
LOW	HIGH	LOW	Quarter Step
HIGH	HIGH	LOW	Eighth step
HIGH	HIGH	HIGH	Sixteenth Step

- RESET = to reset the driver work (enable (5Volt), disable (0 Volt))

- SLEEP = to set driver sleep for a while (enable (5Volt), disable (0 Volt))
- STEP = to set how many step motor will turning/ rotate
- DIR = to set motor turning Clockwise (CW) or Counterclockwise (CCW)
- VMOT = motor voltage supply (8 ~ 35 Volt DC)
- GND = motor ground supply
- 1A & 1B = bipolar stepper motor 1st winding/ core
- 2A & 2B = bipolar stepper motor 2nd winding/ core
- VDD = driver voltage supply (5 Volt DC)
- GND = driver ground supply

  Small potentiometer = for adjusting the current output (Ampere) to stepper motor [9].

2.2.4 power supply

Figure 2.17 power supply

2.2.4.1 Specification:

- 110-220 V AC Input +-15%
- 12v 20A (Maximum) DC Output
- 240W Power Output
- Terminal Board Design for Easy connections (9 Pin Termina Board L, N,E,COMx3,+Vx3)
- Short Circuit Protection
- Passive Cooling Design with Heat Sink
- Sturdy Steel Body
- Adjustable output Voltage from (10V to 12.5V DC)
- Dimensions: L=20cm, B=12 cm, H=5.5 cm
- Mounting Fitting: Screw mount via Bottom or Side Panel
- Application: SMD Led Strip, Battery Charger, Toys, Lights, CCTV ,Wall Clocks, Night Lamps, Door Bell, Water Overflow Tank, School and College Projects etc. 100% Quality control via in-house test facilit with automating machine and experience technicians Sales Package Content:1 x 12v 20A DC power Supply[10].

2.2.4.2 Dc Male Connector 17 ,24 Volts

Fig. 2.18 Dc Male Connector

Chapter Three

Construction of CNC

3.1 Frame Structure of CNC

Building an aluminium structure in the shape of cube the length of the side of the cube from the inner side to the other side shall be equal to the length of the shaft 3 cm (3cm for installation purpose) as shown in figures (3-1) and (3-2) .

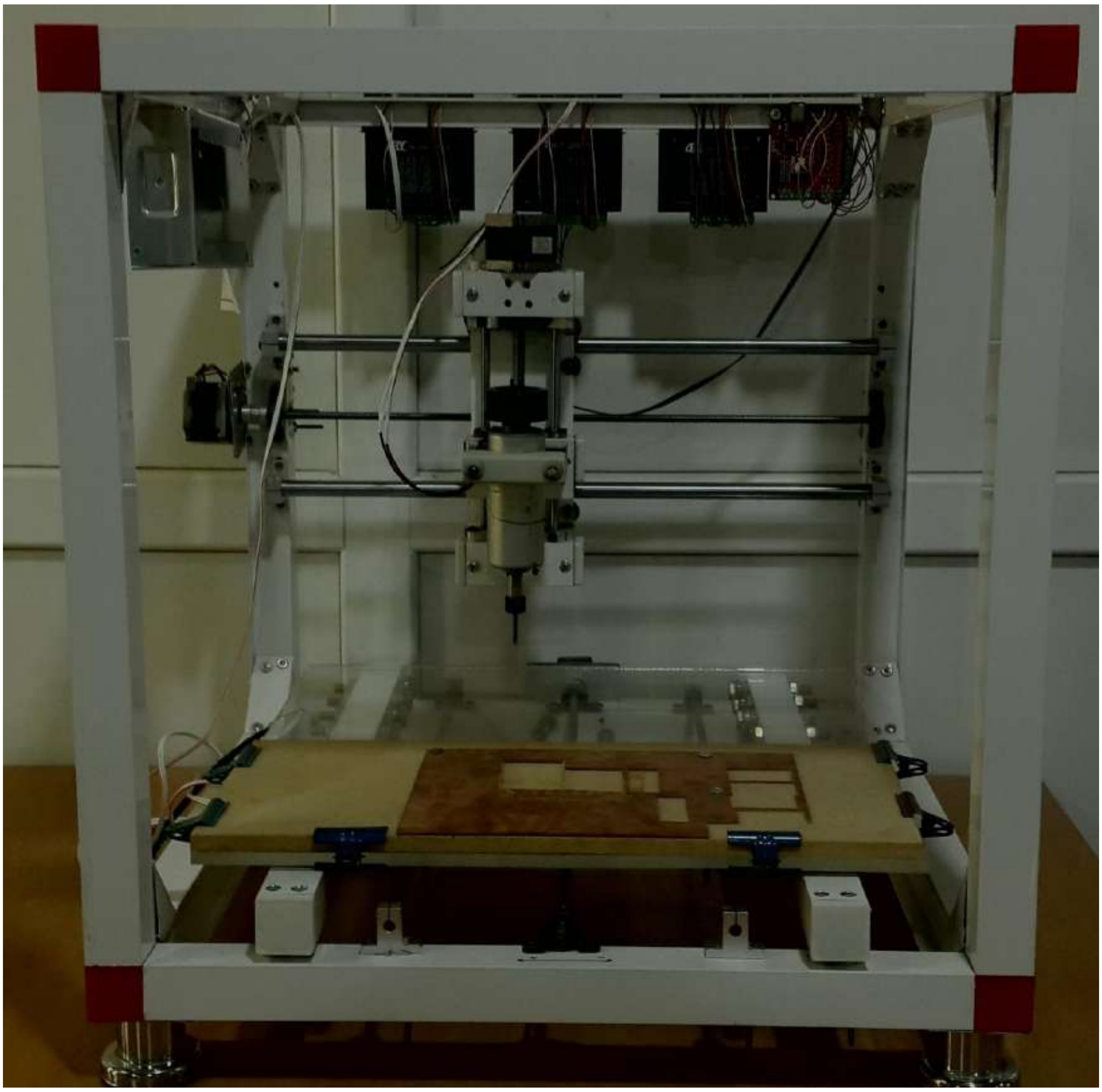


Fig. 3.1 Construction of CNC

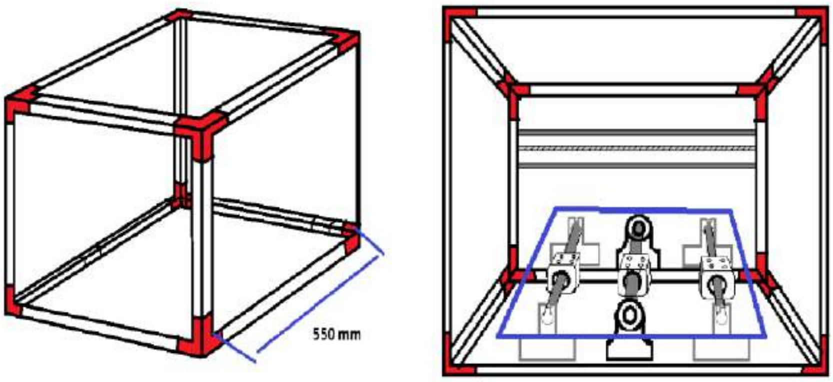


Fig. 3.2 frame structure of CNC

3.2 Installation of axis assembly (y)

(y) axis consists of:

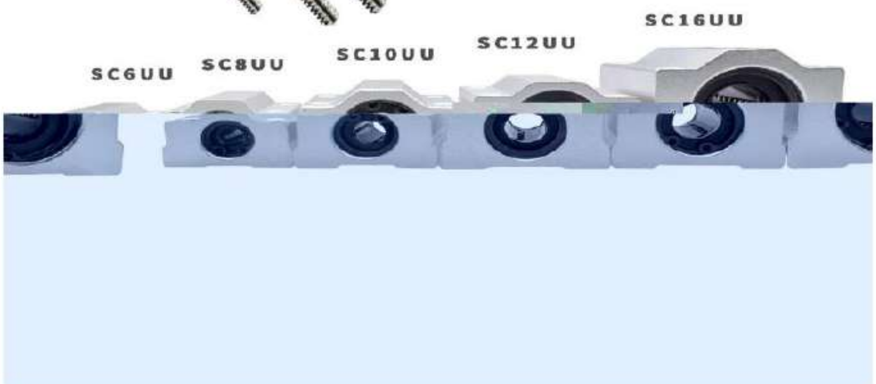
- ❖ Two pieces of shafts in dimension 600 mm in length and dimeter 8mm.
- ❖ Lead screw 600 mm in length and dimeter is 8mm.
- ❖ Linear rail shaft sk8 (4 pcs) diameter (8mm).
- ❖ Kp08 bearing bracket (2 pcs) diameter (8mm).
- ❖ Liner ball bearing sc8aa (4pcs) diameter (8mm).
- ❖ Stage plastic plate type acrylic.

linear rail shaft SK8 shaft

acrylic

KP08 bearing bracket

lead screw



linear ball bearing SC8UU

Fig. 3.3 Mechanical parts y axis

3.3 Installation of axis assembly (z):

Building from aluminium cube with length 30 cm and width 10 cm.

(z) axis consists of:

- ❖ Two pieces of shafts in dimension 20 cm in length and dimeter 8mm.
- ❖ Lead screw 20 cm in length and dimeter is 8mm.
- ❖ Linear rail shaft SHf8 (4 pcs) diameter (8mm).
- ❖ Horizontal Kfl08 bearing bracket diameter (8mm) (2pcs).
- ❖ Long type linear bearing LMH8Luu (2pcs) diameter (8mm).

horizontal KFL08bearing bracket

lead screw

linear rail shaft SHF8

long type linear bearing LMH8LUU

shaft

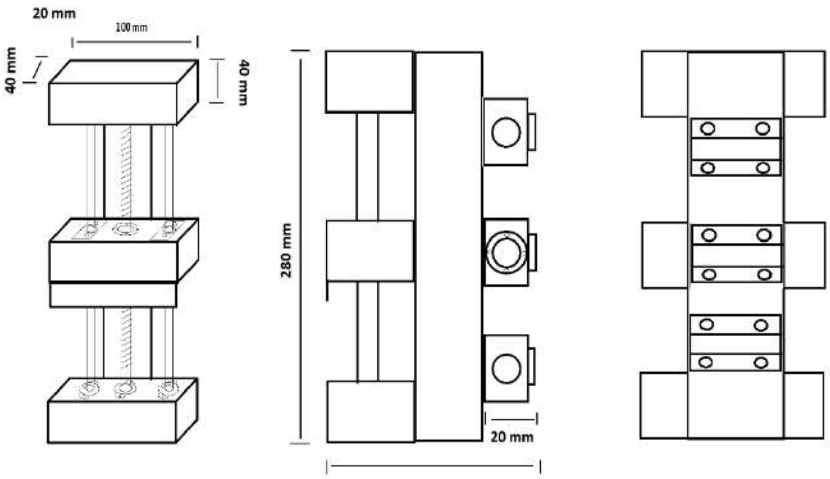


Fig. 3.4 parts of Z axis

3.4 installation of axis assembly (x):

(x) axis consists of:

- ❖ Two pieces of shafts in dimension 600 mm in length and dimeter 16mm.
- ❖ Lead screw 600 mm in length and dimeter is 8mm.
- ❖ Linear rail shaft SK 16 (4 pcs) diameter (16mm).
- ❖ Kpo8 bearing bracket (2pcs) diameter (8mm).
- ❖ Linear ball bearing sc 16uu (4pcs) used to carry (z) axis.

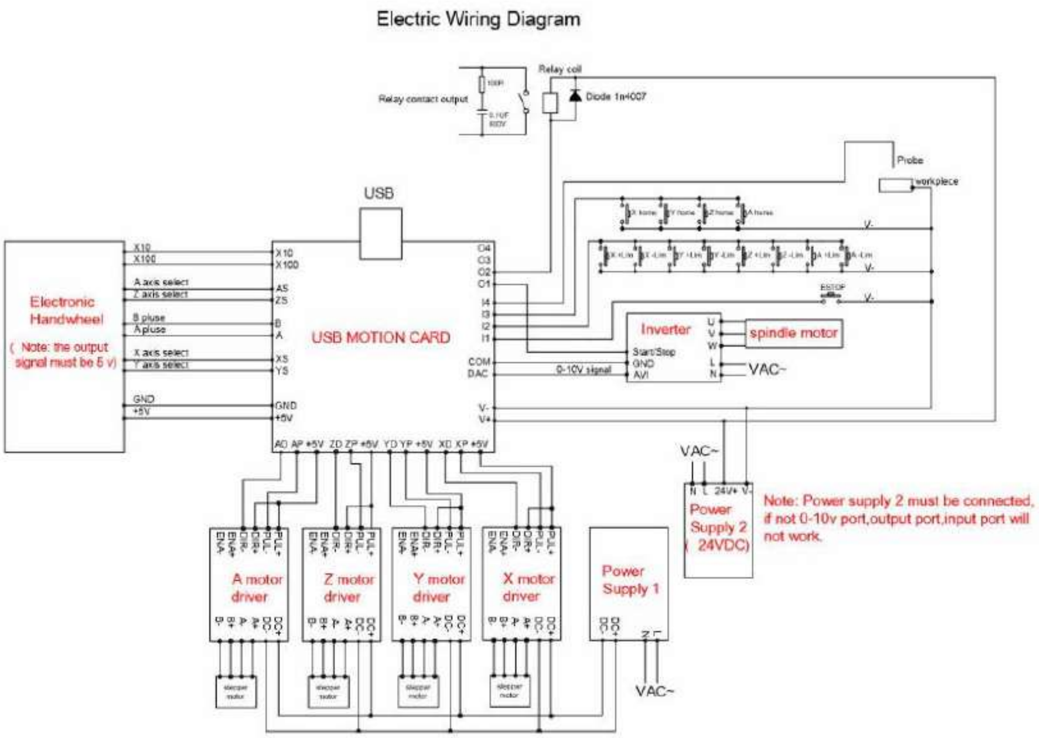
3.5 Stepper Motor for Each Axis

Install the stepper motor for each axis with connecting each motor with lead screw by flexible shaft coupler.

Fig. 3.6 parts of (x) axis

3.6 Install (Power Supply, Drivers, CNC Card)

with connect the circuit wires.



Connection Diagram

Fig. 3.7 circuit wires

3.7 Install the spindle motor with bits.

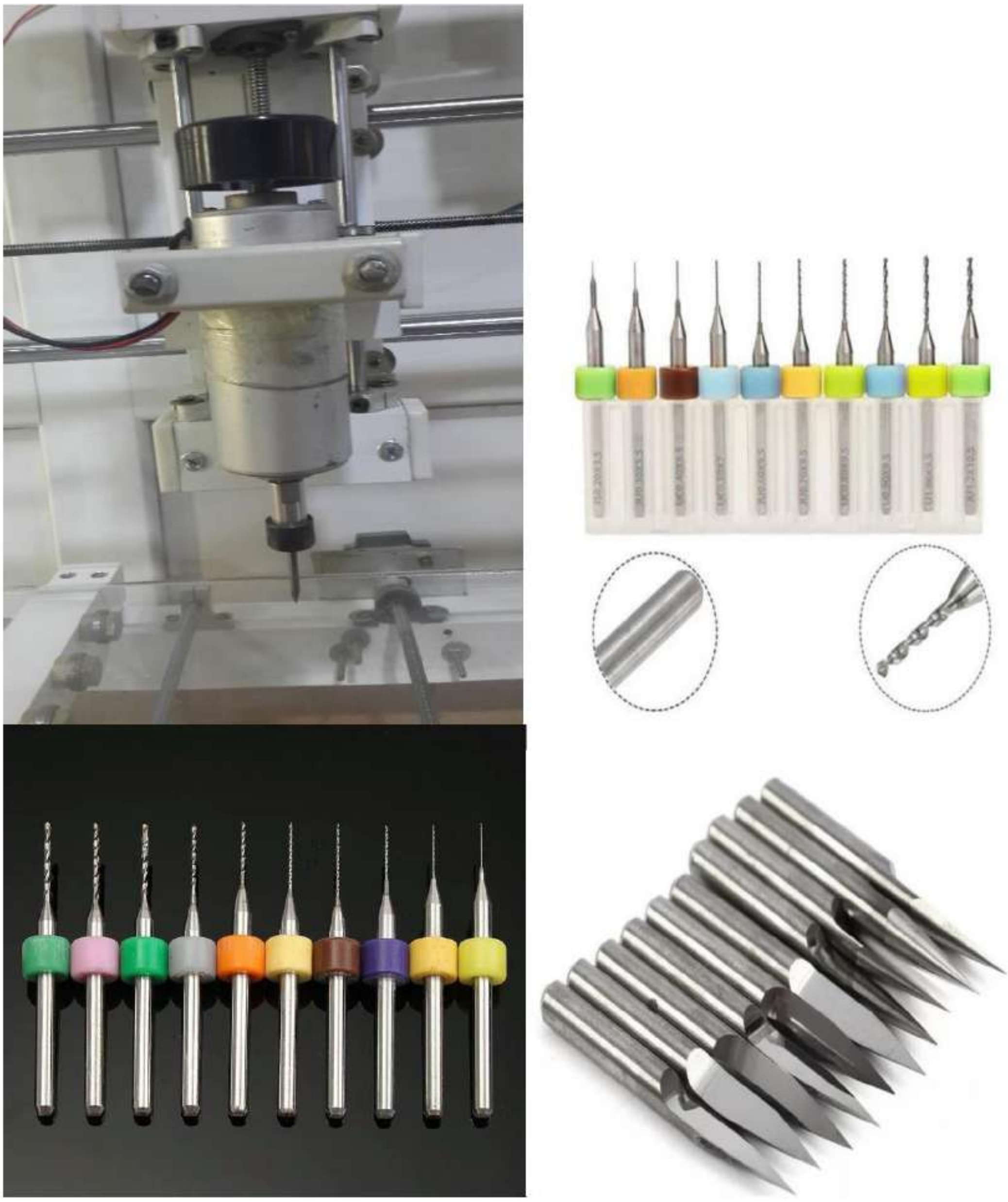


Fig. 3.8 spindle motor

Chapter 4

A First Look at EAGLE

4.1 The Control Panel

The Control Panel normally appears after starting EAGLE, and this is the program's control center. All the files specific to EAGLE are managed here, and some basic settings can be made. It is similar to the familiar file managers used by a wide variety of applications and operating systems. Each EAGLE file is displayed in the tree view by means of a small symbol [11-13].

A context menu is opened by clicking with the mouse on an entry in the tree view. This allows you, depending on the object, to carry out a variety of actions (rename, copy, print, open, create new etc.).

The Control Panel supports Drag Drop. This can also be done between different programs. You can, for instance, copy files, move them, or create links on the desktop. User Language programs or script files that are pulled with the aid of the mouse out of the Control Panel and into an editor window are started automatically. If, for instance, you pull a board file with the mouse into the Layout Editor, the file is opened.

The tree structure provides a quick overview of the libraries, Design Rules, User Language programs, script files, CAM jobs and projects. Special libraries, text, manufacturing and documentation files can belong to a project as well as schematic diagrams and layouts.

The first time it is called, the Control Panel will appear very much as shown in the following diagram. If an object is selected in the tree view, further relevant information is displayed in the right hand part of the window.

Simply click onto various folders and files in order to experiment with the Control Panel's facilities.

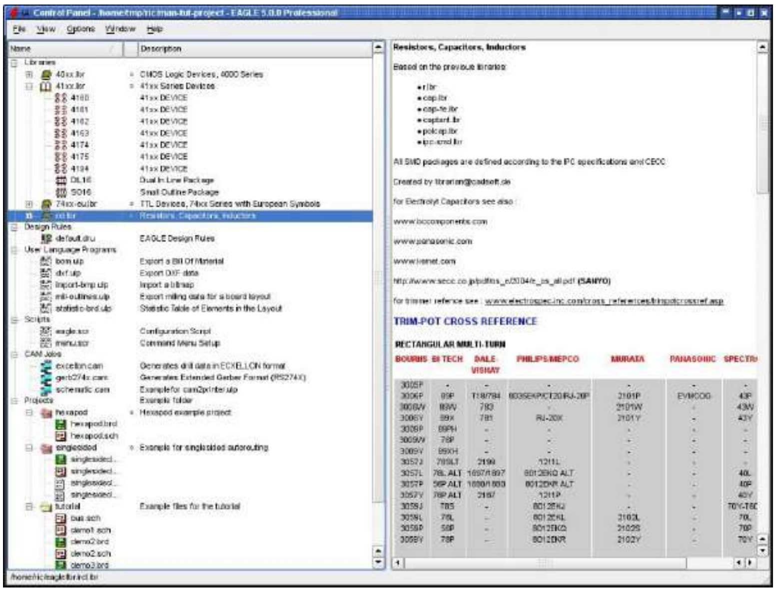


Fig.4.1 Control Panel: On the right, the description of the RCL library

4.2 Library Summary

The possibility of displaying the contents of the libraries is particularly interesting. It provides a very rapid overview of the available Devices. Expand the Libraries entry, and you can see the available libraries [14-15]. In the Description field you can see a brief description of the contents. If a library is selected, you will see more extensive information about the library in the right hand part of the Control Panel. If you then expand a library entry, the contents will be displayed together with a short description of each element. Devices and Packages are marked with a small icon.

The description of the Device and a graphical representation of it appear on the right. The available Package and technology variants are listed. If you click onto one of the Package versions, the Package preview shown above will change. If a Schematic Editor window is open, the entry ADD will be shown right of the variant name. Click it and the Device is attached to the mouse cursor as soon as it is over the Schematic Editor window. Now you can drop it in the schematic.

If you are only working with the Layout Editor, this will of course also operate with Packages. It is, additionally, possible to drag a Device from the tree view into a schematic diagram and to place it there by means of Drag Drop. If it has more than one Package version, the ADD dialog opens automatically, so that the desired Package can be selected.

The green marker behind the library entry indicates that this library is in use. This means that it can be used in the current project. Devices in this library will be examined by the search function in the ADD dialog of the schematic diagram or of the layout.This makes them available for the project. The library will not be examined if the marking is gray. If starting EAGLE without a project (no eagle.epf file is read, the project has been closed before exiting EAGLE last time) and creating a new project (⇒ File/New/Project) all libraries will be in use automatically. However, opening an already existing project, where only certain libraries are in use before creating the new project, will adopt this selection.

If the Library Editor window is open you can Drag&Drop a complete Device set or Package definition from the Control Panel into the library window. This way you can copy it from one library into another.

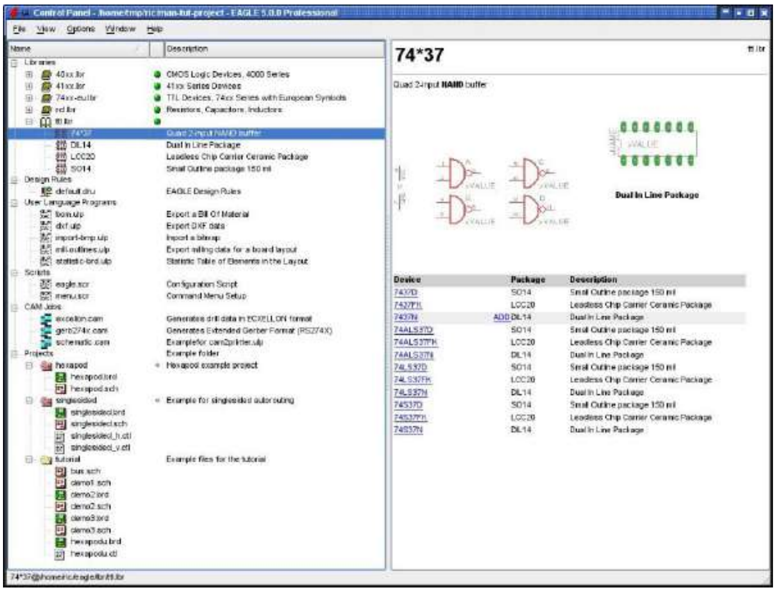


Fig. 4.2 Control Panel: Library summary with Device vie

4.3 Design Rules

Special Design Rules can be specified in EAGLE to govern the board design. These can be saved as data sets in special files (*.dru). The parameter set that is to govern the current project is specified in the Design Rules branch of the tree view. If no data has been provided for the Design Rules (DRC command), EAGLE will itself provide parameters. The marking to the right of the file entry specifies the default parameter set for the current project. The layout will be checked by the DRC in accordance with these criteria.

4.4 User Language Programs, Scripts, CAM Jobs

These entries show the contents of the ulp, scr and cam directories. They contain various User Language programs (*.ulp), script files (*.scr) and CAM jobs (*.cam) for the output of data using the CAM Processor. If one of these files is selected in the Control Panel, you will see a full description of the file. The paths can be set by means of the

Options/Directories menu. This is discussed in more detail later in this chapter.

4.5 Projects

The various projects are managed from the Control Panel. A click onto the Projects entry displays various folders. These are located under the path set under Options/Directories/Projects. It is allowed to define more than one path there.

A project usually consists of a folder which represents the project by its name and the project's configuration file eagle.epf. The folder usually contains all files that belong to your project, for example, schematic and board file, special library files, script files and so on.

Project directories that contain the project file eagle.epf will be marked with a red folder icon. 'Normal' folders will be marked with a yellow icon.

The project to be edited is selected in the Projects branch. On the right of the project's name you will find a marker which is either gray or green. With the help of this marker one can open or close projects. Clicking onto a gray marker, loads the project. The marker appears green now. Clicking onto the green marker again or clicking onto another gray marker closes the current project respectively opens another project after closing the current one. This way one can switch easily from one project to another.

As an alternative you can open or close a project by double-clicking onto the entry in the tree view or by pressing the Space or Enter key.

While closing a project the settings of the currently opened Editor windows will be stored in the corresponding project file eagle.epf, provided that the option Automatically save project file is set in the

Options/Backup menu. If the project file was generated by another EAGLE version than currently used, you will be asked, if it is allowed to overwrite the file.

New projects are created by clicking the right mouse button onto a folder entry in this branch. A context menu opens which permits new files and directories to be created and the individual projects to be managed. Selecting the option New/Project invokes a new folder which has to be given the project's title. The project file eagle. epf will be created automatically You can also use the File/Open/Project or the File/New/Project menu to open or create a new project.

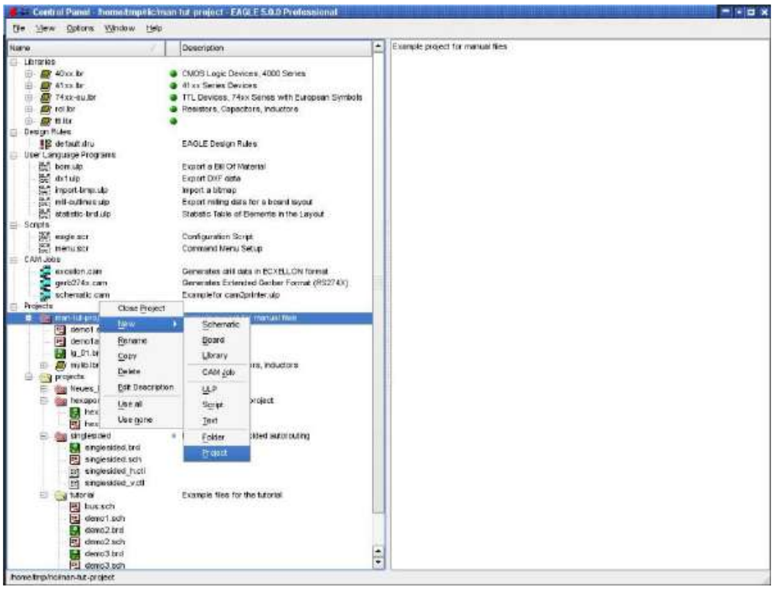


fig 4.3Context menu for project management

The context menu contains the Edit Description item. description of the project can be entered here, and this is then displayed in the description box.

4.6 Menu Bar

The Control Panel allows various actions to be executed and settings made through pull-down menus that are explained below.

A- File Menu

The File menu contains the following items:

1- New

Creates a new layout (board), schematic, library, CAM job, ULP, script or text file. The Project option creates a new project. This initially consists simply of a new directory in which the files for a new project are handled. These will consist as a rule of the schematic diagram and layout, possibly of special libraries, script files, User Language programs, documentation files etc. and of the file eagle.epf, in which project-specific settings are stored.The default directories for the various file types are defined in the Options/Directories menu. CAM jobs are definitions for generating output data with the CAM Processor. Script and ULP files are text files containing command sequences in the EAGLE command language or the EAGLE User Language. They can be created and edited with the EAGLE Text Editor or with an external text editor.

2- Open

Opens an existing file of the types mentioned above.

3- Open recent projects

Lists recently used projects.

4- Save all

All changed files are saved. The current settings for the project are saved in the file eagle. epf, even if the option Automatically save project file in the menu Options/Backup... is switched off. User specific settings are stored in the file eaglerc. Ussr (Windows) or .eaglerc (Linux/Mac).

5- Close project

The project will be closed. Project specific settings are saved in the eagle. epf file of the current project directory.

6- Exit

The program is terminated. When EAGLE is started again, the last program status is restored, i.e. the windows and other working environment parameters appear unchanged. If there was no project loaded only the Control Panel will be opened next time. The current status is also saved when you leave EAGLE with Alt-X from any program part.

B- View Menu

1- Refresh

The contents of the tree view are updated.

2- Sort

The contents of the tree view will be sorted by name or by type.

C- Options Menu

1- Directories

The default directories for particular EAGLE files are entered in the directories dialog box. More than one path may be entered for each of these. In the Windows version the entries are separated by semicolons, while a colon is used in the Linux and Mac version. The Projects directory is the default directory for the Text Editor.

The Projects directory contains subdirectories, each of which represents a particular project. Each of the project directories contains an EAGLE project file (eagle.epf). A project directory and its subdirectories usually contain all the files that are associated with one particular project, such as the schematic diagram and the layout, text files, manufacturing data, documentation files and so on.

Fig 4.4 The directories dialog in the Options menu

Type the path directly into the corresponding box, or select the desired directory by clicking the Browse button. The default settings can be seen in the diagram above. $EAGLEDIR stands for the installation's EAGLE directory. You may also use $HOME for your home directory under Linux. Under Windows it is possible to define this environment variable wit the SET command.

If a HOME variable has not been set within the Environment variable, then under Windows EAGLE will suggest the directory Application Data.

This directory is defined in the Windows registry in: HKEY_CURRENT_USER\Software\Microsoft\Windows\CurrentVersion ion\Explorer\Shell Folders\App Data In this folder you can also find the user specific configuration file eaglerc. usr. It is of course also possible to specify paths with an absolute format.

4.7 Backup

When files are saved, EAGLE creates backup copies of the previous files. The maximum backup level field allows you to enter the maximum number of backup copies (default: 9). Backup files have different file extensions, enumerated sequentially. Schematic files receive the ending s#x, board files b#x, and library files l#x. x can run from 1 to 9. The file with x = 1 is the newest one.

The automatic backup function also permits the backup to be scheduled. The time interval can be between 1 and 60 minutes (default: 5 minutes). The backup files have the endings b##, s## and l## respectively.All these backup files can be further processed in EAGLE if they are renamed and given the usual file endings (brd, sch, lbr).

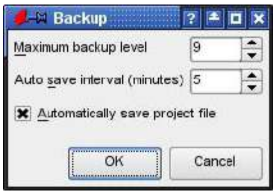


Fig. 4.5 Backup dialog

If the option to Automatically save project file is chosen, your project is automatically saved when you close the current project or leave the program.

4.8 User Interface

The User Interface dialog allows the appearance of the editor windows for the layout, schematic diagram and library to be adjusted to your preferences. You can also access this menu from the Editor windows.

In the Controls box you specify which objects are to be displayed in the editor window. If you deactivate all the Controls, only the command line will remain for entry. This maximizes the free area available.

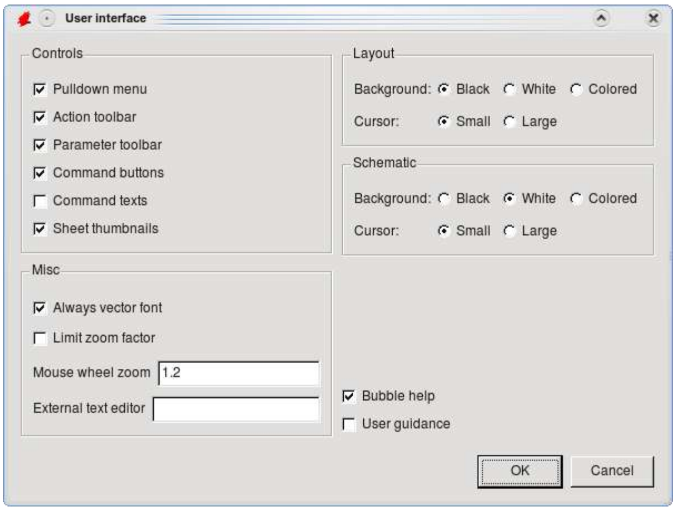


Fig. 4.6 Settings for the User Interface

The option Always vector font shows and prints texts with the built-in vector font, independently from the originally used font. Using the Vector font guarantees that the output with a printer or the CAM Processor is exactly the same as shown in the editor window. Fonts other than vector font depend on the systems' settings and cannot be controlled by EAGLE. The output of non- vector fonts may differ from the editor's view.

Opening the User Interface dialog from one of the Editor windows (for example, the Layout Editor) the Always vector font option offers an additional item Persistent in this drawing. Setting this option causes EAGLE to save the Always vector font setting in the current drawing file. So you can be quite sure that the layout will be shown with vector font at another's person computer (for example, at a board house).

Please see the help function for details (TEXT command). Limit zoom factor limits the maximum zoom factor in an editor window. At maximum zoom level the width of the drawing is about one Millimetre (approx. 40 mil). Switching off this option allows you to zoom until the 0.1 Micron (0.004 mil) grid will become visible. If you are working with a wheel mouse, you can zoom in and out by turning the mouse wheel. Mouse wheel zoom determines the zoom factor. The value 0 switches this function off. The wheel is used for scrolling then.

The field External text editor allows you to specify an alternative for the built- in EAGLE text editor. Further details on this can be found in the help function in the section Editor windows/Text editor.

The background color and the appearance of the drawing cursor can be separately adjusted for the layout and the schematic diagram editors. The background may be black, white or shown in any other color (Colored). The background color definition is described on page 100.

The cursor can be displayed optionally as small cross or as large cross-hairs. The check boxes in the Help area allow to switch on or off the popup texts for the command icons (Bubble help) and the help texts for the commands in the status line (User guidance). Selecting the User guidance check box displays, addition information about the selected object, like the net or signal name, the net class, or the part's name and value (with NET, MOVE, ROUTE, SHOW...), instructions about the possible mouse actions in the status bar of the editor window.

1- Window Positions

Here you can store the positions and the sizes of the currently open Editor windows. Each file that will be opened from now on appears in its Editor window at the given position and size parameters that were stored.

If you delete the stored positions again, EAGLE determines the location of an Editor window and uses a fixed size for it, which is the default setting.

2- Window Menu

From the Window menu you can choose the window (schematic, board, etc.) to be displayed in the foreground. The number on the left is the window number. It allows you to choose a window when combined with the Alt key (e.g. Alt+1 selects window 1).The combination Alt+0 can be used anywhere in the program to bring the Control Panel into the foreground.

3- Help Menu

The Help menu contains an item for calling the help function, as well as items for installing a new license (Product registration) and getting information about the program version etc. (Product information).

4.9 Product registration

The registration dialog is called automatically when you start EAGLE the first time. If you want to install an upgrade you must start this dialog from the Help menu, and then enter the necessary information according to the License/Product Registration section of the help function. All editor windows have to be closed before. Read the notes in the chapter Installation for more information.

Check for Update

By default EAGLE checks for newer EAGLE versions on the CadSoft web server. In case there is a newer version available you will be informed by a message window. EAGLE informs you about a new software release, but won't download it automatically. Click the Configure button to let EAGLE look for the software version daily, weekly, or never. If you are working with a Proxy server, type in it's name.

In case you are interested in the newest beta version, you can activate this check, too.

4.10 The Schematic Editor Window

The Schematic Editor window opens when you load an existing schematic or create a new one. There are several ways of opening files in EAGLE. You can, for instance, load a schematic diagram by means of the File/Open/Schematic menu in the Control Panel. Alternatively double-click onto a schematic diagram file in the tree view. If you want to create a new schematic, select the menu File/New/Schematic. This will open a schematic with the name untitled.sch in the current project directory.

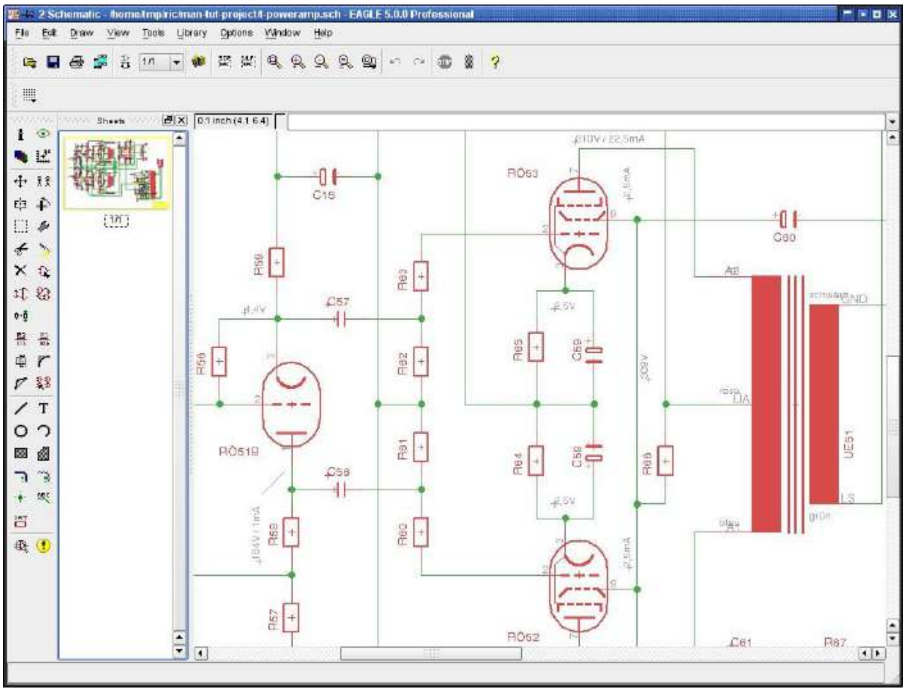


Fig. 4.7 The Schematic Editor

If you want to create a schematic diagram straight away in a new project, you may for example click with the right mouse button onto a project in the Projects entry of the tree view, and select the New project option from the context menu. The new project receives a name. Then click onto this entry with the right mouse button. Now select New/Schematic from the context menu.A new schematic opens in this project directory. On top you will see the **title bar**, which contains the file name, and then the **menu bar**, and the **action toolbar**. Below the action toolbar there is the **parameter toolbar,** which contains different icons, depending on the active command.

Above the working area you will find the **coordinate display** on the left, with the **command line**, where commands can be entered in text format, to the right of it. EAGLE accepts commands in different but equivalent ways: as mouse clicks, text via keyboard, or from command (script) files. On the left of the work space you find the **command toolbar**, which contains most of the Schematic Editor's commands. In the **status line**, at the bottom of the screen, instructions for the user appear, if a command is active. On the left you you can see the preview of the schematic sheets. You can sort the sheets via Drag&Drop.

Each of the toolbars can be displayed or hidden using Options/User Interface. It is also possible to rearrange the toolbars within certain limits with the aid of the mouse. The command toolbar, for instance, can also be placed on the right, or the action and parameter toolbars can be placed together on one line.

4.11 How You Obtain Detailed Information About a Command

1- Bubble Help And Tool Tips

If the mouse cursor remains above an icon for longer than a certain time, the name of the EAGLE command appears. You also see a short explanation below in the status line. For example, move the cursor over the WIRE icon. Bubble help with the word Wire appears directly by the cursor. The short description, Draw lines, appears in the status line.

If you select the command, a short note appears below in the status line, indicating what would normally be expected as the next action. For instance, if you click onto the WIRE icon, the status line will display the instruction: Left-click to start wire . These functions can be activated or cancelled in the Control Panel by means of the Options/User Interface menu.

2- Help Function

If you want to learn more about a command, e.g. the WIRE command, click its icon in the command toolbar, then click the help icon. As an alternative you can type HELP WIRE □ in the command line. The □ character symbolizes the Enter key. The contents of the EAGLE Help is stored in a single HTML file and can be viewed for example with a web browser, as well. It also offers a full-text search. After typing in a search term in the Find line, EAGLE help no longer shows all pages but only the pages containing this expression. The keys F3 and Shift+F3 allow you to go to the next or previous location. Each search term found will be marked. Green indicating the currently found term, yellow for all others.

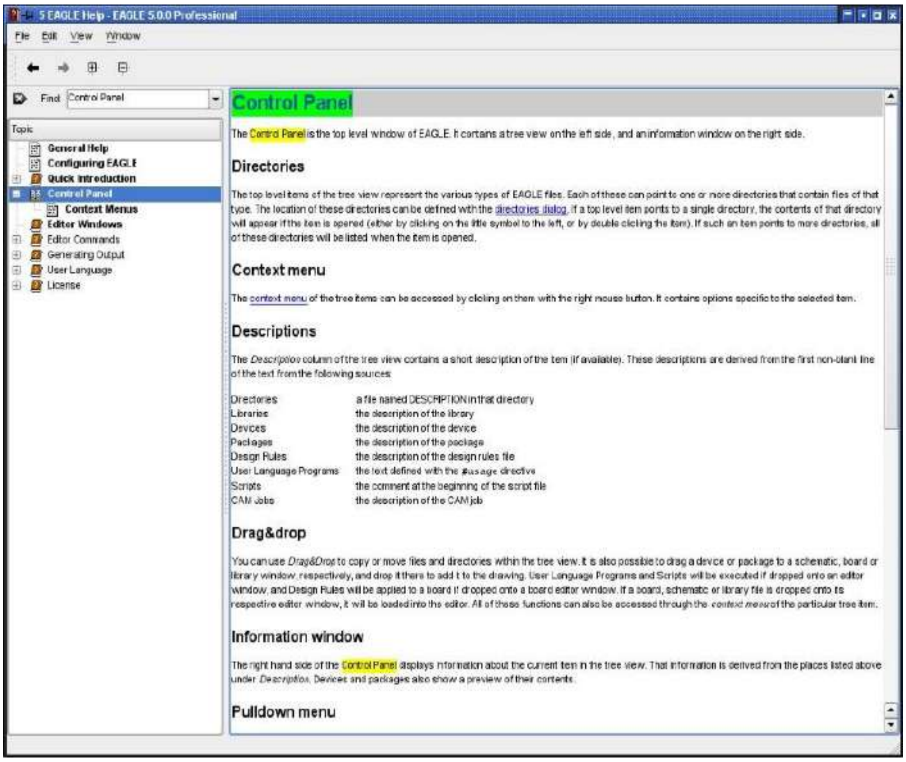


Fig. 4.8 EAGLE Help window

3- Command Parameters

A number of EAGLE commands need additional parameters. Refer to the help pages for a description of the textual entry of parameters (via command line or script file). Most of the parameters can be entered by clicking the appropriate icons in the parameter toolbar, which changes according to the selected command. These icons also show bubble help explanations. This is how the parameter toolbar appears when the NET command is activated.

Fig. 4.9 Parameter toolbar of the NET command

On the left is the GRID icon for setting the grid pitch. To the right are buttons for the bend mode (SET WIRE_BEND) of the net line, followed by the miter radius for smoothing line joints with the options straight or rounded (see MITER command). Next to this is the Style menu where the type of line is defined. On the far right is a value menu for assigning a Net class.

4-  **GRID**

This icon is available at any time. It is used to adjust the grid and to select the current unit. In EAGLE, any value relates to the current unit.

A right-click onto the icon opens a popup menu that contains the entry Last. So you can switch back to the previously chosen grid setting. The New... entry allows to define so-called Aliases. More about this in chapter.

5- The Action Toolbar

This toolbar is composed of the following icons:

From the left: Open file, save file, print file, call CAM Processor, open/create corresponding board file (BOARD command).

Load, remove, or create a new schematic sheet.

 **USE**

Select libraries which will be taken into consideration by the ADD dialog. Can also be done with the Library/Use menu item or by clicking the markers in the Libraries branch of the Control Panel's tree view. The context menu of the entry Libraries or of its subfolders contains the entries Use all and Use none for a quick and simple selection/deselection of all libraries (of the folder). This command has to be used in script files in order to choose the library you want to take parts from.

6- SCRIPT 

Execute a script file. This enables you to execute any comma sequence with a few mouse clicks. A right click onto the icon shows a list of recently executed script files.

7- RUN 

Start a User Language program (ULP). A right click onto the icon shows a menu that contains a list of recently used User Language Programs.

WINDOW  These icons represent different modes of the WINDOW command: Fit drawing into the screen (WINDOW FIT, Alt-F2), zoom in (F3), zoom out (F4), redraw screen (WINDOW or F2), select new area. To move the current drawing window, click the middle mouse button

8- UNDO/REDO 

These commands allow you to cancel previous commands and to execute commands which have previously been cancelled. Default function keys: F9 and F10.

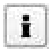
9- Stop Icon 

Terminates the execution of EAGLE commands (Edit/Stop command).

10- Go Icon 

Starts the execution of an active EAGLE command, which allows further parameters to be entered by the user, like it is with the AUTO or the MARK command.

4.12 The Command Toolbar of The Schematic Editor

1- INFO 

Shows the properties of the selected object. If you know the name of the object, you can use it as a parameter in the command line. Depending on the selected object some of the properties can be altered

2- SHOW

Highlights the object to be selected with the mouse. It's also possible to enter the object's or Gate's name (even several names at once) in the command line. You may use the characters * and ? as wildcards, as well. Ctrl + SHOW toggles the show state of the selected object. If you are looking for very small objects, it can be useful to use the SHOW command with the @ option, like in SHOW @ C12; The location of part C12 will be recognized at once, because the part is marked with a surrounding frame.

If the searched object is not located on the current sheet, the SHOW window opens and informs you about the sheet where it is located. In case of objects that consist of more than one part, like elements with several gates or nets that spread over several sheets, the window will list several entries. Clicking on one of the entries center the selected object on the screen. If the searched object is not found in the whole schematic, the Sheet column will be marked with a minus sign '-'.

3- **DISPLAY** 

Select and deselect the layers to be displayed. See the Appendix for the meaning of the layers DISPLAY LAST shows the recently used layer combination that was previously selected for display. For further details please see help function.

4- **MARK**

The following mouse click defines the new origin for the coordinate display. Relative coordinates (R x-value y-value) and polar values (P radius angle) are shown in addition to absolute coordinates in the coordinate display box. If you first click the MARK icon and then the traffic light icon, only the absolute coordinate values will be displayed again.

5- **MOVE**

Move any visible object. The right mouse button rotates the object while it is attached to the mouse cursor. If you move a net over a pin, no electrical connection will be established. If you move the pin of a Gate over a net or another pin, an electrical connection will be create To move groups of objects:

Define the group with the GROUP command, click the MOVE icon, press the Ctrl key, then click into the drawing with the right mouse button, and move it to the desired location.

If you don't press the Ctrl key, the context menu pops up after clicking with the right mouse button. It contains an entry Move:Group that allows you to move the group, too. The right mouse button rotates the group by 90 degrees while it is attached to the mouse cursor.

If you like to move the group onto another sheet, click the sheet combo box in the action toolbar or select it from the Sheets preview. Place the group there MOVE can be used in the command line with various options. See the help function for details.

6- **COPY**

Copy parts and other objects. When copying nets and buses the names are retained, but in all other cases a new name is assigned. COPY can be used with groups and works the same as it would be with CUT and PASTE within a drawing. The content of the group does not go to the clipboard of your operating system!

7- **MIRROR**

Mirror objects.

 8- **ROTATE**

Rotate objects by 90 degrees (also possible with MOVE).

 9- **GROUP**

Define a group which can then be moved, rotated, or copied with CUT and PASTE to another drawing or whose properties are to be changed. After the icon has been clicked, a rectangular group can be defined by holding down the left mouse button and dragging the cursor to the diagonal corner of the rectangle. If you want to define a group by a polygon, use the left mouse button to determine the corners of the polygon. Then click the right mouse button to close the polygon. GROUP ALL in the command line selects all objects on the current sheet, if the respective layers are displayed. The following command (ROTATE, CHANGE, MOVE...) has to be applied to the group with the right mouse button while the key is pressed. Exception is CUT: Here a left mouse click is expected. If you like to add further groups to an already existing one, press the Shift key and define the first corner of the selection area with a mouse click. In case you want to add an object to or remove it from the group, press the Ctrl key and click onto the object in question. Press Ctrl + Shift to toggle the membership of an object and its hierarchically superior objects: Clicking for example, on a net segment in the Schematic inverts the group membership.

 **10- CHANGE**

Change the properties of an object, e.g. the width of a line, the Package variant or the size of text. See help for details.

An object's properties can be checked and even changed, where applicable, by the Properties entry of the context menu. To access the context menu, click onto the object with the right mouse button.

 11- **CUT**

Transfer the objects of a previously defined group into the paste buffer. Activate the CUT command and click with the left mouse button into the group to set a reference point.

All other commands that can be executed with a group expect a right mouse click while the Ctrl key is pressed.

12- PASTE

Insert objects from the paste buffer into the drawing. Restrictions: see help function. Not identical to the Windows paste function.

13- DELETE 

Delete visible objects. Also, in combination with GROUP command. If a group has been defined, it can be deleted with the right mouse button while the Ctrl key is pressed. The DELETE command deletes an entire part in the Schematic when clicking onto a Gate with the Shift key pressed. In that case, the tracks connected to the Package in the board, if already existing, will stay unchanged.

Clicking onto a net or bus wire with the Shift key pressed deletes the entire net or bus segment.

14- ADD 

Add library elements to the schematic. A search function helps Devices to be found quickly. USE specifies which libraries are available. A right click onto the ADD icon opens a popup menu that lists recently fetched Devices.

15- **PINSWAP** 

Swap two nets connected to equivalent pins of a Device, provided the pins have been defined with the same Swap level.

16 -**GATESWAP** 

Swap two equivalent Gates of a Device, provided the Gates have been defined with the same Swap level. In EAGLE terminology, a Gate is a part of a Device which can be individually placed on a schematic (e.g. one transistor from a transistor array).

17- **REPLACE** 

Replace a component (Device) with another one from any library. This can only work if the new component has at least as many pins as the current one and the pins as well as the pads have identical names or the same positions. A right-click onto this icon opens a popup menu that shows a list of recently replaced Devices.

 18- **NAME**

Give names to components, nets, or buses.

 19- **VALUE**

Provide values for components. Integrated circuits normally get the type (e.g. 74LS00N) as their value. A right click onto this icon opens a list of already used values. Select an entry and apply it to one or more components by clicking onto them successively.

 20- **SMASH**

Separate name, value, and, if any, attribute texts from a Device, so that they can be placed individually. The size of detached (smashed) texts can also be individually changed. Also, in combination with GROUP. If a group is defined, you can smash it with a right mouse click while the Ctrl key is pressed. Use DELETE to hide smashed texts.

Keep the Shift key pressed while using the SMASH command in order to unsmash text. Text is not editable anymore and appears at original position(s) after a window refresh (also possible in the context menu with unSmash). Alternatively you can also switch on or off the option Smashed in the context menu's Properties entry.

 21- **MITER**

Round off or bevel wire joints (also possible for nets, buses, polygon contours). The grade of mitering is determined by the miter radius. Positive sign results in a rounded joint, negative sign in a bevel.The miter radius influences some wire bends, too (see help function: SETcommand, Wire_Bend).

 22- **SPLIT**

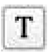
Insert an angle into a wire or net.

23- **INVOKE**

Devices that consist of more than one Symbols (Gates) can be fetched Gate by Gate, for example in certain order (Gate D before Gate C), if wanted. INVOKE can also be used to fetch power supply Gates that do not appear automatically in the Schematic. This is useful and required, for example, when you are adding decoupling capacitors to your design.This command allows you also to add a Gate from a Device which is located on another sheet. In such a case, type the name of the Device (e.g. IC1) into the command line after the INVOKE command has been selected.

 24- **WIRE**

Draw line (this command is called WIRE because it is used to define electrical connections, i.e. wires, in the Layout Editor). The type of line can be changed with CHANGE STYLE. Clicking the right mouse button changes the bend mode (SET WIRE_BEND). WIRE can also be used to draw arcs. Please note the particularities in combination with the Ctrl and Shift key in the help function: If you press, for example, the Ctrl key while starting to draw a wire, the wire begins exactly at the end of an already existing wire nearby. Even if this wire is not in the currently set grid. Wire width, style and layer will be adopted from the already existing wire.

25- TEXT 

Placing text. Use CHANGE SIZE to alter the height of the text. If the text is using a vector font, CHANGE RATIO will alter the thickness. CHANGE TEXT is used to alter the text itself. CHANGE FONT alters the typeface. You can change label texts by assigning a different name to the bus or to a net by means of the NAME command. See also LABEL command.

 26- **CIRCLE**

Draw a circle. Circles with a width of 0 are drawn as filled circles.

 27- **ARC**

Draw an arc (also possible with WIRE). CHANGE CAP FLAT | ROUND defines straight or rounded ends for arcs.

 28- **RECT**

Draw a rectangle.

 29- **POLYGON**

Draw a polygon (copper areas in any shape).

 **29- BUS**

Draw a bus line. The meaning of a bus is more conceptual than physical. It is only a means to make a schematic easier to read. Only nets define an electrical connection. Nets, however, can be dragged out of a bus.

30- NET 

Draw a net. Nets with the same name are connected (even if located on different sheets). Nets and pins which appear to the eye to be connected are not necessarily electrically connected. Please check with the SHOW command, the ERC, or by exporting a netlist or pin list (EXPORT NETLIST or PARTLIST). See also the help for MOVE.

31- JUNCTION 

Place the symbol for a net connection. In general, junctions are placed automatically, but nets which cross over can also be joined manually by the JUNCTION command.

32- LABEL 

Place the name of a bus or net as a label. Labels cannot be changed with CHANGE TEXT but rather with the NAME command because the label represents the net name. If the label option XREF (in the parameter toolbar or by CHANGE XREF ON) is set, a cross reference pointing to an further instance of the chosen net on the next sheet is generated automatically. The cross reference label format can be defined in the menu Options/Set/Misc, Xref label format. See the help function of the LABEL command for the meaning of the placeholders that can be used. For a proper location of the object you should use a drawing frame with classifications for columns and rows. Such frames can be defined with the FRAME command. The library frames' already contains such frames.

 33- ATTRIBUTE

Defines an attribute for a component. Attributes are free definable and can contain any information. Through the menu Edit/Global attributes.. you can define attributes that are valid for all components respectively for the whole schematic.

 34- ERC

Perform an Electrical Rule Check and a consistency check for schematic and board, if already existing. A positive consistency check allows the Forward & Back Annotation engine to run.

4.13 Commands Not Available in the Command Toolbar

Menu items already explained in the Control Panel section are not discussed here. The following commands can be entered into the command line as text inputs. Some of them are available as menu items.

Most of them can be used in the Schematic and in the Layout and even in the Library Editor.

ASSIGN

Assign function keys. The most convenient way of doing this is to use the Options/Assign menu.

CLASS

Select and define net classes (Edit/Net classes...). A net class specifies the width of a track, the clearance from neighbouring signals, and the diameter of vias for the Autorouter and the ROUTE command. These settings are also used in polygons. See also page 117.

CLOSE

Text command for closing an editor window (File/Close).

EDIT

Text command for loading a file or a library object. You can, for instance, load a board from the Schematic Editor (EDIT name. brd).

FRAME

Define a drawing frame for the Schematic (Draw/Frame). Also possible for a board drawing.

EXPORT

Output lists (especially netlists), directories, script files, or images (File/Export...).

LAYER

Choose or define the drawing layer. When using drawing commands the layer can be chosen in the parameter toolbar. To create, for example, a new layer with number 200 and layer name My layer, type in the command line: LAYER 200 My layer In case you created a Layout, for example, with the EAGLE Light Edition and upgraded to the Standard Edition because you would like to use two additional inner signal layers, you have to create these layers with the LAYER command first: LAYER 2 Route2

MENU

Specifies the contents of the text menu. See also the example in the appendix. The text menu can be made visible with the aid of Options/User Interface. See help function for details.

OPEN

Text command for opening a library for editing (Library/Open). This command is not identical to the File/Open menu item of the Schematic Editor, which only lets you select schematics. You can use the OPEN command as an alternative to the File menu of the Control Panel.

PACKAGE

In case there is more than one Package variant defined in the library for a part (Device), a typical example would be a resistor from rcl.lbr, it is possible to change the currently used Package with the PACKAGE or with the CHANGE PACKAGE command. This can be done in the Schematic or in the Layout Editor.

PRINT

Call up the print dialog with the printer icon in the action toolbar  or from the menu item File/Print.... Usually the PRINT command is used to print schematics or for checking the drawings needed for the PCB production. The actual production data are generated with the CAM Processor. If you want to output your drawing in black and white check the Black option (and Solid, if you don't want layers to be printed in their different fill styles). The caption text is suppressed unless you check Caption. Set Page limit to 1, if your drawing is to be fitted on one page. If you prefer to print the currently visible drawing window instead of the whole drawing, select Window instead of Full in the Area option.

QUIT

Quit EAGLE. Identical with the menu item File/Exit or Alt-X

REMOVE

Delete files or schematic sheets. REMOVE .S3 ← for instance, deletes sheet 3 of the loaded schematic.

SET

Set system parameters and modes. Best done via the Options/Set menu item. Please note that not all of the possibilities are available through this dialog. Presetting can be defined in the script file eagle. scr by using text commands. Further information can be found

TECHNOLOGY

If a part (Device) has been defined with various technologies in the library, see typical examples in 74xx.lbr, it is possible to change the currently used technology with the TECHNOLOGY or with the CHANGE TECHNOLOGY command. This can be done in the Schematic or in the Layout Editor.

UPDATE

The UPDATE command checks the parts in a board or schematic against their respective library objects and automatically updates them if they are different. (Library/Update... or Library/Update all). The context menu in the Control Panel's' tree view offers the Options Use all and Use none for a quick selection of libraries.

WRITE

Text command for saving the currently loaded file. Please note that, in contrast to Save as, the name of the currently edited file is never changed when the WRITE command is used

Mouse Keys

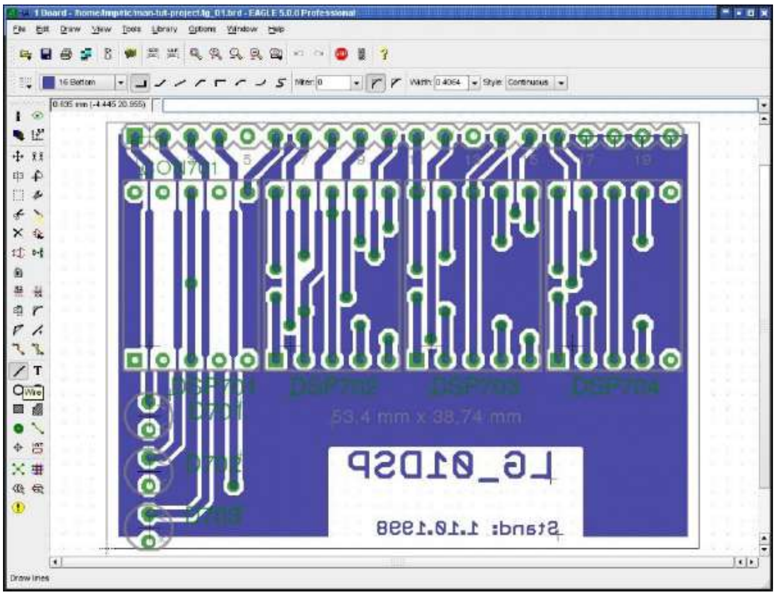
The middle and right mouse button have a special meaning for a number of commands. You can use the middle mouse button only if the operating system knows your mouse is a 3-button mouse, that is your mouse must be installed this way. If you are working with a wheel mouse, you can zoom into and out of the drawing with the help of the mouse wheel. The option Mouse wheel zoom in the Options/User Interface menu determins the zooming in/out factor per step. The value is set to 1.2 by default. Selecting a value of 0 allows you to use the wheel for scrolling. Keep the mouse wheel or the middle mouse button pressed for panning. Mouse clicks in combination with the Shift, Ctrl, and Alt key can have various functions, for example, while selecting objects with MOVE or while drawing lines with WIRE. The help section on Keyboard and Mouse and the help of the referring command gives you more details.

4.14 Selecting Neighboring Objects

If one of two objects which are very close together is to be selected, the individual objects are highlighted one after the other. The user can select the highlighted object with the left mouse button, or proceed to the next one with the right mouse button. The status bar of the editor window shows information about the pre-selected object. See also help function (SET command, SELECT_FACTOR).

4.15 The Layout Editor Window

The Layout Editor window opens when you open an existing board file or create a new board. If you own the Schematic Editor you will normally draw a schematic first and then generate the board file with the BOARD command, or by clicking the Board icon.



4.10 Layout Editor window

The Layout Editor window appears very much like the Schematic Editor window. Even if you don't work with the Schematic Editor, you should study the previous section, as most of the information there applies to the Layout Editor, too. Only the commands in the command toolbar are discussed again, as some commands differ in their use.

Descriptions of commands that cannot be reached through the command toolbar are also to be found in the section concerning the Schematic Editor window. All of the commands can also be reached through the pull-down menus in the menu bar. This also applies, of course, to the Schematic and Layout Editor windows.

4.16 The Commands on the Layout Command Toolbar

 **INFO**

Shows the properties of the selected object. Typing INFO IC1 in the command line results in the properties dialog of the object named IC1. Depending on the selected object some of the properties can be altered here.

 **SHOW**

Highlights the object to be selected with the mouse. It's also possible to enter the object's name (even several names at once) in the command line. * and ? are allowed to be used as wildcards, as well. Ctrl + SHOW toggles the show state of the selected object.

 **DISPLAY**

Select and deselect the layers to be displayed. Components on the top side of the board can only be selected if the layer 23, tOrigins, is displayed. The same applies to components on the bottom side of the board and layer 24, b Origins. Only those signal layers that are defined in the Layer setup are shown. See Appendix for the meaning of the layers.

The DISPLAY command supports so-called aliases. This allows you to name certain combinations of layers and use it as a parameter with the LAYER command. A quick change from one view to another layer combination is possible with this command. DISPLAY LAST switches to the last displayed layer combination. Detailed information about the DISPLAY command can be found in the help function.

MARK

The following mouse click defines the new origin for the coordinate display. Relative coordinates (R x-value y-value) and polar values (P radius angle) are shown in addition to absolute coordinates in the coordinate display box. If you first click the MARK icon and then the traffic light icon, only the absolute coordinate values will be displayed again.

MOVE

Move any visible object. The right mouse button rotates the object. The MOVE command cannot connect signals even if a wire (trace) is moved over another wire or a pad. Use ROUTE or WIRE to route signals. Keeping the Ctrl key pressed while selecting an object selects it in a particular manner. Please consult the help function for details (CRICLE, ARC, WIRE, MOVE, ROUTE etc.). For moving groups,.

COPY

Copy parts and other objects. When copying objects, a new name will be assigned, but the value will be retained. When copying a single wire, the copy will have the same name. COPY can be used with groups and works the same as it would be with CUT and PASTE within a drawing.

MIRROR

Mirror objects. Components can be placed on the opposite side of the board by using the MIRROR command.

ROTATE

Rotate objects (also possible with MOVE). Keep the left mouse button pressed to rotate the selected object by moving the mouse. The parameter toolbar shows the current angle. This can be done with groups (GROUP and right mouse button)as well. ROTATE can be used with groups, as well. Activate ROTATE, press the Ctrl key and click with the right mouse button into the drawing to set the center of rotation. The group will be rotated counterclockwise by the given angle. Alternatively type in the angle in the Angle box or in the command line. Details about the syntax can be found in the help function.

GROUP

Define a group which can then be moved, rotated, or copied with CUT and PASTE to another drawing or whose properties should be changed. After clicking the icon, a rectangular group can be defined by holding down the left mouse button and dragging the cursor to the diagonal corner of the rectangle, or if you want to define a non-rectangular group area, use the left mouse button to determine the corners of the polygon. Then click the right mouse button to close the polygon. GROUP ALL in the command line selects all objects. To be sure that all objects are selected DISPLAY ALL layers before. On the other hand, deselecting specific layers can exclude certain objects from the selection.

CHANGE 

Change the properties of an object, for example the width of a wire or the size of a text. If the Esc key is pressed after changing a property, the previously used value menu will appear again. In this way a new value can be conveniently chosen. See also the help function. Alternatively, object properties can be viewed and some of them even changed with the context menu's Properties entry. The context menu opens after a right mouse click onto the object.

 **CUT**

Transfer the objects of a previously defined group into the paste buffer. Activate the CUT command and click with the left mouse button into the group to set a reference point. All other commands executed with a group expect a right mouse click while the Ctrl key is pressed.

PASTE 

Insert objects from the paste buffer into the drawing. Restrictions: see help. Not identical to the Windows paste function.

 **DELETE**

Delete visible objects. If a group has been defined, it can be deleted with the right mouse button while the Ctrl key is pressed. DELETE SIGNALS in the command line erases all tracks and signals in the layout, provided there is no consistent schematic loaded. The DELETE command deletes an entire polygon when clicking on a polygon wire with the Shift key pressed.

Keeping the Ctrl key pressed while clicking with the left mouse button on a wire bend will delete the bend. A new direct connection between the next bends will be drawn now.

If objects cannot be deleted, the reason can lie with error polygons related to the DRC command. They can be deleted with the ERRORS command (ERRORS CLEAR). If layer 23, tOrigins, or 24, bOrigins, is not displayed, components cannot be deleted.

ADD

Add library elements to the drawing. It offers a convenient search function for Packages here. USE specifies which libraries are available. A right-click onto the ADD icon opens a popup menu that contains a list of recently placed Devices.

PINSWAP

Swap two signals connected to equivalent pads of a component, provided the pins have been defined with the same Swap level.

REPLACE

Replace a component (or a Package, if there is no schematic) by another one from any library. If you want to change the Package variant only and not the whole Device, use CHANGE PACKAGE or the PACKAGE command. A right-click onto the REPLACE icon opens a popup menu that shows a list of recently replaced components.

LOCK

Locks the position and orientation of a component on the board. If a component is locked, you can't move it or duplicate it with CUT and PASTE. Shift + LOCK unlocks the component. This is also possible with the unLock entry of the context menu. To be able to distinguish locked from unlocked components, the origin cross of a locked component is displayed like a 'x' instead of a '+'. The position of a locked component can be changed, however, by typing in new coordinate values in the properties dialog.

 **NAME**

Give names to components, signals, vias, and polygons.With NAME it's possible to move a polygon from one signal to another.

 **VALUE**

Provide values for components. A resistor, for example, gets 100k as its value. A right-click onto this icon opens a list of already used values. Select an entry and apply it to one or more components by clicking onto them successively.

SMASH 

Separate name, value, and attribute (if any) texts from a Device, so that they can be placed individually. The size of detached (smashed) texts can also be individually changed. Also in combination with GROUP. If a group is defined, you can smash it with a right mouse click while the Ctrl key is pressed. Use the DELETE command to hide smashed texts. Keep the Shift key pressed while using the SMASH command in order to unsmash texts. They are not editable anymore and appear at their original positions after a window refresh (also possible with unSmash in the context menu). Alternatively you can switch on or off the option Smashed in the context menu's Properties entry.

 **MITER**

Round off or bevel wire joints (also possible for polygon contours). The grade of mitering is determined by the miter radius. Positive sign results in a rounded joint, negative sign in a bevel. The miter radius influences some wire bend modes, too (see help function: SET, Wire_Bend).

 **SPLIT**

Insert a bend into a wire. If you want to change, for example, the layer for a section of an already routed track, you can insert two wire bends with the SPLIT command and change the layer of the newly created segment with the CHANGE LAYER. EAGLE will set vias automatically at the position of the wire bends. You can use the SPLIT command for a quick re-routing of an already existing track. Click onto the track to insert a wire bend. Now move the mouse and route it anew. To remove the previous track use the RIPUP command or DELETE in combination with the Ctrl key.

 **OPTIMIZE**

Joins wire segments in a signal layer which lie in one straight line.

 **ROUTE**

Route signals manually. Airwires are converted to wires. If your EAGLE license comes with the Autorouter module, the ROUTE command supports the Follow-me router mode which automatically processes the trace of a selected signal. This command offers several options with the different mouse buttons, also in combination with the Ctrl and Shift key.

Ctrl + Left starts routing at any given point along a wire or via **Shift + Left** if the airwire begins at an already existing wire and this wire has a different width, the new wire adopts this width **Center** selects the layer **Right** changes the wire bend style **Shift +Right** reverses the direction of switching bend styles **Ctrl + Right** toggles between corresponding bend styles **Shift + Left** places a via at the end point of the wire **Ctrl + Left** defines arc radius when placing a wire's end point More information can be found in the help function.

RIPUP

Convert routed wires (tracks) into unrouted signals (airwires). Change the display of filled (calculated) polygons to outline view. Using signal names in the command line allows you to ripup only certain signals, to exclude particular signals, or to execute the command exclusively for polygons. More details can be found in the help function. Wires not connected to components must be erased with DELETE.

WIRE

Draw lines and arcs. If used in the layers 1 through 16, the WIRE command creates electrical connections. The Style parameter (CHANGE) determines the line type. The DRC and the Autorouter always treat a WIRE as a continuous line, regardless of what Style is used. Clicking the right mouse button changes the wire bend (SET WIRE_BEND). Please note the particularities in combination with the Ctrl and Shift key in the help function: If you press, for example, the Ctrl key while starting to draw a wire, the wire begins exactly at the end of an already existing wire nearby. Even if this wire is not in the currently set grid. Wire width, style and layer will be adopted from the already existing wire.

TEXT

Placing text. Use CHANGE SIZE to alter the height of the text. If the text is using a vector font, CHANGE RATIO will alter the thickness. CHANGE TEXT is used to alter the text itself. CHANGE FONT alters the typeface. The option Always vector font (Options/User Interface) shows and prints all texts in vector font, regardless of which font is actually set for a particular text.

If you want to have inverted text in a copper layer, you have to enter the text in the layers 41, tRestrict, or 42, bRestrict, and draw a copper plane in Top or Bottom layer around the text with the POLYGON command. The polygon keeps the restricted areas (which is the text) free from copper. It is strongly recommended to write texts in copper layers as vector font! So you can be sure that the CAM Processor's output is identical with the text shown in the Layout Editor. See also help function.

CIRCLE

Draw a circle. This command creates restricted areas for the Autorouter/Follow-me router, if used in the layers 41, tRestrict, 42, bRestrict, or 43, vRestrict. Circles with wire width = 0 are drawn as filled.

ARC

Draw an arc (also possible with WIRE). CHANGE CAP FLAT | ROUND defines straight or rounded ends for arcs. If the arc is a part of a trace and both ends are connected to a wire, caps will be round. Arcs with flat caps are emulated when generating manufacturing data in Gerber format with the CAM Processor. That means they will be drawn with small short straight lines. Arcs with round caps won't be emulated.

RECT

Draw a rectangle. This command creates restricted areas for the Autorouter or Follow-me router, if used in the layers 41, tRestrict, 42, bRestrict, or 43, vRestrict.

POLYGON

Draw a polygon.

Polygons in the signal layers are treated as signals. They keep an adjustable distance to objects belonging to other signals (copper pouring, flood fill). This enables you to realize different signal areas on the same layer and make isolated regions for your design. The contour of a polygon in the outline mode is displayed as a dotted line. The POLYGON command creates restricted areas for the AutoRoute/Follow- me router, if used in the layers t Restrict, b Restrict, or v Restrict, For other possibilities of the POLYGON command

VIA

Place a plated-through hole. Vias are placed automatically if the layer is changed during the ROUTE command. You can assign a via to a signal with the NAME command by changing it's name to the name of the signal. Vias can have different shapes in the outer layers (round, square, octagon) , but are always round in inner layers.

SIGNAL

Manual definition of a signal. This is not possible if the Forward&Back Annotation is active. In that case you have to define the connection with the NET command in the Schematic Editor.

HOLE

Define a mounting hole (not plated-through).

ATTRIBUTE

Defines an attribute for a component. Through the menu Edit/Global attributes.. you can define attributes that are valid for the whole layout.

RATSNEST

Calculate the shortest airwires possible and the real mode (filled) display of polygons. Use the RATSNEST command with a signal name in order to calculate and display or hide a certain airwire. A preceding exclamation mark hides the airwires of the given signal name. More information can be found in the help function. The polygon calculation can be deactivated with the SET command. Either through the menu Options/Set/Misc or by typing in the command line: SET POLYGON_RATSNEST ON | OFF or in short: SET POLY ON | OFF. RATSNEST will be executed automatically for the selected signal while drawing a wire with ROUTE. While RATSNEST is active the status bar of the Layout Editor displays the name of the currently calculated signal.

AUTO

Start the AutoRoute. If you type AUTO FOLLOWME in the command line, the AutoRoute Setup window opens in the follow me mode, which allows to set the parameters for the follow me router only.

ERC

Perform a consistency check for schematic and board.

DRC

Define Design Rules and perform Design Rule Check.

ERRORS

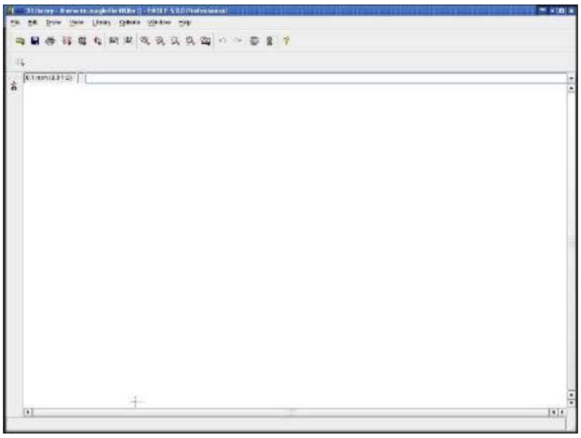
Show errors found by the DRC. If you haven't already processed a Design Rule Check for the board, it will be done automatically before showing the error list, if there are any errors found. There are further commands for the Layout Editor, as they are in the Schematic, that are not available in the Command Menu. Please take a look at the section beginning with page 59. Most of them are valid in Schematic and Layout.

4.17 The Library Editor Window

The Library Editor window opens when you load a library for creating or editing components. A library normally has three different elements: Packages, Symbols and Devices. A Package is a Device's housing, as will be used in the Layout Editor (on the board). The Symbol contains the way in which the Device will be shown in the schematic.

The Device represents the link between one (or more) Symbol(s) and a Package. Here we define the connection between a pin of a Symbol and the referring pad of the Package. We call it a Device set if the component exists in more than one Package and/or technology variant. Even if you do not have the Schematic Editor, you can still create and edit Symbols and Devices.

A library need not contain only real components. Ground or supply symbols as well as drawing frames can also be stored as Devices in a library. These Symbols do not normally contain any pins. There are also libraries that only contain Packages. These libraries can only be used in the Layout Editor. Extensive examples of the definition of library elements are to be found in a section entitled Component Design Explained through Examples, starting on page 201 in this manual. When a library is loaded the following window appears



4.11 Library Editor window: No element has yet been loaded

4.18 Load or Rename Package, Symbol, or Device

The following commands are important for navigating within a library:

 **EDIT**

Load Device or Package (if you only have the Layout Editor) for editing. From the left: Load Device, load Package, load Symbol. These icons are shown in the action toolbar. If you click on one of these icons with the right mouse button, or long-click with the left mouse button on one of theses icons, a list with the recently edited objects will pop up.

REMOVE

Delete Device/Package/Symbol from library. Available only through the Library menu or the command line. See help function or the chapter about Library and Part Management in this manual for more.

RENAME

Rename Device/Package/Symbol in library. Available only through the Library menu or the command line. See help function or the chapter about Library and Part Management in this manual for more.

The Package Editing Mode

The definition of a component is described briefly below. There is a more extensive guide in the Component Design Explained through Examples section. The icons available in the command toolbar are equivalent to the identical icons of the Schematic or Layout Editor.

Design New Package

You change into Package editing mode through the Package icon  in the action toolbar. Type in the name of a package, and reply to the confirming question Create new package 'packagename'? with yes. Place pads (though-hole contacts) or SMDs (SMD contact areas) with the following commands which are only available in the Package Editor.

 **PAD**

Place the pad of a conventional (through-hole) component. The pad comes with a plated-through drill that goes through all signal layers. The pad shape can be round, square, octagon or long in the outer signal layers. In the inner signal layers pads are always round.

 **SMD**

Place a SMD pad. You can change the name of the pads or SMDs with the NAME command. Use the WIRE, ARC, etc. commands to draw the symbol for the silkscreen on layer 21, tPlace, additional graphical information for the documentation print into layer 51, tDocu. Draw restricted areas for the Autorouter, if needed, in layers 41, tRestrict, 42, bRestrict, or 43, vRestrict, or in layers 39, tKeepout, or 40, bKeepout, by using the commands CIRCLE, RECT, or POLYGON. Place mounting holes with the HOLE command, if needed. Use the TEXT command to place the string >NAME in layer 25, tNames, serving as a text variable containing the name of the component, the string >VALUE in layer 27, tValues, serving as a text variable containing the value of the component. Use the DESCRIPTION command to add a description for the Package. HTML text format can be used for this. You will find further information in the help pages.

The Symbol Editing Mode

Defining a Symbol means defining a part of a Device which can be placed individually in a schematic. In the case of a 74L00 this could be one NAND gate and the two power pins, defined as another Symbol. In the case of a resistor, the Device contains only one Symbol which is the representation of the resistor. You now change into Symbol editing mode through the Symbol icon  in the action toolbar. Enter the name of the Symbol, and reply to the confirming question Create new symbol 'symbolname'? with Yes.

Design a New Symbol

Use the commands WIRE, ARC, etc. to draw the schematic representation of the Symbol into layer 94, SymbolsPlace the pins by using the following PIN command, which is only available in the Symbol editing mode:

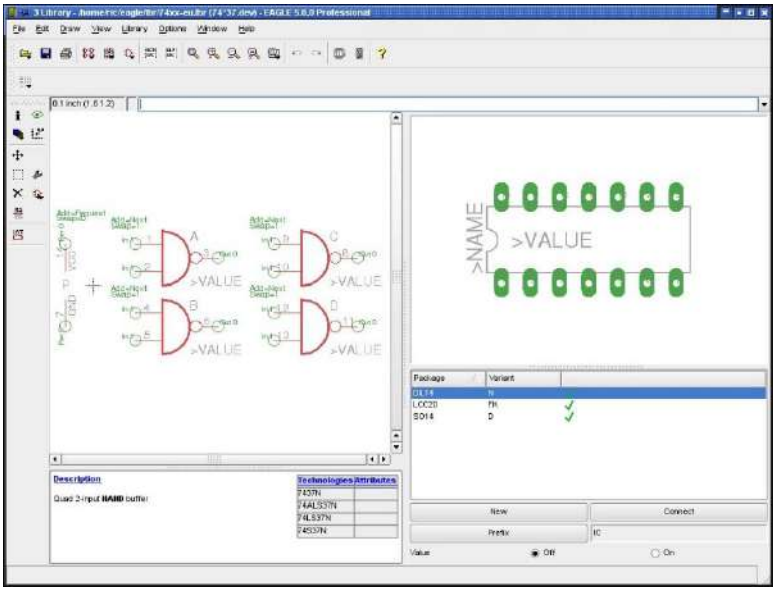
PIN

Place pins. You can adjust the pin parameters (name, direction, function, length, visible, Swaplevel) in the parameter toolbar while the PIN command is active, or later with the CHANGE command. The pin parameters are explained starting on page 206 and in the help pages under the keyword PIN. Pin names are changed using the NAME command. Use the TEXT command to place the string >NAME in layer 25, tNames, serving as a text variable containing the name of the component, the string >VALUE in layer 27, tValues, serving as a text variable containing the value of the component.

The Device Editing mode

Components are defined as Devices. In the Device editing mode you do not draw anything, but you define the following: which Package variant is used, which Symbol(s) is/are used (called Gate within the Device), which names are provided for the Gates (e.g. A, B which technologies are available (e.g. 74**L**00, 74**LS**00, 74**HCT**00) if the Device should have additional user-definable attributes, if there are equivalent Gates which can be interchanged (Swaplevel) how the Gate behaves when added to a schematic (Addlevel), the prefix for the component name, if a prefix is used, if the value of the component can be changed or if the value should be fixed to the Device name, which pins relate to the pads of the Package (CONNECT command whether a description for this component should be stored in the library.

The following diagram shows the fully defined 7400 Device with four NAND gates and a supply gate in various Package and technology versions. If you click onto one of the gates with the right mouse button, the context menu with the executable commands pops up. Furthermore you can display the Properties of the gate. Click on Edit Symbol to open the Symbol Editor.



4.12 Device Editor window

Create Actual Components from Symbols and Packages

Switch to the Device editing mode by clicking the Device icon  in the action toolbar. Type in the Device name and confirm the question Create new device 'device name'? with Yes. Use the following commands to create a Device.

 **ADD**

Add a Symbol to a Device. Gate name, Swaplevel, and Addlevel can be defined in the ADD command in the parameter toolbar, or redefined later with the CHANGE command. The Swaplevel specifies whether there are equivalent Gates. The Add level defines, for instance, if a Gate is to be added to the schematic only on the users request. Example: the power gate of an integrated circuit which is normally not shown on the schematic.

 **NAME**

Change Gate name.

 **CHANGE**

Change Swap level or Add level.

PACKAGE

Define and name Package variant(s). The PACKAGE command is started by clicking on the New button in the Device Editor window, or by typing on the command line. Choose the requested Package variant.

More information about this can be found on page 251.

CONNECT

Define which pins (Gate) relate to which pads (Package).

PREFIX

Provide prefix for the component name in the schematic (e.g. R for resistors).

VALUE

In the Device mode, VALUE is used to specify whether the component value can be freely selected from within the schematic diagram or the layout, or whether it has a fixed specification. On: The value can be changed from within the schematic (e.g. for resistors). The component is not fully specified until a value has been assigned.

Off: The value corresponds to the Device name, including, when present, assignment of the technology and the Package version (e.g. 74LS00N).

Even if Value is Off, the value of a component can be changed. A query checks if this action is intended. The altered value of the component remains unchanged, if the Technology or the Package version is altered.

TECHNOLOGY

If necessary, various technologies can be defined, for example, for a logic component. Click on Technologies therefore.


ATTRIBUTE

Click on Attributes to define any additional attribute for the Device. A detailed description can be found in the chapter about libraries in this manual.

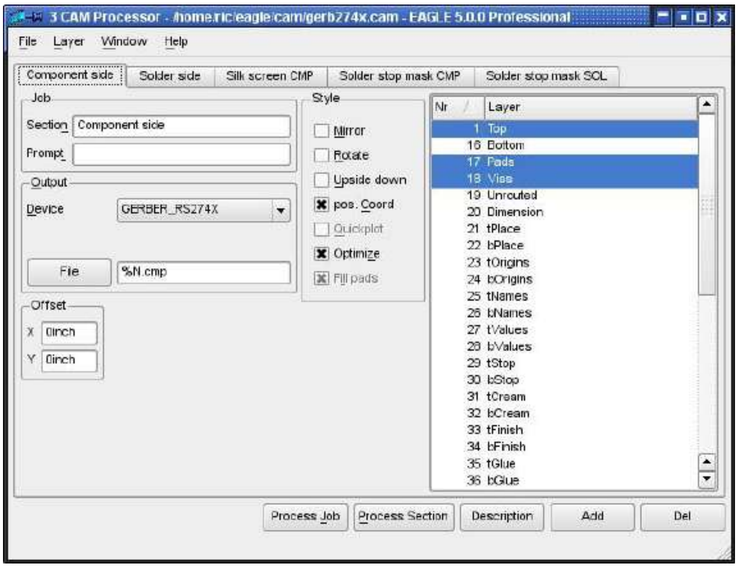
DESCRIPTION

Compose a description of the Device which can also be examined by the search function associated with the ADD dialog. Information about Copying of Packages, Symbols and Devices can be found from page 256 on.

4.5 The CAM Processor

Manufacturing data is generated by means of the CAM Processor. A number of drivers for the data output are available. The drivers are defined in the file eagle.def, which can be edited with any Text Editor.

Output to matrix printers, however, is not created with the CAM Processor but with a PRINT command. The EAGLE license conditions allow you to give the CAM Processor to your board manufacturer. You only need to supply them with a copy of the EAGLE Light Edition. The CAM Processor of EAGLE Light runs without limitation. Alternatively, EAGLE Light can be downloaded from CadSoft's web server.



4.13 The CAM Processor

The CAM Processor can also be started directly from the command line. A number of command line parameters can be passed to it when it is called. These are listed in the appendix.

Generate Data

Starting the CAM Processor

There are different ways to start the CAM Processor: You can do this directly from the Layout or Schematic Editor window with the CAM Processor icon  in the action toolbar or through the menu File/CAM Processor. The current schematic or board will be loaded automatically from the Control Panel by clicking on one of the entries in the Tree View's CAM Jobs branch. Then the selected CAM Job will be loaded automatically. You still have to load the schematic or board from which you want to make the CAM Processor manufacturing data from through the File/Open menu by using the command prompt (Windows command prompt, Terminal or Console window) without graphical user interface. Particular information can be found in the appendix chapter about EAGLE Options.

Load Job File

A job defines the sequence of several output steps in an automatic data creation task. You can, for example, use a job to generate individual files containing the Gerber data for several PCB layers. A job is loaded with the File menu of the CAM Processor or with a double- click on one of the Tree view's CAM Jobs entries in the Control Panel. A job is not absolutely essential for output. All the data can be made step by step manually.

Load Board

Before you can generate an output you must open the File menu and load a board file, if not already loaded automatically while stating the CAM Processor from an Editor window.

Chapter Five

Conclusion and Future Work

5.1 Conclusion

It can be concluded that the work of the CNC using Arduino was successful, and the CNC control system consists of the CNC mach3 card 4 axis USB 100kh, driver motor, these parts are effective and inexpensive in terms of cost. This system or work is aimed at producing, quantifying, providing the working hand and accomplishing the work quickly as well as accuracy, precise measurements, access or making parts that are hard to make or reach Thus, we will get a larger output quantity, high speed and precise measurements.

5.2 Suggestion for Future Works

This machine can be developed by using a more powerful drilling machine with a drilling head capable of drilling iron and thus becoming a machine to dig on iron instead of wood, or add a burning laser to write on wood after drilling it and thus get a wonderful taste of a beautiful nature, also can use the heater instead of the perforating machine which works on heating plastic materials can be used to make 3D parts useful. Also we can add more axis's (fourthly axis , fifthly axis).

References

[1] Viktor Stenberg, “Student CNC Guide”, KTH Royal Institute of Technology, Stockholm, Sweden, 2015-12-07.

[2] Tapas Kumar Mahato,“Experimental Investigation Of Micro drilling Operation of Printed Circuit Board,” A thesis submitted in partial fulfilment of the requirement for the degrees of Bachelor of Technology in Mechanical Engineering, NIT Rourkela, India, 2013.

[3] Arpita Srivatava, “DC Motor Speed Control Using ATMEGA89S52 and MATLAB GUI Application”, International Journal of Science and Research (IJSR), Volume No.5, Issue 12, Page No. 19412-19414, 2015.

[4] Shaikh Noor Farooque, Ansari Mohammed Faizan, Javed Shaikh, PragatiPal “Automated PCB Drilling Machine with Efficient Path Planning,” International Journal of Advanced Research in Computer and Communication Engineering, Vol. 4, Issue 4, Page 108-110, April 2015.

[5] Rodrigo Basniak, Márcio Fontana Catapan,“Design of a PCB Milling Machine,” ABCM Symposium Series in Mechatronics, Section VIII - Sensors & Actuators, Vol. 5, Page 1339-1348, 2012.

[6] N. Balasu bramanyam and Prof. Smt. G. Prasanthiy “Design and Fabrication of an Automatic PC-Based Drilling Machine,” HCTL Open Int. J. of Technology Innovations and Research HCTL Open IJTIR, Volume 7, January 2014.

[7] Mohammed Abdalla. A. Ali, Ahmed Mohamed A. ELShaikh, Sharief F. Babiker “Controlling the CNC Machine using Microcontroller to Manufacture PCB,” IEEE Conference of Basic Sciences and Engineering Studies (SGCAC), pp. 116-120, 2016.

[8] Anjali K M, Niveditha P S, P Shyama, Sreeja Sreedharan V, Susmi P S, “PCB Plotter and Retracer,” International Journal of Industrial Electronics and Electrical Engineering, ISSN, Volume No.: 4, Issue No: 5, May 2016.

[9] Mohammad Kamruzzaman Khan Prince, Muhsi-Al Mukaddem Ansary, Abu Shafwan Mondol "Implementation of a Low-cost CNC Plotter Using Spare Parts," International Journal of Engineering Trends and Technology (IJETT) – Volume-43 Number-6 -January 2017.

[10] Nguyen Huu Phuong, Ho Van Thoi "Design and Build an Automatic PCB Drilling Machine", University of Natural Sxiences, Vietnam National University HCM,

[11] Nareen Hafidh Obaeed1, Mostafa Adel Abdullah1, Momena Muath, Maryam Adnan, Hind Amir , "Study The Effect of Process Parameters of CNC Milling Surface Generation Using Al-alloy 7024", Diyala Journal of Engineering Sciences Vol. 12, No. 03, September 2019, pages 103-112

[12] P.L.S.CAlwis, A.S Premarathna, Y.P Fonseka, S.M Samarasinghe, J.V. Wijayakulasooriya "Automated Printed Circuit Board (PCB) Drilling Machine with Efficient Path Planning", SAITM Research Symposium on Engineering Advancements 2014 (SAITM – RSEA 2014.

[13] Niranjan. G, Chandini. A, Mamatha. P "Automated Drilling Machine withDepth Controllability" International Journal of Science and Engineering Applications Volume 2 Issue 4, 2013, ISSN-2319-7560 (Online).

[14] KetulSheth, "Social Network For Smart Devices Using Embedded Ethernet", Computer Science & Engineering: An International Journal (CSEIJ), Vol. 3, No. 2, April 2013.

[15] Geeetech Wiki 'Stepper Motor 5V 4-Phase 5-Wire & ULN2003 Driver Board for Arduino